ABSTRACT

Sierra/SD provides a massively parallel implementation of structural dynamics finite element analysis, required for high-fidelity, validated models used in modal, vibration, static and shock analysis of weapons systems. This document provides a user’s guide to the input for Sierra/SD. Details of input specifications for the different solution types, output options, element types and parameters are included. The appendices contain detailed examples, and instructions for running the software on parallel platforms.
## CONTENTS

<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.1</td>
<td>Release 5.6</td>
<td>3</td>
</tr>
<tr>
<td>0.2</td>
<td>Release 5.4</td>
<td>4</td>
</tr>
<tr>
<td>0.3</td>
<td>Release 5.2</td>
<td>6</td>
</tr>
<tr>
<td>0.4</td>
<td>Release 5.00</td>
<td>8</td>
</tr>
<tr>
<td>0.5</td>
<td>Release 4.58</td>
<td>11</td>
</tr>
<tr>
<td>1.</td>
<td>HOW TO RUN Sierra/SD</td>
<td>15</td>
</tr>
<tr>
<td>1.1</td>
<td>Accessing Sierra/SD</td>
<td>15</td>
</tr>
<tr>
<td>1.2</td>
<td>Modules and Executables</td>
<td>15</td>
</tr>
<tr>
<td>1.3</td>
<td>The Sierra/SD salinas Executable</td>
<td>16</td>
</tr>
<tr>
<td>1.4</td>
<td>MPI Parallel Execution</td>
<td>17</td>
</tr>
<tr>
<td>1.4.1</td>
<td>Number of MPI Processes Needed</td>
<td>18</td>
</tr>
<tr>
<td>1.4.2</td>
<td>Mesh Decomposition</td>
<td>19</td>
</tr>
<tr>
<td>1.4.3</td>
<td>Running the Sierra/SD Executable in Parallel</td>
<td>19</td>
</tr>
<tr>
<td>1.4.4</td>
<td>Post Processing in Parallel</td>
<td>19</td>
</tr>
<tr>
<td>1.5</td>
<td>File system concerns</td>
<td>20</td>
</tr>
<tr>
<td>1.6</td>
<td>Workflow Examples</td>
<td>20</td>
</tr>
<tr>
<td>1.7</td>
<td>Thread Parallelism</td>
<td>22</td>
</tr>
<tr>
<td>1.8</td>
<td>Troubleshooting</td>
<td>22</td>
</tr>
<tr>
<td>1.8.1</td>
<td>Stand-Alone Tools</td>
<td>22</td>
</tr>
<tr>
<td>1.8.2</td>
<td>Using Sierra/SD To Troubleshoot</td>
<td>23</td>
</tr>
<tr>
<td>1.8.3</td>
<td>Modal Analysis</td>
<td>24</td>
</tr>
<tr>
<td>1.8.4</td>
<td>Evaluating Memory Use</td>
<td>24</td>
</tr>
<tr>
<td>1.8.5</td>
<td>Identifying Problematic Subdomains</td>
<td>24</td>
</tr>
<tr>
<td>1.8.6</td>
<td>Limitations of SD Finite Elements</td>
<td>24</td>
</tr>
<tr>
<td>1.8.7</td>
<td>Problematic Elements and Connectivity</td>
<td>25</td>
</tr>
<tr>
<td>1.9</td>
<td>Over-determined Constraints and Loss of Rigid Body Modes</td>
<td>27</td>
</tr>
<tr>
<td>2.</td>
<td>GENERAL COMMANDS</td>
<td>28</td>
</tr>
<tr>
<td>2.1</td>
<td>Input deck format</td>
<td>28</td>
</tr>
<tr>
<td>2.2</td>
<td>Input mesh geometry file</td>
<td>31</td>
</tr>
<tr>
<td>2.2.1</td>
<td>Geometry_file</td>
<td>31</td>
</tr>
<tr>
<td>2.2.2</td>
<td>Exodus Database Naming Conventions</td>
<td>32</td>
</tr>
<tr>
<td>2.2.3</td>
<td>ASSEMBLY section</td>
<td>33</td>
</tr>
<tr>
<td>2.2.4</td>
<td>Exodus Naming Limitations</td>
<td>33</td>
</tr>
<tr>
<td>2.2.5</td>
<td>Additional Comments About Output</td>
<td>34</td>
</tr>
<tr>
<td>2.3</td>
<td>Parameters</td>
<td>35</td>
</tr>
<tr>
<td>2.4</td>
<td>Solution Options</td>
<td>46</td>
</tr>
<tr>
<td>2.4.1</td>
<td>Flush</td>
<td>47</td>
</tr>
</tbody>
</table>
4.1.2. Orthotropic ........................................... 210
4.1.3. Anisotropic ............................................ 211
4.1.4. Lamé Material ......................................... 211

4.2. Acoustic ...................................................... 213

4.3. Linear Viscoelastic ............................................. 214

4.3.1. Limitations of Viscoelastic Use ......................... 216

4.4. Properties ..................................................... 217

4.4.1. Density ............................................... 217
4.4.2. High Cycle Fatigue ..................................... 217
4.4.3. S-N curve Definitions ................................ 218
4.4.4. S-N Curve Units ....................................... 219
4.4.5. Typical Material Data for Fatigue .................... 220
4.4.6. Temperature dependence ................................ 222
4.4.7. Spatially Variant Material Properties ............... 223
4.4.8. Specific Heat .......................................... 224
4.4.9. Frequency dependence ................................ 225

4.5. Piezoelectric Material ........................................... 226

4.6. Dielectric Material ............................................. 227

4.7. Block ......................................................... 228

4.7.1. Block Parameters ...................................... 228
4.7.2. General Block Parameters ............................... 231

4.8. Damping ...................................................... 235

4.8.1. Nonlinear transient solutions with damping .......... 238
4.8.2. Nonlinear Distributed Damping ....................... 238

5. ELEMENTS ......................................................... 241

5.1. Hex8 ......................................................... 241
5.2. Hex20 ........................................................ 242
5.3. Wedge6 ....................................................... 242
5.4. Wedge15 ...................................................... 242
5.5. Tet4 ......................................................... 243
5.6. Tet10 ........................................................ 243

5.7. Two-Dimensional Shell and Membrane Elements .............. 243

5.7.1. QuadT, Quad8T, and Tria6 ............................. 243
5.7.2. QuadM ............................................... 245
5.7.3. Nquad/Ntria .......................................... 247
5.7.4. TriaShell .............................................. 249
5.7.5. Tria3 ................................................. 249
5.7.6. Stiffness Scaling ...................................... 250
5.7.7. Shell Coordinate Systems ............................... 250
5.7.8. Layered Shells ......................................... 252
5.7.9. Offset Shells ........................................... 254
5.7.10. Spatially Dependent Properties via Exodus Attributes . 255

5.8. HexShell ...................................................... 256
5.9. Beam2 ........................................................ 261
5.10. Nbeam ....................................................... 264
5.11. TiBeam ....................................................... 268
5.12. Truss ......................................................... 268
5.13. Ftruss ........................................................ 268
5.14. ConMass ...................................................... 270
5.15. Spring ........................................................ 271
  5.15.1. Spring Parameter Values ................................ 272
5.16. RSpring ....................................................... 272
5.17. Spring3 - nonlinear cubic spring .................................. 273
5.18. Dashpot ...................................................... 274
5.19. SpringDashpot................................................. 275
5.20. Hys .......................................................... 277
5.21. Joint2G ..................................................... 279
  5.21.1. Specification ........................................... 279
  5.21.2. Constitutive Behavior .................................. 280
5.22. Line Weld ..................................................... 288
5.23. Gap element ................................................... 291
5.24. Gap2D ....................................................... 294
5.25. GasDmp ...................................................... 295
5.26. Nmount ....................................................... 296
5.27. Rrod ......................................................... 298
5.28. Rbar ......................................................... 299
  5.28.1. Interaction of Rbars .................................... 300
5.29. RBE2 ........................................................ 300
5.30. RBE3 ........................................................ 300
5.31. Superelement .................................................. 302
5.32. Dead ......................................................... 308
5.33. Rigidset ....................................................... 308
5.34. Rrodset ....................................................... 310
5.35. Tied Joint ..................................................... 311
  5.35.1. Input Specification ..................................... 311
6. BOUNDARY CONDITIONS AND INITIAL CONDITIONS ................. 318
6.1. Boundary conditions ............................................ 318
  6.1.1. Prescribed Displacements and Pressures ..................... 321
  6.1.2. Prescribed Voltage ....................................... 322
  6.1.3. Prescribed Accelerations .................................. 322
  6.1.4. Prescribed Frequency-Varying Displacements .............. 324
  6.1.5. Node_List_File ....................................... 324
  6.1.6. Nonreflecting Boundaries ................................ 325
  6.1.7. Impedance Boundary Conditions ........................... 326
  6.1.8. Slosh ................................................. 326
  6.1.9. Infinite Elements ........................................ 327
  6.1.10. Perfectly Matched Layers ................................ 329
  6.1.11. Periodic Boundary Conditions ............................ 331
  6.1.12. Usage Guidelines ...................................... 332
6.2. Exodus Mesh Boundary Condition Input ............................................. 333
  6.2.1. SpatialBC Functions ......................................................... 334
  6.2.2. Input an Acoustic Point Source from a Volume ............................ 335
  6.2.3. Input an Acoustic Point Source from a Node Set ............................ 336
  6.2.4. ReadSurface functions .................................................... 337
  6.2.5. ExodusRead functions ...................................................... 338
  6.2.6. In Core Transfer Functions ............................................... 338

6.3. Loads ......................................................................................... 340
  6.3.1. Load .................................................................................... 341
  6.3.2. Scale Factors for the Load .................................................... 341
  6.3.3. Sideset Loading ..................................................................... 342
  6.3.4. Spatial Variation .................................................................... 344
  6.3.5. Required Section .................................................................... 344
  6.3.6. Follower Stiffness .................................................................. 344
  6.3.7. Acoustic Loads ...................................................................... 345
  6.3.8. Thermal Loads ...................................................................... 348
  6.3.9. Energy Deposition Input and Loads .......................................... 352
  6.3.10. Consistent Loads ............................................................... 352
  6.3.11. Pressure_Z .......................................................................... 352
  6.3.12. Surface Charge .................................................................... 353
  6.3.13. Static Loads ........................................................................ 353
  6.3.14. Time Varying Loads ............................................................ 354
  6.3.15. Random Pressure Loads ....................................................... 354
  6.3.16. Frequency Dependent Loads ................................................ 358
  6.3.17. Modal Force Loading ........................................................... 358
  6.3.18. Rotational frames ............................................................... 359
  6.3.19. Rigid Body Filter for Input ................................................. 363
  6.3.20. RanLoads ........................................................................... 364

6.4. Initial Conditions ........................................................................ 366
  6.4.1. Reading Initial Conditions from the Mesh File ............................ 367
  6.4.2. Setting Initial Conditions in the Input Deck ............................... 367

6.5. Use cases for initial acceleration .................................................... 368

7. OUTPUTS ...................................................................................... 370
  7.1. Exodus ...................................................................................... 370
  7.1.1. Surface Projection of Element Variables ..................................... 371
  7.1.2. Database Name ...................................................................... 373
  7.1.3. Properties ............................................................................. 373
  7.1.4. Maa ...................................................................................... 374
  7.1.5. Material ............................................................................... 374
  7.1.6. material_direction ............................................................... 374
  7.1.7. Kaa ...................................................................................... 374
  7.1.8. Faa ...................................................................................... 375
  7.1.9. MLumped ............................................................................. 375
  7.1.10. MPhi .................................................................................... 375
  7.1.11. ElemEigChecks ................................................................. 376
7.1.12. ElemQualchecks .......................................376
7.1.13. Displacement ...........................................377
7.1.14. Velocity...............................................377
7.1.15. Acceleration ..........................................377
7.1.16. Strain ................................................378
7.1.17. Strain = GP ...........................................379
7.1.18. Stress ................................................379
7.1.19. Principal Stresses .....................................380
7.1.20. von Mises stress .......................................380
7.1.21. Signed von Mises Stress ...............................381
7.1.22. Rainflow Cycle Counting .............................381
7.1.23. Fatigue Damage .......................................381
7.1.24. Stress = GP ..........................................382
7.1.25. Vrms ................................................384
7.1.26. Rotational_displacement ................................384
7.1.27. Rotational_acceleration ...............................384
7.1.28. Energy ...............................................384
7.1.29. GEnergies .............................................384
7.1.30. Globals ...............................................385
7.1.31. Block_Energies .......................................385
7.1.32. Mesh_Error ..........................................386
7.1.33. MFile ................................................386
7.1.34. Force ...............................................388
7.1.35. Constraint force ......................................388
7.1.36. Reaction Force .......................................388
7.1.37. Right-hand side ......................................389
7.1.38. EForce ...............................................389
7.1.39. Line_Weld ............................................391
7.1.40. Relative_Disp ........................................391
7.1.41. Residuals .............................................392
7.1.42. TIndex ...............................................393
7.1.43. EOrient ..............................................394
7.1.44. Pressure ..............................................394
7.1.45. NPressure .............................................395
7.1.46. APressure .............................................395
7.1.47. APartVel .............................................395
7.1.48. Constraint_Info .......................................395
7.1.49. Statistics ..............................................396
7.1.50. KDiag ................................................397
7.1.51. ADiag ................................................398
7.1.52. ddamout ..............................................398
7.1.53. Temperature ..........................................400
7.1.54. Voltage and Charge ....................................400

7.2. User Output ...................................................400
7.2.1. Block Statistics ........................................400

vii
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>7.2.2. Closest Distance</td>
<td>401</td>
</tr>
<tr>
<td>7.2.3. Derived Variables</td>
<td>402</td>
</tr>
<tr>
<td>7.3. History</td>
<td>403</td>
</tr>
<tr>
<td>7.3.1. Global History Output Near a Location</td>
<td>405</td>
</tr>
<tr>
<td>7.4. Frequency</td>
<td>407</td>
</tr>
<tr>
<td>7.5. Linesample</td>
<td>408</td>
</tr>
<tr>
<td>7.6. Stresses and Strains</td>
<td>409</td>
</tr>
<tr>
<td>7.6.1. Stress/Strain Truth Table</td>
<td>410</td>
</tr>
<tr>
<td>7.6.2. Solid Elements</td>
<td>412</td>
</tr>
<tr>
<td>7.6.3. Shell Elements</td>
<td>412</td>
</tr>
<tr>
<td>7.6.4. Beam Elements</td>
<td>413</td>
</tr>
<tr>
<td>7.7. Echo</td>
<td>414</td>
</tr>
<tr>
<td>7.7.1. Mass Properties</td>
<td>416</td>
</tr>
<tr>
<td>7.7.2. Multipoint constraints</td>
<td>416</td>
</tr>
<tr>
<td>7.7.3. ModalVars</td>
<td>417</td>
</tr>
<tr>
<td>7.7.4. Subdomains</td>
<td>417</td>
</tr>
<tr>
<td>7.7.5. Memusage</td>
<td>418</td>
</tr>
<tr>
<td>8. CONTACT</td>
<td>419</td>
</tr>
<tr>
<td>8.1. Tied Surfaces</td>
<td>419</td>
</tr>
<tr>
<td>8.1.1. Contact Normal Vectors</td>
<td>420</td>
</tr>
<tr>
<td>8.1.2. Mortar Methods</td>
<td>422</td>
</tr>
<tr>
<td>8.1.3. Node to Face</td>
<td>422</td>
</tr>
<tr>
<td>8.2. Contact Definition</td>
<td>425</td>
</tr>
<tr>
<td>8.2.1. Defining Contact Surfaces</td>
<td>426</td>
</tr>
<tr>
<td>8.2.2. Setting up Contact Interactions</td>
<td>428</td>
</tr>
<tr>
<td>8.2.3. Gap removal and element quality</td>
<td>430</td>
</tr>
<tr>
<td>8.2.4. Examples</td>
<td>431</td>
</tr>
<tr>
<td>8.2.5. Notes and Usage Guidelines</td>
<td>432</td>
</tr>
<tr>
<td>8.2.6. Differences Between SM and SD Defaults</td>
<td>433</td>
</tr>
<tr>
<td>8.3. Lofted Surfaces and Gap Removal</td>
<td>433</td>
</tr>
<tr>
<td>8.3.1. Example</td>
<td>434</td>
</tr>
<tr>
<td>8.3.2. Projection Approach</td>
<td>434</td>
</tr>
<tr>
<td>8.4. Moving MPCs</td>
<td>436</td>
</tr>
<tr>
<td>9. EXAMPLE INPUT FILES</td>
<td>438</td>
</tr>
<tr>
<td>9.1. Eigenvalue problem</td>
<td>438</td>
</tr>
<tr>
<td>9.2. Anisotropic Material</td>
<td>439</td>
</tr>
<tr>
<td>9.3. Multiple materials</td>
<td>440</td>
</tr>
<tr>
<td>9.4. Modaltransient</td>
<td>442</td>
</tr>
<tr>
<td>9.5. ModalFrf</td>
<td>444</td>
</tr>
<tr>
<td>9.6. Directfref</td>
<td>445</td>
</tr>
<tr>
<td>9.7. Statics</td>
<td>447</td>
</tr>
</tbody>
</table>

**Bibliography**

449

**Index**

455
LIST OF FIGURES

Figure 1-1. Single Spring element. .......................... 26
Figure 1-2. Truss Decomposition Issues. .................. 26
Figure 2-3. Example MFile Format Results. ............... 44
Figure 2-4. Notes on Eigen Restart. ......................... 51
Figure 2-5. Eigen Sensitivity Example Data ................. 75
Figure 2-6. Semi-Analytic Methods for Sensitivity Analysis. 77
Figure 2-7. Coordinate system definition vectors: note that $\mathbf{XZ}$ POINT determines the $\mathbf{X}$ axis, but need not lie along it, which is the case depicted in this figure. 81
Figure 2-8. Coordinate System Examples .................... 82
Figure 2-9. Conical Coordinate System Definition at $\mathbf{XZ}$ plane .................. 82
Figure 2-10. Ellipsoidal Coordinate System Using axis_stretching=(2.0, 1.1, 1.0). $\mathbf{r}$ is red, $\mathbf{s}$ is green, and $\mathbf{t}$ is blue. .................................. 83
Figure 2-11. Coordinate system behavior near the $\mathbf{Z}$ axis. $\mathbf{r}$ is red, $\mathbf{s}$ is green, and $\mathbf{t}$ is blue. ............................................................ 85
Figure 2-12. Full model with different coordinate system uses ........................ 86
Figure 2-13. Linear function "ignored_point". .............. 88
Figure 2-14. Linear function "extrapolation". ................ 89
Figure 2-15. Linear function #5. "multiple_fun". ........... 89
Figure 2-16. RandomLib Temporal Interpolation. ........... 95
Figure 2-17. Spherical Wave Geometry ....................... 106
Figure 2-18. Fluid-Structure Interaction (FSI) Infrastructure. 109
Figure 2-19. Illustration of first crossing blended function. .................... 110
Figure 2-20. Illustration of Nth crossing blended functions. ............... 111
Figure 3-21. Craig-Bampton Reduction ......................... 132
Figure 3-22. Superposition Data Flow Diagram. ............ 189
Figure 3-23. Waterline Coordinate Definition ................ 205
Figure 3-24. Net Force vs depth for a Rigid Body ............ 206
Figure 4-25. S-N Curve for Steel Sheet ....................... 221
Figure 5-26. QuadT Element. ................................. 244
Figure 5-27. Quad8T Element. ................................. 244
Figure 5-28. Tria6 Element. .................................... 245
Figure 5-29. Function for nquad_eps_max. .................... 248
Figure 5-30. Projection of global coordinate system to shell. ............... 251
Figure 5-31. Rotation of global coordinate system about axis prior to projection to shell. 251
Figure 5-32. Rotation of local system about normal after to projection to shell ............... 252
Figure 5-33. Stacking arrangement for a multi layer shell element. .................. 253
Figure 5-34. Offset examples for shell of thickness 0.1. ............... 255
Figure 5-35. Beam Orientation and Local Coordinate System .................. 262
Figure 5-36. Beam Offset and Local Coordinate System. ........................................ 263
Figure 5-37. Nbeam Orientation, Offset and Local Coordinate System. .................. 265
Figure 5-38. Hys element parameters. ............................................................... 278
Figure 5-39. Iwan Constitutive Model. .............................................................. 282
Figure 5-40. Hysteresis Microslip Variation with $\beta$ ...................................... 283
Figure 5-41. Hysteresis Macroslip Variation with $\beta$ ...................................... 284
Figure 5-42. Reduced Iwan Load Displacement Curve. ...................................... 285
Figure 5-43. Eplas Model. .................................................................................. 286
Figure 5-44. Line weld definitions for attaching the purple and cyan shells. The line
weld follows the red beam, which is contiguously meshed with the purple shell. ...................... 288
Figure 5-45. Line weld Joint2G connections. The Joint2G elements are initially zero-
length, and the green and red nodes coincident, but they are depicted
with a finite deformation to delineate the individual components. ......................... 289
Figure 5-46. Gap element Force-Deflection Curve. ........................................ 292
Figure 5-47. Mass bouncing off a Gap ............................................................... 293
Figure 5-48. Gap2D force diagram. .................................................................... 295
Figure 5-49. Nmount Orientation ....................................................................... 298
Figure 5-50. Rigidset/TiedJoint Centernode Connection ................................. 309
Figure 5-51. Rrodset Constraints ....................................................................... 311
Figure 5-52. Tied Joint Geometry ...................................................................... 311
Figure 5-53. Tied Joint Surface Normal Definition ........................................ 314
Figure 5-54. Construction of tied joint with side=average. ............................. 315
Figure 5-55. Construction of tied joint with side=rigid. .................................. 315
Figure 5-56. Construction of tied joint with side=Rrod. .................................. 315
Figure 5-57. Example of Interpolation of Exodus Data to Analysis Steps. ........... 334
Figure 6-58. Coordinate Frame Projection for Traction .................................. 344
Figure 6-59. RandomPressure Loading Approximations. ............................... 355
Figure 7-60. Convergence of maximum stress at element centroids and surfaces. .. 372
Figure 7-61. Example KDiag output. ................................................................. 398
Figure 7-62. Tria3 Stress Recovery ................................................................. 413
Figure 8-63. Shell Normal in Contact or Tied Interactions. ............................. 421
Figure 8-64. Search Tolerance definition ............................................................ 423
Figure 8-65. Normal Definitions on Faceted Geometry .................................... 423
Figure 8-66. Smoothing Parameters for Surface Normal Vectors ..................... 424
Figure 8-67. Lofted Constraint Example. ........................................................... 434
LIST OF TABLES

Table 1-1. Command Line Options. .......................................... 17
Table 1-2. Determining Number Of Processors Needed. ......................... 18
Table 1-3. Elements using other elements condition number. ..................... 25
Table 2-4. Comment String Options. ......................................... 29
Table 2-5. Available keywords in the Parameters section. ......................... 36
Table 2-6. Available keywords in the Parameters section related to code coupling
and handoff.  ............................................................. 37
Table 2-7. Developer keywords in the Parameters section. ........................... 37
Table 2-8. Some useful combinations of units. ................................ 37
Table 2-9. Eigenvector Normalization Methods. ....................................... 43
Table 2-10. Sierra/SD Solution Options. ....................................... 47
Table 2-11. Supported restart capabilities for transient integrators in Sierra/SD. . 51
Table 2-12. Restart File Format and Names. ....................................... 52
Table 2-13. GDSW Section Options. (Basic) ................................... 56
Table 2-14. GDSW Section Options 1. (Advanced) .............................. 57
Table 2-15. GDSW Section Options 2. (Advanced) .............................. 58
Table 2-16. GDSW Section Options. (Supplemental Output) ....................... 64
Table 2-17. GDSW Section Options (Advanced). ................................ 70
Table 2-18. GDSW Section Print Options....................................... 71
Table 2-19. GDSW Section Options (Helmholtz). ................................ 72
Table 2-20. GDSW Section Options (Solvers). ................................... 72
Table 2-21. Sensitivity Analysis Keywords. ..................................... 73
Table 2-22. Sensitivity Analysis Solution Type Availability. ....................... 77
Table 2-23. Coordinate Names for history files................................... 84
Table 2-24. SamplingRandom function parameters. .............................. 92
Table 2-25. RandomLib function parameters. ................................... 93
Table 2-26. Predefined Analytic Input Variables. ................................101
Table 2-27. Planar Step Wave Parameters. ....................................105
Table 2-28. Spherical Wave Parameters.........................................105
Table 2-29. Undex Load Parameters ...........................................107
Table 2-30. Blended Function Parameters.......................................109
Table 2-31. Table Section Options. ............................................115
Table 2-32. General MPC commands...........................................117
Table 2-33. MPC Equation lines...............................................117
Table 3-34. Eigen Solution Types. .............................................119
Table 3-35. Modal Solution Types. ............................................119
Table 3-36. Direct Solution Types. ..........................................120
Table 3-37. Preprocessing and Postprocessing Solution Types. .................120
Table 3-38. Multicase Options ................................................. 122
Table 3-39.Checkout Solution Case Parameters. ................................123
Table 3-40. CJdamp Solution Case Parameters. ................................. 124
Table 3-41. Craig-Bampton reduction Solution Case Parameters. .............. 125
Table 3-42. CBModel Parameters. ............................................ 129
Table 3-43. Data output for Craig-Bampton Reduction. ........................ 131
Table 3-44. preddam Solution Case Parameters. ............................... 134
Table 3-45. DDAM Solution Case Parameters. .................................. 135
Table 3-46. DirectFRF Solution Case Parameters. ............................... 138
Table 3-47. Dump Solution Case Parameters. ................................... 139
Table 3-48. Eigen Solution Case Parameters. ................................... 140
Table 3-49. AEigen Solution Case Parameters. .................................. 145
Table 3-50. AEigen Verbosity Table. ............................................ 146
Table 3-51. Largest_Ev Solution Case Parameters. ............................. 147
Table 3-52. Fatigue Solution Case Parameters. .................................. 147
Table 3-53. Buckling Solution Case Parameters. ................................. 150
Table 3-54. ModalFilter Solution Case Parameters. ............................. 151
Table 3-55. Modal Filter Keywords. ............................................ 153
Table 3-56. Modal Participation Factor Solution Case Parameters. .......... 153
Table 3-57. MPF Summary data ................................................ 155
Table 3-58. ModalFrf Solution Case Parameters. ............................... 157
Table 3-59. ModalRanVib Solution Case Parameters. ........................... 162
Table 3-60. ModalRanVib Output to Exodus File ............................... 165
Table 3-61. ModalShock Solution Case Parameters. ............................ 167
Table 3-62. ModalTransient Solution Case Parameters. ......................... 168
Table 3-63. QEVP Solution Case Parameters. .................................. 170
Table 3-64. A 2005 Comparison of Quadratic Eigenvalue Problem Methods. Although Ceigen and Anasazi have changed substantially since 2005, this table has not been updated to reflect those changes. ....................... 172
Table 3-65. Options for qevp Anasazi Solutions. ................................ 172
Table 3-66. ceigen Solution Case Parameters. .................................. 173
Table 3-67. Ceigen Tests. ...................................................... 174
Table 3-68. SA_Eigen Options. .................................................. 176
Table 3-69. Verification Summary for SA_Eigen. ............................... 177
Table 3-70. Projection_Eigen Options. ........................................ 178
Table 3-71. NLStatics Solution Case Parameters. ............................... 179
Table 3-72. NLTransient Solution Case Parameters. ........................... 181
Table 3-73. Receive_Sierra_Data Solution Case Parameters. .................. 182
Table 3-74. Nodal data used in receive_sierra_data. ........................... 184
Table 3-75. Element data used in receive_sierra_data. Only the Neo-Hookean and hyperfoam Lamé models are supported; see Section 4.1.4. ................. 185
Table 3-76. Data Transferred in receive_sierra_data specific to orthotropic_layer materials. Some of this data is transferred automatically. Other data requires the from_transfer keyword in the material definition (denoted by an asterisk in the Effect column). ................................. 186
Table 6-121. Load Specification Keywords .......................................................... 343
Table 6-122. Random Pressure Inputs ................................................................. 356
Table 6-123. Rotating Frame Parameters ............................................................. 360
Table 7-124. OUTPUT Section Options ............................................................... 371
Table 7-125. Exodus Property Output Options ..................................................... 373
Table 7-126. Hex20 Gauss Point Locations ......................................................... 383
Table 7-127. Data Files Written Using the MFile Option ....................................... 387
Table 7-128. Typical Output ................................................................................. 390
Table 7-129. FRF Output ...................................................................................... 390
Table 7-130. ModalRanVib Exodus Output .......................................................... 390
Table 7-131. ModalRanVib Frequency Output ..................................................... 391
Table 7-132. Typical Output ................................................................................. 392
Table 7-133. ModalRanVib Frequency Output ..................................................... 392
Table 7-134. ModalRanVib Exodus Output .......................................................... 392
Table 7-135. TIndex parameters ......................................................................... 393
Table 7-136. Element Orientation Outputs ......................................................... 394
Table 7-137. Element Orientation Interpretation ............................................... 394
Table 7-138. Supported Statistical Data types ..................................................... 397
Table 7-139. Selected Dynamic Matrix Definitions ............................................. 398
Table 7-140. Variables that are output from ddam analysis ................................. 399
Table 7-141. Frequency Value Specification Methods ......................................... 408
Table 7-142. Element Stress Truth Table ............................................................. 411
Table 7-143. Echo Section Options .................................................................... 415
Table 8-144. Tied Surface Parameters ................................................................. 424
Table 8-145. Coordinates of Face (red) and Nodes (blue) .................................... 434
Table 8-146. Conventional Constraint Equations ............................................... 435
Acknowledgments

The Sierra/SD software package is the collective effort of many individuals and teams. A core Sandia National Laboratories based Sierra/SD development team is responsible for maintenance of documentation and support of code capabilities. This team includes Dagny Beale, Gregory Bunting, Mark Chen, Nathan Crane, David Day, Clark Dohrmann, Sidharth Joshi, Payton Lindsay, Julia Plews, Brian Stevens, and Johnathan Vo.

The Sierra/SD team also works closely with the Sierra Inverse and Plato teams to jointly enhance and maintain several capabilities. This includes contributions from Wilkins Aquino, Brett Clark, Murthy Guddati, Sean Hardesty, Chandler Smith, Benjamin Treweek, and Timothy Walsh.

The Sierra/SD team works closely with other Sierra teams on core libraries and shared tools. This includes the DevOps, Sierra Toolkit, Solid Mechanics, Fluid Thermal Teams. Additionally, analysts regularly provide code capabilities as well as help review and verify code capabilities and documentation. Other individuals not already mentioned directly contributing to the Sierra/SD documentation or code base during the last year include Ryan Alberdi, Samuel Browne, Victor Brunini, Mike Glass, Cameron McCormick, Scott Miller, Tolu Okusanya, Heather Pacella, Kendall Pierson, Clay Sanders, Tim Shelton, Greg Sjaardema, Jeremy Trageser, Mike Tupek, and Alan Williams.

Historically dozens of other Sandia staff, students, and external collaborators have also contributed to the Sierra/SD product its documentation.

Many other individuals groups have contributed either directly or indirectly to the success of the Sierra/SD product. These include but are not limited to;

- Garth Reese implemented the original Sierra/SD code base. He served as principal investigator and product owner for Sierra/SD for over twenty years. His efforts and contributions led to much of the current success of Sierra/SD.

- The ASC program at the DOE which funded the initial Sierra/SD (Salinas) development as well as the ASC program which still provides the bulk of ongoing development support.

- Line managers at Sandia Labs who supported this effort. Special recognition is extended to David Martinez who helped establish the effort.

- Charbel Farhat and the University of Colorado at Boulder. They have provided incredible support in the area of finite elements, and especially in development of linear solvers.

- Carlos Felippa of U. Colorado at Boulder. His consultation has been invaluable, and includes the summer of 2001 where he visited at Sandia and developed the HexShell element for us.

- Danny Sorensen, Rich Lehoucq and other developers of ARPACK, which is used for eigenvalue problems.
• Esmond Ng who wrote *sparspak* for us. This sparse solver package is responsible for much of the performance in *Sierra/SD* linear solvers.

• The *metis* team at the university of Minnesota. *Metis* is an important part of the graph partitioning schemes used by several of our linear solvers. These are copyright 1997 from the University of Minnesota.

• Padma Raghaven for development of a parallel direct solver that is a part of the linear solvers.

• The developers of the SuperLU Dist parallel sparse direct linear solver. It is used through GDSW for a variety of problems.

• Leszek Demkowicz at the University of Texas at Austin who provided the HP3D library and has worked with the *Sierra/SD* team on several initiatives. The HP3D library is used to calculate shape functions for higher order elements.

This work was supported by the Laboratory Directed Research and Development (LDRD) program.
Release Notes

0.1. Release 5.6

New or Improved Features

- **vonMises** output (section 7.1.20) has been standardized.
  Requesting **vonMises** will now output the (maximum) von Mises stress as it always has, but now it will also output the von Mises stress at each layer for surface elements (section 7.6.3), and each stress recovery point for beam elements (section 7.6.4).

- Analytic functions can now be used in variable initialization for nodal field variables.
  This allows mathematical operations to be performed on variables as they are read in, such as scaling, or analytically defined preload fields. See input 3.9 for an example use case.

- Additional stress output metrics for DDAM solutions have been enabled. These include stress output at each mode, hydrostatic stress for 3D elements, and maximum shear stress for 3D elements. NRL-sums of each of the above is also provided. See section 7.1.52 for more information.

- Closest distance calculations (section 7.2.2) have been extended to support higher-order elements.

Deprecated Features

- The **mpmd_transfer_version** parameter (table 2-6) option “old” has been deprecated in favor of “NSC” to be more descriptive.

Bug Fixes

- **vonMises** output (section 7.1.20) has been standardized.
  Previously, the von Mises stress reported from requesting **vonMises** output could potentially be different than what would be reported if the full stress tensor was requested (**stress**). Specifically, this was the case with volumetric elements with thermal stress.
  
  The behavior with beam elements containing stress recovery points has also been slightly modified to be in line with the existing documentation and intent.
  Previously, the default "von Mises" quantity in the absence of stress recovery points (the absolute value of the axial stress), was included in the maximum von Mises calculation even when stress recovery points were present. This is no longer the case. See section 7.6.4 for more details.
• Minor bug fixes and robustness improvements of the following capabilities:
  – contact (8.2).
  – mesh_error output (section 7.1.32)
  – tsr_preload solution case (section 3.31)
  – Applied pressure output (section 7.1.44) for elements with multiple sides in a sideset

**Documentation Updates**

• Equations have been added that represent the mesh_error output (section 7.1.32).
• The equation for the NRL-sum computation has been added to the ddam output section (7.1.52).
• Several coordinate system figures were updated for correctness and avoid potential confusion (figures 2-8 and 2-11 and input 2.3).
• The inverse problem documentation has been moved into a stand-alone manual.
• An example of how to use matlab output (mfile) has been added (section 7.1.33.1).
• Piezoelectric and dielectric material examples (inputs 4.5 and 4.6) have been updated for correctness.
• The receive_sierra_data documentation (section 3.24) has been updated for correctness and style.

**0.2. Release 5.4**

**New or Improved Features**

The closest distance user output functionality has been expanded to also be able to compute a nodal field which is the closest distance at each node. See Section 7.2.2.

The TSR Preload verification document was added to the Verification Manual.

New commands were added to allow scaling and shifting of XY functions. See Section 2.8.1.

New user output options for computing the max, min, average, or standard deviation of element variables. See Section 7.2.3.

**Sierra/SD** now works consistently with the sierra script -pre option. This option will automatically detect the mesh file used by the Sierra/SD analysis and decompose it in parallel if this has not yet been done.
It is now possible to provide alternative names for exodus fields (such as displacement, stress, etc.) when importing data from a previous solution. See Section 3.24.

Material properties can now be defined as general functions of element or nodal variables. See Section 4.4.7.

**Deprecated Features**

The *Sierra/SD* syntax for including files in an input file, such as `#include material.inp` or `include material` is deprecated, in favor of the Aprepro syntax used by other Sierra codes. For example `{include(material.inp)}`.

Other features that have been newly deprecated or previously deprecated and removed:

- Constraint output syntax `slave_constraint_info`
- Contact `master` and `slave` options. Use `side b` and `side a` instead.
- Solution case `QmodalFrf`
- Echo `none`, `mpc_nonorthog` options
- GDSW parameter `atLeastOneIteration`
- RBE3 option `method old|new`.
- non-aprepro input deck file includes
- Option `thickness varies by element`
- File `sierra_input_file` command
- Old Coordinate section syntax. Use `begin rectangular coordinate system`, `begin cylindrical coordinate system` etc. instead.
- Material `name` option.

**Bug Fixes**

Maintenance and development prioritizes use cases. Input is implemented using Functions. Some Functions can use data from either the input deck or the Exodus mesh file. These Functions can interpolate, extrapolate and integrate (once or twice).

Changes were made so that the Linear, 1D Table, SM and Spatial BC Functions have consistent behavior. Spatial BC Functions include Random Lib Functions. SM refers to Sierra SM Piecewise Linear Functions. In several cases, the behavior changed from possibly inconsistent to consistent. The Functions are now unit tested and documented.

Table Functions are linear Functions optimized for constant time step size. A Table Function dimension can be 1,2,3, or 4. One dimensional Table Functions have high priority. The handling of edge cases for 1D Table Functions improved. It is possible, albeit unlikely,
that simulation results will change. Higher dimensional Table Functions have the old behavior. They are complex. Their handling of edge cases is not known by current developers. The priority of 2D Table Functions requires clarification. 3D and 4D Table Functions have low priority.

The behavior of SpatialBC and RandomLib Functions was improved. Previously, these Functions were incorrect in a way that would have artificially forced the value to be zero at $t = 0$, and also would have shifted all the values as if the origin was at $t = 0$. Regression test results were changed (see sha a743640134329a08ddeaab8ab69 in the Test repository). Spatial BC Functions have low priority.

Contact tolerances for shell-to-shell contact have been improved. Previously tolerances only used half the shell thickness in the search tolerances and could miss contact.

The contact cutoff which is used to exclude nodes from contact based on field variable values now only considers the field value on the node of the node-on-face constraint. This is more consistent with how Sierra/SM writes such variables, the handoff of which is the primary use case for this capability.

Fixed a bug where the attribute order could be scrambled in the output results mesh. This would cause issues if that output mesh was used as the input of a subsequent SD analysis.

Now using a solver_options for global GDSW parameters will override the any global options specified. Previously the global options would always be used.

0.3. Release 5.2

Sierra/SD is considered production ready for the ATS-2 platform which utilizes GPUs. Most commonly used solution cases such as Eigen and transient show substantial speedup using this platform’s GPUs. Detailed documentation on how to run Sierra/SD on ATS-2 can be found in the online wiki “Running Sierra/SD on ATS-2”. Contact sierra-help@sandia.gov to locate this information.

A capability has been implemented for in-core transfer of loads from a simultaneously running SPARC analysis. See Section 6.2.6.

Several improvements have been made to automatically handle potentially over-determined constraints produced by self contact or contact between surfaces that also share nodes.

Thanks to Dagny Beale for reviewing the Coordinates section (2.8).

User mounts may now be incorporated into Sierra/SD at runtime.

The nskip parameter can now be set independently in the linesample input.

Analytic functions can now be spatially dependent functions of coordinates or other variables. See Section 2.8.9 for more details.
Rain flow cycle counting of stress cycles and fatigue estimation from these cycles is now available in transient solution. See Section 7.1.22. Additionally an example of this capability has been added to the “How To” manual.

Sierra/SD will disallow use of certain reserved words for thing like block names as they would cause ambiguity in parsing. For example a block named “end”. See Section 2.2.4 for more details on how to determine reserved names and also how to override this error handling for legacy models.

Several improvements have been made to the eigen value sensitivity output. Previously the column labels for output did not accurately describe what these outputs represented.

Many improvements have been made to robustness for output and history files. This was accomplished making it possible to request every output in any solution case. If additional issues are seen, then please report them to sierra-help@sandia.gov.

A new QUIVER video is available on solver debugging. Additionally, training videos recorded from a several day in-person class are available on the Computational Simulation Knowledge Base website. Contact sierra-help@sandia.gov to locate them.

The input deck is now by default echoed to the Sierra/SD log file. This echoing can be turned off by specifying “input off” in the echo block.

The default solver shift for eigen solution has been changed from “-1” to “-1.0e+6”. Ideally the shift should be set to the negative of the first eigen value of for the system (the square of two pi times the first flexible frequency.) The new default is much more likely to be in the right ballpark for most engineering systems. However, consideration still needs to be given to shift for hard to solve problems. Additionally, the default maximum size of type 1 constraints was raised from 100 terms to 250 terms to improve solver robustness.

The input syntax to create exodus assemblies within the Sierra/SD input deck was renamed from MESH GROUP to BEGIN ASSEMBLY for greater consistency with other Sierra applications.

The TriaShell mass matrix calculation has been improved. Previously it was missing the inertia terms relating to the shell thickness. In practice the effect of these terms is usually slight as shell structures should be relatively thin compared to element size.

**Deprecated Features**

Features that have been newly deprecated or previously deprecated and removed:

- Material syntax name=string, T_current=real, s_isotropic
- Constraint output syntax slave_constraint_info
- Tied data and contact syntax master, slave
- Joint2g option Shys
- Solution case QmodalFrf
- Solution case Qmodaltransient
- ncfout (output4, op4) utility,
- Solution parameter no_symmetrize_struc_acous (has been replaced by symmetrize_struc_acous)
- RBE3 option method old|new.
- GDSW parameter atLeastOneIteration
- Old Coordinate section syntax. Use begin rectangular coordinate system, begin cylindrical coordinate system etc. instead.
- Echo none, mpc_nonorthog options
- File sierra_input_file command
- Coupling to the deprecated “Aero” application
- Runtime compiled functions (RTC)

0.4. Release 5.00

A new chapter of User’s Manual describes how to run Sierra/SD §1.1 and other fundamental usability issues. Also, Sierra/SD tutorials for new users are available https://snl-wiki.sandia.gov/display/CKB/How-to+articles

An Exodus hierarchical data structure, the Assembly, has been developed to make building models easier. Sierra/SD has full support for Assemblies §2.2.2

The GDSW linear solver is compatible with certain Graphical Processing Units. Users may notice more precise timing information of GDSW. This release includes a full featured interface §2.5 to the SuperLU Dist parallel sparse direct linear solver.

The default GDSW preconditioner is designed for poorly conditioned problems, and is a less efficient option for well conditioned problems. An adaptive block diagonal preconditioner §2.5.6 can be used by setting preconditioner_type to 3. The options §2-15 identify_low_quality_elements and max_element_condition enrich the diagonal preconditioner based on mesh element quality.

Sierra/SD and Sierra/SM use consistent input syntax and share source code for coordinate systems §2.7. Coordinate system definition options have been added such as defining coordinate system by nodesets. Ellipsoidal coordinate systems are now available.

The syntax for spatial boundary condition functions that read data from the mesh is more consistent. It is necessary to specify the variable that the function reads. For consistency with the methods for input from the mesh, the keyword variable_name has been replaced by the keyword exo_var.
Traction input is supported. The **ReadSurface** function §6.2.4 will input spatially dependent and time dependent data from the mesh file. The data is automatically integrated to determine the corresponding forcing function.

Huge improvements in the performance of the Modaltransient §3.20 and ModalFrf §3.17 methods have made it possible to remove the Qmodaltransient and QmodalFrf methods. The output capabilities of Modaltransient and ModalFrf solutions have been extended to reproduce all the capabilities of Qmodaltransient and QmodalFrf.

Another capability to compute all the eigenvalues and eigenvectors for problems with modest numbers of dofs has been extended to handle several important cases such as models with a singular mass matrix and models with multipoint constraints.

The geometric rigid body mode solution case now works for models with any Sierra/SD element including Superelements (c.f. Jira Compsimhd-5610)

Users may use the input **Exodus** mesh to set spatially dependent material properties fields §4.4.7.

Acoustic boundary conditions §6.3.7 include the **point_volume_acceleration**. An acoustics problem can use void elements in the structural mesh to represent air. The void elements move with the solid elements. The flux air volume causes an acoustic pressure wave to propagate out. One approach is to map the volume flux term in the solid to the closest acoustic element. A new approach is to map the volume flux term to the void elements representing the air.

HexShell element §5.8 input syntax is compatible with new coordinate frame syntax. Users must specify the thickness direction of the elements. One way to do this is through a coordinate frame. The frame may also be specified by name instead of the id, which is clearer.

All Sierra/SD elements support stress output. The corresponding issues are Jira Compsimhd-9550 and Nesm-2853.

The lumped mass matrix, **MLumped**, can now be written to the **Exodus** file. Also, the modes shapes from the Eigen solution case multiplied by the mass matrix, **MPhi**, can be written to the **Exodus** file too. The corresponding ticket is Jira Compsimhd-9537.

Principal stress §7.1.19 is output to the **Exodus** file using the **principal_stresses** keyword. Stress output includes a signed von Mises stress §7-124 and §7.1.21. It has been added to support fatigue estimation using Rain Fall cycle counting.

If rigid body filtering is applied using **FilterRbmLoad**, then Force output includes the filtered forces using syntax consistent with Sierra/SM.

Coordinate frames specified in MPCs §2-32 can be defined by either the coordinate id or the coordinate name.

The output for the local or element coordinate frame using EOrient §7.1.43 is consistent for all element types. The local coordinate frame can also be output using §7.1.6. This capability is required for anisotropic materials §4.1.3.
begin contact definition §8.2.3 is more general. Sliding is supported by using the frictionless friction model. The capability provides a linearization of equivalent Sierra/SM syntax. In response to Jira Compsimhd-9517, Contact search failure, gap removal diagnostic information includes an Element Quality table. This applies to tied data too. Note that Sierra/SD reports an element condition number (greater than or equal to one), and Sierra/SM reports the element quality (less than one), which is multiplicative inverse of the condition number.

Sierra/SD has an undocumented interface to a code named Nemo. This release cycle, Sierra/SD added support for two messages. The FailedElementIds message with mpi_tag 1601 reports an integer boolean for every shared element face. It was added for consistency with Sierra/SM. A face is either inactive, 0, or active 1. Sierra/SD reports all elements as active. The SidesetIds message with mpi_tag 1701 reports the integer sideset global ID associated with each coupled face. It is used for diagnostic information between the codes. The IDs are the global ID numbers of the Exodus mesh. Further work is needed on the case of an element face belonging to multiple coupled sidesets.

Deprecated Features

The Sparsepak sparse serial linear solver is deprecated and has been removed. GDSW with the new option krylov_solver = none produces bit-wise equivalent results.

Most deprecated features are ready to be removed immediately after release: material syntax name=string, T_current=real, s_isotropic, constraint syntax slave_constraint_info, master, slave, property line Shys, solution case QmodalFrfr, solution case Qmodaltransient, ncfout (output4, op4) utility, no_symmetrize_struc_acous (to be replaced by symmetrize_struc_acous).

Certain deprecated features may be removed before the next release: RBE3 option method old|new and rtc user functions.

Newly deprecated features include: elmat, old syntax for coordinate frames, constraint method transform.

The SD coordinate system syntax is changing and the old syntax is deprecated. To aid in the syntax transfer, a Python 2 script named convertCoordinateSystems.py is available to convert existing coordinate blocks in an input deck to the new syntax.

$ python convertCoordinateSystems.py FILE_1 FILE_2 FILE_3 ...

Files are overwritten by this script. A message is printed for each file touched. All inputs other than coordinate definitions remain untouched. Comments are preserved everywhere in the file except within the coordinate system definition. It accepts any number of file arguments, so you can chain it together other Linux tools,
Example: In the current directory, convert any *.inp file containing "coordinate":
$ python convertCoordinateSystems.py $(grep -l -i coordinate ./*.inp)
Example: Below the current directory, convert any *.inp file, recursively:
$ python convertCoordinateSystems.py $(find . -name *.inp)

Bug Fixes

Loads are applied correctly at the junction between dead element blocks and element blocks that are not dead for any partition of the mesh into subdomains. Jira Compsimhd-9411 had reported an error in the case in which, for one of the subdomains sharing the junction, all the elements attached to the junction were in dead element blocks. The parallelWithDeadBlocks nightly verification test was added.

An Air Force code verification project found defects with Sierra/SD layered shell elements (Jira Compsimhd-9822). First in a shell element with one layer the layer keyword is redundant and optional. The incorrect behavior, now fixed, was that a single-layer shell with a non-trivial fiber orientation and without the layer keyword, Sierra/SD incorrectly used trivial fiber orientation. A unique attribute value can be specified for each element in an element block in the Exodus file. If specified in the input deck, then an element attribute becomes a block parameter. For block parameters a single value is applied to all the elements in the element block. For layered shell elements thickness and fiber orientation attributes in the Exodus file were considered to be block parameters. One value was applied to all the elements in the element block. This is now fixed. The layerShells test was added to cover these cases.

Jira Compsimhd-9343 does not involve a code defect, but is notable due to reporting incorrect results for a transient simulation. It is important to bear in mind that some solution cases are reliable only if used correctly. In this case users are expected to select sufficiently small step sizes and linear solver tolerances.

Multiple users had trouble using the Seacas tool Aprepro. All issues have been resolved.

Issues using tied data and begin contact definition have continued, leading to usability improvements of begin contact definition. The gap removal QUIVER video https://snl-wiki.sandia.gov/display/CKB/How-to+articles is on tied data and begin contact definition.

0.5. Release 4.58

Deprecated Features

Algorithmic terminology referencing master/slave has been changed. The purpose is both to increase technical clarity and use more inclusive terminology. Most of these changes effect documentation only. One behavior change is that the output variable 'slave_constraint_info' has been deprecated use 'constraint_info' instead. Updates have
also been made to the exodus field names 'constraint_info' output produces. Additionally, master/slave references in the 'contact definition' and 'periodic bc' capabilities have been updated to be consistent with Sierra/SM terminology in the equivalent capabilities.

Previously deprecated features have been removed.

- Output warning level cannot be an integer,
- solution methods blk_eigen and old_transient,
- feti-dp linear solver & option rbm_tol_svd,
- history and frequency output option auto-join,
- patch negative elements (parameter),
- exodus_file & restart_file functions,
- Eigen option nmodes_acoustic,
- spherical_wave option point_source_origin,
- planar_step_wave option C0,
- tiedjoint options edge tolerance and Newton tolerance.
- Boundary conditions smooth angle, smoothing distance, smoothing resolution,
- echo timing.

New Features

Sierra/SD supports coupled electro-mechanical physics in order to simulate the electro-mechanical behavior of piezoelectric materials when subjected to an electric field or mechanical stress. This support includes static, transient, Eigen, and direct frequency response solution methods, piezoelectric and dielectric material models, and electric potential and surface charge boundary conditions. For the inverse solution methods, electro-mechanical physics enables the user to define experimental data in terms of measured voltages.

A new option has been added to contact definition to remove contact constraints based on an exodus mesh variable. The target use case is removal of open frictional constraints based on a solid mechanics preload. See Section 8.2.1 for details.

Output of reaction and constraint forces for static and transient analyses is a production feature.

A new output relative_disp is available to track the relative displacement between the ends of a Joint2G. The output contains a statistical description of the gap for modal random vibration or a numerical value for the gap in Eigen, static, and transient analysis. See Section 7.1.40.
An XML description of the valid commands can be generated with a `-create-xml` command line argument. The primary purpose of this file is to provide a description of valid command lines to SAW.

A conical coordinate system is available. See 2.7.

It is possible to connect multiple tied joints to the same block.

All element types may use temperature dependent material properties. Previously only solid elements could use the temperature dependent properties.

A new capability was added to scale density by block. See 4-91.

The line weld has an option to either remove or not remove gap in the constraints it generates.

Syntax that matches the Sierra/SM piecewise linear function is supported in Sierra/SD.

The documentation has been reorganized following. The Table of Contents and Index have been rewritten to make finding information easier, especially for new users.

**Bug Fixes**

The cbmap MATLAB format superelement output has been corrected. Previously this map was missing rows corresponding to internal degrees of freedom.

Multiple bugs have been corrected in the Qmodaltransient and QmodalFrf capability relating to scrambled output or inconsistent use of modal filters. The Qmodal capability is expected to be deprecated in a future release in favor of a faster Modaltransient and ModalFrf capabilities.

Several robustness improvements have been made to the FilterRbmLoad option. It works properly with both static and transient analysis.

The buckling capability works with gravity loads and constraints.

Modal random vibration properly handles coordinate system in the frequency file outputs.

The element quality metric printed to the log file has been improved. Tetrahedron quality was off by a factor of three, and it could be misreported for elements with small volume.

A `geometric_rigid_body_modes` analysis interacts more correctly with the Eigen solution.

An error was fixed where a single layer hexshell with temperature dependent material properties would use incorrect material properties.

The Navy triangular shell finite element incorrectly transmitted loads. Only the Navy uses the element. Correcting the element lead to re-blessing at least five regression tests.
Syntax and Defaults Changes

Modal random vibration truncation method default has changed from displacement to none. The displacement truncation could truncate modes important to the solution in some cases, such as modes that are localized to a small portion of the domain.
1. **HOW TO RUN Sierra/SD**

**Sierra/SD** provides a massively parallel implementation of structural dynamics finite element analysis. This capability is required for high fidelity, validated models used in modal, vibration, static and shock analysis of weapons systems. General capabilities for modal, statics and transient dynamics are provided. The **Sierra/SD** software evolved from the “Salinas” package.

**Sierra/SD** tutorials for new users are available [https://snl-wiki.sandia.gov/display/CKB/How-to+articles](https://snl-wiki.sandia.gov/display/CKB/How-to+articles)

This section describes the command line arguments and workflows typical of **Sierra/SD** analysis. This section is primarily applicable to analysts at Sandia. Sections 2 through 8 describe **Sierra/SD** capabilities invoked via the input deck. Section 9 provides example problems.

### 1.1. **Accessing** Sierra/SD

At Sandia Sierra is installed on many Unix-based High Performance Computing (HPC) systems. This includes dedicated and shared blades, shared computational clusters (e.g. cee-compute) and large queued clusters (Institutional Clusters such as Eclipse). To run Sierra applications you need to have access to one of these machines. You also must request “SIERRA Analysts Code Access” through WEBCARS.

### 1.2. **Modules and Executables**

Configuration of Sierra applications is controlled via modules. The first step to running a Sierra application is to load the appropriate module for the desired version. For example:

```
workstation_prompt> module load sierra
```

Several modules are commonly used.

- **sierra**: This is the latest released version of Sierra and is generally the recommended version to be used.

- **sierra/X.YY**: This will load a specific Sierra version, for example “sierra/4.58”. Sierra releases are done twice a year and generally the previous four releases are available.

- **sierra/sprint**: This is the latest “sprint snapshot” version of Sierra. A new sprint snapshot is installed every three weeks. Sprint snapshots contain the latest developed features but have less quality assurance testing than the main releases. Sprint snapshots should primarily be used to do beta evaluation of newly developed features.
• **sierra/daily**: This is the development version of Sierra built on the previous night. No quality assurance is done on this version. This version should only be used when directly working with the development team to help evaluate a bug fix.

### 1.3. *The Sierra/SD salinas Executable*

The primary executable for running structural dynamics analysis is called "salinas". It uses a text format input deck to configure a simulation. This users manual primarily describes the input deck options and format. An example invoking a basic serial Sierra/SD analysis is:

```
workstation_prompt> module load sierra
workstation_prompt> salinas -i input.inp
```

The commonly used command line arguments to the salinas executable are given in Table 1-1.
Table 1-1. – Command Line Options.

<table>
<thead>
<tr>
<th>String</th>
<th>Descriptor</th>
</tr>
</thead>
<tbody>
<tr>
<td>-h</td>
<td>Prints help message</td>
</tr>
<tr>
<td>-i</td>
<td>Path and name of the input deck. The input deck is traditionally given</td>
</tr>
<tr>
<td></td>
<td>the extension .inp. However, any name is allowed. If no ‘-i’ is given</td>
</tr>
<tr>
<td></td>
<td>the last argument of the line is assumed to be the input deck name.</td>
</tr>
<tr>
<td>-l or -o</td>
<td>Name for the output log file. By default, output to &lt;input_deck_name&gt;.rslt</td>
</tr>
<tr>
<td></td>
<td>in serial or &lt;input_deck_name&gt;_0.rslt in parallel.</td>
</tr>
<tr>
<td>-n</td>
<td>Do not overwrite any existing output log and diagnostic .dat files. Do</td>
</tr>
<tr>
<td></td>
<td>write a new file appending a 1, 2, 3, etc. to the name.</td>
</tr>
<tr>
<td>-d</td>
<td>Change working directory for the run. Equivalent to cd to that directory</td>
</tr>
<tr>
<td></td>
<td>and running salinas</td>
</tr>
<tr>
<td>-define</td>
<td>Define variables for Aprepro pre-processing of the input deck. See Section</td>
</tr>
<tr>
<td></td>
<td>2.1 for details</td>
</tr>
<tr>
<td>--check-syntax</td>
<td>Do initial syntax checking of the input deck and then stop before reading</td>
</tr>
<tr>
<td></td>
<td>mesh or doing any large calculations. This option can be used to debug</td>
</tr>
<tr>
<td></td>
<td>model input in serial prior to submitting to a queued system.</td>
</tr>
<tr>
<td>-beta</td>
<td>Enable use of ‘beta’ capabilities. These are capabilities in active</td>
</tr>
<tr>
<td></td>
<td>development, are subject to change without notice, and may have less</td>
</tr>
<tr>
<td></td>
<td>rigorous testing.</td>
</tr>
<tr>
<td>-nt</td>
<td>Number of threads to use per MPI rank. Using more threads than physical</td>
</tr>
<tr>
<td></td>
<td>cores (i.e., hyper-threading) may result in decreased performance. This</td>
</tr>
<tr>
<td></td>
<td>option is only supported on a few platforms. Ask the Sierra/SD team for</td>
</tr>
<tr>
<td></td>
<td>more information.</td>
</tr>
<tr>
<td>-ndevices</td>
<td>Number of GPUs to use per hardware node. This option is in development</td>
</tr>
<tr>
<td></td>
<td>and not ready for general use.</td>
</tr>
<tr>
<td>-create-xml</td>
<td>Create an xml format command specification. Primarily for interacting</td>
</tr>
<tr>
<td></td>
<td>with graphical input file creation utilities such as SAW.</td>
</tr>
</tbody>
</table>

1.4. **MPI Parallel Execution**

Sierra/SD may be run either in serial or in parallel with MPI. In parallel Sierra/SD has demonstrated efficiency to thousands of processor cores. Some basic work-flow examples for parallel execution are provided below. Information on using on the current platforms is available at High Performance Computing website [https://computing.sandia.gov](https://computing.sandia.gov)

Additional steps are necessary to execute in parallel instead of serial. The following examples use the input deck `input.inp` and the Exodus mesh file `mesh.exo`. 

---

17
1.4.1. **Number of MPI Processes Needed**

MPI is an optimization that can decrease run time. The configuration of Sierra depends on the architecture. A central processing unit (CPU) has multiple cores and possibly a graphical processing unit (GPU) accelerator. These notes only discuss the CPU. A user must know one thing about the CPU, the number of cores per node (or processor). One way to determine the cores per node is to use the `lscpu` command and look for **Core(s) per socket**. Another way is `cat /proc/cpuinfo`. Look for **cpu cores**.

Running **Sierra/SD** in parallel requires the user to specify how many MPI ranks will be used. Choosing the number of cores so that the number of elements per subdomain approximately 5000 typically works fine. Generally a computational “node” will run several MPI ranks and many computational nodes may be used simultaneously. Selecting the number of ranks to use is based on three concerns. First, each computational node has finite memory and enough nodes must be used to fit the problem in memory. Second, use of more MPI ranks and more computational nodes can reduce time to solution. Third, efficient use of machines should be considered as no analysis will achieve perfect parallel efficiency and using many computational cores on a small problem may provide suboptimal machine utilization. The appropriate number of MPI ranks to use is primarily determined by how many degrees of freedom (DOF) are in the model. Though machines are variable Table 1-2 can be used as a rough guide help guide selection of the number of MPI ranks and thus computational cores to use.

<table>
<thead>
<tr>
<th>memory per core</th>
<th>dofs per core</th>
<th>Num Cores Needed</th>
</tr>
</thead>
<tbody>
<tr>
<td>1GB</td>
<td>15,000</td>
<td><em>dofs</em>/15,000</td>
</tr>
<tr>
<td>2GB</td>
<td>30,000</td>
<td><em>dofs</em>/30,000</td>
</tr>
<tr>
<td>4GB</td>
<td>50,000</td>
<td><em>dofs</em>/50,000</td>
</tr>
<tr>
<td>8GB</td>
<td>70,000</td>
<td><em>dofs</em>/70,000</td>
</tr>
<tr>
<td>16GB</td>
<td>90,000</td>
<td><em>dofs</em>/90,000</td>
</tr>
</tbody>
</table>

*Table 1-2. – Determining Number Of Processors Needed.*

Memory use depends on a variety of factors including the element type used, the solution strategy and the output processing. The numbers in the table are generally conservative. Fortunately, in most cases it is not critical unless there is not enough memory on the compute node.

For high memory use analyses it may be necessary to run **Sierra/SD** with half the maximum number of cores available per computational node. The reason is that using half MPI ranks doubles the memory assigned to each rank. As there is significant memory overhead for an MPI rank, memory limited analyses will benefit more by adding more memory to each rank than spreading the mesh to a larger number of ranks.
1.4.2. Mesh Decomposition

If running in parallel, prior to running the salinas executable first the Exodus mesh file must be parallel partitioned. Mesh decomposition involves dividing the input mesh file into several pieces, one of which will be read by each MPI rank. Several tools exist for this mesh decomposition step, the recommended tool is stk_balanceparallel computing!stk_balance. For more details on stk_balance see.\(^{59}\)

In the next example stk_balance is executed on a non queued blade.

```
workstation_prompt> launch -n 16 stk_balance mesh.exo split
```

This command will decompose the mesh file into 16 parts with names like mesh.exo.16.00, mesh.exo.16.01, etc. and place those spread files into the directory 'split'. Storing the decomposed mesh files in a separate directory is optional. The location of the input mesh file is specified via commands in the input deck as described in Section 2.2.

1.4.3. Running the Sierra/SD Executable in Parallel

With multiple ranks, the Sierra/SD executable must also be run in parallel.

An example of running Sierra/SD in parallel on a non queued blade would be

```
workstation_prompt> launch -n 16 salinas -i input.inp
```

See Section 1.6 for more examples on different systems.

1.4.4. Post Processing in Parallel

In parallel Sierra/SD creates several output files as described in Section 7.

- **log**: In serial the default log file name is <input_deck_name>.rslt. In parallel the default name is <input_deck_name>_0.rslt. This is a text output file giving vital information about the run. The '_0' indicates the is file written by processor zero. Information relevant to the global model behavior is accumulated to processor zero for output.

- **Mesh based Exodus**: The output name will depend on the structure of the input deck, but usually follows a pattern like <mesh_name_base>-out.<mesh_name_extension>. This includes mesh based information such as nodal displacement. Similar to the decomposed input mesh this output Exodus file will be written as a decomposed file with one portion per MPI rank. Some post processing tools such as Ensight or Paraview can read the decomposed Exodus file directly. For other tools the file must first be combined to a single file with the Seacas tool 'epu'.\(^{25}\)
• **Diagnostics**: This could include detailed information from the solver (dd_solver.dat) or other text and MATLAB format output files.

### 1.5. File system concerns

**Sierra/SD** can output large files and works best when writing to fast output systems. Running analyses on scratch drives such as ‘/nscratch’ on HPC system or local ‘/scratch’ on blades are ideal. Large shared disks such as ‘/gpfs1’ are viable though may incur some runtime overhead. Drives that are not mounted by computational nodes of HPC systems such as /tmp should be avoided.

### 1.6. Workflow Examples

There are many ways to run **Sierra/SD** depending on the analysis size and machine used. Several common examples are presented.

The next commands run a serial job on a blade and examine the log file.

```bash
workstation_prompt> module load sierra
workstation_prompt> salinas -i input.inp
workstation_prompt> cat input.rslt
```

Run a parallel job using four processors of a blade. Look at exodus output results via the textual output query tool explore.

```bash
workstation_prompt> module load sierra
workstation_prompt> launch -n 4 stk_balance mesh.g
workstation_prompt> launch -n 4 salinas -i input.inp
workstation_prompt> module load seacas
workstation_prompt> epu --auto mesh-out.g.4.0
workstation_prompt> explore mesh-out.g
```

Run a parallel job on a queued system using 72 MPI ranks and 2 computational nodes manually calling `sbatch`, then combine to a single file, all with a single queue submission. In this case assuming a machine like Eclipse where each computational node can run up to 36 MPI ranks. The Seacas tool `epu` can run in parallel, but is usually run on only a few MPI ranks.

```bash
Contents of submit.sh
#!/bin/bash
launch -n 72 stk_balance -i mesh.g
launch -n 72 salinas -i input.inp
launch -n 8 epu --auto mesh-out.g.72.00
```
Run a parallel job on a queued system using 8 MPI ranks spread to 4 computational nodes to get access to more memory per computational process. In this case assuming a machine like Eclipse where each computational node defaults to 36 ranks.

### Contents of submit.sh

```bash
#!/bin/bash
launch -n 8 stk_balance -i mesh.g
launch -n 8 salinas -i input.inp
launch -n 8 epu --auto mesh-out.g.8.0
```

Run a parallel job on a queued system using the “sierra” script. If the mesh file was not yet decomposed for parallel execution the sierra script will automatically decompose it prior to running the salinas executable. Epu would run in serial on the HPC head node. Note the sierra script can be used to submit Sierra/SD analyses. However, the sierra script ‘–post’ option is currently not functional with Sierra/SD (for other Sierra tools the post option automatically combines output mesh files, similar to the below “epu” command.) The sierra script can potentially be helpful to translate the number of MPI ranks to the correct number of nodes to use on a given machine. Additionally, the same sierra script submission command can be used on queued HPC systems and non queued blades and clusters.

```bash
module load sierra
sierra -j 72 -T 4:00:00 salinas -i input.inp
module load seacas
epu --auto mesh-out.g.72.00
```

Run coupled Sierra/SD Fuego analysis on a non queued system for fluid structure coupling. In this case salinas will use 8 MPI ranks and fuego will use 16 MPI ranks for a total use of 24 MPI ranks. Note, this is an advanced use case.

```bash
module load sierra
launch -n 8 stk_balance salinas_mesh.g
launch -n 16 stk_balance fuego_mesh.g
launch -n 8 salinas -i salinas_input.inp
    --fuego-coupling : -np 16 fuego -i fuego_input.inp
```
1.7. **Thread Parallelism**

In addition to decomposition based MPI parallelism, **Sierra/SD** also supports thread parallelism on some platforms (currently Trinity and GCC development platforms). Threads are activated by the command line option “-nt <numThreads>”. The ‘numThreads’ given will be the number of OpenMP threads to use on each MPI rank. Threaded execution is most valuable on large models. Using a mixture of thread parallelism and MPI parallelism can give optimal performance when the number of MPI processes required would otherwise be very large. As a rule of thumb thread parallelism will provide benefit when exceeding about 200 MPI processes or when more cores are required than mpi ranks to obtain more memory. When using thread parallelism, the number of threads used times the number of mpi ranks used should be setup to be equal the total number of processor cores available on compute nodes.

**Note:** while the number of threads used in **Sierra/SD** is controlled by the command line option “-nt”, it is recommended that the user also set the environment variable ‘OMP_NUM_THREADS’ to be the same value. While **Sierra/SD** doesn’t depend on ‘OMP_NUM_THREADS’, there might be other aspects of your workflow that would, and so we recommend setting both to be consistent. In fact, **Sierra/SD** will output a warning if ‘OMP_NUM_THREADS’ does not match the value set by “-nt”.

1.8. **Troubleshooting**

A variety of issues can cause an analysis to fail. There are still bugs in the **Sierra/SD** software, and these will continue to be found. However, most problems are identified with problems in the model or other input to the software. This section may help to identify these issues with the goal of completing the analysis properly. The best troubleshooting strategy is to try to eliminate the modeling issues, and only then treat the problem as a potential bug.

Users can troubleshoot **Sierra/SD** issues through stand-alone tools or using **Sierra/SD** capabilities. The following sections will describe some ways to do this. First stand-alone tools are described. Second ways of using **Sierra/SD** capabilities to troubleshoot are described.

Queuing systems are inevitable in high performance computing. Any extra step lowering the risk of there being a mistake in the input file is worthwhile.

1.8.1. **Stand-Alone Tools**

Currently, two tools exist which can help the user debug their mesh file, i.e., **Exodus** file: **Explore** and **cubit**.
1.8.1.1. Explore  Explore is an ACCESS/SEACAS utility that can be used to interrogate the Exodus file. One of the commands in Explore that can be used is check. It is used as follows:

```
prompt> explore cube.exo
.
.
EXPLORE> check

  Database check is completed

EXPLORE>
```

If there are any warning or errors, they will appear before the Database check is completed message.

1.8.1.2. Cubit  The Cubit team has developed a GUI-based tool named cubit. It can be used to visualize mesh quality parameters of the Exodus file. For questions about Cubit, please contact the Cubit team at cubit-dev@sandia.gov.

1.8.2. Using Sierra/SD To Troubleshoot

The checkout solution method 3.3 checks the input for errors. It skips matrix formation and solves. The dump solution method 3.9 also check the input for errors. The dump solution does form matrices, but like checkout skips the solves.

Syntax checking 2.3 is helpful.

The user has to take additional steps before executing the parallel version of Sierra/SD. One of the steps is to run stk_balance to decompose the finite element model. Using the Exodus file and the load balancing file, the next step is to run nem_spread to create the partitioned files on the parallel platform where Sierra/SD will be executed. Finally, the commands needed to run Sierra/SD on the parallel platform need to be learned so that execution of Sierra/SD can begin. Many of these steps can cause frustration to the user, but problems with any of these steps are often easily addressed.

All of these steps are worthwhile before running a large model on a queued system. Running the large model on a smaller computer (without a queue) to confirm that the simulation fails in the expected way (by running out of virtual memory) is also worthwhile.

The output includes the Approximate Matrix infinity norm ratios table. This table shows information about the stiffness, mass and dynamic matrices. The value shown is the ratio of the largest diagonal entry to the smallest diagonal entry. If the matrix has a 0 on the diagonal, then 0 is shown.
1.8.3. Modal Analysis

It is possible for the eigen solution method (discussed in Section 3.10) to diverge. Section 2.3 mentions the eig_tol parameter. The default value is about $10^{-16}$ (for the Eigen solution case). Adequate modes can be determined with much larger values of eig_tol. Values such as $10^{-8}$ are reasonable.

At times an analyst may choose to use the eigen method to diagnose a problem. This can be done by using as large a value of eig_tol as is needed. The number of modes would also be as small as needed. After the issue is resolved, if eigenvectors are still needed, don’t forget to reset eig_tol to a small value.

The modal analysis algorithm depends on a linear solver, and assumes that the linear systems are solved accurately. The shift is almost always negative. Increasing the magnitude of a negative shift forms linear systems that are easier to solve, and makes the eigenvalue problem harder to solve. The iterative linear solver (GDSW) parameters trade off between speed and accuracy. For example, a small value of the solver_tol increases accuracy and computational expense. Solving the linear system more accurately makes the eigenvalue problem easier to solve.

1.8.4. Evaluating Memory Use

The Sierra/SD software tends to use a lot of memory. Matrices are generated and solved, and while this is often the fastest method of solution, it results in large memory demands. Parallel computing has its own issues for memory use.

Memory use diagnostics can be requested in the “ECHO” section of the input (see Section 7.7).

1.8.5. Identifying Problematic Subdomains

The partition of a finite element mesh into subdomains may be visualized by rejoining the partitioned files into a new file using epu with the -add_processor_id option.

1.8.6. Limitations of SD Finite Elements

Element quality is a function of both the element geometry and the type of element. The Hex8, Wedge6, Tet4, Tria3, and QuadT elements implement a condition number. Note that the QuadT consists of two triangular elements.

Overall second and higher order finite element methods are much less sensitive to element quality than low order methods. However, there is a tipping point. A high order element will invert before the corresponding low order element. With that in mind, it is noteworthy that Sierra/SD ignores this issue.
<table>
<thead>
<tr>
<th>element</th>
<th>uses condition number of</th>
<th>element</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hex20</td>
<td>Hex8</td>
<td></td>
</tr>
<tr>
<td>Wedge15</td>
<td>Wedge6</td>
<td></td>
</tr>
<tr>
<td>Tet10</td>
<td>Tet4</td>
<td></td>
</tr>
<tr>
<td>CuTet10</td>
<td>Tet4</td>
<td></td>
</tr>
<tr>
<td>Tria6</td>
<td>Tria3</td>
<td></td>
</tr>
<tr>
<td>TriaShell</td>
<td>Tria3</td>
<td></td>
</tr>
<tr>
<td>NLayeredShell</td>
<td>QuadT</td>
<td></td>
</tr>
<tr>
<td>QuadS_GY</td>
<td>QuadT</td>
<td></td>
</tr>
<tr>
<td>NQuad</td>
<td>unknown</td>
<td></td>
</tr>
</tbody>
</table>

Table 1-3. – Elements using other elements condition number.

These approximations are optimistic. Although the NLayeredShell uses a bilinear mapping, the QuadT uses two affine mappings.

Some Sierra/SD elements that have condition numbers and can invert do not implement a condition number. Instead, Sierra/SD reports a condition number of 1. All Sierra/SD HexShell, InfiniteElement, PHex, PTet, PWedge and PmlIsoSolid elements have the optimistic condition number 1. Rigid, Superelement, 1 dimensional and 0 dimensional elements do not have a condition number, and Sierra/SD reports the condition number 1.

1.8.7. Problematic Elements and Connectivity

Many problems are caused by “bad” elements. Following are a few issues that come up periodically.

Rotational Invariance can be lost for certain elements such as springs if they are not of zero length. The spring shown in Figure 1-1 is invariant to rotation about the x-axis, but not invariant to rotation about y or z. If we consider an undeformed rotation about the center of the beam along the z axis we would find that \( u_y(1) < 0 \) and \( u_y(2) = -u_y(1) \). If the spring has \( KY \neq 0 \), then this undeformed rotation results in strain energy, \( E = 2 \ KY \ u_y^2 \). Thus, the rigid body rotation is no longer a zero energy mode.

This is important for a variety of line type elements including spring, Joint2G and gap elements.

Bad element shape is a major source of problems. For example, we have examined models that have “triangles” where one side is 1/200th the length of the other sides. This produces poor element matrices. In some cases this can destroy the condition of the entire system. Such elements can sometimes be found using the kdiag output option described in Section 7.1.50.
Figure 1-1. – Single Spring element.

$\begin{array}{c}
\begin{array}{c}
\text{y} \\
1 \\
\text{x} \\
2
\end{array}
\end{array}$

**Decomposition weakness** The broken figure is an issue for trusses (or rods) and some other elements. The truss in the top part of Figure 1-2 is self-sustaining when made of truss elements. However, because truss elements have no rotational stiffness, the decomposed model in the lower part of the figure contains mechanisms. Note that there is no way to decompose the model without introducing such mechanisms.

Figure 1-2. – Truss Decomposition Issues.

complete truss

decomposed model

Truss elements can be used in **Sierra/SD**. Simulation issues have been traced to poor decompositions in the past. In practice these issues have not come up with stk_balance (see 1.1).

**Poor Connectivity** A structure that has poorly connected regions can be difficult to analyze. If elements have not been properly equivalenced, there can be thousands of zero energy modes in the model. **Sierra/SD** can identify up to a few dozen redundant modes in the best of cases.

**Poor Units** In models with shell elements, rotational degrees of freedom are active. The rotations scale like the displacements divided by the length. An unfortunate choice of length unit will cause ill conditioning. In theory a unit of length that is much too large could lead to inaccurate rotations. And a unit of length that is much too small could lead to inaccurate displacements. This issue has not been investigated. A smaller linear solver tolerance would generally mitigate the issue.
1.9. Over-determined Constraints and Loss of Rigid Body Modes

Building models with tied surfaces or other multi-point constraints (MPCs) may involve more than one iteration. Computing eigenvalues is a common way to check a model. Gap Removal is a light-weight solution method enabling users to obtain information about their tied surfaces earlier in the process, before having to do all the work necessary to successfully set up, submit, and run the massively parallel simulation of the modes.

Solving eigenvalue problems has issues of its own. The eigenvalue decomposition of a model with no prescribed boundary conditions would generally be expected to produce six rigid body modes. A variety of internal constraint issues can cause loss of these rigid body modes. Such artificial constraint of a model can substantially degrade the quality of the overall solution. Sierra/SD has several internal correction algorithm and warnings to detect and repair issues, such as GDSW con_tolerance described in Section 2.5. However, care should be taken to minimize the complexity of constraint equations. Generally having different types of constraints using the same nodes can be potentially problematic. The following are the types of constraint combinations are known to be particularly problematic:

- Intersection of tied data or contact constraints where one node is tied to two or more faces.
- Chains or cycles tied/contact constraints. This could involve a node on side 'A' of an interface being tied to faces on side 'B' and simultaneously nodes on side 'B' being tied to the faces of side 'A'. This is especially problematic in self contact type situations (a tied surface folding over on itself.)
- Intersection of tied/contact constraints and rigid set constraints where a tied node of the node-face constraint is also part of the rigid set.
- Manually specified MPCs that form long chains, cycles, or extensive intersections.
- Intersection of complex manually specified MPCs with any other type of constraint such as rigid set or tied data.
2. GENERAL COMMANDS

No one seems to know what makes a command general. That is why the explanation is missing here.

2.1. Input deck format

The input is a finite element mesh (in Exodus format) paired with an input deck setting both the materials used in the model and what to simulate. Once mesh and materials are set, any type of analysis can be performed. An input deck (or text input file) consists of sections. The order of the sections does not matter, nor does the order of the directives within a section. Commands are case-insensitive.

The Sierra tools related to Sierra/SD include the Cubit geometry and mesh generation toolkit, the ACCESS/SEACAS tool suite for manipulating exodus files and the Nasgen program for convert NASTRAN files to both the exodus file and the text input file.

Traditionally the sections of an input file are ordered so that directives that are most likely to change are near the top. The solution section selects an analysis type. The file section sets the Exodus file name. An Exodus file for a structural model typically contains many element blocks. The input file must have a block section for each element block. And a material section specifies each of the materials used in one or more element blocks.

Typically, the input file has an extension of .inp, although any extension is permitted. If the .inp extension is used, Sierra/SD may be invoked on the input without specifying the extension.

The input file is logically separated into sections. Each section begins with a keyword (Solution, block, etc), and ends with the reserved word end. Words within a section are separated with “white space” consisting of tabs, spaces, and line feeds. Comments are permitted anywhere within the file, and follow the C++ convention, i.e. a comment begins with the comment character (i.e. //) and ends with the end of the line. ¹

Comments

Several options are available for a comment specifier. These are listed in Table 2-4. For any string used to specify a comment, all characters following the comment string are skipped.

¹To be safe, define comments as one of the allowable comment characters (i.e. “//”) followed by a space.
### Table 2-4. – Comment String Options.

<table>
<thead>
<tr>
<th>String</th>
<th>Descriptor</th>
</tr>
</thead>
<tbody>
<tr>
<td>//</td>
<td>C++ style comments.</td>
</tr>
<tr>
<td>/<em>...</em>/</td>
<td>C++ style block comments.</td>
</tr>
<tr>
<td>#</td>
<td>Hash character. Used in scripts and Sierra input.</td>
</tr>
<tr>
<td>$</td>
<td>Dollar sign. Aprepro/Seacas default.</td>
</tr>
</tbody>
</table>

### Meta-characters

**Sierra/SD** supports the use of several meta-characters in the input file. They are listed below, along with what they represent.

- **%P** The number of processors for the run.
- **%B** The base file name. For example, if the input file is `example.inp`, `%B` will be replaced with `example`.
- **%T** The **Sierra/SD** start time. This is the same time that is echoed to the screen at the beginning of the run, but in a shorter and more file-friendly format. For example, if the **Sierra/SD** start time was **Tue Sep 24, 2019, 16:05:41**, `%T` will be replaced with `2019_09_24_16_05_41`.

### Skipping Sections

Occasionally an entire section may need to be commented out. This may be done using "//" on each line of the section, or surrounding the section with the block comment characters "/*" and "*/". A third way to comment out an entire section is to begin it with double "$$" characters. In the following, block 1 is commented out, and block 4 is active.

```plaintext
$$ BLOCK 1 // this section skipped
    material 1
END

BLOCK 4 // this section valid
    material 1
END
```

Except for special cases such as file names, the input file is case-insensitive. Either the single quote ' or the double quote " may be used throughout the input to keep multiple words together (e.g. a name or title). Quotes may be nested by using both single and double quotes, as in either 'a string with "embedded" quotes' or "a string with 'embedded' quotes".
Preprocessing with Aprepro

The Algebraic PREPOcessor, aprepro, in the ACCESS/SEACAS tool suite can be run either stand alone, or as part of the analysis. The use of Aprepro is enabled by default, but can be disabled with the -noaprepro command line flag. Aprepro can be used for a variety of purposes.

1. Define variables on the command line. This is especially useful for automated runs such as optimization and uncertainty quantification.
2. Simplify input by allowing algebraic expressions, e.g. \( Y = \{ 4 * 3 \} \).
3. Automatically include text of other files.
4. Manage various systems of units, e.g. \( Y = \{ 10 * \text{psi} \} \).

For details on Aprepro in general, and for stand alone documentation, please refer to the Seacas documentation.\(^5\) All the rules for command line substitution apply to the built-in capability. Definition of command line variables during analysis requires specification of a command line argument, -define, as used in the example.

\[
\text{sierra salinas --define "E_val=10E6 nu_val=0.30" -i example.inp}
\]

In this example, the text “E_val” in the input file, example.inp will be replaced with “10E6”. Likewise, “nu_val” will be replaced by 0.30.

Including Files

The input parser supports nested includes using Aprepro. Files may be included to any depth.

\[
\{\text{include("english_materials")}\}
\]

The include command may occur anywhere on the line (though for readability and consistency we recommend that it be the start of the line). Case sensitivity will be preserved.

Input Summary

Summarizing, a minimum of two files are needed to run Sierra/SD, namely, a text input file, e.g. example.inp, and an Exodus input file,\(^4\) e.g. example.exo, which contains the finite element model. The finite element model is specified in example.inp as the geometry file (see Section 2.2).

Each of the Sierra/SD input sections is described in the following section.
**Integer List** An integer list may be required as a parameter for a number of keywords. The list is of a format similar to that of MATLAB. A simple list such as “1,2,3,4” is possible. One may also provide a sequence such as “1:4” which is completely equivalent to the previous example. A *step* value may also be provided, as in “2:2:20”. The second term between the colons is the step. For this example, we list all even values between (and including) 2 and 20. Such combinations can also be combined, as in “1,2,3:2:7,11,13,17,19”.

It is recommended that if such lists have spaces they be placed in (single or double) quotes, i.e. ‘1 2’ is preferred, but 1 2 without quotations is also acceptable.

### 2.2. Input mesh geometry file

Disk files names are specified in the *file* section. The parameters for the *file* section are,

<table>
<thead>
<tr>
<th>Option</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>geometry_file</td>
<td>Indicates which Exodus file to use</td>
</tr>
<tr>
<td>user subroutine file</td>
<td>File containing user subroutines, currently relevant only to user mount element</td>
</tr>
</tbody>
</table>

#### 2.2.1. Geometry_file

The geometry file is used for input of the mesh geometry including the nodes, elements, connectivity and attributes. It is typically a binary Exodus file.

**2.2.1.1. Multiple processors** In a multiprocessor environment, the file name is determined by appending the “dot qualified” processor number and processor id to the geometry file specification. For example, if the user specifies,

```
gameometry_file='temp1/example.par'
```

There are 4 processors, then the following files will be opened.

```
temp1/example.par.4.0
temp1/example.par.4.1
temp1/example.par.4.2
temp1/example.par.4.3
```

---

2In other words, the user specifies the path name of the first parallel file, but omits the processor count information. This method permits specification of the file name independent of the number of processors used.
In this example the input file is not in the current working directory.

Specifying the **Exodus** mesh input file in a way that does not depend on the number of input files (or MPI ranks) saves time in the long run. If the input file is `example.exo` and the partitioner is configured to generate the files `example.exo.4.0`,... then the same **Sierra/SD** text input file works in serial and in parallel.

### 2.2.1.2. Single processor

On a single processor, the file is not “spread”, and the full file path is provided. A representative serial file section follows.

```plaintext
file
  geometry_file example.exo
end
```

Note:

- If the file name is not included in quotes, it will be converted to lower case.
- A single processor run, even using mpi protocol, will not append the number of processors and processor ID to the file name.
- Section 1.1 shows the steps involved in the parallel execution of **Sierra/SD**.

### 2.2.2. Exodus Database Naming Conventions

When the mesh file has an **Exodus** format, there are three basic conventions that apply to user input for various command lines. The conventions concern side sets (surfaces), node sets and blocks.

First, for a mesh file with the **Exodus** format, the **Exodus** side set is referenced as a surface. In **Sierra**, a surface consists of element faces plus all the nodes and edges associated with these faces. A surface definition can be used not only to select a group of faces but also to select a group of edges or a group of nodes that are associated with those faces. For nodal boundary conditions that use the surface specification, all the nodes associated with the faces on a specific surface will have this boundary condition applied to them.

A group of elements can also be used to select other mesh entities. In **Sierra**, a block consists of elements plus the associated faces, edges, and nodes. The block and surface concepts are similar in that both have associated derived quantities.

Blocks, sidesets, or nodesets can be grouped together into **assemblies**. However, an assembly must contain entities of the same "type", where "type" is either a block, sideset, nodeset, or another assembly. An assembly cannot contain itself, either directly or indirectly. Currently, not all tools support assemblies. One way to create them is with `io modificarite`. In addition, assemblies can be created directly in the **Sierra/SD** input deck using the **ASSEMBLY** section (section 2.2.3).
The specification for a surface identifier can have the form surface_id, where id is the integer identifier for the surface. Surfaces can also be identified by the id alone. For example, if the side set identifier is 125, the surface could be referred to as surface_125 or simply 125. It is also possible to name a surface in some mesh generation programs, and that name can be used in the input file. The specification for element blocks and nodesets are defined similarly ("block"_id/id/name and "nodelist"_id/id/name). Assemblies can also be specified similarly ("assembly"_id/name), but specification of assemblies by id alone is not allowed. Assemblies can be substituted anywhere in the input deck where a list of blocks, sidesets, or nodesets is expected. If the assembly itself contains sub-assemblies, the set of unique leaf blocks, sidesets, or nodesets will be used, depending on context.

2.2.3. ASSEMBLY section

Assemblies can be created using the ASSEMBLY section of the Sierra/SD input deck as shown below (syntax 2.1). The assemblies created in this way can then be used elsewhere in the Sierra/SD input deck. If an integer is given as the assembly name, then the integer will be used as the assembly identifier and the assembly name will be assembly_id, where id is the assembly identifier. Note that a ASSEMBLY section can only refer to assemblies that either already exist in the input mesh or were defined in a prior ASSEMBLY section. If an assembly already exists, either on the input mesh or from a previous ASSEMBLY section, defining it again using a new ASSEMBLY section will result in modifying the assembly so that it contains the union of the entities in the original and the new instance. Additionally, a warning will be printed in the Sierra/SD results file about modifying an existing assembly.

```plaintext
BEGIN ASSEMBLY <string>
  // exactly one of the following:
  block = <list(block)>
  block = all
  sideset = <list(sideset)>
  sideset = all
  nodeset = <list(nodeset)>
  nodeset = all
END
```

Syntax 2.1. ASSEMBLY example

2.2.4. Exodus Naming Limitations

Sierra/SD supports the use of mesh names in the input deck and lists of mesh objects. As a result, it becomes ambiguous which keywords are mesh names and which are intended as input syntax. For example, a nodeset named "fixed" creates ambiguity when the user
specifies "nodeset 1 fixed". This could mean that nodeset 1 is fixed in space, or it could be a list of nodesets '1' and 'fixed'.

To resolve this ambiguity, Sierra/SD maintains a list of reserved keywords which may not appear in the mesh. This list has over 400 entries, and changes with each release. The rules of this list are simple; any valid syntax which appears adjacent to a nodeset list may not appear in the mesh's nodesets. The same rule applies to sidesets and blocks. Since assemblies are valid in all three contexts, they inherit the restrictions of all three.

There are no keywords in the Sierra/SD input deck which begin or end with an underscore. Therefore, '_fixed_' is a valid name for any mesh object in any context. Similarly, 'assembly_5_front' is obscure enough to be safe from this restriction.

Pure integer values are not allowed as mesh names for the same reasons. This is not a restriction on names like 'Block_5', but blocks named '5' are not allowed.

A warning is issued if reserved characters are found in the mesh names. These warnings are not fatal, but having reserved characters in the mesh names severely limits our ability to parse them in the input deck. The reserved character list includes comment characters (//,/*,/*,#, and $), and special-use characters (=,'',"and :), which have extra meaning in Sierra/SD input decks.

Since the list of reserved keywords is based on the allowable syntax in Sierra/SD, it will change based on the current version of the code. The current list of reserved keywords for the version of the code you're using can be printed to screen at any time using the “–keywords” command-line option. Additionally, the ambiguous naming checks can be changed to a warning or ignored entirely using the reserved_keywords parameter (section 2.3).

2.2.5. Additional Comments About Output

A text log or results file can be written for the run. Details of the contents of the results file are controlled in the echo section (see Section 7.7). The results file name is determined by the name of the input file, and will be in the same directory as the input text file, regardless of whether Sierra/SD is being executed in serial or parallel. However, if executing in parallel, using the subdomains option in the echo section allows control of the number of results files. For example, if running on 100 processors, up to 100 result files may be output. Using subdomains “0:2” will only output three files, from subdomains 0, 1, and 2. The default is to output a results file only for processor zero. The results file name uses the base name of the input, with an extension of .rslt. In a parallel computation, the results file names use the base name of the input file, followed by an underscore and the processor number, then followed by the .rslt extension.

For calculations in which geometry based output requests are included (see Section 7), an output Exodus file will be created. An Exodus file is a binary file (NetCDF). It contains the original geometry and the requested output variables. The output Exodus file name is
determined from the geometry file name. The base name of the output is taken from the geometry file by inserting the text “-out” before the file name extension. The output Exodus file will be written to the same directory where the geometry file is stored. If executing Sierra/SD on a parallel machine, the Exodus output files should be written to the raid disks for reasonable performance.

2.3. Parameters

This optional section provides a way to input parameters that are independent of the solution method or solver. Only one parameters section is recognized in each file. The parameters and their meanings are listed below and in Table 2-5 and 2-6. For reference, Table 2-7 are parameters occasionally used by developers, but not recommended for common use.

\textbf{wtMass} This variable multiplies all mass and density on the input, and divides out the results on the output. It is provided primarily for the English system of units where the natural units of mass are units of force. For example, the density of steel is 0.283 $\text{lbs/in}^3$, but “lbs” includes the units of $g=386.4 \text{in/s}^2$.

Using a value of WtMass of 0.00259 (1/386.4), density can be entered as 0.283, the outputs will be in pounds, but the calculations will be performed using the correct mass units. Sierra/SD, like most finite element codes, does not manage the units of the analysis. The selection of a consistent set of units is left to the analyst. For example, if the analyst uses the SI system (Kg, m, s) the units of pressure must be Pascals. Frequencies are reported in Hz. For micromachines these units are awkward. It may be better to use units of grams, millimeters and microseconds. The analyst must ensure that all material properties and loads are converted to these units.

Some examples of useful units are shown in Table 2-8.

\textbf{NegEigen} Unconstrained structures have zero energy modes which may evaluate to small negative numbers due to machine round off. The eigenvalues and associated frequencies are reported as negative numbers in the results files. However, many post processing tools (such as Ensight) require non-negative frequencies. By default, Sierra/SD converts all negative eigenvalues to near zero values in the output Exodus files \(^1\). To retain the negative eigenvalues in the output file, select parameter NegEigen.

\textbf{output_sideset_data} This option turns on sideset output for any requested element output. Currently, sideset output is only enabled for (centroid) volumetric stress and strain (sections 7.1.16 and 7.1.18).

\(^1\)Because many post processing tools are written for transient dynamics, they expect monotonically increasing, positive values for the time. Since eigenvalues are written in the time columns of the output file, they are converted to be monotonically increasing, positive values. Note that the numerically computed eigen frequencies are also stored as global variables in the file.
<table>
<thead>
<tr>
<th>Keyword</th>
<th>Type</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>AllowExodusAttributes</td>
<td>bool</td>
<td>true</td>
<td>Seek undefined attributes on the mesh</td>
</tr>
<tr>
<td>AllowExodusDistFacts</td>
<td>bool</td>
<td>true</td>
<td>Apply distribution factors from the mesh</td>
</tr>
<tr>
<td>AllowInvalidExodusParts</td>
<td>bool</td>
<td>false</td>
<td>Ignore error due to requesting an invalid Exodus part (e.g. sideset/block)</td>
</tr>
<tr>
<td>ComplexStress</td>
<td>yes/no</td>
<td>no</td>
<td>output complex stress in FRF solutions</td>
</tr>
<tr>
<td>condition_limit</td>
<td>Real</td>
<td>1e6</td>
<td>element quality output control</td>
</tr>
<tr>
<td>constraint_correction</td>
<td>yes/no</td>
<td>no</td>
<td>orthogonalize constraints to RBMS</td>
</tr>
<tr>
<td>defaultSpecificHeat</td>
<td>Real</td>
<td>none</td>
<td>material specific heat</td>
</tr>
<tr>
<td>DoInitialMassSolve</td>
<td>bool</td>
<td>true</td>
<td>transients: perform initial mass solve</td>
</tr>
<tr>
<td>eig_tol</td>
<td>Real</td>
<td>auto</td>
<td>Eigenvalue tolerance</td>
</tr>
<tr>
<td>eigen_norm</td>
<td>string</td>
<td>mass</td>
<td>“visualization” or “unit” input of energy data</td>
</tr>
<tr>
<td>energy_time_step</td>
<td>int</td>
<td>1</td>
<td></td>
</tr>
<tr>
<td>energy_exo_var</td>
<td>string</td>
<td>TEMP</td>
<td>Exodus energy variable name</td>
</tr>
<tr>
<td>FilterRbmLoad</td>
<td>noFiltering</td>
<td></td>
<td>rigid body components of loads to filter</td>
</tr>
<tr>
<td>ignore_gap_inversion</td>
<td>bool</td>
<td>false</td>
<td>Ignore element quality changes due to gap and overlap removal</td>
</tr>
<tr>
<td>Info</td>
<td>int</td>
<td>1</td>
<td>screen output control</td>
</tr>
<tr>
<td>MatrixFloor</td>
<td>Real</td>
<td>0</td>
<td>control of matrix fill</td>
</tr>
<tr>
<td>MaxmpcEntries</td>
<td>int</td>
<td>10,000</td>
<td>maximum # entries in any mpc</td>
</tr>
<tr>
<td>MaxResidual</td>
<td>Real</td>
<td>1</td>
<td>maximum residual for eigen</td>
</tr>
<tr>
<td>MortarMethod</td>
<td>string</td>
<td>dual</td>
<td>dual or standard mortar method</td>
</tr>
<tr>
<td>MFile_format</td>
<td>string</td>
<td>sparse_function</td>
<td>control output format for MATLAB</td>
</tr>
<tr>
<td>NegEigen</td>
<td>none</td>
<td></td>
<td>negative eigenvalue flag</td>
</tr>
<tr>
<td>nonlinear_default</td>
<td>yes/no</td>
<td>yes</td>
<td>nonlinear element blocks</td>
</tr>
<tr>
<td>num_rigid_mode</td>
<td>int</td>
<td>0</td>
<td>number of system rigid body modes</td>
</tr>
<tr>
<td>output_sideset_data</td>
<td>bool</td>
<td>false</td>
<td>Output element values on all sidesets or “test”</td>
</tr>
<tr>
<td>RandomNumberGenerator</td>
<td>string</td>
<td>“rand”</td>
<td>filter constraints</td>
</tr>
<tr>
<td>RbmTolerance</td>
<td>Real</td>
<td>1e-10</td>
<td>tolerance for rigid body zero</td>
</tr>
<tr>
<td>RemoveRedundancy</td>
<td>yes/no</td>
<td>yes</td>
<td>filter constraints</td>
</tr>
<tr>
<td>reorder_Rbar</td>
<td>none</td>
<td></td>
<td>constraint reordering flag</td>
</tr>
<tr>
<td>RequireMatchedBlocks</td>
<td>string</td>
<td>ignore</td>
<td>“ignore”, “warn”, or “fatal”</td>
</tr>
<tr>
<td>SkipmpcTouch</td>
<td>none</td>
<td></td>
<td>control of MPCS</td>
</tr>
<tr>
<td>syntax_checking</td>
<td>string</td>
<td>fatal</td>
<td>“ignore”, “warn”, or “fatal”</td>
</tr>
<tr>
<td>reserved_keywords</td>
<td>string</td>
<td>fatal</td>
<td>“ignore”, “warn”, or “fatal”</td>
</tr>
<tr>
<td>restart_consistency_checking</td>
<td>string</td>
<td>fatal</td>
<td>“ignore”, “warn”, or “fatal”</td>
</tr>
<tr>
<td>TangentMethod</td>
<td>string</td>
<td>element</td>
<td>method of computing tangent</td>
</tr>
<tr>
<td>WtMass</td>
<td>Real</td>
<td>1</td>
<td>Mass multiplier</td>
</tr>
</tbody>
</table>

Table 2-5. – Available keywords in the Parameters section.
<table>
<thead>
<tr>
<th>Keyword</th>
<th>Type</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>nodesets_with_disp</td>
<td>list</td>
<td></td>
<td>node sets with prescribed displacements</td>
</tr>
<tr>
<td>thermal_time_step</td>
<td>int</td>
<td>1</td>
<td>input of thermal data</td>
</tr>
<tr>
<td>thermal_exo_var</td>
<td>string</td>
<td>TEMP</td>
<td>Exodus temperature variable name</td>
</tr>
<tr>
<td>mpmd_transfer_version</td>
<td>NSC/new</td>
<td>new</td>
<td>In core transfer coupling library</td>
</tr>
<tr>
<td>mpmd_transfer_type</td>
<td>fuego/copy</td>
<td></td>
<td>Which code transfer is occurring from</td>
</tr>
<tr>
<td></td>
<td>sparc/interpolation</td>
<td></td>
<td>Surfaces on which transfer is occurring</td>
</tr>
<tr>
<td>mpmd_transfer_sidesets</td>
<td>string_list</td>
<td></td>
<td>transfer is occurring</td>
</tr>
</tbody>
</table>

Table 2-6. – Available keywords in the Parameters section related to code coupling and handoff.

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Arg</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>OutputInitialTime</td>
<td>boolean</td>
<td>false</td>
<td>write output at t=0</td>
</tr>
<tr>
<td>SharedAcousticStiffness</td>
<td>yes/no</td>
<td>no</td>
<td>share stiffness across processors</td>
</tr>
<tr>
<td>linear_solver_warn_factor</td>
<td>Real</td>
<td>10</td>
<td>GDSW warnings</td>
</tr>
<tr>
<td>linear_solver_bailout_factor</td>
<td>Real</td>
<td>100</td>
<td>GDSW fatal errors</td>
</tr>
</tbody>
</table>

Table 2-7. – Developer keywords in the Parameters section.
Developer keywords in the Parameters section. These keywords may be used, but their use is discouraged, and they are not fully tested.

Table 2-8. – Some useful combinations of units.

<table>
<thead>
<tr>
<th>length</th>
<th>mass</th>
<th>time</th>
<th>WtMass</th>
<th>density</th>
<th>force</th>
<th>modulus</th>
<th>internal mass</th>
</tr>
</thead>
<tbody>
<tr>
<td>m</td>
<td>Kg</td>
<td>sec</td>
<td>1</td>
<td>Kg/m³</td>
<td>N</td>
<td>N/m² or Pa</td>
<td>Kg</td>
</tr>
<tr>
<td>ft</td>
<td>slug</td>
<td>sec</td>
<td>1</td>
<td>slug/ft³</td>
<td>lbf</td>
<td>lb/ft²</td>
<td>slug</td>
</tr>
<tr>
<td>ft</td>
<td>lbm</td>
<td>sec</td>
<td>1/32.2</td>
<td>lbm/ft³</td>
<td>lbf</td>
<td>lb/ft²</td>
<td>slug</td>
</tr>
<tr>
<td>in</td>
<td>lbm</td>
<td>sec</td>
<td>1/386.4</td>
<td>lbm/in³</td>
<td>lbf</td>
<td>psi</td>
<td>lbm/386.4</td>
</tr>
<tr>
<td>mm</td>
<td>µg</td>
<td>µs</td>
<td>1</td>
<td>Kg/m³</td>
<td>N</td>
<td>MN/m² or MPa</td>
<td>µg</td>
</tr>
<tr>
<td>mm</td>
<td>g</td>
<td>sec</td>
<td>1</td>
<td>g/mm³</td>
<td>µN</td>
<td>N/m² or Pa</td>
<td>gram</td>
</tr>
<tr>
<td>mm</td>
<td>mg</td>
<td>sec</td>
<td>1/1000</td>
<td>g/cm³</td>
<td>µN</td>
<td>N/m² or Pa</td>
<td>gram</td>
</tr>
</tbody>
</table>
**eig_tol** This is the tolerance used in the **eigen** solution method for eigenvalues. The default is machine precision. This parameter can sometimes be loosened to aid convergence. See Section 1.8.3 for details.

**nonlinear_default** In nonlinear transient dynamics or nonlinear statics, computing the fully nonlinear response of all the elements in the mesh can be computationally expensive. In some cases it is unnecessary. For example, for a simulation that only involves Joint2g elements and solid (3D) elements, the analyst may determine that the nonlinear effects of the solid elements are negligible. In such cases, it is advantageous to be able to control the nonlinear response of elements block-by-block. And there is a block-level parameter described in Section 4.7.2 that selects the optional nonlinearities for specific blocks. Instead of entering this parameter for each block, **nonlinear_default** sets the default for all blocks. If **no**, then all blocks default to linear behavior, unless specified otherwise in the BLOCK section. If **yes**, then all elements default to nonlinear behavior. Note that the block-level flags override the **nonlinear_default** keyword. There are two possible cases for this keyword.

- **nonlinear_default=no** All elements default to linear behavior.
- **nonlinear_default=yes** All elements default to nonlinear behavior.

As noted in Section 4.7.2, there are limitations for using linear materials in nonlinear analysis.

**TangentMethod** The tangent stiffness matrix may be used in a full Newton update in nonlinear statics or transient dynamics (see Sections 3.22 and 3.23). By default, each of the elements can compute its own tangent stiffness matrix. There are cases (particularly when elements are under development) when it is better to use a tangent matrix computed from finite difference methods. There are three possible values for this keyword.

- **TangentMethod=element** The standard element method.
- **TangentMethod=difference** Use finite difference.
- **TangentMethod=compare** Use the standard method, but also compute the matrix by the difference method. Output of the difference of every element matrix in the model will be sent to the results file.  

**info**

**GDSW!prt_debug** Option **info** selects diagnostic information for standard out. There are four different levels. Each level increasingly allows more output to standard out. The GDSW option “prt_debug” overrides “info” for GDSW output. The four levels of control are:

- **0. Silent** – output warnings and std error to the scree (untested)

---

2In parallel solutions the results file is written only for the first processor unless the “subdomains” option is specified in the echo section (7.7).
1. **Normal** – output the data most analysts would use (untested)

2. **Detailed** – Convergence, solution addressing issues (unimplemented, tested).

3. **Debug** – Silent, normal, detailed and diagnostic information (tested).

Info is regularly used for debugging eigenvalue problems. Option `prt_debug` is defined in Section 2.5.6. Option `prt_debug` causes GDSW to write load balance information to a file named `subdomainData.dat`. Its contents are described in Section 2.5.3

Example of usage:

```plaintext
Parameters
  info=0
End
```

This sets the “info” control level to Silent.

**syntax_checking** Sierra/SD has the ability to check input files for syntax and spelling errors. This option controls this behavior. By default, a violation is printed to the screen and execution is terminated. If the user wishes, violations can be printed while execution continue, or the checking for violations can be disabled completely.

The three levels of control are:

- **ignore** silently ignores those entries.
- **warn** provides a warning for those entries.
- **fatal** stops the analysis. This is the default.

**reserved_keywords** Sierra/SD has the ability to check exodus entities for potentially ambiguous or confusing names. This option controls this behavior. By default, a violation is printed to the screen and execution is terminated. If the user wishes, violations can be printed while execution continue, or the checking for violations can be disabled completely. Note that this option will only control the behavior when checking ambiguous names: we will always issue a warning if certain reserved characters are found within a name. For more information, see section 2.2.4.

The three levels of control are:

- **ignore** silently ignores those entries.
- **warn** provides a warning for those entries.
- **fatal** stops the analysis. This is the default.
**restart_consistency_checking** by default **Sierra/SD** will check that the restart data file is consistent with the analysis mesh. These checks include confirming identical node ids, element id, node coordinates, and element connectivity. By default, if the restart file is inconsistent **Sierra/SD** will output a fatal error and terminate execution. Generally such restart inconsistencies indicate a serious problem. For example trying to map the restart data from the wrong analysis onto a new mesh will produce nonsensical behavior. Use **restart_consistency_checking** to convert these checks to a warning or skip them.

The three levels of control are:

- **ignore** silently ignores restart consistency issues.
- **warn** provides a warning for restart consistency issues.
- **fatal** stops the analysis for restart consistency issues. This is the default.

**SkipmpcTouch** **Sierra/SD** uses a unique method of determining an active degree of freedom set. Unlike codes like NASTRAN which use an automatic single point constraint method, **Sierra/SD** loops through all elements and activates only degrees of freedom that are required for elements. Multipoint constraints pose a particular problem because some codes (like NASTRAN) may include multipoint constraints to unused degrees of freedom. Since these are eliminated with the auto-spc, this poses no problem to these codes, but may confuse **Sierra/SD** significantly. On the other hand, usually degrees of freedom associated with MPCs should be included in the active set, and leaving them out can produce errors.

As a stopgap measure, we provide the parameter **SkipmpcTouch**. If this parameter is set, no degrees of freedom will be activated through multipoint constraints.

**condition_limit** Element quality checks are important for evaluating the effectiveness of the mesh. By default, elements with moderately bad topology are reported. However, sometimes there are so many of these warnings, that the bad elements may get missed. The **condition_limit** parameter permits user control of the reporting. Setting this parameter to a larger number will eliminate message from marginal elements. Element checking can also be disabled (see the **ElemQualChecks** parameter in the **outputs** Section 7.1.12). The default value is 1e6.

> Note that the **condition_limit** parameter is ignored if the **ElemQualChecks** option has been disabled in the **output** section (7.1.12).

**reorder_Rbar** An **Rbar** is a type of rigid element. This option reorders all **Rbar** elements to minimize the number of them connected to a single node. Having many **Rbar** elements connected to the same node results in a problematic matrix sparsity pattern which can reduce efficiency. Reducing the number of connections can shorten run time.

If redundant **Rbars** are present (i.e. connections forming a cycle), they are removed.
Specify `reorder_Rbar yes` or `reorder_Rbar no`. The default value is `yes`, which means Rbars will be reordered.

**thermal_time_step** For thermal analysis solution procedures (i.e. statics or transient dynamics with a `thermal_load` body load), or for any solution procedure that uses temperature dependent material properties, the temperature distribution of the structure must be read in from the Exodus file. Typically, the input Exodus files in this case would be the output files from a thermal analysis, and thus would contain the necessary temperature data. Since such an analysis could contain several time steps of temperature data, the parameter `thermal_time_step` allows the analyst to select which set of temperature data is to be read into Sierra/SD. The following gives an example.

```
Parameters
    thermal_time_step 10
End
```

In this case the user would be requesting that the temperature data corresponding to the 10th time step be read into Sierra/SD.

**energy_time_step** This variable is identical to the “thermal_time_step” above, but applies to cases where the energy density is input and must be converted to a temperature. Either energy density or temperature can be input, but not both.

**thermal_exo_var** If a material specifies a coefficient of thermal expansion and a reference temperature, then the corresponding thermal strain is computed, and the corresponding thermal load 6.3.8 is applied. Otherwise, a material property may be an arbitrary function of temperature. Temperature dependent materials 4.4.6 are supported. Temperature dependent viscoelastic materials 4.3 are also supported, using the corresponding input syntax. In either case to read the temperature from the input Exodus file, the name of the temperature field has to be specified. And that is what `thermal_exo_var` is for. The default variable name is 'TEMP'. In the following example the name is changed to temperature.

```
Parameters
    thermal_exo_var "temperature"
End
```

**energy_exo_var** This variable is identical to the “thermal_exo_var” above, but applies to cases where the energy density is input and must be converted to a temperature. Either energy density or temperature can be input, but not both. The only difference is that the energy density will be divided by the specific heat to arrive at the temperature.

```
Parameters
    energy_exo_var "EDEP"
End
```
mpmd_transfer_version Defines which version of the runtime application coupling library to use. Currently, the **NSC** option is used for coupling to Nemo, and the **new** option is used to couple to SPARC or Fuego. See Section 6.2.6.

mpmd_transfer_type Defines which code **Sierra/SD** is coupling to during the run. The **fuego** are **copy** options are equivalent and one of them must be used when coupling to Fuego. The **sparc** and **interpolation** are equivalent and one of them must be used when coupling to SPARC. See Section 6.2.6.

mpmd_transfer_sidesets Defines the sidesets on which coupling are occurring. When data is mapped from one code to the other, the transfer will only occur on these sidesets. See Section 6.2.6.

FilterRbmLoad Establishes a filter for rigid body components of the input load. The options are described in the table. It defaults to unfiltered. The parameter may need to be combined with the RbmTolerance and solver parameters. The **FilterRbmLoad** parameter is only supported for transient and static solution cases. For other solution cases this parameter will have no effect on the solution.

During rigid body mode filtering a net force in a rigid body mode will be counter-balanced by a force with a distribution defined by the rigid body mode shape times the mass matrix. For example to counter-balance a net force in X direction effectively a gravity load would be applied to the body where the sum of the forces of that gravity load is equal and opposite to the net force to be balanced. For rigid body filtering a well-defined mass matrix is required for both transient and static solution.

The rigid body load filtering should only be used of a model that has exactly the six standard rigid body modes.

<table>
<thead>
<tr>
<th>Option</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>NoFiltering</td>
<td>skip all RBM filtering for the load</td>
</tr>
<tr>
<td>AllStructural</td>
<td>apply filtering to all 6 structural RBM</td>
</tr>
<tr>
<td>RotationOnly</td>
<td>apply filtering to rigid body rotation only</td>
</tr>
</tbody>
</table>

See Section 6.3.19 for more details about the use of this option.

RbmTolerance Rigid body filters depend upon accurate rigid body modes. The application checks the matrix product of the stiffness matrix to ensure that these vectors are in the null space of the stiffness matrix. If any of the requested vectors are not in the null space, the application terminates. The default is $1e^{-10}$.

RbmTolerance provides user control of the threshold for that error. It depends on the stiffness matrix, $K$, and a rigid body vector, $\phi_r$,

$$ tolerance = \frac{\|K\Phi_r\|_2}{\|Kd\|_\infty\|\Phi_r\|_2} $$

MatrixFloor Primarily a debugging option. The nearly zero terms in a matrix can be removed using this parameter. Values below this floor are eliminated from the
matrix. This can reduce fill, but if used improperly too much of the matrix can be
affected. It can be important when running on different platforms, where round off
can affect the matrix fill, and make it difficult to compare solutions. This is a relative
value, so 1.0E-6 would remove terms in the matrix that are a million times less than
the largest term. Default is zero.

defaultSpecificHeat The specific heat is used to convert energy to temperature with the
following equation, \( \bar{E} = C_v \Delta T \), see Section 4.4.8 for details. The specific heat is set
on a material by material basis. A default value for the specific heat can be set using
the keyword defaultSpecificHeat which can then be overridden by defining the
specific heat for a material. A fatal error is given if the specific heat is not defined for
a block of material containing energy input and the defaultSpecificHeat keyword
can be used to avoid this.

MaxmpcEntries Soft limit on the number of mpc entries in any single multipoint
constraint. Normally the default will be sufficient, but large RBE3 type entries may
exceed this in rare cases. The limit is there to avoid errors reading the input, and
because such large constraints can consume memory.

eigen_norm Eigenvectors may be arbitrarily normalized. Three common approaches are
listed in Table 2-9. All methods retain orthogonality of the eigenvectors, but the
normalization differs. The default, mass normalization, is most commonly used as it
ensures that the inner products of eigenvectors with the mass matrix is identity.
However, this normalization is not well suited to output visualization. The
“visualization” normalization mimics what is automatically done in MSC/Patran,
and should provide a reasonable visualization without rescaling each mode. In
“visualization” normalization, the maximum translational displacement is normalized
to be less than 10 percent of the maximum model extent, while also insuring that the
model rotation remains below 1 radian. Unit normalization ensures that the largest
value of the eigenvector is one. 1 A global variable, EigenVectScale, provides the
scale factor by which the mode was scaled.

<table>
<thead>
<tr>
<th>Method</th>
<th>Algorithm</th>
<th>Comment</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass</td>
<td>( \phi_i^T M \phi_i = 1 )</td>
<td>Default. Simplifies numerics</td>
</tr>
<tr>
<td>Visualization</td>
<td>max(( \phi_i ))=(model size)/10</td>
<td>Simplifies visualization</td>
</tr>
<tr>
<td>Unit</td>
<td>max(( \phi_i ))=1</td>
<td></td>
</tr>
</tbody>
</table>

Table 2-9. – Eigenvector Normalization Methods.

constraint_correction Ensure that each multipoint constraint generated is orthogonal to
all rigid body modes. This is useful for lofted surfaces. If the surfaces are tied as if

\(^1\)The “unit” method of normalization computes max(\( \phi \)), which is computed only on translational dis-
placement degrees of freedom. Note also that only displacements are renormalized. No effort is made to
renormalize element variables such as strains, stresses or energies. Thus, if these are requested in an eigen
analysis, they will not be consistent with the renormalized eigenvectors, but will retain mass normalized
values.

43
they were coincident, the constraints are incorrect, and eliminate some or all the rigid body modes as worked out in subsection Orthogonality of MPC to Rigid Body Vectors in Section Linear Algebra Issues of the Theory Manual.

Parameters

Constraint_Correction=yes

End

MFile_Format Most of our matrix data can be written as MATLAB readable files. By default, these are written as sparse matrices, as functions. Other formats are also available. The “full” format does not use the sparse methods and is thus compatible with Octave or other tools. Alternatively, the “3column” format can be used. In this format, the file is loaded using the MATLAB “load” command. The data is then converted to a sparse matrix using the MATLAB “sparse” command. The “3column” format may be significantly faster in some cases, but it does require more user interaction. Figure 2-3 compares a simple example for the three formats. In all cases, the matrix symmetry is the same. A fourth format, “CSV”, is also available for compatibility with other external tools.  

<table>
<thead>
<tr>
<th>Sparse_Function</th>
<th>Full</th>
<th>3column</th>
</tr>
</thead>
<tbody>
<tr>
<td>function s=Kssr()</td>
<td>s=Kssr()</td>
<td>1 1 0.11</td>
</tr>
<tr>
<td>s=[ 1 1 0.11</td>
<td>s=zeros(2,2);</td>
<td>1 2 0.12</td>
</tr>
<tr>
<td>1 2 0.12</td>
<td>s(1,1)=0.11;</td>
<td>2 2 0.22</td>
</tr>
<tr>
<td>2 2 0.22];</td>
<td>s(1,2)=0.12;</td>
<td></td>
</tr>
<tr>
<td>s=sparse(s(:,1),s(:,2),s(:,3));</td>
<td>s(2,2)=0.22;</td>
<td></td>
</tr>
</tbody>
</table>

Figure 2-3. – Example MFile Format Results.

RemoveRedundancy RemoveRedundancy affects node/face constraints created by ‘Tied Data’ or ‘Contact Definition’. Redundant constraints cause most solvers to fail. Redundant constraints are often introduced when two surface pairs are tied next to each other, but there are a variety of sources for these redundancies. Exact redundancies are always automatically eliminated, but that is often not sufficient. This parameter removes constraints when a node is tied to more than one face or if the node shows up both as a node of a node-face constraint and attached to a face of a different node-face constraint. By default, it is “true”.

RandomNumberGenerator The default random number generator, “rand”, is the standard generator available from system libraries. It should be the best random

\[Note\, that\, the\, CSV\, format\, should\, be\, readable\, by\, Microsoft\, Excel,\, but\, there\, are\, often\, limits\, on\, the\, number\, of\, columns\, that\, can\, be\, read.\]
number generator in terms of the quality implementation. In a few cases the analyst may want a more repeatable random number generator, that is independent of the platform. The “test” random number generator can be used in this case. It is not recommended for general use, and the statistics of the generator are not well-established.

**MortarMethod** Two mortar methods are available in Sierra/SD: standard and dual (see\textsuperscript{37}). By default, the dual method is selected as it is almost always more efficient in memory use.

**ComplexStress** Most often, analysts do not want output of stress variables in frequency response function analysis. Such output is complex, and huge volumes can be generated. Selecting “ComplexStress=yes”, along with “stress” in the echo section permits output of this data. The default is “ComplexStress=no”.

**num_rigid_mode** Is used to signal to the linear solver that the system is singular and that the singularity is associated with structural and/or acoustic rigid body modes. This is used, for example, in the solution of statics problems without any essential boundary conditions or frequency response analysis with the modal acceleration method. Where possible, other methods should be used to eliminate the singularity. For example, in eigen analysis a negative shift is recommended. Currently, allowed values for this parameter are 1 (acoustic mode only), 6 (structural modes only), or 7 (structural and acoustic modes). We also note that when using the FilterRbmLoad parameter, it is necessary to specify num_rigid_mode to correspond to the number of rigid body modes that will be filtered. For example, if FilterRbmLoad was set to AllStructural, then num_rigid_mode should be set to 6.

**ignore_gap_inversion** Initial overlap removal is another name for gap removal. Gap removal may change element quality. A fatal error occurs if element quality gets much worse. To ignore this poor element quality set the ignore_gap_inversion parameter to true. The gap removal solution case 3.35 making debugging easier.

**DoInitialMassSolve** Determining the initial acceleration is necessary\textsuperscript{28} for quadratic convergence. By default, a consistent initial acceleration is found through an initial mass solve, see Section 3.30.1.2. Some combinations of MPCs can result in a singular mass matrix, leading to errors in the initial mass solve. To skip the initial mass solve use the command,

\begin{verbatim}
   DoInitialMassSolve=false
\end{verbatim}

**OutputInitialTime** In transient simulations, the output can be written at time t=0, prior to the first time step by using the command

\begin{verbatim}
   OutputInitialTime=true
\end{verbatim}

The default is set to false. This will not write the initial time step if reading from a restart file because the time is not 0.
**linear_solver_warn_factor** The GDSW linear solver approximates the solution $u$ of a linear system, $Ku = f$ so that the predicted norm of the $k^{th}$ residual $f - Ku_k$ is less than a user specified tolerance times the initial residual norm $f - Au_o$ (the default is $1.e-6$). However, the true residual may be larger, and a careful analyst must check the true residuals. To automate the task of the checking, a warning message is written if the true residual is more than `linear_solver_warn_factor` times the predicted residual. The default value is 10.0.

```
linear_solver_warn_factor=10.0
```

**linear_solver_bailout_factor** The GDSW linear solver approximates the solution $u$ of a linear system, $Au = f$ so that the predicted norm of the $k^{th}$ residual $f - Au_k$ is less than a user specified tolerance times the initial residual norm $f - Au_o$ (the default `solver_tol` is $1.e-6$). However, the true residual may be larger. A fatal error is thrown if the true residual is more than `linear_solver_bailout_factor` times the predicted residual. The default value is 100. For instance, in an ill-conditioned problem as the `solver_tol` is decreased to improve resolution, one may choose to increase the bailout factor to mitigate the risk of a fatal error.

```
linear_solver_bailout_factor=100.
```

**RequireMatchedBlocks** Sierra/SD always requires that each Exodus block have a corresponding entry in the `.inp` file. This parameter controls behavior if there are blocks defined in the `.inp` file that do not appear in the mesh.

- `ignore` silently ignores those entries.
- `warn` provides a warning for those entries. This is the default.
- `fatal` stops the analysis.

**nodesets_with_disp** This parameter, when used in conjunction with the `nUpdateConstraints` transient option (see section 3.30.1), enables specifying prescribed displacements over a subset of all nodes via nodeset output.

`nodesets_with_disp` is currently BETA release. Enable with the “--beta” command-line option.

### 2.4. Solution Options

The options described in Table 2-10 and in the following paragraphs are part of the Solution section in the input file. None of the keywords are required. Note that in mult case solutions most of these parameters may be applied separately within each case (see Section 3.2.1).
Table 2-10. – Sierra/SD Solution Options.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Type</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>restart to change the restart input file can be given to read a restart</td>
<td>none</td>
<td>read</td>
<td>write</td>
</tr>
<tr>
<td>lumped</td>
<td>&lt;string&gt;</td>
<td>auto</td>
<td>lumped_consistent</td>
</tr>
<tr>
<td>solver</td>
<td>scattering</td>
<td>false</td>
<td>true</td>
</tr>
<tr>
<td>symmetrize...</td>
<td></td>
<td>false</td>
<td></td>
</tr>
</tbody>
</table>

2.4.1. Flush

The parameter flush controls how often the Exodus output file buffers should be flushed. Flushing the output ensures that all the data that has written to the file buffers is also written to the disk. This parameter also controls the frequency of output of restart information if requested. Too frequent buffer flushes can affect performance. However, in a transient run, data integrity on the disk can only be assured if the buffers are flushed. A flush value of -1 will not flush the Exodus output file buffer until the run completes. The default value is to flush the buffers every 50 time steps.

2.4.2. Restart – option

Option restart controls restart file // 48:202 processing. Restart files permit the solution to be saved for later use. Only a limited capability is provided, but it is intended to meet most of the typical needs for structural dynamics. Note that except for eigen restarts, the restart files are independent of the Exodus output, but the restart options may significantly affect the Exodus outputs. Eigen restarts directly use the Exodus output. Application of restarts in specific sections is detailed in the following paragraphs. See Figure 2-4 for details on eigen restart.

There are four values associated with this option.
none indicates that restart files will be ignored. They will be neither read, nor written.
Existing restart files will not be altered in any way. Restart=none is the default selection if no restart options are entered in the solution block.

read indicates that existing restart files will be read, but no output restart files will be written. If the restart files do not exist, a fatal error will result.

write indicates that existing restart files will be ignored, but restart files will be written.

auto is a combination of read and write. However, unlike read, the existence of previous restart files is optional, i.e. there will be no error message if there are no existing restart files. Invalid restart files will produce a warning, but not a fatal error.

The restart read and auto options can take an optional from clause to define the file containing the restart data. Examples are given below.

Restarts are designed to ensure accuracy of the solution. However, restarts in Sierra/SD are not transparent in the sense that there will be small differences in two solutions to a problem when one solution involves a restart. Restarts may also have an expense. For example, GDSW uses an acceleration technique where the values of previous solutions are used as a starting place for new solves. The information associated with previous solutions is not stored in the file. Also, restarts discard Lagrange multipliers.

For transient dynamics, the state of the machine at the most recent time step is recorded. To avoid problems with corruption of a database, the three nodal vectors (disp, velocity, acceleration) and applicable element data are recorded at each time step, but on alternate locations in the file. If previous Exodus files exist, they will be appended. Data is written at the same interval as the Exodus output.

When restarting a multicasa solution, the current time is used to determine which case the restart will begin. For example, assume the following solution block is defined.

```plaintext
Solution
  case one
    transient
    restart=auto
    time_step 1e-6
    nsteps 200
  case two
    transient
    restart=auto
    time_step 1e-5
    nsteps 300
End
```

If restarting at time=1E-4, case “one” has a final time value of \( T_F = T_0 + 200 \times 1E-6 = 2E-4 \), assuming \( T_0=0 \). Since \( \text{time} < T_F \), case ‘one’ will restart the solution. If restarting
at \textit{time=}2E-4, case ‘one’ will not perform any calculations and restart will begin with case ‘two’.

An important note for restarts is care must be taken to ensure the restart files are given the expected names. By default, restart file names are defined based on the geometry file and the case name. The following example uses the default behavior and will write a restart file with the name \texttt{mesh-one.rst_trans}.

```plaintext
Solution
  case one
    transient
    restart=write
    time_step 1e-6
    nsteps 200
  End
File
  geometry_file = 'mesh.g'
  End
Output
End
``` 

The default restart (read) input deck would then be as follows. The mesh name and case name will be used to define the restart input file name \texttt{mesh-one.rst_trans}.

```plaintext
Solution
  case one
    transient
    restart=read
    time_step 1e-6
    nsteps 200
  End
File
  geometry_file = 'mesh.g'
  End
``` 

The default restart file names work fine if you are running a model multiple times in one directory, with the same number of processors, say to computing additional transient time steps or eigenvalues with each restart. Alternatively a \texttt{from} clause can be used with the \texttt{read} or \texttt{auto} restart options to explicitly define the restart input file name.

It is possible to read a restart file that is decomposed for a different number of processors than are being used for the run by setting \texttt{num_procs}. Restart files will always be written using the same base name and paths as the \texttt{outputs} files.

For the following example assume the analysis run is submitted on 4 processors. The combination of the restart \texttt{from} and \texttt{num_procs} clauses will explicitly set the name and path from which to read the parallel restart files to
As the output database name is set Sierra/SD will write restart to the file name 'output/my_new_file.rst_trans.4.0', 'output/my_new_file.rst_trans.4.1', 'output/my_new_file.rst_trans.4.2', and 'output/my_new_file.rst_trans.4.3'. Restart and output files will always be written decomposed on the same number of processors that are being used by the analysis.

The restart options from and num_procs can be used to execute the common use case of N to M restart. For example it can be efficient to execute an expensive eigen decomposition once on hundreds of processors then read that eigen decomposition on a handful of processors to do many subsequent modal transient or modal random vibration analyses.

2.4.2.1. Restart solution support

Restarts are not supported in all solutions types. They are supported for the following.

- Eigen
- transient
- NLtransient
- modaltransient
- QEVP (with Anasazi and sa_eigen methods)

Note that none of the modal solutions except modaltransient support restart. Typically most of the computation time for these solutions is in the eigen analysis. It is recommended that the multicase solution be used, with a restart in the eigen analysis portion of the solution.
Table 2-11 illustrates the current supported restart options for transient analysis. As shown here, one can restart a modal transient analysis from a transient analysis, and vice versa.

<table>
<thead>
<tr>
<th>Integrator</th>
<th>NLtransient</th>
<th>transient</th>
<th>modaltransient</th>
</tr>
</thead>
<tbody>
<tr>
<td>NLtransient</td>
<td>Tested</td>
<td>?</td>
<td>?</td>
</tr>
<tr>
<td>transient</td>
<td>Tested</td>
<td>Tested</td>
<td>Tested</td>
</tr>
<tr>
<td>modaltransient</td>
<td>Tested</td>
<td>Tested</td>
<td>Tested</td>
</tr>
</tbody>
</table>

**Table 2-11.** – Supported restart capabilities for transient integrators in Sierra/SD.

If the QEVP solution restarts at all, then every requested mode must be provided. Therefore, you cannot ask for more modes than are in the restart file and have Sierra/SD calculate only the additional modes. You can however save previously calculated modes for use in follow on solution methods.

All restart files are in the exodusII format. This permits evaluation and manipulation of the data using standard tools. Details of the file names and formats used for restart are included in Table 2-12.

The eigen solution case avoids the complexity of writing a special restart file. Instead the Exodus output is reused. Restart does require that originally displacements were output to the Exodus file. However, the use of the standard file format facilitates error diagnosis. Reuse of the Exodus output does impact the options available as follows.

| NONE       | ensures that no data will be read from existing Exodus files. However, output, written in the normal fashion, will replace existing files. As a consequence, the results of the previous analysis will be modified. |
| auto       | Data is read from available input files. After a run, data is written to the appropriate files. They will be updated to include additional calculations. |
| READ       | Read from the output of the previous modal or transient analysis. In the case of a transient analysis, however, no output is written. Also, if no previous transient analysis wrote a restart file, then there will be nothing to read in this case. |
| WRITE      | No restart files are read. Results are output after analysis. For modal analyses, this option is effectively identical to none. For transient analyses, however, either write or auto is required to write a file that can be used to restart. |

**Figure 2-4.** – Notes on Eigen Restart.

The auto option makes it difficult to know whether or not restart occurred. To clarify if restart is being used, it is best to explicitly define the restart read, write, none options.

2.4.2.2. Restarts with virtual nodes and elements Currently, Sierra-SD creates virtual nodes and elements when a model has features like Tied Joints, infinite elements, and
superelements. These virtual nodes and elements are created internally during execution of the code, and result in the analysis having more degrees of freedom than are specified in the Exodus file.

When the restart files described in Table 2-12 are written to disk, these virtual elements and nodes will also be written, resulting in Exodus files that have more nodes and elements than were in the original Exodus file. The restart Exodus output files can be visualized in Ensight and the data can be processed in MATLAB. Thus, the restart files provide this additional benefit of being able to post-process the virtual nodes and elements. For example, one may wish to visualize the Rbars created for a Tied Joint in Ensight. This can be done by visualizing the restart files.

2.4.3. Solver

As Sierra/SD evolves, various solvers are available for computation of the solution. Each solver brings with it different capabilities and sometimes unwanted features. Currently, available solvers are listed in the following.

<table>
<thead>
<tr>
<th>Solution</th>
<th>file name</th>
<th>Details</th>
</tr>
</thead>
<tbody>
<tr>
<td>eigen</td>
<td>example-out.exo</td>
<td>Use the standard Exodus output for restart. Displacements must be written or no restart is possible. Other variables (such as strain energy) may also be written.</td>
</tr>
<tr>
<td>qevp</td>
<td>example-out.exo</td>
<td>Uses standard Exodus output for restart. No additional modes may be computed, but the superposition is possible.</td>
</tr>
<tr>
<td>transient</td>
<td>example-out.rst_trans.exo</td>
<td>The two most recent time steps are written. They are only written at the “flush” interval.</td>
</tr>
<tr>
<td>NLtransient modaltransient</td>
<td>example-out.rst_trans.exo</td>
<td></td>
</tr>
</tbody>
</table>

Table 2-12. – Restart file format and names. Details of the restart files for a file named “example.exo” for various solutions.
constraints. The GDSW solver is currently under development and supported by the Sierra-SD team. The most recent development efforts for the GDSW solver have been focused on implementation and testing of a new Helmholtz solver for direct frequency response analysis.

Generally no user input is required for specification of a solver. Indeed, up to version 1.0.5 of Sierra/SD, only one solver was ever available at any time (i.e. we built separate executables if another solver was desired). Usually the specification can be left off, or specified as auto. If a solver is requested and unavailable, a warning will be issued, and auto will be selected.

The solver may be specified as a default (above the case keywords as detailed in Section 3.2.1), or it may be individually specified within the case framework. The default value is auto. In the example shown below GDSW will be used for the eigen analysis, and the auto selection for the for transient dynamics and direct frequency response. If “input_summary” is specified in the “ECHO” section (see Section 7.7) then the solver information will be echoed to the results file.

```
SOLUTION
  solver=auto
  case eig
    eigen nmodes=50
    solver=gdsw
  case nltransient
    NLtransient
    (other parameters)
  case frf
    directFRF
END
```

2.4.3.1. Limitation There is a limitation of restarts that the number of processors (i.e. MPI ranks) used to restart must be the same as used originally. There is one exception however. For the eigen solution method, it is possible to restart on a single processor. Beforehand, the distributed output file must be joined into a single file. Similarly for a transient solution, it is possible to restart in serial. In this case both the distributed output files and the distributed restart file must be joined into single files.

2.4.4. Lumped – option

To use a lumped mass matrix for all solution cases add option lumped anywhere in the Solution section. Most off diagonal terms are set to zero. The diagonal terms are increased to conserve mass.
The drilling degrees of freedom associated with beams and shells can generate spurious modes when they are lumped. As a consequence, Sierra/SD does not fully lump these degrees of freedom. They are lumped in the element coordinate frame, but transforming the mass matrix to the physical coordinates results in a $3 \times 3$ matrix.

2.4.4.1. Computational Acoustics In acoustics simulations with hexahedrons, in theory a special mass matrix minimizes dispersion error. Adding `lumped_consistent` to the Solution selects the special mass matrix. Many have conjectured that the special mass matrix enhances simulations of linear elasticity. The CFL number $c \, dt/dx$ depends on the sound speed, $c$, the element size, $dx$, and the time step size, $dt$. A study of the `lumped_consistent` mass matrix reached the following conclusions.

- In the most accurate simulations, time step size is scaled with mesh size to maintain a constant CFL number.
- Simulation accuracy depends on the temporal (e.g. Newmark) as well as the spatial (finite element) discretization.
- A `lumped_consistent` mass matrix can enhance accuracy.
- With Newmark Beta time integration, the most accurate simulations used a consistent mass matrix and a CFL near 7/10.

2.4.5. ConstraintMethod – option

The ConstraintMethod option is defined in the Solution section to indicate how multipoint constraints (MPC) will be applied. MPCs applied using Lagrange multipliers. Inverse problems sometimes use an experimental penalty method.

2.4.6. Scattering – option

For some acoustics and structural acoustics problems, it is advantageous to define the loads in terms of an incident pressure instead of a total pressure. The solutions for the scattered pressures follow the same differential equations as those of the total pressures. It may be necessary to combine the incident and scattered terms to compute a total pressure. A review is presented in subsubsection Acoustic Scattering subsection Acoustic and Structural Acoustic Boundary Conditions section Acoustics and Structural Acoustics of the Theory Manual Note that the scattering keyword applies to all loads in the solution case. It is nonsensical to mix scattering pressure inputs with total pressure inputs.

Scattering solutions require this keyword in the solution block. In addition, loads should be applied properly in the LOADS block. The user must apply a load to both the structural
and the acoustic side of a wet surface. This is typically done using a function tailored for that purpose.

### 2.4.6.1. symmetrize_struc_acous – option

By default, coupled structural acoustic discretizations are symmetric. This is accomplished by scaling the acoustic equation by a factor of -1. In some cases (e.g., infinite elements) scaling is impossible, and cases the code internally reverts to the nonsymmetric formulation. The nonsymmetric formulation is always available by setting `symmetrize_struc_acous` to false. In a multicase solution, `symmetrize_struc_acous` can vary from case to case. The pros and cons of two approaches have never been studied.

### 2.5. GDSW

GDSW is the workhorse for parallel solutions. It is the default linear solver on two or more processors. Many Sierra/SD features require that GDSW be the linear solver. In this manual, “the solver” is the linear solver. This section describes the GDSW parameters. Table 2-13 describes the basic solver parameters. Parameters for advanced usage are given in Tables 2-14 and Table 2-15. Report problems using the GDSW solver with the default solver parameters to the Sierra help system at `sierra-help@sandia.gov`.

To use a non-default linear solver (i.e., GDSW in serial) in the solution section set `solver=GDSW`. Non-default parameters are set in an optional GDSW section. The GDSW section applied to each solution case. The first subsection of this Section 2.5.1 presents a way to specify different GDSW parameters for each solution case. Subsection 2.5.1 describes additional parameters. Subsection 2.5.2 describes GDSW output. The corresponding parameters are summarized in Table 2-16. Subsection 2.5.3 describes some techniques for improving performance and reliability. Subsection 2.5.4 reviews theory related to solution accuracy. The GDSW Helmholtz solver is a relatively new capability. Subsection 2.5.5 presented the parameters for the parallel frequency response linear solver. The parameters are summarized in Table 2-19. Report any problems using the new Helmholtz solver to `sierra-help@sandia.gov`.

- **solver_tol** It is important to control the accuracy of the solution. For all our linear solvers, `solver_tol` is the requested accuracy of the computed solution as measured by the relative residual error. In other words, the 2-norm of the residual vector for the computed solution divided by the 2-norm of the right-hand side force vector should be no greater than `solver_tol`.

- **orthog** Convergence may be accelerated by storing and recycling search directions from previous solves. This feature requires additional memory, but may significantly reduce iterations. It is possible for the application of these vectors to be unstable.

---

1 the “plane_wave”, “planar_step_wave” and “shock_wave” functions compute both appropriate pressures on the structure, and normal velocities on the acoustic medium. See sections 2.8.10, 2.8.12 and 2.8.14.
Table 2-13. – GDSW Section Options. (Basic)

<table>
<thead>
<tr>
<th>Variable</th>
<th>Values</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>max_iter</td>
<td>integer</td>
<td>1000</td>
<td>maximum number of iterations</td>
</tr>
<tr>
<td>solver_tol</td>
<td>real</td>
<td>1e-6</td>
<td>relative residual convergence tolerance</td>
</tr>
<tr>
<td>overlap</td>
<td>integer</td>
<td>2</td>
<td>number of layers of overlapping nodes for preconditioner</td>
</tr>
<tr>
<td>orthog</td>
<td>integer</td>
<td>1000</td>
<td>number of stored search directions used to accelerate solver convergence (also see num_vectors_keep)</td>
</tr>
<tr>
<td>prt_summary</td>
<td>integer</td>
<td>3</td>
<td>output flag: 0 - no summary 1 - summary 3 - detailed summary</td>
</tr>
</tbody>
</table>

This rare event is avoided by trying orthog=0. The default value of orthog is 1000. For the Helmholtz linear solver the corresponding parameter is orthogH.

**krylov_method** A variety of Krylov iterative methods are available as shown in Table 2-14. The default should work fine in most instances. If convergence problems arise, switching to classic right preconditioned GMRES is recommended. (krylov_method = gmresClassic) without the use of any stored search directions (orthog = 0).

**default_solver** The subdomain sparse direct linear solver is specified either by name (Esmond, Pardiso, NoPivot) or by the corresponding integer (1, 3, 6).

**num_rigid_mode** note: see parameters, Section 2.3. This keyword must not appear in the GDSW solver section. It is read from the Parameters section.

**constrain_rbms** Tells GDSW to numerically constrain rigid body modes of a structure in order to do a static solution of a free floating structure. Must be used in conjunction with the FilterRBMLoad parameter to ensure no net rigid body load on the constrained structure. The rigid body modes associated with x, y, z, rotx, roty, rotz can be selectively constrained, as can p the pressure degree of freedom in acoustic analysis. See Section 6.3.19 for more details on this use case.

**max_numterm_C1** Constraints for the GDSW solver are classified by two types:

**Type 1**: simple constraints like those applied by an Rbar, tied contact, or rigid surfaces.

**Type 2**: more complex, averaging constraints like those in an RBE3.

Type 1 constraints have few terms, and Type 2 constraints have many terms in each constraint equation. Solution of problems with Type 2 constraints using Type 1 methods is possible and desirable if they are small enough, but the memory requirements could be prohibitive if the number of terms N in any constraint equation is too large. Specifically, storage of a dense matrix with at least $N^2$ terms
<table>
<thead>
<tr>
<th>Variable</th>
<th>Values</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>krylov_method</td>
<td>integer</td>
<td>1</td>
<td>0-pcg: preconditioned conjugate gradients</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>1-GMRES: right preconditioned GMRES (generalized conjugate residual version)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>2-lgmres: left preconditioned GMRES</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>3-flexgmres: flexible right precond GMRES</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>4-flexgmres2: variant of flexgmres</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>5-gmresClassic: right preconditioned</td>
</tr>
<tr>
<td>default_solver</td>
<td>integer</td>
<td>1</td>
<td>1-direct: Esmond Ng’s sparse direct solver</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>3 - Pardiso for Pardiso sparse direct solver (Intel MKL only), 6-NoPivot:</td>
</tr>
<tr>
<td>num_rigid_mode</td>
<td>x y z</td>
<td></td>
<td>Enables solution of static system with rigid body modes</td>
</tr>
<tr>
<td>constrain_rbms</td>
<td>rotx</td>
<td></td>
<td></td>
</tr>
<tr>
<td>max_numterm_C1</td>
<td>integer</td>
<td>250</td>
<td>max num terms for Type 1 constraints</td>
</tr>
<tr>
<td>coarse_option</td>
<td>integer</td>
<td>1</td>
<td>0 - additive coarse correction, 1 - multiplicative coarse correction</td>
</tr>
<tr>
<td>SC_option</td>
<td>integer</td>
<td>1</td>
<td>0-no/1-yes: eliminate subdomain interior unknowns using static condensation</td>
</tr>
<tr>
<td>weight_option</td>
<td>integer</td>
<td>2</td>
<td>1 - to not use weighted residuals for overlapping subdomain problems</td>
</tr>
<tr>
<td>coarse_size</td>
<td>string</td>
<td>auto</td>
<td>coarse space reduction option</td>
</tr>
<tr>
<td>reorder_method</td>
<td>string</td>
<td></td>
<td>auto: automatic selection</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>small: use reduced coarse space</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>large: use larger coarse space</td>
</tr>
<tr>
<td>num_GS_steps</td>
<td>integer</td>
<td>1</td>
<td>number of Gram-Schmidt orthogonalization steps for stored search directions</td>
</tr>
<tr>
<td>con_tolerance</td>
<td>real</td>
<td>2.5e-9</td>
<td>singularity tolerance for processing constraints</td>
</tr>
<tr>
<td>con_row_tolerance</td>
<td>real</td>
<td>0.1</td>
<td>pivoting tolerance for processing constraints</td>
</tr>
<tr>
<td>scale_option</td>
<td>integer</td>
<td>0</td>
<td>0 - no scaling in factorizations</td>
</tr>
<tr>
<td>diag_scaling</td>
<td>string</td>
<td>none</td>
<td>1 - use scaling in factorizations</td>
</tr>
<tr>
<td>PTAP_solver</td>
<td>integer</td>
<td>1</td>
<td>no - no scaling of operator matrix</td>
</tr>
<tr>
<td>bailout</td>
<td>keyword</td>
<td></td>
<td>diagonal - symmetric diagonal scaling</td>
</tr>
<tr>
<td>coarsening_ratio</td>
<td>integer</td>
<td>1000</td>
<td>If keyword is found, ignore errors</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>coarsening ratio for multilevel solver</td>
</tr>
<tr>
<td>Variable</td>
<td>Values</td>
<td>Default</td>
<td>Description</td>
</tr>
<tr>
<td>----------------------------------</td>
<td>----------</td>
<td>---------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>minCoarseLevels</td>
<td>integer</td>
<td>1</td>
<td>min number of coarse levels (for testing only)</td>
</tr>
<tr>
<td>maxCoarseLevels</td>
<td>integer</td>
<td>1</td>
<td>max number of coarse levels</td>
</tr>
<tr>
<td>maxCoarseSize</td>
<td>integer</td>
<td>3000</td>
<td>max size for coarsest problem</td>
</tr>
<tr>
<td>enforceActualResidual</td>
<td>integer</td>
<td>0</td>
<td>0-no/1-yes</td>
</tr>
<tr>
<td>graphPartitioner</td>
<td>integer</td>
<td>0</td>
<td>graph partitioner for multilevel solver</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>0-Parmetis, 1 PHG in Zoltan</td>
</tr>
<tr>
<td>num_vectors_keep</td>
<td>integer</td>
<td>900</td>
<td>related to orthog; see discussion below</td>
</tr>
<tr>
<td>stag_tol</td>
<td>real</td>
<td>0.01</td>
<td>Used to detect stagnation</td>
</tr>
<tr>
<td>dd_solver_output_file</td>
<td>string</td>
<td>dd_solver.dat</td>
<td>Output name for domain decomposition solver diagnostic file</td>
</tr>
<tr>
<td>krylov_solver_output_file</td>
<td>string</td>
<td>krylov_solver.dat</td>
<td>Output name for Krylov solver diagnostic file</td>
</tr>
<tr>
<td>useParallelDirectSolver</td>
<td>integer</td>
<td>0</td>
<td>0-no/1-yes</td>
</tr>
<tr>
<td>useParallelCoarseSolver</td>
<td>integer</td>
<td>0</td>
<td>0-no/1-yes</td>
</tr>
<tr>
<td>numCoarseProcs</td>
<td>integer</td>
<td>all available</td>
<td>number of processors for the coarse problem</td>
</tr>
<tr>
<td>useSuperLUDist</td>
<td>integer</td>
<td>0</td>
<td>0-no/1-yes: whether to use SuperLU-Dist</td>
</tr>
<tr>
<td>numProcRowSuperLUDist</td>
<td>integer</td>
<td></td>
<td>number of rows in SuperLU-Dist process grid</td>
</tr>
<tr>
<td>identify_low_quality_elements</td>
<td>bool</td>
<td>false</td>
<td>identify poorly shaped elements for special attention by solver</td>
</tr>
<tr>
<td>max_element_condition</td>
<td>double</td>
<td>Infinity</td>
<td>condition above which to consider element low quality</td>
</tr>
</tbody>
</table>
would likely be required. The parameter `max_numterm_C1` specifies the maximum number of terms that can appear in a Type 1 constraint following a constraint pre-processing step. Constraints with more than `max_numterm_C1` terms are then considered to be Type 2. The algorithm used to enforce Type 2 constraints in the preconditioner is generally not as efficient as the one for Type 1 constraints. If feasible, avoiding Type 2 constraints by increasing the `max_numterm_C1` in conjunction with a multiplicative coarse correction (`coarse_option=1`) will lead to a more efficient solution.

Avoiding Type 2 constraints generally reduces run times. To eliminate Type 2 constraints set the GDSW option `max_numterm_C1` to a sufficiently large number. The value that GDSW is using for `max_numterm_C1` is in the `dd_solver.dat` file. It is called `maxNumTermsForType1Constraints`. Also, in the `dd_solver.dat` file note the value of `maxNumNonZeros in Tran Matrix`. To eliminate Type 2 constraints `max_numterm_C1` must be larger than the value of `maxNumNonZeros in Tran Matrix`.

`coarse_size` is used to specify a reduction strategy for the coarse problem size. There is no need to consider this parameter for problems run on fewer than a few hundred processors. However, as the number of processors (subdomains) becomes large, solving the coarse problem can become a bottleneck. The default (`auto`) automatically selects to use the `small` coarse space only if the number of processors exceeds 1000. Specifying a `small` rather than a `large` coarse space can often reduce the amount of memory needed by the solver.

`reorder_method` Allows one to specify a reordering method for a sparse direct solver. Currently, it is only available for `default_solver = direct` (see Table 2-14).

`num_GS_steps` is the number of orthogonalization steps of the stored search directions. Sometimes increasing it from its default to 2 is helpful. I know of no case of a value larger than 2 being necessary. It turns out that `num_GS_steps` does not apply to `gmresClassic`.

`con_tolerance` The GDSW solver uses a sparse LU decomposition algorithm to process the constraint equations. This involves choosing pivot rows for numerical stability (much like Gaussian elimination with partial pivoting). A constraint equation is deemed linearly dependent if the magnitude of its pivot is less than `con_tolerance`. The `con_tolerance` is the minimum acceptable ratio of the constraint pivot to the maximum matrix pivot. The `con_tolerance` can viewed as a dimensionless parameter associated with numerical round off. The number of numerically redundant constraints in a model will typically be reduced as the `con_tolerance` is increased. Excessively high `con_tolerance` values could start to remove legitimate constraints.

Messages of the form,

```
min/max pivot for constraint factorization = some number
You may want to consider increasing the con_tolerance parameter in the GDSW solver block.
```
are issued if the ratio of magnitudes of the smallest to largest pivots is less than 0.01. This provides a recommendation to carefully examine the constraints in the model for any potential problems. Additionally, a message will be written if there exists constraints on the cusp of being removed as redundant. Again presence of such nearly redundant constraints should be investigated closely for correctness.

Redundant or nearly redundant constraints can be generated in a variety of circumstances. For example a closed loop of Rbars, Rbars between nodes of a rigid set, or overlapping rigid sets. Although the SD constraint filtering algorithm is designed to detect and handle such redundancies it is strongly recommended that redundant constraint be avoided in model definition as much as possible. See Section 1.9 for more details.

**scale_option** There are presently two options for matrix scaling in the GDSW solver. Including `scale_option yes` or, equivalently, `scale_option 1` in the GDSW solver block will apply symmetric diagonal scaling to all matrices prior to them being passed to Esmond Ng’s sparse direct solver. Notice for parallel runs that both the subdomain matrices and the coarse problem matrix will be scaled. In exact arithmetic, this option should have no effect on the number of iterations for each solve of a parallel run.

**diag_scaling** Including `diag_scaling diagonal` in the solver block will apply symmetric diagonal scaling to the original operator matrix and is not tied to a specific sparse direct solver. In contrast to the scale_option parameter, this parameter changes the number of iterations for each solve of a parallel run since GDSW is solving the scaled problem $DADy = Db$ to a specified relative residual tolerance rather than the original problem $Ax = b$ (note substitution of $x = Dy$, where $D$ is a diagonal scaling matrix) for that same tolerance.

**coarsening_ratio** Is a target ratio between the number of subdomains prior to and after coarsening by the multilevel solver. For example, if there are originally 8000 subdomains (processors) and `coarsening_ratio` is chosen as 100, then the number of subdomains after coarsening will be 80.

**maxCoarseLevels** Is the maximum number of coarse levels allowed by the multilevel solver. For a standard 2-level method this parameter has a value of 1.

**maxCoarseSize** Is the largest size for the coarsest problem allowed before another level of coarsening is made. The solver parameter `maxCoarseLevels` takes precedence over `maxCoarseSize`.

**enforceActualResidual** This option is used to enable strict enforcement of the solver tolerance. If needed, iterative refinement steps are taken within the solver in an attempt to satisfy the convergence criterion. It should be noted that it may not be possible to drive the relative residual to values no greater than the solver tolerance. This can happen when the problem is poorly conditioned and/or the requested solver tolerance is too small. An error message is issued and the analysis stops if the specified solver tolerance cannot be achieved.
**graphPartitioner** Specifies which graph partitioning software to use when coarsening the subdomains.

**num_vectors_keep** applies to the classic version of GMRES (krylov_method gmresClassic). The parameter orthog (default 1000) controls the number of stored search directions. We store search directions in order to make the linear solver faster. Generally more is better but not always. The point to understand is which search directions are stored. The first 1000 search directions are stored. On later solves, the first 900 are saved and recycled. 100 search directions from the current solve are used. The number 900 is the default value of num_vectors_keep. Sometimes the solution has changed significantly and none of the old search help the solver. num_vectors_keep = 0 tells GDSW to never recycle any of the stored search directions between solves.

**num_GS_step_gmres** is the parameter for the Krylov method gmresClassic corresponding to the num_GS_steps for other Krylov methods.

**useParallelDirectSolver** Whether to use parallel sparse direct solver for the whole problem. Best suited to directfrf solution case.

**useParallelCoarseSolver** Whether to use a parallel sparse direct solver for the coarse problem. This can be helpful whenever the coarse problem is large enough that its single-processor solution becomes a bottleneck.

**numCoarseProcs** Number of processors to use if useParallelCoarseSolver = yes.

**useSuperLUDist** Whether to use SuperLU-Dist parallel sparse direct solver even if Cluster Pardiso is available. Applies to both the full problem (useParallelDirectSolver and coarse problem (useParallelCoarseSolver) cases.

**numProcRowSuperLUDist** SuperLU-Dist uses a rectangular grid of processes. This option allows the user to override the default choice, which is to make the number of rows about equal to the number of columns. The parameter is ignored unless it divides the total number of processors. It has been suggested that for large problems, the number of columns should be larger than the number of rows, but we have not yet tested cases where this choice proved to be beneficial.

The generalized conjugate residual (GCR) version of right preconditioned GMRES is the default Krylov method. GCR has the limitation that orthog must be at least as large as max_iter. On the other hand with gmresClassic max_iter and orthog may be specified independently.

### 2.5.1. Options

The GDSW section specifies GDSW linear solver parameters for all the solution cases. Different solution cases may call for different parameters. For example, an eigenvalue problem may use a shift parameter to eliminate rigid body modes, but a statics analysis cannot shift. Linear solver parameters for an individual solution case are set using the
solver_options keyword in the solution case, and adding the corresponding solver_options section.

The solver_options sections allow multiple definitions of the parameter. Each “solver_options” section may be called out from a separate case in a multicase solution block as illustrated in Input input 2.1. A “solver_options” section applies only to the GDSW solver.

Note that cases using a solver_options section will ignore any “global” options defined in the GDSW section, and will instead use the internal GDSW defaults for any options that aren’t explicitly defined.

```
Solution
  case preload
    statics
    solver_options for_preload
      load=10
  case eig
    eigen
      nmodes=100
      solver_options for_eig
End
Solver_options for_preload
  solver_tol = 1e-4
  constrain_RBMs="x y z"
End
Solver_options for_eig
  solver_tol = 1e-8
End
```

Input 2.1. Multiple Solver Options Example

### 2.5.2. Diagnostics

There is a variety of solver diagnostic information output to the files dd_solver.dat and krylov_solver.dat. Here we describe six different measurements of solution accuracy.

Note: the default names of the dd and Krylov solver diagnostic files (“dd_solver.dat” and “krylov_solver.dat”) can be overridden by the dd_solver_output_file and krylov_solver_output_file options in the GDSW and solver_options sections.

**actual final residual** Let’s say we want to solve the linear system $Ax = b$ for the vector of unknowns $x$ given the right-hand side vector $b$. If $x_a$ is the approximate solution from the solver, then the actual final residual is $\|b - Ax_a\|$, where $\|b\|$ denotes the 2-norm of $b$. The actual final residual is reported in krylov_solver.dat after each solve.
**actual relative residual**  This is $\|b - Ax_a\|/\|b\|$ and shows up in column 6 of dd_solver.dat. In other words, this is the actual final residual divided by the norm of right-hand side vector $b$.

**recursive final residual**  During GMRES or CG iterations, we can recursively calculate $b - Ax_a$ without having to directly calculate $Ax_a$ (for efficiency reasons). In exact arithmetic the actual and recursive finals residuals are identical, but in practice they can be different because of round off errors. If the effects of round off are not too big, then the actual and recursive final residuals should be close. The recursive final residual is reported after each solve in krylov_solver.dat.

**recursive relative residual**  This is the recursive final residual divided by $\|b\|$ and shows up in column 5 of dd_solver.dat.

**constraint error residual**  The constraint equations for a problem can be expressed as $Cx = 0$, where $C$ is the constraint matrix. The constraint error residual is a normalized measure of $Cx_a$ and should be small relative to 1 if the approximate solution $x_a$ satisfies the constraints well. This residual is reported after each solve in krylov_solver.dat.

**equilibrium error**  For problems with constraint equations, we solve the linear system

$$
\begin{bmatrix}
A & C^T \\
C & 0
\end{bmatrix}
\begin{bmatrix}
x \\
\lambda
\end{bmatrix} = \begin{bmatrix} b \\
0
\end{bmatrix},
$$

where $x$ is the vector of unknowns, $\lambda$ is the vector of Lagrange multipliers, and $C^T$ denotes the transpose of the constraint matrix $C$. For the approximate solution vector $x_a$, the equilibrium error is $\|b - Ax_a - C^T\lambda\|$ and is reported after each solve in krylov_solver.dat. In the absence of round off errors, the equilibrium error and the actual final residual should be identical.

All these residual measures may be more than is usually of interest, but they can provide valuable information for cases when solver convergence is an issue.

Picking a suitable solver tolerance for GDSW or any other iterative solvers requires close attention. If the solver tolerance is too high, then simulation results may not have sufficient accuracy. Likewise, if the solver tolerance is too low, then more analysis time may be spent obtaining a needlessly over-accurate solution. Solving a problem with two or more values for the solver tolerance can be useful to help avoid unnecessarily accurate solves and to also ensure that the solves are accurate enough. For example, let’s say you do a modal analysis with a solver tolerance of 1e-4 and 1e-5. If you don’t see a concerning change in the modal frequencies, then either choice for the solver tolerance is probably fine for the modal frequencies themselves. If, however, the modal solution is used as the basis for a subsequent analysis such as modal transient, then it is recommended that the effects of solver tolerance on the final results also be considered. Similar comments regarding the choice of a solver tolerance also hold for other solution cases such as direct transient analysis.

We could provide users with estimates for how small a solver tolerance is needed to guarantee a certain measure of accuracy, but these estimates would usually be way too pessimistic to be of any practical value.
Additional details and troubleshooting strategies for the GDSW solver can be found in §2.5.3 and documentation available on the compsim.sandia.gov website. Relevant documentation includes GDSW 101 and the GDSW Solver Tutorial. Solver strategies for dealing with poor mesh decompositions caused by the presence of constraints equations or multiple physics (i.e. structural-acoustics problems) are described in the GDSW Solver Tutorial. These include rebalancing algorithms internal to the solver that can be accessed using GDSW solver parameters. We hope this will provide a useful interim solution for challenging problems prior to the deployment of alternative decomposition tools that effectively address these issues prior to the solution phase. Input 2.2 provides recommended options for minimum memory use.

```
GDSW
  overlap=0
  max_iter=50       // set to minimum required for a solution
  krylov_method = gmresClassic
  orthog=0
  precision_option_O=single
  precision_option_coarse=single
END
```

Input 2.2. Minimum Memory Recommended Options. These options, while not usually optimal for speed, may use the lowest memory.

<table>
<thead>
<tr>
<th>Variable</th>
<th>Values</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>prt_coarse</td>
<td>integer</td>
<td>0</td>
<td>0-no/1-yes: print coarse matrix</td>
</tr>
<tr>
<td>prt_constraint</td>
<td>integer</td>
<td>0</td>
<td>0-no/1-yes: print constraint matrix</td>
</tr>
<tr>
<td>prt_memory</td>
<td>integer</td>
<td>0</td>
<td>0-no/1-yes: print memory information</td>
</tr>
<tr>
<td>prt_timing</td>
<td>integer</td>
<td>0</td>
<td>0-no/1-yes: print timing information</td>
</tr>
<tr>
<td>prt_interior</td>
<td>integer</td>
<td>0</td>
<td>0-no/1-yes: print interior matrices</td>
</tr>
<tr>
<td>prt_overlap</td>
<td>integer</td>
<td>0</td>
<td>0-no/1-yes: print overlap matrices</td>
</tr>
<tr>
<td>write_orthog_data</td>
<td>integer</td>
<td>0</td>
<td>0-no/1-yes: write orthogonalization data to file</td>
</tr>
</tbody>
</table>

2.5.3. Troubleshooting

Ideally, the linear solver should always return a solution that satisfies the requested accuracy in terms of the relative residual tolerance ($10^{-6}$ by default). This subsection provides some troubleshooting guidelines for situations when this is not the case. Additional information is available at compsim.sandia.gov in the Sierra/SD online documentation *GDSW Solver Tutorial* and *GDSW 101*. If problems persist, please submit
a Sierra help ticket or reach out to a member of the Sierra/SD development team for assistance.

- **Default Solver Parameters:** In case of solver convergence problems, it is recommended that one first verify that the default solver parameters do not work. A notable exception is for problem types like structural acoustics where a smaller solver tolerance than the default may be needed to obtain accurate structural and acoustic responses.

- **Negative Shift for Modal Analyses:** A common mistake is to not specify a negative shift for modal analyses of structures with no essential boundary conditions. If one or more rigid body modes are present, then the stiffness matrix will be singular and the solver will likely have problems converging except in special cases (see Singular Solves bullet below). The recommended shift is $-(2\pi f)^2$, where $f$ is an estimate of the natural frequency (in Hz) of the first flexible mode. Of course the modes are not known in advance. The point is that the shift does not need to be an accurate estimate. For modeling an ordinary (larger than a paper clip, smaller than a house) steel component in Imperial units $-10^6$ almost always works. Caution: specifying a negative shift that is too large in magnitude may help the linear solver, but it can cause the algorithm that solves the eigenvalue problem to either require too many linear solves or not converge at all.

- **Memory Considerations:** The linear solver requires that enough memory be available to store the factorizations of subdomain and coarse space matrices. If the subdomain or coarse matrices are too large, then the memory capacity of the computing resource will be exceeded and a run will fail. There are a number of different ways to address such problems. First, one may request a smaller number of processors per compute node. For example, the CTS-1 machine *eclipse* has 36 cores per compute node, but one may request that only half or even a smaller number of cores be used per node. This has the effect of providing more memory for each subdomain (MPI rank). In the same spirit, another option is to run the problem on a larger number of processors. This will result in smaller subdomain matrices, but the size of the coarse matrix will increase. For large problems, it may be necessary to use a second coarse level to limit the coarse matrix size. Memory resources might also be exceeded by the storage of search directions used by the Krylov method (e.g. GMRES). The default maximum number of iterations is 1000, and reducing this number will result in memory savings. Additional information on memory usage is available in the online documentation *GDSW Memory Use Tutorial*.

- **Convergence Issues:** Due to the limitations of finite precision arithmetic, it may not always be possible to provide a solution which satisfies the specified relative residual tolerance. This is especially the case for poorly conditioned linear systems or unrealistically small solver tolerances. Discussion of this topic continues in the online document *Solver Accuracy Notes*.

Increasing the maximum number of iterations is one simple option that can result in successful solves. Other options for improving convergence are described below in
separate bullets. If convergence to the specified tolerance still cannot be achieved, then one option is to increase the relative residual tolerance until the solves are successful. To confirm that the quantities of interest have converged, one or more additional runs with even smaller tolerance are recommended.

- **Subdomain Overlap**: The default value of the `overlap` solver parameter is two. This means that the original subdomains are extended by two layers of elements when determining their overlap. Increasing the overlap often reduces the number of iterations needed for convergence, but the memory requirements increase and each iteration will require more time. Try a value for overlap of 3, 4, or more and see what effects it has. Even larger values of overlap can be used for models with only shell elements due to the reduced demands on the subdomain solver.

- **Static Condensation**: For non direct frequency response problems, the linear solver eliminates subdomain interior residuals at each iteration by default and solves the resulting Schur complement system of equations. For models with higher-order elements (i.e. polynomial degree greater than 2), it is recommended that the `SC_option` parameter be set to `none` since elimination of residuals interior to the subdomain can be problematic (i.e. effects of round off errors can be more pronounced). Similar concerns are present for direct frequency response problems, but the default option for such problems is `none`.

- **Frequency Response Analysis**: The iterative solution of direct frequency response problems can be challenging. The coefficient matrix $K + i\omega C - \omega^2 M$ can be both indefinite and complex depending on the input circular frequency $\omega$ and the damping matrix $C$. A more thorough discussion of trouble shooting the Helmholtz linear solver is in the Sierra/SD How To manual in section Frequency response linear solver.

A distributed memory sparse direct solver is recommended if the problem is not too large (to fit in memory). A direct solver works well for structures that are not blocky and shell element models. In the GDSW solver block set `useParallelDirectSolver = yes`. If available, this uses the Intel code Cluster Pardiso. Otherwise, SuperLU-Dist is used. If it is desired to use SuperLU-Dist on a platform where Cluster Pardiso is available, the option `useSuperLUDist = yes` must be specified.

If the distributed memory sparse direct solver option is not viable, then convergence may be improved as the number of processors increases. This leads to smaller subdomains and better performance of the coarse part of the GDSW preconditioner. Some users have also seen improved convergence by using the non-default solver parameter settings `orthogH = 0` and `precondUpdateFreq = 1`. If the problem already has a fair amount of damping, it may also help to set `structural_damping = 0`. This manual documents the interface 2.5.5.

Fiscal year 2021 planned work includes the capability to solve for multiple subdomains per MPI process, which should improve performance without the need to run on more processors.
• **Constraint Equations:** Certain types of constraint equations like those introduced by RBE3 elements can cause challenges for the linear solver (see earlier discussion of the max_numterm_C1 solver parameter, Type 1 constraints, and Type 2 constraints). Models with Type 2 constraints are generally harder to solve than those without them. Two lines in the solver output file dd_solver.dat relevant to avoiding Type 2 constraints are maxNumNonZeros in Tran Matrix and maxNumTermsForType1Constraints. The current value of the max_numterm_C1 solver parameter is reported by maxNumTermsForType1Constraints. Type 2 constraints can be avoided by setting max_numterm_C1 to be no less than maxNumNonZeros in Tran Matrix, but in some cases this can lead to excessive memory requirements for the solver.

Another source of potential difficulty is the presence of dependent or nearly dependent constraints. This means that the coefficient matrix $\hat{C}$ in the constraint equations $\hat{C}x = 0$ has either an infinite or large condition number. There are filters inside of Sierra/SD to eliminate linearly dependent constraints, but some may still be passed to the solver. The solver also identifies and eliminates dependent or nearly dependent constraints, but this is not always foolproof. It is advised that an analyst carefully check their model’s contact definitions to avoid potential problems in terms of model correctness or for the linear solver. Additional information on constraint equations and their interactions with the linear solver is available in the online documentation *Constraints*.

• **Singular Solves:** The stiffness matrix for a structure with no essential boundary conditions is singular. Thus, static analyses or modal analyses with a zero shift require special considerations. First, any rigid body mode component of the applied loads must be removed (see FilterRbmLoad and num_rigid_mode in the Parameters section). The GDSW solver is able to solve singular systems, but it is important that the rigid body modes actually exist. For example, consider a model with curved surfaces that are connected using node-to-face tied surfaces. If a dependent node is not located on the independent surface, then one or more of the rotational rigid body modes will be lost. If the solver expects there to be six rigid body modes but there are say only three, then solver convergence problems are likely to occur. To avoid this problem, ensure that your model does indeed have the specified rigid body modes.

• **Very Small Solver Tolerances:** As noted earlier, it may not be possible for the solver to provide a solution which satisfies the requested relative residual tolerance. Nevertheless, there are some situations where it may be possible to reduce the actual relative residual below what is possible using default solver parameters. Sierra/SD has a guardrail to error out if the actual relative residual is more than 100 times of that requested (see linear_solver_bailout_factor in the Parameters section). By increasing this parameter while reducing the solver tolerance, it may be possible to reduce the actual relative residuals for challenging problems like structural acoustics.

• **Scalability:** If you add prt_debug=1 or prt_debug=yes in the GDSW section, then the information needed to measure the load balance is written to a file. The file name is subdomainData.dat. A row of data in the subdomainData file has 8 columns of

67
integers. There is one row per subdomain (or MPI rank or core). Typically, the integers in each column are defined as follows.

1. Interior unknowns (i.e. number of unknowns for static condensation).
2. Non-zeros in the factors of the interior matrix.
3. Unknowns on subdomain boundary.
4. Owned unknowns on subdomain boundary.
5. Unknowns in subdomain.
7. Unknowns in overlapping subdomain.
8. Non-zeros in the factors of the overlap matrix.

In the special case of a diagonal preconditioner (preconditioner_type=DIAG), only 2 integers are printed.

1. Unknowns in subdomain.
2. Owned unknowns in subdomain.

Each iteration sometimes accesses each of these matrices a constant number of times. Often times the non-zeros in the factors of the overlap matrix predict the load balance. Larger numbers are generally more important. If the interior unknowns are eliminated, then non-zeros in the factors of the interior matrix will sometimes predict the load balance. The unknowns in a subdomain is governs the load balance of matters related to orthog. As orthog or orthogH increases, the influence of the unknowns per subdomain increases.

2.5.4. Mathematical Conditioning Issues

The performance of the solver is closely tied to the mathematical conditioning of the equation system it is solving. Conditioning can be thought of as the ratio of the largest eigenvalue of the system to the smallest eigenvalue. The larger this ratio becomes the more difficult it will be to solve the system and the larger potential error there will be in the solution.

Many model features can increase (or worsen) model conditioning.

- Rigid body modes (which have zero eigenvalues) are a special case which is handled as carefully as is theoretically possible, but which nonetheless have robustness problems.
- Mesh refinement increases gradients.
- Poorly shaped elements can lead to ill conditioned linear systems
• Understanding Type 1 versus Type 2 constraints as discussed in the definition of the GDSW option `max_numterm_C1` can lead to substantial performance improvements.

• Models in which a stiff material abuts a soft material are poorly conditioned.

• Mixtures of high density and low density components.

• Meshes with mixtures of solid and structural elements.

A poorly conditioned system can cause difficulties for the solver, both from an accuracy and robustness standpoint.

• Long solve times or many iterations required for convergence

• Difficulty obtaining a low target residual during the solve

• Large difference between the ‘recursive’ and ‘actual’ residuals reported by the solver. This can also indicate the solver result has lower accuracy than intended.

• Significant localized errors in the result obtained at some nodes

2.5.5. Frequency Response Functions

Several additional SD training documents\textsuperscript{42} and presentations describe the solver behavior, debugging, and usage guidelines in more detail. See the Sierra/SD training documents on 'GDSW Solver Accuracy Notes', 'GDSW 101', 'GDSW Memory Use Tutorial.

2.5.6. Parameters

SubdomainData is described in Section 2.5.3.

The custom option for `Hprecond` in Table 2-19 preconditions the matrix

\[-\omega^2(\alpha_M + i\beta_M)M + i\omega C + (\alpha_K + i\beta_K)K,\]

where \(\omega\) is the circular frequency of excitation, \(i\) is the imaginary unit, and \(M, C\) and \(K\) are the mass, damping, and stiffness matrices, respectively. With \(\gamma = \text{structural\_damping}\) and \(\beta = \text{viscous\_damping}\) the non-zero parameters for the other preconditioning options are \(\alpha_K = 1\) for stiffness, \(\alpha_K = 1, \alpha_M = -1\) for Laird-Giles, \(\alpha_K = 1, \alpha_M = 1, \beta_M = -\gamma\) for shifted Laplacian, and \(\alpha_K = 1, \beta_K = \gamma + \beta\omega, \alpha_M = 1\) for operator. Notice one should not use the stiffness preconditioning option for \(\omega\) near zero for structures with rigid body modes since \(K\) is singular or near singular in this case.
<table>
<thead>
<tr>
<th>Variable</th>
<th>Values</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>version</td>
<td>integer</td>
<td>2</td>
<td>GDSW version</td>
</tr>
<tr>
<td>num_vectors_keep</td>
<td>integer</td>
<td>-1</td>
<td>number of stored search directions to keep after culling</td>
</tr>
<tr>
<td>max_previous_sols</td>
<td>integer</td>
<td>0</td>
<td>maximum number of previous solutions used to accelerate convergence</td>
</tr>
<tr>
<td>num_sub_per_proc</td>
<td>integer</td>
<td>1</td>
<td>number of subdomains per processor</td>
</tr>
<tr>
<td>num_iter_improve_I</td>
<td>integer</td>
<td>0</td>
<td>number of iterative improvement steps for I_solver = LDM and precision_option_I = single</td>
</tr>
<tr>
<td>num_iter_improve_O</td>
<td>integer</td>
<td>0</td>
<td>number of iterative improvement steps for O_solver = LDM and precision_option_I = single</td>
</tr>
<tr>
<td>num_iter_improve_coarse</td>
<td>integer</td>
<td>0</td>
<td>number of iterative improvement steps for coarse_solver = LDM and and precision_option_coarse = single</td>
</tr>
<tr>
<td>overlap_method</td>
<td>integer</td>
<td>0</td>
<td>0 - standard graph-based</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>1 - element-based</td>
</tr>
<tr>
<td>cull_method</td>
<td>integer</td>
<td>1</td>
<td>0 - none: reach maximum and then stop</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>1 - simple: reach maximum and then remove most recent ones</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>2 - eigen: cull search directions based on solution to eigenvalue problem</td>
</tr>
<tr>
<td>num_vectors_keep</td>
<td>integer</td>
<td>-1</td>
<td>number of stored search directions to keep after culling (-1 automatic selection)</td>
</tr>
<tr>
<td>reduced_option_coarse</td>
<td>integer</td>
<td>3</td>
<td>same as reduced_option but for subregions</td>
</tr>
<tr>
<td>coarse_size</td>
<td>integer</td>
<td>0</td>
<td>0 - automatic selection of coarse space size</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>1 - select small coarse space</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>2 - select large coarse space</td>
</tr>
<tr>
<td>preconditioner_type</td>
<td>integer</td>
<td>2</td>
<td>1 - BDDC</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>2 - GDSW</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>3 - DIAG</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>4 - NODAL</td>
</tr>
<tr>
<td>interface_precond</td>
<td>integer</td>
<td>0</td>
<td>0 - do not require interface precondition</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>1 - require interface precondition</td>
</tr>
<tr>
<td>coarse_connectivity_option</td>
<td>integer</td>
<td>1</td>
<td>algorithm number for coarse elem connect</td>
</tr>
<tr>
<td>viscous_damping</td>
<td>real</td>
<td>0</td>
<td>viscous damping coefficient for preconditioner operator</td>
</tr>
<tr>
<td>structural_damping</td>
<td>real</td>
<td>0.12</td>
<td>structural damping coefficient for preconditioner operator</td>
</tr>
<tr>
<td>rbm_tolerance</td>
<td>real</td>
<td>1e-12</td>
<td>tolerance used to flag rigid body modes in dd_solver.dat. Recommend usage of the rbmtolerance in the parameters section.</td>
</tr>
<tr>
<td>con_tolerance</td>
<td>real</td>
<td>2.5e-9</td>
<td>singularity tolerance for processing constraints</td>
</tr>
<tr>
<td>con_row_tolerance</td>
<td>real</td>
<td>1e-1</td>
<td>pivoting tolerance for processing constrains</td>
</tr>
<tr>
<td>ML_max_level</td>
<td>integer</td>
<td>7</td>
<td>maximum number of levels for multilevel local solver</td>
</tr>
<tr>
<td>ML_max_coarse</td>
<td>integer</td>
<td>1000</td>
<td>maximum number of unknowns for coarsest level</td>
</tr>
<tr>
<td>use_epetra_coarse</td>
<td>integer</td>
<td>0</td>
<td>0 - do not use Epetra matrices for coarse correction</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>1 - use Epetra matrices for coarse correction</td>
</tr>
<tr>
<td>parmetis_option</td>
<td>integer</td>
<td>0</td>
<td>Parmetis option for coarse problem partitioning:</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>0 - PartKway</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>1 - PartGeomKway</td>
</tr>
</tbody>
</table>
### Table 2-18. GDSW Section Print Options.

<table>
<thead>
<tr>
<th>Variable</th>
<th>Values</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ML_print_coarse</td>
<td>integer</td>
<td>0</td>
<td>0 - no output</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>1 - print coarse stiffness matrix</td>
</tr>
<tr>
<td>ML_print_Phi</td>
<td>integer</td>
<td>0</td>
<td>0 - no output</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>1 - print interpolation matrix</td>
</tr>
<tr>
<td>pardiso_message_level</td>
<td>integer</td>
<td>0</td>
<td>0 - no messages</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>1 - print messages</td>
</tr>
<tr>
<td>prt_matrix</td>
<td>integer</td>
<td>0</td>
<td>0 - no output</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>1 - print out matrix in 3-column format</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>2 - print out matrix in CSR format</td>
</tr>
<tr>
<td>prt_subdomain_coarse</td>
<td>integer</td>
<td>0</td>
<td>0 - do not print subdomain coarse matrix</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>1 - print subdomain coarse matrix</td>
</tr>
<tr>
<td>prt_subdomain_PU</td>
<td>integer</td>
<td>0</td>
<td>0 - do not print sub partition of unity</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>1 - print sub partition of unity</td>
</tr>
<tr>
<td>prt_stabilization</td>
<td>integer</td>
<td>0</td>
<td>0 - do not print coarse stabilization matrices</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>1 - print coarse stabilization matrices</td>
</tr>
<tr>
<td>prt_interior</td>
<td>integer</td>
<td>0</td>
<td>0 - do not print interior matrices</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>1 - print interior matrices</td>
</tr>
<tr>
<td>prt_overlap</td>
<td>integer</td>
<td>0</td>
<td>0 - do not print overlap matrices</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>1 - print overlap matrices</td>
</tr>
<tr>
<td>prt_memory</td>
<td>integer</td>
<td>0</td>
<td>0 - do not print gdsw memory diagnostics</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>1 - print gdsw memory diagnostics</td>
</tr>
<tr>
<td>prt_debug</td>
<td>integer</td>
<td>0</td>
<td>0 - do not print subdomainData.dat</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>1 - print subdomainData.dat</td>
</tr>
<tr>
<td>write_orthog_data</td>
<td>integer</td>
<td>0</td>
<td>0 - do not write orthogonalization data to file</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>1 - write orthogonalization data to file</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>1 - ignore memory reporting</td>
</tr>
<tr>
<td>ML_print_coarse</td>
<td>integer</td>
<td>0</td>
<td>0 - do not print coarse stiffness matrix</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>1 - print coarse stiffness matrix</td>
</tr>
<tr>
<td>ML_print_Phi</td>
<td>integer</td>
<td>0</td>
<td>0 - do not print interpolation matrix</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>1 - print interpolation matrix</td>
</tr>
</tbody>
</table>
### Table 2-19. – GDSW Section Options (Helmholtz).

<table>
<thead>
<tr>
<th>Variable</th>
<th>Values</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hprecond</td>
<td>integer</td>
<td>5</td>
<td>Helmholtz preconditioner:</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>0-stiffness: Stiffness based</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>1-LG: Laird-Giles</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>2-custom: Custom</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>3-SL: shifted Laplacian</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>5-operator: Operator with damping</td>
</tr>
<tr>
<td>orthogH</td>
<td>integer</td>
<td>20</td>
<td>maximum number of stored search directions for Helmholtz problems</td>
</tr>
<tr>
<td>max_previous_sols</td>
<td>integer</td>
<td>0</td>
<td>maximum number of previous solutions used to accelerate convergence</td>
</tr>
<tr>
<td>precondUpdateFreq</td>
<td>integer</td>
<td>10</td>
<td>frequency to update preconditioner as operator changes</td>
</tr>
<tr>
<td>viscous_damping</td>
<td>real</td>
<td>0</td>
<td>viscous damping coefficient (see text)</td>
</tr>
<tr>
<td>structural_damping</td>
<td>real</td>
<td>0.12</td>
<td>structural damping coefficient (see text)</td>
</tr>
<tr>
<td>alphaK</td>
<td>real</td>
<td>0</td>
<td>custom precond stiffness coefficient (see text)</td>
</tr>
<tr>
<td>betaK</td>
<td>real</td>
<td>0</td>
<td>custom precond stiffness coefficient (see text)</td>
</tr>
<tr>
<td>alphaM</td>
<td>real</td>
<td>0</td>
<td>custom precond mass coefficient (see text)</td>
</tr>
<tr>
<td>betaM</td>
<td>real</td>
<td>0</td>
<td>custom precond mass coefficient (see text)</td>
</tr>
<tr>
<td>krylov_methodH</td>
<td>integer</td>
<td>5</td>
<td>same as krylov_method but for Helmholtz problems</td>
</tr>
<tr>
<td>SC_optionH</td>
<td>integer</td>
<td>0</td>
<td>same as SC_option but for Helmholtz problems</td>
</tr>
</tbody>
</table>

### Table 2-20. – GDSW Section Options (Solvers).

<table>
<thead>
<tr>
<th>Variable</th>
<th>Values</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>default_solver</td>
<td>integer</td>
<td>1</td>
<td>1-direct: Esmond Ng’s sparse direct solver</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>2-LDM: Clark’s LDM’ sparse direct solver</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>6-NoPivot: Clark’s sparse direct solver for conjugate gradient matrix</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>0-diag: diagonal (holds in exact arithmetic)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>1-full: full $\Phi^T A \Phi$ matrix</td>
</tr>
<tr>
<td>PTAP_solver</td>
<td>integer</td>
<td>1</td>
<td></td>
</tr>
</tbody>
</table>
2.6. Sensitivity

Sensitivity to parameters is available for modal analysis, Craig-Bampton reduction (CBR), static solutions and some transient solutions. An example input file for modal analysis is given in the Section 9. In the case of CBR analysis, we refer to sections 3.5 and 3.5.1 for a detailed discussion of how to perform sensitivity analysis. The sensitivity section controls global parameters related to sensitivity analysis. Sensitivity analysis is not performed in Sierra/SD unless this section is present in the input file. The following example illustrates the legal keywords. Valid keywords are identified in Table 2-21. Lists of numbers should follow the rules for integer lists detailed in Section 2.1.

```
SENSITIVITY
  values all
  vectors 1,3,5,7:9
  iterations 8
  tolerance 1e-7
Attune
AttuneNodeset sensitivity_nodeset
END
```

<table>
<thead>
<tr>
<th>Keyword</th>
<th>argument</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>values</td>
<td>“all”/“none” or list</td>
<td>eigenvalue selection</td>
</tr>
<tr>
<td>vectors</td>
<td>“all”/“none” or list</td>
<td>eigenvector selection</td>
</tr>
<tr>
<td>iterations</td>
<td>integer</td>
<td>number of eigenvector iterations</td>
</tr>
<tr>
<td>tolerance</td>
<td>float</td>
<td>convergence tolerance for eigenvectors</td>
</tr>
<tr>
<td>attune</td>
<td>n/a</td>
<td>enable attune output</td>
</tr>
<tr>
<td>AttuneNodeset</td>
<td>string</td>
<td>nodeset for reduced model</td>
</tr>
</tbody>
</table>

Table 2-21. – Sensitivity Analysis Keywords.

The keywords values and vectors are used to control what types of sensitivities are computed for which cases in the analysis. In modal analysis, these refer to the eigenvalues and eigenvectors, respectively, and the case numbers represent the mode numbers. In static analysis, vectors refers to the displacement vector results, and values has no meaning. Also, in modal analysis, eigenvalue sensitivities are always computed when eigenvector sensitivities are requested for a mode. Allowable values are:

```
vectors all       // compute for all cases/modes
vectors none      // compute for no cases/modes
vectors 1:3,5      // range of cases/modes
```

Omitting the keyword vectors (or values) is equivalent to not requesting those sensitivities; in other words, it is equivalent to vectors none. The keywords iterations and tolerance are used in computing eigenvector derivatives. The default values are 10.
and $1.0 \times 10^{-6}$, respectively. If the eigenvector sensitivity approximation fails to converge to the desired tolerance in the specified number of iterations, a fatal error occurs.

### 2.6.1. Attune

An interface is provided to the Attune test/analysis correlation code supplied by ATA engineering. The data is written to an external text file, with a file name based on the input (*.inp) file name. A surrogate of the finite element model is determined using eigenvectors. Attune applies only to eigen sensitivity analysis, and the eigen modes must be selected using values. For output through this interface, the following two parameters must be defined.

- **attune**: request interface output.
- **AttuneNodeset**: identification of a nodeset to be associated with the test degrees of freedom. Note that even if test mode shapes are not available, Attune requires the definition of a reduced space model (using this nodeset). It is required for mode tracking.

To use Attune, please refer to the ATA website and on line documentation.

### 2.6.2. Sensitivity Output

Sensitivity results are output to the same file as the nominal results. The arrangement of the output depends on the analysis. The statics nominal result is output, followed by the sensitivity result for each parameter. In eigenvalue problems, the nominal frequencies and eigenvectors are output, followed by the eigenvalue and eigenvector sensitivities with respect to the first parameter, the second parameter, etc. The eigenvalue sensitivities are placed in the time field of each output record, like the frequencies are for the nominal modal parameters. For transient analysis, the nominal response for each time step is output, followed by the sensitivities for that time step. Then the nominal results for the next time step are output. See Figure 2-5 for an example of eigen sensitivity.

The change of parameter (or tolerance) may be specified in any of three ways.

1. Specify an absolute tolerance by entering “+/-” followed by the number, e.g. “+/- 1.05e-4”.
2. Specify a relative tolerance by entering “+/-” followed by a number and the keyword “percent”. Each field should be separated by a space. For example,
   
   56 +/- 2.0 percent
3. Use the default tolerance by entering only the “+/-” by itself. The default tolerance is 2 percent.
EXPLORE> select step 4
  Time = 3.504E+3 (time step 4 of 30)
EXPLORE> gvar
  Time = 3.504E+3 (time step 4 of 30)
  Global Time Step Variables
  ModeNumber = 4.0000E+00
  EigenFrequency = 3.5042E+03
  EigenVectScale = 1.0000E+00
  deriv_Block_1000_area = 3.9362E+03
  deriv_Block_101_thickness = 1.4426E+07

The order of parameters can be determined from the global variables. It is also available in the results file. The sensitivities may be extracted using the global variables.

EXPLORE> times
  Number of time steps = 18
  Step 1) 725.3E+0
  Step 2) 725.3E+0
  Step 3) 3.005E+3
  Step 4) 3.504E+3
  Step 5) 3.504E+3
  Step 6) 4.929E+3
  Step 7) 602.1E+0
  Step 8) 602.1E+0
  Step 9) 6.512E+0
  Step 10) 3.936E+3
  ...

The first nmodes (6 in this example) eigenvalues and vectors are associated with the nominal structure. The next nmodes values are the $d\lambda/dP_1$ values associated with the first parameter, $P_1$. The corresponding vectors are $d\phi_1/dP_1$.

**Figure 2-5.** – Eigen Sensitivity Example Data. In this example, both eigenvalue and eigenvector sensitivities are computed. The data is probed using “explore”. Global variable include sensitivities to area in block 1000 and thickness in block 101.
The selection of parameters is controlled by the inclusion of a +/- symbol following a parameter in the input deck. Examples of valid sensitivity parameter definitions are:

```
MATERIAL 1
  E 10e6 +/- 1e6 // absolute tolerance specified
  density 2.59e-4 +/- // no tolerance, use default
END

BLOCK 1
  area 0.10 +/- 5 percent // relative tolerance specified
END

BLOCK 2
  thickness +/- 1 percent // relative to Exodus attribute
END

Loads
  nodeset 1
  force 0. 0. 1000 +/- 0 0 10 // tolerance for vector parameter
end
```

Note that the tolerances are specified on the parameters where they normally appear in the input file. That is, these definitions do not appear in the sensitivity section.

The sensitivity quantities output to the exodus file are derivatives, and can be used to compute a first-order approximation to the change in an output quantity with respect to a parameter. For example, the change in an eigenvalue \( \lambda \) that depends on a parameter \( p \) can be approximated

\[
\lambda(p + \Delta p) = \lambda(p) + \frac{d\lambda}{dp} \Delta p + o(\Delta p).
\]

The quantity written to the exodus output is \( d\lambda/dp \). The tables printed in the results and standard output also include the linear approximation to the change in \( \lambda \),

\[
\frac{d\lambda}{dp} \Delta p.
\]

This approximation may be inaccurate if \( \Delta p \) is too large. The size of \( \Delta p \) is specified by the user in the input deck for each parameter included in the sensitivity study.

### 2.6.2.1. Derived Output Quantities

The sensitivity analysis methods use a semi-analytic method. Primary variables (usually displacement) are computed in terms of changes in stiffness and mass matrices, and resultant displacements are then computed analytically. See Figure 2-6. Many other output quantities are computed in terms of these primary variables using the standard output routines. Most of these output quantities depend linearly on the primary variables. For
example, computation of the derivatives of strain can be readily computed using the chain rule, and may also employ the same procedures for strain computation. Let $\varepsilon = \kappa u$ define the strain/displacement relationship at a given point in the model. Here $\kappa$ is a constant.

$$\frac{d\varepsilon}{dp} = \frac{d\varepsilon}{du} \frac{du}{dp} = \kappa \frac{du}{dp}$$

Code that computes $\varepsilon$ from $u$ may be used to compute $\frac{d\varepsilon}{dp}$ from $\frac{du}{dp}$.

Other variables are not linear with respect to the primary variables. For example, the strain energy or the von Mises stress include quadratic terms in displacement.

$$E_s = u^T Ku$$

$$\frac{dE_s}{dp} = \frac{dE_s}{du} \frac{du}{dp} = u^T K \frac{du}{dp}$$

These variables may not use the same code paths. Data is written, but is not correct for these variables.

<table>
<thead>
<tr>
<th>Statics</th>
</tr>
</thead>
<tbody>
<tr>
<td>$Ku = f$</td>
</tr>
<tr>
<td>$\Delta u = K^{-1}(\Delta f - \Delta Ku)$</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Eigen</th>
</tr>
</thead>
<tbody>
<tr>
<td>$(K - \lambda M)\phi = 0$</td>
</tr>
<tr>
<td>$\Delta \lambda_i = \phi^T \Delta K \phi - \lambda \phi^T \Delta M \phi$</td>
</tr>
</tbody>
</table>

**Figure 2-6.** – Semi-Analytic Methods for Sensitivity Analysis.

### 2.6.2.2. Solution Types

Sensitivity analysis is available only for the solution types shown in Table 2-22. The primary application is in eigenvalue problems where the semi-analytic solutions can provide significant computation and accuracy benefit over a finite difference approach.4

<table>
<thead>
<tr>
<th>Name</th>
<th>Section</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>eigen</td>
<td>3.10</td>
<td>Normal Modes</td>
</tr>
<tr>
<td>statics</td>
<td>3.26</td>
<td>Linear Statics</td>
</tr>
</tbody>
</table>
2.6.2.3. Sensitivity Limitations

1. Subsection Eigen Sensitivity Analysis section Solution Procedures of the Theory Manual explains how sensitivity analysis may be performed using most solvers.

2. Outputs are limited to variables linear in displacement (2.6.2.1). In particular, output of von Mises stress is output for these solutions, but is not correct. That is the derivative of the von Mises stress is not the von Mises stress of the derivative of the stress.

2.7. Coordinate

Local coordinate systems may be defined to orient directional materials (section 4.7.2.3), define constraints, boundary conditions, or loads in local orientations (sections 2.9, 6.1 and 6.3), or transform outputs to a local coordinate system (sections 7.3 and 7.4). The basic/default coordinate frame can also be explicitly referenced as coordinate “0”, if desired.

The ability to visualize and verify correctness of coordinate system definitions is a key element of the overall workflow. At present, support is limited to the material direction output for anisotropic isosolid elements; see section 4.1.3.

BEGIN RECTANGULAR COORDINATE SYSTEM <string>name
ORIGIN = <real> <real> <real>
Z POINT = <real> <real> <real>  # Point on \( \hat{Z} \) axis
XZ POINT = <real> <real> <real>  # Point on \( \hat{X}\hat{Z} \) plane
ROTATE <real> ABOUT AXIS X|Y|Z  # angle in radians
END

BEGIN CYLINDRICAL COORDINATE SYSTEM <string>name
ORIGIN = <real> <real> <real>
Z POINT = <real> <real> <real>  # Point on \( \hat{Z} \) axis
XZ POINT = <real> <real> <real>  # Optional: point on \( \hat{X}\hat{Z} \) plane; sets
# location of azimuthal angle \( \theta = 0 \)
ROTATE <real> ABOUT AXIS X|Y|Z  # angle in radians
END

Syntax 2.2. Rectangular Coordinate System Syntax
BEGIN CYLINDRICAL COORDINATE SYSTEM <string>name
ORIGIN NODESET = <nodelist> # Single-node nodesets
Z POINT NODESET = <nodelist> # Node on \( \ddot{Z} \) axis
XZ POINT NODESET = <nodelist> # Optional: node on \( \ddot{X}\ddot{Z} \) plane; sets
# location of azimuthal angle \( \theta=0 \)
ROTATE <real> ABOUT AXIS X|Y|Z # angle in radians
END

Syntax 2.3. Cylindrical Coordinate System Syntax

BEGIN SPHERICAL COORDINATE SYSTEM <string>name
ORIGIN = <real> <real> <real>
Z POINT = <real> <real> <real> # Point on \( \ddot{Z} \) axis
XZ POINT = <real> <real> <real> # Optional: point on \( \ddot{X}\ddot{Z} \) plane; sets
# location of azimuthal angle \( \theta=0 \)
ROTATE <real> ABOUT AXIS X|Y|Z # angle in radians
END

BEGIN SPHERICAL COORDINATE SYSTEM <string>name
ORIGIN NODESET = <nodelist> # Single-node nodesets
Z POINT NODESET = <nodelist> # Node on \( \ddot{Z} \) axis
XZ POINT NODESET = <nodelist> # Optional: node on \( \ddot{X}\ddot{Z} \) plane; sets
# location of azimuthal angle \( \theta=0 \)
ROTATE <real> ABOUT AXIS X|Y|Z # angle in radians
END

Syntax 2.4. Spherical Coordinate System Syntax

BEGIN CONICAL COORDINATE SYSTEM <string>name
ORIGIN = <real> <real> <real>
Z POINT = <real> <real> <real> # Point on \( \ddot{Z} \) axis
XZ POINT = <real> <real> <real> # Optional: point on \( \ddot{X}\ddot{Z} \) plane
ROTATE <real> ABOUT AXIS X|Y|Z # angle in radians
ANGLE = <real>
END

BEGIN CONICAL COORDINATE SYSTEM <string>name
ORIGIN NODESET = <nodelist> # Single-node nodesets
Z POINT NODESET = <nodelist> # Node on \( \ddot{Z} \) axis
XZ POINT NODESET = <nodelist> # Optional: Node on \( \ddot{X}\ddot{Z} \) plane
ROTATE <real> ABOUT AXIS X|Y|Z # angle in radians
ANGLE = <real>

END
Syntax 2.5. Conical Coordinate System Syntax

```
BEGIN ELLIPSOIDAL COORDINATE SYSTEM <string>name
    ORIGIN = <real> <real> <real>
    Z POINT = <real> <real> <real>  # Point on \( \hat{Z} \) axis
    XZ POINT = <real> <real> <real>  # Point on \( \hat{X} \hat{Z} \) plane
    ROTATE <real> ABOUT AXIS X|Y|Z  # angle in radians
    AXIS STRETCHING = <real> <real> <real>  # angle in radians
END
```

```
BEGIN ELLIPSOIDAL COORDINATE SYSTEM <string>name
    ORIGIN NODESET = <nodelist>  # Single node nodeset
    Z POINT NODESET = <nodelist>
    XZ POINT NODESET = <nodelist>
    ROTATE <real> ABOUT AXIS X|Y|Z  # angle in radians
    AXIS STRETCHING = <real> <real> <real>  # angle in radians
END
```

Syntax 2.6. Ellipsoidal Coordinate System Syntax

The prefix to `COORDINATE SYSTEM` specifies the type of local coordinate system. Supported types are: RECTANGULAR (Cartesian), CYLINDRICAL (Polar), SPHERICAL, CONICAL (a cylindrical system with an aperture angle), and ELLIPSOIDAL (a spherical system stretched by AXIS STRETCHING).

A local element coordinate system is defined by a set of three control points. These points may be defined using absolute locations (ORIGIN, Z POINT, XZ POINT) or using nodelists in the mesh file (ORIGIN NODESET, Z POINT NODESET, XZ POINT NODESET) that contain exactly one node each. The control points define a local \( \hat{X} \hat{Y} \hat{Z} \) Cartesian coordinate system centered at the coordinate system origin. For cylindrical, spherical, conical, and ellipsoidal systems the orientation of the basis vectors at individual nodes and elements is spatially dependent. In this documentation, the local basis vectors at each node or element will be denoted as \( \hat{r} \), \( \hat{s} \), and \( \hat{t} \).

Note: coordinate systems defined by nodesets use only the initial location of the nodes and do not update with model displacement.

- **ORIGIN**: Required. Specifies the origin location for the new coordinate system.
- **Z POINT**: Required. Specifies a point on the \( \hat{Z} \) axis for the new coordinate system. For SPHERICAL, this defines where the zenith (polar) angle \( \phi \) is 0.
• **XZ POINT**: Required only for **RECTANGULAR** and **ELLIPSOIDAL**. Third point to specify the $\tilde{X}\tilde{Z}$-plane of the new system. Must not lie on the $\tilde{Z}$ axis. If not specified, an arbitrary axis will be chosen based on the global coordinate system. For **CYLINDRICAL** and **SPHERICAL**, this defines where the azimuthal angle $\theta$ is 0. **XZ POINT** also affects behavior at ill-defined points such as the origin of **SPHERICAL** systems, or along the $\tilde{Z}$ axis of **CYLINDRICAL** systems (see figure 2-11).

• **ANGLE** (**CONICAL COORDINATE SYSTEM** only): Required. Defines the angle in radians to rotate the axial direction away from the $\tilde{Z}$ axis (see figure 2-9).

• **AXIS STRETCHING** (**ELLIPSOIDAL COORDINATE SYSTEM** only): Required. Defines the aspect ratio to stretch a spherical system (see figure 2-10).

• **ROTATE**: Processed last. Specifies a rotation (in radians) of the coordinate system about one of its own axes. The conical system is effectively a cylindrical system with a local rotation about the $\tilde{Y}$ axis. Only one rotation may be specified.

The three coordinate control points are illustrated in figure 2-7. The coordinate system types are depicted in figures 2-8 to 2-10.

![Coordinate System Diagram](image)

**Figure 2-7.** – Coordinate system definition vectors: note that **XZ POINT** determines the $\tilde{X}$ axis, but need not lie along it, which is the case depicted in this figure.
Three examples (rectangular, spherical, and cylindrical) of transformed coordinate systems are given. The center of the rectangular block (6 × 2 × 3) is located at \((3, -1, 5)\), and rotated 30 degrees about the \(X\) axis. The center of the sphere (radius of 2) is located at \((5, 4, -2)\). The center of the cylinder (radius of 1, height of 2, and rotated 90 degrees about the \(X\) axis) is located at \((-5, 2, 0)\). The new coordinate systems are defined respectively of each of the geometries, and are indicated by \(\tilde{X}\tilde{Y}\tilde{Z}\) and the subscripts \((r, s, c)\).

Conical Coordinate System Definition at \(\tilde{X}\tilde{Z}\) plane
Examples of rectangular, spherical, and cylindrical coordinate systems are given in figure 2-8. In those examples, we wish to define coordinate systems for three geometries: a brick, a sphere, and cylinder. The corresponding input deck is shown below.

```
begin rectangular coordinate system rc
  origin 3 -1 5
  z point 5 -1 5
  xz point 3 0.7321 6
end

coordinate spherical coordinate system ballLike
  origin 5 4 -2
  z point 5 2 -2
  xz point 3 4 -2
end

begin cylindrical coordinate pinSystem
  origin -5 2 0
  z point -5 4 0
  xz point -5 2 2
end
```

**Input 2.3. Example coordinate system input**

For cylindrical, conical, spherical, and ellipsoidal systems, the local basis vectors \( \hat{r}, \hat{s}, \) and \( \hat{t} \) are not all well-defined when sampled along the \( \hat{Z} \) axis. At the origin, all coordinate
systems fall back to a local Cartesian system with \( \hat{r} \) along the \( \tilde{X} \) axis, \( \hat{s} \) along the \( \tilde{Y} \) axis, and \( \hat{t} \) along the \( \tilde{Z} \) axis. This is also true for the cylindrical and conical systems on the \( \tilde{Z} \) axis. Spherical and ellipsoidal systems on the \( \tilde{Z} \) axis will instead maintain the first vector \( (\hat{r}) \) pointing radially outward, the third vector \( (\hat{t}) \) pointing along the \( \tilde{X} \) axis, and the second vector \( (\hat{s}) \) as either the positive or negative \( \tilde{Y} \) axis (generating a right handed system). This maintains the normal and tangential properties of the three vectors for spherical coordinates. See figure 2-11 for a visual representation of this behavior.

The \textit{Exodus} mesh input file also has the ability to define local coordinate frames. These frames will be read and available during an analysis. In the case of a coordinate frame being defined in both the \textit{Exodus} file and the input file, the input file definition will be used, and a warning will be logged.

In spherical coordinates, it may help to consider the Cartesian frame \((\tilde{X},\tilde{Y},\tilde{Z})\) with the same orientation as \((r,\theta,\phi)\):

\[
\begin{align*}
\tilde{X} &= r \sin(\phi) \cos(\theta) \\
\tilde{Y} &= r \sin(\phi) \sin(\theta) \\
\tilde{Z} &= r \cos(\phi),
\end{align*}
\tag{2.1}
\]

If the user specifies a coordinate system in the \textbf{History} section, notice that its applicability may be somewhat limited, though convenient. In particular, only a single history file is written in each analysis, and only one coordinate frame may be outputted per node (see section 7.3). The history file will display variables as Cartesian regardless of coordinate choice. Table 2-23 shows the corresponding values for cylindrical and spherical coordinates.

Finally, we provide a full example of how we can use the coordinate definitions. We use the same coordinate definitions from figure 2-8 and input 2.3, and we define the brick as block 1, the sphere as block 2, and the cylinder as block 3, respectively. We also specify a sideset

<table>
<thead>
<tr>
<th>Coordinate System</th>
<th>History Variable</th>
<th>Corresponding Coordinate</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cylindrical</td>
<td>( X )</td>
<td>( r )</td>
</tr>
<tr>
<td></td>
<td>( Y )</td>
<td>( \theta )</td>
</tr>
<tr>
<td></td>
<td>( Z )</td>
<td>( z )</td>
</tr>
<tr>
<td>Spherical</td>
<td>( X )</td>
<td>( r )</td>
</tr>
<tr>
<td></td>
<td>( Y )</td>
<td>( \theta )</td>
</tr>
<tr>
<td></td>
<td>( Z )</td>
<td>( \phi )</td>
</tr>
</tbody>
</table>

\textbf{Table 2-23.} – Coordinate Names for history files.
Figure 2-11. – Coordinate system behavior near the $\tilde{Z}$ axis. $\hat{r}$ is red, $\hat{s}$ is green, and $\hat{t}$ is blue.
100 on the brick, a nodeset 200 on the sphere, and a nodeset 300 on the cylinder as shown in figure 2-12. We wish to apply a radial force emanating from the middle of the cylinder, and we want to record the output accelerations on the sphere. The corresponding input deck is attached, with associated coordinate systems defined in input 2.3.

```
Loads
   nodeset 300
   coordinate ball_like
   force = 1 0 0
end
History
   nodeset 200
   coordinate pin_system
   accel
end
```

Tractions can also be specified on a particular coordinate frame, but special care is required. We refer the reader to section 6.3.3 for using arbitrary coordinates with tractions.

![Diagram showing nodesets and sideset](image)

**Figure 2-12.** – An example of how coordinate systems are used. Accelerations are measured on the sphere (nodeset 200) due to a radial force applied on the cylinder (nodeset 300).

### 2.8. Function

Time, frequency and/or spatially dependent functions for transient and frequency response analysis can be defined using the **function** section. The following are simple examples of
the use of a function.

```plaintext
FUNCTION test_func1
    type LINEAR
    data 0.0 0.0
    data 0.0150 0.0
    data 0.0152 1.0
    data 0.030 0.0
END

FUNCTION poly_fun
    // a pulse of duration .05
    // peak value 6/7 at 1/49 sec
    // pulse(t)=8*sqrt(t)-686*t^2
    name "Smooth Pulse with Duration 0.05"
    type POLYNOMIAL
    data 0. 0.
    data .5 8.
    data 2. -686.
END
```

The function identifier string is given after function keyword. This identifier is used to reference the function in other parts of the input deck. It can be any string, but once this identifier could only be an integer. The keywords for these function definitions are:

1. TYPE to define the functional form,
2. NAME Additional reference information for the function. This may be printed to error messages or other informational output regarding the function, however it is the identifier that is always used to call the function from other parts of the input deck.
3. DATA for the functional parameters.

Other function definitions may require more parameters.

2.8.1. Function Offset/Shifts

Function input/output values can be offset and scaled. The syntax for defining these transformations is below.

```plaintext
X\text|abscissa offset = <real, default=0>
Y\text|ordinate offset = <real, default=0>
X\text|abscissa scale = <real, default=1>
Y\text|ordinate scale = <real, default=1>
```
Given a function $y = f(x)$, the transformed output would be:

$$y = y_{scale} \cdot \left[ f(x_{scale} \cdot [x + x_{offset}]) + y_{offset} \right].$$

Note that function offsets/shifts are currently only enabled for single-input, single-output functions. Otherwise, a warning will be issued and the offsets/shifts will be ignored.

### 2.8.2. Linear Functions

For linear functions, the data elements are pairs specifying the independent variable (e.g., time) and the corresponding function value. Evaluation of the function at unspecified points is performed via linear interpolation. In order to enforce uniqueness of the interpolant, 

Sierra/SD ignores points with an independent variable value less than that of the previous point. For example, the last point in the `ignored_point` function is ignored.

```
FUNCTION ignored_point
    type linear
    data 0.00 0.
    data 0.01 1.
    data 0.05 1.
    data 0.04 0. // ignored: column 1 may not decrease
END
```

![Graph of linear function "ignored_point".](image)

**Figure 2-13.** Linear function "ignored_point".

Figure 2-13 shows the graph of function 3.

Extrapolation refers to evaluating the function before the first given time or after the last given time. Linear functions use the value of the nearest data point to extrapolate. For example, in the following function, $f(3/100) = 1/2$.

```
FUNCTION extrapolation
    type linear
    data 0.00 0.
    data 0.01 1.
    data 0.02 0.5
END
```

88
Figure 2-14 shows the graph of the *extrapolation* function.

![Graph of the extrapolation function](image)

**Figure 2-14.** – Linear function 'extrapolation'.

If an independent variable value is listed multiple times, then the function value at that point is the average of the specified function values. For example, in the following function, $f(3/100) = 3/4$.

```plaintext
FUNCTION 5
  name "multiple_fun"
  type linear
  data 0.00 0.
  data 0.01 1.
  data 0.03 1.
  data 0.03 0.5
END
```

Function 5 is pictured in Figure 2-15.

![Graph of the multiple_fun function](image)

**Figure 2-15.** – Linear function #5. "multiple_fun".

2.8.3. **Sierra SM Piecewise Linear Functions**

Some limited Sierra SM syntax is supported for specifying a piecewise linear function in Sierra/SD. The data entered through this syntax is interpreted as a linear function, and follows all the rules specified above. This comes with limitations. Only the *piecewise linear* function type is supported, and a number of features are missing. Most
importantly, ending the function with "END FUNCTION <string>" is not supported. This comes from an incompatibility with existing Sierra/SD syntax; "FUNCTION <string>" happens to be valid Sierra/SD syntax in this scope and context.

```
Begin Function function_name
  Type is Piecewise Linear
  Begin Values
    0.00, 0.
    0.01, 1.
    0.05, 1.
  End Values
End
```

### 2.8.4. Functions using Tables

Functions may be specified by reference to a linearly interpolated table (as discussed in section 2.8.19). The table must be of dimension=1. 1-dimensional tables behave identically to the linear functions described above. However, they will typically be much faster and more memory efficient than linear functions, especially as the data size grows.

The function in the following example is a tabular representation of the data of Figure 2-14 and Function “extrapolation” above.

```
FUNCTION 7
  type table
  tablename=example7
END

Table example7
  dimension=1
  size=5
  datafile='example7.txt'
  origin 0.0
  delta .01
END
```

Within the datafile, “example7.txt”, the following data would be represented.

```
0.0
1.0
0.5
0.5
0.5
```
The linear function can be evaluated for any time, and the table is limited to the range 0-0.04. Table type functions require the `tablename` keyword.

### 2.8.5. Polynomials

For polynomials, the data points given are the exponent of the independent variable and the coefficient. The independent variable taken to any real power will always be evaluated as positive. If powers are repeated, their coefficients will sum. For example,

```
FUNCTION 6
  name "poly_fun"
  type polynomial
  data 0.0 0.
  data 1.0 1.
  data 2.0 0.1
  data 1.0 0.5
END
```

is equivalent to

```
FUNCTION 6
  name "poly_fun"
  type polynomial
  data 0.0 0.
  data 1.0 1.5
  data 2.0 0.1
END
```

The function value as a function of the independent variable $t$ is,

$$f(t) = 1.5t + 0.1t^2.$$

### 2.8.6. LogLog Functions

In frequency domain analysis, log/log functions are commonly used for application of loads. This is particularly true for random vibration inputs which are commonly specified on log/log plots. The `LogLog` option allows linear interpolation on a log/log plot so that only the corner frequencies need be specified. An example follows.

```
FUNCTION 1
  name "my_LogLog"
  type LogLog
  data 1.0 1e-8
```

2.8.7. SamplingRandom

The random pressure loading defined in subsection Random Pressure Loading section Loads and Materials of the Theory Manual provides a means of applying a pressure load with a specified spatial and temporal correlation. In many cases, the desired random function is a function of time only. In those cases the SamplingRandom function provides a mechanism for applying the load.

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Values</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>type</td>
<td>string</td>
<td>required</td>
<td>must be “SamplingRandom”</td>
</tr>
<tr>
<td>cutoff_freq</td>
<td>Real</td>
<td>required†</td>
<td>cutoff frequency (in Hz)</td>
</tr>
<tr>
<td>omega_c (ωc)</td>
<td>Real</td>
<td>required†</td>
<td>cutoff frequency (in rad/s)</td>
</tr>
<tr>
<td>deltaT</td>
<td>Real</td>
<td>π/ωc</td>
<td>coarse time step</td>
</tr>
<tr>
<td>ntimes</td>
<td>integer</td>
<td>5</td>
<td># of terms in time interpolation</td>
</tr>
<tr>
<td>correlation_function</td>
<td>string</td>
<td>defaults to sin(x)/x</td>
<td>function for time interpolation</td>
</tr>
<tr>
<td>scale_function</td>
<td>string</td>
<td>defaults to σ(z) = 1</td>
<td></td>
</tr>
</tbody>
</table>

†Specify either cutoff_freq or omega_c, but not both.

Table 2-24. – SamplingRandom function parameters.

Note that

\[ \omega_c = 2\pi \times \text{cutoff\_freq}, \]

and that only one of the two parameters omega_c and cutoff_freq can be specified. More detailed descriptions of these parameters are given in Section 6.3.15. Random time functions can be used to specify any type of random load, including pressure loads, force loads, acoustic loads, etc. Below we give an example for the case of an acoustic load.

LOAD
  sideset 1
    function = 1
    acoustic_vel = 1.0
END

FUNCTION 1
  type SamplingRandom
cutoff_freq 1000
The SamplingRandom function is a special case of a zero mean, unit variance Gaussian function. Sampling methods allow a reduced memory method of computing the time realization. In a transient analysis, the time integration step should be less than the coarse time step, “deltaT”. Statistics for the functions may be output by specifying “input_summary” in the “ECHO” section of the input file (see Section 7.7).

2.8.8. RandomLib Functions

There are two tests of the RandomLib function. One is a verification test.

In many cases, a random load on a structure may need a spatial correlation with other loads on the structure. The RandomLib function was created to address this need.  

Run time parameters for “RandomLib” functions are listed in Table 2-25, and an example is provided in input 2.4. Each parameter is described in more detail below.

The RandomLib function operates only by reading data from an external Exodus data file. The data file is an Exodus file that contains nodal scalar boundary conditions on a nodeset that covers the same nodes as the sideset. This nodeset is assumed to have the name “surface_1_nodes” where the “1” in this case corresponds to sideset 1. These nodal loads are typically generated within MATLAB code and merged with the Exodus file definition.

Currently, this function has been applied only to apply a scalar function on the nodal locations of a single sideset in the model. Such functions can be used to apply pressures (which are applied as piecewise linear functions within the elements). It can also be used to apply prescribed accelerations at the nodal locations.

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Values</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>type</td>
<td>randomlib</td>
<td>required to specify function</td>
</tr>
<tr>
<td>interp</td>
<td></td>
<td>temporal interpolation scheme</td>
</tr>
<tr>
<td></td>
<td>none=nearest</td>
<td>linear=linear interpolation</td>
</tr>
<tr>
<td>sideset</td>
<td>int/string</td>
<td>sideset id/name where pressures are applied</td>
</tr>
</tbody>
</table>

Table 2-25. – RandomLib function parameters.

2The RandomLib function is an external library interface to Sierra/SD. Additional functionality and the interface to other applications are described in separate documentation.
Input 2.4. Example RandomLib Function Specification

**type**  The specification “*type=randomlib*” is *required* to reference the *randomlib* function and its capabilities.

**interp**  A restart file contains time samples of a random function. *Sierra/SD* references these values at each time step to properly load the function. Figure 2-16 shows how interpolation influences the actual value returned.

**sideset**  Pressure is applied over a single sideset of the model. This sideset must match the definition in the load section. ¹

**exo_var**  Specification of the name of the nodeset variable that represents the nodal loading keyword *exo_var* is required. The format should be “*exo_var scalar pressure*,” where ‘pressure’ is the name of the variable used in the *Exodus* file.

¹Data for the *Exodus* file is usually provided using specialized tools such as *mkrandloadrst*. A sideset provides information about the extent of the load, and for pressure loads, it is required to identify the faces upon which the load is applied. Actual time history data is associated with a nodeset which includes the same nodes as the sideset.
Figure 2-16. – RandomLib Temporal Interpolation. Because of different time steps in the RandomLib data library and the Sierra/SD time step algorithm, the function value returned depends on the time interpolation algorithm. With interp=none, the first value returned to Sierra/SD is about -1.0, as that is the nearest time sample in the data. With interp=linear, the value returned is about -0.6. Note that round off can cause odd behavior with interp=none, even if the two data sets have the same fundamental time step.

2.8.9. Analytic Functions

Analytic functions can be used define complex mathematical functions directly in the input deck. An example of the input for this type of function is:

```plaintext
FUNCTION sine
    type analytic
    evaluate expression = "sin(2 * pi * t)"
END
```

Sierra/SD uses the STK expression parser. Analytic functions cannot be used for acceleration boundary conditions. Integration of analytic functions is not implemented.
Rules and options for composing algebraic expressions. If you choose to use the EVALUATE EXPRESSION command line, you will need to write the algebraic expressions. The algebraic expressions are written using a C-like format. Each algebraic expression is terminated by a semicolon (;). The entire set of algebraic expressions, whether a single expression or several, is enclosed in a single set of double quotes (" ").

Note: analytic function variables are case insensitive, including any local variable definitions, and there is currently no warning for multiply-defined variables (the last variable name “wins”), so caution should be taken to ensure that duplicate variable names aren’t used.

Expressions compute an output value as a function of input values. The input value of an expression is called an independent variable of the expression. An expression can use any name for the expression independent variable. The examples shown here use x for the independent variable.

Example: Return \( \sin(x) \) as the value of the function.

```plaintext
function sinx
    type is analytic
    evaluate expression is "\( \sin(x) \)"
end
```

Example: Return a piecewise linear interpolated ramp function. For \( x \) values less than 0.0 the function will return 0.0. For \( x \) values between 0.0 and 0.5 the function will return y values linearly interpolated between 0 and 100. For \( x \) values greater than 0.5 the function will return 100:

```plaintext
function pressure
    type is analytic
    evaluate expression is "
        \( (x <= 0.0) ? (0.0) : (x < 0.5) ? (x*200.0) : (100.0) \)
    "
end
```

Operators and Functions Available within Expressions The following functionality is currently implemented for the expressions:
Operators  Valid arithmetic and Boolean operations are as follows:

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Operation</th>
</tr>
</thead>
<tbody>
<tr>
<td>+, -</td>
<td>plus, minus</td>
</tr>
<tr>
<td>*, /</td>
<td>times, divide</td>
</tr>
<tr>
<td>^</td>
<td>power</td>
</tr>
<tr>
<td>==, !=</td>
<td>equivalent, not equivalent</td>
</tr>
<tr>
<td>&gt;, &lt;</td>
<td>greater/less than</td>
</tr>
<tr>
<td>&gt;=, &lt;=</td>
<td>greater/less than or equal to</td>
</tr>
<tr>
<td>!</td>
<td>not</td>
</tr>
<tr>
<td>&amp;</td>
<td>and</td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td>?:</td>
<td>ternary if, then, else</td>
</tr>
</tbody>
</table>

Parentheses  (), represents standard mathematical meaning of operation ordering.

Component indexing  value[i], accesses the ith component of a multi-component variable such as a vector, tensor, etc. This index is one-based. For a vector vec,

\[
\begin{align*}
\text{vec}_x &= \text{vec}[1] \\
\text{vec}_y &= \text{vec}[2] \\
\text{vec}_z &= \text{vec}[3]
\end{align*}
\]

for a symmetric tensor sym,

\[
\begin{align*}
\text{sym}_x &= \text{sym}[1] \\
\text{sym}_y &= \text{sym}[2] \\
\text{sym}_z &= \text{sym}[3] \\
\text{sym}_{xy} &= \text{sym}[4] \\
\text{sym}_{yz} &= \text{sym}[5] \\
\text{sym}_{zx} &= \text{sym}[6]
\end{align*}
\]

and for a full tensor full,

\[
\begin{align*}
\text{full}_x &= \text{full}[1] \\
\text{full}_y &= \text{full}[2] \\
\text{full}_z &= \text{full}[3] \\
\text{full}_{xy} &= \text{full}[4] \\
\text{full}_{yz} &= \text{full}[5] \\
\text{full}_{zx} &= \text{full}[6] \\
\text{full}_{yx} &= \text{full}[7] \\
\text{full}_{zy} &= \text{full}[8] \\
\text{full}_{xz} &= \text{full}[9]
\end{align*}
\]

Math functions  Valid mathematical operations are as follows:
<table>
<thead>
<tr>
<th>Function</th>
<th>Operation</th>
</tr>
</thead>
<tbody>
<tr>
<td>abs(x)</td>
<td>absolute value of x</td>
</tr>
<tr>
<td>mod(x,y)</td>
<td>modulus of x</td>
</tr>
<tr>
<td>min(x,y)</td>
<td>minimum value of x, y</td>
</tr>
<tr>
<td>max(x,y)</td>
<td>maximum value of x, y</td>
</tr>
<tr>
<td>sign(x)</td>
<td>sign operator: −1 if x is negative, 1 if positive</td>
</tr>
<tr>
<td>ipart(x)</td>
<td>integer part of x</td>
</tr>
<tr>
<td>fpart(x)</td>
<td>fractional part of x</td>
</tr>
</tbody>
</table>

**Power functions** Valid functions for expressions containing exponents are as follows:

<table>
<thead>
<tr>
<th>Function</th>
<th>Operation</th>
</tr>
</thead>
<tbody>
<tr>
<td>pow(x,y)</td>
<td>(x^y)</td>
</tr>
<tr>
<td>pow10(x)</td>
<td>(10^x)</td>
</tr>
<tr>
<td>sqrt(x)</td>
<td>(\sqrt{x})</td>
</tr>
</tbody>
</table>

**Trigonometric functions** Valid trigonometric operations are as follows:

<table>
<thead>
<tr>
<th>Function</th>
<th>Operation</th>
</tr>
</thead>
<tbody>
<tr>
<td>acos(x)</td>
<td>arccosine of x</td>
</tr>
<tr>
<td>asin(x)</td>
<td>arcsine of x</td>
</tr>
<tr>
<td>asinh(x)</td>
<td>inverse hyperbolic sin of x</td>
</tr>
<tr>
<td>atan(x)</td>
<td>arctangent of x</td>
</tr>
<tr>
<td>atan2(y,x)</td>
<td>arctangent of (y/x), signs of (x) and (y)</td>
</tr>
<tr>
<td>cos(x)</td>
<td>cosine of x</td>
</tr>
<tr>
<td>cosh(x)</td>
<td>hyperbolic cosine of x</td>
</tr>
<tr>
<td>sin(x)</td>
<td>sine of x</td>
</tr>
<tr>
<td>sinh(x)</td>
<td>hyperbolic sine of x</td>
</tr>
<tr>
<td>tan(x)</td>
<td>tangent of x</td>
</tr>
<tr>
<td>tanh(x)</td>
<td>hyperbolic tangent of x</td>
</tr>
</tbody>
</table>

**Logarithm functions** Valid logarithmic operations are as follows:

<table>
<thead>
<tr>
<th>Function</th>
<th>Operation</th>
</tr>
</thead>
<tbody>
<tr>
<td>log(x)</td>
<td>natural logarithm of x</td>
</tr>
<tr>
<td>log10(x)</td>
<td>base-10 logarithm of x</td>
</tr>
<tr>
<td>exp(x)</td>
<td>(e^x)</td>
</tr>
</tbody>
</table>

**Rounding functions** Valid mathematical rounding operations are as follows:

<table>
<thead>
<tr>
<th>Function</th>
<th>Operation</th>
</tr>
</thead>
<tbody>
<tr>
<td>ceil(x)</td>
<td>smallest integer value not less than x</td>
</tr>
<tr>
<td>floor(x)</td>
<td>largest integer value not greater than x</td>
</tr>
</tbody>
</table>

**Random functions** Valid functions for generating random data are as follows:
Function | Operation
--- | ---
random() | random real number $0.0 \leq x < 1.0$
random(x) | seeds the random number generator
time() | elapsed time in seconds since January 1, 1970

**Coordinate conversion functions** Valid functions for converting to and from polar and Cartesian coordinate systems are as follows:

<table>
<thead>
<tr>
<th>Function</th>
<th>Operation</th>
</tr>
</thead>
<tbody>
<tr>
<td>deg(x)</td>
<td>converts x from radians to degrees</td>
</tr>
<tr>
<td>rad(x)</td>
<td>converts x from degrees to radians</td>
</tr>
<tr>
<td>recttopolr(x,y)</td>
<td>magnitude of vector (x, y)</td>
</tr>
<tr>
<td>recttopola(x,y)</td>
<td>angle of vector (x, y)</td>
</tr>
<tr>
<td>poltorectx(r,th)</td>
<td>$x$-coordinate of angle $\theta$ at distance $r$</td>
</tr>
<tr>
<td>poltorecty(r,th)</td>
<td>$y$-coordinate of angle $\theta$ at distance $r$</td>
</tr>
</tbody>
</table>

**If, then, else ternary syntax** $A \ ? \ B : C$, where $A$ is a Boolean statement such as $(x < 2.0)$, $B$ is a statement (or set of statements) to be evaluated if $A$ is true, and $C$ is a statement (or set of statements) to be evaluated if $A$ is false. **Note:** these statements can only be used for return values, i.e. you cannot define local variables inside a ternary statement.

**Other utilities** Miscellaneous other utility functions are as follows:

- **cos_ramp**(x, xstart, xend) defines a cosine ramp loading function. For $x < x_{\text{start}}$, it returns 0.0. For $x > x_{\text{end}}$, it returns 1.0. If $x_{\text{start}} \leq x \leq x_{\text{end}}$, the function has the value $(1.0 - \cos(\pi((x-x_{\text{start}})/(x_{\text{end}}-x_{\text{start}}))))/2.0$. The primary purpose for the cosine ramp is to provide smooth loading. If attempting to run a nearly quasistatic problem with the dynamic solver, a displacement boundary condition applied via the **cos_ramp** will generally give a smooth response for a given loading time.

- **cycloidal_ramp**(x, xstart, xend) defines a cycloidal ramp function. For $x < x_{\text{start}}$ it returns 0.0. For $x > x_{\text{end}}$, it returns 1.0. If $x_{\text{start}} \leq x \leq x_{\text{end}}$, the function has the value $(x-x_{\text{start}})/(x_{\text{end}}-x_{\text{start}})-1/(2*\pi)*\sin(2*\pi/(x_{\text{end}}-x_{\text{start}})\ast(x-x_{\text{start}}))$. The primary purpose for the cycloidal front ramp is to provide smooth loading for prescribed displacements and velocities. This function has continuous acceleration derivatives at $x_{\text{start}}$ and $x_{\text{end}}$, providing a smoother response than the **cos_ramp** for prescribed displacement boundary conditions.

- **haversine_pulse**(x, xstart, xend) defines a haversine pulse function. For $x < x_{\text{start}}$, it returns 0.0. For $x > x_{\text{end}}$, it returns 0.0. If $x_{\text{start}} \leq x \leq x_{\text{end}}$, the function has the value $\text{pow}(\sin(\pi((x-x_{\text{start}})/(x_{\text{end}}-x_{\text{start}}))),2)$. Design shock loads for components are often specified as haversine acceleration pulses; this function is included to make it easier to apply this loading.
**Constants.** There are three predefined constants that may be used in an expression. These three constants are \( e \), \( \pi \), and \( \text{two}\_\pi \). Note that these constants are reserved variable names and thus cannot be redefined in an expression.

<table>
<thead>
<tr>
<th>Math symbol</th>
<th>Expression</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>( e )</td>
<td>( e )</td>
<td>2.7182818284...</td>
</tr>
<tr>
<td>( \pi )</td>
<td>( \pi )</td>
<td>3.1415926535...</td>
</tr>
<tr>
<td>( 2\pi )</td>
<td>( \text{two}_\pi )</td>
<td>2 * 3.1415926535...</td>
</tr>
</tbody>
</table>

Also, there are two predefined constant functions that can be used, \text{SIERRA}\_\text{CONSTANT}\_\text{FUNCTION}\_\text{ZERO} and \text{SIERRA}\_\text{CONSTANT}\_\text{FUNCTION}\_\text{ONE}. These two functions are equivalent to defining functions with constant expressions “0.0” or “1.0”.

**Multiple inputs** By default, we assume that analytic functions have a single input value. Typically, the input would represent time, but the variable name can be whatever you choose (e.g. “x”, “t”, or “time”). Sometimes, however it is beneficial to define functions that depend on more than one input, for example both time and the current displacement. In that case, the user must supply additional information to define the meaning of each input variable:

```
expression variable <string> = time|coord|disp|velocity|acceleration|nodeId
expression variable <string> = nodal <string>
```

Note that an 'expression variable' line is required for each independent variable, even if the name and meaning happen to be the same (e.g. \text{expression variable time = time}). Also, note that analytic functions are evaluated at each application node on the structure (i.e. each node in a nodeset). The list of supported input variables is given in table 2-26.

Nodal variables may be used in the 'expression variable' line to define independent analytic function variables. Current testing only guarantees accuracy for variables read from the input mesh, as they do not change with time. Output variables change over time, and current testing does not guarantee proper evaluation order of functions and outputs. Nonetheless, output variables are accessible through this interface.

Analytic functions are case-insensitive. \text{PI}, \text{Pi}, \text{pI} and \( \pi \) are equivalent. If it isn’t, then that is a bug. Note however that if you define both \( \pi \) and \( \Pi \) as different variables, the expression variable parser treat them as the same variable, and only use the last definition.

This section combines documentation from Sierra SM and Sierra SD. Having a shared document has pitfalls. \text{Sierra/SM} and Sierra/SD have different syntax. For example this section uses the “is” and “are” of Sierra SM. In SM usually ‘=/is/are’ can be used interchangeably. Readers should be aware that “are” ain’t defined in Sierra SD. It is not obvious what will happen if someone uses the “is” Sierra SM syntax in Sierra SD. One way to translate the Sierra SM syntax to Sierra SD syntax is to replace “is” and “are” by =.
Some parts of the documentation recommend using time as the independent variable. Sometimes the independent variable is time. Other parts of the documentation states that time() is a built in function that measures time is a surprising way. This part of the documentation discourages using time as the independent variable.

The documentation says that each algebraic expression is terminated by a semicolon. There are no examples. No one has time to add examples. There are examples in the regression test suite.

A function has a name parameter. The name is not documented. No one has time to document it.

Scientific notation is not mentioned in the documentation. If it is possible to rewrite 100000 as 1.0e5, then this would be worth documenting. What about E, d or D? The scientific notation "1.0e-5" works. No one knows about other notation.

```plaintext
function left
    type analytic
    name "other_left"
    evaluate expression = "1500*pow(sin(1.0e5*t*pi/9),2)"
end
```

### 2.8.10. Plane Wave (Time Domain)

Plane wave functions are applied primarily in scattering problems where a load on a surface is analytically described as an incident plane wave. We define the wave in terms of the following parameters.

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Values</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>type</td>
<td>plane_wave</td>
<td>identifier keyword</td>
</tr>
<tr>
<td>Direction</td>
<td>3 reals</td>
<td>direction of the wave, ( e_k = k_0/</td>
</tr>
<tr>
<td>material</td>
<td>string</td>
<td>acoustic material</td>
</tr>
<tr>
<td>K0</td>
<td>real</td>
<td>wavenumber, ( k_0 =</td>
</tr>
<tr>
<td>origin</td>
<td>3 reals</td>
<td>wave origin, ( x_0 )</td>
</tr>
</tbody>
</table>

Table 2-26. – Predefined Analytic Input Variables.
The material specification provides the wave speed $c_0$ and the fluid density $\rho_0$, and the pressure amplitude $p_0$ is specified in the \textbf{loads} section. The angular frequency is $\omega = 2\pi f$ and $k_0 = \omega/c_0$. A time-harmonic plane wave can then be written as

$$p = p_0 \cos[k_0 c_0 t - k_0 \cdot (x - x_0)],$$

(2.2)

The velocity is also computed for scattering problems,

$$v = e_k \frac{p}{\rho_0 c_0}.$$  \hspace{1cm} (2.3)

\begin{verbatim}
loads
  sideset 1 // acoustic
    acoustic_vel = 1.0
    function = 65
  sideset 2 // structure
    pressure = 1.0
    function = 65
end

function 65
  type = plane_wave
  Direction = 1 0 0
  origin = 0 0 0
  K0 = 1000
  material = air
end

material air
  acoustic
  c0=332.0
  density=1.29
end

tied data
  surface 1,2
end
\end{verbatim}

\textbf{Input 2.5. Example Planewave Function Specification}

An example is shown in input 2.5. Note that if the\texttt{ tied data} block is omitted, there is no interaction between the acoustic domain and the structure. Instead, the boundary of the acoustic domain is rigid, and the scattered pressure field is from a rigid boundary instead of from a structure with the specified material properties.

102
### 2.8.11. Plane Wave (Frequency Domain)

An incident plane wave can also be applied in the frequency domain for scattering problems. In the frequency domain, a plane wave is defined in terms of the following parameters.

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Values</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>type</td>
<td>plane_wave_freq (or iplane_wave_freq)</td>
<td>identifier keyword</td>
</tr>
<tr>
<td>Direction</td>
<td>3 reals</td>
<td>direction of the wave, ( \mathbf{e}_k = k_0 /</td>
</tr>
<tr>
<td>material</td>
<td>string</td>
<td>acoustic material</td>
</tr>
<tr>
<td>origin</td>
<td>3 reals</td>
<td>wave origin, ( \mathbf{x}_0 )</td>
</tr>
</tbody>
</table>

The wave speed \( c_0 \) and the density \( \rho_0 \) are specified by the choice of material, the frequency \( f \) is specified in the frequency block, and the particle velocity amplitude \( u_0 = p_0 / \rho_0 c_0 \) is specified in the loads block. A time-harmonic plane wave (with the time-dependence dropped) can then be written as

\[
p(\mathbf{x}) = p_0 e^{-i k_0 (\mathbf{x} - \mathbf{x}_0)}.
\] (2.4)

```plaintext
loads
  sideset 1 // acoustic
    acoustic_vel = 1.0
    function = 66
  sideset 1
    iacoustic_vel = 1.0
    function = 67
  sideset 2 // structure
    pressure = 1.0
    function = 66
  sideset 2
    ipressure = 1.0
    function = 67
end

function 66
  type = plane_wave_freq
  Direction = 1 0 0
  origin = 0 0 0
  material = air
end

function 67
  type = iplane_wave_freq
  Direction = 1 0 0
  origin = 0 0 0
  material = air
end
```
Input 2.6. Example PlanewaveFreq Function Specification

An example is shown in input 2.6. The syntax for the plane wave frequency function does not require a wavenumber, $K_0$, but is otherwise the same as plane_wave function from the previous section. Note that both a real and an imaginary plane wave frequency function are applied to their corresponding real and imaginary loads—which is currently necessary in order to apply the correct phase shift between real and imaginary parts of propagating waves—and that real and imaginary loads should be given identical amplitude, direction, and origin.

2.8.12. Planar Step Wave

The planar step wave, keyword="planar_step_wave" provides a means of applying a traveling exponential step wave to an acoustic scattering problem. The function provides both a pressure on a structure and a velocity load on an acoustic model. Parameters are listed in Table 2-27. The exponential step wave is useful for verification problems in scattering, but is not realizable physically. The pressure definition is similar to the plane wave, but employs a Heaviside step function, $H(t-t')$, where $t' = \frac{d[\vec{x}-\vec{x}_o]}{c_0}$.

$$P = P_o \cdot e^{-\beta(t-t')} H(t-t')$$  \hspace{1cm} (2.5)

A standard planar step wave function can be defined by using $\beta = 0$. This is the default behavior if no beta parameter is specified.
2.8.13. Spherically Spreading Wave

A spherically spreading wave, keyword=spherical_wave, computes the response of a point source excitation in an acoustic medium. The function applies both a pressure on the structure and a velocity load on an acoustic model. Parameters are listed in Table 2-28. Figure 2-17 illustrates the geometry.

A spherical wave is used only in transient dynamics analyses. An example input is described in input 2.7. Function 1 in the example defines the spherical wave function, which describes the geometry of the loading. The time history of the loading is referenced in the function, function 11 in the example, must be a simple function of time. It could be a linear function, a runtime compiled function or a table. It cannot be a function of space and time.
Figure 2-17. – Spherical Wave Geometry.

```
material = 1000  //material for acoustic medium
END

FUNCTION 11
   Data 0.0    0.00000
   Data 1e-6   0.00001
   Data 2e-6   0.00002
END
```

Input 2.7. Spherical Wave Example

2.8.14. Undex Structural Acoustic Loads

For Navy scattering applications, the “multicycle_bubble” and "Undex_shock_wave" functions provide a numerical function for analysis of exterior shock loading. The parameters of the loading are listed in Table 2-29. Details of the theory and implementation are available from the Navy Surface Warfare Center, Carderock Division (NSWC/CD). An example input is shown in input 2.8.

```
function 67
   type = hicks_bubble
```
<table>
<thead>
<tr>
<th>Keyword</th>
<th>Values</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>type</td>
<td>single_decay, double_decay, hicks_bubble, UndexShockwave</td>
<td>required</td>
<td>identifier keyword</td>
</tr>
<tr>
<td>charge_weight</td>
<td>real</td>
<td>required</td>
<td>in pounds of TNT</td>
</tr>
<tr>
<td>charge_location</td>
<td>3 reals</td>
<td>required</td>
<td>explosive location</td>
</tr>
<tr>
<td>waterline_depth</td>
<td>real</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>free_surface_flag</td>
<td>integer</td>
<td>1</td>
<td>acoustic material</td>
</tr>
<tr>
<td>material</td>
<td>string</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 2-29. – Undex Load Parameters. input coordinates are in inches.

```plaintext
charge_weight = 10
charge_location = 100.0 0. 50
waterline_depth = 10
free_surface_flag = 1
material = water
end

material water
  acoustic
c0=4872
density=62.4
end
```

### Input 2.8. Example Hicks Bubble Function Specification

A “free_surface_flag” of one indicates generation of an applicable image source above the surface of the water, where a “free_surface_flag” of 0 indicates a load without a free surface. In this routine, “z” is upwards and normal to the water surface. The depth is the distance below the water surface, i.e.

\[
\text{waterline_depth} = z_{\text{waterline}} - z_{\text{charge}}
\]

where \(z_{\text{charge}}\) is the \(z\) component of the charge location. If the free surface flag is not specified, no effects of the surface are included.

One special type of shock wave (contained within the general “Undex” shock wave function definition), is that of a single decay shock wave. The single decay function has the
following simple analytical solution:

\[ P(w, r, t) = \begin{cases} 
0.0 & t < \text{toa}(r) \\
\frac{P_{\text{max}}(w, r)}{\exp\left(\frac{(t - \text{toa}(r))/\theta(w, r)}{1}\right)} & t > \text{toa}(r) 
\end{cases} \]

Where:

\[ w = \text{charge weight (lb. TNT)} \]
\[ r = \text{standoff distance (ft)} \]
\[ t = \text{time (seconds)} \]
\[ c = \text{sound speed in water (ft/s)} \]
\[ P_{\text{max}}(w, r) = k_1 \left( \frac{\sqrt[3]{w}}{r} \right)^{a_1} \]
\[ k_1, a_1 = \text{similitude constants} \]
\[ \theta(w, r) = k_2 \sqrt[3]{w} \left( \frac{\sqrt[3]{w}}{r} \right)^{a_2} \]
\[ k_2, a_2 = \text{similitude constants} \]
\[ \text{toa}(r) = \frac{r}{c} = \text{time of arrival} \]

### 2.8.15. Fluid Structure Interaction

For fluid-structure interaction (FSI) applications, the FSI keyword provides a means of applying a prescribed nodal pressure load along the wetted surface. The FSI function is referenced in the Sierra/SD input as follows,

```plaintext
Loads
  sideset 1
  pressure 1
  scale 1
  function 1
End

Function 1
  type = FSI
End
```

The above input file assumes sideset 1 is the wetted surface. Sierra/SD will communicate nodal locations of the sideset to the fluid. These are the locations at which pressures are sent to Sierra/SD. Then, Sierra/SD calculates a consistent load based on the values at the nodes. Finally, if restarts are needed, “restart=auto” is required in the solution section.

Sierra/SD also supports two-way coupling for Fluid-Structure interaction. Interpolation from structural nodes to fluid nodes and from fluid nodes to structural nodes is
implemented and unit tested. Figure 2-18 shows the infrastructure for FSI. There are many details to use of this coupling, such coupling is turned with the “acoustic_coupling” transient solution option.

Coupling to the Sierra code Fuego through Sierra Toolkit Transfers is currently being implemented. Users of current sprint releases must be careful to not use the “Fuego_coupling” transient solution option, as this is only partially implemented.

![Fluid-Structure Interaction (FSI) Infrastructure](image)

**Figure 2-18.** – Fluid-Structure Interaction (FSI) Infrastructure.

### 2.8.16. Blending

In some applications, a combination of functions is necessary to accurately describe the loading. For example, a blended shock/bubble function is applied to a ship surface. The shock wave reaches the surface first, and is the effective loading until the bubble function arrives. Once the bubble arrives, the shock may be ignored, as it may not propagate through the cavitated region. This is illustrated for the first crossing case in Figure 2-19, and a typical input for this is shown in input 2.9. Parameters of the blended function are shown in Table 2-30. Several examples for the more general Nth crossing case are shown in Figure 2-20, with a typical input shown in input 2.10.

<table>
<thead>
<tr>
<th><strong>Keyword</strong></th>
<th><strong>Parameter</strong></th>
<th><strong>Description</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td>Method</td>
<td>string</td>
<td>“first_crossing”, “second_crossing” or “Nth_crossing”</td>
</tr>
<tr>
<td>N</td>
<td>int</td>
<td>(for “Nth_crossing”) number of crossings</td>
</tr>
<tr>
<td>Primary Function</td>
<td>string</td>
<td>shock function</td>
</tr>
<tr>
<td>Secondary Function</td>
<td>string</td>
<td>bubble function</td>
</tr>
</tbody>
</table>

**Table 2-30.** – Blended Function Parameters.
**Figure 2-19.** – Illustration of first crossing blended function.

```plaintext
Loads
  sideset 55
    pressure 1
  function 55
end

function 55       // blended shock/bubble
  type = blended
  method = first_crossing
  primary function 551
  secondary function 552
end

function 551
end

function 552
end
```

**Input 2.9.** Blended First Crossing Function Example
Figure 2-20. – Illustration of Nth crossing blended functions.

Load
  sideset 55
  pressure 1
  function 55
end

function 55       // blended shock/bubble
  type = blended
  method = Nth_crossing
  N = 2
  primary function 551
  secondary function 552
end

function 551
end

function 552
end

Input 2.10. Blended Nth Crossing Function Example
This section provides for input of a matrix function as is used in a cross correlation matrix for input to a random vibration analysis. In the limit of a single input these reduce to a single function. Note that a matrix-function can have arbitrary symmetry and can be complex. An important feature of the matrix-function is that each entry of the matrix is a function of frequency (or time).

The Matrix-Function is illustrated in the following example.

```
Matrix-Function 1
    name 'cross-spectral density'
symmetry=Hermitian
dimension=2x2
nominalt=20.1
data 1,1
    real function func11 scale 1.0
data 1,2
    real function func12
    imag function func121 scale -3.0
data 2,2
    real function func22 scale 0.5
End
```

Matrix functions have the following parameters.

**NAME** allows you to optionally enter a string by which the matrix-function will be identified in subsequent messages.

**SYMMETRY** identifies the matrix symmetry. Options are “none”, “full”/“symmetric”, “asymmetric” and “Hermitian”. If the matrix is not square, only “none” can apply. The default for this optional parameter is “symmetry=none”.

**DIMENSION** specifies the dimension of the matrix. If not specified, it defaults to $1 \times 1$. The dimension is specified as the number of rows, an “x” and the number of columns. No space should be entered between the terms.

**DATA** Each data entry specifies one entry in the matrix-function. The data entry must be immediately followed by the matrix location specified as a row, column pair. Again, no spaces may be inserted in the location entry. The `data` parameters uses two keywords.

- “real” identifies the real component of the entry. It must be followed by a function reference (see Section 2.8), and optionally by a scale factor.
- “imag” identifies the imaginary component of the entry. It must be followed by a function definition, and an optional scale factor.
nominalt nominalt Used only for echoing the matrix values. If input_summary is specified as an “ECHO” option (see Section 7.7) general information from the matrix function are written to the log file (the .rslt file). If, a nominalt entry also exists, then the matrix entries are written for that nominal time (or frequency). Only one such output can be specified. It provides a means of checking the input to assure the matrix values are correct at a single time (or frequency) value.

2.8.18. Alternate Table Interface

An alternate Table input is provided. See Section 2.8.19 for details about tables. However, for many inputs, the individual specification of each function on each matrix element is both tedious and inefficient. Table input is provided primarily for efficiency reasons. It cannot be mixed with the individual methods, i.e. if the table keywords are used, the “data” keyword must not be used.

Application of table input to matrix-functions requires three tables for:

1. Real valued data
2. Imaginary valued data
3. A table which associates each nonzero row and column of the matrix-function with appropriate rows of the real valued and imaginary valued data.

Three new keywords are introduced:

Real Table which is a two-dimensional table containing all the real valued entries for each entry in the matrix. Each column contains the frequency data for that entry.

imag Table which is a two-dimensional table containing all the imaginary (complex valued) entries for each entry in the matrix. Each column contains the frequency data for that entry.

Table Index which is a two-dimensional table providing a map from the matrix elements to the data columns in the real and imag tables. This index is a 4 column table. Columns 1 and 2 are the row, column index of the matrix-function. Column 3 is the row index of the real data, while column 4 is the row index of the imaginary data. If the value in column 3 or 4 is zero then the corresponding data is zero See the example in input 2.11.

Note that the table entry has a fixed step, so the same number of values must be provided in each column.

```
Matrix-Function 1
   name 'spectral density'
   dimension=2x2
   symmetry=Hermitian
   real Table real_data
   imag Table imag_data
```
Input 2.11. Example Input for a Matrix-Function using Tables

Each matrix entry in the matrix-function must reference a row of a two-dimensional table. In the table columns contain the frequency response for that entry. The number of rows required for each table depends on the matrix symmetry and on the index in the “Table Index”.

2.8.19. Table

A (1 dimensional) table is implemented by including a file with one value per line. 1-dimensional tables have identical behavior to linear functions (section 2.8.2) while typically being much faster and more memory efficient, especially for a large number of data points.

A small section in the main input deck specifies the initial time and time increment. The data must be sampled at a uniform interval.
Tables are used by being referenced in other sections of the input deck. Tables offer support for multi-dimensional data (up to dimension 4). Note that tables of dimension greater than 1 are complicated.

Each Table includes a number of required and optional parameters, as shown below.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>dimension</td>
<td>optional</td>
<td>number of dimensions in the table</td>
</tr>
<tr>
<td>size</td>
<td>required</td>
<td>table size in each direction</td>
</tr>
<tr>
<td>datafile</td>
<td>required</td>
<td>ASCII file containing the values at each point</td>
</tr>
<tr>
<td>dataline</td>
<td>required</td>
<td>flag indicating that all data values will follow.</td>
</tr>
<tr>
<td>origin</td>
<td>zero</td>
<td>origin of the table (for scaling)</td>
</tr>
<tr>
<td>delta</td>
<td>1</td>
<td>interval between points in each direction</td>
</tr>
<tr>
<td>rowfirst</td>
<td></td>
<td>transpose data on input</td>
</tr>
</tbody>
</table>

The `dimension` identifies the table shape. For example, `dimension=2` indicates a table of xy values. If this parameter is not defined, the dimension will be automatically inferred from the number of entries in `size`. If it is defined, it must match the inferred value, and thus only serves as a check on the table size.

The `size` parameters indicate the individual table hyper-cube dimensions. For example, in a table of `dimension=2`, the `size` parameter indicates the number of rows and columns in the table. The total number of entries is the product of all the terms in the size.

The text file containing the table data values is specified using the `datafile` parameter. Data values are separated by white space. The layout of the file is not important, but the order is important. The first dimension cycles the fastest. For a `dimension=2` table, the file list begins with the entries for column 1. The number of entries in the file must match the table size. Comments are not permitted in the `datafile`, but white space is permitted.

The `dataline` parameter indicates that the tabular data is included in this file following the parameter. If `dataline` is specified then `datafile` must not be specified. The format is identical to the datafile. For performance reasons, it is best to use `dataline` for smaller data sets, and `datafile` for larger.

The `rowfirst` is provided to transpose the data on input. It applies only to 2D tables. If this keyword is present, then the table values will be interpreted as if the table had been transposed.

Both the `origin` and the `delta` parameters are optional values provided for interpolation. The implicit integer entries of the table are converted to real values for function evaluation by use of these parameters.

Function evaluations within the range of the table can be linearly interpolated. The range in each direction is determined by the following.

\[
\text{origin}_i < \text{range}_i < \text{origin}_i + (\text{delta}_i \cdot \text{size}_i)
\]  

(2.6)
Evaluations of the table for regions outside the valid range will use of the value of the nearest data point, just as with linear functions.

In contrast to a function (see Section 2.8), tables require memory only as needed. All processors store the full input file in memory. However, tables can store a large amount of data in the datafile. This file is opened and data is read from it only as needed. For this reason, tables are preferred over functions when only a few processors may need access to a large amount of data. Tables are the only option when a function of more than one variable is required.

An example of a two-dimensional table definition is shown below.

```plaintext
Table example-2D-table
  dimension=2
  size = 200 300 // note: don’t put in an x
  origin 1.0 0.0 // optional. defaults to 0 0
  delta 1.0 0.9 // optional. defaults to 1 1
  datafile 'multi_dimensional_table.txt'
END
```

2.9. Multipoint constraints

A multipoint constraints (MPC) is a constraint equation applied directly to the stiffness matrix. Some analysis codes treat them as pseudo elements. An MPC is not an element, and is inaccessible through Exodus.

An MPC is a displacement constraint

$$\sum_{i} c_i u_i = r,$$

The (nonzero) coefficients $c_i$ are real. The number of (nonzero) coefficients is assumed to be approximately 1. The $u_i$ are displacement of degrees of freedom. By default, $r$ vanishes.

Unlike many Finite Element programs, Sierra/SD does not support user specification of constraint and residual degrees of freedom (DOF). In serial solvers the partition of constrained and retained degrees of freedom is performed simultaneously by Gauss elimination with full pivoting so the constrained degrees of freedom are guaranteed to be independent. For GDSW the constraints are specified as Lagrange multipliers which involves no such partitioning. Redundant specification of constraint equations is handled by elimination of the redundant equations and issue of a warning. User selection of constrained DOF in NASTRAN has inconvenienced analysts who must ensure that the constrained DOF are independent and never specified more than once.

Each MPC is specified in the input file with a section descriptor. Note that a separate section is required for each equation (or degree of freedom eliminated). An optional
coordinate system may be specified on the input \(^2\); see section 2.7. The MPC will be stored internally in the basic coordinate system (coordinate frame 0). The input consists of a triplet listing the node or nodeset, a degree of freedom string, and the coefficient of that degree of freedom. The degree of freedom strings are \(x, y, z, Rx, Ry, Rz\). They are case-insensitive. If the global ID of the node in the MPC does not exist in the model, the code will exit with a fatal error.

GDSW is required for inhomogeneous MPCs. The solution method must not be QEVP. Sierra/SD will exit with a fatal error if a model including non-homogeneous MPCs violates either of these requirements.

The MPC can apply some general commands that apply to the whole MPC as shown in Table 2-32. Additionally, the MPC can include multiple lines that define the MPC equation entries as shown in Table 2-33.

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>coordinate</td>
<td>string</td>
<td>Optional coordinate frame for MPC</td>
</tr>
<tr>
<td>rhs</td>
<td>real</td>
<td>Optional right-hand side constant</td>
</tr>
</tbody>
</table>

**Table 2-32.** – General MPC commands.

<table>
<thead>
<tr>
<th>Command Line</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>integer (x/y/z/\text{rx}/\text{ry}/\text{rz}) real</td>
<td>Equation entry using single global node id</td>
</tr>
<tr>
<td>nodeset (\text{nodeset}_\text{id} (x/y/z/\text{rx}/\text{ry}/\text{rz}) real</td>
<td>Equation entry using nodeset with EXACTLY one node</td>
</tr>
</tbody>
</table>

In this first example the MPC is defined with respect to a coordinate system \textit{dir1}. The displacement at the \(x\) degree of freedom of node 4 is constrained to be equal to the average \(x\) degree of freedom of nodes 2 and 3 plus 5, i.e. \(u_{(4,x)} - 0.5 \times u_{(2,x)} - 0.5 \times u_{(3,x)} = 5.0\).

```plaintext
MPC
  coordinate dir1
  4 x 1.0
  2 x -0.5
  3 x -0.5
  rhs 5.0
END
```

\(^2\)At this time, all the nodes in an MPC must be associated with the same coordinate system.
In this next example, the $x$ degree of freedom of the node in the nodeset 101 is constrained to be equal to the $y$ degree of freedom of the node in the nodeset with name *aft_plate*. Note that both nodesets must contain exactly one node.

```
MPC
  nodeset 101 x 1.0
  nodeset aft_plate y -1.0
END
```

**IMPORTANT**

Constraints are handled in various ways by the linear solvers. In the serial solver, the dependent degrees of freedom are eliminated before the matrices are passed to the solver. In parallel, we use Lagrange multipliers to handle the constraints. There is currently no user control of constraint handling methods.

Note also that there are practical differences between rigid elements (described in the following sections) and constraint equations that are nominally identical. For parallel solutions, we are currently using an augmented Lagrange type solution method with the rigid links. This means that terms are added to the stiffness matrix in parallel with the constraints. In most cases, this renders the matrices positive definite, and increases robustness and solution performance with no penalty for accuracy. Thus, rigid links are recommended whenever possible in parallel solutions.

Replacing rigid links with stiff beams may cause ill conditioning and simulation inaccuracies.
Sierra/SD supports a wide variety of different analyses or solution methods. Input consists of an Exodus mesh file and a text input file. Solution methods are specified in the text input file in the solution section.

The Solution section defines the type of physics to simulate. Analysis types are shown in Tables 3-34, 3-35, 3-36, 3-37. Relevant options are given in the detailed description of each solution case. Also, general options are shown in Table 2-10 and are described in Section 2.4.

Table 3-34. – Eigen Solution Types.

<table>
<thead>
<tr>
<th>Solution Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>eigen</td>
<td>Eigen solution to extract natural vibration modes of K,M</td>
</tr>
<tr>
<td>aeigen</td>
<td>Eigen solution with Anasazi</td>
</tr>
<tr>
<td>buckling</td>
<td>Eigen solution solving for buckling modes</td>
</tr>
<tr>
<td>CBR</td>
<td>Craig-Bampton reduction for creation of superelements</td>
</tr>
<tr>
<td>geometric_rigid_body_modes</td>
<td>Exact analytic definition of the rigid body modes</td>
</tr>
<tr>
<td>blk_eigen</td>
<td>Eigen solution on a block by block basis</td>
</tr>
<tr>
<td>residual_vectors</td>
<td>Modal truncation augmentation</td>
</tr>
<tr>
<td>largest_ev</td>
<td>Largest eigenvalue of K,M</td>
</tr>
<tr>
<td>qevp</td>
<td>Solution to quadratic eigenvalue problem</td>
</tr>
</tbody>
</table>

Table 3-35. – Modal Solution Types.

<table>
<thead>
<tr>
<th>Solution Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ModalFrf</td>
<td>Frequency response using modal displacement or modal acceleration</td>
</tr>
<tr>
<td>modalranvib</td>
<td>Random vibration using modal superposition</td>
</tr>
<tr>
<td>modalshock</td>
<td>Shock response spectra using</td>
</tr>
<tr>
<td>modaltransient</td>
<td>Transient analysis using modal superposition</td>
</tr>
</tbody>
</table>
Table 3-36. – Direct Solution Types.

<table>
<thead>
<tr>
<th>Solution Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>directfrf</td>
<td>Direct computation of frequency response functions</td>
</tr>
<tr>
<td>NLstatics</td>
<td>Nonlinear static solution</td>
</tr>
<tr>
<td>NLtransient</td>
<td>Nonlinear transient solution</td>
</tr>
<tr>
<td>statics</td>
<td>Static solution</td>
</tr>
<tr>
<td>transhock</td>
<td>Shock response spectra using direct implicit transient</td>
</tr>
<tr>
<td>transient</td>
<td>Linear transient solution</td>
</tr>
</tbody>
</table>

Table 3-37. – Preprocessing and Postprocessing Solution Types.

<table>
<thead>
<tr>
<th>Solution Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>receive_sierra_data</td>
<td>Import stress, displacement, and material state from a preload Sierra/SM analysis</td>
</tr>
<tr>
<td>checkout</td>
<td>Check input for errors, skips matrix formation and solves</td>
</tr>
<tr>
<td>dump</td>
<td>Check input for errors, form matrices, skips solve</td>
</tr>
<tr>
<td>CJdamp</td>
<td>Approximate modal damping contributions</td>
</tr>
<tr>
<td>ddam</td>
<td>Dynamic design analysis method (U.S. Navy)</td>
</tr>
<tr>
<td>preddam</td>
<td>Gather data for use by ddam solution case</td>
</tr>
<tr>
<td>gap_removal</td>
<td>Do contact search and apply the contact gap removal algorithm</td>
</tr>
<tr>
<td>superposition</td>
<td>Expand superelement results to physical degrees of freedom</td>
</tr>
<tr>
<td>tangent</td>
<td>Compute tangent stiffness after a nonlinear load step</td>
</tr>
<tr>
<td>tsr_preload</td>
<td>Import data for thermal structural response</td>
</tr>
<tr>
<td>fatigue</td>
<td>Postprocess modal random vibration results to predict high cycle fatigue life</td>
</tr>
<tr>
<td>MPF</td>
<td>Compute and output modal participation factors</td>
</tr>
<tr>
<td>waterline</td>
<td>Determine waterline of a floating structure</td>
</tr>
</tbody>
</table>

3.1. **Defining Solution Cases**

It is important to distinguish the two different kinds of Solution section. A Solution section may do one analysis. In this case a `loads` section is used. Or a Solution section may do multiple analyses. Here each case may specify a `load` section. Here’s an example of a Solution section that does one analysis. The eight lowest eigen modes of a structure are requested.
A more complex analysis could perform a static preload, followed by computation of updated tangent stiffness, and then a linearized eigen analysis with this input:

```plaintext
Solution
  eigen
  nmodes 8
end
```

```plaintext
Solution
  case 'NonlinearStatics'
    nlstatics
    load=10
  case 'tangent'
    tangent
  case 'eig'
    eigen
    nmodes 8
end
```

### 3.2. Multicase Solutions

The solution methods of Tables 3-34, 3-35, 3-36, and 3-37 may be a part of a multicase solution. Using one input deck, it is possible to run any valid sequence of simulations. For example, a static preload, followed by computation of updated tangent stiffness, and then a linearized eigen analysis.

In a multicase analysis, each solution step is delineated by the `case` keyword. The word `case` indicates the beginning of a solution case. Cases are run sequentially from top to bottom. Indenting to clarify this hierarchical relationship is recommended, but not required. Each `case` keyword must always be followed by a label. The label is used to set the output file name and is used in a variety of informational, warning, and error messages. Note: the label name for each case must be unique.

The results and outputs from one case will be used to drive subsequent cases. For example, the modes computed by an `eigen` solution case will be used in a subsequent `modaltransient` solution case. As another example the stresses and displacements read in from `receive_sierra_data` solution case will be used to define the initial geometry and geometric stiffness for a subsequent eigen solution case.

In a multicase solution, the system matrices (mass, stiffness and damping) will typically be computed once. Matrix updates between solutions may be specified by selecting the `tangent` keyword (see Section 3.28).
3.2.1. Multicase Options

A solution type has specific options. Only transient solution cases have time steps for example. On the other hand, Table 3-38 shows options that may apply to each solution case. They may be specified either above the case control sections, or within the section. The specification above the case control section is the default value. Specifications within the case sub-blocks apply to that sub-block. In the example given in 3.2.2, the restart options are thus “none” for most sub cases, but “read” for the eigen analysis and auto for the linear transient.

<table>
<thead>
<tr>
<th>Option</th>
<th>Description</th>
<th>Options</th>
</tr>
</thead>
<tbody>
<tr>
<td>restart</td>
<td>Restart options</td>
<td>see Section 2.4.2</td>
</tr>
<tr>
<td>solver</td>
<td>selection of solver</td>
<td>see Section 2.4.3</td>
</tr>
</tbody>
</table>

The default load applied to each solution case in a multicase solution is prescribed by the loads block if present in the input file. Loading prescribed by a loads block can be overridden by specifying a load in an individual solution case in the multicase solution. Only one load can be applied in each solution case. In the example given in 3.2.2, load ‘10’ will be applied during the nonlinear statics solution case (case 'Nonlinear_Statics'). For more information on loads see Section 6.3 and for load see Section 6.3.1.

3.2.2. Multicase Example

In the example which follows, a nonlinear statics computation is followed by a tangent stiffness matrix update. The updated matrix is then used in an eigen analysis. Two sets of Exodus output files will be written. Output from the statics calculation will be in files of the form ‘example-nlstatics.exo’. Eigen results will be in the form ‘example-eig.exo’. The tangent solution normally produces no output in the Exodus format.

Transient solution cases are run sequentially in time. For the example below ‘trans1’ will start at time 0.0 and step through 100 steps of at a step size of 1e-8 and then through 4000 steps at a step size of 1e-6. The final time value of case ‘trans1’ will be \(10^{-8}\times10^{2} + 10^{-6}\times4\times10^{3} = 0.004001\). Case ‘trans2’ will start at 0.004001 and run an additional \(10^{-4}\times210^{2}\). This will end at time 0.024001.

```plaintext
Solution
  restart=none
  title='example multicase'
  case 'nlstatics'
    nlstatics
    load=10
```
case 'tangent'
  tangent
case 'eig'
  eigen
  restart=read
case 'trans1'
  transient
  restart=auto
  time_step 1e-8 1e-6
  nsteps 100 4000
  flush 50
  rho=0.9
  load=20
case 'trans2'
  transient
  restart=auto
  time_step 1e-4
  nsteps 200
  flush 10
  load=20
end

3.3. Checkout Solution Case

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Type</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 3-39. – Checkout Solution Case Parameters.

The checkout solution method tests out various parts of the code without forming the system matrices or solving the system of equations. This solution method may be used to check for syntax errors, use of nonexistent blocks and sidesets, or completeness of input. Generally checkout would be used to fix all such errors on a local serial platform before allocating expensive resources or waiting in a queue to run a full solution. The checkout solution case performs the same type of input checking as the \texttt{--check-syntax} command line argument.
3.4. CJdamp Solution Case

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Type</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
</table>

Table 3-40. – CJdamp Solution Case Parameters.

The CJdamp solution provides a method to compute the equivalent modal damping terms introduced from material damping in lightly damped viscoelastic materials. It is based on a development by Conor Johnson et al.\textsuperscript{30} CJdamp is an approximate method which assumes that the mode shapes and frequencies are not modified by the damping. The modal damping is related to the fraction of energy in block.

The CJdamp method is effectively a post processing step following an eigen analysis and must be run in a multicase solution. For each of the modes in the eigen analysis, a strain energy is computed on an element basis. These are summed at the block level.

\[
SE^i_j = \sum_{\text{elem}} \phi_i^T K^{\text{elem}} \phi_i \tag{3.1}
\]

The total strain energy \( TSE^i \) is the sum of the strain energy contributions in mode \( i \) from all blocks. We define the block strain energy ratio for mode \( i \) as,

\[
R^i_j = \frac{SE^i_j}{TSE^i} \tag{3.2}
\]

The CJdamp contribution for the modal damping of mode \( i \), is given by,

\[
\zeta_i = \frac{1}{2} \sum_j R^i_j \eta_j(f_i) \tag{3.3}
\]

\( \eta_j(f_i) \) is the CJetaFunction contribution from block \( j \) evaluated at the natural frequency of mode \( i \). More information is available in Section 4.4.9.

\textbf{Note that cases following the CJdamp solution will include the computed damping as part of their damping calculation.}

Example,

```plaintext
Solution
  case eig
    eigen nmodes=30
  case Johnson
```

124
3.5. Craig-Bampton reduction Solution Case

### Parameter Table

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Type</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>nmodes</td>
<td>integer</td>
<td></td>
<td>Number of fixed interface modes</td>
</tr>
<tr>
<td>shift</td>
<td>Real</td>
<td>-1.0e6</td>
<td>Shift to apply to matrix system to allow solving singular systems. See Section 3.10.2</td>
</tr>
<tr>
<td>correction</td>
<td>node/values/vectors</td>
<td>values</td>
<td>Correction method for rigid body modes</td>
</tr>
<tr>
<td>RbmDof</td>
<td>string</td>
<td>123456</td>
<td>Defines which rigid body modes to which correction=vectors applies</td>
</tr>
<tr>
<td>ModalFilter</td>
<td>string</td>
<td>none</td>
<td>Modal filter to define modes to retain. See Section 3.10.2.2</td>
</tr>
</tbody>
</table>

Table 3-41. – Craig-Bampton reduction Solution Case Parameters.

It can be advantageous to reduce a model to its interface degrees of freedom. This is important in satellite work, where the model of the satellite may be much larger than the model of the remainder of the missile. Reduction of the satellite model to a linearized, Craig-Bampton model makes it possible to share the dynamic properties of the model without requiring details of the interior. There are many types of component mode synthesis techniques (or CMS), of which the Craig-Bampton approach is one of the more popular. In this approach the model is reduced to a combination of fixed interface and constraint modes. The fixed interface modes are eigenvectors of the system with all interface degrees of freedom clamped. A constraint mode is the deformation that results if one interface degree of freedom receives a unit displacement, and all other interface degrees of freedom are zero. There is one constraint mode per interface degree of freedom.

The Craig-Bampton reduction solution reduces an entire structural model to its reduced system and transfer matrices. Options are listed in the table below, and correspond to the parameters required for an eigen analysis (Section 3.10). In addition, a CBModel section must be defined elsewhere (see Section 3.5.1). Any boundary conditions specified are applied before reducing the model.
We note that sensitivity analysis can be performed in Craig-Bampton reduction, though the process is somewhat different from other types of sensitivity analysis. Section 3.5.1 contains more information about sensitivity analysis in Craig-Bampton models.

The method will write system matrices and general information. Each of the parameters is described below.

**nmodes**: The CB model is composed of fixed interface modes and constraint modes. The number of constraint modes is determined by the interface. *nmodes* selects the number of fixed interface modes. The fixed interface modes are eigenvectors of the interior of the structure, and provide a basis for internal deformation. Any number of these modes may be specified. Typically, frequencies up to about twice the system frequency are required for accuracy.

**correction**: As shown in the theory manual, the null space of the stiffness matrix is determined by the sum of two large terms: \( \kappa_{cc} = K_{cc} + K_{cv} \psi \). With parallel iterative solvers, it may be difficult to determine this quantity as accurately as desired. In particular, it is possible for errors in the solver to render the reduced matrices negative definite, which can cause instability in subsequent transient analysis. It is strongly recommended that low solution tolerances be used in developing CB models. In addition, the matrix may be post processed to correct these errors. The post processing options are as follows:

- **none** no correction will be applied.
- **values** (default) no corrections will be made to the eigenvector space, but the negative eigenvalues will be adjusted to zero.
- **vectors** This option is available in models that have no boundary conditions, so that the CBR model is floating and has six rigid body modes. If boundary conditions are present, there is a fatal error. Additionally, if the model does not have six rigid body modes for another reason (for example a gap contact constraint that impedes rotation) then the correction=vectors option is not well posed, should not be used, and could cause a serious degradation of model behavior. The correction=vectors will compute Zero energy eigenvectors geometrically (exactly), and these are used to correct both the eigenvalues and the eigenvectors. This is more involved than correcting the eigenvalues alone, but it is not a significant computational cost, and can improve the usefulness of the resulting model.

  If **correction=vectors** is selected, one may also optionally determine which zero energy modes are required. This word **RbmDof** activates this. It is followed by a string indicating which dofs are active on the interface. The string contains the numbers 1 through 6, where 1 represents translation in the \( x \) coordinate direction. These specifications apply in the basic coordinate frame.
Inertia Tensor for Craig-Bampton Reduction. A reduced inertia matrix, $I_{vv}$ may be output from a CBR (Craig-Bampton Reduction) analysis. The Inertia Tensor may be used to apply initial conditions to the superelement in some applications. The input file syntax is described in the CBModel section, 3.5.1. $\Phi$ is the matrix of mode shapes used for the CBR analysis. It consists of both fixed-interface modes and constraint modes. $I_{vv}$ is defined by

$$I_{vv} = \Phi^T R,$$

The number of rows in $I_{vv}$ is the number of CBR modes, and the number of columns is the number of rigid body vectors. For example, for the three translational rigid-body modes and assuming three degrees-of-freedom per finite element node,

$$R = \begin{bmatrix}
1 & 0 & 0 \\
0 & 1 & 0 \\
0 & 0 & 1 \\
1 & 0 & 0 \\
0 & 1 & 0 \\
0 & 0 & 1 \\
\vdots & \vdots & \vdots
\end{bmatrix}.$$

Mass Inertia Matrix for Craig-Bampton Reduction. The Mass Inertia Matrix is used in some applications to apply a load to the interior degrees of freedom of a model. 3 The mass inertia matrix is defined as,

$$I_m = \Phi^T M R,$$

where $M$ is the mass matrix of the unreduced model.

---

3 Neither the inertia tensor, nor the mass inertia matrix may be applied in Sierra/SD. They are output quantities.
Note the following limitations for the CBR method.

- MPCs may no share nodes with interface nodes.
- The entire reduced order model and associated transfer matrix must fit into memory. On a parallel machine, this memory is required on every processor. The dimension of the model is the sum of the numbers of constraint and fixed interface modes.
- Static solutions with all interface degrees of freedom clamped are part of the reduction process. If the interface dofs do not fully constrain the system, then the linear system may be singular. In such cases the solution is not reliable. Due diligence includes verifying the reduced order eigen solution against the full system. One may also compare the retained mass.

3.5.1. CBModel

The CBModel section provides a method of specifying information related to a Craig-Bampton model reduction of the entire structure. It is required by the CBR method (section 3.5).

The “interface” is that portion of the model which will interface to the external structure. The interface is defined by collections of nodes specified as nodesets or sidesets. After eliminating boundary conditions, the active degrees of freedom on the nodes become the interface.

The input deck keywords shown in Table 3-42 are described below.

nodeset: The nodeset keyword specifies the nodes to be placed in the interface. Nodesets are defined in the Exodus file. A nodeset ID or name must follow the nodeset keyword. Alternatively, a list of nodesets (in MATLAB type format) can be specified. This is identical to the history file definition of Section 7.3, and follows the rules for integer lists detailed in Section 2.1.

sideset: A sideset may also be used to specify the interface nodes. Any number of nodeset and sideset combinations are allowed. The interface is the union of all such entries.

format: The preferred format is the netcdf format. This is a superset of the Exodus format. It is the format that must be used if the reduced model is to be inserted into another Sierra/SD model as a superelement. The DMIG format is for use with NASTRAN. It contains only the reduced system matrices (no maps, coordinates, etc). The MATLAB format is a convenience.

file: The file keyword is required to specify the output file name.
Table 3-42. – CBModel Parameters.

<table>
<thead>
<tr>
<th>Keyword</th>
<th>type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>nodeset</td>
<td>int/string/list</td>
<td>Exodus nodeset name(s) and/or id(s).</td>
</tr>
<tr>
<td>sideset</td>
<td>int/string/list</td>
<td>Exodus sideset name(s) and/or id(s).</td>
</tr>
<tr>
<td>format</td>
<td>string</td>
<td>Specifies the output format.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>MATLAB - MATLAB.m format</td>
</tr>
<tr>
<td></td>
<td></td>
<td>DMIG - NASTRAN DMIG format</td>
</tr>
<tr>
<td></td>
<td></td>
<td>DMIG* - NASTRAN long DMIG format</td>
</tr>
<tr>
<td></td>
<td></td>
<td>netcdf - netcdf format †</td>
</tr>
<tr>
<td>file</td>
<td>string</td>
<td>Specifies the file name for output.</td>
</tr>
<tr>
<td>GlobalSolution</td>
<td>bool</td>
<td>‘yes’ to compute the eigen solution of the reduced system.</td>
</tr>
<tr>
<td>inertia_matrix</td>
<td>bool</td>
<td>‘yes’ to compute the inertia tensor</td>
</tr>
<tr>
<td>sensitivity_method</td>
<td>string</td>
<td>Specifies the method to compute CBR sensitivities.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>constant_vector - constant vector method</td>
</tr>
<tr>
<td></td>
<td></td>
<td>finite_difference - finite difference method</td>
</tr>
<tr>
<td>spoint_offset</td>
<td>integer</td>
<td>Offset for spoints in DMIG format</td>
</tr>
</tbody>
</table>

†The netcdf format is the database upon which exodusII is built. Seacas tools, Matlab and Python can all read netcdf files.

**GlobalSolution:** As a convenience, we will optionally compute the eigenvalues of the reduced system. It is strongly recommended that these values be compared with the eigenvalues of the full system to ensure that the model has converged over the frequency of interest. Moreover, users are strongly advised to review section Craig-Bampton Model Reduction of Sierra/SD How To\(^2\) before computing superelements.

**inertia_matrix:** The inertia matrix defined in Section 3.5 is optionally computed and written to the super-element files with \( i \) the reduced mass and stiffness matrices.

**sensitivity_method** Currently, the constant vector and finite difference methods are available for computing sensitivities for Craig-Bampton reduction. The default is the constant vector method.

**spoint_offset** NASTRAN DMIG output defines “spoint” variables that store generalized degrees of freedom. The identifiers the generalized output variables must be unique. The range of identifier output may be specified with this option. By default, “spoint_offset” is 10000 which means spoints are numbered as 10001, 10002, 10003, etc.

Specify “displacement” in the outputs section (7) to output the constraint modes and fixed interface modes that were used as a basis to generate the reduced order system to the Exodus file. First the fixed interface modes are output, followed by the constraint modes. The eigenvalues of the fixed interface modes correspond to the mode shapes. For the
constraint modes an integer index replaces the eigenvalues. These modes may be visualized and evaluated using any of the standard tools.

Data in Table 3-43 will be written to a file. The Output Transfer Matrix (or OTM) depends on data in the history section (see Section 7.3). Specifically, the output nodes and elements, and the output variables are specified in the history file \textit{as if they were to be output to a history file}. For simplicity, and because the OTM describes a linear transfer matrix, only a limited subset of results are provided. In particular, displacements and the natural strains and stresses may be written. The transfer matrix provides the following computation.

\[
\begin{bmatrix}
  u \\
  \varepsilon \\
  \sigma
\end{bmatrix}_{\text{out}} = \text{OTM} \begin{bmatrix}
  q \\
  u_{\text{int}}
\end{bmatrix}
\]

\[
= \begin{bmatrix}
  \Phi_u & \Psi_u \\
  \Phi_\varepsilon & \Psi_\varepsilon \\
  \Phi_\sigma & \Psi_\sigma
\end{bmatrix} \begin{bmatrix}
  q \\
  u_{\text{int}}
\end{bmatrix}
\]

Here \( q \) is the amplitude of the internal constraint modes (typically computed in the next level analysis), and \( u_{\text{int}} \) is a vector of interface displacements. The fixed interface modes (eigen modes of a clamped boundary) are represented by \( \Phi \), and the constraint modes by \( \Psi \).

The left-hand side vectors represents internal results (displacement, strain and stress) which are computed from the interface results. Any of the output results may be omitted, and the OTM will retain only nonzero components. For example, if only displacements are required, the matrix reduces to \( [\Phi_u \ \Psi_u] \). The OTM matrix is a rectangular matrix, and it is typically full. An example \texttt{CBModel} section follows.

```
CBMODEL
  nodeset=1:2 // nodes from nodeset 1 and 2
  format=netcdf // use a netcdf format file
  file='junk.ncf'
END
```

The reduced inertia matrix \( I \) for Craig-Bampton Reduction defined in Section 3.5 may be computed and written to the super-element files with the reduced mass and stiffness matrices. \( I \) can be written to the results file in either netcdf, MATLAB or DMIG format. In the \texttt{CBModel} section

```
CBMODEL
  nodeset 1
  format = netcdf
  file = model.ncf
  GlobalSolution = yes
  inertia_matrix = yes
END
```
Table 3-43. – Data output for Craig-Bampton Reduction.

<table>
<thead>
<tr>
<th>Variable</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>NumC</td>
<td>number of constraint modes</td>
</tr>
<tr>
<td>NumEig</td>
<td>number of fixed interface modes</td>
</tr>
<tr>
<td>Kr</td>
<td>Reduced stiffness matrix.</td>
</tr>
<tr>
<td>Mr</td>
<td>Reduced mass matrix.</td>
</tr>
<tr>
<td>Cr</td>
<td>Reduced damping matrix. Only available for dashpots and block proportional damping.</td>
</tr>
<tr>
<td>cbmap</td>
<td>A two column list providing a map from each interface degrees of freedom to the node and coordinate direction of the global model. The first column of this list is the node number (1:N) in the structure. The second column indicates the coordinate direction as follows.</td>
</tr>
<tr>
<td></td>
<td><strong>Number</strong></td>
</tr>
<tr>
<td></td>
<td>1</td>
</tr>
<tr>
<td></td>
<td>2</td>
</tr>
<tr>
<td></td>
<td>3</td>
</tr>
<tr>
<td></td>
<td>4</td>
</tr>
<tr>
<td></td>
<td>5</td>
</tr>
<tr>
<td></td>
<td>6</td>
</tr>
<tr>
<td></td>
<td>7</td>
</tr>
<tr>
<td>OutMap</td>
<td>A map of the nodes in the output transfer matrix. OutMap(i) is the global node number for each node in the output. There are always 6 rows of output for each node. Thus, OutMap(1) corresponds to rows 1 through 6 in the OTM.</td>
</tr>
<tr>
<td>OTM</td>
<td>Output Transfer Matrix to provide a transfer function from the interface dofs to internal degrees of freedom or other results.</td>
</tr>
<tr>
<td>OutElemMap</td>
<td>A map of the elements in the output transfer matrix, OTME. OutElemMap(i) is the global element number for each element in the output. There are always 6 rows of output for each element.</td>
</tr>
<tr>
<td>OTME</td>
<td>Output Transfer Matrix to provide a transfer function from the interface dofs to internal elements.</td>
</tr>
</tbody>
</table>
Figure 3-21. – An initial analysis using CBR can be applied to reduce a complex component to much smaller matrices. In subsequent analyses the superelement replaces the complex component in the system analysis. There is little loss of accuracy, but significant computational benefit.

the *inertia_matrix = yes-no* line requests the output of the inertia tensor. The default value of *inertia_tensor* is no.

**NOTE:**
The OTM output capability permits an analyst to output the reduced order model of the entire structure for use in another code that supports superelements (such as MSC/NASTRAN). In a subsequent release, we will add the capability to input these matrices as a superelement in *Sierra/SD*. At that point one could perform a Craig-Bampton reduction to generate a reduced order model of that portion of the structure. A follow-up analysis could use this as a superelement. See details in Figure 3-21.
3.5.2. Sensitivity Analysis for Craig-Bampton models

Sensitivity output for Craig-Bampton reduction requires both the sensitivity block described in Section 2.6, and the sensitivity_method, keyword in the CBModel block. The sensitivity_method defaults to the constant vector method unless otherwise specified.

The sensitivity output from a Craig-Bampton reduction is different from that typically seen in eigenvalue or transient solutions. In the case of a Craig-Bampton reduction, the sensitivities that are output consist of partial derivatives of the reduced mass and stiffness matrices with respect to the parameters. We give a brief description here, and refer to the CBR Sensitivity Analysis discussion in the section Solution Procedures of the Theory Manual for further details.

The reduced stiffness matrices in Craig Bampton reduction is computed via the transformation

\[ \kappa = T^T K T \]  

where \( T \) is the Craig-Bampton transformation matrix, \( K \) is the stiffness matrix, and \( \kappa \) is the reduced stiffness matrix. A similar expression can be written for the reduced mass matrix. Sensitivities of \( \kappa \) with respect to a parameter \( p \) can be computed with the constant vector and finite difference approaches.

The sensitivity_method , approach makes the approximation that the transformation matrix \( T = T_o \) does not change with the parameter \( p \). Consequently, the sensitivity can be computed as

\[ \frac{d\kappa}{dp} \approx \frac{T_o^T (K(p + \Delta p) - K(p)) T_o}{\Delta p} \]  

(3.5)

Since \( T_o \) represents a truncated modal space, this approximation is not always accurate.

The finite_difference method uses forward differences. Given updated system stiffness \( K_1 = K(p + \Delta p) \) and transformation matrix \( T_1 = T(p + \Delta p) \), direct forward differences are used to evaluate the sensitivity

\[ \frac{d\kappa}{dp} \approx \frac{T_1^T K(p + \Delta p) T_1 - T_o^T K(p) T_o}{\Delta p} \]  

(3.6)

Unlike the constant vector approach, the finite difference method will converge to the exact sensitivities as \( \Delta p \) goes to zero. This is true provided that there are no repeated modes in the system.

One additional complexity that arises in sensitivity analysis of Craig-Bampton reduced models is when there are repeated modes in the transformation matrix \( T \). Since the ordering of repeated modes in any eigenvalue problem is arbitrary, the perturbed transformation matrix \( T_1 \) could have a different ordering of the repeated modes than \( T_o \). This would corrupt the difference operation in equation 3.6. However, in the case of the constant vector method this is not an issue since there is only one transformation matrix \( T_o \). For these reasons, in the case where the user suspects repeated modes may be present
in the structure, we currently recommend using the constant vector rather than the finite difference approach.

The output from a sensitivity analysis of a Craig-Bampton model is different from other types of sensitivity analysis. The output is written to either a MATLAB or netcdf format, depending on the format parameter (see Table 3-42). The default is netcdf.

The outputted quantities are the derivatives of the stiffness and mass matrices with respect to the various parameters. Thus, if there were two sensitivity parameters \( p_1 \) and \( p_2 \), the output quantities would be

\[
\frac{\partial \kappa}{\partial p_1}, \frac{\partial \kappa}{\partial p_2}
\]

where \( \kappa \) would be the reduced mass and/or stiffness matrix. The dimensions of these sensitivity matrices would be the same as the dimensions of the corresponding reduced mass and stiffness matrices.

The output matrix derivatives given in equation 3.7 are useful for studying how the reduced matrices change with the parameters. These matrix derivatives can also be used in subsequent analysis with the corresponding superelements. For more details, we refer to Section 5.31.

### 3.6. preddam Solution Case

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Type</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ModalFilter</td>
<td>string</td>
<td>none</td>
<td>Modal filter to define modes to retain. See Section 3.10.2.2</td>
</tr>
<tr>
<td>load</td>
<td>string</td>
<td>none</td>
<td>Load section for gravitational loading</td>
</tr>
</tbody>
</table>

Table 3-44. – preddam Solution Case Parameters.

The preddam solution case is intended to be ran as part of a multicase solution, as a preparatory step for a subsequent DDAM analysis (section 3.7). It should be preceded by an eigen solution case (section 3.10), but does not necessarily have to be followed by a DDAM solution case.

preddam utilizes the eigenvectors and system mass matrix produced in the preceding eigen solution to calculate and filter the modal participation factors, modal weights, individual modal weight percentage, cumulative modal weight, and cumulative modal weight percentage. See table 3-44 for a list of valid preddam section parameters.

modalfilter is implemented as a part of Sierra/SD and may be used as a part of other solution methods. It provides a means of filtering data taken from the modal analysis and
the participation factors. More information may be found in section 3.10.2.2 and section 3.15.

Finally, a load block should be defined. This load block (in Sierra/SD DDAM analyses) applies to the value of gravitational loading (-386.4). Its direction must match DDAM analysis direction defined in the modalfilter block and in the subsequent DDAM solution case.

### 3.6.1. Eigen analysis notes

The standard Sierra/SD specification for eigen analysis is nmodes=<number>, where <number> is an integer value for the requested number of modes (see section 3.10). This capability can compute approximately half the modes of the system at most. For verification purposes a limited capability exists to compute all the eigen modes. This capability runs only in serial, and only on small models. It is selected by nmodes=all.

Sierra/SD doesn’t have the capability to select modes by frequency limits. The user must choose the number of modes, nmodes, in an initial eigen solution, and continually increase nmodes and restart or rerun eigen analysis until the desired frequency level is reached. This step is recommended to be completed before the subsequent preddam and DDAM solution cases are included and analyzed.

### 3.7. DDAM Solution Case

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Type</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>analysis_direction</td>
<td>vertical</td>
<td>athwartship</td>
<td>fore_aft</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>athwartship → Y-direction</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>fore_and_aft → X-direction</td>
</tr>
<tr>
<td>ship_type</td>
<td>surface_ship</td>
<td>submarine</td>
<td></td>
</tr>
<tr>
<td>mount_type</td>
<td>hull</td>
<td>deck</td>
<td>shell_plating</td>
</tr>
<tr>
<td>response_type</td>
<td>elasticplastic</td>
<td>elastic</td>
<td></td>
</tr>
<tr>
<td>velocity_coeffs</td>
<td>&lt;real&gt;(4)</td>
<td></td>
<td>See NAVSEA documentation</td>
</tr>
<tr>
<td>acceleration_coeffs</td>
<td>&lt;real&gt;(4/5)</td>
<td></td>
<td>See NAVSEA documentation</td>
</tr>
</tbody>
</table>

Table 3-45. – DDAM Solution Case Parameters.
The U.S. Navy Dynamic Design Analysis Method (DDAM) is an established procedure employed in the design of ship equipment and foundations for shock loading requirements. The details of the formulation, specific procedures for application, acceptance criteria, etc., are documented in NAVSEA Report 250-423-30 and NAVSEA 0908-LP-000-3010. Support for performing DDAM calculations, as implemented in the Sierra/SD Finite Element Code, is documented both in the Sierra/SD User’s manual and in the DDAM Primer. The user is expected to be fully familiar with both NAVSEA publications.

DDAM is focused on five main phases: the problem formulation phase, mathematical modeling phase, coefficient computation phase, dynamic computation phase, and evaluation phase. DDAM as implemented in Sierra/SD will focus primarily on the evaluation phase.

A DDAM analysis in Sierra/SD is divided into three solution cases: case 1 (eigen), case 2 (preddam), and case 3 (DDAM). Case DDAM may only be run following preddam and eigen.

DDAM uses filtered eigenvalues and mode shapes from case 1 eigen and filtered modal participation factors and modal weights from case 2 preddam to calculate shock design coefficients and values, filtered modal outputs (force, displacement, stress, etc.), and the NRL sums of those outputs.

Nodal displacements, velocities, accelerations, and forces; element-wise stresses and derived quantities; and NRL sums of the above are all written to the output Exodus file using the ddamout output keyword (see section 7.1.52). Some of the commonly used post-processing tools for Exodus outputs are Paraview for graphical visualization and explore, blot, and exo2mat for data extraction.

An example input follows.

```
SOLUTION
  case 1
    eigen
      nmodes = 4
  case 2
    preddam
      MODALFILTER vertical
      load 1
  case 3
    DDAM
      analysis_direction VERTICAL
      ship_type SURFACE_SHIP
      mount_type HULL
      response_type
      velocity_coeffs 1.4 5.2 220.1 12.2
      acceleration_coeffs 1.0 2.0 3.0 4.0 5.0
END
MODALFILTER vertical
```
Note: Ship directions (\textbf{analysis\_direction}) must match coordinate directions. See table 3-45 for the meaning of each value. The direction must also match the \textbf{modalfilter} and \textbf{load} sections used in the \textbf{preddam} solution case.

Highlights of the DDAM capabilities in \textbf{Sierra/SD} are as follows:

1. Include modal masses that include at least 1\% individually of the total modal mass.
   The user must manually add any extra needed modes using \texttt{add} in the \textbf{modalfilter} block.

2. Incorporate a 6g minimum load requirement. This is not a user option.

3. Complex models and multiple element types.

4. Superelement integration.

5. Symmetric boundary conditions.

6. Parallel computation.

The user may verify \textbf{preddam} (section 3.6) and DDAM by examining filtered modes, participation factors, modal weights, shock design coefficients, and values found in the following text files:

1. \texttt{preddam} $\rightarrow$ \texttt{PREDDAM\_RESULTS.txt}

2. \texttt{DDAM} $\rightarrow$ \texttt{DDAM\_RESULTS.txt}

DDAM does have a few limitations. The equipment to be analyzed must be represented as a linear elastic system with discrete modes. Damping is neglected. For very low frequency (VLF) systems, DDAM may not be appropriate, and, where closely-spaced modes exist, DDAM may produce excessive responses.
3.8. DirectFRF Solution Case

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Type</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>interpolate points</td>
<td>integer</td>
<td>0</td>
<td>Number of additional points to interpolate for Padé expansion. If zero, no interpolation is performed.</td>
</tr>
<tr>
<td>interpolate order</td>
<td>integer</td>
<td>20</td>
<td>Order of the rational function for Padé expansion.</td>
</tr>
<tr>
<td>flush</td>
<td>integer</td>
<td>50</td>
<td>Defines how often results are written to the exodus results file. See Section 2.4.1</td>
</tr>
</tbody>
</table>

Table 3-46. – DirectFRF Solution Case Parameters.

Option `directfrf` is used to perform a direct frequency response analysis. In other words, we compute a solution to the Fourier transform of the equations of motion, i.e.,

\[
\begin{align*}
\bar{\Pi} & = \frac{\bar{K} + i\omega \bar{C} - \omega^2 \bar{M}}{\bar{A}(\omega)} = \bar{f}(\omega)
\end{align*}
\]

where \( \bar{\Pi} \) is the Fourier transform of the response \( \bar{u} \), and \( \bar{f} \) is the Fourier transform of the applied force. The matrix equation is then solved for each frequency. When a direct solver is used, this means that a complex factorization must be performed once per output. This is time-consuming, and the `ModalFrF` may be a better option for many situations (see Section 3.17).

The force function must be explicitly specified in the load section, and `must` have a function definition. Note that the force input provides the real part of the force at a given frequency, i.e., it is a function of frequency, not of time.

The frequency response function is evaluated at the frequencies specified as described in Section 7.4. In a frequency section set `freq_step`, `freq_min`, and `freq_max`. Also set an application region. Examples are presented in inputs 3.6, 3.7 and 3.11.

In addition to the output that is sent to the `frq` file, output may also be written to the Exodus file, provided that the keywords (such as acceleration) are specified in the outputs section. If nothing is specified in the outputs section, then nothing is written to the Exodus output files.
The expression “frf” is often interpreted as the ratio of output/input. There are reasons for using that ratio, including the confusion that can come from scaling the Fourier transform. The Sierra/SD code computes the output and does not compute a ratio. If the ratio is required, use a function with unit load as the input.

3.8.1. Padé Expansion

Computation of each frequency response is expensive because the system matrices must be computed, factored and solved once at each frequency. A cost-effective approach is to use a much coarser computational grid for full computation, and use a rational function (or Padé) expansion for intermediate points. The two additional parameters required for the expansion are listed in table 3-46. They are described below, and an example is shown in input 3.1. The theory is described in reference.

```plaintext
solution
case out
directfrf
  Interpolate Points = 50
  Interpolate Order = 18
end
```

Input 3.1. Padé Expansion Input Example. In this example, each exactly computed direct frequency response point will be separated by 50 interpolated values. These values will be determined using a Padé expansion of order 18.

3.9. Dump Solution Case

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Type</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
</table>

Table 3-47. – Dump Solution Case Parameters.

The keyword dump will cause Sierra/SD to form matrices. No solve will be done. The main reason to use this solution case is to output the matrices to matlab (section 2.3) for debugging or custom post processing.

4A rational function expansion is similar to a Taylor series expansion, but is capable of approximating resonant behavior.
3.10. Eigen Solution Case

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Type</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>nmodes</td>
<td>Integer</td>
<td>10</td>
<td>Number of modes to extract. See Section 3.10.1</td>
</tr>
<tr>
<td>shift</td>
<td>Real</td>
<td>-1.0e6</td>
<td>Shift to apply to matrix system to allow solving singular systems. See Section 3.10.2</td>
</tr>
<tr>
<td>untilfreq</td>
<td>Real</td>
<td>Inf</td>
<td>Target frequency to reach. See Section 3.10.2.1</td>
</tr>
<tr>
<td>ModalFilter</td>
<td>string</td>
<td>none</td>
<td>Modal filter to define modes to retain. See Section 3.10.2.2</td>
</tr>
<tr>
<td>fluidloading</td>
<td>yes/no</td>
<td>no</td>
<td>Turns on added mass for approximate Wet Modes calculation.</td>
</tr>
</tbody>
</table>

Table 3-48. – Eigen Solution Case Parameters.

The `eigen` solution case computes eigenvalues and mode shapes of a system representing the natural vibration modes. The parameters `NegEigen`, `eig_tol`, and `eigen_norm` are described in sections 2.3 and 2.3.

Eigenvectors, $\Phi$, are stored in Exodus files as time dependent displacements. The time of an eigenvector is the frequency. Seacas library tools expect times to be unique, occasionally causing benign warning messages when there are multiple modes at the same frequency. The mode shapes multiplied by the consistent mass matrix 7.1.10 are also available.

3.10.1. Option nmodes

Nmodes is the number of modes to compute. The eigenvalues are computed beginning with the lowest frequency mode and working up. The calculation continues until `nmodes` have converged. Modal analysis uses iterative Lanczos procedures. These methods build a Krylov subspace from which the solution is determined. The Krylov subspace method is designed to find the lowest frequencies, which is typically what is needed for structural analysis. More linear solves and longer run times are required to compute more modes and reach higher frequencies.

If `nmodes=all`, then all the finite modes of the structure are computed using dense linear algebra. No shift is required. The mass matrix may be either full rank or singular. Constraints may be present. This option is only available in serial. The number of dofs must be < 1000.
3.10.2. Solving Singular Systems with Shifts

There is a trade off between reducing the number of linear solves required to solve an eigenvalue problem or reducing the cost per linear solve. This trade off is controlled using the shift. The number of solves per eigenvalue decreases as the shift decreases in magnitude, and at the same time the cost per linear solve increases. Direct linear solvers require a shift for singular (floating) systems. Iterative methods are more reliable when applied to positive definite linear systems than they are when applied to positive semi-definite linear systems. The eigenvalue problem is defined as,

\[(K - \omega^2 M)\phi = 0.\]  (3.9)

\(K\) and \(M\) are positive semi-definite matrices. If the shift \(\sigma < 0\), then \(K - \sigma M\) is positive definite. Solution cases generally involve positive definite matrices except direct frequency response.

Here \(K\) and \(M\) are the stiffness and mass matrices respectively, and \(\omega\) and \(\phi\) are the eigenvalues and vectors to be determined. The problem may be solved using a variety of methods — the Lanczos algorithm is used in Sierra/SD. In this method, a Krylov subspace is built by repeated solving equations of the form \(Ku = b\). For floating structures, or structures with zero-energy mechanisms, \(K\) is singular and special approaches are required to solve the system. The two approaches used in Sierra/SD are described below.

Deflation. If it is possible to identify the singularity in \(K\), then the null vectors of \(K\) are eigenvectors (with \(\omega = 0\)), and the system can be solved by ensuring that no component of the null vectors ever occurs in \(b\). This approach is equivalent to computing the pseudo inverse of \(K\).

The strength of deflation is that if the eigenvectors can be determined exactly, the Lanczos algorithm is unaltered and the remaining vectors can be determined somewhat optimally. The difficulty is ensuring that we have correctly determined the eigenvectors, especially when mechanisms or multipoint constraints exist in the model.

As an example, the geometric rigid body modes capability 3.33 can be used to analytically compute the standard rigid body modes and deflate them out of a singular system.

Shifting. The second method involves solution of a modified (or shifted) eigenvalue problem.

\[((K - \sigma M) - \mu M)\phi = 0.\]  (3.10)

This system has the properties that the eigenvectors, \(\phi\), are unchanged from the original equation, and the eigenvalues, \(\mu\), are related to the original values. Namely, \(\mu = \omega^2 - \sigma\).
On serial platforms, a small negative shift is normally sufficient to solve the problem due to the high accuracy of serial direct solvers. For parallel solution, a reasonable shift value is usually given by $\sigma = -\omega_{elas}^2$, where $\omega_{elas}$ is the expected first nonzero (or elastic) eigenvalue. Because Sierra/SD cannot compute an optimal shift value a priori, a default of $-1$ is used and a warning is written if the shift used is well outside of the expected range. In practice, a shift of $-1e6$ is often used in Sierra/SD input, as this gives reasonable results for extracting frequencies anywhere near 150Hz, assuming the time unit is seconds. This is often the general frequency range of engineering interest. However, an adequate shift may vary some depending on the units and the properties of an analysis.

The shifted problem benefits from the fact that $K - \sigma M$ can be made non-singular (except in rare situations). This is done by choosing $\sigma$ to be a large negative value. Unfortunately, the Lanczos routine convergence is affected if $\sigma$ is chosen to be too far from the recommended value of $\sigma = -\omega_{elas}^2$.

If $\sigma$ is too large, many solves will be required to determine the eigenvalues, which consequently slows convergence. If $\sigma$ is too small each linear solve may be near singular, requiring many solver iterations or being unable to reach the target residual at all resulting in a 'SOLVER OUT OF BOUNDS' error.

Another consequence of an excessively large or small negative shift is that potentially not all redundant zero eigenvalues may be found. These modes may be found by correcting the shift, tightening tolerances, or by restarting.

The shifted eigenvalue problem is more reliable. Set the grbm_tol to a small value (e.g. 1e-20) (or use the default), and manually enter a negative shift. The output should still be examined to ensure that no global rigid body modes are detected.

If the model is not floating and has no mechanisms and no zero energy modes, the system is not singular and a shift is unnecessary.

**Example**

A representative Solution section for an eigen analysis with a shift of $-10^6$ followings. A dozen modes are requested. This shift would be appropriate for a system where the first elastic mode is approximately 150Hz.

```plaintext
Solution
  eigen
  nmodes 12
  shift -1.0e6
end
```
3.10.2.1. UntilFreq Option and Eigen Restart

The untilfreq keyword provides an additional method of controlling the eigen spectrum to be computed. If this value is provided, then the analysis will be automatically (and internally) restarted until the frequency of the highest mode is at least the value of the untilfreq. This restart capability is somewhat crude. There are always nmodes new modes computed on each calculation. Also, because there can be inaccuracies associated with restarting the eigen solver, Sierra/SD restarts a maximum of 5 times.

Sierra/SD uses the ARPACK Lanczos solver for the eigen problem. This solver maintains the orthogonality of the eigenvectors for a single batch of modes. However, when restarted, the solver must deflate out the previously computed modes. There can thus be a slight loss of orthogonality due to round off. With repeated restarts, the effect can significantly reduce accuracy.

Additionally, eigen restart can be manually controlled via the multicase solution. In the example below, the first 1000 modes are computed in 'eig1'. Next, eig2 restarts and reads and deflates those 1000 modes from the system. In this solution case, an additional 500 modes are computed, for a total of 1500 modes. Restarting eigen analysis in this way can compute modes with less memory than computing all modes at once. An excessive number restarts of this nature (more than about five) will lead to accuracy loss from round off errors during deflation.

```plaintext
Solution
  case 'eig1'
    eigen
    nmodes 1000
  case 'eig2'
    eigen
    restart = read
    nmodes 1500
end
```

3.10.2.2. ModalFilter Option

The optional modalfilter keyword provides a means of reducing the modes retained for output and for subsequent analysis. For more details, see Section 3.15.

Use of this parameter within the eigen solution case is deprecated in favor of the new modalfiltercase solution case (Section 3.15).

You can also put the modal filter into a separate case, called preddam, 3.7.

3.10.2.3. Fluid loading Option

Although a coupled structural/acoustic eigenvalue problem is quadratic, under certain approximations, the fluid may be represented as an added mass on the structure, and a real eigen problem results as described in subsection Wet Modes or Added Mass section.
Solution Procedures of the Theory Manual. The fluidloading enables that added mass calculation. For wet modes all material blocks that use an acoustic material are treated alike as a fluid domain for mass loading.

In an early implementation the lowest modes of the acoustic stiffness matrix were used to approximate the added mass. As a sanity check, one mode of the acoustic stiffness matrix is approximated.

Fluid loading is not applicable to models that use infinite elements. There are two reasons for this. First, infinite elements lead to a nonsymmetric eigenvalue problem, which is not supported. Second, with infinite elements, the fluid region stiffness matrix is singular, implying and infinite added mass. A fatal error is returned if infinite elements are detected.

3.10.2.4. Rigid Body or Zero Energy Modes
Rigid body modes represent rigid body translation or rotation of a structure. A normal free-free structure would be expected to have 6 rigid body modes. A structure with constraints may have fewer than 6 rigid body modes. On the other hand a structure with multiple disconnected pieces may have more than 6 rigid body modes. To compute any rigid body modes, one must request all of them. For structures with 6 rigid body modes, nmodes ≥ 6. Otherwise, the solution case is likely to fail. Prior to computing the wet modes, rigid body modes are generated by adding a Geometric Rigid Body Modes solution case as described in Section 3.33. As described there the parameter num_rigid_modes must be set to 6.
### 3.11. **AEigen Solution Case**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Type</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>nmodes</td>
<td>Integer</td>
<td>10</td>
<td>Number of modes to extract. See Section 3.10.1</td>
</tr>
<tr>
<td>shift</td>
<td>Real</td>
<td>-1.0e6</td>
<td>Shift to apply to matrix system to allow solving singular systems. See Section 3.10.2</td>
</tr>
<tr>
<td>untilfreq</td>
<td>Real</td>
<td>Inf</td>
<td>Target frequency to reach. See Section 3.10.2.1</td>
</tr>
<tr>
<td>ModalFilter</td>
<td>string</td>
<td>none</td>
<td>Modal filter to define modes to retain. See Section 3.10.2.2</td>
</tr>
<tr>
<td>implicit_restarts</td>
<td>integer</td>
<td>5</td>
<td>Number of allowed restart steps, default same as eigen</td>
</tr>
<tr>
<td>anblocksize</td>
<td>integer</td>
<td>1</td>
<td>Number of subspace vectors added each step, default same as eigen</td>
</tr>
<tr>
<td>anrho_shift</td>
<td>Real</td>
<td>0.0</td>
<td>Shift of infinite eigenvalues of constrained problem</td>
</tr>
<tr>
<td>anverbosity</td>
<td>integer</td>
<td>17</td>
<td>Level of verbosity of log files, see below for details</td>
</tr>
<tr>
<td>subspace_size</td>
<td>integer</td>
<td></td>
<td>Maximum number of subspace block Krylov-Schur basis vectors. By default, picks based on number of requested eigen modes.</td>
</tr>
</tbody>
</table>

*Table 3-49. – AEigen Solution Case Parameters.*

The symmetric generalized eigenvalue problem of equation 3.9 is solved for the lowest frequency modes. Eigenvalue problems are solved using a linear solver. The iterative solution of the eigenvalue problem is a nonlinear iteration that requires multiple linear solves.

The standard eigenvalue methods of Section 3.10 use the ARPACK\(^{32}\) package. It is a powerful, public domain solver and has been successful. Other approaches are being developed. The AEigen keyword selects the Krylov-Schur method in the Anasazi\(^{7}\) package.

Modal solution methods all control accuracy using the parameter **eig_tol**.
A Block Krylov-Schur method is used that is similar to the ARPACK solver, discussed in Section 3.10. A subspace is built by repeatedly solving systems of the form \((K - \sigma M)\), where \(\sigma\) is the shift specified by keyword \texttt{shift}; see 3.10.2 for more discussion on shifts.

Unlike ARPACK, Anasazi optionally employs block solvers. That is, the subspace generated by the Krylov-Schur iteration adds multiple vectors at each step, where ARPACK adds a single vector. The number of vectors is specified by the keyword \texttt{anblocksize}.

The amount of diagnostic information or level of verbosity is changed used \texttt{anverbosity}. The default value is 17, implying that the Anasazi solvers will output errors, warnings, and timing details. Each verbosity type is controlled by a single bit in the integer \texttt{anverbosity}. These types are listed in Table 3-50. Each combination is valid, the combinations being formed by adding different verbosity values. For example, setting \texttt{anverbosity} to 25 = 0 + 1 + 8 + 16 requests output for Errors, Warnings, Final Summary and Timing Details.

<table>
<thead>
<tr>
<th>Verbosity type</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Errors</td>
<td>0</td>
</tr>
<tr>
<td>Warnings</td>
<td>1</td>
</tr>
<tr>
<td>Iteration Details</td>
<td>2</td>
</tr>
<tr>
<td>Orthogonalization Details</td>
<td>4</td>
</tr>
<tr>
<td>Final Summary</td>
<td>8</td>
</tr>
<tr>
<td>Timing Details</td>
<td>16</td>
</tr>
<tr>
<td>Status Test Details</td>
<td>32</td>
</tr>
<tr>
<td>Debug</td>
<td>64</td>
</tr>
</tbody>
</table>

Table 3-50. – AEigen Verbosity Table.

Input 3.2 illustrates the “Solution” section of an input file using the Krylov-Schur method. A dozen of the lowest frequency modes are requested. The \texttt{shift} is \(-10^6\). The shift is appropriate for a floating system where the first elastic mode is approximately 150 Hz. This produces an eigen analysis equivalent to the example given in 3.10 for \texttt{eigen}. The verbosity level specifies that after finishing the eigen analysis, the solver will print status information (number of iterations, current eigenvalues) and timing statistics.

```plaintext
Solution
aeigen
  nmodes 12
  shift -1.0e6
  anblocksize 1
  anverbosity 25
end
```

Input 3.2. Aaigen example
### 3.12. *Largest_Ev Solution Case*

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Type</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Table 3-51.** – Largest_Ev Solution Case Parameters.

The *Largest_Ev* analysis determines the largest eigenvalue of the system, i.e.

\[(K - \lambda M)\phi = 0\]

There are no arguments to the solution method. The eigenvalue not physically meaningful. This solution is used primarily for debugging.

### 3.13. *Fatigue Solution Case*

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Type</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>method</td>
<td>narrow-band/wirsching</td>
<td>narrow-band</td>
<td>Fatigue calculation algorithm to use.</td>
</tr>
<tr>
<td>duration</td>
<td>&lt;real&gt;</td>
<td>1</td>
<td>Time duration, (\tau) over which to integrate fatigue damage.</td>
</tr>
</tbody>
</table>

**Table 3-52.** – Fatigue Solution Case Parameters.

High Cycle Fatigue occurs after long periods of alternating stresses in the elastic range.

The input parameters for fatigue-failure are shown in Tables 3-52 and 4-88. An example is shown in input 3.3. Fatigue analysis requires inputs in several sections.

1. The fatigue section in the **Solution** block must be immediately preceded by a modalranvib solution, with noSVD option (3.18). This defines the random stress moments needed for computation of the stress crossing rates.

2. The **Fatigue** section in the Solution block defines general fatigue parameters. Relevant parameters are outlined in Table 3-52.

3. The **material** section must include parameters necessary for computation of fatigue and damage. Sections without this input will not have a fatigue parameter output. Typical materials parameters required for fatigue analysis are found in Table 4-88, found in the materials section, 4.4.2.
Input 3.3. High Cycle Fatigue Input Example. Fatigue is specified by block. In this example, there are two blocks. The material properties for block aisi4140 include fatigue parameters, but the material properties for block steel do not include fatigue parameters. The fatigue calculation is performed for block 100.

3.13.0.1. User Output  If the fatigue method is specified in the Solution section then output for fatigue will be provided. There is no need for a specification in the outputs section. The following fields will be output.
**NarrowBandDamageRate:** There are two possible equations for $D_{NB}$ depending on if the material parameters specified for the S-N curve are based on a stress range or peak stress. The Narrow Band damage per unit time, defined as:

$$D_{NB} = \frac{\nu_o^+}{A} (\sqrt{2} \sigma_s F_{SS})^m \Gamma \left( \frac{m}{2} + 1 \right)$$

(3.11)

for materials defined based on Peak Stress, and

$$D_{NB} = \frac{\nu_o^+}{A} (2 \sqrt{2} \sigma_s F_{SS})^m \Gamma \left( \frac{m}{2} + 1 \right)$$

(3.12)

for materials defined based on the Stress Range. Equation (3.11) is used by default. A stress range type material can be specified in the input (MaterialType = StressRange). See the theory manual for details of the parameters.

**WirschingDamageRate:** The Wirsching damage per unit time. This may be thought of as a scaled value of the NB damage intensity.

$$\hat{D}_W = \lambda \hat{D}_{NB}.$$

**ZeroCrossingRate:** the expected positive zero-crossings intensity.

$$\nu_o^+ = \sqrt{\frac{M_2}{M_o}}$$

**Damage:** The selected damage rate multiplied by time.

$$D = \tau \hat{D}$$

(3.13)

Here $\hat{D}$ can be either the Narrow Band or Wirsching Damage Rate.

**PeakFrequency:** the expected peak occurrence frequency.

$$\nu_p = \sqrt{\frac{M_4}{M_2}}$$

The even moments, $M_x$ (with $x=0,2,4$), are output as part of the random vibration computation, see (3.18). For example, $M2 = (\text{VRMS}^2/2\pi)^2$. 

149
### 3.14. Buckling Solution Case

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Type</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>nmodes</td>
<td>Integer</td>
<td>10</td>
<td>Number of modes to extract. See Section 3.10.1</td>
</tr>
<tr>
<td>shift</td>
<td>Real</td>
<td>-1.0e6</td>
<td>Shift to apply to matrix system to allow solving singular systems. See Section 3.10.2</td>
</tr>
<tr>
<td>untilfreq</td>
<td>Real</td>
<td>Inf</td>
<td>Target frequency to reach. See Section 3.10.2.1</td>
</tr>
<tr>
<td>ModalFilter</td>
<td>string</td>
<td>none</td>
<td>Modal filter to define modes to retain. See Section 3.10.2.2</td>
</tr>
<tr>
<td>bucklingSolver</td>
<td>ARPACK</td>
<td>Anasazi</td>
<td>ARPACK</td>
</tr>
</tbody>
</table>

Table 3-53. – Buckling Solution Case Parameters.

The buckling keyword is used to obtain the buckling modes and eigenvalues of a system. The parameters which can be specified for a buckling solution are tabulated below. Users are encouraged to review the discussion of buckling analyses in.[42]

The shift parameter indicates the shift desired in a buckling analysis. The shift value represents a shift in the eigenvalue space (i.e. $\omega^2$ space). It is set using the shift parameter. Determining an effective shift is problem dependent.

The nmodes parameter specifies the number of requested buckling modes. Its default value is 10.

Unlike eigen analysis, buckling solution cases require a loads block. This is because buckling is always specified with respect to a particular loading configuration. For example, for a pressure load applied on a sideset, the buckling analysis would indicate the critical amplitude of the applied pressure needed to cause buckling. The critical buckling load is computed as the product of the first (lowest) eigenvalue times the amplitude of the applied load. Thus, for the case

```
Loads
  sideset 1
  pressure = 10.0
end
```

The lowest obtained eigenvalue is 100.0. The critical buckling pressure would be $P_{cr} = 100.0 \times 10.0 = 1000.0$. This would indicate that buckling would occur if the loading were applied as,
Similar conclusions can be drawn about force loads on nodesets.

Buckling of floating structures is not supported at this time. If global rigid body modes are present, the solution may not be correct. The use of beams and shells in the buckling solution case is not supported. For models with multipoint constraints, the default serial linear solver is not available.

Example A Solution section for buckling analysis with a shift of $-10^6$ looks like the following, if 1 mode is needed (i.e. if the use is confident that the modes are well separated).

```
Solution
   buckling
   nmodes 1
   shift -1.0e6
end
```

### 3.15. ModalFilter Solution Case

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Type</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ModalFilter</td>
<td>string</td>
<td>none</td>
<td>Modal filter to define modes to retain. See Section 3.10.2.2</td>
</tr>
<tr>
<td>write_files</td>
<td>all</td>
<td>none</td>
<td>output</td>
</tr>
</tbody>
</table>

Table 3-54. – ModalFilter Solution Case Parameters.

A ModalFilter filters modes computed by an eigen solution case (Section 3.10). The filtered modes are then used by subsequent solution cases.

Since the eigen solution case already outputs the full set of modes to disk, the filtered modes are not written to disk by default. The write_files parameter selects output of the filtered modes.

Controlling the modes retained for subsequent analyses can significantly reduce run time with little effect on the desired response. For example, a shell structure may have many...
hundreds of modes contributing to the normal mode response, and only a few that interact with the loads. 1

During eigen analysis or CBR reduction, the usual number of modes (\textit{n modes}) are computed. These modes are filtered, and only a subset are written to the \textit{Exodus} file or used in subsequent analysis. An example input is shown in input 3.4.

\begin{verbatim}
SOLUTION
  case eig
  eigen
    nmodes=500
  case filter
    modalfiltercase
    ModalFilter=MPF1
  END

ModalFilter MPF1
  remove 1:999
  cumulative mef 0.8 0.8 0.8 0.2 0.2 0
  add 99:101,103
  END
\end{verbatim}

Input 3.4. Example ModalFilter Input

For this example, the following actions are performed in the filter.

1. The first 999 modes are removed. In this case 500 modes are computed, and all modes are removed.

2. The modes contributing the most to a cumulative modal effective mass are added. Modes sufficient for 80\% contributions to the \textit{x}, \textit{y}, and \textit{z} directions are added. Modes needed to achieve 20\% of the rotational terms for \textit{x} and \textit{y} are added. Since the contribution for rotation about \textit{z} is zero, no modes are added there.

3. Modes 99, 100, 101 and 103 are added if they are not already included.

Each entry in the modal filter section consists of two parts: an action (like \textit{remove} or \textit{add}) and an application space. The 	extit{application space} for the “add” and “remove” space is an integer list with a format much like MATLAB. See section 2.1 for more details. Valid action keywords are listed in Table 3-55.

\textbf{remove} Removes modes in the application space from output.

\textbf{add} Adds modes in the action space.

1The modes of large ship are an example. Only a few of the modes contribute to global bending or torsional modes. The remaining modes are local, and may not be of interest to the analysis.
<table>
<thead>
<tr>
<th>Keyword</th>
<th>Application Space</th>
</tr>
</thead>
<tbody>
<tr>
<td>remove</td>
<td>integer list or “all”</td>
</tr>
<tr>
<td>add</td>
<td>integer list or “all”</td>
</tr>
<tr>
<td>cumulative mef</td>
<td>6 fractions</td>
</tr>
<tr>
<td>cumulative nmef</td>
<td>6 fractions</td>
</tr>
</tbody>
</table>

**Table 3-55.** – Modal Filter Keywords.

**cumulative mef** Adds modes which contribute to the modal effective mass. Following this keyword sequence, 6 fractions are entered, one for each of the 6 rigid body mode contributions. The modes are sorted and modes are kept that contribute most to the modal effective mass. When the fractional contribution exceeds the threshold, no more modes are added for that direction. Contributions from each direction are combined (union) and added to the list of modes kept.

The 6 fractions following the keyword indicate the threshold for each coordinate direction. Each fraction must be between 0 and 1, inclusive. A value of zero means no modes are retained. A value of unity retains all modes.

**cumulative nmef** Adds modes which contribute to the normalized modal effective mass. This option is identical to the “cumulative mef” option except that the terms are normalized such that the total contribution from all computed modes sums to one.

The Modal Effective Mass equals the Modal Participation Factor. The Modal Participation Factor is defined in the next section.

The Normalized Modal Effective Mass is not defined.

### 3.16. Modal Participation Factor Solution Case

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Type</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>write_table</td>
<td>Yes</td>
<td>No</td>
<td>Yes</td>
</tr>
<tr>
<td>blockwise</td>
<td>Yes</td>
<td>No</td>
<td>Yes</td>
</tr>
<tr>
<td>RCID</td>
<td>string</td>
<td>123</td>
<td>A string representing rigid body modes to include in the calculation. 123 represents the translational degrees of freedom. 123456 includes all six degrees of freedom.</td>
</tr>
</tbody>
</table>

**Table 3-56.** – Modal Participation Factor Solution Case Parameters.
A Modal Participation Factor (MPF) is a quality of a mode shape. Another name for the Modal Participation Factor is the Modal Effective Mass. A MPF is a direction cosine of an eigenvector along one of the 6 rigid body modes. It measures the interaction of the modes with a gravity load or a base excitation.

The rigid body modes \( \{ R_i \}_{i=1}^6 \) ignore any boundary conditions.

The modal participation factor of an eigenvector \( v \) of the constrained system is determined from the representation of \( v \) in the unconstrained space, using a lumped mass matrix 2.4.4,

\[
\Gamma_{ij} = \frac{R_i^T M v_j}{\sqrt{(R_i^T M R_i) (v_j^T M v_j)}}.
\]

(3.14)

\( \Gamma_{ij} \) is a mass normalized measure of the contribution of a given rigid body term, \( \gamma_i \), to the vector, \( v_j \). A summary term which represents the total fraction of a vector that is spanned by all rigid body modes is also useful.

\[
\text{MPF}_j = \sum_i \Gamma_{ij}^2
\]

(3.15)

The MPF method computes these participation factors for the eigenvectors of a system. This method must be used as part of a multicase solution, and the previous case must be an eigenvalue problem (see Section 3.10). Further, this method (by default) computes the modal participation factor on a block by block basis. Thus, those portions of the model that most contribute to the rigid body motion may be determined. \(^2\) Then,

\[
\Gamma_{ij}^k = \frac{R_i^T M^k v_j}{\sqrt{(R_i^T M R_i) (v_j^T M v_j)}}
\]

(3.16)

Options for the MPF method are listed in Table 3-56.

Summary data from the calculation is written to the results file as described in Table 3-57. In addition, unless write_table=no, data will be written to an external text file. The format for the file is specified in the results file. It contains the block wise modal participation factors, \( \Gamma_{ij}^k \) of equation 3.16. An example is provided in input 3.5.

The external text file is intended to be easily read by external programs such as the MATLAB “load” command. It therefore has no header information. The data ordering is exactly the same as the table written to the echo file (which contains that header information). Each column is grouped first by block (in the order of the blocks in the genesis file), and then by degree of freedom. Usually there are either 3 or 6 dofs per block entry. Each row corresponds to a single mode.

The optional external text file with file name extension mpf contains block-wise modal participation factors. The data is presented in tabular format, separated by white space.

\(^2\)The overall modal contribution is not the sum of the block wise contributions, and contributions from individual blocks may cancel other blocks. See Table 3-57.
Data | Value | Description
--- | --- | ---
MPF | $\sum_i (\Gamma_{ij})^2$ | Overall mode $j$ MPF
MPF-B$_k$ | $\sqrt{\sum_i (\Gamma_{kij})^2}$ | MPF for block $k$, mode $j$
MPF by RBM$_i$ | $\sum_j (\Gamma_{ij})^2$ | MPF for direction $i$

Table 3-57. – MPF Summary data. Each mode, $v_j$, has contributions from each of these summary values.

Most data analysis software tools can easily import this type of data for analysis and plotting (e.g., Microsoft Excel, OpenOffice Calc, Python, MATLAB, Octave, etc.). The MPF file contains no header information, so it is important to understand what each column and row represents. Each row of data corresponds to a mode. Columns represent modal participation factors calculated for each block and requested coordinate (controlled with “rcid”). The columns are grouped first by block, and then by degree of freedom. Blocks are written out in the order they are found in the genesis file (note: they are not sorted by Block ID or by the order they appear in the input file). Hence, if the exodus file contains two blocks and rcid=123 (default), the *.mpf file will contain six columns in the following order: Block1_x, Block1_y, Block1_z, Block2_x, Block2_y, Block2_z. If rcid=123456, then six columns per block (x, y, z, Rx, Ry, Rz) will be written out and there will be 12 columns.

Solution
case eig
eigen nmodes=10
  shift=-1e5
case out
  mfp
  blockwise=yes
  RCID=123
  write_table=yes
end

Input 3.5. Modal Participation Factor (MPF) Example

Modal Effective Mass There are several definitions of the modal participation factor. The value from equation 3.14 is unit normalized such that the sum of the squares of $\Gamma_{ij}$ over all modes equals unity. A related term is the modal effective mass.

$$\text{MEFF}_{ij} = \sqrt{M_{oi} \cdot \Gamma_{ij}}$$ (3.17)

Here $M_{oi}$ is total mass associated with each rigid body mode. The sum of the squares of MEFF$_{ij}$ over all modes, $j$, for a given rigid body mode, $i$ is the total mass associated with
that RBM. Modal truncation decreases the sum. The modal effective mass table is output to the result file immediately after the modal participation factors. At the bottom of the table, the sum of the squares of the modal effective mass is given for each rigid body mode. The total mass and moments of inertia of the system are also given.

**Lumped or Consistent Mass**  We always use the lumped mass for computation of the geometric rigid body vectors used in the modal participation factor calculation. These vectors are mass orthogonalized, and use of the consistent mass matrices for these efforts, especially when there are MPCs can be complicated in parallel. There is a small error introduced when the modes are computed using a consistent mass, and the rigid body vectors use a lumped mass. Refining the mesh reduces the problem, but most accurate results are obtained when the lumped mass is used (see Section 2.4.4).
3.17. **ModalFrf Solution Case**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Type</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>nmodes</td>
<td>Integer</td>
<td>10</td>
<td>Number of modes to extract. See Section 3.10.1</td>
</tr>
<tr>
<td>shift</td>
<td>Real</td>
<td>-1.0e6</td>
<td>Shift to apply to matrix system to allow solving singular systems. See Section 3.10.2</td>
</tr>
<tr>
<td>untilfreq</td>
<td>Real</td>
<td>Inf</td>
<td>Target frequency to reach. See Section 3.10.2.1</td>
</tr>
<tr>
<td>ModalFilter</td>
<td>string</td>
<td>none</td>
<td>Modal filter to define modes to retain. See Section 3.10.2.2</td>
</tr>
<tr>
<td>complex</td>
<td>True/False</td>
<td></td>
<td>If true require use of a complex eigen solution. If false require use of a real valued eigen solution. If unset use the existing eigen solution.</td>
</tr>
<tr>
<td>lfcutoff</td>
<td>Real</td>
<td>-Inf</td>
<td>Exclude any modes below this frequency from the modal computation. Often used to exclude rigid body modes.</td>
</tr>
<tr>
<td>usemodalaccel</td>
<td></td>
<td></td>
<td>If set, use modal acceleration method</td>
</tr>
<tr>
<td>nrbms</td>
<td>Integer</td>
<td>0</td>
<td>Number of rigid body modes, needed for usemodalaccel</td>
</tr>
<tr>
<td>write_files</td>
<td>all/ none/ output/ frequency</td>
<td>all</td>
<td>Controls which result files are written during this solution.</td>
</tr>
</tbody>
</table>

Table 3-58. – ModalFrf Solution Case Parameters.

Option **ModalFrf** is used to perform a modal superposition-based frequency response analysis. In other words, **ModalFrf** provides an approximate solution to the Fourier transform of the equations of motion. If $\tilde{u}$ is the Fourier transform of the displacement, $u$, and $\tilde{f}$ is the Fourier transform of the applied force, then

$$ \left( K + i\omega C - \omega^2 M \right) \tilde{u} = \tilde{f}(\omega). $$

If the damping matrix is zero, or if it can be diagonalized by the undamped modes, then **ModalFrf** uses the undamped modes for the superposition. Otherwise, for general damping matrices $C$, complex modes are used for the superposition. In either case,
ModalFrF is performed in a multicae approach, where the modes (real or complex) are computed in a first case, and then ModalFrF is computed in a subsequent case.

Modal damping can be applied with either real eigenvalues computed by `eigen` or complex eigenvalues shapes computed by `qevp`. However, proportional damping is currently available only with real modes. For more details on damping, see Section 4.8.

### 3.17.0.1. ModalFrF with Real-Valued Modes

In the case where the undamped real modes are used for the superposition, two options are available for the ModalFrF solution: the modal displacement method, and the modal acceleration method. In the case when complex modes are used, the modal displacement method is available. In both the modal displacement and modal acceleration methods, the approximate solution is found by linear modal superposition. Once the modes have been computed, there is little cost in computation of the frequency response. The solution does suffer from modal truncation, but in the case of the modal acceleration method, a static correction term partially accounts for the truncated high frequency terms. Thus, that method is generally more accurate than the modal displacement method. The most accurate approach, though also the most computationally expensive, is the `directfrf` method described in Section 3.8.

For real modes using the modal displacement method, the relation used for modal frequency response is given below.

\[ u_k(\omega) = \sum_j \frac{\phi_{jk}\phi_{jm}\bar{T}_m(\omega)}{\omega_j^2 - \omega^2 + 2i\gamma_j\omega_j\omega} \]

Here \( u_k \) is the Fourier component of displacement at degree of freedom \( k \), \( \phi_{jk} \) is the eigenvector of mode \( i \) at dof \( k \), and \( \omega_j \) and \( \gamma_j \) represent the mode frequency and associated fractional modal damping respectively. In the case of complex modes, the equations need to be linearized as described in subsection Quadratic Modal Superposition section Solution Procedures of the Theory Manual.

For the modal acceleration method, the procedure for computing the modal frequency response is more complicated. The response is split into the rigid body contributions, and the flexible contributions. The number of global rigid body modes must be specified in the input file. Also, see subsection Modal Frequency Response Methods section Solution Procedures of the Theory Manual.
The modal acceleration method more accurately computes the poles (or peaks) of the response and much more accurately compares the zeros of a function. The cost is an additional factor and solve. It can be used on floating structures, and the additional factor involves the stiffness terms (which are singular) and has no mass terms to stabilize the solution. Thus, it may be much more difficult to perform that solve than the other solves involved in the eigen analysis. In eigen analysis we recommend a negative shift for floating structures to remove the singularity associated with rigid body modes. No such approach is possible if you are using the modal acceleration method. Thus, significant “tweaking” of the linear solver parameters may be required to accurately determine the global rigid body modes required for success of this method.

The force function must be explicitly specified in the load section, and MUST have a “function” definition. Note that the force input provides the real part of the force at a given frequency, i.e. it is a function of frequency, not of time.

The following table gives the parameters needed for ModalFrf section.

The nnmodes parameter controls the eigen analysis (see Section 3.10). The optional keyword, usemodalaccel, is used to determine whether to use the modal displacement or the modal acceleration method. If this keyword is specified, modal acceleration is used, otherwise the modal displacement method is invoked. If usemodalaccel is used, then the number of global rigid body modes must be specified using nrbms.

The parameters freq_step, freq_min, and freq_max are used to define the frequencies for computing the shock response spectra. They are identified in the frequency section with the application region (see Section 7.4). The range of the computed frequency spectra is controlled by freq_min and freq_max, while freq_step controls the resolution. The accuracy of the computed spectra does not depend on the magnitude of freq_step. This parameter controls the quantity of output. Examples are shown in inputs 3.6, 3.7 and 3.11.

We note that, in addition to the output that is sent to the frq file, output is also written to the Exodus file during a ModalFrf, provided that the keywords are specified in the outputs section. If nothing is specified in the outputs section, then nothing is written to the Exodus output files.

In the case of undamped modes, the following is a multicase example of how the ModalFrf could be specified.
### Input 3.6. ModalFrf Example Input

#### 3.17.0.2. ModalFrf with Complex Modes

In the case when complex modes are used, the modal displacement method is available. In this case the `qevp` solution case is used to compute the modes. There are currently three methods that can be used with the `qevp` solution case, and they are the `sa_eigen` method, the *Anasazi* method, and the `ceigen` method. For more details, we refer to Section 3.21.0.1. We note that in the case of complex modes, modal superposition is currently implemented for the `sa_eigen` method and the *Anasazi* method. The `ceigen` method does not support a subsequent modal superposition.

Also, when computing the complex modes in preparation for a modal superposition, we recommend using the `reorthogonalization` flag. When turned on, this flag searches for repeated modes and reorthogonalizes the eigenvectors of those modes. In many cases, repeated modes coming out of the eigen solvers are linearly independent, but not orthogonal. For more details, we refer to Section 3.21.0.1.

In the case of complex modes, the following is an example.

```plaintext
Solution
  case qevp
    qevp
      method = sa_eigen
      reorthogonalize = Y
      nmodes=20
      nmodes_acoustic = 5
      nmodes_structural = 5
  case out
    ModalFrf
end
```
Input 3.7. Complex ModalFrF Example Input

```plaintext
complex = y
end
Frequency
  freq_step=300
  freq_min=100
  freq_max=2500
  nodeset=12
  acceleration
end
```
### ModalRanVib Solution Case

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Type</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>nmodes</td>
<td>Integer</td>
<td>10</td>
<td>Number of modes to extract. See Section 3.10.1</td>
</tr>
<tr>
<td>shift</td>
<td>Real</td>
<td>-1.0e6</td>
<td>Shift to apply to matrix system to allow solving singular systems. See Section 3.10.2</td>
</tr>
<tr>
<td>untilfreq</td>
<td>Real</td>
<td>Inf</td>
<td>Target frequency to reach. See Section 3.10.2.1</td>
</tr>
<tr>
<td>ModalFilter</td>
<td>string</td>
<td>none</td>
<td>Modal filter to define modes to retain. See Section 3.10.2.2</td>
</tr>
<tr>
<td>noSVD</td>
<td></td>
<td></td>
<td>Selection of method for von Mises stress computation. Less expensive, and provides additional stress moments.</td>
</tr>
<tr>
<td>lfcutoff</td>
<td>Real</td>
<td>0.1</td>
<td>Exclude any modes below this frequency from the modal computation. Often used to exclude rigid body modes.</td>
</tr>
<tr>
<td>truncation-Method</td>
<td>none/</td>
<td>none</td>
<td>Truncates modes with low activity</td>
</tr>
<tr>
<td></td>
<td>displacement/acceleration</td>
<td></td>
<td></td>
</tr>
<tr>
<td>keepModes</td>
<td>integer</td>
<td>Inf</td>
<td>Keep the specified number of modes, kept modes are selected based on the highest modal activity.</td>
</tr>
<tr>
<td>checkSMatrix</td>
<td>true/false</td>
<td>true</td>
<td>Evaluate the correlation matrix $S$. This matrix is generated at each frequency, and must be positive semi-definite. A correlation matrix that is not positive semi-definite results in negative PSD results, which are not physically meaningful. Note, matrix evaluations are enabled if PSD output is requested.</td>
</tr>
</tbody>
</table>

Table 3-59. – ModalRanVib Solution Case Parameters.
Option `modalranvib` is used to perform a modal-superposition-based random vibration analysis in the frequency domain. Root-mean-square (RMS) outputs, including von Mises stress, are computed for a given input random force function. The resulting power spectral density functions may also be output at locations specified in the `frequency` section. The forcing functions (one for each input) must be explicitly specified in the `ranloads` section (6.3.20). It must reference a `matrix-function` definition (see Section 2.8.17).

`modalranvib` accepts `eigen` solution parameters such as `nmodes`. These are used to compute an eigen solution if one does not yet exist. The `eigen` parameters should not be used in a multicase solution.

The `optional` keyword `noSVD` determines the method used to compute the RMS von Mises stress output. If `noSVD` is specified, then the simpler method which does not use a singular value decomposition is used. Additionally, that simpler method causes the second and fourth moments associated with von Mises stress to be computed and to be written to Exodus output. (The RMS von Mises stress and these two moments, along with the appropriate material properties, can be used in a manner suggested in [64] and discussed in [65] to estimate fatigue life in broad-band random excitation. Also, see the `fatigue` section (3.13). However, this method provides no information about the statistics of the stress. Only the RMS value and moments are reported.

The `optional` keyword `lf cutoff` provides a low frequency cutoff for random vibration processing. Usually, rigid body modes are not included in this type of calculation if RMS stress is computed. The `lf cutoff` provides a frequency below which the modes are ignored. The default for this value is 0.1 Hz. Thus, by default rigid body modes are not included in random vibration analysis. A large negative value will include all the modes.

The `optional` keyword `TruncationMethod` provides control over selection of the retained modes. By default, modes are retained if they have any contribution to the stress. As stresses are proportional to displacement, the default method is `displacement`. It is possible to not truncate at all (none), or to truncate based on accelerations (acceleration). Acceleration contributions are weighted to higher frequencies. Often zero energy modes contribute to a bad truncation, and a preferred means of controlling the truncation is to use the `lf cutoff` parameter and to ensure the integration does not go to zero frequency.

The `optional` keyword `keepmodes` is a method of truncating modes. By default, its value is `nmodes`. If a value is provided, the modes with the lowest modal activity will be truncated until `keepmodes` remain. Note that this procedure is much different from truncating the higher-frequency modes. Modal truncation is important because all the operations compute responses that require $O(N^2)$ operations. Even if `keepmodes` is not entered, modes with modal activity less than 1 millionth of the highest active mode will be truncated.

The parameters `freq_step`, `freq_min`, and `freq_max` are used to define the frequencies for computing the random vibration spectra. They are identified in the `frequency` section along with an optional application region (see Section 7.4). The range of the computed
frequency spectra is controlled by \texttt{freq\_min} and \texttt{freq\_max}, while \texttt{freq\_step} controls the resolution. The accuracy of the computed spectra \textit{does} depend on the magnitude of \texttt{freq\_step} since it is used in the frequency domain integration. Examples are presented in inputs 3.6, 3.7 and 3.11.

In random vibration, the \texttt{frequency} block serves two purposes. First, it is used for the integration information for the entire model. Thus, $\Gamma_{qq}$ for the referenced papers\textsuperscript{48,39} is integrated over frequency and used for all output. In addition, if an output region is specified in the \texttt{frequency} block, output acceleration and displacement power spectra may be computed for the given region at the required frequency points. At this time, \texttt{acceleration} and/or \texttt{displacement} may be specified in the frequency block for random vibration analysis. This output is described in more detail below.

Random vibration analysis is trickier than most input. A number of blocks must be specified.

1. The \texttt{Solution} block must have the required input for \texttt{eigen} analysis, and the keyword \texttt{modalranvib}.

2. The \texttt{RanLoads} block contains a definition of the spectral loading input matrix and other input. Note that the input, $S_{FF}$ is separated into frequency and spatial components. The spatial component is specified here using \texttt{load} keywords. See Section 6.3.20. The spectral component is referred to here, but details are provided in the matrix-function section.

3. The \texttt{matrix-function} section contains the spectral information on the loading. It references functions for the details of the load. The real and imaginary function identifiers for this input are specified here (2.8.17).

4. There must be a \texttt{function} definition for each referenced spectral function. Functions of time or frequency are further described in Section 2.8.

5. There must be a \texttt{frequency} block that is used for integration and optionally also for output of displacement and acceleration output. See Section 7.4.

6. As an undamped system is singular, some type of \texttt{damping} is required. Modal damping terms are required.\footnote{Proportional damping, such as is applied with the \texttt{alpha} and \texttt{beta} terms, will \textit{not} work in \texttt{modalranvib}.}

7. \texttt{boundary} conditions are supplied in the usual way, but the standard \texttt{loads} block is replaced by the input in the ranloads section. The loads block will be quietly ignored in random vibration analysis.

8. The \texttt{outputs} and \texttt{echo} sections will require the keyword \texttt{vrms} for output of RMS von Mises stress. If the \texttt{stress} keyword is also found, then the natural stresses for solid elements will be output.\footnote{These stresses are linear functions of the displacement.} The keywords \texttt{rotational\_displacement} and \texttt{rotational\_acceleration} in the \texttt{outputs} and \texttt{frequency} sections will output the
corresponding RMS and PSD quantities respectively. Quantities output are listed in Table 3-60.

All other input should remain unchanged.

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Output Variable</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Vrms</td>
<td>vrms</td>
<td>Root Mean Squared von Mises Stress</td>
</tr>
<tr>
<td></td>
<td>D1...D5</td>
<td>von Mises Stress SVD moments. Details in.</td>
</tr>
<tr>
<td></td>
<td>M0, M2, M4</td>
<td>von Mises Stress moments. noSVD</td>
</tr>
<tr>
<td></td>
<td>Xrms</td>
<td>X component of RMS displacement</td>
</tr>
<tr>
<td></td>
<td>Yrms</td>
<td>Y component of RMS displacement</td>
</tr>
<tr>
<td></td>
<td>Zrms</td>
<td>Z component of RMS displacement</td>
</tr>
<tr>
<td></td>
<td>Axrms</td>
<td>X component of RMS acceleration</td>
</tr>
<tr>
<td></td>
<td>Ayrms</td>
<td>Y component of RMS acceleration</td>
</tr>
<tr>
<td></td>
<td>Azrms</td>
<td>Z component of RMS acceleration</td>
</tr>
<tr>
<td>rotational_displacement</td>
<td>RotXrms</td>
<td>Rotational X component of RMS displacement</td>
</tr>
<tr>
<td>rotational_displacement</td>
<td>RotYrms</td>
<td>Rotational Y component of RMS displacement</td>
</tr>
<tr>
<td></td>
<td>RotZrms</td>
<td>Rotational Z component of RMS displacement</td>
</tr>
<tr>
<td>rotational_acceleration</td>
<td>RotAxrms</td>
<td>Rotational X component of RMS acceleration</td>
</tr>
<tr>
<td></td>
<td>RotAyrms</td>
<td>Rotational Y component of RMS acceleration</td>
</tr>
<tr>
<td></td>
<td>RotAzrms</td>
<td>Rotational Z component of RMS acceleration</td>
</tr>
</tbody>
</table>

Table 3-60. – ModalRanVib Output to Exodus File. The stress spectral moments are neither computed nor output if noSVD is selected. The stress moments are available if noSVD is selected, and may be used for fatigue. The RMS values of displacements and acceleration are components of a Hermitian tensor. See Section 3.18.0.1 for details.

3.18.0.1. Power spectral density When requested in the frequency block, one output from the random vibration analysis is a power spectral density or PSD (for displacement or acceleration). The power spectral density is a measure of the output content over a frequency band, and usually measured in units of cm$^2$/Hz or some similar unit. Acceleration PSDs are often measured in units of g$^2$/Hz. 5

Like the input cross spectral forces, the output quantities are Hermitian, with 9 independent translational quantities per frequency, at each output node for each type of output. The method for transforming these quantities in alternate coordinate systems are in subsection Modal Frequency Response Methods section Solution Procedures of the Theory Manual. The rotational terms can be requested using the keywords rotational_displacement or rotational_acceleration. Note that the cross correlation

---

5Power spectral density output is requested in the frequency block. A collection of nodes is indicated and the displacement or acceleration keyword is entered. PSDs of displacement or acceleration are available.
terms for rotation are not output.

\[
\begin{bmatrix}
A_{xx} & A_{xy} + iA_{xyi} & A_{xz} + iA_{xzi} & - & - & - \\
A_{xy} - iA_{xyi} & A_{yy} & A_{yz} + iA_{yzi} & - & - & - \\
A_{xz} - iA_{xzi} & A_{yz} - iA_{yzi} & A_{zz} & - & - & - \\
- & - & - & A_{\text{Rot}_x,\text{Rot}_x} & - & - \\
- & - & - & - & A_{\text{Rot}_y,\text{Rot}_y} & - \\
- & - & - & - & - & A_{\text{Rot}_z,\text{Rot}_z}
\end{bmatrix}
\]

Because the inputs are specified in terms of force cross-correlation functions, the standard procedure for applying loads often involves application of a large concentrated mass at the input location. The force may then be applied to the mass and the acceleration determined from \( a = f/m \), where we assume that \( m \) is much larger (100 to 1000 times larger) than the mass of the remainder of the structure. Some confusion can arise in the scaling of the force.

The output PSD for acceleration is defined as follows.

\[
G_{ij} = H_{ki}^* S_{kl} H_{lj}
\]

where \( H_{lj} \) is the transfer function giving \( a_j/f_l \), and \( S_{kl} \) is a power spectral density input (with units of force\(^2\)/Hz).

Consider a single input, i.e. \( k = l \), and with \( f_k = m_k a_k \).

\[
G_{ij} = H_{ki}^* \langle m_k a_k, a_k m_k \rangle H_{lj} = (m_k^2) H_{kl} \langle a_k, a_k \rangle H_{kj}
\]

(3.18)

(3.19)

Thus, the acceleration PSD must be multiplied by the square of the mass to get the force PSD. Sierra/SD applies the scale factor to the spatial force distribution (which is also squared), so the scale factor in Sierra/SD should be \( m_k \).
3.19. **ModalShock Solution Case**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Type</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>nmodes</td>
<td>Integer</td>
<td>10</td>
<td>Number of modes to extract. See Section 3.10.1</td>
</tr>
<tr>
<td>shift</td>
<td>Real</td>
<td>-1.0e6</td>
<td>Shift to apply to matrix system to allow solving singular systems. See Section 3.10.2</td>
</tr>
<tr>
<td>untilfreq</td>
<td>Real</td>
<td>Inf</td>
<td>Target frequency to reach. See Section 3.10.2.1</td>
</tr>
<tr>
<td>ModalFilter</td>
<td>string</td>
<td>none</td>
<td>Modal filter to define modes to retain. See Section 3.10.2.2</td>
</tr>
<tr>
<td>srs_damp</td>
<td>Real</td>
<td>0.03</td>
<td>Damping coefficient used for the shock response spectra calculation</td>
</tr>
</tbody>
</table>

Table 3-61. – ModalShock Solution Case Parameters.

The **modalshock** solution method is used to perform a modal-superposition-based implicit transient analysis followed by computation of the shock response spectra for the degrees of freedom in a specified node set. More information about the about shock response spectra solution cases is given in Section 3.29 describing implicit-transient-based SRS.

A **frequency** block must also be included in the input file for **modalshock** solution cases to define the frequencies and nodesets for computing the shock response spectra (see Section 7.4). The parameters `freq_step`, `freq_min`, and `freq_max` are used to define the frequencies for computing the shock response spectra. They are identified in the **frequency** section along with an application region (see Section 7.4). The range of the computed frequency spectra is controlled by `freq_min` and `freq_max`, while `freq_step` controls the resolution. The accuracy of the computed spectra is not dependent on the magnitude of `freq_step`. This parameter controls the quantity of output. Examples are presented in inputs 3.6, 3.7 and 3.11.
### 3.20. ModalTransient Solution Case

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Type</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>time_step</td>
<td>Real</td>
<td></td>
<td>Time step size. See Section 3.30.0.1</td>
</tr>
<tr>
<td>nsteps</td>
<td>Integer</td>
<td></td>
<td>Number of time steps to take. See Section 3.30.0.2</td>
</tr>
<tr>
<td>start_time</td>
<td>Real</td>
<td>0.0</td>
<td>Solution case start time. See Section 3.30.0.3</td>
</tr>
<tr>
<td>nskip</td>
<td>Integer</td>
<td>1</td>
<td>Results output frequency. See Section 3.30.0.4</td>
</tr>
<tr>
<td>rho</td>
<td>Real</td>
<td>1</td>
<td>Select time integrator. See Section 3.30.1.1</td>
</tr>
<tr>
<td>load</td>
<td>Integer</td>
<td></td>
<td>Load to apply during solution case. See Section 3.30.0.5</td>
</tr>
<tr>
<td>write_files</td>
<td>all</td>
<td>none</td>
<td>output</td>
</tr>
<tr>
<td>lfcutoff</td>
<td>Real</td>
<td>-Inf</td>
<td>Exclude any modes below this frequency from the modal computation. Often used to exclude rigid body modes.</td>
</tr>
<tr>
<td>flush</td>
<td>integer</td>
<td>50</td>
<td>Defines how often results are written to the exodus results file. See Section 2.4.1</td>
</tr>
</tbody>
</table>

**Table 3-62.** -- ModalTransient Solution Case Parameters.

Option **modaltransient** is used to perform a modal-superposition-based implicit transient analysis. Damping for the model is defined in Section 4.8.
The parallel solution of modal transient may be slower than expected: while the eigen solution parallelizes well, there is not enough computation to parallelize the modal calculation. Only those DOFs requested for output are included in the solution. Thus, requesting output on only a small subset of the model can improve solution speed.

If output is required at few locations, we recommend setting write_files to history, or specifying an empty outputs definition. Note that an empty outputs definition applies to all solution cases, whereas write_files applies only to the current solution case. Forces and displacements on modal degrees of freedom are also available via the keyword modalvars in the echo definition.

modaltransient supports restart in the eigen part of the analysis, the modaltransient part, or both. In the latter case, two things would happen first. Any modes from the modal restart file are read, and the time history data from any previous transient restart files (direct or modal) is read. Afterwards stepping in time continues. A general discussion of restart is given in Section 2.4.2, which also discusses restart test coverage (Table 2-11).

An example of restart with the modaltransient solution is given below. In this case, the eigen solution is restarted prior to the modaltransient solution. The eigen solution would proceed as follows

```
Solution
  case eigen
  eigen
   nmodes 10
   restart=write
end
```

and, subsequently, the eigen restart and modaltransient would be:

```
Solution
  case eigen
  eigen
   nmodes 20
   restart=read
  case modaltrans
  modaltransient
   nsteps 100
   time_step 1.0e-3
   restart=write
end
```
The previous modal transient cannot restart from the last computed time step when computing additional time steps. To restart it is necessary to add option write in the modal transient case. For example, one could then do the following:

```plaintext
Solution
    case 'eigen'
        eigen
            nmodes 20
            restart=read
    case 'modaltrans'
        modaltransient
            nsteps 100
            time_step 1.0e-3
            restart=write
    case 'modaltrans'
        modaltransient
            nsteps 200
            time_step 1.0e-3
            restart=read
end
```

### 3.21. Q EVP Solution Case

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Type</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 3-63. – Q EVP Solution Case Parameters.

Note: Options for the qevp solution case depend on the type of quadratic eigen solver being used and are documented below.

#### 3.21.0.1. Quadratic Eigenvalue Methods Comparison

The quadratic eigenvalue problem is defined as,

\[
(K + D\lambda + M\lambda^2)u = 0
\]  

(3.20)

The solution of the quadratic eigenvalue problem (3.20), has applications in a variety of physics solutions including coupled structural acoustics, general eigenvalue systems with damping, and gyroscopic systems for rotating structures. Various methods have been developed to address the solution to these problems. The solution to the problem is difficult, and knowledge of the types of systems encountered can help significantly in addressing the robustness of each of the methods. The methods are listed and described in the following paragraphs. Table 3-64 lists recommended procedures for different problem sets.
Anasazi: It is possible to use the Anasazi method, although more testing is needed. It can be used to address two problem areas, 1) the coupled structural acoustics problem, and 2) gyroscopic systems from rotating frames. Currently, it requires that both the mass and the stiffness matrix be non-singular. Previous versions of Sierra/SD used the solution case `qevp` with no method keyword to denote the Anasazi method, and it is the default method to keep consistent with this syntax.

A couple of the parameters for the Anasazi solver for quadratic eigenvalue problems are described in the Table 3-49. Here shifts are not supported, and a warning message may be avoided by setting `shift` to zero. Restarts are not supported either; set `implicit_restarts` to 1. Restarts make the capability easier to use. Without restarts the user is required to set the subspace size, `subspace_size`, to be sufficiently large. The way that the algorithm works is to compute all the modes, and then compare a relative residual to `eig_tol`. If the residuals are large, the modes are not returned to the user. It can be helpful to use a larger value of `eig_tol` than the default, say $10^{-8}$. In this situation, it is helpful to set `anverbosity` to a large value, say And then examine other diagnostic information, including the data written to the screen, to ascertain the accuracy of the modes.

CEIGEN: The `ceigen` method uses methods in ARPACK to solve the quadratic eigenvalue problem. Of methods in Sierra/SD, it is the oldest, and probably the least robust.

SA_EIGEN: The `sa_eigen` method solves a coupled structural acoustics problem by solving a linear, uncoupled eigen problem on each of the domains, and using them as a basis to reduce the coupled equations to a dense system. The dense system is solved using LAPACK routines. The method is applicable to structural/acoustic systems. It is robust. Modal truncation can introduce significant errors. Some solutions can fail (or convergence may be slow) because the decomposition tools know nothing about the two domains.

PROJECTION_EIGEN: The `projection_eigen` method solves the quadratic eigenvalue problem by projecting the problem into a subspace corresponding to the real-valued modes. This smaller subspace is constructed by neglecting the damping matrix, symmetrizing the stiffness matrix, and solving the eigenvalue problem, $Ku = \lambda Mu.$ (3.21)

This smaller problem is then used as a basis for solving the original quadratic eigenvalue problem, which takes the form

$$Ku + \lambda Cu + \lambda^2 Mu = 0$$ (3.22)

The original quadratic eigenvalue problem is then pre and post multiplied by the eigenvectors obtained from the subspace eigenvalue problem. This results in a small quadratic eigenvalue problem which is then solved with a LAPACK method. Finally, the modes from the reduced space are projected to the space corresponding to the original quadratic eigenvalue problem.
As with the `sa_eigen` method, truncation error is a concern with the `projection_eigen` method. The more modes one takes, the smaller the truncation error.

<table>
<thead>
<tr>
<th>Problem</th>
<th>Ceigen</th>
<th>SA_eigen</th>
<th>Anasazi</th>
<th>Projection_eigen</th>
</tr>
</thead>
<tbody>
<tr>
<td>Damped Systems</td>
<td>Acceptable</td>
<td>Acceptable</td>
<td>Fails</td>
<td>Acceptable</td>
</tr>
<tr>
<td>structural acoustics</td>
<td>Fails</td>
<td>Acceptable</td>
<td>Acceptable</td>
<td>Acceptable</td>
</tr>
<tr>
<td>Rotational systems</td>
<td>N/A</td>
<td>N/A</td>
<td>Acceptable</td>
<td>Acceptable</td>
</tr>
<tr>
<td>Damped structural/acoustics</td>
<td>Fails</td>
<td>Acceptable</td>
<td>Fails</td>
<td>Acceptable</td>
</tr>
</tbody>
</table>

**Table 3-64.** – A 2005 Comparison of Quadratic Eigenvalue Problem Methods. Although Ceigen and Anasazi have changed substantially since 2005, this table has not been updated to reflect those changes.

Use the keyword `method` followed by the name of the method (Anasazi, ceigen, sa_eigen, Projection_eigen) to select a `qevp` method. Below is a more detailed description of each `qevp` method, their parameters, and examples of how to use them.

### 3.21.1. Anasazi

The *Sierra/SD* interface to the Trilinos package Anasazi solves the quadratic eigenvalue problem defined as

\[
(K + D\lambda + M\lambda^2)u = 0
\]  

(3.23)

See Section 3.21.0.1 for a comparison of these methods for this problem. As currently implemented, the Anasazi method applies to systems with a non-singular mass and stiffness matrix. The damping matrix may be asymmetric. Options for input are described in Table 3-65. An example is given below.

<table>
<thead>
<tr>
<th>Option</th>
<th>Argument</th>
<th>Default</th>
<th>Comment</th>
</tr>
</thead>
<tbody>
<tr>
<td>nmodes</td>
<td>Integer</td>
<td>10</td>
<td>number of modes</td>
</tr>
<tr>
<td>shift</td>
<td>Real</td>
<td>0</td>
<td>ignored</td>
</tr>
<tr>
<td>reorthogonalize</td>
<td>Yes/No/Full</td>
<td>“Yes”</td>
<td>Reorthogonalize vectors</td>
</tr>
<tr>
<td>check_diag</td>
<td>Yes/No/Full</td>
<td>“Yes”</td>
<td>Check that vectors diagonalize linearized system</td>
</tr>
<tr>
<td>ANverbosity</td>
<td>Integer</td>
<td>17</td>
<td>Anasazi verbosity</td>
</tr>
<tr>
<td>ANblocksize</td>
<td>Integer</td>
<td>1</td>
<td>Anasazi Block Size</td>
</tr>
</tbody>
</table>

Although modal truncation methods are an industry standard, the results sometimes are inaccurate. A solution may not to change when the number of modes increases, and still be inaccurate. Verification is entirely up the user in every case with modal truncation methods.
Users have never prioritized enhancing the usability of Anasazi or CEigen. In addition to the shift, users must select \texttt{eig\_tol}. But discerning an effective value depends on the problem, especially on the shift and the linear solver accuracy. Too large a threshold ($>1.e-4$) degrades solution accuracy. Too small a threshold ($<1.e-13$) leads to divergence. However, with these methods, diagnostic information is provided (written to the standard output stream) to guide users with problem configuration and solution verification.

```
Solution
  case qevp
    qevp
      method=Anasazi
      nmodes=14
      anverbosity=27
  end
```

### 3.21.2. Damped Eigenvalue Problems

The \texttt{qevp} solution with “method=ceigen” is used to select complex eigen analysis using the ARPACK package. This computes the solution to the quadratic eigenvalue problem,

$$
\left( K + D\lambda + M\lambda^2 \right) u = 0
$$

(3.24)

Note that two other solution methods may also be used to evaluate the quadratic eigenvalue problem. Each of these methods has its strengths and weaknesses. A comparison of these methods is provided in Section 3.21.0.1.

The following table gives the parameters needed for complex eigen analysis. Additionally, the eigenvalue tolerance can be set with the \texttt{eig\_tol} parameter in the parameters block (Section 2.3).

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Type</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>nmodes</td>
<td>Integer</td>
<td>100</td>
<td>Number of modes to compute, reported as complex conjugate pairs</td>
</tr>
<tr>
<td>viscofreq</td>
<td>Real</td>
<td>1e-6</td>
<td>Frequency at which to evaluate material damping</td>
</tr>
</tbody>
</table>

Table 3-66. – ceigen Solution Case Parameters.

The optional \texttt{viscofreq} keyword indicates the frequency at which the damping properties of viscoelastic materials will be computed. It must be non-negative. The \texttt{viscofreq}
Table 3-67. – Ceigen Tests.

<table>
<thead>
<tr>
<th>Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ceig</td>
<td>stiffness proportional damping</td>
</tr>
<tr>
<td>ceig_visco</td>
<td>viscoelastic damping</td>
</tr>
<tr>
<td>ceig_dash</td>
<td>dashpot damping</td>
</tr>
<tr>
<td>steel_in_foam</td>
<td>complex mixed materials</td>
</tr>
</tbody>
</table>

parameter can be confusing. In particular, viscoelastic materials typically have high damping at lower frequencies, and lower damping at high frequencies. The viscofreq parameter sets a frequency from which we estimate all the viscoelastic damping. Thus, if viscofreq is small, the damping is large. In particular, if viscofreq is below the glass transition frequency, then damping appropriate to the low frequency modes will be used. This high value of damping is applied to the entire spectrum. It is generally better to over-estimate viscofreq than to underestimate it.

The reason for this difficulty is that even linear viscoelastic materials generate a more complex equation than that shown in equation 3.24. With a single term in the Prony series, the equation of motion for a damped viscoelastic structure can be written in the frequency domain.

\[
\left( K + D \frac{s}{s + \omega_g} + M s^2 \right) u = f(s) \quad (3.25)
\]

Here \( s \) is the Laplace transform variable and \( \omega_g = 1/\tau \) is the reciprocal of the relaxation constant. This system is not a simple quadratic in \( s \). Effectively, viscofreq approximates this system with the linearized system below.

\[
\left( K + D \frac{s}{2\pi \cdot \text{viscofreq} + \omega_g} + M s^2 \right) u = f(s) \quad (3.26)
\]

Computation of quadratic eigen modes is much more difficult than real eigen analysis. The system of equations is more difficult, and more “tricks” must be used to resolve issues that are generated, such as decreasing eig_tol. Even the post processing can be complicated. Like real eigen analysis, one must request displacement output in the output section (see 7.1.13). The output file contains 12 fields (six real and six imaginary). Few post processing tools handle complex mode shapes. Also, see subsection Complex Eigen Analysis — Modal Analysis of Damped Structures section Solution Procedures of the Theory Manual.

Because of the difficulties with complex eigen analysis, it is important to understand the problems for which we have evaluated and tested it. The tests in the test suite are listed in Table 3-67.
3.21.3. **SA_eigen**

The `qevp` procedure with method `SA_eigen` provides a means of computing the modal response of a coupled structural acoustic system, using a modal truncation basis. The quadratic eigenvalue problem describing this system can be written as follows.

\[
\begin{pmatrix}
K_s & 0 \\
0 & K_a
\end{pmatrix} + \lambda \begin{pmatrix}
C_s & L \\
-\rho_aLT & C_a
\end{pmatrix} + \lambda^2 \begin{pmatrix}
M_s & 0 \\
0 & M_a
\end{pmatrix} \begin{pmatrix}
\phi_s \\
\phi_a
\end{pmatrix} = 0
\]  
(3.27)

Here the subscripts refer to structural or acoustic domains, \(\rho_a\) is the density of the fluid and \(L\) is a coupling matrix. Note that for this formulation, \(\phi_a\) represents the acoustic velocity potential, which relates to the time derivative of the acoustic pressure, \(\phi_a = \nabla_\cdot \dot{u}_a\). It helps to understand the capabilities and limitations of this analysis.

The `sa_eigen` method solves this system by solving for the uncoupled eigen modes in the two domains, using them as a basis to reduce the coupled equations to a dense system, and solving the dense system. Thus, it uses a modal reduction technique similar to the Craig-Bampton methods (section 3.5) to generate a dense system of equations that are solved and results propagated to the physical space. More details are available in the theory manual.

Options of the analysis are provided in Table 3-68, and an example is provided in input 3.8. Boundary conditions are applied exactly as for the generalized eigenvalue problem. Exterior, non-reflecting boundary conditions may be applied, but modal convergence is poorer. Loads are irrelevant. Output is complex, as for the `ceigen` case (3.21.2).

```
Solution
  case sa_eigen
    qevp
      method=sa_eigen
      nmodes=20
      nmodes_acoustic=50
      nmodes_structure=26
      acoustic_lfcutoff=-1
      structural_lfcutoff=-1
      sort method = frequency
  end
```

**Input 3.8. SA_Eigen Example**

**Limitations:** This is a modal superposition method. The **Sierra/SD** interface to the Trilinos package Anasazi is a more complete but less robust method which does not rely on modal truncation. The SA_eigen method is accurate for many structural acoustic environments. Damping may be provided, but does tend to slow convergence. The method also depends on the solution to separate structural and
<table>
<thead>
<tr>
<th>Option</th>
<th>Args</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>nmodes</td>
<td>int</td>
<td>Number of requested eigen modes</td>
</tr>
<tr>
<td>nmodes_acoustic</td>
<td>int</td>
<td>Number of free-free acoustic modes in the reduction. Defaults to 2*(nmodes).</td>
</tr>
<tr>
<td>nmodes_structure</td>
<td>int</td>
<td>Number of free-free structural modes in the reduction. Defaults to 2*(nmodes).</td>
</tr>
<tr>
<td>acoustic_lfcutoff</td>
<td>Real</td>
<td>Low frequency cutoff to filter acoustic modes. By default, all modes are retained.</td>
</tr>
<tr>
<td>structural_lfcutoff</td>
<td>Real</td>
<td>Low frequency cutoff to filter structural modes. By default, all modes are retained. Used to eliminate negative modes</td>
</tr>
<tr>
<td>shift</td>
<td>Real</td>
<td>Eigen shift used in computation of the subregion modes. See 3.10.</td>
</tr>
<tr>
<td>sort method</td>
<td>string</td>
<td>magnitude: complex magnitude of $\lambda$ frequency: Sort by frequency and then damping. damping: Sort by damping and then frequency. truefreq: Sort by frequency... avoiding zero energy round off. none: Multicase requires None</td>
</tr>
<tr>
<td>linearization</td>
<td>int</td>
<td>1 $A = [0 \ I; -K \ -C]$; $B = [I \ 0; 0 \ M]$</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2 $A = [-K \ 0; 0 \ M]$; $B = [C \ M; M \ 0]$;</td>
</tr>
<tr>
<td></td>
<td></td>
<td>4 $A = [0 \ -K; M \ 0]$; $B = [M \ C; 0 \ M]$;</td>
</tr>
<tr>
<td></td>
<td></td>
<td>These follow the linearizations in Tisseur</td>
</tr>
<tr>
<td>reorthogonalize</td>
<td>string</td>
<td>no: no reorthogonalization yes: reorthogonalize all modes full: check all modes</td>
</tr>
<tr>
<td>check_diag</td>
<td>string</td>
<td>no: no check for orthogonalization yes: check redundant modes full: check all modes</td>
</tr>
</tbody>
</table>

Table 3-68. – SA_Eigen Options.
acoustic subregion eigen problems. These solutions are not as robust as full system
eigen analysis. Please see the notes in the verification manual for convergence details.
Table 3-69 summarizes the status of this procedure.

**Low Frequency Cutoff:** The parameters acoustic_lfcutoff and structural_lfcutoff remove
low frequency modes before initiating the qevp. This will reduce the number of
modes (nmodes_acoustic and nmode_structure) in the analysis. Negative cutoff
frequencies are allowed.

<table>
<thead>
<tr>
<th>Analytic Reference</th>
<th>Verification Section</th>
<th>Tested</th>
<th>Parallel Test</th>
<th>User Test</th>
</tr>
</thead>
<tbody>
<tr>
<td>31</td>
<td>33</td>
<td>Y</td>
<td>Y</td>
<td>some</td>
</tr>
</tbody>
</table>

Table 3-69. – Verification Summary for SA_Eigen.

**Specialized Output:** There are a few items that are output specifically for the sa_eigen
procedures that can be helpful in assessing the solutions.

**StructuralFraction** It is useful to know which modes participate in which regions.
This is computed as follows.

Let $\phi$ be the eigenvector computed on the reduced space. We divide $\phi$ into its
structural and acoustic components. i.e.,

$$\phi = \begin{bmatrix} \phi_s \\ \phi_a \end{bmatrix}$$

We compute,

$$F_{structure} = \frac{\phi_s^\dagger \cdot \phi_s}{\phi_s^\dagger \cdot \phi_s + \phi_a^\dagger \cdot \phi_a}$$

(3.28)

where $\phi^\dagger$ represents the transpose and complex conjugate of $\phi$. Note that these
products are computed in the reduced space which has coordinates associated
with each structural or acoustic eigen mode. In the reduced space, the mass
matrix is identity, and the vector product, $\phi^\dagger \cdot \phi$ represents an energy norm.

**AcousticFraction** The acoustic fraction is the analogue of the structural fraction (eq.
3.28) applied the acoustic domain. It represents the portion of the system level
complex eigen mode that is associated with the acoustic domain.

**ErrorNorm** We define a normalized modal energy residual.

$$E_{\text{resid}}^n = \frac{|\phi^\dagger(k + \lambda c + \lambda^2 M)\phi|}{\phi^\dagger K \phi}$$

(3.29)

Here $\phi$ and $\lambda$ are the estimates of the eigenpairs computed using the modal
approximation technique. The matrices, $k$, $c$ and $m$ are the fully assembled
stiffness, coupling and mass matrices. This residual norm is a measure of the
relative accuracy of the eigenvalue solution. It is available in both the text
results files and the output Exodus files, and should be consulted to determine
the convergence.
<table>
<thead>
<tr>
<th>Option</th>
<th>Args</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>nmodes</td>
<td>int</td>
<td>Number of requested eigen modes</td>
</tr>
<tr>
<td>shift</td>
<td>Real</td>
<td>Eigen shift used in computation of the subregion modes. See 3.10.</td>
</tr>
</tbody>
</table>
| reorthogonalize | string | no: no reorthogonalization  
|              |      | yes: reorthogonalize all modes  
|              |      | full: check all modes |
| check_diag   | string | no: no check for orthogonalization  
|              |      | yes: check redundant modes  
|              |      | full: check all modes |
| sort method  | string | magnitude: complex magnitude of \( \lambda \)  
|              |      | frequency: Sort by frequency and then damping.  
|              |      | damping: Sort by damping and then frequency.  
|              |      | truefreq: Sort by frequency... avoiding zero energy round off.  
|              |      | none: |

Table 3-70. – Projection_Eigen Options.

3.21.4. Projection_eigen

The *Projection_Eigen* method is the most robust of all the solvers available for quadratic eigenvalue problems. Options of the *Projection_Eigen* solver are provided in Table 3-70. These parameters are identical to those for the *sa_eigen* method.
### 3.22. NLStatics Solution Case

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Type</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>tolerance</td>
<td>Real</td>
<td>1e-6</td>
<td>Controls completion of the Newton iteration.</td>
</tr>
<tr>
<td>max_newton_iterations</td>
<td>Integer</td>
<td>100</td>
<td>If the iteration count exceeds this value before reaching <strong>tolerance</strong>, the Newton loop is considered to have failed.</td>
</tr>
<tr>
<td>update_tangent</td>
<td>Integer</td>
<td>101</td>
<td>How often the tangent stiffness matrix is rebuilt during the Newton iterations.</td>
</tr>
<tr>
<td>num_newton_load_steps</td>
<td>Integer</td>
<td>1</td>
<td>Number of load steps used to incrementally step up to the final equilibrium position</td>
</tr>
</tbody>
</table>

Table 3-71. – NLStatics Solution Case Parameters.

If stiffness matrix $K$ is a function of the displacement $u$ in

$$K(u) = f,$$

then use **NLstatics** for nonlinear statics. Newton’s method applied to the residual force equations to drive the residual $r = p - f$ to zero. The residual vector $r$ is the difference between the internal force vector $p$ and the external force vector $f$. The internal force vector is a function of the structural displacements (and possibly velocities). External forces can also be a function of the structural displacements in the case of follower loads such as surface pressure loads.

The **tolerance** keyword provides control over the completion of the Newton iteration. Once the change in the $L^2$-norm of the displacement decreases below **tolerance**, the loop completes successfully.

The **num_newton_load_steps** keyword controls the number of load steps used to incrementally step up to the final equilibrium position. Large loads may cause the Newton algorithm to diverge. If this occurs, increase the number of load steps applied. Displacements will be output after each load step which may be animated similar to transient dynamics simulations.

The **update_tangent** keyword controls how often the tangent stiffness matrix is rebuilt during the Newton iterations. The default is set to update the tangent stiffness matrix at the beginning of a load step. Setting **update_tangent** to 1 is equivalent to using a full-Newton algorithm where the tangent stiffness matrix is rebuilt after each Newton
iteration. For nonlinear (difficult) problems, this option may be optimal, but for most problems the extra cost of assembling a preconditioning the tangent stiffness matrix should be amortized over several solves. Note that for this option to improve Newton’s method, the element types in the model must have the tangent stiffness method implemented.

An example **Solution** section is shown below.

```plaintext
Solution
  title 'Example of a nonlinear statics solution'
  nlstatics
  tolerance = 1e-6
  max_newton_iterations = 100
  num_newton_load_steps = 10  // split load into 10 increments
  update_tangent = 1  // full-newton algorithm
end
```
### 3.23. NLTransient Solution Case

The NLTransient solution method is used to perform a direct implicit nonlinear transient analysis. A projector-corrector step is used. Note that for a linear system, the NLTransient analysis will require two solves per time step.

The tolerance keyword provides control over the completion of the Newton iteration. Once the change in the $L^2$-norm of the acceleration decreases below tolerance, the loop completes successfully. Note the difference viz a viz NLstatics (3.22), where tolerance instead refers to the displacement.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Type</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>time_step</td>
<td>Real</td>
<td></td>
<td>Time step size. See Section 3.30.0.1</td>
</tr>
<tr>
<td>nsteps</td>
<td>Integer</td>
<td></td>
<td>Number of time steps to take. See Section 3.30.0.2</td>
</tr>
<tr>
<td>start_time</td>
<td>Real</td>
<td>0.0</td>
<td>Solution case start time. See Section 3.30.0.3</td>
</tr>
<tr>
<td>nskip</td>
<td>Integer</td>
<td>1</td>
<td>Results output frequency. See Section 3.30.0.4</td>
</tr>
<tr>
<td>rho</td>
<td>Real</td>
<td>1</td>
<td>Select time integrator. See Section 3.30.1.1</td>
</tr>
<tr>
<td>load</td>
<td>Integer</td>
<td></td>
<td>Load to apply during solution case. See Section 3.30.0.5</td>
</tr>
<tr>
<td>write_files</td>
<td>all/ none/ output/ history</td>
<td>all</td>
<td>Controls which result files are written during this solution.</td>
</tr>
<tr>
<td>tolerance</td>
<td>Real</td>
<td>1e-6</td>
<td>Controls completion of the Newton iteration.</td>
</tr>
<tr>
<td>max_newton_iterations</td>
<td>Integer</td>
<td>100</td>
<td>If the iteration count exceeds this value before reaching tolerance, the Newton loop is considered to have failed.</td>
</tr>
<tr>
<td>update_tangent</td>
<td>Integer</td>
<td>101</td>
<td>How often the tangent stiffness matrix is rebuilt during the Newton iterations.</td>
</tr>
<tr>
<td>flush</td>
<td>integer</td>
<td>50</td>
<td>Defines how often results are written to the exodus results file. See Section 2.4.1</td>
</tr>
</tbody>
</table>

Table 3-72. – NLTransient Solution Case Parameters.
If the iteration count in a given time step exceeds `max_newton_iterations`, the Newton loop is considered to have failed. Thus, note that `max_newton_iterations` is not the limit for the total number of Newton iterations, as it is in the `NLstatics` (3.22) case, but is instead the limit on the number of iterations per time step.

In a `NLstatics` (3.22) analysis, load stepping can be used to help the convergence of the Newton loop by cutting the total load into a series of incremental steps. This is controlled with the `num_newton_load_steps` keyword. However, in `NLtransient` analysis, load stepping makes no sense because the dynamic response of a structure subjected to a total load is different from the response to a series of incremental loads. In effect, the load stepping is replaced by time stepping in the case of nonlinear transient analysis. Thus, the keyword `num_newton_load_steps` is inactive for nonlinear transient analysis.

For `NLtransient` problems, if Newton’s method diverges, either the tangent stiffness matrix has to be updated more often (see `update_tangent`) or the time-step should be decreased.

The `update_tangent` controls how often the dynamic tangent stiffness matrix is rebuilt during the Newton iterations. The default is set to 101, and thus unless a given Newton loop takes more than 101 iterations, the tangent matrix will not be updated by default. Setting `update_tangent` to 1 is equivalent to using a full-Newton algorithm where the dynamic tangent stiffness matrix is rebuilt after each Newton iteration. Note that currently there is no option for forcing a tangent update at the beginning of each time step, unless the `update_tangent` keyword is set to exactly the number of Newton iterations taken per time step. For non-linear problems, some control of this option is recommended. Note, for this option to improve Newton’s method, the element types in the model must have the dynamic tangent stiffness method implemented.

### 3.24. Receive_Sierra_Data Solution Case

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Type</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>include_internal_force</code></td>
<td><code>off/on</code></td>
<td><code>on</code></td>
<td>Include or disable computation of initial internal force</td>
</tr>
<tr>
<td><code>no_geom_stiff</code></td>
<td></td>
<td></td>
<td>Turn off the geometric stiffness term</td>
</tr>
</tbody>
</table>

`Table 3-73. – Receive_Sierra_Data Solution Case Parameters.`

Solution case receive_sierra_data inputs a deformed or preloaded model state. Calculations in SIERRA codes such as Sierra/SM may be transferred to Sierra/SD. This provides the ability to compute nonlinear responses in a separate code, followed by a supported solution case in Sierra/SD. For SM/SD hand-off also see chapter 182.
“Sierra/SM to Sierra/SD Coupling” and. In particular see the “Sierra/SM output parameters for Sierra/SD modal analysis using file transfer.” section.

Solution method receive_sierra_data helps with the following.

1. Update the initial geometry from the previously computed displacements.

2. Update the element stiffness matrices due to preload stresses. This is the geometric stiffness correction. The geometric stiffness correction is supported for volumetric and membrane elements at this time. Stress transfer for shells and beams is not enabled. The geometric stiffness calculation may be disabled with the no_geom_stiff keyword.

3. Compute and apply an initial force associated with the input stress state. Disable this by setting include_internal_force=off command.

4. Update of tangent stiffness of materials based the preloaded material state.

receive_sierra_data solutions require a multicase solution. An example follows. In this example preload data is received from the input Exodus file, that preload alters the stiffness matrix, and eigenvalues are computed using this updated stiffness matrix.

```
Solution
  case transfer
    receive_sierra_data
  case eig
    eigen
    nmodes=40
    shift=-3e6
end
```

The receive_sierra_data solution is specifically designed to read initial conditions and tangent stiffness state computed in a Sierra/SM analysis. Note that connections between different parts of the model specified in the Sierra/SM input deck are not transferred, e.g., contact conditions, MPCs, and joints. These conditions may be equivalently specified in Sierra/SD. The following options are available to receive_sierra_data.

include_internal_force When set to the default value (on), the imported stress state is integrated to generate an internal force body load. This load is included in the right-hand side of subsequent static or transient analysis. If the Sierra/SM preload analysis is in static equilibrium, the externally applied forces from boundary conditions will be in balance with the internal forces generated by the elements. By default, the internal forces will be computed again in Sierra/SD. Thus, if the same forces are included in the Sierra/SD input deck that causes the preload in Sierra/SM, then the internal force should be included to keep the model in static equilibrium. An alternative is to exclude the boundary conditions causing the preload from Sierra/SD and set include_internal_force=off. In this case, Sierra/SD will compute no initial internal force, and the initial state of the model will be
treated as perfect equilibrium. See the verification manual chapter “Sierra/SM to Sierra/SD Coupling” for more detail on this topic.

**no_geom_stiff** Can be used to ignore the preloaded stress contribution to the geometric stiffness matrix. Generally, elements in tension have higher effective stiffness, and elements in compression have lower effective stiffness. At sufficiently large compression, element stiffness can go negative, which causes severe problems for solver stability. Turning off geometric stiffness can be used for debugging purposes, or to evaluate what effect the geometric stiffness term has on model response. See the verification manual chapter “Sierra/SM to Sierra/SD Coupling” for more detail on this topic.

### 3.25. Data Transfer

The purpose of `receive_sierra_data` is transfer of data from a previous Sierra/SM analysis to Sierra/SD. Data relevant to the load is mostly transferred automatically based on expected naming conventions. For the Lamé hyperfoam model (section 4.1.4), and orthotropic layers in shells (section 4.1.2 and table 3-76), the option `from_transfer` must be used to turn on the material tangent update for specific material parameters. The fields relevant to `receive_sierra_data` are given in tables 3-74 to 3-76.

<table>
<thead>
<tr>
<th>Exodus Label</th>
<th>Description</th>
<th><code>&lt;key&gt;</code> (for initialize variable name)</th>
<th>Effect</th>
</tr>
</thead>
<tbody>
<tr>
<td>displ_x</td>
<td>nodal displacement</td>
<td>displacement(x)</td>
<td>Added to mesh coordinates and used as initial mesh configuration. See below for details</td>
</tr>
<tr>
<td>displ_y</td>
<td>displacement</td>
<td>displacement(y)</td>
<td></td>
</tr>
<tr>
<td>displ_z</td>
<td></td>
<td>displacement(z)</td>
<td></td>
</tr>
<tr>
<td>vel_x</td>
<td>nodal velocity</td>
<td>velocity(x)</td>
<td>Used to set initial conditions for transient analysis</td>
</tr>
<tr>
<td>vel_y</td>
<td>velocity</td>
<td>velocity(y)</td>
<td></td>
</tr>
<tr>
<td>vel_z</td>
<td></td>
<td>velocity(z)</td>
<td></td>
</tr>
<tr>
<td>rv_x</td>
<td>and rotational velocity</td>
<td>rotational_velocity(x)</td>
<td></td>
</tr>
<tr>
<td>rv_y</td>
<td></td>
<td>rotational_velocity(y)</td>
<td></td>
</tr>
<tr>
<td>rv_z</td>
<td></td>
<td>rotational_velocity(z)</td>
<td></td>
</tr>
<tr>
<td>force_internal_x</td>
<td>force from stress state</td>
<td>force_internal(x)</td>
<td>Used for equilibrium diagnostics</td>
</tr>
<tr>
<td>force_internal_y</td>
<td></td>
<td>force_internal(y)</td>
<td></td>
</tr>
<tr>
<td>force_internal_z</td>
<td></td>
<td>force_internal(z)</td>
<td></td>
</tr>
</tbody>
</table>

*Table 3-74. – Nodal data used in `receive_sierra_data`.*

Only accurately labeled data will be transferred from the Exodus file. An exact match is typically required, although some variables have multiple valid names. For example, `stress_xx` or `stressxx` are allowed. The data, typically written from Sierra/SM, may require special output requests (in the Sierra/SM input file) for proper naming. See the How To or Verification manual for examples.
<table>
<thead>
<tr>
<th>Exodus Label</th>
<th>Description</th>
<th>&lt;key&gt; (for initialize variable name)</th>
<th>Effect</th>
</tr>
</thead>
<tbody>
<tr>
<td>stress_xx</td>
<td>volumetric stress components</td>
<td>stress(xx)</td>
<td>Used to compute geometric stiffness (turn off with no_geom_stiff) and internal force (turn off with include_internal_force=off).</td>
</tr>
<tr>
<td>stress_yy</td>
<td></td>
<td>stress(yy)</td>
<td></td>
</tr>
<tr>
<td>stress_zz</td>
<td></td>
<td>stress(zz)</td>
<td></td>
</tr>
<tr>
<td>stress_xy</td>
<td></td>
<td>stress(xy)</td>
<td></td>
</tr>
<tr>
<td>stress_yz</td>
<td></td>
<td>stress(yz)</td>
<td></td>
</tr>
<tr>
<td>stress_zx</td>
<td></td>
<td>stress(zx)</td>
<td></td>
</tr>
<tr>
<td>left_stretch_xx</td>
<td>Element Stretch</td>
<td>N/A</td>
<td>Used for material tangent stiffness calculations for Lamé material models.</td>
</tr>
<tr>
<td>left_stretch_yy</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>left_stretch_zz</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>left_stretch_xy</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>left_stretch_yz</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>left_stretch_zx</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>rotation_xx</td>
<td>Element Rotation</td>
<td>N/A</td>
<td>Used for material tangent stiffness calculations for Lamé material models.</td>
</tr>
<tr>
<td>rotation_yy</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>rotation_zz</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>rotation_xy</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>rotation_yx</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>rotation_yz</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>rotation_zx</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>rotation_xz</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>lame_state_&lt;model&gt;</td>
<td>Material State</td>
<td>N/A</td>
<td>Used for material tangent stiffness calculations for Lamé material models. Supported &lt;model&gt;s are “neo_hookean” and “hyperfoam”</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Table 3-75.** – Element data used in receive_sierra_data. Only the Neo-Hookean and hyperfoam Lamé models are supported; see Section 4.1.4.
<table>
<thead>
<tr>
<th>Exodus Label</th>
<th>Description</th>
<th>〈key〉 (for initialize variable name)</th>
<th>Effect</th>
</tr>
</thead>
<tbody>
<tr>
<td>memb_stress_xx</td>
<td>fiber membrane stress components</td>
<td>memb_stress(xx)</td>
<td>Geometric stiffness and internal force</td>
</tr>
<tr>
<td>memb_stress_yy</td>
<td></td>
<td>memb_stress(yy)</td>
<td></td>
</tr>
<tr>
<td>memb_stress_zz</td>
<td></td>
<td>memb_stress(zz)</td>
<td></td>
</tr>
<tr>
<td>memb_stress_xy</td>
<td></td>
<td>memb_stress(xy)</td>
<td></td>
</tr>
<tr>
<td>memb_stress_yz</td>
<td></td>
<td>memb_stress(yz)</td>
<td></td>
</tr>
<tr>
<td>memb_stress_zx</td>
<td></td>
<td>memb_stress(zx)</td>
<td></td>
</tr>
<tr>
<td>bottom_stress_xx</td>
<td>fiber membrane stress components</td>
<td>bottom_stress(xx)</td>
<td>Geometric stiffness and internal force</td>
</tr>
<tr>
<td>bottom_stress_yy</td>
<td></td>
<td>bottom_stress(yy)</td>
<td></td>
</tr>
<tr>
<td>bottom_stress_xy</td>
<td></td>
<td>bottom_stress(xy)</td>
<td></td>
</tr>
<tr>
<td>fibermod</td>
<td>fiber modulus</td>
<td>fibermodulus</td>
<td>Tangent stiffness *</td>
</tr>
<tr>
<td>ax</td>
<td>primary fiber direction</td>
<td>fiberdir</td>
<td>Tangent stiffness *</td>
</tr>
<tr>
<td>ay</td>
<td>secondary fiber direction</td>
<td>fiberdir2</td>
<td>Tangent stiffness *</td>
</tr>
<tr>
<td>fiberdensity</td>
<td>fiber density</td>
<td>fiberdens</td>
<td>Mass *</td>
</tr>
<tr>
<td>fiberthickness</td>
<td>thickness</td>
<td>fiberthickness</td>
<td>Tangent stiffness *</td>
</tr>
<tr>
<td>E1</td>
<td>material modulus</td>
<td>N/A</td>
<td>Tangent stiffness *</td>
</tr>
<tr>
<td>E2</td>
<td>material modulus</td>
<td>N/A</td>
<td>Tangent stiffness *</td>
</tr>
<tr>
<td>nu12</td>
<td>Poisson ratio</td>
<td>N/A</td>
<td>Tangent stiffness *</td>
</tr>
<tr>
<td>G12</td>
<td>material modulus</td>
<td>N/A</td>
<td>Tangent stiffness *</td>
</tr>
<tr>
<td>density</td>
<td>material density</td>
<td>N/A</td>
<td>Mass *</td>
</tr>
</tbody>
</table>

Table 3-76. – Data Transferred in receive_sie rra_data specific to orthotropic_layer materials. Some of this data is transferred automatically. Other data requires the from_transfer keyword in the material definition (denoted by an asterisk in the Effect column).
Alternatively, Sierra/SD also supports user-defined setup of some input variables using `initialize variable name = <key>` in the FILE section, where the appropriate `<key>` for each variable (if applicable) is listed in tables 3-74 to 3-76. The `read variable` and `variable type` may be used to read input variables from a non-default field name. The `function` and `variable type` may be used to initialize nodal quantities via an analytic function.

An example for displacements is shown below, where nodal displacements are stored as “dx”, “dy”, and “dz” on the input geometry file `input_mesh.g`.

```plaintext
FUNCTION my_disp_z
  type analytic
  expression variable dz_Input = nodal dz
  evaluate expression = "0.5 * dz_Input"
END

FILE
  geometry_file = input_mesh.g

  initialize variable name = displacement(x) # x-component of displacement
  variable type = node # is stored in a nodal field
  read variable = dx # on input_mesh.g named "dx"

  initialize variable name = displacement(y) # y-component of displacement
  variable type = node # is stored in a nodal field
  read variable = dy # on input_mesh.g named "dy"

  initialize variable name = displacement(z) # z-component of displacement
  variable type = node # calculated via analytic
  function = my_disp_z # expression, in this case
  # by scaling "dz" from mesh
END
```

Input 3.9. Analytic function based on input displacements

A shorthand notation of the user-defined label mapping is also supported and shown below for the same example of displacement. For vector fields, ‘x’, ‘y’ and ‘z’ is appended to the variable name, and for tensor fields, “xx”, “yy”, “zz”, “xy”, “yz” and “zx” is appended. For tensor fields, the order of off-diagonal indices is irrelevant (“xy” is the same as “yx”).

```plaintext
FILE
  geometry_file = input_mesh.g

  initialize variable name = displacement # displacement is stored
```
Coordinate Update  The receive_sierra_data method uses Sierra/SM displacement to update the initial coordinates of the mesh. Mass matrices are computed in the undeformed configuration, to better ensure conservation of mass in this hand-off. However, the initial coordinates for Sierra/SD are updated with the transferred displacements.

\[ \vec{x} = \vec{x}_o + \vec{u} \]

Here \( x_o \) is the location of the undeformed coordinates. Stiffness matrices, forces, and tied MPCs are then computed in the updated frame. Tied MPCs can be defined via tied surfaces (Section 8.1) or contact (Section 8.2).

3.26. Statics Solution Case

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Type</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>load</td>
<td>Integer</td>
<td></td>
<td>Load to apply during solution case. See Section 3.30.0.5</td>
</tr>
</tbody>
</table>

Table 3-77. – Statics Solution Case Parameters.

The statics keyword is required if a static solution is needed, i.e. the solution to the system of equations \( [K]\{u\} = \{f\} \). An example Solution section is shown below.

```
Solution
  title "Example of a statics solution"
  statics
end
```

3.27. Superposition Solution Case

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Type</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>reduced_file</td>
<td>String</td>
<td></td>
<td>Ignored?</td>
</tr>
</tbody>
</table>

Table 3-78. – Superposition Solution Case Parameters.
Superposition is currently BETA release. Enable with the “--beta” command-line option.

The superposition provides superelement recovery capability. This recovers physical space solutions from generalized degrees of freedom.

A CB model generates a transformation matrix consisting of a combined set of fixed interface and constraint modes. See section 3.5. This modal basis may be stored in an exodus file. A netcdf file containing the reduced order model is also created at this time. Subsequently, this reduced model is inserted into a residual model for superelement analysis, say a transient analysis. That analysis outputs the standard exodus results, and may also generate output on the netcdf file. These data may be post-process using linear superposition to determine output quantities on the original interior degrees of freedom of the superelement. This is illustrated in Figure 3-22.

![Figure 3-22. – Superposition Data Flow Diagram.](image)

The superposition method requires a single argument, the file name of the reduced order model containing the output results. In addition, the geometry_file specified in the FILE section must contain the modal basis for the Craig-Bampton reduction. See the example in input 3.10.

```plaintext
solution
    case superposition
        superposition
            reduced_file=rom-out.ncf
        end
    end

file
    geometry_file basis-out.exo
end
```

Input 3.10. Superposition Example. Output will be to “basis-out-superposition.exo”
Limitations  The superposition method is under development and contains the following limitations.

- Solutions are supported in serial at this time.
- Only displacement, velocity and acceleration may be output.
- Superposition is tested for Eigen and Transient solutions. There is no support for frequency domain solutions at this time.
- Data recovery is possible for a single superelement in each run.

3.28. Tangent Solution Case

<table>
<thead>
<tr>
<th>Parameter</th>
</tr>
</thead>
<tbody>
<tr>
<td>Type</td>
</tr>
<tr>
<td>Default</td>
</tr>
<tr>
<td>Description</td>
</tr>
</tbody>
</table>

Table 3-79. – Tangent Solution Case Parameters.

The tangent solution step requires a multicase solution (see paragraph 3.2). It forces an update of the tangent stiffness matrix. It is typically used following a nonlinear solution step to ensure that the following step begins using the tangent stiffness matrices computed from the previous result. However, it may also be used following a linear solution step, in which case the stiffness matrix is computed again based on the current value of displacement.

During tangent the stiffness is computed again based on element configuration in the deformed state. Tangent also adds any stress stiffening effects from the preceding load case. Stress resultants from the preload step are not added to stress resultants of subsequent cases after the tangent solution, but do contribute to increasing the effective stiffness computed in the tangent stiffness. For example, the stress stiffening that occurs from tension of a guitar string would be successfully captured using the tangent solution case. As another example the stiffness of a warped plate may be significantly different from the stiffness of a flat plate, tangent will take into account this change in stiffness.

The tangent stiffness matrix is assembled at the subdomain level from computations at the element level. It represents the partial derivative of the force with respect to the displacement, i.e.

\[ K_{\text{tangent}} = \frac{\partial f}{\partial u} \]  

(3.30)

In eigen analysis, the tangent stiffness matrix replaces the linear stiffness matrix in the eigenvalue equation. This permits computation of modal response following a preload.
### 3.29. TranShock Solution Case

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Type</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>time_step</td>
<td>Real</td>
<td></td>
<td>Time step size. See Section 3.30.0.1</td>
</tr>
<tr>
<td>nsteps</td>
<td>Integer</td>
<td></td>
<td>Number of time steps to take. See Section 3.30.0.2</td>
</tr>
<tr>
<td>start_time</td>
<td>Real</td>
<td>0.0</td>
<td>Solution case start time. See Section 3.30.0.3</td>
</tr>
<tr>
<td>nskip</td>
<td>Integer</td>
<td>1</td>
<td>Results output frequency. See Section 3.30.0.4</td>
</tr>
<tr>
<td>rho</td>
<td>Real</td>
<td>1</td>
<td>Select time integrator. See Section 3.30.1.1</td>
</tr>
<tr>
<td>load</td>
<td>Integer</td>
<td></td>
<td>Load to apply during solution case. See Section 3.30.0.5</td>
</tr>
<tr>
<td>write_files</td>
<td>all/ none/ output/ history</td>
<td>all</td>
<td>Controls which result files are written during this solution.</td>
</tr>
<tr>
<td>SRS_damp</td>
<td>Real</td>
<td>0.03</td>
<td>Damping coefficient used for the shock response spectra calculation</td>
</tr>
</tbody>
</table>

| Table 3-80. – TranShock Solution Case Parameters. |

The TranShock solution method is used to perform a direct implicit transient analysis followed by computation of the shock response spectra for the degrees of freedom in a specified node set. The options for configuring a transient solution case described in Section 3.30 are all applicable in the TranShock solution case too. An example of a TranShock solution case and frequency block are shown in input 3.11. Damping for the implicit transient portion of the simulation is defined in Section 4.8. The srs_damp is not used in the modal transient portion of the simulation.

A frequency block must also be included in the input file for TranShock solution cases to define the frequencies and nodesets for computing the shock response spectra (see Section 7.4). The frequencies for computing the shock response spectra are defined by setting freq_step, freq_min, and freq_max. They are identified in the frequency section along with an application region (see Section 7.4). The range of the computed frequency spectra is controlled by freq_min and freq_max, while freq_step controls the resolution. The accuracy of the computed spectra is independent of the magnitude of freq_step. This parameter controls the output resolution. Examples are presented in inputs 3.6, 3.7 and 3.11.
The shock spectrum procedure will compute acceleration results. The options specified in the outputs and echo blocks are used in the transient portion of the analysis, but are ignored for the post processing of the transient results into shock spectra. Thus, if displacement, velocity, and/or acceleration is selected in the outputs and/or echo sections for a shock spectra analysis, the results echoed to the output listing or the Exodus output file will be time history results as requested, but the shock spectra results will be for acceleration response for the nodes in the specified node set. *The calculated shock spectra are written to the frequency file (*.frq); they are not output to the Exodus results file.*

```
Solution
  transhock
    time_step .00005
    nsteps 500
    nskip 1
    srs_damp .03
  end

Frequency
  freq_min 100.
  freq_max 10000.
  freq_step 100.
  nodeset 3
  acceleration
end
```

**Input 3.11. Transhock Example Input**
### 3.30. Transient Solution Case

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Type</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>time_step</td>
<td>Real</td>
<td></td>
<td>Time step size. See Section 3.30.0.1</td>
</tr>
<tr>
<td>nsteps</td>
<td>Integer</td>
<td></td>
<td>Number of time steps to take. See Section 3.30.0.2</td>
</tr>
<tr>
<td>start_time</td>
<td>Real</td>
<td>0.0</td>
<td>Solution case start time. See Section 3.30.0.3</td>
</tr>
<tr>
<td>nskip</td>
<td>Integer</td>
<td>1</td>
<td>Results output frequency. See Section 3.30.0.4</td>
</tr>
<tr>
<td>rho</td>
<td>Real</td>
<td>1</td>
<td>Select time integrator. See Section 3.30.1.1</td>
</tr>
<tr>
<td>load</td>
<td>Integer</td>
<td></td>
<td>Load to apply during solution case. See Section 3.30.0.5</td>
</tr>
<tr>
<td>write_files</td>
<td>all/none/output/history</td>
<td>all</td>
<td>Controls which result files are written during this solution.</td>
</tr>
<tr>
<td>PredictorCorrector</td>
<td>Integer</td>
<td>-1</td>
<td>predictor-corrector implementation</td>
</tr>
<tr>
<td>ConstraintCorrectionFrequency</td>
<td>Integer</td>
<td>1</td>
<td>time-step frequency for constraint correction</td>
</tr>
<tr>
<td>ConstraintErrorDiagnostics</td>
<td>yes/no</td>
<td>no</td>
<td>prints constraint errors</td>
</tr>
<tr>
<td>flush</td>
<td>integer</td>
<td>50</td>
<td>Defines how often results are written to the exodus results file. See Section 2.4.1</td>
</tr>
<tr>
<td>nUpdateConstraints</td>
<td>Integer</td>
<td>0</td>
<td>Frequency to update constraints</td>
</tr>
</tbody>
</table>

Table 3-81. – Transient Solution Case Parameters.

The transient solution method is used to perform a direct implicit transient analysis.

#### 3.30.0.1. Time_step Option

The time_step defines the time step size. The time step size is constant for a time period. Multiple time periods can be given as shown in example 3.30.0.6.
3.30.0.2. Nsteps Option
The nsteps defines the number of time steps to take. As shown in example 3.30.0.6, multiple time periods can be defined with different numbers of steps.

3.30.0.3. Start_time Option
The start_time sets the initial time of a transient analysis. By default, a transient case begins where a previous transient solution case ended, or the time transferred from a receive_sierra_data or other preload solution case. Otherwise, the default start time is 0.0. To set the start time in the input deck specify start_time.

3.30.0.4. Nskip Option
The nskip controls how many integration steps to take between outputting results. It defaults to 1, which is equivalent to outputting all time steps. Because transient analysis often takes little computational time per step, the overall runtime can be significantly reduced by choosing not to output the results at every step, i.e., setting nskip greater than 1. The nskip in the solution section can be overridden for history (Section 7.3) and linesample (Section 7.5) output by specifying nskip in those sections.

3.30.0.5. Load Option
The load defines the name of the “load” block to apply during the solution case. See Section 6.3.1 for details.

3.30.0.6. Defining Multiple time Periods
We note that multiple time step values, along with the corresponding number of steps, can be specified for transient analysis. This can be useful for separating the simulation into a section of tiny time steps followed by a section of larger time steps, or vice versa. The following provides an example of the use of multiple time steps.

```
solution
  time_step 1.0e-5 1.0e-3
  nsteps 100 500
  nskip 10 1
end
```

In this case, the user requested 100 time steps of $\Delta t = 10^{-5}$, followed by 500 time steps with $\Delta t = 10^{-3}$. There is no practical limit on the number of such regions that may be specified.
3.30.0.7. PredictorCorrector Option

PredictorCorrector indicates whether predictor-corrector implementation is to be used (see theory manual). It defaults to 1. If it is 1, predictor-corrector is always used, while 0 indicates that it is never used. ConstraintCorrectionFrequency and ConstraintErrorDiagnostics are related to the constraint errors arising in the predictor-corrector implementation, with the first controlling how often the correction is applied, while the second can be used to examine the evolution of constraint errors of displacement, velocity and acceleration.

3.30.1. nUpdateConstraints Option

nUpdateConstraints is currently BETA release. Enable with the “--beta” command-line option. nUpdateConstraints defines the frequency to update constraints (e.g. MPCs and contact). When it is undefined or set equal to 0, the constraints will never be updated. A value of 1 results in the constraints being updated every time-step. In addition to re-building the constraints, the nodal coordinates will also be updated at the corresponding time-steps based on the interpolation of any existing displacements on the mesh (if applicable).

The name of the displacement nodal variables can be defined by the initialize variable name, read variable, and variable type options in the FILE section, as shown in section 3.24.

This option is related to the nodesets_with_disp option (section 2.3), which enables specifying displacements on a subset of all nodes via nodeset output. However, nodeset displacements do not currently support user-defined variable names as with nodal displacements.

3.30.1.1. Numerical damping

Two time integrator schemes are available for direct time integration. The method and the configuration of the integrator are selected using the keyword rho. If this keyword is not found, the time integrator defaults to a standard Newmark beta integration scheme. With rho the Generalized Alpha method is used, and the value of the numerical damping is controlled by rho.

Important
Due to the inexactness of linear solvers, the Newmark beta integrator is conditionally unstable. Without damping, a solution may gradually diverge. Also, see subsection Linear transient analysis section Solution Procedures of the Theory Manual. Either proportional damping or numerical damping is strongly recommended in all cases.
The option \texttt{rho} defines the numerical damping of the Generalized Alpha method. \texttt{rho} varies from 0 (maximal damping case) to 1 (minimal damping case). If \texttt{rho} is not specified in the input file, the integrator defaults to the Newmark beta method. Otherwise, the code uses the value of \texttt{rho} given by the user to configure the Generalized Alpha method. Therefore, there is no value default for \texttt{rho}, as shown in the table above, since if it is not specified the code uses the Newmark beta method instead. If \texttt{rho} is specified to be greater than 1 or negative an error message is printed. \texttt{rho} determines Newmark beta, $\alpha_f$, and $\alpha_m$ of the Generalized Alpha method. More detailed information on the implementation, and references can be found in the description of the method in the \texttt{Sierra/SD} Theory Manual.

The following conditions suffice to achieve second order accuracy and unconditional stability:

\[
\begin{align*}
\alpha_m &< \alpha_f \leq \frac{1}{2} \\
\gamma_n &= \frac{1}{2} - \alpha_m + \alpha_f \\
\beta_n &\geq \frac{1}{4} + \frac{1}{2}(\alpha_f - \alpha_m)
\end{align*}
\]

The parameters are determined to satisfy the conditions. Specifically,

\[
\begin{align*}
\alpha_f &= \rho/(1 + \rho) \\
\alpha_m &= (2\rho - 1)/(1 + \rho) \\
\beta_n &= (1 - \alpha_m + \alpha_f) \cdot (1 - \alpha_m + \alpha_f)/4 \\
\gamma_n &= 1/2 - \alpha_m + \alpha_f
\end{align*}
\]

We note some special cases of interest. If $\rho = 0$, we have that $\alpha_f = 0$ and $\alpha_m = -1$. This is the maximum damping case. If $\rho = 1$, we have that $\alpha_f = \alpha_m = 1/2$, which yields $\beta_n = 1/4$, and $\gamma_n = 1/2$. This is similar to the classical undamped Newmark beta method, although we note that it is a different algorithm since $\alpha_f = \alpha_m = 1/2$ implies some lagging in the time-stepping procedure. The classical undamped Newmark beta method has $\alpha_f = \alpha_m = 0$.

Unlike proportional damping, there is no direct relation between \texttt{rho} and an equivalent modal damping term. A value of \texttt{rho}=0.9 is recommended for most analyses. The Generalized Alpha integrator imparts numerical damping to the solution that most strongly affects high frequency content. Users must check that the damping in the frequency range of interest is physical. For example with a time step size of $1e-5$, damping has the most effect at frequencies above the Nyquist frequency $0.5e+5$.

\subsection*{3.30.1.2. Initial Acceleration}

Determining the initial acceleration is necessary\cite{28} for quadratic convergence. If force, velocity or displacement is specified, an initial solve may be required to determine a consistent acceleration. Remember that determination of the displacement at step $n+1$
depends on values at the previous step. Specifically, the acceleration at the previous step is given by the solution to the equation,

\[ A_n = M^{-1}(F_n + Kd_n + Cv_n) \]

where \( M, K, \) and \( C \) are the corresponding mass, stiffness and damping matrices.

If this mass solve is not performed, it is possible to introduce a spike in acceleration which can oscillate through time. Initialization can be a somewhat tricky process. An example set of use cases is provided in Appendix 6.5. By default, GDSW is used. Some combinations of MPCs can lead to a singular mass matrix that will cause solver errors. For these cases the initial mass solve is deactivated with the following command in the parameters block (see Section 2.3),

```plaintext
Parameters
  DoInitialMassSolve = false
end
```

### 3.31. **TSR_preload Solution Case**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Type</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>linedata_only</td>
<td>True/False</td>
<td>False</td>
<td>Indicates that no system matrices should be computed, but the linedata specified in the linesample file should be computed for verification of data transfer.</td>
</tr>
</tbody>
</table>

Table 3-82. – TSR_preload Solution Case Parameters.

The `tsr_preload` solution method reads an Exodus file with a previously computed Thermal Structural Response (TSR) into Sierra/SD for a subsequent statics or transient dynamics analysis. This is not a fully coupled calculation. Stress results are read from the geometry file, an equivalent internal force is computed, and that internal force is combined with the applied force throughout the transient run. If temperature data is also included in the file, it will be read and used to compute temperature dependent material properties. A `tsr_preload` requires a multicase solution, and it must be followed by a transient dynamics or statics solution (see paragraphs 3.2 and 3.30 respectively).

Note that since the stresses are converted into a force, and since there is no immediate deformation in transient dynamics, the elastic stresses output by Sierra/SD will be small initially, i.e. they will not contain a contribution from the thermal stress. However, at large times, the deformation from the internal force will result in an elastic stress opposite to
that of the thermal stress. The linesample method 7.5 recovers the input thermal stress as an output quantity (in either MATLAB or Exodus format).

The tsr_preload solution method is considered to be a temporary solution to a more complicated problem. In the future, TSR analysis will involve coupling to other mechanics codes. In many cases a thermal load (Section 6.3.8), may provide equivalent capability.

Data in the Exodus file from which TSR data is read must strictly match the following criteria. There must be one time step in the result. That time step must have a number of different element fields defined. These correspond to the six stress values of the stress tensor and the number of stress tensors defined per element must correspond exactly to the number of integration points for that element. For instance, a hex20 element requires exactly 27 stress tensor values per element. An error is produced if the number of data points read in does not match the number of integration points for that element. For instance, a fully integrated hex8 will produce an error if reading in Gauss point data produced by a fully integrated hex20 element. Shell and beam type elements are not supported in tsr_preload.

The labels for the stresses must be as shown in the table below where SigXX is interchangeable with SigMA_XX or STRESS_XX. Both sequential and ijk\(^60\) numbering are supported for integration point data. For sequential numbering, replace \%d with an integer representing the integration point value (0 to 26). Do not zero pad. For ijk numbering,\(^60\) labeling goes as SigXX_Hex20_GP%d where \%d is replaced with the ijk numbering scheme and _Hex20 is replaced with the element type. Do not include _%d for labeling centroid stress data.

<table>
<thead>
<tr>
<th>Name</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>SigXX_%d</td>
<td>(\sigma_{xx}), the (xx) component of stress</td>
</tr>
<tr>
<td>SigYY_%d</td>
<td>(\sigma_{yy}), the (yy) component of stress</td>
</tr>
<tr>
<td>SigZZ_%d</td>
<td>(\sigma_{zz}), the (zz) component of stress</td>
</tr>
<tr>
<td>SigYZ_%d</td>
<td>(\sigma_{yz}), the (yz) component of stress</td>
</tr>
<tr>
<td>SigXZ_%d</td>
<td>(\sigma_{xz}), the (xz) component of stress</td>
</tr>
<tr>
<td>SigXY_%d</td>
<td>(\sigma_{xy}), the (xy) component of stress</td>
</tr>
</tbody>
</table>

The following is an example solution section for a TSR preload followed by transient dynamics.

```
Solution
  title 'Pure bending from initial stress'
  case TSR
    tsr_preload
    load 1
  case bend
    transient
    time_step 1.e-6
    start_time 1.0e-3
```
If executed on a file with `geometry_file = example.exo`, this will produce two output files, `example-tsr.exo` and `example-bend.exo`. The first of these has little useful information. The second contains the displacements (or other variables) from the transient analysis.

### 3.31.0.1. Line Sample

One additional feature for thermal structural response is the ability to do line sampling on the original Exodus file containing the element stresses. This is useful for debugging and verification. It allows the stresses along lines within the structure to be examined. Sampling occurs for data stored on integration points using the variables names described above. Line sample is used for energy deposition (see the Two Element Exponential Decay Variation Hex20 problem). Energy deposition is interchangeable with supplying an applied temperature.

In tsr_preload, the input Exodus file is required to contain at least one of the following fields: stress, temperature or energy deposition. Any field that is not found in the input Exodus file is reported as a zero field in the output line sample output file.

### 3.32. Residual Vectors Solution Case

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Type</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>node_list_file</td>
<td>File</td>
<td>Unknown</td>
<td></td>
</tr>
</tbody>
</table>

Table 3-83. – Residual Vectors Solution Case Parameters.

Residual Vectors is currently BETA release.

Enable with the “--beta” command-line option.

The `residual_vectors` solution method Modal truncation augmentation (MTA) provides a method to represent the modes not retained in an eigen solution. It is particularly useful in component mode synthesis approaches where multiple models are joined together. In NASTRAN, MTA vectors are referred to as ‘residual vectors’. The theory of MTA is established. We use the following terminology:
N | Number of degrees of freedom
nev | Number of retained eigenvalues/eigenvectors from the eigen solution
nf | Number of applied forces and/or moments
M | Mass matrix of size N×N
K | Stiffness matrix of size N×N
Φ | Matrix with eigenvectors as columns, size N×(nev)
Ω² | Diagonal matrix of eigenvalues, size (nev)×(nev)
R₀ | Applied spatial load vector, size N×(nf)
Rs | Modally represented spatial load vector, size N×(nf)
Rₜ | Force truncation vector, size N×(nf)
X | Static displacements due to applied loads, size N×(nf)
ω² | Diagonal matrix of reduced eigenvalues, size (nf)×(nf)
Q | Matrix of reduced eigenvectors, size (nf)×(nf)
P | Matrix of modal truncation (residual) vectors, size N×(nf)

The algorithm for computing MTA vectors is:

1. Solve the generalized eigenvalue problem

   \[ K\Phi = M\Phi\Omega^2 \]

   for nev eigenvalues and eigenvectors. This is done by first specifying eigen in a multicase solution procedure.

2. Compute the force truncation vector

   \[ Rₜ = R₀ - Rₙ = R₀ - M\Phi\Phi^T R₀. \]

3. Compute the static displacements \(X\) due to the force truncation vector \(Rₜ\) by solving \(K X = Rₜ\).

4. If rigid body modes are present, orthogonalize \(X\) to them. The optional input nrbms allows the user to specify the number of rigid body modes present.

5. Form the reduced matrices \(nf \times nf\),

   \[ K = X^T K X, \quad M = X^T M X. \]

6. Solve the reduced generalized eigenvalue problem \(KQ = MQ\bar{\omega}²\)

7. Form the modal truncation (residual) vectors: \(P = XQ\)

8. Construct the pseudo modal set: \(\Phi = [\Phi|P]\).

In Sierra-SD, the multi-case solution strategy is:

1. Solve the eigenvalue problem

2. For each column of \(R₀\), solve a statics problem

3. Solve a residual_vectors problem to form the pseudo modal set

An example solution block is given here for a case with six rigid body modes.
Solution

```plaintext
title 'Sample MTA solution procedure'
case 'eigen'
  eigen
    nmodes=30
    shift=-1000. // needed for floating
  solver=gdsw

case 'residual_vectors'
  residual_vectors
    node_list_file node1
end
```

### 3.33. GeometricRigidBodyModes Solution Case

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Type</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
</table>

Table 3-84. – GeometricRigidBodyModes Solution Case Parameters.

Nominal rigid body modes may be determined from the coordinates. No attempt is made to account for boundary conditions. This solution method requires that the GDSW linear solver be used.

The intent of the examples is to first introduce rigid modes, next use the modes to solve an eigenvalue problem, and then demonstrate the Modal Transient capability 3.20. The third example uses the modes in a modal transient simulation to deflate out the rotations. This section depends on Section 6.3.19.

The geometry rigid body mode capability will always generate six modes and these modes are explicitly ordered +X, +Y, +Z, +RX, +RY, +RZ.

Rigid body modes are requested in the SOLUTION block.

```plaintext
SOLUTION
    geometric_rigid_body_modes
END
PARAMETERS
    num_rigid_mode 6
END
```

Rigid body modes can be incorporated into the modes computed in a modal analysis, and then used for other purposes.
Rigid body modes are the 6 lowest frequency eigenvectors. In this case 4 more modes are computed, for a total of 10.

In this example a modal transient simulation uses the geometric rigid body modes to deflate out the (infinitesimal) rotation, while retaining the translational rigid body modes. This is equivalent to use of the FilterRbmLoad for direct transient solutions (though accomplished differently).
3.34. Waterline Solution Case

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Type</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>max_iterations</td>
<td>Integer</td>
<td>100</td>
<td>Maximum number of solution iterations</td>
</tr>
<tr>
<td>tolerance_force</td>
<td>Real</td>
<td>1.0e-6</td>
<td>Target force balance accuracy</td>
</tr>
<tr>
<td>point_a</td>
<td>Real(3)</td>
<td></td>
<td>Coordinates of point on estimated water surface</td>
</tr>
<tr>
<td>point_b</td>
<td>Real(3)</td>
<td></td>
<td>Coordinates of point on estimated water surface</td>
</tr>
<tr>
<td>point_c</td>
<td>Real(3)</td>
<td></td>
<td>Coordinates of point on estimated water surface</td>
</tr>
<tr>
<td>VizOption</td>
<td>none</td>
<td>Ensight</td>
<td>none</td>
</tr>
</tbody>
</table>

Table 3-85. – Waterline Solution Case Parameters.

It can be advantageous to determine the waterline of a ship prior to commencing more complex analysis. The waterline capability solves the nonlinear geometric equations of equilibrium for a rigid ship in water. An example is shown in input 3.12.

Solution
```plaintext
Solution
  case 'waterline'
    waterline
      max_iterations 100
      tolerance_force 1.0e-6 // absolute tolerance on force convergence
      point_a 0 0 0 // coords of point 'A' on estimated water surface
      point_b 1 0 0 // coords of point 'B' on estimated water surface
      point_c 1 1 0 // coords of point 'C' on estimated water surface
    load 1
  case 'transient'
    ...
end

LOAD 1
  sideset 1 // wetted sideset
    pressure = 1
    function = 1 // this defines rho g h
```
Input 3.12. Waterline Example

The arguments point_a, point_b and point_c indicate the Cartesian coordinates of three points A, B, C on the estimated water surface. These three points define a plane, which serves as the initial guess of the waterline. The waterline normal is determined using the right-hand rule with these points, as shown in Figure 3-23. The Newton’s method implementation then uses this plane as the initial guess, and begins iterations towards force and moment equilibrium. On completion, we write out the coordinates of three points on the final (converged) waterline surface, along with the Cartesian coordinate system defined by these points. This output appears in the result file in text format. A grep,script for moving the body may also be written.

The optimization is configured as follows.

max_iterations sets the maximum number of iterations.

tolerance_force is a normalized force residual. The norm is computed from the residual vector,

\[ F_{\text{residual}} = [F_z/W, M_{\theta_1}/(LW), M_{\theta_2}/(LW)] \]

where \( W = Mg \) is the total weight of the ship, and \( L \) is a characteristic length of the model.

VizOption may be none or Ensight to generate a visualization script.

In addition to the entries in the Solution section of the input, this method requires two load entries and a function. The load entries define the sideset for the wetted surface and the gravity load. The function defines the pressure as a function of depth. In the example of input 3.12, the argument to the function is the depth, \( h \). The function returns \( P = \rho gh \).

\[ ^1 \text{Gravity is specified using the standard load keywords of a body load with a gravity vector. However, for the waterline solution, the magnitude of the gravity vector is relevant. The gravity direction is always directed opposite the normal to the surface for this solution type.} \]
Figure 3-23. – Waterline Coordinate Definition. The plane of the surface is defined by three points: A, B, and C. The $\theta_1$ rotation is about the line from A to B, while the normal is defined using the right-hand rule.

The waterline iteration may output nodal data during the iteration. Select `force` to output the buoyancy force. Select `npressure` to output the nodal pressure. See the Outputs section, 7, for details.

3.34.0.1. Limitations There are a number of limitations to this method.

**gradient-based optimization:** These powerful algorithms are based on nonlinear gradient-based optimization and have subtle limitations. Limitations are listed below.

1. Singular tangent matrices are generated in various conditions, which cause the solution to terminate. A common condition causing a singular tangent matrix is a body completely submerged in a constant-density fluid. For simplicity, consider an unrotated cube $^2$ of edge length $S$. The net force on the cube is,

$$F_{\text{net}} = (A_{\text{bottom}} P_{\text{bottom}} - A_{\text{top}} P_{\text{top}}) - mg$$  \hspace{1cm} (3.31)

$$= \rho_f g S^2 (h_{\text{bottom}} - h_{\text{top}}) - mg$$  \hspace{1cm} (3.32)

$$= \rho_f g S^3 - \rho_s g S^3$$  \hspace{1cm} (3.33)

$^2$The arithmetic is easier for a cube, but the arguments can be shown to be completely valid for any rigid body.
where $\rho_f$ and $\rho_s$ are the densities of the fluid and solid respectively. Significantly, the net force does not depend on the average depth. Thus,

$$K_t = \frac{\partial F_{net}}{\partial z} = 0.$$ 

$K_t = 0$ for a ship that is completely out of the water too.

Real seawater is not constant density. An optimal solution may be found in this case. However, because the pressure is usually expressed as a piecewise linear function, the same problem occurs. Use of a runtime function may allow computation of higher-order derivatives, but this has not been evaluated.

Figure 3-24 plots net force versus depth for a body. Only the partially submerged region has a nonzero tangent matrix that can be determined by a gradient-based optimization scheme.

![Net Force vs Depth for a Rigid Body](image)

**Figure 3-24.** – Net Force vs depth for a Rigid Body. Only the unshaded region, where the body is partially submerged, has a non-singular tangent matrix.

2. Gradient-based solution methods often have trouble with local minima. These can occur in the case of unstable systems, such as a light, tall cylinder floating on a dense fluid. A local minimum occurs for the cylinder standing vertically. A global minimum is achieved when the cylinder is perturbed and falls to the side.

3. Gradients may also go to zero for symmetry reasons. A perfect cylinder floating on the water has no sensitivity to roll.

**sidetset orientation:** The wetted surface defines the pressure surface. It does not need to be closed. However, there can be no contribution to the net force from portions of the model that are submerged, but not part of the sideset.
One and only one sideset defines the wetted surface. Its outward direction should point into the water. There is no check for a reversal of the normal vectors on the sideset. This must be evaluated by the analyst.

*Z-orientation* Current design requires that the initial configuration has gravity approximately aligned with the global \( Z \) coordinate.

### 3.35. Gap Removal Solution Case

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Type</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ignore_gap_inversion</td>
<td>true</td>
<td>false</td>
<td>false</td>
</tr>
</tbody>
</table>

Table 3-86. – Gap Removal Solution Case Parameters.

If two meshes are tied using either **tied data** or **contact definition**, then along the interface opposite elements may initially overlap or leave gaps. Gap removal, which is done by default, attempts to remove these gaps and overlaps. Gap removal is the same as **initial overlap** removal.

The **gap_removal** solution case is used to debug contact setup prior to submitting a full run. The **gap_removal** solution case runs quickly and uses low memory. This solution method enables visualization of the constraints created and the gap removed from **tied data** 8.1 and **contact definition** 8.2 blocks. The **gap_removal** solution method reads in the mesh, applies the contact search and gap removal algorithm and writes out the output mesh with gap removed. An example input is given below.

```plaintext
solution
  gap_removal
end

tied data
  surface 1,2
  name "tied_1-2"
  search tolerance 1.0e-3
end
```

Removal of contact gaps is essential to maintaining rigid body invariance. This is illustrated in Appendix 8.3. However, the removal of gaps in tied surfaces can occasionally result in distorted elements which may make it difficult to impossible for the solver to converge. Thus, it can be advantageous to investigate the results of gap removal before committing to a full and expensive solution case.
Gap removal output will include two element variables. The variable 
\texttt{elementInversionFlag} is set to one on any inverted element. The variable 
\texttt{elementQuality} gives a non-dimensional element quality metric (one is ideal higher is 
worse) for the deformed configuration of every element. The same quality metric is used as 
described in Section 7.1.12.

If gap removal inverts any element the file name extension “-gap” will be appended to the 
output exodus file and a fatal error will be given. One of the parameters described in 
Section 2.3 influences the Gap Removal solution case. The parameter is 
\texttt{ignore_gap_inversion}. It is false by default. Set it to true to suppress both the fatal 
error and the “-gap” output behavior.

Gap removal for lofted surfaces is discussed in Section 8.3.

In addition to the element shape, information diagnostics regarding the contact constraints 
are available. The rslt file will list basic information on the number of constraints found. 
Detailed visual information on constraint locations is obtained by requesting 
\texttt{constraint_info} in the \texttt{outputs} block as described in Section 7.1.48. Furthermore, 
adding \texttt{MPC} to the \texttt{echo} block prints every contact created MPC as described in Section 
7.7.2.

\textbf{Coupled Electro-Mechanical Analysis} Piezoelectricity is the production of electrical 
charges on a surface by the imposition of mechanical stress. \textit{Sierra/SD} supports coupled 
electro-mechanical physics in order to model piezoelectric materials subjected to electrical 
and mechanical forces. This support includes static, transient, eigen, and direct frequency 
response solution methods, piezoelectric and dielectric material models (4.5, 4.6), and 
voltage measurement based inverse methods such as material and source identification. 
Electrical boundary conditions such as charge-based Neumann (6.3.12) and voltage-based 
Dirichlet(6.1.2, 6.1.3) boundary conditions are also supported.\(^{41}\)

\section{Materials} \label{sec:materials}

The material section has a (unique) material identifier (an integer or a string name). The material identifier is used in assigning material properties to element blocks. Material 
types and their parameters are summarized in Table 4-87.

\begin{table}[h]
\centering
\begin{tabular}{|l|l|}
\hline
material type & parameters \\
\hline
isotropic & any two of $K$, $G$, $E$ or $\mu$ \\
optotropic & nine $C_{ij}$ entries \\
optotropic\_prop & E1, E2, E3, $\mu_{23}$, $\mu_{13}$, $\mu_{12}$, $G_{23}$, $G_{13}$, $G_{12}$ \\
anisotropic & 21 $C_{ij}$ entries \\
\hline
\end{tabular}
\caption{Material Stiffness Parameters.}
\end{table}
For example,

```
Material steel
  isotropic
  E 3e7
  nu .3
End
```

A materials may be isotropic, orthotropic, orthotropic_prop, anisotropic, or isotropic_viscoelastic.

The Joint2G element 5.21 has material models for joints including an elastic-plastic model.

### 4.1. Elastic

Elastic materials may be isotropic section 4.1.1, orthotropic section 4.1.2, or anisotropic section 4.1.3. Some material models from the Lamé library are available section 4.1.4.

#### 4.1.1. Isotropic

Isotropic materials require specification of two of the following parameters. They can be defined directly as `parameter = <real>`, as functions of temperature (Section 4.4.6), or as spatially dependent properties (Section 4.4.7.)

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>E</td>
<td>Young’s Modulus</td>
</tr>
<tr>
<td>nu</td>
<td>Poisson’s Ratio</td>
</tr>
<tr>
<td>G</td>
<td>Shear Modulus</td>
</tr>
<tr>
<td>K</td>
<td>Bulk Modulus</td>
</tr>
</tbody>
</table>

Isotropic materials are the default, and the keyword `isotropic` is not required. Of the four parameters, exactly two must be supplied. They are related by

\[
E = 3K(1 - 2\nu), \quad G = \frac{3KE}{9K - E}.
\]

Internally, *Sierra/SD* stores the values of K and G.
4.1.2. Orthotropic

Orthotropic material entry is similar to the anisotropic case.

A difference is that the keyword orthotropic replaces anisotropic, and only 9 $C_{ij}$ entries are specified. These entries correspond to $C_{11}$, $C_{12}$, $C_{13}$, $C_{22}$, $C_{23}$, $C_{33}$, $C_{44}$, $C_{55}$ and $C_{66}$. Like the anisotropic material definition, we follow the stress/strain ordering of $xx$, $yy$, $zz$, $zy$, $zx$, $xy$.

Alternatively, an orthotropic material may be specified using orthotropic_prop and the material parameters $E_1$, $E_2$, $E_3$, $\nu_{12}$, $\nu_{23}$, $\nu_{13}$, $G_{23}$, $G_{13}$, and $G_{12}$ as shown in the following example. As with isotropic materials, temperature-dependent parameters may be defined via a function as parameter = function <string> (see Section 4.4.6). Note that all elastic materials must satisfy requirements that the elasticity matrix is positive definite.

```
Material honeycomb
  orthotropic_prop
  E1 = 508.7
  E2 = 7641.0
  E3 = 14750.0
  nu12 = .2
  nu23 = .0825
  nu13 = .1
  G12 = 115
  G23 = 2320.
  G13 = 450.
  density=0.5
End
```

A single orthotropic layer may be specified using orthotropic_layer. An orthotropic layer must specify 4 of the above parameters ($E_1$, $E_2$, $\nu_{12}$, $G_{12}$). The syntax parameter = from_transfer may also be used to receive_sierra_data solution case to obtain material parameter values from a file transfer. Here is an example:

```
Material 13
  orthotropic_layer
  E1 = 508.7
  E2 = 7641.0
  nu12 = 1.293
  G12 = 115
  density=0.5
End
```
If sensitivity analysis is being performed (see Section 2.6), one indicates the parameters for analysis by following these parameters with the +/- characters. In the first entry method, a sensitivity analysis must be performed on all 9 parameters. In the second, each individual parameter must be requested individually. The concept is that the sensitivity is performed with respect to the labeled parameters, i.e. either the set of $C_{ij}$ parameters, or each individually labeled E1 term.

4.1.3. Anisotropic

Anisotropic materials require specification of a 21 element $C_{ij}$ matrix corresponding to the upper triangle of the $6 \times 6$ stiffness matrix. Data is input in the order $C_{11}, C_{12}, C_{13}, C_{14}, C_{15}, C_{16}, C_{22}, \text{etc.}$ The $C_{ij}$ must be preceded by the keyword Cij. The keyword anisotropic is also required. Materials are specified according to the stress/strain ordering $xx, yy, zz, yz, zx, xy$. This is generally consistent with published Materials Science data. However, NASTRAN and Abaqus use a different convention. An input deck illustrating anisotropic material input is provided in Section 9.2.

If an element block uses a coordinate system the anisotropic material is defined in the $\hat{r}, \hat{s}, \hat{t}$ local frame 7.1.6. If an element block does not use a coordinate system the anisotropic material is defined in the X, Y, Zframe. The frame used by each element may be visualized by requesting material_direction_1, material_direction_2, and material_direction_3 in the outputs block.

For anisotropic (and orthotropic) materials, the check to make sure material properties are acceptable is skipped. A message is printed notifying the user that this check is skipped.

4.1.4. Lamé Material

Lamé material is currently BETA release. Enable with the “--beta” command-line option.

Sierra/SD provides a limited capability to use linearized versions of the non-linear Lamé material models in Sierra/SD. This can be used in conjunction with the capability to hand-off a nonlinear preload to a linear Sierra/SD analysis. This hand-off is accomplished by reading in the element variables stress, left_stretch, rotation, and Lamé state (e.g. lame_state_hyperfoam) at the last time-step of your Exodus input file. One example where this would be useful is computing the tangent stiffness of a compressed foam.

Lamé materials are defined in a material section. As in input 4.1. A Lamé material definition begins with begin-lame-material and ends with end-lame-material. Currently, only Neo-Hookean and Hyperfoam materials are supported.

| Material 1 |
| begin-lame-material |
begin parameters for model Hyperfoam
  bulk modulus = 1.e6
  Poissons ratio = 0.1
  n = 3
  shear = 3.74e6, -3.17e6, 1.18e4
  alpha = 2.536, 2.090, -8.807
  Poisson = 0.5630, 0.5507 0.3662
end
end-lame-material
density = 5.0
End

Input 4.1. Example material section for a Lamé Hyperfoam material model.

Note: unlike the rest of a Sierra/SD input file, the material model definition (emphasized text in input 4.1) must strictly follow syntax rules including newlines. Details of the allowed syntax for each material model are given below. **Lamé Neo-Hookean model**

Acceptable syntax for Neo-Hookean models is given below.

```
BEGIN PARAMETERS FOR MODEL NEO_HOOKEAN
#
# Elastic constants
#
  YOUNGS MODULUS = <real> E
  POISSONS RATIO = <real> ν
  SHEAR MODULUS = <real> G
  BULK MODULUS = <real> K
  LAMBDA = <real> λ
  TWO MU = <real> 2μ
END [PARAMETERS FOR MODEL NEO_HOOKEAN]
```

A detailed discussion of the theory of the Neo-Hookean model can be found in the Sierra Solid Mechanics User Manual.\(^{58}\)

**Lamé Hyperfoam Model** Acceptable syntax for Hyperfoam models is given below.

```
BEGIN PARAMETERS FOR MODEL HYPERFOAM
#
# Elastic constants
#
  YOUNGS MODULUS = <real> E
  POISSONS RATIO = <real> ν
  SHEAR MODULUS = <real> G
  BULK MODULUS = <real> K
```
As with Neo-Hookean models, a detailed discussion of the theory of the Hyperfoam model can be found in the Sierra Solid Mechanics User Manual.\textsuperscript{58}

### 4.2. Acoustic

Linear acoustic materials require the specification of the fluid density, and the linear speed of sound. In addition, the keyword \texttt{acoustic} must be in the material block.

```
Material air
  acoustic
  density 1.293
  c0 332.0
  cavitating
  pvapor 0.0
End
```

Nonlinearity can be activated by the keyword \texttt{nonlinear}. Nonlinear acoustic materials require one additional parameter, \texttt{B\_over\_A}, which is a measure of fluid nonlinearity. For air, \texttt{B\_over\_A} = 0.4. Tables of \texttt{B\_over\_A} for various fluids can be found in.\textsuperscript{26}

Cavitation can be activated by the keyword \texttt{cavitating}. This requires an additional parameter, \texttt{pvapor}, which is the vapor pressure with a default value equal to 0. Cavitating elements also require the definition of vectors for \texttt{hydrostatic\_gravity} and \texttt{free\_surface\_point} in the \texttt{block} section as shown in input 4.2. \texttt{hydrostatic\_gravity} and \texttt{free\_surface\_point} are vectors used to compute the external pressure at the block.

```
BLOCK 1
  material 1
    hydrostatic\_gravity = 0 0 -32.2
    free\_surface\_point = 0 0 34.0
End
```
Input 4.2. This is an example of a block section for a cavitating material.

For computational acoustics see Section 2.4.4.

4.3. Linear Viscoelastic

Linear viscoelastic materials require the specification of the density, and the limiting moduli $E_g$, $E_{\text{inf}}$, $G_g$, $G_{\text{inf}}$. The subscript ‘g’ refers to the glassy modulus, which occurs at $t = 0$, or $\omega = \infty$. The subscript ‘inf’ refers to the rubbery modulus, which occurs at $t = \infty$, or $\omega = 0$. In addition the Prony series for the viscoelastic materials have to be specified using keywords K_coeff, K_relax, G_coeff, and G_relax. All of these parameters are required.

For the bulk modulus $K$, the Prony series parameters are defined by the following equation:

$$K(t) = K_{\text{inf}} + (K_g - K_{\text{inf}}) \sum_i K_{\text{coeff}}[i] * e^{-\frac{t}{K_{\text{relax}}[i]}}$$  \hspace{1cm} (4.1)

A similar equation holds for the shear modulus. Note that, the K_coeff and G_coeff must sum to 1.0 (individually). Otherwise, the formulation is inconsistent. That is,

$$\sum_i K_{\text{coeff}}[i] = \sum_i G_{\text{coeff}}[i] = 1.0.$$  \hspace{1cm} (4.2)

Note that the number of terms in K_coeff and K_relax must be the same, and the number of terms in the G_coeff and G_relax must be the same. However, the number of terms in the K series does not have to equal the number of terms in the G series. Thus, one could simulate a case where the material shear modulus $G$ is viscoelastic, but the bulk modulus is not. In this case, the latter would have no terms in its series.

$E_g$, $E_{\text{inf}}$, $G_g$, $G_{\text{inf}}$ may be constant or may depend on temperature. Temperature functions can specify the value for the limiting moduli, for a given value of temperature. For example, if the limiting moduli typically depend linearly on temperature, a linear function can be specified for the values of $E_g$, $E_{\text{inf}}$, $G_g$, $G_{\text{inf}}$. We refer to the example given below for the specifics on how to set this up.

Optional parameters for viscoelastic materials include reference ($T_0$), glassy ($T_g$). In the event that none of the moduli are specified as functions, the values specified for these parameters determine which model is used for temperature-dependent behavior. The $T_g$ parameter may be given as a constant or as a spatially dependent property (Section 4.4.7.)

Two such models are available: the WLF (Williams-Landel-Ferry) model, and the Hinrich’s model. The WLF model is used when shifting temperatures above $T_g$, while the Hinrich’s model is used shifting temperatures below $T_g$. Both models incorporate the reference temperature $T_0$ (the reference temperature at which the input viscoelastic constants are defined), which may differ from $T_g$. The WLF and Hinrich’s model use...
model-specific constants. We note that any units of temperature can be used, as long as
they are consistent with the values of the constants. The shift factors computed from the
Hinnerich’s or WLF equations are used to scale the coefficients in the Prony series.

The WLF model is
\[
\log_{10}(a_T) = -\frac{C_1(T_{\text{elem}} - T_0)}{C_2 + T_{\text{elem}} - T_0},
\]
where \(T_{\text{elem}}\) is the current temperature in the element, and \(T_0\), \(C_1\), and \(C_2\) are material
parameters that are determined experimentally. Typically, \(T_0\) is the glass transition
temperature of the material of interest. The shift factors computed from the WLF
equation are a strong function of temperature.

The Hinnerich’s model, provided by Terry Hinnerich’s, accurately characterizes many
viscoelastic materials below the glassy transition temperature. Its form is
\[
\log_{10}(a_T) = a_{T1} \times (1 - e^{a_{T2} \times (T_{\text{elem}} - T_0)}),
\]
where \(a_{T1}\) and \(a_{T2}\) are user-specified constants. This equation used to determine an
approximate set of shift factors when experimental data for a particular material is not at
hand.

Note if a model is shifted through \(T_g\) a composite shift is used. For example if \(T_0\) is less
than \(T_g\) and \(T_{\text{elem}}\) is above \(T_g\) first a Hinnerich’s shift is used to shift parameters from \(T_0\)
to \(T_g\) then a WLF shift is used to transition those constants from \(T_g\) to \(T_{\text{elem}}\). These shifts
are automatically computed given \(T_0\), \(T_g\), block temperature, \(C_1\), \(C_2\), \(a_{T1}\), and \(a_{T2}\).
Note that if these shifting parameters are not specified in the input file, then no shifting
will be done and the relaxation times as specified in the input deck will be used. If \(T_g\) is
unspecified then it defaults to the provided value of \(T_0\). Either specify all the shifting
coefficients, or specify none of them. Partial specification rarely succeeds.

After computing the shift factors using one of the two approaches given above, the
relaxation times are shifted. This occurs before computations begin using the relations,
\[
K_{\text{relax}}[i] = a_T K_{\text{relax}}[i]
\]
\[
G_{\text{relax}}[i] = a_T G_{\text{relax}}[i].
\]

Example 4.3 demonstrates how Hinnerich’s model is used to set a linear viscoelastic material.

```
Material foam
  isotropic_viscoelastic
  T_0= 10
  T_g = exo_var scalar t_g_input
  C_1=15.
  C_2=35.
  aT_1=6.
  aT_2=.0614
  K_g = function 1
```
Input 4.3. Hinnerich’s viscoelastic material specification

Note that the coefficients of both $K$ and $G$ sum to 1.0. This is necessary for a consistent formulation. Also, in this case we specify a temperature function for $K_g$. Thus, the value of $K_g$ used in the simulations is the value of function 1, at the particular element temperature $T_{elem}$. The $T_g$ value is shown reading from an input mesh exodus field named ‘t_g_input’. $T_g$ can also be specified as a constant like the other parameters.

4.3.1. Limitations of Viscoelastic Use.

Linear viscoelastic materials are “linear” in the sense that (linear) transient dynamics accommodates them exactly. There are limitations for the use of these materials in Sierra/SD.

1. When using viscoelastic materials in a nonlinear transient simulation, it is necessary to specify “nonlinear=no” in the BLOCK section of the viscoelastic block. This is because different internal force mechanisms are called for linear and nonlinear cases, and viscoelastic materials in Sierra/SD only support a linear constitutive model and small deformation.

2. When viscoelastic materials are used in a statics simulation, the material is assigned the properties $G_{inf}$ and $K_{inf}$. This is because in a slow (static) loading, the material responds with these material properties since they are the long-time or slow response properties.

3. Likewise, Eigen solutions and the modal based solutions derived from them apply only the first terms of the Prony series for $G_{inf}$ and $K_{inf}$, which are used to define the elastic constants of an isotropic elastic material. This is because the real modal solution must use a constant mass and stiffness matrix, and has no damping contribution.

4. It is possible to evaluate the eigenvalues expanded about a given frequency, viscofreq. See the discussion of ceigen in Section 3.21.2. The damping matrix, $C$, is taken into account in the eigenvalue problem. Few modal solutions are adapted to use these complex modes.
Complex Viscoelastic  Complex isotropic viscoelastic materials may be defined for directFRF solution cases with the *isotropic_viscoelastic_complex* keyword. This material type has 4 required parameters, representing the real and imaginary shear and bulk moduli: $G_{\text{real}}, G_{\text{imag}}, K_{\text{real}},$ and $K_{\text{imag}}$. (The real and imaginary components are commonly known as the storage and loss moduli, respectively.) Each of these complex viscoelastic parameters must be defined by a frequency-varying function, e.g. \texttt{parameter = function <string>}. 

Example 4.4 demonstrates how a complex linear viscoelastic material is set.

```
Material 99
  isotropic_viscoelastic_complex
  Kreal = function 1
  Kim = function 2
  Greal = function 3
  Gim = function 4
  density 1.0
End
```

**Input 4.4. Viscoelastic material specification**

4.4. **Properties**

4.4.1. **Density**

For solutions requiring a mass matrix, all material specifications must define density. This can be set via the keyword \texttt{density} followed by a scalar value. Alternatively, the density can be defined as a temperature dependent property (Section 4.4.6, as a spatially dependent property (Section 4.4.7.), or be defined via a file transfer to receive\_sierra\_data solution case with \texttt{density = from\_transfer}).

4.4.2. **High Cycle Fatigue**

Material parameters for high cycle fatigue (Section 3.13) may be provided to define the statistical failure behavior of materials. These properties are summarized in Table 4-88.
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Type</th>
<th>Default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fatigue_A1</td>
<td>Real</td>
<td>0.0</td>
<td>S-N curve constant</td>
</tr>
<tr>
<td>Fatigue_A2</td>
<td>Real</td>
<td>-3.0</td>
<td>S-N curve slope</td>
</tr>
<tr>
<td>Fatigue_A3</td>
<td>Real</td>
<td>0.0</td>
<td>S-N curve translation, $S_{eq} = (1 - R)^{A_3}$</td>
</tr>
<tr>
<td>Fatigue_A4</td>
<td>Real</td>
<td>0.0</td>
<td>S-N curve endurance limit. This parameter is not used in Sierra/SD fatigue computations.</td>
</tr>
<tr>
<td>Stress_Ratio</td>
<td>Real</td>
<td>-1.0</td>
<td>$R$, the ratio of max/min stress. The default -1 indicates oscillation about a zero-mean stress state</td>
</tr>
<tr>
<td>Fatigue_A</td>
<td>Real</td>
<td>1.0</td>
<td>S-N curve constant</td>
</tr>
<tr>
<td>Fatigue_m</td>
<td>Real</td>
<td>3.0</td>
<td>S-N curve coefficient</td>
</tr>
<tr>
<td>Fatigue_Stress</td>
<td>Real</td>
<td>1.0</td>
<td>Stress unit conversion factor</td>
</tr>
<tr>
<td>std_err</td>
<td>Real</td>
<td>0.0</td>
<td>to shift S-N curve by material uncertainty</td>
</tr>
<tr>
<td>t_dist</td>
<td>Real</td>
<td>0.0</td>
<td>to shift S-N curve by material uncertainty</td>
</tr>
</tbody>
</table>

Table 4-88. – Material Section Parameters for Fatigue Parameters.

4.4.3. S-N curve Definitions

There are two competing S-N curve definitions in literature, which are equivalent for S-N curves that are linear in log-log space, and are both supported by Sierra/SD. The first is used by Wirsching, Paez, and Ortiz in their book Random Vibrations Theory and Practice,\(^{65}\) and is supported with the Fatigue_A, and Fatigue_m parameters:

$$NS^m = A$$

The second is adaptable to a wider variety of materials, and is defined in chapter 9 of MMPDS\(^{20}\) (otherwise known as MIL-HDBK-5) as:

$$\log_{10}(N) = A_1 + A_2 \log_{10}( S (1 - R)^{A_3} - A_4 )$$

For real materials, $A_1$ is always positive, and $A_2$ is always negative. $A_2$ gives the S-N curve its negative slope, and $A_1$ represents the crossing of the S-axis on the S-N curve. For the
case of an S-N curve which is linear in log-log space, \( A_4 = 0 \) and the analysis is assumed to operate at a constant stress ratio such that:

\[
S_{eq} = S(1 - R)^{A_3} = S(2^{A_3})
\]

and

\[
\log_{10}(N) = A_1 + A_2 \log_{10}(S_{eq})
\]

These functions are equivalent, and the material properties can be mapped to each other by:

\[
A_1 = \log_{10}\left(\frac{A}{(1 - R)^{A_2 \cdot A_3}}\right), \quad A_2 = -m
\]

Note that \( A_4 \) is equivalent to the endurance limit of the material, if it exists. Parameters \( A_1, A_2, A_3 \), and \( A_4 \) can all be found in MIL-HDBK-5\textsuperscript{20} as empirically derived values for most metallic materials. See the example in Section 4.4.5. Since the Narrowband approach has an inherent assumption of zero mean stress, the default is \( R = -1 \). While the user can specify a different R-ratio, such usage would be inconsistent with the Narrowband and Wirsching methods.

### 4.4.4. S-N Curve Units

While Sierra/SD requires only a consistent set of units, the introduction of experimental data with their own units can confuse the solution. This is particularly challenging because the units for some parameters are mixed, and the data is gathered and presented in only a single unit system.

The stress scaling parameter helps reduce that problem. Consider the scaled equation for narrow band damage.

\[
D_{NB} = \frac{\nu^{+\tau}}{A} (\sqrt{2} \sigma_s F_{ss})^m \Gamma \left( \frac{m}{2} + 1 \right)
\]

Here \( F_{ss} \) is the stress scaling parameter. This parameter lets the user convert from the unit system of the analysis to the unit system of the test data. Table 4-89 provides common conversions of these stresses.

<table>
<thead>
<tr>
<th>Model Units</th>
<th>Experimental Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>PSI (lbs/in(^2))</td>
<td>1</td>
</tr>
<tr>
<td>Ksi (K lbs/in(^2))</td>
<td>1000</td>
</tr>
<tr>
<td>SI (N/m(^2))</td>
<td>.000145037738</td>
</tr>
<tr>
<td>CG (dynes/cm(^2))</td>
<td>.0000145037738</td>
</tr>
</tbody>
</table>

*Table 4-89.* Common Unit Scalings using **Fatigue Stress Scale.** The MIL-HNBK typically uses Ksi for experimental units.
4.4.5. Typical Material Data for Fatigue

Figure 4-25 shows typical fatigue data from MIL-HDBK-5. For a range of stress ratios in this material, the number of cycles to failure is represented by the equation,

$$\log_{10} N_n = 9.65 - 2.85 \log_{10}(S_{eq} - 61.3)$$

In this range, and for this material, we have the definitions given in Table 4.4.5.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
<th>Comment</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fatigue_A1</td>
<td>9.65</td>
<td>offset in S-N curve. $A_1 = \log (A)$</td>
</tr>
<tr>
<td>Fatigue_A2</td>
<td>-2.85</td>
<td>slope of S-N curve, $-m$</td>
</tr>
<tr>
<td>Fatigue_A3</td>
<td></td>
<td>S-N curve translation, $S_{eq} = (1 - R)^{A_3}$</td>
</tr>
<tr>
<td>Fatigue_A4</td>
<td>61.3</td>
<td>endurance limit</td>
</tr>
<tr>
<td>Stress_Ratio</td>
<td>-0.60</td>
<td>$R$, determines equivalent stress, $S_{eq} = (1 - R)^{A_3}$</td>
</tr>
<tr>
<td>Fatigue_Stress_Scale</td>
<td>0.001</td>
<td>Stress unit conversion to Ksi</td>
</tr>
</tbody>
</table>
Figure 4-25. Best-fit S/N curves for unnotched 4130 alloy steel sheet, normalized, longitudinal direction.

**Correlative Information for Figure 2.3.1.2.8(a)**

**Product Form:** Sheet, 0.075 inch thick

**Properties:** TUS, ksi  
117  
TYS, ksi  
99  
Temp., °F  
RT

**Specimen Details:** Unnotched  
2.88-3.00 inches gross width  
0.80-1.00 inch net width  
12.0 inch net section radius

**Surface Condition:** Electropolished

**Test Parameters:**  
Loading - Axial  
Frequency - 1100-1800 cpm  
Temperature - RT  
Environment - Air

**No. of Heats/Lots:** Not specified

**Equivalent Stress Equations:**

For stress ratios of -0.60 to +0.02  
\[ \log N_r = 9.65 - 2.85 \log (S_{eq} - 61.3) \]

\[ S_{eq} = S_{max} (1-R)^{0.41} \]

Std. Error of Estimate, Log (Life) = 0.21  
Standard Deviation, Log (Life) = 0.45  
\[ R^2 = 78\% \]

**Sample Size** = 23

For a stress ratio of -1.0  
\[ \log N_r = 9.27 - 3.57 \log (S_{max} - 43.3) \]

Note that material parameters depend on the unit system.
4.4.6. Temperature dependence

Material properties in *Sierra/SD* can be specified to be temperature dependent. Temperature dependent material properties are supported when temperatures are read in from an *Exodus* file, or when they are specified on a block-by-block basis. In the case of *Exodus* temperatures, the material properties would vary from element to element, since the temperatures vary with each element. The temperature dependent material properties are calculated from an elemental average of the *Exodus* temperatures. When temperatures are specified on a block-by-block basis, the temperature-dependence of the material properties can be specified explicitly in the input deck. If temperatures are specified in the *Exodus* file and block-by-block in the input deck, then the input deck values take precedence.

For linear elastic materials, an example of specifying temperature dependent properties is given below.

```
Material 1
  E function=eTempFunc1
  alphat .001
  tref 100
  nu 0.0
  density 7700.0
End

Material 2
  E function=eTempFunc2
  alphat .001
  tref 100
  nu 0.0
  density 7700.0
End

FUNCTION eTempFunc1
  type LINEAR
  data 0.0 4.0
  data 5.0e9 4.0
End

FUNCTION eTempFunc2
  type LINEAR
  data 0.0 3.0
  data 5.0e9 3.0
End
```

In this case, the elastic modulus of material 1 is specified by function eTempFunc1, and the
elastic modulus of material 2 is specified by function eTempFunc2. The moduli of each
element will be determined from its temperature and an interpolation on the function. In
this example, the functions are trivial, and thus the moduli of materials 1 and 2 will be 4
and 3, respectively. Note that the moduli, density and any of the 4 elastic constants $k$, $g$, $e$, $\nu$ can be specified as temperature dependent, and can be given different functions. In this
example, the Poisson’s ratio is constant and only the elastic modulus is temperature
dependent.

For viscoelastic materials, functions do not need to be specified in the material block to
designate temperature dependence of the shift factors. This is accounted for automatically.
See Section 4.3 on viscoelastic materials for more details.

In addition to linear elastic and linear viscoelastic material properties, the coefficient of
thermal expansion $\alpha$ may be given temperature-dependent material properties.

If the thermal_time_step keyword is used the temperature that effects material
properties will be based on the temperature read from the provided time step. Alternatively
the nUpdateDynamicMatrices keyword can be given which will update the material
stiffness based on the last read temperature. The last read temperature is controlled by the
nUpdateTemperature keyword. Updating the dynamic matrices is computationally
expensive and should be done only when temperature has changed significantly.

4.4.7. Spatially Variant Material Properties

Multiple methods exist to define element-to-element spatial dependence in material
properties.

Some material properties in Sierra/SD can be read on an element-by-element basis from
the Exodus mesh file. The syntax for this is parameter=exo_var scalar <string>. Here
<string> is the provided exodus field name. This must be a scalar field (one component)
defined on each element using the material.

For linear elastic materials, an example of specifying exodus based properties is given
below.

```
Material 1
    E = exo_var scalar e_input
    nu = exo_var scalar my_Poisson
    density = exo_var_scalar elem_density
End
```

In this case, the elastic modulus of material 1 is specified by the exodus mesh field
‘e_input’, the Poisson’s ratio by ‘my_Poisson’, and the density by ‘elem_density’. The
exodus based properties can be used to define complex spatial dependence of material
properties as may be found in partially compressed foams for example. No time-variance of
material properties is considered, if the input mesh exodus file has multiple time steps the
material property fields should be constant over time to avoid confusion.
Additionally it is possible to define material properties through the general function syntax. Syntax and requirements are detailed in table 2-26. Unless otherwise specified material property functions are evaluated as a function of temperature. The temperature can be defined via a block temperature int the input deck, an element-by-element temperature in the exodus input mesh, or a nodal temperature (which is then interpolated to the element centroid temperatures) in the exodus input mesh.

```
Material based_on_function
  E = function e_temperature_function
  nu = 0.3
  density = 1
End

Function e_temperature_function
  type linear
  data 0 1.0e+6
  data 400 1.0e+6
  data 500 0.9e+6
  data 900 0.3e+6
End
```

Alternatively specific input variables other than temperature can be explicitly defined in analytic functions. Nodal function responses are mapped to element material properties by evaluating the function at the nodes, and applying element shape functions to interpolate those values to the centroid.

```
Material reads_from_nodal
  E = function e_input_function
  nu = 0.3
  density = 1
End

Function e_input_function
  type analytic
  expression variable X = nodal some_mesh_var
  expression variable Y = nodal another_mesh_var
  evaluate expression "sqrt(X) + Y"
End
```

### 4.4.8. Specific Heat

Conversion of energy deposited in a structure to a change in temperature may be effected by a specific heat.

\[ Q = \rho V C \Delta T. \]  \hspace{1cm} (4.8)
Here \( Q \) is the total heat energy, \( \rho \) is the density, \( V \) is the volume, \( C \) is the specific heat and \( \Delta T \) is the change in temperature. It is up to the analyst to ensure that consistent units are employed. Note also that the analyst must determine under what conditions the specific heat is applied (constant pressure or constant volume).

Specific heat is used only in applying boundary conditions. Energy deposited within a structure is converted to temperature using equation 4.8. Once converted to temperatures, thermal stresses and temperature dependent material properties may be applied. A fatal error is encountered if the specific heat is not specified for each material containing an energy load. The keyword \texttt{defaultSpecificHeat} defined in the “parameters” section, can be used to specify a default specific heat for all materials.

```
Material 'Steel-SI'
  E=2e11 // Pa
  NU=0.28
  density=7850 // kg/m^3
  specific heat = 0.45 // J/(gK)
  tref = 300 // K
  alphat = 0.001
End
```

The reference temperature is used only for temperature dependent material properties, such as in viscoelastic materials. In other words,

\[
\Delta T = \frac{Q}{\rho V C}
\]

\[
T_{\text{elem}} = T_{\text{ref}} + \Delta T
\]

\[
\epsilon_{\text{thermal}} = \alpha T (T_{\text{elem}} - T_{\text{ref}}).
\]

Energy loads use the energy per unit mass or specific energy,

\[
\tilde{E} = \frac{Q}{\rho V},
\]

as described in Section 6.3.9.

### 4.4.9. Frequency dependence

For the \texttt{CJdamp} solution method (see Section 3.4), a frequency dependent damping coefficient, \( \eta(f) \), may be specified. All other solution methods will ignore this keyword. The \texttt{CJetaFunction} keyword requires as a parameter the identifier of a function. Its use is specified in the following example. See Section 2.8 for details in specifying the function. If no function is specified, the block will be treated as if the function were identically zero everywhere.

\( \eta \) is twice the normal modal damping coefficient. Thus, if \( \text{eta}=0.02 \) for all materials, the equivalent modal damping will be 1 percent.
The function specifies the frequency and amplitude pairs for \( \eta \). The frequencies are in Hertz. The CJdamp solution process interpolates the function at the eigen frequencies to determine the effective damping for any particular mode.

### 4.5. Piezoelectric Material

Sierra/SD supports two material models which possess voltage degrees of freedom in addition to displacements and rotations: dielectric and piezoelectric materials. The piezoelectric material is characterized by an electro-mechanical coupling in the stiffness matrix. In general, piezoelectric materials are defined by three constitutive tensors: a rank four orthotropic elasticity tensor, a rank two anisotropic permittivity tensor, and a rank three piezoelectric coupling tensor. See theory manual for further details on the constitutive tensors. Piezoelectric material tensors may be specified in a material block by including the keyword `orthotropic_piezoelectric` followed by the required three material tensor specifications. The nine parameters defining an orthotropic elasticity tensor are given by the keyword `Cij` followed by the upper triangle of a six by six matrix. The piezoelectric coupling tensor is given by the keyword `e_ij` followed by a six by three coupling matrix. Sierra/SD assumes the coupling matrix is in its stress-charge form (units charge/Area). The anisotropic permittivity tensor is provided by the keyword `permittivity_ij` followed by a three by three matrix. The permittivity matrix should be populated by absolute permittivity values (not normalized by the permittivity of free space).

Here is an example of a PZT5A piezoelectric material where \( e_0 \) is the permittivity of free space:

```plaintext
MATERIAL 1
   ORTHOTROPIC_PIEZOELECTRIC
```
Input 4.5. Piezoelectric Material

4.6. **Dielectric Material**

A dielectric material, the second available material possessing a voltage degree of freedom, is used to model the electro-static behavior of materials that do not exhibit electro-mechanical coupling (i.e. non-piezoelectric). Dielectrics can be generated with the keyword `dielectric`, and are defined with only its permittivity tensor. The following is an example of a dielectric input example.

```plaintext
Input 4.6. Dielectric Material

Material 1
  DIELECTRIC
  permittivity_ij = 916 * e0 0 0
                   0 916 * e0 0
                   0 0 830 * e0
End
```

Input 4.6. Dielectric Material
4.7.  Block

Each element block in the Exodus file must have a corresponding block section in the input file. The converse is not true — there can be block entries in the input deck that do not have corresponding entries in the Exodus file. There are two cases where this can happen:

- Virtual blocks. These are blocks that have entries in the input deck and are intended to be part of the model, but have no corresponding entries in the Exodus file. At this time, only Joint2g elements (see Section 5.21) can be defined to be virtual blocks.
- Extra blocks that have entries in the input deck but are not intended to be part of the model. These blocks are silently ignored by Sierra/SD.

It is an error to have multiple definitions for the same block. However, Sierra/SD does not report the error. The behavior of Sierra/SD in this case is not defined. This section contains information about the properties of the elements within the block.

4.7.1.  Block Parameters

There are two main types of block parameters:

1. Parameters common to most elements include:
   - Material property references are required for most elements. The material reference is of the form, material=material_id, where material_id is a string representing the material identifier (see Section 4).
   - coordinate frames - optional
   - nonlinear behavior - optional
   - block damping - optional
   - non-structural mass - optional

2. Element-specific parameters. These properties depend on the element type. A block parameter applies to all elements in the block. In contrast, a unique attribute may be specified for each element in a block.

Currently, four groups of elements have parameters. Shells have membrane and bending factor factors 5.7.5. The Infinite Element 6.1.9 and Perfectly Matched Layer 6.1.10 are configured using several unique block parameters. And the Beam2 5.9 and Nbeam 5.10 both have several parameters.

An example is provided in input 4.7. The block names can be provided in three forms as shown in the example. An integer number can be given, the number refers to the Exodus block id number in the mesh. Alternatively this number be provided as 'BLOCK_##' which is compatible with Sierra/SM syntax. Also, an option the block name can be given such as 'aft_cover_plate', again this is the name of the block in the specified in the
**Exodus** input mesh. Note that the material ID specified for BLOCK 32 uses an index (material 2), whereas BLOCK aft_cover_plate uses a specified material ID string “aluminum” These refer to materials defined by blocks “MATERIAL 32 ... END” and “MATERIAL aluminum ... END” respectively (see Sec. 4 for details).

```
BLOCK 32
  material 2
  tria3
  thickness 0.01
END

BLOCK aft_cover_plate
  material aluminum
END

BLOCK block_3
  coordinate 1
  spring
  Kx=1e6
  Ky=0
  Kz=0
  BlkBeta=0.0031
END

Material aluminum
  ....
END
```

**Input 4.7. Example Block input**

A list of the applicable attributes\(^{41}\) for different element types is shown in Table 4-90. Each element type is outlined in Section 5.
<table>
<thead>
<tr>
<th>Element Type</th>
<th>keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ConMass</td>
<td>1</td>
<td>Mass</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>Ixx</td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>Iyy</td>
</tr>
<tr>
<td></td>
<td>4</td>
<td>Izz</td>
</tr>
<tr>
<td></td>
<td>5</td>
<td>Ixy</td>
</tr>
<tr>
<td></td>
<td>6</td>
<td>Ixz</td>
</tr>
<tr>
<td></td>
<td>7</td>
<td>Iyz</td>
</tr>
<tr>
<td></td>
<td>8,9,10</td>
<td>offset</td>
</tr>
<tr>
<td>Beam</td>
<td>1</td>
<td>Area</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>I1</td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>I2</td>
</tr>
<tr>
<td></td>
<td>4</td>
<td>J</td>
</tr>
<tr>
<td></td>
<td>5,6,7</td>
<td>Orientation</td>
</tr>
<tr>
<td></td>
<td>8,9,10</td>
<td>offset</td>
</tr>
<tr>
<td>Spring</td>
<td>1</td>
<td>Kx</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>Ky</td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>Kz</td>
</tr>
<tr>
<td>Triangle</td>
<td>1</td>
<td>thickness</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>fiber orientation (theta)</td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>offset</td>
</tr>
<tr>
<td>Quad</td>
<td>1</td>
<td>thickness</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>fiber orientation (theta)</td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>offset</td>
</tr>
</tbody>
</table>
4.7.2. General Block Parameters

Parameters that are generally applicable to almost all blocks are listed in Table 4-91. More detailed descriptions are available in the following paragraphs.

4.7.2.1. Nonlinear Behavior  The nonlinear behavior of the block in nonlinear solutions is controlled by the nonlinear keyword. The global default for block-level nonlinear behavior is set in the parameters section (2.3). Within each block, we can override that default value. For example, to set a block to default to linear behavior, we would have the following BLOCK definition.

```plaintext
BLOCK 3
    nonlinear=no
    material 2
    tria3
    thickness 0.01
END
```

Similarly, to turn on the nonlinear behavior for the block, we would have,

```plaintext
BLOCK 3
    nonlinear=yes
    material 2
    tria3
    thickness 0.01
END
```

Note that these block-level nonlinear flags override the global nonlinear_default keyword that is set in the parameters section (2.3).

Some elements types are incapable of nonlinear behavior. This includes RBE3s, Rbars, SUPERELEMENTs, and Rrods. By default, use of these elements in nonlinear analysis will generate a fatal error. Use of the command nonlinear=no in these element blocks will enable overriding this fatal error and use pure linear behavior for these elements in the nonlinear analysis. Care should be taken with this option, pure linear elements have some incompatibilities in nonlinear analysis. For example use of pure-linear elements in nonlinear analysis can artificially constraint large rotations.

Limitations:
Linear element behavior in a nonlinear solution is limited to the linear range of the element. For example, rotations are stored incrementally in nonlinear solutions. This permits us to use geometrically nonlinear element formulations (such a corotational formulation). However, it limits the linear behavior in such solutions to rotations less than 360°.
### Table 4-91. General Block Parameters

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Values</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>nonlinear</td>
<td>yes/no</td>
<td>blockwise nonlinear behavior</td>
</tr>
<tr>
<td>material</td>
<td>string</td>
<td>material identifier</td>
</tr>
<tr>
<td>rotational_type</td>
<td>Eulerian or Lagrangian or none</td>
<td>blockwise behavior for rotational dynamics terms</td>
</tr>
<tr>
<td>coordinate</td>
<td>string</td>
<td>reference coordinate frame</td>
</tr>
<tr>
<td>blkalpha</td>
<td>Real</td>
<td>blockwise mass proportional damping</td>
</tr>
<tr>
<td>blkbeta</td>
<td>Real</td>
<td>blockwise stiffness proportional damping</td>
</tr>
<tr>
<td>nsm</td>
<td>Real</td>
<td>blockwise non-structural mass</td>
</tr>
<tr>
<td>density_scale_factor</td>
<td>Real</td>
<td>blockwise density scaling factor</td>
</tr>
<tr>
<td>stiffness_scale_factor</td>
<td>Real</td>
<td>blockwise stiffness scaling factor</td>
</tr>
<tr>
<td>T_current</td>
<td>Real</td>
<td>Temperature to set on every element of the block.</td>
</tr>
</tbody>
</table>

#### 4.7.2.2. Rotational Loading Matrices

For problems involving rotational loads, the `rotational_type` keyword allows the analyst to specify which type of rotational formulation to use for a given block of elements. The Eulerian formulation involves a fixed (non-rotating) coordinate system. The Lagrangian formulation attaches a rotating coordinate system to the block. If the None option is chosen, then rotational loads are ignored for this block. Thus, a structure with a rotating disk would only have the rotational terms applied to the spinning disk, and not the entire structure. The default is for the `rotational_type` keyword to be None.

#### 4.7.2.3. Coordinate Frame Reference

The reference coordinate system may be defined in a block. This definition applies to all the elements of the block and the associated materials. At this point, the coordinate system is only recognized for a subset of the elements (solid elements and springs). Further information on coordinate systems may be found in Section 2.7.

#### 4.7.2.4. Block Specific Damping

In Section 4.8, various methods of specifying the damping parameters for a model are identified. In addition to these methods, block specific damping parameters may be applied. These apply a stiffness (or mass) proportional damping matrix on an element by element basis within the block. Thus, if a model is made of steel and foam, one could apply a 5% stiffness proportional damping term to the foam, but leave the steel undamped.

There is no physical justification for proportional damping, and there is no expectation that it will accurately represent damping mechanisms in a structure. However, it is easy to apply, and there are cases where proportional damping may reveal a need for more
accurate damping models. As with all damping models, the effects depend on the solution type. For example, both Statics and Eigen analysis ignore the damping matrix.

The damping matrix generated from block specific damping is defined as follows.

\[
D = \sum_{i}^{nblks} \alpha_i M_i + \beta_i K_i
\]

(4.12)

Where \(D\) is the real system damping matrix, and \(\alpha_i\) and \(\beta_i\) are the proportional mass and damping coefficients for block \(i\). These coefficients are completely analogous to the system level coefficients described in Section 4.8. The damping contributions from these block parameters are always added to the other contributions.

Block specific damping is applied using the \texttt{blkalpha} and \texttt{blkbeta} parameters. Block proportional damping generates a damping matrix that would couple modal based solutions. It is not currently available in modal solutions such as \texttt{modaltransient}. Also, see Section 4.4.9 for material modal like damping.

4.7.2.5. Non-Structural Mass  
Non-structural mass (NSM) is specified per element block in the input deck. It is added to the internal mass of the element. One reason to add a NSM is to use a gravity load to simulate an external load. Another reason is to stabilize solutions with mass-less nodes. The units depend on the element type as defined in Table 4-92.

4.7.2.6. Non-Structural Mass Corner Cases  
'Three Dimensional' is determined from the mesh topology. HexShell elements are three-dimensional. Layered shell elements add non-structural mass once per meshed element, not once per layer. Some conditions can cause non-structural mass to be silently ignored. These cases do not trigger a warning, and are shown in Table 4-93.

The following is an example of how to use non-structural mass in the input file:

```plaintext
//nsm specified in pounds per square inch
BLOCK 3
    material 2
    tria3
    thickness 0.01
    nsm 0.005
END

MATERIAL 2
    density 0.5
END
```
Table 4-92. – Non-Structural Mass Units.

<table>
<thead>
<tr>
<th>Element Type</th>
<th>Units</th>
<th>Example</th>
</tr>
</thead>
<tbody>
<tr>
<td>ConMass</td>
<td>Mass Per Element</td>
<td>lbs</td>
</tr>
<tr>
<td>Spring or Joint2g</td>
<td>Mass Per Element</td>
<td>lbs</td>
</tr>
<tr>
<td>Beam or Truss</td>
<td>mass/length</td>
<td>lbs / in</td>
</tr>
<tr>
<td>Two Dimensional</td>
<td>mass/area</td>
<td>lbs / sq-in</td>
</tr>
<tr>
<td>Three Dimensional</td>
<td>mass/volume</td>
<td>lbs / cu-in</td>
</tr>
</tbody>
</table>

Table 4-93. – Unhandled Corner Cases.

<table>
<thead>
<tr>
<th>Element Type</th>
<th>State</th>
<th>Result</th>
</tr>
</thead>
<tbody>
<tr>
<td>Beam or Truss</td>
<td>Area=0</td>
<td>NSM Silently Ignored</td>
</tr>
<tr>
<td>Two Dimensional</td>
<td>Thickness=0</td>
<td>NSM Silently Ignored</td>
</tr>
</tbody>
</table>

4.7.2.7. Blockwise Density Scaling An element block may define a scale factor to be applied to the density of the material. This can be used to calibrate the exact mass in each block and account for discretization errors, or to reuse materials that only differ in density. The interaction of this feature with non-structural mass is documented in Table 4-94.

The following is an example of blockwise density scaling in the input file:

```plaintext
BLOCK 3
   material 2
   density_scale_factor = 1.0025
END

MATERIAL 2
   density 0.5
END
```

Table 4-94. – Combining NSM with Density_Scale_Factor.

<table>
<thead>
<tr>
<th>Element Type</th>
<th>Mass Per Element</th>
</tr>
</thead>
<tbody>
<tr>
<td>ConMass</td>
<td>NSM + Mass</td>
</tr>
<tr>
<td>Spring or Joint2g</td>
<td>NSM</td>
</tr>
<tr>
<td>Beam or Truss</td>
<td>NSM<em>Length + Density</em>Scale*Volume</td>
</tr>
<tr>
<td>Two Dimensional</td>
<td>NSM<em>Area + Density</em>Scale*Volume</td>
</tr>
<tr>
<td>Three Dimensional</td>
<td>NSM<em>Volume + Density</em>Scale*Volume</td>
</tr>
</tbody>
</table>
4.7.2.8. Blockwise Stiffness Scaling  An element block may define a scale factor to be applied to the linear stiffness of the material, but the capability has been limited to isotropic material models only. This can be used to tune the precise stiffness of components without requiring separate material definitions, usually when adapting a model to match test results. The `stiffness_scale_factor` is applied to isotropic materials consistently, even if the Young’s modulus is not defined explicitly by the user. Any valid combination of material constants will still be valid.

Note that `stiffness_scale_factor` is ignored on blocks without a material, such as spring elements, and attempting to use `stiffness_scale_factor` with any anisotropic material will result in an error.

The following is an example of blockwise stiffness scaling in the input file:

```plaintext
BLOCK 3
    material 2
        stiffness_scale_factor = 1.0025
    END

MATERIAL 2
    isotropic
    E 1e7
    nu 0.3
    density 0.5
    END
```

4.7.2.9. Piezoelectric Material Damping  Only stiffness (\(\text{blkbeta}\)) and mass (\(\text{blkalpha}\)) proportional damping can be applied to electro-mechanical materials, and damping models can only be specified at the block level. Global damping is prohibited on any model containing piezoelectric or dielectric materials. Voltage degrees of freedom do not couple with mass or damping (see theory manual). Hence, a piezoelectric element’s damping matrix, defined by stiffness and(or) mass proportional damping, is zero at all voltage degrees of freedom. The effects of mechanical damping will only impact the voltage degree of freedom responses due to the electro-mechanical stiffness coupling.

4.8.  Damping

This section allows input of simple global viscous damping models, using either modal damping rates or stiffness and mass proportional damping. The various options for the DAMPING section are shown in Table 4-95.

The damping matrix or modal damping coefficient is determined by summing contributions from all damping parameters given in Table 4-95. For modal superposition-based transient
Table 4-95. – DAMPING Section Options.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>alpha</td>
<td>mass proportional damping parameter (real)</td>
</tr>
<tr>
<td>beta</td>
<td>stiffness proportional damping parameter (real)</td>
</tr>
<tr>
<td>gamma</td>
<td>uniform modal damping ratio (fraction of critical) applied to all modes (real)</td>
</tr>
<tr>
<td>mode</td>
<td>ADDITIONAL modal damping ratio applied to individual mode(fraction of critical) (integer, real)</td>
</tr>
<tr>
<td>ratiofun</td>
<td>index of function to define modal damping ratios</td>
</tr>
<tr>
<td>FilterRbm</td>
<td>remove rigid body mode contribution to damping</td>
</tr>
<tr>
<td>maxRatioFlexibleRbm</td>
<td>controls check for 6 RBM with FilterRbm</td>
</tr>
</tbody>
</table>

Analysis, **modal transient**, all the given parameters are defined. For linear direct implicit transient analysis, the modal damping parameters apply only to modes for which eigenvalues and eigenvectors have previously been computed. This depends on the presence of the keyword **nmodes** in the solution section of the input file. In the case of a **modalranvib** (or **ModalFrf** analysis in the case of complex modes), modal damping is available, but the proportional damping parameters **alpha** and **beta** are currently ignored.

The effect of the mass and stiffness proportional parameters on modal damping depends on the frequencies of the modes. For modal-based analysis, the damping rate for mode \( i \) with radial frequency \( \omega_i \) is given as

\[
\zeta_i = \frac{\alpha}{2\omega_i} + \frac{\beta}{2} + \Gamma + \text{mode}(i) + \text{ratiofun}(i),
\]

where the viscous damping term in the modal equilibrium equation is \( 2\zeta_i \omega_i \). For example, the following damping input section could be used in a modal transient analysis.

<table>
<thead>
<tr>
<th>DAMPING</th>
</tr>
</thead>
<tbody>
<tr>
<td>alpha 0.001     //</td>
</tr>
<tr>
<td>beta 0.00005    // C = .001 * M + .00005 * K</td>
</tr>
<tr>
<td>gamma 0.005     // 0.5% critical</td>
</tr>
<tr>
<td>mode 1 0.01     // gamma+mode_1 = 1.5% of critical</td>
</tr>
<tr>
<td>mode 2 0.005    // gamma+mode_2 = 1.0% of critical</td>
</tr>
<tr>
<td>mode 3 0.015    // gamma+mode_3 = 2.0% of critical</td>
</tr>
<tr>
<td>END</td>
</tr>
</tbody>
</table>

It produces the following damping ratios.

\(^2\)Use of block specific proportional damping is explained in Section 4.7.2.
In direct transient analysis, the full mass and stiffness matrices are integrated for the solution. Specification of a modal damping method triggers construction of a damping contribution from the previous modal solution. This contribution is combined with other damping terms such as the proportional damping. Thus, the same damping input section would produce the damping ratios shown above for the first three modes. Modal damping is applied to modes computed in a previous solution case.

The `ratiofun` keyword permits definition of modal damping terms based on a frequency dependent function. The associated function definition (see Section 2.8) provides a table look up for damping ratios. For example, consider a system with modes at 200 and 500 Hz. The following example will establish modal damping ratios of .03 and .06 respectively. The function describes a line defined by $\text{ratio}(f) = 0.01 + 0.1/1000f$.

```
DAMPING
  ratiofun=100
END

FUNCTION 100
  type=linear
  data 0 0.01
  data 1000 0.11
END
```

The `FilterRbm` keyword permits proportional damping without damping the rigid body response. Thus, mass proportional damping can be used with no impact on the rigid body response. The theory behind this method of damping is described in subsection Damping of Flexible Modes Only section Solution Procedures of the Theory Manual.

In order for this method of damping to work properly, the structure must have the conventional six rigid body modes of three translations and three rotations. A check of this condition is made inside of `Sierra/SD`, and a fatal error results if this condition is not satisfied. Specifically, the condition is met if

$$
\text{RatioFlexibleRbm} = \frac{\| K\Phi_r \|_2}{\| K_d \|_\infty \| \Phi_r \|_2} \leq \epsilon
$$

(4.13)

---

3i.e. non-modal based, but linear transient
4A previous modal solution case must have been specified to use modal damping, otherwise `Sierra/SD` will warn the user and abort.
where $K$ is the stiffness matrix, $\Phi_r$ is the matrix of six rigid body modes, and $\|K_d\|_\infty$ is the largest entry on the diagonal of $K$. The scalar tolerance $\epsilon$ can be specified using the `maxRatioFlexibleRbm` keyword.

```plaintext
DAMPING
  alpha=0.1
  FilterRbm
  maxRatioFlexibleRbm=0.001 // default is 1e-10
END
```

### 4.8.1. Nonlinear transient solutions with damping

Using the stiffness proportional damping parameter $\beta$ in a `NLtransient` analysis will generate damping terms using the initial (or linear) stiffness matrix. The tangent stiffness matrix is not used. This reason is that the tangent matrix would be required to compute the damping terms at each iteration.

> Nonlinear solutions do not support standard modal damping.

While nonlinear solutions do not currently support standard modal damping, they may be damping using the Distributed Damping method of the next section (4.8.2). Like modal damping, this is a system level damping model.

### 4.8.2. Nonlinear Distributed Damping using Modal Masing Formulation

The purpose of this formulation is to implement a subsystem or system level nonlinear distributed damping model into Sierra/SD. The theory on this method is found in the Sierra/SD Theory Manual.\(^{41}\) Distributed damping is a method developed to model the nonlinear damping response of a subsystem. It implements the damping in a nonlinear manner with the use of an internal force term. The damping is modeled by either an Iwan model or a linear damper, and distributed to the subsystem by a modal expansion. This method augments the internal force vector through a modal Masing formulation.\(^2\)

Previous to the nonlinear transient solution which computes the distributed damping, eigenvectors must be computed. This is done in a previous solution 'case' option using "eigen" methods.

The damping section is used to define the type of damping behavior. Currently, only two types of damping behavior are defined: a linear damper, `damper`, and an Iwan model, `Iwan`.\(^41\) Each mode will have a keyword defined after it with an associated parameter

\(^2\)Masing and Iwan models are used almost interchangeably in this document. Iwan models are a subset of more general Masing models.
number. The parameters are used to define the damping behavior. If nothing is specified for a mode, then no damping for that mode is defined. An example input is shown below.

```
SOLUTION
  case 'eig'
    eigen
      nmodes 16
      shift -1e5
  case 'nonlinear'
    NLtransient
      nsteps = 200
      time_step = 5.0e-3
      rho = 0.8
  END

DAMPING
  mode 1 damper 1
  mode 2 damper 2
  mode 3 damper 2
  mode 4 damper 2
  mode 5 damper 2
  mode 6 damper 2
  mode 7 Iwan 4
  mode 8 Iwan 4
  mode 9 Iwan 4
  mode 10 Iwan 3
  mode 11 Iwan 3
  mode 12 Iwan 3
  mode 13 Iwan 3
  mode 14 Iwan 3
  mode 15 Iwan 3
  mode 16 Iwan 3
  END

Property 1
  Mu = 0.001
  K = 0
  end

Property 2
  Mu = 0.02
  K = 0
  end
```
Property 3
  chi = -0.82139
  phi_max = 1.0325e-04
  R = 7.608594e+06
  S = 5.616950e+06
end

Property 4
  chi = -0.82139
  phi_max = 1.0325e-04
  R = 7.608594e+06
  S = 5.616950e+06
end
5. Element Library

Short descriptions of each of the types of elements follow. Materials are described in the materials section. Some elements have attributes. An attribute may be specified in the Exodus file per element or the input deck for per block or both. Input deck values override values from the Exodus file. Attributes are described in this section. An attribute is either required or optional.

5.1. Hex8

The Hex8 is a standard 8 node hexahedron with three degrees of freedom per node. The Hex8 element has 8 integration points. The shape functions are trilinear. It supports isotropic and anisotropic materials.

There are three variations of Hex8. The default element is a bubble hex element, which can be specified by Hex8b, or by no specification at all. The bubble element still has 8 nodes and 3 degrees of freedom per node, and thus from a user’s perspective it is indistinguishable from the standard Hex8. The Hex8b element uses bubble functions to augment the standard element shape functions. It bends more accurately than the hex8.

The Hex8u specifies an under integrated Hex with properties similar to those of most commercial finite element codes. There are two versions of this element. The first is a standard under integrated element, and the second is a mean quadrature element with selective deviatoric control. In both cases, the under integration produces an element that is soft relative to a fully integrated element. Both elements are specified using the keyword Hex8u, along with the parameter sd_factor. If sd_factor is specified, then the mean quadrature element with selective deviatoric is invoked with the value of sd_factor specified. If the parameter sd_factor is not specified, the standard under integrated element is invoked. For example, to use the under integrated element, one could specify

```
block
  hex8u
  material 1
end
```

On the other hand, the following block would use the mean quadrature element with a selective deviatoric parameter of 0.9

```
block
  hex8u
  material 1
  sd_factor 0.9
end
```
Note that \texttt{sd\_factor} must be between 0 and 1. With a value of 0, the element is a mean quadrature element. With a value of 1, the element is again mean quadrature, but with fully integrated deviatoric component. More information is given in subsection Isoparametric Solid Elements. Selective Integration section \texttt{Sierra/SD Elements of the Theory Manual.}

The fully integrated Hex is specified by \texttt{Hex8F}. While it performs adequately when the element shape is nearly cubic, it performs poorly for larger aspect ratios. For most problems involving bending the Hex8b is recommended.

The only \textit{required} parameter for these elements is the material specification. Any material may be applied.

For computational acoustics, see Section 2.4.4.

\textbf{5.2. Hex20}

The 20 node variety of Hex element provides quadratic shape functions. It is a far better element than the Hex8, and should be used if possible. The Hex20 element in \texttt{Sierra/SD} is similar to elements found in most commercial codes. A material specification is required, and any structural material may be used.

Shape Function and Gauss point locations for the Hex20 are described in table 7-126, and in subsubsection Shape Functions and Gauss Points subsection Quadratic Isoparametric Solid Elements section \texttt{Sierra/SD Elements of the Theory Manual.}

The stress may be output at the Gauss points as described in Section 7.1.24.

\textbf{5.3. Wedge6}

The Wedge6 is a compatibility element for the Hex8, it is not recommended that the entire mesh be built of \texttt{Wedge6} elements. They are primarily intended for applications where triangles are naturally generated in mesh generation. A material specification is required, and any structural material may be used.

\textbf{5.4. Wedge15}

The Wedge15 element adds mid-side nodes to the Wedge6. Like the Hex20 and Tet10, it has quadratic shape functions, and is recommended. A material specification is required, and any structural material may be used.
5.5. **Tet4**

This is a standard 4 node tetrahedral element with three degrees of freedom per node. The **Tet4** element has one integration point. The shape functions are linear. It is not recommended to use only Tet4 elements for the entire mesh because standard, linear tetrahedron are typically much too stiff for structural applications. The **Tet4** is provided primarily for those applications where a mesh may be partially filled with these elements. If a model is constructed of all tetrahedral elements (as by an automatic mesh generator), the **Tet10** is strongly recommended over the **Tet4**.

A material specification is required, and any structural material may be used.

5.6. **Tet10**

This is a standard 10 node tetrahedral element with three degrees of freedom per node. The **Tet10** uses 4-point integration for the stiffness matrix and 16-point integration for the mass matrix. The shape functions are quadratic. This element is recommended for use in most structural analyses.

A material specification is required, and any structural material may be used.

5.7. **Two-Dimensional Shell and Membrane Elements**

**Sierra/SD** supports a variety of topologically 2D elements that capture shell or membrane behavior. A specific 2D element formulation can be selected by explicitly specifying the element type in the input block with keywords such as **quadt**, **quadm**, **nquad**, etc. For three noded triangles if no specific element formulation is given then by default one of the Tria3 (isotropic materials) or TriaShell (orthotropic or layered materials) shell formulations will be used. For all other topologies (4-node quad, 8-node quad, and 6-node triangle) the default element is composed of sub-triangles using the Tria3 or TriaShell shell formulations as described in Section 5.7.1

5.7.1. **QuadT, Quad8T, and Tria6**

The 4-node quad **QuadT**, 8-node quad **Quad8T**, and 6-node triangle **Tria6** are all internally composed of sub-triangles. Each of these elements is the **Sierra/SD** default formulation for its respective topology. These elements have both membrane and bending stiffness. The element stiffness and mass matrices are derived by composing internally generated triangle elements, as illustrated in Figure 5-26, 5-27, and 5-28. Output quantities such as stress are the average stress over the sub-triangles. Composing elements from low-order triangles, though not optimal, is adequate for most applications.
Figure 5-26. – QuadT Element.
The element is generated by internally combining two Triangle elements.

The sub-triangles may be based on either the Tria3, or on the TriaShell element depending on the material properties. The Tria3 is used for isotropic, single-layer elements. More complex materials require use of the TriaShell. The underlying triangle formulation is determined automatically by Sierra/SD, and cannot be selected by the user. See the description of the Tria3 and TriaShell for details of the formulations of the triangle elements that compose the QuadT, Quad8T, and Tria6 elements.

Table 5-96 lists the supported inputs.

Sierra/SD example input files that use this element can be found in

Salinas_rtest/patchtests/quadt/quadt-patch8_test
Salinas_rtest/patchtests/quadt/quadt-patch9_test
5.7.2. \textbf{QuadM}

\textbf{QuadM} is a 4-node quadrilateral membrane element. It has only membrane stiffness and no rotational degrees of freedom. Membranes are well suited to structures with very low bending stiffness, such as fabric. Use of shell elements for such low-bending-stiffness structures can generate a problematic near-singularity.

In the input deck a \texttt{block} section indicating \texttt{QuadM} is required.

For two-dimensional problems, the \texttt{QuadM} reduces to the standard plane elasticity element. For three-dimensional problems, it behaves like the plane elasticity element in the plane, and like a stretched balloon out-of-plane. The out-of-plane stiffness for the element comes from the preload. Without a preload, the out-of-plane stiffness vanishes and the element can be singular. The out-of-plane behavior results from an additional stiffness term that is applied to the out-of-plane degrees of freedom. The stiffness resembles the stiffness associated with Laplace’s equation. This additional stiffness is derived in classical textbooks.\textsuperscript{31}

\begin{table}[h]
\centering
\begin{tabular}{ll}
\hline
Keyword & Description \\
\hline
thickness & Thickness of the shell \\
offset & offset of the shell midplane: see Section 5.7.9 \\
material & Linear elastic material used by element \\
layer & Layer properties: see Section 5.7.8 \\
coordinate & Base global coordinate system: see Section 5.7.7 \\
rotate about axis & Coordinate system rotation: see Section 5.7.7 \\
rotate about normal & Coordinate system rotation: see Section 5.7.7 \\
membrane_factor & Stiffness scale factor: see Section 5.7.6 \\
bending_factor & Stiffness scale factor: see Section 5.7.6 \\
\hline
\end{tabular}
\caption{QuadT, Quad8T, Tria6 Inputs.}
\end{table}
Table 5-97 lists the supported inputs.

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>thickness</td>
<td>Thickness of the membrane, required</td>
</tr>
<tr>
<td>sd_factor</td>
<td>Selective deviatoric parameter used for numerical integration</td>
</tr>
<tr>
<td>material</td>
<td>Linear elastic material used by element, required</td>
</tr>
<tr>
<td>coordinate</td>
<td>Base global coordinate system: see Section 5.7.7</td>
</tr>
<tr>
<td>rotate about axis</td>
<td>Coordinate system rotation: see Section 5.7.7</td>
</tr>
<tr>
<td>rotate about normal</td>
<td>Coordinate system rotation: see Section 5.7.7</td>
</tr>
<tr>
<td>membrane_factor</td>
<td>Stiffness scale factor: see Section 5.7.6</td>
</tr>
</tbody>
</table>

Table 5-97. – QuadM inputs.

Both full and selective integration methods are available for the membrane. The full integration is the default. Selective deviatoric integration can be specified by using the parameter `sd_factor`. For example, for full integrated membrane, one would specify

```plaintext
block
  QuadM
  material 1
  thickness 0.1
end
```

On the other hand, the following block would use the mean quadrature element with a selective deviatoric parameter of 0.9

```plaintext
block
  QuadM
  material 1
  sd_factor 0.9
  thickness 0.1
end
```

Note that `sd_factor` must be between 0 and 1. With a value of 0, the element is a mean quadrature element. With a value of 1, the element is again mean quadrature, but with fully integrated deviatoric component. More details on the theory behind these elements is given in the theory manual.

This element could be used in any situation where a preload is applied to the elements before the analysis of interest (e.g., a static preload followed by eigen analysis), or even in cases where no preload is applied but the membranes are sufficiently constrained (such as a hex element with a layer of membrane elements on the surface).

The QuadM element can be used in coupled simulations. In these cases, Adagio performs the preload calculation, and the preload information is passed to Sierra/SD for later analysis. In these cases the element is non-singular due to the preload calculation.
5.7.2.1. Known Issues

- The membrane element does not currently compute stress, strain, strain energy, or strain energy density outputs. All these outputs will be reported as zero for the element.

5.7.3. Nquad/Ntria

The Nquad and Ntria elements are isoparametric shells with membrane and bending stiffness. They are shear-deformable elements with six degrees of freedom (DOF) per node which support isotropic, orthotropic, and layered materials. The formulation of the Nquad/Ntria is generated by decoupling the membrane and bending DOF. These elements currently only have linear behavior implemented. If using a non-linear solution method, these elements will not calculate a true internal force, but a linear force.

The Nquad/Ntria isotropic stiffness matrix is based on the plane elasticity and shear deformable (Mindlin) formulations as outlined in Reddy.\(^{38}\) The layered shell stiffness matrix uses a composite laminate formulation.\(^{35}\)

In the input deck a block definition indicating either Nquad or Ntria is required. The block definition must also have a material keyword referencing the isotropic material properties (Section 4) or orthotropic layer properties (Section 4.1.2) with properties \(E_1, E_2, \nu_{12},\) and \(G_{12}\). An example element block for a single layer isotropic material is shown below:

```
block 2
  Nquad
  thickness 0.1
  material 2
end
block 3
  Ntria
  thickness 0.4
  material 4
end
```

Inputs are given in Table 5-98.

The stabilization method from Belytschko\(^9\) is used for the Nquad element. Using single-point integration \(K_{s}^{[1x1]}\) for the shear stiffness matrix leads to hourglass modes for some problems. Using full integration \(K_{s}^{[2x2]}\) can cause shear locking in some problems. Belytschko recommends a shear stiffness matrix that is a linear combination of the reduced integration and full integration shear stiffness matrices,

\[
K_{s} = (1 - \varepsilon)K_{s}^{[1x1]} + \varepsilon K_{s}^{[2x2]}.
\]

The fraction, \(\varepsilon = rt^2/A\) is a function of thickness and area. Here \(r = 0.03\), \(t\) is the element thickness and \(A\) is the area of the shell. This automatic selection of \(\varepsilon\) works well for thin
plates, but can be a problem for thicker elements; $\varepsilon$ should never exceed 1. To limit shear locking, the fraction may be capped using \texttt{nquad_eps_max}, as shown in the example below.

```
block 1
  nquad
  thickness 1
  nquad_eps_max 0.1
end
```

The value for $\varepsilon$ is adjusted using the function $\hat{\varepsilon} = \frac{\varepsilon_{\text{max}}}{\sqrt{1+\varepsilon^4}}$. This is done to address problems with “elbow functions” in the code. Figure 5-29 shows this function for nquad_eps_max = 1.

![Figure 5-29. - Function for nquad_eps_max.](image-url)

5.7.3.1. Known Issues

- The Ntria with orthotropic materials sets $G_{23}$ and $G_{13}$ to the input value of $G_{12}$.
5.7.4. **TriaShell**

The **TriaShell** is a 3-noded triangular element with 6 degrees of freedom (DOF) per node. The formulation of the **TriaShell** is generated by decoupling the membrane DOF and the bending DOF. Allman’s Triangular (AT) element\(^2\) models the membrane DOF, while the Discrete Kirchhoff Triangle\(^8\) (DKT) models the bending DOF. These two elements are combined into the **TriaShell** element.

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>thickness</td>
<td>Thickness of the shell</td>
</tr>
<tr>
<td>offset</td>
<td>Offset of the shell midplane: see Section 5.7.9</td>
</tr>
<tr>
<td>material</td>
<td>Linear elastic material used by single layer element</td>
</tr>
<tr>
<td>layer</td>
<td>Layer properties: see Section 5.7.8</td>
</tr>
<tr>
<td>coordinate</td>
<td>Base global coordinate system: see Section 5.7.7</td>
</tr>
<tr>
<td>rotate about axis</td>
<td>Coordinate system rotation: see Section 5.7.7</td>
</tr>
<tr>
<td>rotate about normal</td>
<td>Coordinate system rotation: see Section 5.7.7</td>
</tr>
<tr>
<td>membrane_factor</td>
<td>Stiffness scale factor: see Section 5.7.6</td>
</tr>
<tr>
<td>bending_factor</td>
<td>Stiffness scale factor: see Section 5.7.6</td>
</tr>
</tbody>
</table>

**Table 5-99.** – TriaShell input options.

In general, the **Tria3** element is preferred to the **TriaShell** because it is less prone to shear locking behavior, and it is computationally efficient. The TriaShell element is required for orthotropic or layered materials.

- TriaShells support orthotropic or anisotropic materials.
- TriaShells support layered materials. Note however that mass lumping is not allowed with layered TriaShell elements.

5.7.5. **Tria3**

The **Tria3** is a three-dimensional triangular shell with membrane and bending stiffness. There are 6 degrees of freedom per node. In most respects it is similar to the **TriaShell**. It is the default element for triangular meshes. The **Tria3** was provided by Carlos Felippa of CU Boulder. It only supports only isotropic single-layer materials.

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>thickness</td>
<td>Thickness of the shell, required</td>
</tr>
<tr>
<td>offset</td>
<td>Offset of the shell midplane see Section 5.7.9</td>
</tr>
<tr>
<td>material</td>
<td>must be isotropic, required</td>
</tr>
<tr>
<td>membrane_factor</td>
<td>Stiffness scale factor, see Section 5.7.6</td>
</tr>
<tr>
<td>bending_factor</td>
<td>Stiffness scale factor, see Section 5.7.6</td>
</tr>
</tbody>
</table>
An example element block is shown below.

```
block 3
  Tria3
  Thickness 0.01
  material 71
  membrane_factor=0 // turns off membrane stiffness
end
```

### 5.7.6. Stiffness Scaling

The stiffness of the 2D element bending and membrane responses are computed independently and can be independently arbitrary scaled using the commands `membrane_factor` and `bending_factor`. Each of these parameters default to 1.0. Reasons for scaling the element stiffness could include accounting for damage to the structure or tuning of model response to match experimental data.

### 5.7.7. Shell Coordinate Systems

For orthotropic materials a coordinate system must be defined in the element to set the local material orientation. First, a user-defined global coordinate system is referenced with the `coordinate` keyword (Section 2.7.) The global coordinate system is evaluated at the element centroid to define a local \( \hat{r}, \hat{s}, \hat{t} \) system for the element.

This \( \hat{r}, \hat{s}, \hat{t} \) coordinate system is projected onto the surface of the shell as shown in Figure 5-30. This projection will rotate the coordinate system such that the \( \hat{t}_p \) axis aligns with the shell normal and \( \hat{r}_p \) and \( \hat{s}_p \) are as close as possible to the original orientations of \( \hat{r} \) and \( \hat{s} \). It is recommended to use a global coordinate system in which \( \hat{t} \) lies as close as possible to the shell normal vectors to minimize ambiguity in this coordinate system alignment step.

Two additional inputs are available in the shell block to alter the element local coordinate system. First a rotation can be applied to the \( \hat{r}, \hat{s}, \hat{t} \) vectors prior to projection onto the shell plane with the command:

```
rotate <real> about axis <int>
```

The rotation angle is given in degrees. The axis integer 1, 2, or 3, represents the \( \hat{r}, \hat{s}, \hat{t} \) coordinate axes. After rotation of the coordinate system the \( \hat{r}', \hat{s}', \hat{t}' \) system is aligned to the plane of the shell. See Figure 5-31 for an example.

A second input option can rotate the the element local coordinate system after projection with the command:

```
rotate <real> about normal
```
Figure 5-30. – Projection of global coordinate system to shell.

Figure 5-31. – Rotation of global coordinate system about axis prior to projection to shell.
The angle is given in degrees. This will rotate the projected system around the \( \hat{t}_p \) vector as shown in Figure 5-32.

Once a shell-local system is defined, the \( \hat{r}_p \) and \( \hat{s}_p \) vectors will define the orientation of orthotropic materials. The fiber_orientation command can do one more local rotation of this orientation layer-by-layer as described in Section 5.7.8.

5.7.8. Layered Shells

Several of the shell element formulations allow composing the element via a stack of layers of different materials. This is used to model layered composites. When using layers, the available materials are isotropic and orthotropic_layer. Each layer must specify a material and thickness. A fiber orientation, which is a rotation of the layer material coordinate system with respect to the element coordinate system, may optionally be given. Thickness and fiber orientation for a multi-layer material must be specified layer by layer in the input deck. Exodus attributes may not be used.

Keyword layer defines a new layer for the current shell. Layers of the shell are stacked from the bottom to the top based on the order of the layer keyword in the input deck. The layer_ID input is an identifier provided by the user and is not used to select stacking order. A shell may have up to 250 different layers defined. Figure 5-33 shows a simple schematic explaining how layers are stacked in Sierra/SD. An example element block for a four-layer orthotropic layered shell is shown below.  

\[\text{For layered shells, the thickness parameter specifies the actual thickness of that layer of the shell. This is in contrast to the HexShell which specifies a relative thickness: see Section 5.8.}\]
An important parameter for the layered shells with orthotropic materials is the specification of a user-defined coordinate system with the `coordinate` option, see Section 5.7.7. In the example shown here, a coordinate system is defined for the shell block and orthotropic material properties are defined via a fiber orientation rotation with respect to that base element system.

```
block 2
  TriaShell
  coordinate 1
  layer <layer_ID>
    material 1
    thickness 0.02
    fiber orientation 40
  layer <layer_ID>
    material 2
    thickness 0.04
    fiber orientation 44
  layer <layer_ID>
    material 3
    thickness 0.03
    fiber orientation 54
  layer <layer_ID>
    material 4
    thickness 0.01
    fiber orientation 4
```
Input 5.1. Layered Shell Example

Stress output for shells with more than one layer can be written to an Exodus file or can be obtained from the result file by specifying stress in the echo section. The layer stresses will be computed only at the midpoint of each layer. Thus, layer stresses at the top and bottom of each layer are not supported.

5.7.8.1. Known Issues

- The Navy Layered Shell with orthotropic materials sets $G_{23}$ and $G_{13}$ to the input value of $G_{12}$.

5.7.9. Offset Shells

By default, the meshed shell element lies at the midplane of the material volume represented by the shell. Shells may be offset from the midplane by specifying the offset input. The offset vector is the element unit normal vector scaled by the offset. An example is shown in Figure 5-34.

The resulting mass and stiffness properties are equivalent to the stiffness generated by translating the shell by the offset vector, and constraining the resulting offset nodes to the untranslated nodes using rigid links. The performance of offset shells is better than that of the constraint approach. Note that for curved surfaces there may be modeling issues with offset elements since there is no change in curvature with the change in radius.

In the .inp file the element offset is specified as,

```
offset=-3.14e-2
```

Offsets may also be specified in the Exodus file via attributes (see Section 5.7.10.) Some limitations of element offsets are described in Section 5.7.9.1
5.7.9.1. Offset Shells and Lumped Mass
Offset elements can provide more accurate modeling of some structures, such as shells cladded on top of a volume. Offset elements necessarily involve coupling between the rotational and translational degrees of freedom. This results in off-diagonal coupling terms in the element stiffness and mass matrices.

Generally, the element stiffness matrix is fully populated and seldom is reduced. However, the mass matrix may be lumped (diagonalized) as described in Section 2.4.4.

Lumping the mass matrix uncouples the translational and rotational degrees of freedom, which is inaccurate for offset shells. Specifically, while the total mass is conserved, the center of gravity and mass moments are not. The lumped mass looks as if it had not been offset. This is true even with mesh refinement. The models of the consistent and lumped mass are fundamentally different when element offsets are included. Mass lumping with offset shell elements is discouraged.

5.7.10. Spatially Dependent Properties via Exodus Attributes

Certain 2D element properties can be defined either in the input deck or via attributes on the input Exodus mesh. When a property is defined in the input deck it has a constant value for all elements of the block. The advantage of Exodus attributes is that a different value can be used in each element. This can be used to model properties such as variations in thickness in tapered shells.

The supported attributes are shown in Table 5-100.
If an input deck value is given for a shell property, such as thickness, it will override any value given in the Exodus attributes. Attributes cannot be used with multi-layer shells. For Multi-layer shells all the element properties must be defined in input deck.

5.8. HexShell

The 8 noded HexShell is a hybrid solid/shell element. It is meshed as a standard hex element, but the formulation of the element is similar to that of a shell. Unlike a shell element, the thickness is determined by the mesh. But, the element is designed to operate with many of the same features as shell elements even when it becomes thin. Details of the element formulation are available in a separate report. An introduction to HexShells is readily available in, and the verification manual discusses the results of the verification problems from for Sierra/SD.

The HexShell has a preferential thickness direction that must be set correctly. There are three ways to specify the thickness direction.

1. Using the tcoord, it may be specified by a coordinate frame.

2. An Exodus side set may be attached to one face of all the elements in a block using the keyword sideset. The thickness direction will be defined to be the normal to the sideset’s surface. For example, if the sideset is placed on a side of the structure that lies on the x-y plane, then the thickness direction of the HexShell will be defined as the z direction, since that is the normal to the x-y plane.

3. Sierra/SD may attempt to determine the thickness direction from the topology. This is the default option (because it is the easiest for the user), but it is also the least robust.

Sierra/SD attempts to identify the element orientation first using tcoord. The tcoord keyword abbreviates thickness coordinate, and is only defined for HexShells. If tcoord is not specified, then Sierra/SD attempts to identify the element orientation second from the corresponding sideset. These methods do not depend on the decomposition, but the third method does depend on the decomposition. Lastly if no sideset is specified, Sierra/SD attempts to determine the thickness direction from the topology.
The element orientation may be identified in the output using the `eorient` keyword. See Section 7.1.43.

**Thickness Determination by Topology**

When the element thickness must be determined by the topology, the mesh must follow these requirements. The elements in the block must form a sheet. More than one disconnected portion of the sheet is possible, but all portions must adhere to these requirements.

- Every element in the sheet must have at least two neighbors, e.g. the sheet can’t be a single element. NOTE... at this time, this is true for the parallel decomposed mesh too. The portions of the sheets found in each subdomain can not be a single element. We must be able to eliminate the thickness direction of each element by its neighbor connectivity.

- The elements in the sheet may vary in thickness, but the sheet must be exactly one element thick.

- The elements must be connected as a single sheet. Thus, if the sheet turns a corner, it must do so gently. The algorithm will fail if any element in the sheet is connected on the top or bottom to another element in the sheet.

Determining element thickness from the topology has known limitations, and is not planned for an update. This topology method is the oldest and depends on the body being laid out in a layer one element thick. Unfortunately, it is not well parallelized, as we do not have ghost elements. The other two methods do not depend on the decomposition.

**HexShell Parameters**

The `HexShell` requires a material specification. Optional parameters include the sideset or the coordinate frame and coordinate direction used to determine the thickness direction. The sideset keyword must be associated with a defined sideset in the model. The `tcoord` keyword requires a string and integer argument. The first is the Name of the coordinate system referenced. The second is the direction (1, 2 or 3) associated with the coordinate system.

<table>
<thead>
<tr>
<th>#</th>
<th>Keyword</th>
<th>Arguments</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>sideset</td>
<td>ID/name</td>
<td>sideset to specify thickness direction</td>
</tr>
<tr>
<td>2</td>
<td>tcoord</td>
<td>ID/name and direction</td>
<td>coordinate frame and coordinate direction</td>
</tr>
<tr>
<td>3</td>
<td>autolayer</td>
<td># of layers and material</td>
<td>creates specified number of uniform layers of specified material</td>
</tr>
</tbody>
</table>

An example specification for a multi-layer HexShell is shown in input 5.2.
Input 5.2. Example HexShell Input. Block 88 is multi-layer input with thickness direction determined using a sideset. Block 89 defines the thickness direction using a coordinate frame and the tcoord keyword.

HexShell Multilayers

The formulation of the HexShell supports multiple layers of orthotropic materials. Each layer has an associated material, normalized thickness and coordinate. The coordinate is provided to permit specification of the material coordinate. The thickness specifies the relative thickness of each layer. The total thickness is determined from the element.
topology, but relative thicknesses for each layer must be specified. If only one layer is specified, then the layer keyword is not required, and the relative thickness is irrelevant (and not required).

There are two methods to specify multiple layers in a HexShell. The first, illustrated in input 5.2, provides complete flexibility over the material specification, orientation and thickness of each layer. An autolayer capability provides are much more limited specification that is useful for models of a single material with temperature dependence across the thickness. This autolayer feature creates the specified number of layers, of uniform thickness, of a single specified material.

Materials for all HexShell specifications can be defined as a function of temperature, with the temperatures defined through the exodus file as element variables. The temperature can vary over both the elements and layers in the block.

```plaintext
block 1
  HexShell
  sideset 1
  autolayer 4
  material 1
end

Material 1
  name "steel"
  E function=1
  nu .3
  density 0.288
end

FUNCTION 1
  type Linear
  data 0 30e6
  data 1e6 20e6
END
```

Input 5.3. HexShell Autolayer Example. Here, exodus element variables define the temperature for each element on the block. Exodus layers must be of uniform thickness, and must be labeled “layer_temp1”, “layer_temp2”, etc.

When using temperature dependent materials, the temperature is obtained from the exodus file. The modulus is calculated as a function of temperature, and used in the modulus.

---

4Layers for HexShells must specify the relative thickness of the layer. This is in contrast to layered shells which specify the absolute thickness (Section 5.7.8).
element stiffness formulation. The temperature can vary both with layers and with elements. Any of the material parameters in either an isotropic or orthotropic material can be set to be temperature dependent. In the case of an isotropic material, any pair of two of the properties \( G, K, E, \) or \( \nu \) can be temperature dependent.

The number of layers in the input file does not need to match the number of layers in the exodus file. The temperatures in the exodus file will be interpolated piecewise linearly to the center of the layer in the input file.

Temperature dependent orthotropic materials are supported for HexShells only. Temperature dependent densities are also supported.

<table>
<thead>
<tr>
<th>Feature</th>
<th>Analytic Reference</th>
<th>Verification Section</th>
<th>Tested</th>
<th>Parallel Test</th>
<th>User Test</th>
</tr>
</thead>
<tbody>
<tr>
<td>general</td>
<td>yes†</td>
<td>43</td>
<td>Y</td>
<td>Y</td>
<td>some</td>
</tr>
<tr>
<td>multiple layers</td>
<td>no†</td>
<td>43</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

†Felippa’s report contains some verification. It has not been carried into Sierra/SD.

**Table 5-101.** – HexShell Verification Summary.

The mass properties of a layered HexShell are computed approximately as follows.

1. The volume fraction, \( f_i \), and density, \( \rho_i \), of each layer is determined.
2. The contribution of the mass of the element is added to the nodes as if an element of density \( \bar{\rho} = \sum_i \rho_i f_i \) filled the entire element.

The net affect of this is that the mass is computed as if an average density were applied. This could introduce minor errors if the element is thick and is much denser on one side than another.

For a HexShell if using \texttt{tcoord}, it is important to remember that the material definition may also use a non-default coordinate frame. In the next example, the thickness coordinate, \texttt{tcoord}, and the material definition use the same coordinate system.

```plaintext
begin cylindrical coordinate system 1000
  origin 0.0 0.0 0.0
  z point 1.0 0.0 0.0
  xz point 0.0 1.0 0.0
end
block 13
  HexShell
  tcoord 1000 1
  material 8
  coordinate 1000
end
```
5.9. **Beam2**

The **Beam2** element formulation is described in Section 3.14 of. This element is similar to the standard NASTRAN CBAR element, but it does not include a definition for a product of inertia or area shear factors. A product of inertia and area shear factors are included in the CBAR element in NASTRAN and are supported by the **Nbeam** element described in Section 5.10.

The use of a **Beam2** element requires a block definition with a **beam2**. The block definition must also have a material keyword referencing an isotropic material. Finally, the **Beam2** element must have a defined set of geometric parameters. Attributes may be defined in the input deck as follows.

```plaintext
block block_id
  Beam2
  material = material_id
  Area = area
  I1 = inertia_about_1
  I2 = inertia_about_2
  J = polar_moment_inertia
  orientation = x_orient y_orient z_orient
  offset = x_offset y_offset z_offset
end
```

The keywords are defined in following sections.

We define the local coordinate system before defining the attributes. The local coordinate system is defined by three axes – \( x_{elem}, y_{elem}, \) and \( z_{elem} \). The \( x_{elem} \)-axis is the axial direction. It lies along the length of the beam. The other two axes, the \( y_{elem} \)-axis and \( z_{elem} \)-axis, are determined by the orientation vector, \( V \). The local coordinate system and the orientation vector are shown in Figure 5-35. The orientation vector \( V \) lies in the plane defined by the \( x_{elem} \)-axis and the \( y_{elem} \)-axis – plane 1 in Figure 5-35. The \( z_{elem} \)-axis has the same orientation as \( x_{elem} \times V \). Finally the \( y_{elem} \)-axis has the same orientation as \( z_{elem} \times x_{elem} \).

The \( x_{elem} \)-axis and \( z_{elem} \)-axis define plane 2 in Figure 5-35. The \( y_{elem} \)-axis, which lies in the 1-plane, corresponds to a local 1-axis defined in a cross-sectional plane, a plane normal to the \( x_{elem} \)-axis. The \( z_{elem} \)-axis, which lies in the 2-plane, corresponds to a local 2-axis defined in the cross-sectional plane.

The **Beam2** element attributes are complicated. Four attributes are always required. A cross-sectional area, **area**, must be defined. The cross-sectional area can be defined with an AREA keyword. Two bending moments of inertia are also required. A bending moment of inertia for the 1-plane (bending about the \( z_{elem} \)-axis) is defined by the **I1** keyword. Bending moments in the 2-plane (or bending about the \( y_{elem} \)-axis) is defined using the **I2** keyword. \( I_1 \) and \( I_2 \) are the bending moments in their corresponding planes. \( I_1 \) and \( I_2 \) are
not bending about their axes. This convention is consistent with many commercial codes including NASTRAN. A polar moment of inertia, \( \text{polar\_moment\_inertia} \), for torsion about the \( x_{\text{elem}} \)-axis is required. The polar moment of inertia can be defined with the \( J \) keyword.

If the cross-section has the symmetry, \( I_1 = I_2 \), then the orientation vector \( V \) may be an optional attribute. Otherwise, \( V \) is a required attribute. It is necessary for the bending properties to have the correct global orientation. The components of the orientation vector can be specified with the values \( x_{\text{orient}}, y_{\text{orient}}, \) and \( z_{\text{orient}} \) using an \text{ORIENT} keyword.

For attributes are provided in the \text{Exodus} mesh file, if an offset vector will be set through the attributes, then the orientation vector is always required.

The origin of the 1,2 coordinate system at the beam endpoints is the corresponding grid points by default. A user specified offset vector \( V_{\text{off}} \) translates the geometric location of the coordinate system origin. This offset vector is shown in Figure 5-36. For the \text{Beam2} element, one offset vector translates both ends of the beam. The \text{OFFSET} keyword is optional. The offset vectors move the beam neutral axis (the \( x_{\text{elem}} \)-axis) off the line that passes between the two grid points defining the connectivity of the beam. An offset is defined by a vector with values \( x_{\text{offset}}, y_{\text{offset}}, \) and \( z_{\text{offset}} \). These values are associated with an \text{OFFSET} keyword.

When the offset option is used, the offset stiffness properties are equivalent to the stiffness generated by translating the beam by the offset direction and constraining the resulting
offset nodes back to the untranslated nodes using rigid links. In addition, the offset mass properties are equivalent to the mass generated by translating the beam by the offset direction and constraining the resulting offset nodes back to the untranslated nodes using rigid links. For the Beam2 element, only the component of the offset vector orthogonal to the element is used to compute the offset behavior for both the stiffness and mass.

For curved surfaces it is possible for the offset element to be inaccurate. The reason is that the radius changes, but the curvature does not change. Refer to section 5.7.9.1 for limitations of element offsets.

The block parameters area, inertia_about_1, inertia_about_2, polar_moment_inertia, x_orient, y_orient, z_orient, x_offset, y_offset, and z_offset, may also be defined as attributes in the mesh file. Use the mesh file to specify attributes per element and the input block to specify attributes per element block. Input deck attributes override mesh file attributes. Attributes in the mesh file must be in the order specified in Table 5-102.

The Beam2 element supports isotropic materials only.

The following section illustrates the use of the Beam2 keyword in an element block definition. The element block has an integer block identifier of 3. This element block must consist of two node elements.

```
block 3
  Beam2
  Material 7
```
Table 5-102. – Attributes for Beam2.

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Area</td>
</tr>
<tr>
<td>2</td>
<td>I1</td>
</tr>
<tr>
<td>3</td>
<td>I2</td>
</tr>
<tr>
<td>4</td>
<td>J</td>
</tr>
<tr>
<td>5,6,7</td>
<td>Orientation</td>
</tr>
<tr>
<td>8,9,10</td>
<td>offset</td>
</tr>
</tbody>
</table>

Area 0.71
I1 .05
I2 5e-2
J 0.994
Orientation 1.0 -1.0 0.9
Offset -3.14e-2 0.11 0.99
end

5.10. Nbeam

The Nbeam element was developed from the COSMIC/NASTRAN open source CBAR element. Unlike the Beam2 element discussed in the previous section, the Nbeam element includes a definition for a product of inertia and definitions for area shear factors. The Nbeam element, currently, only has linear behavior implemented. If using a nonlinear solution method, the Nbeam element will not calculate a true internal force, but a linear force.

The use of a Nbeam element requires a block definition with a Nbeam keyword. The block definition must also have a material keyword referencing an isotropic material. Finally, the Nbeam element must have a defined set of geometric parameters. Most parameters for the Nbeam element may be entered either as attributes in the mesh file or through keywords in the block definition. Some parameters can be reset from default values only by use of the keyword definitions in the block definition. The general form of the block definition is as follows:

block block_id
    Nbeam
    material = material_id
    Area = area
    I1 = inertia_about_1
    I2 = inertia_about_2
    J = polar_moment_inertia
I12 = product_inertia_12
Shear_factor_1 = sfactor1
Shear_factor_2 = sfactor2
orientation = x_orient y_orient z_orient
offset = x_offset y_offset z_offset
end

The various keywords in the above block definition are described in following paragraphs.

**Local coordinate frame:** Before describing the parameters for the Nbeam element, it is necessary to define the local coordinate system that is set up for beam elements in general. The local coordinate system is defined by three axes – \( \mathbf{x}_{\text{elem}} \), \( \mathbf{y}_{\text{elem}} \), and \( \mathbf{z}_{\text{elem}} \). The \( \mathbf{x}_{\text{elem}} \)-axis lies along the length of the offset beam. The other two axes, the \( \mathbf{y}_{\text{elem}} \)-axis and \( \mathbf{z}_{\text{elem}} \)-axis, are determined by an orientation vector, \( \mathbf{V} \). The local coordinate system and the orientation vector are shown in Figure 5-37. The orientation vector \( \mathbf{V} \) lies in the plane defined by the \( \mathbf{x}_{\text{elem}} \)-axis and the \( \mathbf{y}_{\text{elem}} \)-axis – plane 1 in Figure 5-35. The \( \mathbf{z}_{\text{elem}} \)-axis is derived from the orientation vector \( \mathbf{V} \) and the \( \mathbf{x}_{\text{elem}} \)-axis by taking the cross-product \( \mathbf{x}_{\text{elem}} \times \mathbf{V} \). Once the \( \mathbf{z}_{\text{elem}} \)-axis is calculated, the cross-product \( \mathbf{z}_{\text{elem}} \times \mathbf{x}_{\text{elem}} \) gives the \( \mathbf{y}_{\text{elem}} \)-axis. As the Nbeam supports arbitrary vector offsets at each end, the orientation of the offset beam may differ from the orientation of the geometry (see “offset” below) that is not offset.

![Figure 5-37. Nbeam Orientation, Offset and Local Coordinate System. The coordinate system is in the plane of the offset beam. The plane is defined by the offset beam and the orientation vector, \( \mathbf{V} \).](image)

The \( \mathbf{x}_{\text{elem}} \)-axis and \( \mathbf{z}_{\text{elem}} \)-axis define plane 2 in Figure 5-35. The \( \mathbf{y}_{\text{elem}} \)-axis, which lies in the 1-plane, corresponds to a local 1-axis defined in a cross-sectional plane, a
plane normal to the $x_{elem}$-axis. The $z_{elem}$-axis, which lies in the 2-plane, corresponds to a local 2-axis defined in the cross-sectional plane.

**Area:** The cross-sectional area, $area$, must be defined either as exodus attributes or in the “block” section. The cross-sectional area can be defined with an **AREA** keyword.

**Bending Moments:** The bending moments of inertia about orientation axes must be defined either in the exodus file, or the “block” section. A bending moment of inertia about the 1-axis (the local cross-sectional axis corresponding to the $y_{elem}$-axis), $\text{inertia\_about\_1}$, can be defined with the $I_1$ keyword. A bending moment of inertia about the 2-axis (the local cross-sectional axis corresponding to the $z_{elem}$-axis), $\text{inertia\_about\_2}$, can be defined with the $I_2$ keyword. Finally, a polar moment of inertia, $\text{polar\_moment\_inertia}$, for torsion about the $x_{elem}$-axis is required. The polar moment of inertia can be defined with the $J$ keyword.

The **Nbeam** element supports a product of inertia specification. The product of inertia about the 1,2-axes, $\text{product\_inertia\_12}$, is specified with the keyword $I_{12}$. If the $I_{12}$ keyword does not appear, the value for $\text{product\_inertia\_12}$ defaults to zero.

**Shear Factor:** The **Nbeam** element also has two area shear factor specifications. An area shear factor is a constant by which an average shearing strain on a beam cross-section must be multiplied in order to obtain the same transverse shear displacement as the transverse shear displacement that will be obtained from the actual shear strain distribution for the cross-section. Typically, the shearing strain varies over a cross-section. See Oden (Ref.36) for a discussion of shear factors. An area shear factor for shear in the 1-direction, $\text{sfactor\_1}$, is specified with a **Shear\_factor\_1** keyword. If no **Shear\_factor\_1** keyword appears, the value for $\text{sfactor\_1}$ defaults to 1.0. An area shear factor for shear in the 2-direction, $\text{sfactor\_2}$, is specified with a **Shear\_factor\_2** keyword. If no **Shear\_factor\_2** keyword appears, the value for $\text{sfactor\_2}$ defaults to 1.0.

**Orientation:** The orientation vector $V$ must be specified to assure that the bending properties of the beam have the correct global orientation relative to the rest of the structure. The components of the orientation vector can be specified with the values $x_{orient}$, $y_{orient}$, and $z_{orient}$ using an **Orientation** keyword.

**Offset:** A user may specify offsets exactly as described Section 5.9, and shown in Figure 5-36. For the **Nbeam** element, the same offset vector is applied to both ends of the beam. The **OFFSET** keyword is optional. The offset vectors move the beam neutral axis (the $x_{elem}$-axis) off the line that passes between the two grid points defining the connectivity of the beam. An offset is defined by a vector with values $x_{offset}$, $y_{offset}$, and $z_{offset}$. These values are associated with an **OFFSET** keyword.

When the offset option is used, the offset stiffness properties are equivalent to the stiffness generated by translating the beam by the offset direction and constraining the resulting offset nodes back to the untranslated nodes using rigid links. For the **Nbeam** element, the full offset vector is used to compute the offset behavior, and
Table 5-103. – Attributes and Parameters for Nbeam.

<table>
<thead>
<tr>
<th>#</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Area</td>
<td>Area of beam</td>
</tr>
<tr>
<td>2</td>
<td>I1</td>
<td>First bending moment</td>
</tr>
<tr>
<td>3</td>
<td>I2</td>
<td>Second bending moment</td>
</tr>
<tr>
<td>4</td>
<td>J</td>
<td>Torsion moment</td>
</tr>
<tr>
<td>5,6,7</td>
<td>Orientation</td>
<td>orientation vector</td>
</tr>
<tr>
<td>8,9,10</td>
<td>offset</td>
<td>beam offset vector</td>
</tr>
<tr>
<td>11,12,13</td>
<td>–</td>
<td>offset of second node</td>
</tr>
<tr>
<td>–</td>
<td>I12</td>
<td>product of inertia</td>
</tr>
<tr>
<td>–</td>
<td>Shear_factor_1</td>
<td>shear factor 1-direction</td>
</tr>
<tr>
<td>–</td>
<td>Shear_factor_2</td>
<td>shear factor 2-direction</td>
</tr>
</tbody>
</table>

different offsets may be applied at each end. (This behavior is different from the Beam2 element, in which only the component of the offset vector orthogonal to the element is used to compute the offset behavior).

Note that for curved surfaces there may be modeling issues with offset elements, since there is no change in curvature with the change in radius. Refer to Section 5.7.9.1 for limitations of element offsets.

Many of the parameters described can also be defined as attributes in the mesh file. Attributes in the mesh file must be in the order specified in Table 5-103. If an attribute is entered in both the mesh file and the input file, the value in the input file will supersede the value in the mesh file.

The Nbeam element is restricted to isotropic materials. No stress or strain output is available for Nbeam elements.

The following section illustrates the use of the Nbeam keyword in an element block definition. The element block has an integer block identifier of 3. This element block must consist of two node elements.

```
block 3
  Nbeam
  Material 7
  Area 1.92
  I1 2.57375
  I2 4.81277
  J 0.025816
  I12 -1.45983
  Shear_factor_1 0.44021
  Shear_factor_2 0.33313
  Orientation 1.0 0.0 0.0
```
5.11. **TiBeam**

This is a Timoshenko beam with consistent mass. For beams with an aspect ration of 100 there is little difference between the beam2 and the tibeam. Beams with an aspect ratio of 10 differ significantly. The tibeam uses the parameters of a the **Beam2**. In addition the effective area in shear should be provided (the default is 2/3). For a beam with square cross section,

```plaintext
block 3
tibeam
  Material 1
  // Shear_factor = 10(1+nu)/(12+11 nu)
  areay = .84967320261438
  areaz = .84967320261438
end
```

In this example, the area, $I_1$, $I_2$, $J$ and the orientation are all set in the Exodus file. Recall that for a circular cross section, the approximate shear factor is given in terms of Poisson’s ratio, $\nu$, by $6(1+\nu)/(7+6\nu)$. At this time, although the parser warns that `areay` and `areaz` are ignored, they are parsed and applied.

5.12. **Truss**

This is the definition for a **Truss** element based on Cook (Ref.17). Trusses have stiffness in extension only. The **Truss** has 1 attribute as shown in the table. A linear elastic, isotropic material is required.

<table>
<thead>
<tr>
<th>#</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Area</td>
<td>Area of truss</td>
</tr>
</tbody>
</table>

No stress or strain output is available for trusses.

5.13. **Ftruss**

The **Ftruss** is a truss element whose stiffness is a function of the truss length.
Trusses have stiffness in only the axial direction. While they exist in a 3 dimensional world, forces orthogonal to the axial direction result in no resistance, i.e. they are singular. The axial force for a \textbf{Ftruss} element is defined as,

\[
\vec{F}(\vec{L}_n, t) = -K(|\vec{L}_n|, t_n) \vec{L}_n
\]  

(5.1)

where \( \vec{L}_n \) is the vector from the first point to the second at time \( t_n \). Note that \(|\vec{L}_n|\) is the instantaneous length of the truss. The force is \textit{always} in the direction of the instantaneous element.

Denote by \( K_o \) the stiffness of a standard truss, and by \( L_o \) the nominal truss length. \( F = -K_o dx \) implies that \( K = \frac{K_o dx}{L_o + dx} \). The definition in equation 5.1 was chosen so that a force may be applied when \( dx \) vanishes.

If a standard (non user) function is used, the stiffness is a function of truss extension only. It may not be a function of both extension and time.

Input to the \textbf{Ftruss} element is similar to that for the truss element. The attributes and parameters are listed in table 5-104, and a demonstration example is provided below.

```
block 88
  Ftruss
  function 88
  scale 1.0
    material 17  //optional material
    area 0.01    //area required if material defined
  end
end
```

If the material keyword is not found, no mass matrix is generated for the element. If a material is found, then \textit{area} must also be defined. Like a standard truss, \textit{area} is the first \textbf{Exodus} attribute. The area and material properties are used only to compute the mass properties of the element, and may be omitted.

The \textit{scale} term can be defined using the input file, or alternatively, it may be defined using the second \textbf{Exodus} attribute.

\footnote{Recall that attributes are ordered data that may be specified in the \textbf{Exodus} file, providing a variable which changes with each element. Parameters may be specified in the input file, and are applied uniformly to all elements in the block.}
Table 5-104. – Ftruss Attributes and Parameters.

<table>
<thead>
<tr>
<th>#</th>
<th>Name</th>
<th>Type</th>
<th>Default</th>
<th>Comment</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Area</td>
<td>Real</td>
<td>0</td>
<td>required if a material is specified.</td>
</tr>
<tr>
<td>2</td>
<td>Scale</td>
<td>Real</td>
<td>1</td>
<td>multiplier for the function</td>
</tr>
<tr>
<td></td>
<td>Function</td>
<td>string</td>
<td>required</td>
<td>function identifier (see section 2.8)</td>
</tr>
<tr>
<td></td>
<td>Material</td>
<td>string</td>
<td>optional</td>
<td>If the material specification is provided, it must point to a valid material (sec. 4), and an area must also be provided.</td>
</tr>
</tbody>
</table>

5.14. **ConMass**

Concentrated masses are used to apply a known amount of mass at a point location. The *Exodus* file element topology is a sphere. Support for concentrated masses as two noded elements in *exodus* has been deprecated.

Parameters for the **ConMass** are listed below. Because of difficulties in translation or generation of the model, the parameters found in the *Exodus* file are not normally used for a **ConMass**. This avoids the confusion generated when mass constant defaults may have been taken from beams for example. As a result, all parameters must be specified in the input or the analysis will fail.

This behavior can be tedious however, if many concentrated masses are found in the model, and if the analyst is confident that the attributes are appropriate for these elements. In this case, use the **ConMass** element. It is identical to the ConMass, but uses the default attributes from the *Exodus* file. Typically seven attributes would be specified there.

A concentrated mass must have a mass. If no inertia tensor is specified, then the concentrated mass has 3 degrees of freedom, displacements. On the other hand a concentrated mass with an inertia tensor has both displacement and rotational degrees of freedom.

<table>
<thead>
<tr>
<th>#</th>
<th>keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Mass</td>
<td>concentrated mass</td>
</tr>
<tr>
<td>2</td>
<td>Ixx</td>
<td>$xx$ moment of inertia</td>
</tr>
<tr>
<td>3</td>
<td>Iyy</td>
<td>$yy$ moment of inertia</td>
</tr>
<tr>
<td>4</td>
<td>Izz</td>
<td>$zz$ moment of inertia</td>
</tr>
<tr>
<td>5</td>
<td>Ixy</td>
<td>$xy$ moment of inertia</td>
</tr>
<tr>
<td>6</td>
<td>Ixz</td>
<td>$xz$ moment of inertia</td>
</tr>
<tr>
<td>7</td>
<td>Iyz</td>
<td>$yz$ moment of inertia</td>
</tr>
<tr>
<td>8,9,10</td>
<td>offset</td>
<td>offset from node to CG</td>
</tr>
</tbody>
</table>

As an example element block,
The **ConMass** moments of inertia are defined at the location of the **ConMass**. The offset can be used to specify inertial terms about a different point.

A **ConMass** element will activate either 3 or 6 degrees of freedom on the node the mass is located. Every **ConMass** element will activate 'DispX', 'DispY', and 'DispZ'. A **ConMass** element with non-zero inertial terms or an offset will activate 'RotX', 'RotY', and 'RotZ'. In a case such as a spring-mass system where only one translational degree of freedom is desired, the mass should be constrained in the other directions. If **ConMass** elements are attached to solid elements, through shared nodes or a 2D element, either the inertial terms should be set to zero or the rotational degrees of freedom should be constrained. Failing to properly constrain the **ConMass** may result in a solver out of bounds error or incorrect results.

### 5.15. **Spring**

The **Spring** element provides a simple spring connection between two nodes in a model. Note that the direction of application of the spring should be parallel to a vector connecting the nodes of the spring. It is usually preferable to have the nodes of the spring be coincident. Springs are defined in the **Exodus** database using **Beam** or **Bar** elements.

The **Spring** element has three required parameters (translational spring stiffness). Rotational parameters are supported using the **RSpring** element described in section 5.16. Currently, there is no way to attach off-diagonal elements, i.e. there is no $K_{xy}$ spring element. If that is required, a combination of a spring and a multi-point constraint must be used.

Springs can be defined in user defined coordinate systems.

<table>
<thead>
<tr>
<th>#</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Kx</td>
<td>translational spring constant in $X$</td>
</tr>
<tr>
<td>2</td>
<td>Ky</td>
<td>translational spring constant in $Y$</td>
</tr>
<tr>
<td>3</td>
<td>Kz</td>
<td>translational spring constant in $Z$</td>
</tr>
</tbody>
</table>

As an example element block,
5.15.1. Spring Parameter Values

It is strongly recommended that all three values of the spring constants be nonzero. This is especially important in parallel analysis performed using domain decomposition. Many domain decomposition tools may partition the model such that zero spring constants lead to singular domain stiffness matrices. This is true even if other elements may eliminate the singularity.

While setting nonzero spring stiffness helps to avoid solver problems, underlying domain decomposition problems may still exist for parallel calculations. Domain decomposition tools employ heuristics for connection of springs to solids; the models are not compatible. Finite length springs often result in constraints on rigid body modes.\(^6\) Springs fill an important analysis need, but analysts may find that in many cases it may be better to replace the spring elements by solid element meshes which more accurately represent the physical connection. While there are more degrees of freedom in the calculation, the accuracy is enhanced, and domain decomposition problems are mitigated.

5.16. RSpring

The RSpring element provides a simple rotational spring connection between two nodes in a model. It is usually preferable to have the nodes of the spring be coincident. RSprings are defined in the Exodus database using Beam or Bar elements.

The RSpring element has three required parameters (rotational spring stiffness). It is strongly recommended that all three components have some stiffness. This is particularly important when doing parallel analysis (see the discussion in section 5.15.1). Translational stiffness require the use of the Spring element described in Section 5.15. Currently, there is no way to attach off diagonal elements, i.e. there is no \(K_{xy}\) spring element. If that is required, a combination of an RSpring and a multi-point constraint must be used.

RSprings can be defined in user defined coordinate systems. The relevant parameters are listed in the table.

---

\(^6\)This is not specific to parallel solutions. Most often, finite length springs introduce strain for a model rotation.
5.17. **Spring3 - nonlinear cubic spring**

The **Spring3** element provides a nonlinear spring connection between nodes in a model. Note that the direction of application of the spring should be parallel to a vector connecting the nodes of the spring. It is usually preferable to have the nodes of the spring be coincident. Springs are defined in the **Exodus** database using **Beam** or **Bar** elements.

The **Spring3** element has nine required parameters (translational spring stiffness). There is no way to attach off diagonal elements, i.e. there are no $K_{xy}$ spring elements. If that is required, a combination of a spring and a multi-point constraint must be used.

The force applied by the **Spring3** is defined as a cubic polynomial in each of the coordinate directions. Thus,

$$ F_x = Kx1 \cdot u_x + Kx2 \cdot u_x^2 + Kx3 \cdot u_x^3 $$  \hspace{1cm} (5.2)

For linear analyses, only the first term is used.

Cubic springs may be defined in user defined coordinate system.

<table>
<thead>
<tr>
<th>#</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Kx1</td>
<td>translational linear spring constant in $X$</td>
</tr>
<tr>
<td>2</td>
<td>Ky1</td>
<td>translational linear spring constant in $Y$</td>
</tr>
<tr>
<td>3</td>
<td>Kz1</td>
<td>translational linear spring constant in $Z$</td>
</tr>
<tr>
<td>4</td>
<td>Kx2</td>
<td>translational quadratic spring constant in $X$</td>
</tr>
<tr>
<td>5</td>
<td>Ky2</td>
<td>translational quadratic spring constant in $Y$</td>
</tr>
<tr>
<td>6</td>
<td>Kz2</td>
<td>translational quadratic spring constant in $Z$</td>
</tr>
<tr>
<td>7</td>
<td>Kx3</td>
<td>translational cubic spring constant in $X$</td>
</tr>
<tr>
<td>8</td>
<td>Ky3</td>
<td>translational cubic spring constant in $Y$</td>
</tr>
<tr>
<td>9</td>
<td>Kz3</td>
<td>translational cubic spring constant in $Z$</td>
</tr>
</tbody>
</table>

As an example element block,

```
block 52
   Rspring
   coordinate 7
   Krx=1e6
   Kry = 1.11E7
   Krz 0.1
end
```
As an example element block,

```plaintext
block 51
  spring3
  coordinate 7
  Kx1 1e6
  Ky1 1.11e7
  Kz1 0
  Kx2 0
  Ky2 0
  Kz2 0
  Kx3 1e4
  Ky3 1.11e5
  Kz3 0
end
```

### 5.18. **Dashpot**

A dashpot represents a damping term proportional to velocity. Dashpot elements combine a viscous friction damper with a simple linear spring. The spring is included to avoid singular stiffness matrices when dashpots are connected without springs. Dashpots are currently only used in transient dynamic, direct frf and complex eigen analyses. For other analyses only the spring term will be used.

The damping factor is the damping matrix entry. It has units of \( \text{force} \cdot \text{time/length} \). For a single degree of freedom system with a mass=\( M \), the following equation is satisfied.

\[
K \cdot \dot{u} + c \cdot \ddot{u} + M \cdot \dddot{u} = f(t)
\] (5.3)

Currently, dashpots are defined in the basic coordinate system only. Because they are single degree of freedom elements, the direction must also be defined (i.e. cid=1, 2 or 3). There are three parameters. All are required.

<table>
<thead>
<tr>
<th>#</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>K</td>
<td>translational linear spring constant</td>
</tr>
<tr>
<td>2</td>
<td>c</td>
<td>damping factor</td>
</tr>
<tr>
<td>3</td>
<td>cid</td>
<td>coordinate direction (1, 2 or 3)</td>
</tr>
</tbody>
</table>

As an example element block,

```plaintext
block 51
  dashpot
  cid=1  // dashpot is in the X direction
  K=1e6
```
Dashpots may be represented in the Exodus file with any linear element. The Truss element most closely mimics the dashpot’s single degree of freedom behavior, and may be the best definition for domain decomposition tools.

Caution should be exercised when using dashpots (or any single degree of freedom element). The remaining degrees of freedom must be properly accounted for, or the system matrices will be singular. Care should also be exercised to ensure that if the nodes of the dashpot are not coincident, that the constraint force lies along the axis of the element - failure to do this can result in models that have nonzero rotational modes. There may also be important domain decomposition issues with dashpots. See Section 5.15 for a discussion.

5.19. SpringDashpot

The SpringDashpot element provides a general, fully coupled spring and dashpot connected to a pair of nodes. It is a linear element only, and is not corotational. It supports stiffness and damping in the translational and/or rotational degrees of freedom. The relevant parameters are described in Table 5-105.

As shown in the table, all the elements of the matrices may be entered for this element. An example follows.

```
block 100
  SpringDashpot
    Kxx = 1e4
    Kyy = 1e4
    Kzz = 1e4
    Kxy = -1e4
    Kyz = -1e4
    Kyz = 3.2
  end
```
<table>
<thead>
<tr>
<th>#</th>
<th>Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Kxx</td>
<td>Translation Stiffness, $K_{xx}$</td>
</tr>
<tr>
<td>2</td>
<td>Kyy</td>
<td>Translation Stiffness, $K_{yy}$</td>
</tr>
<tr>
<td>3</td>
<td>Kzz</td>
<td>Translation Stiffness, $K_{zz}$</td>
</tr>
<tr>
<td>4</td>
<td>Kxy</td>
<td>Translation Stiffness, $K_{xy}$</td>
</tr>
<tr>
<td>5</td>
<td>Kxz</td>
<td>Translation Stiffness, $K_{xz}$</td>
</tr>
<tr>
<td>6</td>
<td>Kyz</td>
<td>Translation Stiffness, $K_{yz}$</td>
</tr>
<tr>
<td>7</td>
<td>Krxx</td>
<td>Rotation Stiffness, $K_{rxx}$</td>
</tr>
<tr>
<td>8</td>
<td>Kryy</td>
<td>Rotation Stiffness, $K_{ryy}$</td>
</tr>
<tr>
<td>9</td>
<td>Krzz</td>
<td>Rotation Stiffness, $K_{rz}$</td>
</tr>
<tr>
<td>10</td>
<td>Krxy</td>
<td>Rotation Stiffness, $K_{rxy}$</td>
</tr>
<tr>
<td>11</td>
<td>Krxz</td>
<td>Rotation Stiffness, $K_{rxz}$</td>
</tr>
<tr>
<td>12</td>
<td>Kryz</td>
<td>Rotation Stiffness, $K_{ryz}$</td>
</tr>
<tr>
<td>13</td>
<td>Bxx</td>
<td>Translation Damping</td>
</tr>
<tr>
<td>14</td>
<td>Byy</td>
<td>Translation Damping</td>
</tr>
<tr>
<td>15</td>
<td>Bzz</td>
<td>Translation Damping</td>
</tr>
<tr>
<td>16</td>
<td>Bxy</td>
<td>Translation Damping</td>
</tr>
<tr>
<td>17</td>
<td>Bxz</td>
<td>Translation Damping</td>
</tr>
<tr>
<td>18</td>
<td>Byz</td>
<td>Translation Damping</td>
</tr>
<tr>
<td>19</td>
<td>Brxx</td>
<td>Rotation Damping</td>
</tr>
<tr>
<td>20</td>
<td>Bryy</td>
<td>Rotation Damping</td>
</tr>
<tr>
<td>21</td>
<td>Brzz</td>
<td>Rotation Damping</td>
</tr>
<tr>
<td>22</td>
<td>Brxy</td>
<td>Rotation Damping</td>
</tr>
<tr>
<td>23</td>
<td>Brxz</td>
<td>Rotation Damping</td>
</tr>
<tr>
<td>24</td>
<td>Bryz</td>
<td>Rotation Damping</td>
</tr>
<tr>
<td>25</td>
<td>coordinate</td>
<td>coordinate frame</td>
</tr>
</tbody>
</table>

**Table 5-105.** – SpringDashpot Parameters.
5.20. **Hys**

The Hys element provides a simple, one dimensional approximation of a joint going through microslip. Many simple joints can be represented by their hysteresis loop, a curve in the displacement vs. force plane. The relevant parameters of this element are indicated in the table, and illustrated in Figure 5-38.

<table>
<thead>
<tr>
<th>#</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Kmax</td>
<td>maximum slope of $f$ vs $u$ curve</td>
</tr>
<tr>
<td>2</td>
<td>Kmin</td>
<td>minimum slope of $f$ vs $u$ curve</td>
</tr>
<tr>
<td>3</td>
<td>fmax</td>
<td>maximum possible force</td>
</tr>
<tr>
<td>4</td>
<td>dmax</td>
<td>maximum possible displacement</td>
</tr>
</tbody>
</table>

The $f_{max}$, $d_{max}$ pair define the limits of applicability of the element. The element will fail if the internal force exceeds $f_{max}$ or the displacement exceeds $d_{max}$. The slope of the curve at the origin is $k_{max}$. It represents the small amplitude response of the system. The slope at the extremum, i.e. at $(d_{max}, k_{max})$ is $k_{min}$.

A Hys element uses a Beam or truss element in the Exodus file. At the current time, the element may only be defined in the $X$ direction. An example of the Sierra/SD input is shown below.

```plaintext
block 2
  Hys
  Kmax 4.5e+7
  Kmin 3.0e6
  fmax 5.92
  dmax 0.9833e-6
end
```
Figure 5-38. – Hys element parameters.
5.21. Joint2G

The Joint2G element was added with the Iwan models. It models the adherence between surfaces.

Each Joint2G element connects a pair of nodes (or grids, hence the “G” in Joint2G). It permits users to specify independently the constitutive behavior of each of the degrees of freedom connecting its node pair.

The constitutive behavior is implemented through a constitutive class that provides generalized scalar forces in response to corresponding generalized displacements. The constitutive response was decoupled from the element to support general constitutive models.

Joint deformation under cycling loading is complex. The Iwan element can accurately model bolted joints, but depends on parameters that must be calibrated (using experimental data).

5.21.1. Specification

The meshed objects that map into the Joint2G element are defined in the Exodus database using Beam or Bar elements. The Joint2G ignores the Exodus file attributes. Properties are be specified in the block and PROPERTY cards. In the example below, properties are assigned to element block “2”.

```
block 2
  coordinate 5
  shear_axis 2
  Joint2G
  kx=iwan 1
  ky=elastic 1.0e6
  kz=elastic 1.0e6
  krx=null
  kry=null
  krz=null
end
```

The above statement declares block 2 to be of type Joint2G. It also declares the constitutive response in the $x$ to be the 4 parameter Iwan model. The model parameters are specified in “Property 1” defined below. In this case, the four parameters chosen are χ, φ\_max, $R$, and $S$. The Iwan properties can be specified alternatively by the parameter set chi, phi\_max, F\_S, and beta.

```
property 1
  chi   =  -0.82139
  phi_max =  1.0325e-04
```
The constitutive behavior in the \( y \) and \( z \) directions is elastic with stiffness specified by the third argument - \( 1.0 \times 10^6 \) in this case.

In this example, there is no specification for constitutive behavior in the three rotational directions. The \textit{Null} specification merely means that those degrees of freedom in the relevant nodes are not activated (“touched”) by this element. Because of artifacts associated with parallelization, it is recommended that if any of the rotational degrees of freedom are active (not \textit{Null}), they all should be active.

The directions (“\( x \)”, “\( y \)”, and “\( z \)”) employed above are those associated with the coordinate system declared for the block. In the example shown, there is an explicit reference to coordinate system 5. If there is no such explicit reference to a coordinate system, then the “\( x \)”, “\( y \)”, and “\( z \)” directions are those of the global coordinate system.

In the case when the \texttt{joint2G} element is used in conjunction with a Tied Joint, then the \texttt{shear_axis} can be used to specify the “\( x \)” direction for the constitutive response of the \texttt{joint2G}. Note that the \texttt{shear_axis} parameter is only meaningful when the \texttt{joint2G} is used in conjunction with a Tied Joint.

The \texttt{shear_axis} parameter allows the user to specify the “\( x \)” direction for the constitutive behavior. Since \texttt{shear_axis} is set to 2 in the above example, the “\( x \)” direction will be derived from the second component of coordinate 5. For more information on the \texttt{shear_axis} parameter, we refer to Figure 5-53 and Section 5.35.

### 5.21.2. Constitutive Behavior

**Elastic** Undamped, linear elastic behavior is defined by the \texttt{elastic} followed by the value of the parameter. No \texttt{property} section is required.

**Damper** Linear, damped behavior is obtained using a \texttt{damper} in the \texttt{Joint2G} definition, and using a property definition to specify the stiffness and damping terms. Typically each direction will require a different property definition.
4-Parameter Iwan Model (iwan)  The Iwan element is a collection of spring slider elements designed to provide a predicted model of joint behavior (including energy loss). Joint modeling with Iwan elements is described elsewhere. Descriptions of the relationship of the Iwan element to other joint elements are also available.

The schematic of the Iwan model is shown in figure 5-39. Parameters for the behavior may be specified using either an older definition (Table 5-106), or a newer set (Table 5-107). The newer parameters are summarized below.

<table>
<thead>
<tr>
<th>#</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>chi</td>
<td>Exponent, $\chi$, describing slope of force-dissipation curve at small amplitudes</td>
</tr>
<tr>
<td>2</td>
<td>R</td>
<td>Constant coefficient in distribution</td>
</tr>
<tr>
<td>3</td>
<td>phi_max</td>
<td>Maximum break free pseudo-force</td>
</tr>
<tr>
<td>4</td>
<td>S</td>
<td>Strength of singularity in break free force distribution</td>
</tr>
<tr>
<td></td>
<td>alpha</td>
<td>Geometric factor specifying nonuniform spacing of dphi (optional, default = 1.2)</td>
</tr>
<tr>
<td></td>
<td>sliders</td>
<td>Number of slider elements (optional, default = 50)</td>
</tr>
</tbody>
</table>

Table 5-106. – Older Iwan 4-parameter model.

<table>
<thead>
<tr>
<th>#</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>chi</td>
<td>Exponent, $\chi$, describing slope of force-dissipation curve at small amplitudes</td>
</tr>
<tr>
<td>2</td>
<td>beta</td>
<td>Shape parameter of force/dissipation curve</td>
</tr>
<tr>
<td>3</td>
<td>K_T</td>
<td>Tangent stiffness at low loads</td>
</tr>
<tr>
<td>4</td>
<td>FS</td>
<td>Maximum break free pseudo-force</td>
</tr>
<tr>
<td></td>
<td>alpha</td>
<td>Geometric factor specifying nonuniform spacing of dphi (optional, default = 1.2)</td>
</tr>
<tr>
<td></td>
<td>sliders</td>
<td>Number of slider elements (optional, default = 50)</td>
</tr>
</tbody>
</table>

Table 5-107. – Revised Iwan 4-parameter model.

$\chi$: determines the slope of the dissipation-force curve. Typically $0 < \chi < -1$. A value of zero corresponds to a coulomb type loss in Mindlin solutions. A value of $\chi = -1$ corresponds to a viscous like (but amplitude dependent) loss with dissipation proportional to the square of the amplitude. Dissipation follows the relation,

$$\text{Dissipation} \approx (\text{Amplitude})^{\chi+3}$$
Figure 5-39. – Iwan Constitutive Model.
beta: determines the shape of the dissipation-force curve. Beta affects both the shape of the hysteresis curve within microslip (Figure 5-40), and the abruptness of the transition from microslip to macroslip as shown in Figure 5-41. $0 \leq \beta < \infty$.

KT: determines the slope of the force-displacement curve at low amplitudes. This is equivalent to a spring constant, and is used as such in analyses for which the element is treated linearly.

FS: determines the force at which the last slider gives out, and element goes into macroslip. The Iwan element is a statistical distribution of spring/slider elements. This is a point on that distribution.

The Reduced Iwan improves on the Iwan by requiring less calibration. Mechanisms to capture nonlinearity and dissipation are provided. Accuracy however depends on the fit with the experimental data under varying loading conditions.

**Reduced Iwan Plus Pinning (Riwan)** Reduced Iwan model, described later in the section, can be used as a constitutive model for a Joint2G definition using the riwan. This can be used for any desired direction, using the following format that requires associated property block:

```plaintext
block 3
    Joint2G
    kx=damper 1
    ky=riwan 2
    kz=damper 3
```

Figure 5-40. – Dimensionless hysteresis curves for the four-parameter Iwan model with $\chi = -1/2$ and two values of $\beta$. 
Figure 5-41. – Dimensionless static loading curves for the four-parameter Iwan model with \( \chi = -1/2 \) and three values of \( \beta \), as the model goes into macroslip.

\[
\begin{align*}
F_{RIwan} &= F_{Pin} + F_{Sliding}. \\
(5.4)
\end{align*}
\]

Description: The RIwan element is a modification made by Brake (Ref.\textsuperscript{12}), of the revised 4 parameter Iwan model described in the previous section. The modifications include (a) simplification of the Iwan force by assuming that, upon load reversal, the distribution of friction elements resembles scaled version of the original distribution, and (b) incorporation of pinning forces through approximation of Hertzian contact. The monotonic load-displacement curve for this element is shown in figure 5-42, where the initial nonlinear curve corresponds to microslip, the plateau corresponds to macroslip, and the steep linear curve represents pinning. The reader is referred to Ref.\textsuperscript{12} for further details, but the model can be summarized as follows:

\[
\begin{align*}
F_{RIwan} &= F_{Pin} + F_{Sliding}. \\
(5.4)
\end{align*}
\]
where the pinning force is given by the linear relationship,

\[ F_{Pin} = \begin{cases} 
K_p(u - \delta_p), & u > \delta_p \\
0, & \delta_p \leq u \leq \delta_p \\
K_p(u + \delta_p), & u < -\delta_p 
\end{cases} \quad (5.5) \]

where \( \delta_p \) is the pinning displacement shown in 5-42. The sliding force is given by

\[ F_{Sliding} = \begin{cases} 
F_0 + \frac{F_S - F_0}{F_S} F_{Iwan} \left( u - \frac{F_S}{F_S - F_0} \right), & \text{loading} \\
F_0 - \frac{-F_S - F_0}{-F_S} F_{Iwan} \left( -u - \frac{F_S}{-F_S - F_0} \right), & \text{reverse loading} 
\end{cases} \quad (5.6) \]

where \( F_0 \) is the shifted central force with \(-F_S < F_0 < F_S\). Table 5-107 and figure 5-42 define \( F_S, K_T, \chi, \beta, K_p, \delta_p \). We use the 4 parameter model Iwan force. \( F_{Iwan} = \)

\[ \frac{F_S(\chi+1)}{\phi^{\chi+2}_{MAX} \left( \frac{\beta+\chi+1}{\chi+2} \right) \left( \frac{1}{\chi+2} - \frac{1}{\chi+1} \right) u^{\chi+2} + \frac{\phi^{\chi+1}_{MAX}}{\chi+1} u} \]

\[ + \frac{F_S}{\phi^{\chi+2}_{MAX} \left( \frac{\beta+\chi+1}{\chi+2} \right)} \min(u, \phi_{MAX}) \quad (5.7) \]

One Dimensional Gap Model (gap) The Gap element model attempts to represent the behavior of a gap closure with a bilinear elastic element. For proper numerical behavior, the stiffness of the open gap should not be more than a few orders of magnitude less than
Figure 5-43. – Eplas Model.

the stiffness when the gap is closed. The Joint2G implementation of the Gap model is identical to the axial behavior of NASTRANs cgap/pgap element and the axial behavior of the stand alone version of the Gap element implemented in Sierra/SD (Section 5.23).

<table>
<thead>
<tr>
<th>#</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Ku</td>
<td>Unloaded Stiffness</td>
</tr>
<tr>
<td>2</td>
<td>Kl</td>
<td>Loaded Stiffness</td>
</tr>
<tr>
<td>3</td>
<td>U0</td>
<td>Initial Gap Opening</td>
</tr>
<tr>
<td>4</td>
<td>F0</td>
<td>Preload (force at U0)</td>
</tr>
</tbody>
</table>

```plaintext
property 1
  ku = 1e5
  kl = 1e6
  U0 = 0.01
  F0 = 200
end
```

**Elastic Plastic Hardening Model (eplas)**  
eplas element is an elastic-plastic 1-dimensional element with linear isotropic hardening. Both the plastic strain and the hardening variable are initialized to zero. The parameters are illustrated in Figure 5-43.

<table>
<thead>
<tr>
<th>#</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>k</td>
<td>Linear Stiffness</td>
</tr>
<tr>
<td>2</td>
<td>kp</td>
<td>Hardening Stiffness</td>
</tr>
<tr>
<td>3</td>
<td>fyield</td>
<td>Force at Yield</td>
</tr>
</tbody>
</table>
One Dimensional Spring-Dashpot Model (damper)  

A damper represents a damping term proportional to velocity. Damper elements combine a viscous friction damper with a linear or cubic spring. The spring is included to avoid singular stiffness matrices when dampers are connected without springs. Dampers are currently only used in transient dynamic, direct frf and complex eigen analyses. For other analyses only the spring terms will be used. The behavior of this element is similar to dashpot, but also includes cubic terms.

The damping factor is the damping matrix entry. It has units of \textit{force \cdot time/length}. For a single degree of freedom system with a mass=\(M\), the following equation is satisfied.

\[
K \cdot u + \mu \cdot \dot{u} + M \cdot \ddot{u} + K_3 \cdot u^3 + \mu_3 \cdot \dot{u}^3 = f(t) 
\]  \hspace{1cm} (5.8)

<table>
<thead>
<tr>
<th>#</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>K</td>
<td>stiffness</td>
</tr>
<tr>
<td>2</td>
<td>Mu</td>
<td>viscous damper coefficient</td>
</tr>
<tr>
<td>3</td>
<td>K3</td>
<td>\textit{Optional} cubic stiffness coefficient</td>
</tr>
<tr>
<td>4</td>
<td>Mu3</td>
<td>\textit{Optional} cubic damping coefficient</td>
</tr>
</tbody>
</table>

Additional Constitutive Behavior  
The philosophy employed in the implementation of the Joint2G element of decoupling the constitutive behavior from the element machinery should facilitate the implementation of other constitutive models. Among those whose implementation is foreseen are the following:

- Bouc-Wen hysteresis model
- Preisach hysteresis model
5.22. **Line Weld**

Line welds offer a way to join two shells by a collection of virtual Joint2G elements (section 5.21). The position of the line weld is defined by a collection of beams which are meshed contiguously with one of the shells. Using a contiguously-meshed beam instead of the shell directly to define the line weld position enables more general weld definitions (e.g. skip welds).

![Diagram of line weld](image)

**Figure 5-44.** – Line weld definitions for attaching the purple and cyan shells. The line weld follows the red beam, which is contiguously meshed with the purple shell.

In figure 5-44, the line weld will be used to join block 3 (in red) to surface 1 on the cyan shell. This is accomplished in three steps, each illustrated by figure 5-45:

- Create a new virtual node (in green) coincident with every existing block node
- Tie each virtual node to the nodes of the shell on which it lies
- Create a zero-length, virtual Joint2G between every block node and its coincident virtual node

Currently, line welds in **Sierra/SD** are only available for the elastic Joint2G axial constitutive model. These can be thought of as a general translational/rotational spring, with axes as shown in figure 5-44. Input 5.4 gives an example input for the line weld shown in figure 5-44. Note that rotational stiffness is only supported along the direction of the beam (the “r-axis”). Any rotational components along the other two axes will be ignored.

In keeping with Sierra/SM syntax, line welds may be specified using a force vs. deflection function as shown in Input 5.5. The current line weld capability is purely linear, and the line weld stiffness $k$ is determined by estimating the derivative with a forward finite-difference stencil, i.e.,

\[
    k \approx \lim_{h \to 0^+} \frac{f(h) - f(0)}{h}.
\]

A warning is issued if the corresponding backward stencil indicates that the derivative is not well-defined, i.e. if

\[
    \lim_{h \to 0^-} \frac{f(h) - f(0)}{h} \neq \lim_{h \to 0^-} \frac{f(h) - f(0)}{h}.
\]
Figure 5-45. – Line weld Joint2G connections. The Joint2G elements are initially zero-length, and the green and red nodes coincident, but they are depicted with a finite deformation to delineate the individual components.

A fixed step size is used, so although the code will detect the case where the limits are not equal, it will not know whether the one-sided limits are well-defined. To aid the user in assessing the validity of these operations, the estimated stiffness values and other critical information is reported in the .rslt file.

Gap removal is enabled by default, and can be disabled using the gap removal option. The gap removal solution method 3.35 can make debug easier. Disabling gap removal can cause artificial grounding of rigid body modes, but can avoid collapsing elements when the search tolerance is large or when element quality is low. As with contact, the output mesh represents the model after gap removal.

```plaintext
begin line weld
    surface = surface_1
    block = block_3
    search tolerance = 1e-4
    r displacement elastic = 1.0e6
    s displacement elastic = 1.0e6
    t displacement elastic = 1.0e6
    r rotation elastic = 1.0e6
    gap removal = on
end
```

Input 5.4. Line weld input corresponding to figure 5-44

```plaintext
begin line weld
    surface = surface_1
    block = block_3
    search tolerance = 1e-4
```
Input 5.5. Force function syntax for line welds

Adding line_weld to an outputs or history block selects line weld output. To get the line weld output in the history file, the beam block from which the line welds were created must be included in the history section using block. The keyword line_weld adds the additional outputs shown in table 5-108 to the beam elements from which the line weld was created. The new virtual line weld elements created in the mesh have the usual output associated with a Joint2G element, such as eforce.

Line welds output force per unit length in the weld local coordinate system as line_weld_force_rst and line_weld_moment_rst.

<table>
<thead>
<tr>
<th>Name</th>
<th>type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>line_weld_weld_active</td>
<td>int</td>
<td>A value of one marks beams within search tolerance to the surfaces that have active welds</td>
</tr>
<tr>
<td>line_weld_initial_weld_length</td>
<td>real</td>
<td>Length of the line weld beam element</td>
</tr>
<tr>
<td>line_weld_force</td>
<td>vector</td>
<td>Force per unit length produced by the line weld in the global XYZ system</td>
</tr>
<tr>
<td>line_weld_moment</td>
<td>vector</td>
<td>Moment per unit length produced by the line weld in the global XYZ system</td>
</tr>
<tr>
<td>line_weld_force_rst</td>
<td>vector</td>
<td>Force per unit length produced by the line weld in the element local RST system</td>
</tr>
<tr>
<td>line_weld_moment_rst</td>
<td>vector</td>
<td>Moment per unit length produced by the line weld in the element local RST system</td>
</tr>
</tbody>
</table>

Table 5-108. Line weld output: note that these are intended to exactly match the equivalent outputs in Sierra/SM.
Gap elements are modeled after the non-adaptive NASTRAN CGAP/PGAP elements. They are intended to provide a simple, penalty type element suitable for modeling simple connections. Note that these elements (like all beam-like elements) when embedded in solid meshes can result in difficult domain decompositions, and lead to load imbalance.

The **Gap element** is inherently nonlinear. In linear analysis, the element behaves approximately like a spring with the stiffness determined by $K_L$ and $K_U$ and a transverse stiffness, $K_T$. The parameters of the element are listed in the table below and shown graphically in Figure 5-46.

<table>
<thead>
<tr>
<th>#</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>$K_U$</td>
<td>Unloaded stiffness</td>
</tr>
<tr>
<td>2</td>
<td>$K_L$</td>
<td>Loaded stiffness</td>
</tr>
<tr>
<td>3</td>
<td>$K_T$</td>
<td>Transverse stiffness (closed)</td>
</tr>
<tr>
<td>4</td>
<td>$U_0$</td>
<td>Initial gap opening</td>
</tr>
<tr>
<td>5</td>
<td>$F_0$</td>
<td>Preload, i.e. force at $U_0$</td>
</tr>
<tr>
<td>6</td>
<td>coordinate</td>
<td>Coordinate frame.</td>
</tr>
</tbody>
</table>

The unloaded stiffness, $K_U$, represents the stiffness of the element when the gap is open. It must be greater than zero. The loaded stiffness, $K_L$, represents the stiffness when the gap is closed (as shown in the figure). The stiffness is $K_L$ when $U_A - U_B$ is greater than $U_0$.

The initial gap opening and preload define the corner point in the force/deflection curve as shown in Figure 5-46. Typically, these will be zero.

A Gap element provides for transverse stiffness and friction. When the gap is closed, the transverse stiffness is $K_T$. If the gap is open, the transverse stiffness is reduced to $K_T' = K_T \times K_U/K_L$.

The coordinate frame is an optional attribute of the gap element. The gap open and closes along the $X$ axis of the frame. Note that the direction of the coordinate frame is important. The element determines a quantity $U_A - U_B$ along this coordinate axis. *This axis may not align with the coordinate alignment of the elements, which can lead to confusion.* If the coordinate frame is not provided, each Gap element will have a coordinate frame generated such that the gap opens and closes along the line between the two points. If the points are coincident, then a coordinate frame *must* be provided.

The Gap element is a simple penalty type element that somewhat mimics the effect of a physical gap. Choice of the value of $K_L$ is important to success of the element. Good values are somewhat in the range of the neighboring element stiffness. Too large a value can lead to matrix condition problems. Too small a value results in excessive softness and penetration in the gap.

Because the element is nonlinear, it has a significant impact on solutions. As described in Section 3.22 (and the **update_tangent** keyword), the default behavior for the nonlinear...
solver is a partial Newton iteration. This means that the tangent stiffness matrix is not updated between iterations. Thus, if KL and KU are different, the solver will be using the wrong slope in the newton loop. Many, many iterations may be required for convergence. You may want to turn on the ‘nlresiduals’ option in the echo section (see 7.7) which will put convergence information into the results file.

An example is shown below.

```
block 2
  GAP
  KL 4.5e+7
  KU 3.0e6
  KT=1e6
  f0 5.92
  u0=0.9833e-6
  coordinate 5
end
```

![Figure 5-46. – Gap element Force-Deflection Curve.](image)

**Gap Issues** The Gap element is definitely more complex than most elastic elements. Here is a partial list of “gotchas” that we have observed.

- Gaps should normally be zero length elements. Like springs, a gap that has a physical length will not be invariant to rigid body rotation. See Section 1.8.7. One approach to this would be to use a combination of beam and Gap elements. Note however, that if KT is zero, and the gap opens and closes along the line between the beam endpoints, the element is invariant to rotation.

- The Gap element may use a coordinate frame to define its direction. *In this case the direction is not set by the nodal coordinates.*

- The direction of the Gap element must correlate to the displacement difference from \( UB - UA \). It is easy to get this direction reversed.
Figure 5-47. – Mass bouncing off a Gap. With this large time step the model is not conserving energy. Reducing the time step is required to correct the problem.

- If you set $U_0$, you must also set $F_0$. This element does not constrain the force/displacement curve to go through zero. The input must do this. The Gap element may thus be used to enforce an initial displacement or force. That may not be what you want. It can cause slow convergence on the initial time step.

- Significant numerical damping may be required for convergence. Closing the gap can cause energy to be moved into higher frequencies. Without numerical damping, this energy can multiply until the solution becomes unstable. Numerical damping is best introduced by setting “rho” in the time integrator. Values of “rho=0.2” to “rho=0.7” have worked well. It is problem dependent.

Physically closing a gap would cause some energy loss, either by microslip, or by a small amount of local plastic deformation. Numerical damping can dissipate this energy that is removed from the physical system by means that are not included in the finite element model.

- This Gap element may not conserve energy. This is demonstrated in Figure 5-47, where a mass is dropped onto a gap. A completely elastic rebound would take the mass back to zero. Instead, it rebounds significantly above zero. This issue comes about because of time discretization. The mass “penetrates” the gap region too far, which stores too much energy in the element. It is then expelled with too much velocity. The only solution with this element is to reduce the integration step.
• Setting either $K_U$ or $K_L$ to zero is a recipe for disaster in parallel. Use a small positive value even if physically the unloaded stiffness may be zero.

### 5.24. Gap2D

The **Gap element** of the previous section provides a useful construct for planar type interactions. A common modeling issue is a bolt hole that is too large. To model this interaction an ellipsoidal Gap element (or **Gap2D** may be required.

The **Gap2D** element operates like the Gap element except that the gap could open in 2 dimensions. The gap is open provided that the element displacement is within an ellipse defined by the major and minor axes.

\[
\left(\frac{u_x}{U_{0X}}\right)^2 + \left(\frac{u_y}{U_{0Y}}\right)^2 < 1
\]  

(5.9)

The major and minor axes of the ellipse are defined in the $x$ and $y$ direction of the coordinate frame.

Parameters of the **Gap2D** element are listed below.

<table>
<thead>
<tr>
<th>#</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>KU</td>
<td>Unloaded stiffness</td>
</tr>
<tr>
<td>2</td>
<td>KL</td>
<td>Loaded stiffness</td>
</tr>
<tr>
<td>3</td>
<td>KT</td>
<td>Transverse stiffness ($z$ direction)</td>
</tr>
<tr>
<td>4</td>
<td>U0X</td>
<td>Initial gap opening, major direction</td>
</tr>
<tr>
<td>5</td>
<td>U0Y</td>
<td>Initial gap opening, minor direction</td>
</tr>
<tr>
<td>6</td>
<td>coordinate</td>
<td>Coordinate frame.</td>
</tr>
</tbody>
</table>

While the gap geometry is defined as an ellipse, stiffness is not. In the open section of the element, the in-plane stiffness is $K_U$, and is independent of direction. Likewise, in the closed gap region, the in-plane stiffness is independent of direction, and is defined by $K_L$. The out of plane stiffness for this element is *always* $K_T$. Note that the transverse stiffness behavior is significantly different from that of the standard **Gap element**.

The definitions above define the gradient of the force only, and for this nonlinear force, the value of the force depends on the path chosen for integration. For this element, we define the force as the integral along the shortest line from the origin.

In Figure 5-48, two possible integration paths are shown for arriving at the point $(x_1, y_1)$. In the first path, we integrate to $(x_1, 0)$ and then up to $(x_1, y_1)$. The $y$ component of force is $f_y^{(1)} = K_L \cdot y_1$. In path 2, we follow the straight line through $(x_b, y_b)$. The associated force is $f_y^{(2)} = K_U \cdot y_b + K_L(y_1 - y_b)$. For this element, we always choose the shortest line path (path 2). This ensures that the force is not history dependent.
5.25. **GasDmp**

GasDmp is currently BETA release. Enable with the “--beta” command-line option.

The **GasDmp** element is a nonlinear, beam-like element that simulates the damping forces on MEMS devices due to gas pressure as MEMS beams vibrate. The element has no stiffness, but has damping roughly proportional to velocity/L^3, where L is the distance from the beam to the substrate. The element is experimental. Contact Troy Skousen of *Sandia National Labs* or Professor Burak Ozdoganlar at Carnegie Mellon for details.

Inputs to the **GasDmp** element are as follows.

<table>
<thead>
<tr>
<th>#</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>W</td>
<td>Beam width (length units)</td>
</tr>
<tr>
<td>2</td>
<td>dL</td>
<td>Considered length of beam (length)</td>
</tr>
<tr>
<td>3</td>
<td>mm</td>
<td>Molecular mass of gas (mass)</td>
</tr>
<tr>
<td>4</td>
<td>p0</td>
<td>Ambient pressure of gas (pressure)</td>
</tr>
<tr>
<td>5</td>
<td>T</td>
<td>Ambient temperature of gas (temperature)</td>
</tr>
<tr>
<td>6</td>
<td>muRef</td>
<td>Reference viscosity (pressure * time)</td>
</tr>
<tr>
<td>7</td>
<td>TRef</td>
<td>Reference temperature (temperature)</td>
</tr>
<tr>
<td>8</td>
<td>ww</td>
<td>Viscous temperature exponent</td>
</tr>
</tbody>
</table>

Currently, the parameters are implemented through the input file and not through the Exodus_II file.
5.26. **Nmount**

The **Nmount** element is a Navy-specific mount element that provides an external force at user-specified points in the model. These forces are formed from a constitutive equation that is supplied by the user in the form of a subroutine. The Nmount capability provides an interface capability that allows the user to input their own subroutine that evaluates the constitutive equation.

An example of the user interface is shown in input 5.6. Mount orthogonal directions must be provided either as attributes in the **Exodus** file, or using the “Orientation” keyword in the “block” section. The relation of the orientation vector to internal element coordinates is shown in Figure 5-49. Remaining information is provided in the “block” section. Each mount type requires a separate block entry. Mount parameters are provided as text input in the block section. A list of built in Nmount types is listed in Table 5-109.

Each mount type may require a different number of mount parameters. If more parameters are provided than required for this mount, the additional parameters are ignored *without warning*. If fewer parameters are provided than are anticipated for the mount, the last parameters are set to zero, a warning is printed, and the analysis continues.

Mount type “7” is a special user subroutine mount. This mount type relies on Fortran or C functions compiled into the code at runtime. For this option **user subroutine file** must be defined in the file section. The user subroutines are incorporated into a custom build of the **Sierra/SD** executable with a command line like “sierra –make salinas -i my__input.inp”. See the Sierra/SM user manual for more details on the mount user subroutine formats and requirements.

```
block 41378
  Nmount
  mount type = 99
  parameters = 1.414 3.141 2.713
  orientation = 0 0.7 1
end
```

**Input 5.6. Sierra/SD Mount Interface**

**Stability** The Nmount element applies a force to the joining nodes in much the same way as an externally applied force. It provides no contribution to the stiffness matrix, and as such resembles an explicit element. Thus, stability issues can arise with this formulation. For certain models, damping has been shown to stabilize the formulation. The user may need to experiment with time step and damping levels to determine appropriate parameters for a stable solution.
<table>
<thead>
<tr>
<th>Name</th>
<th>Index</th>
<th>Comment/Parameters</th>
</tr>
</thead>
</table>
| fail_truss | 1     | 1 = Axial stiffness, Kr  
2 = torsional stiffness  
3 = critical value of tensile strain  
4 = critical value of rotational strain |
|            |       | **SpringDashpot**                                                                 |
|            | 2     | 1 = Axial stiffness, Kr  
2 = Stiffness in S frame, Ks  
3 = Stiffness in T frame, Kt  
4 = Axial damping, Cr  
5 = Damping coefficient in S frame, Cs  
6 = Damping coefficient in T frame, Ct |
|            |       | **Snubber**                                                                    |
|            | 3     | 1 = Model Direction (1=axial, 2=radial)  
2 = Model Type (1=10k, 2=20k)  
3 = Snubber gap  
4 = offset  
5 = input weight on compression  
6 = input angle of radial action |
|            |       | **TorsionalSpring**                                                                |
|            | 4     | 1 = Toronto spring constant, $K_\theta$  
$M = K_\theta(\theta_2 - \theta_1)$ |
|            |       | **Test-spring**                                                                   |
|            | 5     | 1 = $K_x$  
2 = $K_y$  
3 = $K_z$  
4 = $K_{\theta_x}$  
5 = $K_{\theta_y}$  
6 = $K_{\theta_z}$  
$F_j = K_{j_x}x_j$ and $M_j = K_{\theta_j}\theta_j$ |
|            |       | **NLSpring**                                                                    |
|            | 6     | 1 = $K_x$  
2 = $K_y$  
3 = $K_z$  
4 = $W_x$  
5 = $W_y$  
6 = $W_z$  
$\vec{F} = K_x + W_x^2$ |
| subroutine | 7     | Mount behavior defined by  
mount force subroutine = <string>  
mount init subroutine = <string>  
mount size subroutine = <string> |

*Table 5-109.* – Built in Nmount Models.
Figure 5-49. – Nmount Orientation

\( X_{elem} \) Normalized vector from node 1 to node 2. May change as the structure deforms.

\( \vec{V} \) User provided orientation vector.

\[ Z_{elem} = \frac{\hat{X}_{elem} \times \vec{V}}{|\hat{X}_{elem} \times \vec{V}|} \]

\[ Y_{elem} = \hat{Z}_{elem} \times \hat{X}_{elem} \] A normalized vector in the \( X_{elem} \vec{V} \) plane, and orthogonal to \( X_{elem} \).

5.27. \textbf{Rrod}

An \textit{Rrod} is a pseudoelement which is infinitely stiff in the extension direction. The constraints for an \textit{Rrod} may be conveniently stated that the dot product of the translation and the beam axial direction for an \textit{Rrod} is zero. There is one constraint equation per \textit{Rrod}.

The \textit{Rrod} is specified using beams or trusses in the \textbf{Exodus} database, with a corresponding \textbf{block} section in the \textbf{Sierra/SD} text input file. No material is required. A block may contain any number of connected or disconnected \textit{Rrod} elements. The following is an example of the input file specification for an \textit{Rrod} if the \textbf{Exodus} database contains beams in block id=99.

```plaintext
block 99
  Rrod
end
```
<table>
<thead>
<tr>
<th>Attribute</th>
<th>default</th>
<th>description</th>
</tr>
</thead>
<tbody>
<tr>
<td>RB_ID</td>
<td>-</td>
<td>translation identifier</td>
</tr>
<tr>
<td>CID_FLAG_INDEP</td>
<td>123456</td>
<td>independent coordinate flag</td>
</tr>
<tr>
<td>CID_FLAG_DEPEND</td>
<td>123456</td>
<td>dependent coordinate flag</td>
</tr>
</tbody>
</table>

Table 5-110. – Rbar Exodus Attributes.

5.28. **Rbar**

An Rbar is a *pseudo* element which is infinitely stiff in extension, bending and torsion. The constraints for an Rbar may be summarized as follows.

1. the rotations at either end of the Rbar are identical,
2. there is no extension of the bar, and
3. translations at one end of the bar are consistent with rotations.

The Rbar is specified using beams or trusses in the Exodus database, with a corresponding block section in the input file. No material is required and any number of connected or disconnected Rbar elements may be placed in a block. The following is an example of the input file specification for Rbar elements if the Exodus database contains beams in block id=99.

```plaintext
block 99
  Rbar
end
```

Rbar elements can be reordered so that the number of them connected to a single node is minimized. Having a large number connected to the same node results in a populated matrix and a slow computation. Therefore, reducing the number of connections can shorten run time. (see the reorder_Rbar parameter in the parameters Section 2.3).

The Rbar attributes are listed in Table 5-110, and are described below.

**RB_ID** Sometimes a collection of Rbars is a description of a rigid body. This occurs for example when translating a NASTRAN model containing RBE2 elements. During translation these bars are grouped into rigid bodies based on their connectivity. The RB_ID is an index to that grouping.

**CID_FLAG_INDEP** By default, all degrees of freedom are active on both nodes of the Rbar. Independent dofs are activated on the first node. The “CID_FLAG_INDEP” allows control over which degrees of freedom are activated. The flag is specified as an integer which is sum of components.\(^2\)

\(^2\)It is an unusual descriptor, but it was designed to somewhat mimic the NASTRAN cid flag.
100000  $X$ degree of freedom
20000   $Y$ degree of freedom
3000    $Z$ degree of freedom
400     $R_x$ degree of freedom
50      $R_y$ degree of freedom
6       $R_z$ degree of freedom

Thus, ‘123456’ activates all dofs, and ‘123000’ activates only translations.

**CID_FLAG_DEPEND** By default, six dofs are eliminated from the bar. By setting this attribute to a non-default value, constraint equations may be skipped. The values are the same as the “CID_FLAG_INDEP” described above.

5.28.1. Interaction of Rbars

If two Rbars or sets of Rbars share a node then they are effectively merged into a single rigid set. E.g. if an Rbar connects nodes 1 and 2, and another Rbar connects nodes 2 and 3 then the effect is the set of nodes 1, 2, and 3 act as a single rigid set of nodes. The same rule applies to intersection of rigid sets 5.33 or the intersection of Rbars and rigid sets as the rigid set capability is effectively a means to automatically generate Rbars.

5.29. **RBE2**

*Sierra/SD* has no support for the NASTRAN RBE2 element. However, in most cases the RBE2 element is not that different from a collection of Rbar rigid elements.

5.30. **RBE3**

The RBE3 pseudo-element’s behavior is taken from the eponymous NASTRAN element. More detail is found in the Theory Manual section *Sierra/SD* Elements subsection Rigid Elements subsubsection RBE3

The element is used to apply distributed forces to many nodes while not stiffening the structure as an Rbar would. The RBE3 uses the concept of a reference node.

Because all the nodes in an RBE3 are not equivalent, each RBE3 requires its own block ID. In the Exodus file, all links connecting to a single RBE3 are defined in a single element block. The input file then specifies that this is an RBE3 element block, as shown in the example below. If the model requires many RBE3 elements, a separate block must be specified for each.
Usage  The optional parameters for the RBE3 pseudo-element are shown in the table below. These parameters must be specified in the input file, not as attributes of the Exodus file.

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>refc</td>
<td>string</td>
<td>reference node coordinates</td>
</tr>
<tr>
<td>WT</td>
<td>6 reals</td>
<td>relative weight of coordinates</td>
</tr>
</tbody>
</table>

refc  The REFC parameter sets the degrees of freedom to activate on the reference node. For instance REFC='12' activates equations that constrain degrees of freedom associated with X and Y translations. No other degrees of freedom are affected. If the REFC keyword is not provided, it defaults to REFC='123456', i.e. constraint relations will be provided for the 6 structural degrees of freedom on the reference node.

WT.  The contributions of each of the coordinates of the independent nodes may be scaled by WT. Most typically this would be used to determine the relative weight of rotational degrees of freedom on the independent nodes to the computation of the reference node rotations. The default value is WT=111000 which means that the rotations do not contribute to the RBE3.

Generally we recommend there be no contribution from the rotations. The rotation of the element may then be determined solely from the translational degrees of freedom on the independent nodes.

The formulation of the RBE3 is based directly on the published method from MSC/NASTRAN. Details of the method are described in the Theory Manual section Sierra/SD Elements subsection Rigid Elements subsubsection RBE3.

Cautions in using RBE3  Albeit convenient, the RBE3 is not a true element. It can introduce complexity in the solution.

- A RBE3 may connect a large portion of the model. This degrades linear solvers efficiency. As a consequence convergence may be slow.
- A RBE3 connected to many nodes requires a lot of memory. This memory is stored on a single processor.
- No two MPCs should be linked together. Linear solvers may fail in this case.
- Accelerations (see Section 6.1.3) cannot be prescribed on an RBE3 or any other MPC.
- The element has no logic to determine which degrees of freedom of the independent nodes are active. Thus, if you specify WT = 111111 the element will try to determine its rotation based on a combination of the translational and rotational degrees of freedom on the independent nodes. If the rotational degrees of freedom are inactive, they are treated as zero. This is rarely what is wanted.
• Care must be taken to ensure that only one node of the RBE3 has multiple connections to its links. Further, every link in the RBE3 must be connected to the reference node.

• A Joint2G with side averaging uses two RBE3 elements.

• Many user issues are caused by RBE3 elements.

**Example RBE3**  The following is an example of the input file specification for an RBE3 if the Exodus database contains beams in block id=99.

```
block 99
  RBE3
    refc=123456
    wt=1 1 1 0 0 0
end
```

### 5.31. Superelement

<table>
<thead>
<tr>
<th>Keyword</th>
<th>value</th>
<th>default</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>file</td>
<td>string</td>
<td></td>
<td>input file containing matrices</td>
</tr>
<tr>
<td>format</td>
<td>string</td>
<td>netcdf</td>
<td>format of superelement input file: netcdf or DMIG</td>
</tr>
<tr>
<td>savememory</td>
<td>yes/no</td>
<td>no</td>
<td>controls storage of matrices in memory</td>
</tr>
<tr>
<td>diagnostic</td>
<td>integer</td>
<td>1</td>
<td>0 - run no diagnostics, 1 - compute Kr * RBM, 2 - compute eig(Kr,Mr)</td>
</tr>
<tr>
<td>map</td>
<td>integer</td>
<td></td>
<td>table of node/cid pairs</td>
</tr>
<tr>
<td>map</td>
<td>string</td>
<td></td>
<td>“ascending_id” or “sorted” or “locations”</td>
</tr>
<tr>
<td>skip_output</td>
<td>yes/no</td>
<td></td>
<td>option to disable netcdf output</td>
</tr>
<tr>
<td>sensitivity_param</td>
<td>integer/real</td>
<td>parameter may be used</td>
<td>multiple times to specify different parameters</td>
</tr>
</tbody>
</table>

A superelement is an abstract concept with different realizations in commercial codes. Sierra/SD does not support a full automatic superelement capability. The Sierra/SD Craig-Bampton reduction 3.5 reduces an entire model to a reduced-order model. Importing a reduced model (or superelement) into Sierra/SD is also supported, typically using mksuper. Sierra/SD supports superelements that can import of a mass and stiffness matrix into a full system model. This linearized approach may be used in any type of analysis.
Evaluating the suitability of a superelement approximation is left to the user. Few if any users have used superelements correctly based on only the information in User’s Manual. Tutorials particularly the ones in\textsuperscript{42} are recommended.

Limitations

- The superelement must be small enough (have a sufficiently small number of degrees of freedom) to fit (in the virtual memory) a single MPI rank. No consideration for superelements which span processors is made.
- Nodes on the superelement interface may be shared across processors. Interior degrees of freedom are local to a single processor.
- Output of the interface node degrees of freedom will be made in the base model in the usual way. Output of internal superelement quantities will be made in the superelement database file. The superelement modal degrees of freedom will be stored in the \textbf{Exodus} and MATLAB output on the X-degree of freedom of newly created virtual nodes.
- No automatic data recovery is available.
- Only a single level of superelement is supported.
- The mass properties report is computed by lumping mass to the interface dofs. For a superelement formed from a free-free system this will preserve the total translational mass of the superelement. However, the rotational inertia and center of gravity will not be exactly preserved in the superelement mass properties. If the superelement has an internal constraint or if for other reasons the superelement cannot exactly reproduce rigid body modes then the translational mass properties will not necessarily be preserved by the superelement.
- No geometric stiffness effects are currently accounted for in superelements. The default at this time is for these elements to return zero geometric stiffness.

User Input

The following input is provided by the user. If format=DMIG, the connectivity information is read from the DMIG file, and cannot be specified in any other way.

**connectivity (Exodus):** Note that codes such as NASTRAN input superelements by connecting to the nodes directly. Like any element with \textbf{Sierra/SD} the superelement must be mapped to a single processor. The superelement must be in the finite element database used to partition the elements. To provide the geometric connectivity to the model, the connectivity must be added to the \textbf{Exodus} file in one way or another.

If the superelement has the same number of nodes as a standard element, the analyst may choose to use such an element to provide the connectivity. This can facilitate
visualization of the model. **Sierra/SD** does support an element with *more* nodes than required for the connectivity map. Thus, a Hex-8 could be used to define the connectivity for a superelement with 7 nodes on the interface. The connectivity map cannot have more nodes than the element.

The **mksuper** utility function will add a superelement to an **Exodus** database. See.\textsuperscript{42}

**connectivity map (Exodus):** The equations for the system matrices must be associated with the nodes and degrees of freedom in the model. The following example creates a map for an eight degree of freedom reduced order matrix. The first column of the map is associated with the node index in the element. The second degree of freedom defines the coordinate direction (typically 1 to 6 for \(x, y\), etc).

```
// node cid
map 0 0
  0 0
  1 1
  1 2
  1 3
  2 1
  2 2
  2 3
```

In this example, the first two rows of the system matrices are associated with internal degrees of freedom (DOFs) such as fixed interface modes. These interior dofs are indicated by a zero for both the node index, and the coordinate direction. Row 3 of the matrix is associated with the first node in the element connectivity, and with the \(x\) coordinate direction. Row 8 is associated with the second node, and the \(z\) coordinate direction.

There must be exactly as many rows in the connectivity map as there are rows in the system mass and stiffness matrices.

If the node index is negative, the row of the matrix associated with that degree of freedom will not be mapped to the system matrix. This can be used to “clamp” a generalized degree of freedom.

> The node index is not the node number in the Exodus file. It is the index into the element connectivity. Thus, for a four node element, the index must never exceed 4. This permits the use of gjoin and other tools without the need to reorder these terms in the input file.

Alternate formats may be used to provide the map between rows of the system matrices and degrees of freedom of the residual structure. For these alternate formats to be used, the netcdf file containing the superelement data must include the **cbmap**
data, which provides an *internal* mapping between internal rows and columns and the internal nodes. These methods include the following.

**map ascending_id or sorted** If the user specifies the node number connectivity of the superelement in an ascending node order, then we can automatically generate the map. Note, either `ascending_id` or `sorted` may be used here, they refer to identical algorithms.

**map locations** If the nodal coordinates of the superelement are stored in the netcdf reduced order model file, then the best match among coordinates of the residual and the superelement can be used to determine the map. This method works best if the superelement and residual have the same coordinate locations and if there are no collocated nodes in the interface. Each superelement interface node will be mapped to the closest finite element model node. No search tolerance is needed as the closest node is always found, even if it is far away. However, when using this option care should be taken that superelement interface nodes and finite element nodes match in space. If a superelement interface degree of freedom is mapped onto a finite element node with significantly different coordinates the superelement behavior may be substantially degraded and the superelement may no longer be able to represent rigid body modes. A warning is emitted by *Sierra/SD* if a node match cannot be found within a distance of one one-millionth of the characteristic model size.

**system matrices:** The system matrices may be provided in a netcdf or DMIG file. These matrices are available as output of the CBR reduction process (Section 3.5) and may also be generated with other tools such as Nasgen. The file must contain the following.

- **Kr.** The reduced stiffness matrix. This is required for all analysis.
- **Mr.** Most analyses require a reduced mass matrix as well. Its dimension must match that of the stiffness matrix.
- **Cr.** An optional reduced damping matrix may be used. It must be of the same dimension as **Kr.**

**maps** that connect the degrees of freedom of the superelement to the degrees of freedom of the residual structure.

An accurate reduced **Kr** for 3D analysis should have exactly 6 zero energy modes. It must be symmetric (*Sierra/SD* will try to symmetrize it). Typically, **Mr** would be non-singular. Failure to meet these requirements can confuse the entire solution procedure, and lead to erroneous solutions.

---

3With the **mksuper** application, it is easy for the user to set up an element with ascending order, but most tools do not know how to visualize the element. Visualization may be easier using standard elements, but the restriction that the connectivity have ascending node ids is confusing.
transfer matrices (Exodus): Output of results on interior points in the superelement are facilitated using optional output transfer matrices (OTM). These are described in the section on Craig-Bampton reduction (3.5). These matrices are written to the output Exodus file only if superelement output is requested in the Craig-Bampton reduction output specification. The following matrices apply.

**OTM** Nodal output transfer matrix.

**OTME** Element output transfer matrix.

**OutMap** An optional node map for the OTM.

**OutElemMap** An optional element number map for OTME.

**skip_output:** Optionally provides a means of disabling all output to the netcdf results files. This is particularly useful if the analyst wishes to use the same netcdf data for multiple superelements in the model. Without this keyword, each of these superelement blocks would be writing to the same file location, resulting in corrupted data.

**output specifications:** In the text input file superelement output is selected from either the outputs or echo sections by adding the word superelement.

If requested in the outputs section, then a new Exodus file will be generated from the information and name of the netcdf file. The number of nodes in the new file is the sum of the number of nodes on the interface and the number of nodes in the OTM. The number of elements is the number of elements in the OTME. All elements will be placed in a single element block.

Because we don’t know the connectivity of the elements in the OTME, all such elements will be defined as sphere elements, and will be collocated on a single node in the model. This impedes visualization, but the element data is preserved for other types of post processing.

Likewise, no coordinate information is available for the interior nodes of the model. These elements will be located at the origin of the system.

**sensitivity_param:** If the Craig-Bampton reduction that generated the superelement included a sensitivity analysis, then the netcdf file containing the superelement matrices also contains derivatives of the reduced matrices with respect to the parameters. This information can then be used in the superelement block to set the superelement parameters as needed. This uses the linear Taylor series expansion of the sensitivity information of the Craig-Bampton model to compute the updated reduced matrices, and thus by-passes the need re-generate the Craig-Bampton model when the parameters are perturbed. The sensitivity_param allows the user to input specific values of the parameters for the superelement.

The above parameters are entered in the block section of the input file. For example,
In this case, there are two sensitivity parameters, one for the thickness of a shell block in the Craig-Bampton model, and the other for the Young’s modulus in that same block. Note that the format is assumed to be netcdf because the keyword format is not specified.

DMIG Input Files

<table>
<thead>
<tr>
<th>Matrix</th>
<th>Acceptable Names</th>
</tr>
</thead>
<tbody>
<tr>
<td>Stiffness (Kr)</td>
<td>K2GG,KAX</td>
</tr>
<tr>
<td>Mass (Mr)</td>
<td>M2GG,MAAX</td>
</tr>
<tr>
<td>Damping (Cr)</td>
<td>C2GG,BAAX</td>
</tr>
</tbody>
</table>

Table 5-111. – Acceptable names of matrices within DMIG input files.

- Sierra/SD does not check the file extension of DMIG input files. Files generated by Sierra/SD use the .dmig extension, whereas direct output from NASTRAN uses .pch. Both are acceptable as input.
- Allowable matrix names are listed in table 5-111.
- Information required by Sierra/SD in parsing DMIG files comes from the matrices themselves: all comments are ignored, including e.g., those providing information about the dof map, number of interface modes, etc.
- The stiffness damping matrix K4AXX is ignored.

Finally, we show an example input deck for DMIG format:
5.32. **Dead**

A **dead** element has no mass and no stiffness. It may be of any dimensionality, solid, planar, line or point. Interior nodes to a block of **dead** elements will not be included in the computation of the model. There are no parameters for **dead** elements.

Note that for sideset-based load and boundary conditions, the boundary condition sideset is defined by sides of elements. If an element block is marked dead, then the boundary condition on the sides of those elements will generally also be deactivated. Special attention needs to be given to any sideset that is on the internal boundary between a dead block and an active block. Deactivation of the dead block may deactivate the boundary condition on that internal sideset if the sides are associated with the dead elements. In visualization tools the sideset normal will point out of the element it is attached to. Visualization of the sideset normal can be helpful to determine which block the sideset is attached in ambiguous internal sideset cases. As a general rule it is recommended that use of internal sidesets be avoided due to the model ambiguity they can cause.

5.33. **Rigidset**

Rigid Sets are intended as a usability tool to permit the analyst to treat a set of nodes as completely rigid. The input is straightforward.

```
Rigidset set1
  sideset 1
  sideset 2:5
  nodeset 88-90,wing
END
```

The above definition would establish a single set that is tied together. For purposes of error reporting only, the name “set1” is associated with this example set. If multiple independent sets are required, then multiple rigid set definitions may be made.

Table 5-112 shows the parameters. Any number of rigidset sections may be introduced. Each acts independently. Exodus sideset or nodeset information may be included in the definition. The rules for defining multiple nodesets, sidesets, or blocks at once are the same as the history output section, 7.3.
Table 5-112. – Rigidset Parameters.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>type</th>
<th>description</th>
</tr>
</thead>
<tbody>
<tr>
<td>sideset</td>
<td>int/name/list</td>
<td>sideset id</td>
</tr>
<tr>
<td>nodeset (not recommended)</td>
<td>int/name/list</td>
<td>nodeset id</td>
</tr>
<tr>
<td>CenterNode tiedto node</td>
<td>integer</td>
<td>see below</td>
</tr>
</tbody>
</table>

Tied Node tied node or center node A rigid set can be attached to a Tied Joint 5.35 or Joint2g 5.21. In this case, a “reference” node may be generated and tied to another block or element. This is accomplished with the keywords below.

CenterNode tiedto node XX block YY

The CenterNode command will create a bar element with two nodes, and associate it with block YY. One end of the bar element is node XX. Node XX must exist in the input mesh. The other end node of the bar element will be created on the fly at the centroid of the rigid set. Note this capability only works for blocks with a single element. There are examples in.42 Figure 5-50 illustrates the concept.

![Figure 5-50. – Rigidset/TiedJoint Centernode Connection. The model illustrates the connection of a physical rigid set to a physical reference node via a virtual center node and virtual connection block.]

5.33.0.1. Voltage Rigid Sets For many models using piezoelectric materials, a free surface of the piezoelectric tile may be plated with a purely conductive material such as copper. The conductive layer results in an equipotential surface. To simplify modeling an equipotential surface, Sierra/SD enables voltage rigid sets, which enforce a spatially constant voltage on the nodes associated with specified nodeset or sideset. Specifying voltage rigid sets are done in the rigid set block as shown in the following.
Input 5.7. Voltage Rigid Set

5.33.0.2. Limitations  Rigidsets meet an important need to tie many nodes together. Generally they are much more robust than generating collections of $R_{bar}$ rigid elements or other rigid elements. However, it is easy to generate redundant constraints through this input. Redundant constraints cause most linear solvers to fail, and Sierra/SD may not always provide diagnostics. Generally,

1. Rigidsets should be completely disjoint, i.e. should share no common nodes. If two rigid sets do share any node they are effectively merged as a single larger rigid set.

2. If a $R_{bar}$ is connected to any node of a rigid set then effectively the combination of Rigid set and connected $R_{bars}$ behave as a single rigid system. Similarly, connecting two rigid sets via a $R_{bar}$ effectively merges the two sets into a single larger rigid set.

3. None of the nodes in the rigid set should be constrained (as through a boundary condition).

4. While nodesets can be used to define rigid sets, this is not recommended because parallel decompositions may put only one or two nodes on a processor. So few nodes may introduce local singularities in rotation that impact the linear solver. If possible, use a sideset to define the rigidset.

5.34.  $R_{rodset}$

Like the rigidset of Section 5.33, the $R_{rodset}$ provides a convenient means of tying together a surface. All the limitations of the rigid set apply here. Unlike the rigid set, the $R_{rodset}$ constrains only the distance between nodes on the faces, and no rotational degrees of freedom are constrained. The $R_{rodset}$ acts much like a Kevlar skin; it resists stretching, but does not impede bending.

For a quadrilateral face, the $R_{rodset}$ is equivalent to applying a rigid rod to each of the edges of the face. A constraint is also placed across one of the diagonals of the face as shown in Figure 5-51. An example is shown below.

Like the rigidset, the $R_{rodset}$ may be used to connect a “reference” node to a block.
5.35. **Tied Joint**

The *Tied Joint* models a joint structure. At the heart of the *Tied Joint* is an *Iwan* model. The *Tied Joint* supports flexibly mixing many models. An Iwan element may be used to represent the shear response, and multipoint constraints may be used to represent the normal response. Energy loss of the joint can be approximated by the *Iwan* element, and normal surfaces alignment can be preserved by the constraints.

5.35.1. **Input Specification**

Refer to Figure 5-52 for reference to the model definition. An example input is shown in input 5.8. There are several sections to the model definitions. Parameters of the input are summarized in Table 5-113. Details are below.

![Figure 5-52. – Tied Joint Geometry. The two side set surfaces are shown separated for clarity. A virtual element is created which connects only the shear components of the joint. Normal components are interconnected using Tied MPCs.](image)

<table>
<thead>
<tr>
<th>Tied Joint</th>
</tr>
</thead>
<tbody>
<tr>
<td>Normal Definition = slip</td>
</tr>
<tr>
<td>surface 3,5</td>
</tr>
<tr>
<td>connect to Block 11 // Joint2g block</td>
</tr>
<tr>
<td>End</td>
</tr>
</tbody>
</table>

// definitions for the referenced Joint2g

Block 11

---

Figure 5-51. – Rodset Constraints. The black lines indicate the edge of the element. Red lines are corresponding linear constraints.
Joint2g  
Kx=Iwan 1  
Ky=Elastic 1e6  
coordinate=5 // for anisotropic shear parameters  
End

**Input 5.8. Tied Joint Example**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>surface</td>
<td>2 sidesets connected by the joints, in slip constraints the first sideset is the face-surface and the second the node-surface as in tied data</td>
</tr>
<tr>
<td>tied nodes</td>
<td>a sideset connected by the joint, the face-surface in slip constraints as in tied data</td>
</tr>
<tr>
<td>tied faces</td>
<td>a sideset connected by the joint, the node-surface in slip constraints as in tied data</td>
</tr>
<tr>
<td>normal</td>
<td>if slip, normal-only node-face constraints</td>
</tr>
<tr>
<td>slip</td>
<td>none</td>
</tr>
<tr>
<td>search_tolerance</td>
<td>if slip, defines node-face MPCs</td>
</tr>
<tr>
<td>edge_tolerance</td>
<td>if slip, defines node-face MPCs</td>
</tr>
<tr>
<td>shear</td>
<td>reference block for whole joint</td>
</tr>
<tr>
<td>connect to block side</td>
<td>average, rigid or Rrod</td>
</tr>
</tbody>
</table>

**Table 5-113.** – Tied Joint Parameters.

**Name:** Optional name of this joint. Useful primarily in diagnosing error messages.

**Surface:** Exactly two sidesets should be provided, these two sidesets will be connected via the joint. For node-face MPCs involved in normal direction constraints the nodes of the second surface (node-surface) are constrained to the faces of the first surface (face-surface)

**Normal Definition:** In the *Tied Joint*, the joint behavior in the normal direction can be governed by the joint element or by distributed node-face MPCs on the sidesets.

Slip implies the surfaces will remain in tied contact, and shear effects are managed by the “shear definition”. Tied node-face constraints will be created similar to the Tied Data 8.1 command. These constraints only tie the normal deformation of the sides together. The shear behavior of the joint will be managed by the single joint element. In this case, the joint element will be located at the nearest node to the centroid of the node-surface.
None implies that no specific node-face normal constraints will be generated.

Surfaces may separate or interfere and the joint normal behavior will be controlled only by the whole-joint element. In this case, the joint will be located at the centroid of the node-surface.

**Connect to block:** A reference to a block containing parameters for the whole-joint element, a single element that connects the two sides of the joint. Usually a Joint2G element is used. If the normal definition is none, then the whole-joint element must specify behavior in all six dofs. Otherwise, if the normal definition is slip, then tied constraints are used to constrain the normal motion of the joint and only the three dofs associated with plane motion in the whole-joint element are used.

For non-isotropic shear behavior, the block may include a coordinate command. The frame may be curvilinear (e.g. cylindrical), in which case whole joint quantities are evaluated at the centroid of the surfaces (see coordinates, 2.7). To reference the basic (or default) frame, use coordinate frame “0”. The coordinate frame is specified in the connected element frame.

For curvilinear coordinate frames, it may be difficult to exactly specify the orientation of the centroid of the surface. Any user defined coordinate frame will be projected to the plane of the surface at the centroid, and a new coordinate frame is generated for specification of the orthogonal, in-plane coordinates. The $\tilde{X}$ and $\tilde{Y}$ axes of the user-defined coordinate system are projected to the plane, with the new third axis ($\tilde{Z}'$) in the normal direction.

In the case when the whole-joint element is a Joint2G element, the shear_axis can be used in the block definition to define the coordinate direction used for the first in plane constitutive component. We refer to Figure 5-53 for a description of the local coordinate system used to specify the constitutive behavior of the Joint2G element. The surface normal, $n$, is obtained as the normal on the node that is closest to the centroid of the sidesets that define the Tied Joint. This normal direction defines the $\tilde{Z}'$ axis of the local coordinate system. The shear_axis definition specifies which of the 3 axis of the user-specified coordinate system (in this example coordinate 5) is intended to be the first shear direction for the constitutive response. Thus, if shear_axis is set to 1, then $\tilde{X}'$ is defined as the part of the $\tilde{X}$ axis from the user-defined coordinate system that is orthogonal to $\tilde{Z}' = n$. If the $\tilde{Z}$ axis of the user specified coordinate system lines up exactly with the normal at the node, then the shear direction will be in exactly the same direction as the $\tilde{X}$ axis in the user-defined coordinate system. Generally, they will not line up perfectly, and this is the main reason why the shear_axis is needed. Once $\tilde{Z}' = n$ and $X'$ are defined, the remaining component of the coordinate system $\tilde{Y}'$ can be obtained by a cross product.

**Side:** The side defines additional constraints on the surfaces and how the tied joint Joint2G element spreads load to the surfaces.

A tied joint is between two surfaces $A$ and $B$. To create the tied joint first a point in space is selected near the centroid of the joint. Next two new collocated nodes $A_c$.
Figure 5-53. – The surface normal, \( n \), is defined by the normal of the surface at the centroid point. The shear axis direction (in this example \( \tilde{X} \)) is projected onto the surface as \( \tilde{X}' \). Together, \( \tilde{X}' \) and surface normal provide the basis for the generated coordinate frame.

and \( B_c \) are created at this point. Node \( A_c \) will track the average displacement of side \( A \). Node \( B_c \) will track the average displacement of side \( B \). The whole joint element, a Joint2G, is connected between nodes \( A_c \) and \( B_c \) and controls the stiffness and damping of the joint.

average See Figure 5-54. \( A_c \) and \( B_c \) are connected to side \( A \) and \( B \) via RBE3 constraints. One RBE constrains node \( A_c \) to have the average displacement of side \( A \) and another RBEs constrains node \( B_c \) to have the average displacement of side \( B \). The RBE3 constraints are added as bar elements to the output mesh file. Side average is the default for “normal definition” of “slip”.

rigid provides a means of constraining all the nodes on each surface to move together as a rigid set (see users 5.33). See Figure 5-55. Side \( A \) is turned into a rigid set and side \( B \) is turned into a rigid set. Node \( A_c \) is added to rigid set \( A \) and node \( B_c \) is added to rigid set \( B \). An RBE3 is not required in this case as the rigid set already guarantees that the displacement at \( A_c \) and \( B_c \) track the average displacement of the sides. Side rigid is default for “normal definition” of “none”.

Rrod provides a means of constraining all the nodes on the surface to move together as a Rrodset (see users 5.34). See Figure 5-56. Side \( A \) is turned into a Rrodset and side \( B \) is turned into a Rrodset. As with side=average RBE3 constraints are needed to constrain nodes \( A_c \) and \( B_c \) to the average displacement of the two sides. The RBE3 constraints are added as bar elements in the output mesh file.

The side rigid option will create the stiffest overall joint behavior, average the softest, and Rrod something in between. Using option rigid to avoiding a complex RBE3 constraint can also avoid numerical issues that sometimes come with the average or Rrod options.

Not all Tied Joint specifications are fully consistent. In particular, the specification of the “normal definition” and the “side” descriptions are not fully independent. Table 5-114 summarizes some dependencies between these two parameters.

Output Specifications Because the Tied Joint is not fully represented in the Exodus database (except as a collection of surfaces), standard element output capabilities are
Figure 5-54. – Construction of tied joint with side=average.

Figure 5-55. – Construction of tied joint with side=rigid.

Figure 5-56. – Construction of tied joint with side=Rrod.
RBE3s constrain joint end nodes to have the average displacement of each surface. Additionally, each surface is turned into an Rrodset. Each surface is turned into a rigid set which also contains the joint end nodes. An RBE3 is not needed in this case as the rigid set already constraints the joint end nodes to have the average displacement of the surfaces.

<table>
<thead>
<tr>
<th>Normal Definition</th>
<th>Side</th>
<th>Status</th>
<th>Note</th>
</tr>
</thead>
<tbody>
<tr>
<td>none</td>
<td>average</td>
<td>RBE3s constrain joint end nodes to have the average displacement of each surface. RBE3s constrain joint end nodes to have the average displacement of each surface. Additionally, each surface is turned into an Rrodset. Each surface is turned into a rigid set which also contains the joint end nodes. An RBE3 is not needed in this case as the rigid set already constraints the joint end nodes to have the average displacement of the surfaces.</td>
<td></td>
</tr>
<tr>
<td>slip</td>
<td>average</td>
<td>Node-face normal only constraints are added to the surfaces. RBE3s constrain joint end nodes to have the average displacement of each surface. The shear deformation of the whole-joint model is based on the relative lateral motion of each side. Node-face normal only constraints are added to the surfaces. RBE3s constrain joint end nodes to have the average displacement of each surface. Rrod constraints stiffen the surface. The shear deformation of the whole-joint model is based on the relative lateral motion of each side. Invalid. Overly constrained joint. Fatal error.</td>
<td></td>
</tr>
</tbody>
</table>

Table 5-114. – Tied Joint, “Normal” and “Side” dependencies.
insufficient to represent the data. The data is divided into two categories: configuration and results.

5.35.1.1. Configuration Output  The configuration output is only available in the text output of Sierra/SD, i.e. in the .rslt file. It is requested with the keyword “input_summary” in the “ECHO” section (see 7.7). This includes the following.

1. The type of the normal enforcement.
2. Surface information.
3. Centroid of the surface pairs (if applicable).
4. Owning processor for the shear elements (if applicable).
5. Shear models.

Results Output  The only results output that is currently available for a Tied Joint consists of the forces in the Joint2G element that connect the two surfaces of the Tied Joint together. Currently, these forces can be only obtained in the history (or frequency) file for a transient or nonlinear transient analysis. They cannot be written to the global Exodus output file. If we consider the same example that is given in input 5.8, we could obtain the element forces as follows for a transient analysis

```plaintext
History
   block 11
   EForce
end
```
or, for a frequency domain analysis,

```plaintext
Frequency
   block 11
   EForce
enD
```

where in this example block 11 is the Joint2G block that connect the two surfaces of the Tied Joint together.
6. BOUNDARY CONDITIONS AND INITIAL CONDITIONS

6.1. Boundary conditions

Boundary conditions are specified within the boundary section. Node sets, side sets, blocks, or node lists may be used to specify boundary conditions. The rules for defining multiple nodesets, sidesets, or blocks at once are the same as the history output section, 7.3, and also follows the rules for integer lists detailed in Section 2.1. Specialized coordinate systems that can be used are described in section 2.7. The example in input 6.1 illustrates the method.

```
boundary
nodeset end_nodes // for nodes in nodeset "end_nodes"
  x = 0 // constrain x=0
  coordinate 1 // for x/y/z in coord system 1

nodeset 1 // for nodes in nodeset 1
  x = 0.1 // constrain x=0.1
  y = 0 // and y=0
  RotZ = 0 // and the rotational dof about Z

nodeset 13:15, widget // for nodesets 13-15 and "widget"
  accelx = 0.3 // constrain the x-acceleration
  function=1 // varying in time with function 1
  disp0 = 0.1 // and initial condition
  vel0 = 0.2 // disp0 = 0.1*0.3 and vel0 = 0.2*0.3

sideset 5 // for nodes in sideset 5
  absorbing // use an absorbing bc

sideset 2 // for the acoustic sideset 2
  p = 0 // fixed acoustic pressure
    // (pressure release condition)

sideset acoustic_surface // for sideset "acoustic_surface"
  pdot = 1.0 // constrain the time derivative
    // of acoustic pressure for
    // enforced accelerations
  function = 2 // varying in time with function 2
  p0=1.0 // and initial condition p0 = 1.0

sideset 6 // for sideset 6
  impedance_pressure=0.5 // use a pressure impedance bc
  impedance_shear = 0.5 // and a shear impedance bc
```
sideset 7  // for sideset 7
  slosh = 0.6  // use a slosh bc

sideset 8  // for sideset 8
  infinite_element  // use infinite elements
  use block my_block  // where the infinite element
    // parameters are in the
    // user-defined block "my_block"

sideset piezoelectric_side  // for "piezoelectric_side"
  V = 0  // fixed voltage
    // (electrical ground)

block fixed_block  // for nodes in block "fixed_block"
  fixed  // constrain all dofs

node_list_file='nodes.txt'  // for nodes in the file "nodes.txt"
  fixed  // constrain all dofs
end

### Input 6.1. Example boundary section

The descriptors for the displacement boundary conditions are, \( X \), \( Y \), \( Z \), RotX, RotY, RotZ, \( P \), and fixed. Their application and meaning are listed in Table 6-115. An optional equals sign separates each descriptor from the prescribed value. The value fixed implies a prescribed value of zero for all degrees of freedom.

Note however that the syntax checking in the boundary block does not check for duplicate boundary conditions, and silent failures are possible. For example in the example shown in input 6.2, part of the input block is silently ignored.

```
Boundary
  nodeset 10  // This
    rotx = 0  // is
    roty = 0  // parsed
    rotz = 0  // as
  nodeset 10  // expected
    accelx=370.
    function=1
    accely=380.  // Parsing
    function=2  // silently
    accelz=390.  // ignores
    function=3  // these!
```
<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>X Component of displacement</td>
</tr>
<tr>
<td>Y</td>
<td>Y Component of displacement</td>
</tr>
<tr>
<td>Z</td>
<td>Z Component of displacement</td>
</tr>
<tr>
<td>RotX</td>
<td>Component of Rotation about X axis</td>
</tr>
<tr>
<td>RotY</td>
<td>Component of Rotation about Y axis</td>
</tr>
<tr>
<td>RotZ</td>
<td>Component of Rotation about Z axis</td>
</tr>
<tr>
<td>fixed</td>
<td>Constrain all components of rotation and translation</td>
</tr>
<tr>
<td>P</td>
<td>Acoustic pressure</td>
</tr>
<tr>
<td>V</td>
<td>Voltage</td>
</tr>
</tbody>
</table>

**prescribed displacement keywords**

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>AccelX</td>
<td>scaling factor on X component of motion</td>
</tr>
<tr>
<td>AccelY</td>
<td>scaling factor on Y component of motion</td>
</tr>
<tr>
<td>AccelZ</td>
<td>scaling factor on Z component of motion</td>
</tr>
<tr>
<td>RotAccelX</td>
<td>scaling of rotational motion about X axis</td>
</tr>
<tr>
<td>RotAccelY</td>
<td>scaling of rotational motion about Y axis</td>
</tr>
<tr>
<td>RotAccelZ</td>
<td>scaling of rotational motion about Z axis</td>
</tr>
<tr>
<td>AccelV</td>
<td>second derivative of voltage</td>
</tr>
<tr>
<td>disp0</td>
<td>initial displacement</td>
</tr>
<tr>
<td>vel0</td>
<td>initial velocity</td>
</tr>
<tr>
<td>Pdot</td>
<td>derivative of acoustic pressure</td>
</tr>
<tr>
<td>P0</td>
<td>initial acoustic pressure</td>
</tr>
</tbody>
</table>

**prescribed acceleration keywords**

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DispX</td>
<td>scaling factor on X component of motion</td>
</tr>
<tr>
<td>DispY</td>
<td>scaling factor on Y component of motion</td>
</tr>
<tr>
<td>DispZ</td>
<td>scaling factor on Z component of motion</td>
</tr>
<tr>
<td>RotDispX</td>
<td>scaling of rotational motion about X axis</td>
</tr>
<tr>
<td>RotDispY</td>
<td>scaling of rotational motion about Y axis</td>
</tr>
<tr>
<td>RotDispZ</td>
<td>scaling of rotational motion about Z axis</td>
</tr>
<tr>
<td>FreqV</td>
<td>scaling factor on voltage</td>
</tr>
<tr>
<td>FreqP</td>
<td>scaling factor on acoustic pressure</td>
</tr>
</tbody>
</table>

**prescribed displacement keywords (directfrf-only)**

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Table 6-115.** – Dirichlet Boundary Enforcement Keywords.
Input 6.2. Example Silent Failure of Parsing

In order to apply functions 2 and 3, it is necessary to provide a nodeset for each function as in input 6.3.

<table>
<thead>
<tr>
<th>Boundary</th>
</tr>
</thead>
<tbody>
<tr>
<td>nodeset 10</td>
</tr>
<tr>
<td>rotx = 0</td>
</tr>
<tr>
<td>roty = 0</td>
</tr>
<tr>
<td>rotz = 0</td>
</tr>
<tr>
<td>nodeset 10</td>
</tr>
<tr>
<td>accelx = 370.</td>
</tr>
<tr>
<td>function = 1</td>
</tr>
<tr>
<td>nodeset 10</td>
</tr>
<tr>
<td>accely = 380.</td>
</tr>
<tr>
<td>function = 2</td>
</tr>
<tr>
<td>nodeset 10</td>
</tr>
<tr>
<td>accelz = 390.</td>
</tr>
<tr>
<td>function = 3</td>
</tr>
<tr>
<td>enD</td>
</tr>
</tbody>
</table>

Input 6.3. Corrected syntax

The way that the parser works for the boundary block is that the text is divided up into consecutive chunks by the keywords nodeset, sideset, block, node_list_file, and end. This is the pattern followed in input 6.1. Within each chunk, any number of Dirichlet boundary enforcement keywords (see Table 6-115) may be provided. Surprisingly, only the first function in a chunk is parsed; any others are ignored. There is no warning for ignored text in the boundary block at this time.

6.1.1. Prescribed Displacements and Pressures

In linear statics, one may prescribe a nonzero displacement by entering a value following the coordinate direction, as shown in input 6.4.

<table>
<thead>
<tr>
<th>Boundary</th>
</tr>
</thead>
<tbody>
<tr>
<td>nodeset 1</td>
</tr>
<tr>
<td>x = 3</td>
</tr>
<tr>
<td>y = 0</td>
</tr>
</tbody>
</table>
Input 6.4. Prescribed Displacement for Statics

For acoustics, pressures may be fixed by specifying \( p = 0 \), as in Table 6-115 on sideset 2. This corresponds to a pressure release condition.

For linear statics, there must be no \textbf{function} entry following the entry. Prescribed displacements have the same limitations as prescribed accelerations described in the next section. The load in this case is introduced by the prescribed displacement. However, the \textbf{loads} section must exist (for error checking purposes) even if it is empty.

6.1.2. Prescribed Voltage

For electro-mechanical coupled physics problems, constant voltage boundary conditions may be specified on nodesets or sidesets using the keyword \textit{V} in the Boundary block. In the following example, electrical grounds were set at sideset 1 and nodeset 1, and a non-zero constant voltage boundary condition set at sideset 2.

```
Boundary
  sideset 1
  V = 0
  nodeset 1
  V = 0
  sideset 2
  V = 2
end
```

Input 6.5. Prescribed Voltage

6.1.3. Prescribed Accelerations

In transient dynamics, the acceleration on a portion of the model may be prescribed as a function of time. As shown in Table 6-115, acceleration are specified using \texttt{accelX}, \texttt{accelY}, \texttt{accelZ}, \texttt{RotaccelX}, \texttt{RotaccelY}, \texttt{RotaccelZ}, \texttt{disp0}, \texttt{vel0}, \texttt{Pdot} and \texttt{accelV}
A function must be used to apply the time-dependent boundary accelerations. Optional initial displacement and velocity can also be specified; if not, they default to 0. In the example above, the \( x \) acceleration of nodesets 13-15 and “widget” will be prescribed as \( 0.3 \times f(t) \), where \( f(t) \) is defined in function 1. The \( \text{accelx} \) factor also scales the initial displacement and velocity. Thus initial displacement is given as 0.1\( \times 0.3 \) and the initial velocity is 0.2\( \times 0.3 \).

Prescribed accelerations are ultimately enforced in the code by integrating to produce a prescribed displacement as shown in Equation 6.1. The \textit{start time of the function}, \( t_{\text{start}} \), is not the start time of the analysis. Accounting for this is important in hand-off analyses. A function is required. Not listing a function will generate an error message.

\[
u(t) = \text{scale factor} \times \left[ \int_{t_{\text{start}}}^{t} \left( \int_{t_{\text{start}}}^{t} f(t) \, dt \right) \, dt \right] + (t - t_{\text{start}}) \times v_{0} + u_{0}
\] (6.1)

In the case of an acoustic sideset or nodeset, the prescribed value is the first time derivative of acoustic pressure, denoted above as \( \text{Pdot} \). This is because, internally, \textit{Sierra/SD} solves for the velocity potential, and the first time derivative of the velocity potential is the acoustic pressure. Thus, by specifying the first time derivative of pressure, one is prescribing the acceleration of the velocity potential.

In the case of a prescribed voltage time history, the prescribed value is the second time derivative of the voltage, i.e. voltage acceleration (\( \text{accelV} \)). However, since the first and second time derivatives of voltage do not contribute to the equations of motion (see theory reference for details), \textit{Sierra/SD} also permits direct specification of the voltage time histories. A voltage time history may be directly defined by the function following the keyword \textit{transV}.

An additional point to consider when applying prescribed accelerations is that the initial velocity and displacement (denoted as \( \text{disp0} \) and \( \text{vel0} \)), are also necessary to completely define the boundary condition. These values account for the constants of integration obtained when integrating the prescribed acceleration to obtain the corresponding velocity and displacement on the sideset or nodeset. In the case of acoustics, only one initial condition is needed ( \( p_{0} \) which specifies the initial acoustic pressure), since only the first time derivative of acoustic pressure is specified. In the case of prescribed voltage acceleration, the descriptors \( \text{disp0} \) and \( \text{vel0} \) are used to define, respectively, the initial voltage and initial time derivative of voltage. By default \( \text{disp0} \), \( \text{vel0} \), and \( p_{0} \) vanish.

There are some limitations with the prescribed acceleration capability, which are listed in the following, and in Table 6-116. First, prescribed accelerations are not currently set up to work with multicase solutions. Also, they only work in the standard (Cartesian) coordinate system. Prescribed accelerations can be used in meshes that have nonlinear or viscoelastic elements, as long as the prescribed accelerations are not applied directly to the nonlinear or viscoelastic elements. Note that the nodes involved in prescribed accelerations cannot coincide with nodes that are involved with MPCs.
Finally, note that when prescribed accelerations are used, they induce a load on the structure. Thus, in many cases the loads section serves no purpose, unless an additional external load is applied. In these cases, however, an empty loads block is still needed in the input file. An error message will be generated if the input file has no loads section.

1. No support for multcase.
2. Only in basic coordinate directions.
3. Cannot be used on nodes attached to viscoelastic elements.
4. Cannot be used on nodes attached to nonlinear elements.
5. Cannot be used on nodes connected to rigid elements or MPCs.
6. Load section is required, even if empty.

Table 6-116. – Limitations for Prescribed Acceleration Boundary Conditions.

6.1.4. Prescribed Frequency-Varying Displacements

For the direct frequency response solution method, a portion of the model may be prescribed displacements as a function of frequency. As shown in Table 6-115 the descriptors for prescribed frequency dependent displacements are DispX, DispY, DispZ, RotDispX, RotDispY, RotDispZ, FreqP, FreqV. A function must be used to apply the frequency dependent boundary condition.

6.1.5. Node_List_File

To make it easier to apply boundary conditions, a node_list_file option is provided. In this option, the user provides an additional text file that contains a list of global node ids separated by white space. No comments, or other characters are allowed in the file, as shown in input 6.1. The remainder of the boundary condition specifications are unchanged.

There are several limitations place on collections of nodes specified in this manner.

1. This is an inefficient method of supplying the nodes. It is recommended that nodesets or sidesets be employed when practical.
2. No node distribution factors may be provided.
3. The output Exodus file will have no record of this list.
4. The global node numbers are the mapped Exodus global numbers. This is the arbitrary node numbering provided by the analysts, and may not correspond to the 1 to N ordering used by some other programs. 4

5. There is NO requirement that the nodes be sorted in the list, but repeating a node in the list can have undefined results, i.e. don’t do it.

6.1.6. Nonreflecting Boundaries

Nonreflecting boundary conditions for acoustics and for elasticity may be specified using the “absorbing” keyword.

This section allows the user to specify an exterior boundary for acoustic, elastic, or coupled structural acoustic simulations. Once specified, first-order non-reflecting boundary conditions are applied on this surface. The boundary is specified with a sideset. The sideset can be placed either on acoustic or elastic elements. The code automatically determines whether the sideset is placed on acoustic or elastic elements, and then applies the appropriate boundary conditions.

Only pressure waves need to be absorbed for acoustic elements, and the absorbing boundary could represent an infinite fluid surrounding a structure. For elastic waves both pressure and shear waves need to be absorbed, and the absorbing boundary could represent an infinite elastic medium, such as in a seismic problem.

An example of this syntax is given below.

```
Boundary
  sideset 5
    absorbing
    radius = 1.0
end
```

The parameter “radius” specifies the radius of the sphere that defines the absorbing boundary. For a planar absorbing surface, one can either specify no radius, or a large radius (the radius is equal to infinity for a planar surface). In those cases, the absorbing boundary condition reduces to a plane-wave absorbing condition. We also note that the radius parameter refers to the distance from points on the spherical surface to the center of curvature, not to the origin of the coordinate system. Thus, it is independent of the coordinate system that is specified. For example, one could shift the coordinates of the nodes of the acoustic mesh by any constant, but the radius parameter would remain the same.

4Earlier versions of Sierra/SD used the 1 to N ordering, where N is the maximum number of nodes in the model. Current versions always use the mapped ordering for node references.
6.1.7. Impedance Boundary Conditions

Impedance boundary conditions are partially reflecting and partially absorbing. Thus, they are somewhere in-between a rigid wall and an absorbing boundary condition. They reduce to these special cases for certain choices of the impedance parameters.

An example syntax for an absorbing boundary condition is given below

```
Boundary
    sideset 6 // sideset on acoustic material
    impedance = 0.5
    sideset 7 // sideset on elastic material
    impedance_pressure = 0.5
    impedance_shear = 0.5
end
```

In this case, sideset 6 is attached to acoustic elements, and sideset 7 is attached to elasticity elements. For acoustic elements, only one impedance parameter is needed, and it corresponds to an impedance condition for pressure waves only (acoustic elements support no shear waves). For elasticity elements, the `impedance_pressure` and `impedance_shear` correspond to impedance for pressure and shear waves, respectively.

This example specifies that sideset 6 is to have an impedance of $Z = 0.5\rho c$, where $\rho$ is the density and $c$ is the speed of sound. Thus, the “impedance” parameter that is parsed in is the multiplier on the characteristic impedance $\rho c$. Similarly, for the elasticity element the pressure and shear impedance would be $Z_P = 0.5\rho c_P$ and $Z_S = 0.5\rho c_S$, where $c_P$ and $c_S$ are the speeds of sound for the pressure and shear waves, respectively.

Currently, impedance boundaries are only set up to work with the standard characteristic impedance $\rho c$. Thus, specifying the “radius” parameter with an impedance boundary condition will have no effect.

We note that if the impedance parameters are all set to 1.0, the problem reduces to the absorbing boundary described in the previous section. If set to 0, the impedance condition becomes a pressure-release boundary for acoustics and a free boundary for an elasticity element. If set to a large number, the impedance boundary condition reduces to a rigid-wall condition for acoustics, and a fixed condition for elasticity elements.

6.1.8. Slosh

Slosh boundary conditions are applied at free surfaces that are effected by gravity. This type of free surface is typically only important on the surface of a liquid such as water. It contributes to the mass matrix, resulting in “surface” wave modes.

An example syntax for an absorbing boundary condition is given below
This specifies that sideset 7 is to have a slosh boundary condition. In this case, the slosh coefficient needs to be set to \( \frac{1}{g} \), where \( g \) is the gravity constant. Thus, for SI units, the slosh coefficient is 0.102. Currently, slosh boundary conditions are only valid for acoustic elements. Applying them to elastic elements will generate an error.

### 6.1.9. Infinite Elements

In this section, we describe how to use infinite elements for acoustics. These elements serve as both high-order absorbing boundary conditions, and far-field calculators that allow the analyst to compute the solution at far-field points outside of the acoustic mesh. This latter step is a post-processing step.

The infinite element specification begins with a sideset on the *Exodus* file of interest. Currently, that sideset has to be an ellipsoidal surface or part of an ellipsoidal surface. Thus, a full spherical surface, hemispherical surface, or a quarter of a sphere would all be acceptable. Infinite element accuracy will degrade if the element surfaces on the boundary do not adequately represent the ellipsoidal surface. The finite element surfaces will be faceted, but enough elements on the boundary are needed to represent the ellipsoidal curvature.

Once a sideset is identified for the infinite element surface, the `boundary` section in the input deck would be modified as follows.

```plaintext
Boundary
  sideset 1
    infinite_element
    use block 57
end

BLOCK 57
  infinite_element
  radial_poly = Legendre
  order = 5
  source_origin = 0 0 0
  ellipsoid_dimensions 15 15 30
  neglect_mass = yes
END
```

where block 57 contains the infinite element parameters. The number 57 is arbitrary; the user can pick any number (or name) that is not assigned to a block in the input mesh
(Exodus) file. The parameters are summarized in Table 6-117. Currently, only Legendre polynomials are available for the radial basis. The order of the polynomial can vary from 0 to 19. Order 0 corresponds to a simple absorbing boundary condition. Higher orders will be more accurate, but also more computationally expensive. The source point is the location of the center of the ellipsoid that the infinite elements emanate from.

The ellipsoid_dimensions parameters indicate the axial dimensions of the ellipsoid in the global coordinate system. They are specified as ellipsoid radii instead of ellipsoid diameters. In the case of a sphere, all 3 parameters are equal and the radius of the sphere. These parameters are currently required, and an error will be generated if they are not specified.

The neglect_mass keyword indicates whether to neglect the mass matrix contributions from the infinite elements. By default, neglect_mass is yes. Note that for a spherical surface, the mass matrix contributions from an infinite element are identically zero. However, when numerically generated, small entries will be present in the mass matrix, and thus an option is provided to include these terms in the analysis. Neglecting the mass, yes, is recommended in most cases.

Infinite elements only require a specification of a sideset on the surface of interest. No elements need be set up explicitly on this interface. Internally, Sierra/SD constructs virtual elements and virtual nodes that define the actual infinite elements, but the analyst need not build a layer of elements on the boundary of the sideset.

The infinite element formulation in Sierra/SD uses a Petrov-Galerkin formulation, instead of a standard Galerkin formulation. As a result, nonsymmetric system matrices are encountered with infinite elements. This restricts the solver options to the GDSW solver for time and frequency domains (i.e. directfrf). Infinite elements can be used either with purely acoustic problems, or with coupled structural acoustics. The formulation is the same, and the GDSW solver is required for the solutions since nonsymmetric matrices are encountered.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Options</th>
<th>default</th>
</tr>
</thead>
<tbody>
<tr>
<td>radial_poly</td>
<td>the type of polynomial for radial expansion</td>
<td>Lagrange, Legendre, Jacobi</td>
<td>Legendre</td>
</tr>
<tr>
<td>order</td>
<td>the order of the radial basis</td>
<td>0-19</td>
<td>0</td>
</tr>
<tr>
<td>source_origin</td>
<td>the origin of the ellipsoid</td>
<td>3 real numbers</td>
<td>0 0 0</td>
</tr>
<tr>
<td>ellipsoid_dimensions</td>
<td>radial dimensions of ellipsoid axes</td>
<td>3 real numbers</td>
<td>0 0 0</td>
</tr>
<tr>
<td>neglect_mass</td>
<td>indicates whether to neglect infinite element mass</td>
<td>yes or no</td>
<td>yes</td>
</tr>
<tr>
<td>correct_mass</td>
<td>whether to correct negative mass terms.</td>
<td>yes or no</td>
<td>yes</td>
</tr>
<tr>
<td>useplanelineintersectmethod</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 6-117. – Available parameters for the infinite element section.
6.1.9.1. Far-Field Postprocessing  
The infinite element formulation allows the analyst to compute the response outside of the acoustic mesh as a post processing step. The response can be computed at any point outside the mesh, and for any time interval. Currently, the linesample capability is used to write out the far-field data (see Section 7.5). This data may be written in a readable MATLAB format, which can easily be read in to create plots of the data.

The output will be written to a MATLAB m-file with the name “linedata.m” or “linedata.exo”, depending on which option is selected for output. One file is written per analysis (results are joined analogous to history file output). For example, reading this file in will create vectors FieldTime and displacement. The acoustic pressure is found in displacement1.

We note that the infinite element output in the far-field is always given with respect to some time shift. Details of this are given in the theory notes on infinite elements. The shifted times are included in the linesample output for the analyst to use. These allow for plotting the time histories against the appropriate time vectors.

The shifted time output is available in the linesample output in a nodal array called FieldTime. The dimension of the FieldTime array is the same dimension as the acoustic pressure output, since each node in the linesample output has its own FieldTime array. One FieldTime array is available for each sample point in the linesample output.

The following command in MATLAB will plot the pressure for the first sample point.

\[
\begin{align*}
\text{FieldTime} &= \text{nvar09}; \\
\text{pressure} &= \text{nvar01}; \\
\text{plot} (\text{FieldTime}(1,:), \text{pressure}(1,:))
\end{align*}
\]

The linesample points defined in the linesample file can contain points that are both inside and outside of the acoustic mesh. For points that are inside of the mesh, the FieldTime array for each node will be identically equal to the time array. For points outside of the acoustic mesh (i.e. inside of the infinite element mesh), the FieldTime values will be larger than the corresponding time values in the Time array, since the acoustic waves will take additional time to reach these far-field points.

6.1.10. Perfectly Matched Layers

Perfectly Matched Layer (PML) elements enforce acoustic baffle boundary conditions. A detailed explanation of theory and implementation of our PML formulation is available. These elements serve as an absorbing boundary condition for outgoing acoustic waves, much like infinite elements. Unlike infinite elements, they are linear elements, and do not exhibit the large matrix condition numbers and convergence issues that can accompany infinite elements. Table 6-118 summarizes the parameters.
While PML are a separate block of elements in the finite element boundary, in an input file they are treated like a boundary condition. The boundary section is set up as follows:

```plaintext
Boundary
  sideset 1
  pml_element
  use block 217
end
```

```plaintext
Block 217
  pml_element
  pml_thickness 1
  stack_depth 1
  source_origin = 0 0 0
  ellipsoid_dimensions 15 15 30
  loss_function = polynomial
  loss_params 0 960 960 0
end
```

where block 217 contains the PML parameters. The number 217 is arbitrary; the user can pick any number (or name) that is not assigned to a block in the input mesh (Exodus) file. The output mesh will contain PML elements in block 217.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Options</th>
</tr>
</thead>
<tbody>
<tr>
<td>pml_thickness</td>
<td>length of PML extrusion from boundary</td>
<td>Real</td>
</tr>
<tr>
<td>stack_depth</td>
<td>number of elements through PML thickness</td>
<td>Integer</td>
</tr>
<tr>
<td>source_origin</td>
<td>the origin of the ellipsoid</td>
<td>3 real numbers</td>
</tr>
<tr>
<td>ellipsoid_dimensions</td>
<td>radial dimensions of ellipsoid</td>
<td>3 real numbers</td>
</tr>
<tr>
<td>loss_function</td>
<td>type of function describing PML decay</td>
<td>singular or polynomial</td>
</tr>
<tr>
<td>loss_params</td>
<td>constants in function describing PML decay</td>
<td>4 real numbers</td>
</tr>
</tbody>
</table>

For PML, a loss function is a definition of the rate of decay of the outgoing wave. While the choice of the loss function $\sigma(d)$ is discussed in the literature, papers in the literature use a range of formulations and implementations, and it is still unclear what the best choice is for any given problem. Typically, the loss function starts at a low value (often zero) to minimize numerical reflections, and increases at an increasing rate to maximize the loss terms near the outer boundary. One option is the polynomial loss function, that includes the constant, linear, quadratic, and cubic terms, with four parameters that define the loss function

$$
\sigma(\xi) = c_1 + c_2 \frac{\xi}{l} + c_3 \frac{\xi^2}{l^2} + c_4 \frac{\xi^3}{l^3}
$$

(6.2)
where \( c_1, c_2, c_3, \) and \( c_4 \) are specified in the input file, \( \xi \) is the distance along the normal from the Gauss point to the inner ellipsoid boundary, and \( t \) is the total thickness of the PML layer. The loss function is normalized such that changing the thickness of the PML layer does not change the maximum value of \( \sigma \). Another option is the singular loss function\(^{11} \) (equation (6.3)), which is unbounded at the outer boundary.

\[
\sigma (\xi) = \frac{c_1}{t - \xi}
\] (6.3)

PML elements can be extruded from either Tet4 or Hex8 meshes. Note that Tet4 is the default; for PML elements extruded from Hex8 meshes, it is necessary to specify Hex by modifying the boundary section as follows:

\[
\text{Boundary} \\
\text{  sideset 1} \\
\text{  pml_element} \\
\text{  use block 217} \\
\text{  hex} \\
\text{end}
\]

6.1.10.1. Limitations PML is only supported for certain formulations and element types. The implemented PML formulation only applies in the frequency domain, and will throw a fatal error for eigen or time domain solutions. Additionally, PML is only supported for three and four noded boundary faces, limiting the exterior of the acoustic domains to Tet4, Wedge6, and Hex8 elements.

6.1.11. Periodic Boundary Conditions

Periodic boundary conditions can be applied in Sierra-SD using begin-periodic block. Built on the tied data algorithm, periodic boundary conditions are currently supported for structural/solid surfaces, with the displacement on the side B surface specified relative to the displacement of the side A surface. The A and B surfaces within a begin-periodic block may not share nodes, but surfaces across various begin-periodic blocks can share nodes (e.g. standard representative volume element (RVE) model would have three begin-periodic blocks, with the surfaces in each block sharing nodes with surfaces in other blocks, at the RVE edges). The surfaces can be curved, but the user must ensure that they have the same geometry, translated in space. Meshes on the two surfaces need not match. Matching meshes are however recommended whenever feasible, to ensure accuracy of the computed stresses on the surfaces. See discussion in Tied Surfaces section above regarding inaccuracies in stresses at the tied surfaces with mismatched meshes. The underlying algorithm results in non-homogeneous MPCs from a node-to-face constraint algorithm. Naturally, for matching meshes, these MPCs reduce to node-to-node MPCs.

Like a tied data block, each begin-periodic block represents a single pair of opposing sides connected by periodic boundary conditions, as shown in the example below.
In defining surfaces, care must be exercised to ensure that the normal vectors of the two surfaces point towards each other. The keyword “geometric offset” represents the distance from the A surface to the B surface in x, y and z directions. Similar to the TIED DATA block, the keyword “search tolerance” represents the normal distance from a node on the B surface, shifted by the geometric offset, to the face on the A surface. See Figure 8-64 for further details. The keywords “Ux,” “Uy” and “Uz” represent the displacement of the B surface relative the A surface in x, y and z directions respectively. The description of all the parameters are shown in Table 6-119.

6.1.12. Usage Guidelines

The user is referred to the verification manual for examples of periodic boundary conditions(PBC), one with unidirectional PBC and the other with multi-directional PBC, and their application to simulating periodic volume elements (PVE). Here, we provide some guidelines and cautionary remarks associated with PBC and PVE.

1. Matching Meshes: The two sidesets that are connected by PBC do not need to have matching meshes, but are recommended whenever feasible. Such matching meshes can be built using various tricks at the mesh-generation stage. For example, for symmetric PVEs (with deterministic microstructure), one could split the PVE using plane(s) of symmetry, meshing one part and mirroring the meshed volume to get the complementary volume, eventually building the mesh for the overall volume with matching meshes on the opposite side.

2. Non-matching Meshes: When the microstructure is random or lacks symmetry, PBC may need to be applied to connect non-matching meshes. Such situations may encounter
local oscillations in stresses on the surface, associated with the underlying node-to-face contact strategy. These oscillations are expected to decay quickly going into the volume, making the volumetric average more accurate. Since micromechanics modeling often involves average stresses, the error in homogenized global material properties is expected not to be significant provided that the representative volume element (RVE) is large enough (which will also be a requirement from the standpoint of representing random microstructure).

3. Geometric Offset: Theoretically, geometric offset should not be needed in the periodic-boundary block. However, whenever the opposite faces have non-matching meshes, there can be errors in automatic computation of offset, which involves centroid computation of the sideset that may involve discretization errors. Given this, whenever feasible, it is recommended that the geometric offset be provided in the input. Note that this comment applies only to non-matching meshes; automatic computation of geometric offset would not have any errors for matching meshes.

4. Imposition of Homogenized Strain: The user is referred to the verification manual (problem on PVE) for details of imposing global strain tensor through multi-directional PVE.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>type</th>
<th>description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Name</td>
<td>String</td>
<td>name of periodic boundary condition block</td>
</tr>
<tr>
<td></td>
<td></td>
<td>defaults to periodicBC</td>
</tr>
<tr>
<td>side A</td>
<td>Integer</td>
<td>sideset A id</td>
</tr>
<tr>
<td>side B</td>
<td>Integer</td>
<td>sideset B id</td>
</tr>
<tr>
<td>Geometric Offset</td>
<td>3 Reals</td>
<td>x,y,z components of B sideset location relative to A sideset</td>
</tr>
<tr>
<td></td>
<td></td>
<td>automatically computed if not explicitly specified</td>
</tr>
<tr>
<td></td>
<td></td>
<td>(explicit specification is recommended for non-matching meshes)</td>
</tr>
<tr>
<td>Search Tolerance</td>
<td>Real</td>
<td>search tolerance normal to the face</td>
</tr>
<tr>
<td></td>
<td></td>
<td>defaults to 1e-8</td>
</tr>
<tr>
<td>Ux</td>
<td>Real</td>
<td>x-displacement of B sideset relative to A sideset</td>
</tr>
<tr>
<td></td>
<td></td>
<td>default value is 0.0</td>
</tr>
<tr>
<td>Uy</td>
<td>Real</td>
<td>y-displacement of B sideset relative to A sideset</td>
</tr>
<tr>
<td></td>
<td></td>
<td>default value is 0.0</td>
</tr>
<tr>
<td>Uz</td>
<td>Real</td>
<td>z-displacement of B sideset relative to A sideset</td>
</tr>
<tr>
<td></td>
<td></td>
<td>default value is 0.0</td>
</tr>
</tbody>
</table>

6.2. Exodus Mesh Boundary Condition Input

Several boundary conditions may be set to values specified in the mesh geometry file or input sources. This is used to hand-off loads an earlier finite element analysis, even a previous Sierra/SD simulation. The pressure determined in another code, and output to a
sideset and can put input on the sideset, determining the right-hand side in a Sierr/SD analysis.

The Exodus file input data is defined at one or more time slices that do not necessarily correspond to Sierr/SD time steps. Figure 6-57 summarizes the rules for interpolating the input time slides to Sierr/SD time steps.

The value of a boundary condition at a time …

- … before any input Exodus time steps is the initial input value.
- … between two input Exodus time steps is determined by linear interpolation.
- … after the last input Exodus time step is the last input Exodus time step.

An example of interpolating Exodus data do Sierr/SD time steps is shown in Figure 6-57.

![Figure 6-57. – Example of Interpolation of Exodus Data to Analysis Steps.](image)

6.2.1. SpatialBC Functions

This appears to be a capability that was added for a specific purpose that was mistakenly exposed to users.

The Spatial boundary condition function is used to set a boundary conditions from the input Exodus mesh geometry file. It resembles the randomlib function 2.8.8. The difference is that the randomlib function uses a nodeset associated with the specified sideset. With Spatial boundary condition, the nodeset is specified directly.
The variable input from the Exodus file is specified with the parameter `exo_var`, followed by either `scalar` or `vector`, sets the variable input from the Exodus.

In the example the input scalar is acceleration in the Z direction:

```plaintext
boundary
  nodeset NS_top
  function from_mesh
  accelz = 1
end
function from_mesh
  type = spatialBC
  nodeset NS_top
  exo_var scalar Acc_Z
end
```

Here “Acc_Z” is the exact name of a field on the Exodus nodeset. Exodus field names are case insensitive.

Sierra/SD has only a couple regression tests of SpatialBC, and each test uses exactly the same boundary condition. All of them result in a confusing and misleading warning about missing sidesets. The vector input feature is completely untested.

### 6.2.2. Input an Acoustic Point Source from a Volume

The 3 tests of ReadNodal exercise the capability to input a scalar acceleration from the whole mesh (body) or an element block. In acoustic point source analysis volume velocities may be input from an Exodus file using a ReadNodal function. In theory, the Exodus file contains the first or second derivatives of the volume velocity at the corresponding times. The name of the field input from the Exodus file is specified with the `exo_var` keyword.

Table 6-120 lists the run time parameters for ReadNodal functions. An example is provided in input 6.6. The keyword `exo_var` must be followed by two keywords, specifying first whether the data is a scalar or a vector, and second specifying the name of the variable on the Exodus database. However the vector input feature is untested. Thus, Table 6-120 specifies that a scalar variable with the name `volume_acceleration` should be available on the Exodus database. The `interp` parameter is the same as was described for the randomlib functions in section 2.8.8. It specifies the type of temporal interpolation.

ReadNodal determines what, if anything, to read by accepting the first match found. One after another, it searches for 3 things, and errors out if the last search come up empty. The first search is for a nodal variable with the specified name (i.e., `volume_acceleration` in the example above). Next it looks for an element variable with the same name. Finally it searches for a face variable.
Table 6-120. – ReadNodal function parameters.

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Values</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>type</td>
<td>ReadNodal</td>
<td>required to specify function</td>
</tr>
<tr>
<td>interp</td>
<td></td>
<td>temporal interpolation scheme</td>
</tr>
<tr>
<td></td>
<td>none=nearest</td>
<td>none=nearest</td>
</tr>
<tr>
<td></td>
<td>linear=linear interpolation</td>
<td>linear=linear interpolation</td>
</tr>
<tr>
<td>exo_var</td>
<td>scalar volume_acceleration</td>
<td>either scalar or vector, followed by variable name</td>
</tr>
</tbody>
</table>

Input 6.6. Example ReadNodal Function Specification

6.2.3. Input an Acoustic Point Source from a Node Set

Input volume velocities for acoustic point source analysis from the input Exodus mesh geometry file with the ReadNodalSet function. Velocities are input at each time specified in the Exodus file. Spatially dependent velocities are input at each nodeset node.

Table 6-120 lists run time parameters for ReadNodalSet functions. The only difference from the ReadNodal parameters is the required nodeset parameter. The force is applied over a single nodeset of the model. This nodeset must match the definition in the load section.

Input 6.7. Example ReadNodalSet Function Specification
6.2.4. ReadSurface functions

A ReadSurface function reads in data from either the entire Exodus file, or a nodeset or sideset that covers a surface of interest. If a set is specified in the function block, then data corresponding to that set is read in from the Exodus file. Otherwise, the variable is read from the entire mesh as a nodal variable (rather than a nodeset variable).

Once the data is read, Sierra/SD integrates the data over the surface to create a time and spatially-varying forcing function. One difference between this function and the ReadNodal function is that the data is from ReadSurface is used in a surface integration to generate the load, whereas the data from ReadNodal does not need to be integrated, and thus can be inserted directly into the force vector.

This function is used to read in surface velocities or accelerations which are used as a boundary condition for acoustic analysis. It can also be used for applying time and spatially-dependent pressure or traction loads on a structure. For this case, the load output variable currently only outputs element data for values read from a sideset. Nodal values, like those used in randomlib functions, are output as 0. There is 1 and only 1 test of read surface with a traction load.

As currently implemented, the ReadSurface function operates only by reading data from an external exodus data file. The name of the variable to read from the Exodus file must be specified in the input deck using the exo_var keyword. Also, the variable must be specified to be a scalar or a vector, using the syntax given in input 6.8. Pressures require a scalar variable, and tractions require a vector variable. An example for ReadSurface functions is given in input 6.8. A sideset matching the corresponding Load sideset is required. The interp selected the temporal interpolation algorithm as was described for the randomlib function. The default option, linear interpolation, is the only option available.

In input 6.8, the keyword exo_var specifies the type of data, such as vector or scalar, and the name of that variable in the Exodus file. In the case of a vector, the name of the variable as given in the input deck should be the base name of the variable, without the suffix of ‘x’, ‘y’, or ‘z’. For example, for the data given in input 6.8, a vector nodal variable with a base name of name ‘traction_load’ should be available in the Exodus file. Thus, the data in the Exodus file would have names traction_loadX, traction_loadY, and traction_loadZ. In the case of scalar data, the base name given (i.e. traction_load in input 6.8), should match exactly the name of the nodal variable in the Exodus file.

```
loads
   sideset 1
      traction 1 1 1
   function 55
end
function 55
   type=ReadSurface
   interp=linear
```
Input 6.8. Example ReadSurface Function Specification

ReadSurface functions ignore distribution factors when used to apply loads. See section 6.3.14.1.

6.2.5. ExodusRead functions

A ExodusRead function reads in data from elements. Exodus read is currently only available with the acoustic boundary conditions point_volume_acceleration and point_volume_velocity.

ExodusRead function operates only by reading data from an external exodus data file. The name of the variable to read from the Exodus file must be specified in the input deck using the exo_var keyword. The acoustic load conditions this function type supports are scalars.

Input 6.9. Example ExodusRead Function Specification

6.2.6. In Core Transfer Functions

A limited capability exists to perform in core transfer of variables between physics codes via MPI messages rather than Exodus files. In this use case both codes are run simultaneously with the results from the load producing code constantly being fed to the load using code. The main use case for this capability is to hand-off boundary conditions from one code to another without the need for huge intermediate Exodus files with a large numbers of steps.
6.2.6.1. Transfer from Fuego

One field that may be transferred in core is the divergence of Lighthill’s tensor from Fuego to Sierra/SD. This is a nodal value that can be used as an acoustics load term. An example of the relevant input for this case is:

```plaintext
PARAMETERS
    mpmd_transfer_type = fuego
END
LOADS
    nodeset = 1000
    lighthill = 1.0  #Sets load type and scale factor
    function = from_fuego
END
FUNCTION from_fuego
    type transfer
END
```

An example of the job execution syntax for this case is:

```
$ mpirun -n 1 fuego -i Fuego.i : -n 1 salinas -i Salinas.inp
```

6.2.6.2. Transfer from SPARC

Another field that may be transferred in core is the traction load from SPARC to Sierra/SD. This is a face-based term that can be used for structural loads. An example of the relevant input for this case is:

```plaintext
PARAMETERS
    mpmd_transfer_type = SPARC
    mpmd_transfer_sidesets = 5
END
LOADS
    sideset 5
    traction = 1.0 1.0 1.0  #Sets load type and scale factors
    function = from_sparc
END
FUNCTION from_sparc
    type transfer
END
```

An example of the job execution syntax for this case is:

```
$ mpirun -n 1 sparc -i sparc.i -c 0 : -n 1 salinas -i Salinas.inp
```
6.3. Loads

Loading conditions are specified within the loads section. The following example illustrates the method.

```plaintext
Loads
  nodeset 3
    force = 1.0 0. 0.
    scale = 1000.
    function = my_load_function
  nodeset nose
    coordinate 11
    force = 0. -1 0
  nodeset 7
    point_volume_vel = 1
    scale = 1.0
    function = 1 // time history of dV/dt,
    // where V is the volume of the source
  body
    gravity
    0.0 1.0 0
    scale -32.2
  block wing
    thermal_load
    function = 1
  block 12
    thermal_load
    function = 3
  sideset 7
    pressure 15.0
  sideset tail
    traction = 100.0 20.0 0.0
    coordinate 0
  sideset 13
    acoustic_vel 1.0
    function = 1
  sideset 14
    pressure = 1
    follower=yes
    node_list_file='force.nodes'
    force=1.0 0 0.
    scale = 100.
    function=2
END
```
Loads may be applied to node sets, side sets, blocks, node lists (see Section 6.1.5), or the entire body (in the case of inertial loads). Pressure loads may be applied using side sets. The pressure is always normal to the surface. All loads applications are additive. Forces should not be applied to sidesets.

The components of each load specification are listed in Table 6-121. The syntax followed is to first define the region over which the load is to be applied (either nodeset, sideset, block, node_list_file, or body). Each such region defines a load set. For each such definition, one (and only one) load type may be specified. However, any region definition (except node_list_file) may be repeated so that forces and moments may be applied using the same region.

Following the definition of the load type, a vector (or scalar in the case of pressure loads) must be specified, except in the case of a thermal load, where no vector or scalar multiplier is needed. The vector is the load applied in the basic coordinate frame unless a coordinate frame is also specified (see Section 2.7).

6.3.1. Load

Loading conditions for individual cases in a multicase solution are specified within the load section. See Section 3.2 for information on load specifications for multicase solutions. The load section is identical to the loads section described in the previous section (6.3), except the section begins with the keyword load, and a load step identifier is required. The following example illustrates the required input.

```plaintext
LOAD=57
  nodeset 3
    force = 1.0 0. 0.
    scale = 1000.
    function = 2
  nodeset 5
    force = 0. -1 0
END
```

Unlike the loads section, there may be multiple load sections in the file. Each load can then be applied to different cases in the multicase solution by calling out the load identifier in the case block of the multicase solution, as described in Section 3.2.

6.3.2. Scale Factors for the Load

The total load on each degree of freedom is the product of the load vector, the scale factor, and the nodeset distribution factor found in the Exodus file. For pressures and tractions, the load is also multiplied by the area of the face. Note that in some cases the nodeset

---

5The identifier is usually an integer, but any unique string is acceptable.
distribution factor may be zero. In that case, the total applied force will also be zero. The Exodus distribution factors can be ignored by setting the parameter AllowExodusDistFacts to false.

6.3.3. Sideset Loading

The pressure, acoustic_vel, and acoustic_accel boundary conditions may only be applied to side sets. The total pressure is the product of the scale factor, pressure (scalar) and sideset distribution factors. By default, pressure loads are not follower loads, i.e. pressures are applied in the direction of the undeformed element normal for the entire simulation. The follower keyword may be applied to user defined functions if a follower load is required. See section 6.3.6 for follower stiffness specification.

If the pressure loading is not normal to the sideset, the traction capability should be used. NOTE: Pressure will act on a surface in a compressive sense, while a traction can be specified as any vector which will act on the sideset specified in the direction given by the triple values specified after traction. Also, traction loads are applied on the faces of the shell elements in a piecewise manner, i.e., the traction load acting on a face of the element is assumed constant. If the distribution factors on the nodes of the element vary, the average of the load (element per element) is assumed.

Traction loads may be specified in either the global coordinate frame (default), or in a coordinate frame projected onto the surface.

If the analyst provides a coordinate frame with the traction definition, then that frame is projected onto the surface of each element. Figure 6-58 illustrates that projection. Note that when a coordinate system is used, there can be a mesh dependence on which direction the forces are applied. Note that the third coordinate of the traction will always be applied along the surface normal, and that the third component of the vector will always correspond to the surface normal (and hence will be applied as a pressure).

\[\text{distribution factor may be zero.} \]

\[\text{In that case, the total applied force will also be zero. The Exodus distribution factors can be ignored by setting the parameter AllowExodusDistFacts to false.}\]

---

\[\text{Because the nodeset distribution factors are part of the Exodus file, and may be difficult to check, errors in the distribution factors are common. Analysts are urged to carefully examine the distribution factors.}\]
<table>
<thead>
<tr>
<th>Section</th>
<th>Keyword</th>
<th>Parameters</th>
</tr>
</thead>
<tbody>
<tr>
<td>Region</td>
<td>body</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>nodeset</td>
<td>id/name</td>
</tr>
<tr>
<td></td>
<td>sideset</td>
<td>id/name</td>
</tr>
<tr>
<td></td>
<td>block</td>
<td>id/name</td>
</tr>
<tr>
<td></td>
<td>node_list_file</td>
<td>file name</td>
</tr>
<tr>
<td>Load Type</td>
<td>force</td>
<td>val1 val2 val3</td>
</tr>
<tr>
<td></td>
<td>moment</td>
<td>val1 val2 val3</td>
</tr>
<tr>
<td></td>
<td>gravity</td>
<td>val1 val2 val3</td>
</tr>
<tr>
<td></td>
<td>pressure</td>
<td>val1 val2 val3</td>
</tr>
<tr>
<td></td>
<td>point_volume_vel</td>
<td>value</td>
</tr>
<tr>
<td></td>
<td>point_volume_accel</td>
<td>value</td>
</tr>
<tr>
<td></td>
<td>acoustic_vel</td>
<td>value</td>
</tr>
<tr>
<td></td>
<td>acoustic_accel</td>
<td>value</td>
</tr>
<tr>
<td></td>
<td>Lighthill</td>
<td>value</td>
</tr>
<tr>
<td></td>
<td>surface_charge</td>
<td>value</td>
</tr>
<tr>
<td></td>
<td>traction</td>
<td>value</td>
</tr>
<tr>
<td></td>
<td>thermal_load</td>
<td>value</td>
</tr>
<tr>
<td></td>
<td>energy_load</td>
<td>value</td>
</tr>
<tr>
<td></td>
<td>coordination</td>
<td>id/name</td>
</tr>
<tr>
<td>Coordinate Frame</td>
<td>scale</td>
<td>val1</td>
</tr>
<tr>
<td>(for vector loads only)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Scale Factor Multiplier</td>
<td>function</td>
<td>id</td>
</tr>
<tr>
<td>(Required for transient</td>
<td>follower</td>
<td>yes/no</td>
</tr>
<tr>
<td>analysis)</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 6-121. – Load Specification Keywords.
Consider a user defined coordinate frame, $C_{\text{user}}$ defined by the basis vectors, 

$$(\tilde{X}, \tilde{Y}, \tilde{Z})$$

A surface normal, $n$, is defined by the element normal. The user defined coordinate frame is projected onto the surface as follows.

$$\tilde{Z}' = n$$

$$\tilde{X}' = \hat{Y} \times \tilde{Z}'$$

$$\tilde{Y}' = Z' \times \tilde{X}'$$

This transformation is singular when $\hat{Y} \times n$ is zero. Near that location, the transformation is modified.

$$\tilde{Z}' = n \tilde{Y}'$$

$$\tilde{X}' = \tilde{Y}' \times \tilde{X}$$

When normalized, $(\tilde{X}', \tilde{Y}', \tilde{Z}')$ form the basis for a coordinate frame, $C_p$, on the surface of the element in which to apply the tractions.

**Note:** This transformation is dependent on the direction of the element normal, $n$. If $\hat{Z}$ is in the opposite direction of $n$, the $\hat{Y}$ direction will be preserved, but the $\tilde{X}'$ direction will be the opposite of $\tilde{X}$. This preserves the right-hand rule.

**Figure 6-58.** – Coordinate Frame Projection for Traction

### 6.3.4. Spatial Variation

Variation of the load over space is accomplished using node set or side set distribution factors or a function. If these are provided in the *Exodus* file, the load set is spatially multiplied by these factors. The total loading is the sum of the loads for each load set summed over all the load set regions.

### 6.3.5. Required Section

When prescribed accelerations are applied in the *boundary* section (6.1), they induce a load on the structure. In these cases the *loads* section may serve no purpose, unless an additional external load is applied. In these cases, an empty *loads* block is still needed in the input file. An error is generated if the input file has no *loads* section.

### 6.3.6. Follower Stiffness

The follower stiffness that corresponding to an applied pressure load may be included. A follower pressure load applied to a structure will “follow” the structure during deformation, always remaining normal to the surface where they are applied. As such, the applied force due to a pressure load depends on the deformed state, and this induces a follower stiffness matrix that contributes to the overall stiffness matrix of the structure.

The boundary where the pressure is applied is specified with a sideset. Also, the magnitude of the applied pressure field must be specified, as shown in the example below. The follower stiffness matrix scales linearly with the magnitude of the applied pressure.
In the above example, sideset 1 is used to denote the surface where the pressure is applied. The parameter "pressure" specifies the magnitude of the applied pressure field.

### 6.3.7. Acoustic Loads

The `acoustic_vel`, `acoustic_accel`, `point_volume_vel`, `point_volume_accel`, Lighthill loading conditions are specifically designed for acoustic elements, and thus may only be applied to acoustic elements. In all cases, a time function is required that defines either the time or frequency dependence of the loads.

The `acoustic_vel` and `acoustic_accel` keywords specify the fluid velocity and fluid acceleration in the normal direction of the element faces in the sideset, respectively. Note that these are the counterparts to the `pressure` load for structures in the sense that they are Neumann boundary conditions.

We note that the `acoustic_vel` and `acoustic_accel` approaches should yield the same acoustic response, provided that the `acoustic_vel` time function is precisely the time integral of the `acoustic_accel` function. This time integration must include the constant of integration. If the two time functions for `acoustic_vel` and `acoustic_accel` are complementary in this way, the acoustic pressure output from these approaches will be the same up to first order. They are not exactly the same since the time derivative of velocity potential is needed to generate the acoustic pressure for output, and that time derivative is only first-order accurate.

An example of the `acoustic_vel` keyword is given below.

```
Loads
   sideset = 1
   pressure = 10.0
   follower = yes
END
```

In this case, sideset 1 is given a prescribed normal velocity of amplitude 1, with a time dependence given by function “velocity_function”.

Currently, a given load case cannot contain both an `acoustic_vel` and an `acoustic_accel` input. Only one or the other can be specified in a given load case, though for a multicase
solution the **acoustic_vel** and **acoustic_accel** inputs could be present in separate load cases. We also note that for coupled structural acoustics, only the **acoustic_vel** keyword is applicable. For analysis involving only acoustic elements, either keyword can be used.

The **point_volume_vel** and **point_volume_accel** keywords prescribe an acoustic point source on a nodeset. This force is the product of the fluid density with the first and second derivatives, respectively, of volume of the source. The **function** for the point source contains the time history of the first (for **point_volume_vel**) and second (for **point_volume_accel**) time derivative of volume.

Since the code scales by density in the internal calculations, there is no need to multiply the time history of volume by density to get the acoustic force. Thus, for point sources the **scale** parameter is typically set to 1.0, unless a direct scaling is desired. The units of the input time functions for **point_volume_vel** and **point_volume_accel** are volume per unit time and volume per unit time squared, respectively. The density need not be multiplied by these functions, since the code is already dividing by density internally (see the theory notes on structural acoustics for a more detailed discussion.)

Currently, the point acoustic source is only implemented for the time domain (transient) calculations.

The **Lighthill** keyword prescribes the divergence of the Lighthill tensor at nodes. The double divergence of the Lighthill tensor provides a source term for noise generation in the pressure formulation of acoustics. The divergence of the Lighthill tensor is a time varying vector quantity that can only be applied as a nodeset load using the `readnodalset` function. The Lighthill load is only implemented for the time domain (transient) calculations. The Lighthill load is only valid for the pressure formulation of acoustics and can only be used with **acoustic_accel** loading. For Lighthill loading, the **scale** parameter is typically set to 1.0, unless a direct scaling of the load being read in is desired. The **outputs** keyword **acousticLighthill** outputs the acoustic Lighthill source term.

```plaintext
Loads
  nodeset 1
  point_volume_accel = 1.0
  function = accelFunc
end

Function accelFunc
  type LINEAR
  name "volume_acceleration"
  include inc/volume_acceleration.inp
end

Outputs
  acousticLighthill
end
```
Input 6.10. Example Input for Point Acoustic Load. In this case, nodeset 1 would consist of a single node, and the file "volume_acceleration.inp" would contain the second time derivative of volume velocity of the source, with units of volume per time squared. Note that the amplitude of the point source is taken to be 1.0, and that it does not include the density multiplier.

The sign conventions of the acoustic_vel, acoustic_accel, point_volume_vel, and point_volume_accel keywords are important. For the acoustic_vel and acoustic_accel cases, the equations of motion are given by,

\[
\frac{1}{c^2} \dddot{p} - \Delta p = -\int_{\Gamma} \rho q (a \cdot n) d\Gamma \tag{6.4}
\]

or, in discrete form,

\[
M \dddot{p} + Kp = f \tag{6.5}
\]

where \(\rho\) is the density, \(q\) is the surface shape function, \(a\) is the acceleration vector on the surface, \(n\) is the normal to the surface, and \(\Gamma\) is the portion of the surface where the loading is defined. \(M, K,\) and \(f\) are the mass, stiffness, and discrete force vectors. We denote \(a \cdot n = a_n\) as the normal component of acceleration. We also note that this force has a negative sign in front of the integral, which comes from the variational formulation. This implies an inverse relationship between surface acceleration and acoustic pressure. Thus if the acceleration is oriented in the same direction as the normal, then \(a_n\) will be positive, and thus the total force vector will be negative. Intuitively, this makes sense, since if the acceleration is in the same direction as the surface normal, mass will be ejected from the acoustic space, causing a decrease in pressure. Conversely, if the acceleration is oriented in the opposite direction as the surface normal, then \(a_n\) will be negative. This will cause the total force vector to be positive, resulting in a positive pressure. This makes sense, since in this case mass will be added to the acoustic space, causing an increase in pressure.

For the point_volume_vel and point_volume_accel boundary conditions, the equations of motion are given by

\[
\frac{1}{c^2} \dddot{p} - \Delta p = \rho \frac{\partial^2 V}{\partial t^2} \delta(x - x_0) \tag{6.6}
\]

or, in discrete form,

\[
M \dddot{p} + Kp = f \tag{6.7}
\]

where \(\frac{\partial^2 V}{\partial t^2}\) is the second derivative of the volume change with respect to time, and \(\delta(x - x_0)\) is the Dirac delta function that makes the term zero everywhere except where \(x = x_0\). We note that \(V\) is the volume of fluid added to the surrounding acoustic space, not the volume of the point source per se. Thus, the sign of the acoustic pressure will be related to the sign of \(\frac{\partial^2 V}{\partial t^2}\). A positive \(\frac{\partial^2 V}{\partial t^2}\) would result in a positive acoustic pressure, implying that fluid mass is added to the surrounding acoustic space. Conversely, if \(\frac{\partial^2 V}{\partial t^2}\) is negative, mass will be subtracted from the acoustic space, and thus a negative acoustic pressure will result.
The previous examples involved spatially constant functions of time. Acoustic boundary conditions with spatially-varying functions of time are supported through the ReadNodal and ReadSurface functions as described in Sections 6.2.2 and 6.2.4 respectively.

### 6.3.8. Thermal Loads

The `thermal_load` option is used in conjunction with a spatial temperature specification for the structure. The temperature distribution can either be specified via the input Exodus file, or on a block-by-block basis, as described below. Based on the temperature distribution, a thermal load is computed and then applied to the structure.

If the solution method is selected to be statics, the `thermal_load` option will provide the thermal load necessary to solve the thermal expansion problem. If the solution method is transient dynamics, the same thermal load will be applied as in the statics case, but modulated by the function that is specified below the `thermal_load` keyword. This corresponds to a thermal shock analysis. Thus, for a transient dynamics problem that includes damping, and with a function that is equal to 1.0 for all time, the transient analysis would eventually converge to the same solution as obtained in the statics analysis, which would be the solution from a classical thermal expansion analysis. On the other hand, for a transient dynamics problem with a `thermal_load` in which the associated time function is not equal to 1.0, the thermal load will be scaled according to that time function. For example, in the case of a mesh that has block-by-block values of temperature $T_{\text{current}}$ specified in the input deck, and a thermal load function that ramps up from zero to one, the actual thermal load applied to the structure will be multiplied by that time function. In this case, the full thermal load will only be seen after the ramp in the time function is completed.

If it is desired to apply a thermal preload to a structure, we generally recommend using a statics analysis rather than a transient analysis, since in the latter case the preload that will be computed will be a dynamic preload that will oscillate about the static preload solution. If damping is used, this dynamic preload will converge to what would be obtained from using a statics analysis. However, in some cases such as when rigid body modes are present, a transient analysis may be the only option for applying the preload.

The temperature field can either be read from an Exodus file, which would typically be the result of a thermal analysis, or it can be specified on a block-by-block basis in the input deck. For temperature fields that change from element to element, the temperatures must be read in from the Exodus input file. For more uniform temperature distributions, it is more efficient to specify them block-by-block in the input deck. Note that when using thermal loads, the temperature data is expected to either be in the mesh (exodus) files, or specified using the input deck (i.e. block-by-block). If temperature is specified in the Exodus file and on a block-by-block basis in the input deck, then the input deck values take precedence.

**Output stress and strain.** Sometimes it is of interest to output the stress after a thermal load analysis. In this case, the stresses that are output to the Exodus file will be
the mechanical stress, rather than the combined thermal-mechanical stress. There is a known bug in the way that thermal stresses are computed, particularly when temperature comes from the Exodus file. If thermal stresses are needed extreme care should be taken. Mechanical stress is the same as elastic stress. Strain, elastic_strain, and thermal_strain output are all available.

**Input deck syntax.** If temperatures are specified using the input deck, then each block must be given its own temperature. In the example below, there are 2 blocks, and each is given a different temperature.

```
BLOCK 1
  material 1
  T_current 100
END
BLOCK 2
  material 2
  T_current 200
END
```

Note that if $T_{current}$ is specified for some blocks and not for others, the code will error out.

When temperatures are read in from the Exodus file, the material properties can be specified as temperature-dependent. This implies that each element will have different material properties. More details are given in the section on temperature-dependent material properties.

For thermal statics or thermal transient analysis, each material block must be given two additional parameters, the reference temperature, $T_{ref} = T_{ref}$, and the coefficient of thermal expansion, $\alpha_t = \text{alphat}$. These parameters are defined via the thermal strain, which is given by

$$
\epsilon_{thermal} = \text{alphat} (T_{current} - T_{ref})
$$

An example is the following.

```
MATERIAL 1
  E 10e6
  nu 0.3
  Tref 300.0
  alphat .001
  density 0.1
END
```

The defaults for $T_{ref}$ and $\text{alphat}$ are both 0.0. This implies that if they are not specified, then the material will not contribute to the thermal analysis (see equation 6.8).

Note: currently, isotropic thermal strain is supported for isotropic, isotropic viscoelastic, and anisotropic materials.
Shell and beam type elements are not supported in thermal strains. If a material with a coefficient of thermal expansion is used in a shell, beam, or other unsupported element Sierra/SD will generate an error.

The default Exodus file labels for the temperatures are shown in the table below. This is the default variable format for Sierra/SD. However, it is also possible to read in element variables and variables of different names. Using the keyword thermal_exo_var in the parameters section (2.3) allows you to specify the name of the temperature variable in the Exodus file. A fatal error is generated when thermal_load is used with energy_exo_var. A nodal variable of this name is expected. If there isn't one, an element variable is used. And if no element variable of the given name is found, an error will be generated.

<table>
<thead>
<tr>
<th>Name</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>TEMP</td>
<td>the nodal temperature</td>
</tr>
</tbody>
</table>

The thermal_load load case can be used in a multicas solution method. In that case, the stresses and internal forces from the thermal analysis are used as initial conditions for the next case. For example, for a fixed-fixed cantilever beam that is subjected to a uniform temperature increase, the beam will undergo a stretch due to the thermal static analysis, and will have residual stresses. If this beam were then subjected to an eigen analysis in a subsequent case, the modes would be modified due to the geometric stress stiffening. Conversely, for a fixed-free beam, there would be no residual stresses and thus no effect on subsequent cases. Note that the displacements from thermal analysis are not carried over to subsequent cases. Thus, to get the total displacement from a thermal analysis followed by transient, one would need to add the displacement results from the two cases separately.

When temperature is read from the Exodus input mesh using either the default temperature file name or the thermal_exo_var keyword a constant temperature can be read from a single time step with the keyword thermal_time_step in the parameters block. Alternatively a time variant temperature can be updated periodically from the mesh file with use of the keyword nUpdateTemperature in the solution case input. The nUpdateTemperature keyword defines how often (every n steps) temperature should be updated. In a transient run updating of temperature is moderately expensive thus it may be advantageous to only update temperature at a finite step interval if temperature is changing slowly.

When a new temperature value is read that temperature is used immediately to update the applied thermal strain. Additionally, the updated temperature can be used with the nUpdateDynamicMatrices keyword to use apply the changing temperature to updated material properties. When reading transient temperature data from the input mesh the closest Exodus time step at or below the current time step is used. Temperature does not interpolate between Exodus steps.

The thermal_time_step keyword must be specified in the parameters block, to specify which time step of the previous thermal analysis should be used to extract temperature data. The following gives an example.
The **Exodus** files can contain multiple time steps of temperature data. The user can select which time step is to be used for defining temperature data in **Sierra/SD**, using the keyword `thermal_time_step`. In this example the tenth time step will be read in from the **Exodus** file. The default value for the `thermal_time_step` is 1.

The **nUpdateTemperature** keyword is placed in the solution case to specify how often to update the temperature that is read in.

```plaintext
SOLUTION
  transient
  nUpdateTemperature 5
END
```

Here a new temperature is read in every 5\textsuperscript{th} time step. If the transient solution specifies the last time step from the thermal analysis then the final temperature will be used.

The next example presents input for thermal statics analysis.

```plaintext
SOLUTION
  statics
END

Parameters
  thermal_time_step 10
end

Loads
  body
    thermal_load
END
```
6.3.9. Energy Deposition Input and Loads

Input from energy deposition are similar to thermal loads (section 6.3.8). These loads are specified when energy is deposited directly in the structure as with an X-ray deposition. For consistency with other applications, the energy is defined as specific energy, i.e. the energy per unit mass. Such direct energy deposition is converted to a change in temperature after which thermal strains and loads are computed exactly as for the thermal_load approach.

Energy is converted to a change in temperature using the specific heat of the material (see Section 4.4.8).

\[ \tilde{E} = C_v \Delta T \]

where \( \tilde{E} \) is the specific energy of the body, \( C_v \) is the specific heat capacity for constant volume, and \( \Delta T \) is the change in temperature.

The energy load is specified using the keyword energy_load. All other parameters are identical to thermal_load. Note that by the nature of these loads there is often an exponential decay in energy as a function of depth. For this reason, it is advantageous to specify the loads at Gauss points, particularly when using higher order elements. Energy loads can also be specified at nodes or centroids. Nodal energy loads should be avoided because it is not clear which materials specific heat to use when converting an energy load to a temperature change for nodes on the interface between two materials. For this case Sierra/SD will use the specific heat of the material that is processed last (which is not typically what is required by the analyst). Sierra/SD generates a fatal error if the specific heat is not specified for a material with an energy load.

Energy may also be used as an input for thermally dependent material properties. To ensure that the energies are converted to temperature before determining the material properties, identify the variable name from the Exodus file with the energy_exo_var and energy_time_step keywords, rather than the thermal_exo_var and thermal_time_step keyword. Using thermal_exo_var with energy_load generates a fatal error.

6.3.10. Consistent Loads

The loads for every 3-D and 2-D element is calculated in a consistent fashion when a pressure load is applied. For more details on the implementation, see the programmer’s notes. It is important that consistent loading be used. This is especially true for shell elements where the consistent loading is required to properly apply rotations.

6.3.11. Pressure_Z

Depth dependent pressure loads may of course be applied using a user defined function. To simplify this loading condition, depth dependent pressure may also be applied using the
**pressure_z** keyword. An example is shown in input 6.11. This loading is applied only in the basic coordinate frame, and the analyst must specify that the pressure is either “below” or “above” an offset to the coordinate axis. The pressure is always proportional to the depth. In the example of input 6.11, the pressure is zero at $x = 5$, 10 at $x = 4$, 20 at $x = 3$. At depths above the “waterline”, the pressure is zero. Any of the basic coordinate directions ($x$, $y$, or $z$) may be used as a reference.

```plaintext
// depth dependent pressure for a waterline at x=5.
Loads
   sideset 2
      pressure_z 10.0 below x = 5
      sideset 20 // air
         pressure_z 1e-4 above x = 5
END
```

**Input 6.11. Depth Dependent Pressure Load Example.** This load section applies a pressure to sideset 2 which is proportional to the distance below $x = 5$.

### 6.3.12. Surface Charge

For electro-mechanical coupled materials such as dielectric or piezoelectric materials, surface charges may be applied to a specified sideset using the keyword **surface_charge** in the loads block. The surface charge is mechanically analogous to a pressure, where surface charge represents a charge per unit area and is applied only to the voltage degrees of freedom. Surface charges can be applied for direct frequency response and transient solution methods. An example is provided below.

```plaintext
Loads
   sideset 1
      surface_charge = 1 // scalar multiplying the specified function
      function 1 // surface_charge function
END
```

**Input 6.12. Surface Charge Example**

### 6.3.13. Static Loads

Static loads only require the definition of the load region and load keyword (e.g. force) with its accompanying parameters. A function may be used instead. In this case, the function will be evaluated at time $t = 0$.
6.3.14. Time Varying Loads

Additional options provide the capability of varying the load over time. The load options include,

- **scale** with one parameter provides a scale factor to be applied to the entire load set. Only one scale may be provided per load set.

- **function**. A time varying function may be applied by specifying a function ID. Only one function may be applied per load set. The function is defined in the function section (see section 2.8 on page 86). The loads applied at time $t$ for a particular load set will be the sum of the force or moment vectors summed over the nodes of the region and multiplied by the scale value and the value of the time function at time $t$.

**NOTE:** If no function is applied for a particular load, then the function is defined as 1.0 for all time. All loads will be applied to the transient solution, regardless of whether an explicit time function is defined.

6.3.14.1. Reading Loads from Exodus Data

Loads may be read in from previous analyses when stored in the input exodus file. These are read using an appropriate function. See sections 6.2.2, 6.2.3, and 6.2.4 for functions which read data on nodal values, on a node set and on a surface respectively.

Note that exodus read functions do not trigger follower stiffness calculations when used as follower loads. A warning is issued when the follower stiffness calculation is skipped.

Also, note that exodus read functions ignore distribution factors on sidesets. A warning is issued in this case when the distribution factors are ignored.

6.3.15. Random Pressure Loads

Input for random loads can be complicated, though the loads are not uncommon and are important for many applications. This type of random pressure loading is developed for use of direct transient loading typical of a turbulence load on a hypersonic vehicle. Throughout the development, we maintain a concept of flow direction, and correlation distances that may be different in flow and transverse directions. By computing the random pressure fields as part of the time evolution, we avoid the need to compute these complex quantities before the run. The approach requires a linear solve at each solve to compute the loads.

\[1\] A hypersonic vehicle is a prime example of a random loading. Turbulence provides a time varying loading which has a limited spatial and temporal correlation on the surface of the hypersonic vehicle.

354
The most general type of input is the correlation matrix, which is the inverse Fourier transform of the spectral density matrix. The RandomPressure load option provides a simplified means of specification of the loading. The material in this section is consistent with and builds on Section 2.8.7.

Subsection Random Pressure Loading section Loads and Materials of the Theory Manual\textsuperscript{41} describes the approximations involved in the implementation. These approximations are summarized in Figure 6-59.

The simplified correlation matrix is not general, but may be useful for a large class of problems. It has the following limitations.

1. The system must be time stationary.
2. The correlation function must be separable (a product of temporal and spatial correlations).
3. The same PSD shape must apply throughout the entire hypersonic vehicle body. The PSD may be scaled as a function of $z$, but there may be no change in the shape.
4. The PSD must have a cutoff. The time integration must occur above this cutoff frequency.
5. By default, the temporal function is represented by a sinc function. This may be replaced by a user defined temporal function.

\begin{figure}[h]
\centering
\includegraphics[width=\textwidth]{random_pressure_loadings.png}
\caption{RandomPressure Loading Approximations.}
\end{figure}

The random loading is a component of the loads section. An example is shown here, and described in Table 6-122.

\begin{verbatim}
Loads
 sideset 22
 RandomPressure
   correlation_length_z = 2.0  // required
   correlation_length_r = 0.67 // required
   cutoff_freq = 16.8 // required
   correlation_function = 20 // defaults to sin(x)/x
   psd_scale_function = 10 // defaults to Sigma=1
   NTimes = 5 // defaults to 5
   coordinate 1 // defaults to basic frame
   MinimumNodalSpacing = 1.0e-5 // defaults to 1.0e-8
   NumberOfInitializationSteps = 100 // defaults to 5
END
\end{verbatim}

Details for the parameters to the correlation matrix input are described below.
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Type</th>
<th>Default</th>
<th>Comment</th>
</tr>
</thead>
<tbody>
<tr>
<td>correlation_length_z</td>
<td>real</td>
<td>required</td>
<td>spatial decay in flow direction</td>
</tr>
<tr>
<td>correlation_length_r</td>
<td>real</td>
<td>required</td>
<td>spatial decay orthogonal to the flow</td>
</tr>
<tr>
<td>cutoff_freq</td>
<td>real</td>
<td>required</td>
<td>cutoff frequency</td>
</tr>
<tr>
<td>correlation_function</td>
<td>string</td>
<td>$\frac{\sin(t\omega_c)}{t\omega_c}$</td>
<td></td>
</tr>
<tr>
<td>psd_scale_function</td>
<td>string</td>
<td>$\Sigma(z) = 1$</td>
<td></td>
</tr>
<tr>
<td>NTimes</td>
<td>int</td>
<td>5</td>
<td>defaults to basic frame</td>
</tr>
<tr>
<td>coordinate</td>
<td>string</td>
<td>0</td>
<td>smallest allowable inter-node spacing</td>
</tr>
<tr>
<td>MinimumNodalSpacing</td>
<td>real</td>
<td>$1 e - 8$</td>
<td>random number seed</td>
</tr>
<tr>
<td>Random_Seed</td>
<td>int</td>
<td>ignore</td>
<td>iterations to improve initial spatial distribution</td>
</tr>
<tr>
<td>NumberOfInitializationSteps</td>
<td>int</td>
<td>5</td>
<td></td>
</tr>
</tbody>
</table>

Table 6-122. – Random Pressure Inputs.

**correlation_length_z** Spatial decay in the flow direction, $L_z$. The flow direction is the $Z$ axis of the coordinate frame. The correlation function $C(\Delta Z)$ is proportional to $\exp(-\Delta Z/L_z)$, where $\Delta Z$ is the distance between two points in the flow direction.

**Correlation_Length_R** Correlation_length_r is the spatial correlation distance in the radial or transverse direction. The correlation function is proportional to $\exp(-\sqrt{(\Delta x^2 + \Delta y^2)/L_r})$.

**Cutoff_freq** The cutoff frequency, $F_c$ is important to the operation of the RandomPressure algorithm. No energy may be found in the PSD above this frequency. The time integrator may not sample the system lower than this frequency, i.e. $dt < 1/F_c$.

**NTimes** The matrix is proportional to the number of time values assembled, and affects the interpolation as described in the Theory Manual section Loads and Materials subsection Alternative Derivation Based on Lagrange’s Equations subsection Separation of spatial and temporal components

Typically, only a small number of terms are required. Note that there are $2 \times NTimes + 1$ terms in the sum, and the dimension of the correlation matrix grows commensurately. The number may depend on the interpolation time step and on the shape of the PSD. Default=5 (which produces 11 terms in the sum).

**CORRELATION_FUNCTION** The temporal time function, whose argument is $(t_1 - t_2)$. By default this function is $\sin(x)/x$, with $x = \pi F_c(t_1 - t_2)$. It must be an even function of the argument.

**PSD_SCALE_FUNCTION** provides a means of scaling the power spectral density as a function of flow direction. This type of input requires that the PSD have the same shape at all locations, but the value may be scaled. Scaling the PSD effectively scales the standard deviation of the pressure. Default is no scaling. The function must be positive for all values of the coordinates.
**coordinate** is an optional coordinate frame that is used to define the flow direction. The $\hat{Z}$ component of that frame is the direction of flow. By default, the basic frame is used.

**MinimumNodalSpacing** Some models can contain co-located nodes on the surface where the random pressures are to be applied. This can cause the correlation matrix to be singular, since the repeated nodes would result in two identical rows in the correlation matrix. The **MinimumNodalSpacing** keyword allows the analyst to specify the smallest inter-node spacing (absolute) that is allowed on the surface where the random pressure is being applied. Any nodes that are closer than that tolerance will be treated as identical in the correlation matrix manipulations. The **Exodus** file and corresponding nodal output will not be changed. This will avoid a singular correlation matrix, but does not alter the mesh database.

**NumberOfInitializationSteps** Initially the pressure spatial distribution may be too correlated. Mesh resolution exacerbates this issue. Increasing the **NumberOfInitializationSteps** mitigates the issue. Each initialization step requires about as much CPU time as an implicit time step. Default=5 (values below 5 are discouraged).

**OMEGA_C** Deprecated. Use Cutoff_freq.

**ALPHA_Z** Deprecated. $\alpha_z = 1/L_z$.

**BETA_T** Deprecated. $\beta_t = 1/L_R$.

The computation of the random pressure loads depends on matrix factorizations (subsection Random Pressure Loading section Loads and Materials of the Theory Manual).

However, the Cholesky matrix factorizations are defined only if the correlation matrix is (numerically) non-singular. At this time, the code stops with an error if this occurs. A common cause of this error is using too many time steps NTimes with too small a time step. For this reason, the condition number of the temporal correlation matrix is always evaluated, and, if it is singular, the cutoff frequency is decreased. In this case the warning message

```
Singular temporal correlation matrix
Increasing Delta_T to ...
```

will be printed in the result file for processor 0. Another source of ill conditioning is the use of large correlation lengths correlation_length_z or correlation_length_r, or a fine mesh.

For this reason inverse condition number estimates are printed in the result files. An inverse condition number is the relative distance to a singular matrix, and is denoted Rcond, for reverse condition number. In double precision, an Rcond below $10^{-12}$ indicates that the factorization may fail. The precise statements in the results files are
6.3.16. Frequency Dependent Loads

Frequency dependent loads may be applied for frequency response analysis. The real part of these loads is applied exactly as above with the understanding that the functions referenced apply to frequency not time. Frequency dependent loads may include an imaginary component. This is done by prefixing the load types listed above by the letter “i”. Thus, the imaginary part of the load uses these load types.

<table>
<thead>
<tr>
<th>Option</th>
<th>Parameters</th>
</tr>
</thead>
<tbody>
<tr>
<td>iforce</td>
<td>val1 val2 val3</td>
</tr>
<tr>
<td>imoment</td>
<td>val1 val2 val3</td>
</tr>
<tr>
<td>igravity</td>
<td>val1 val2 val3</td>
</tr>
<tr>
<td>ipressure</td>
<td>val1</td>
</tr>
<tr>
<td>itraction</td>
<td>val1 val2 val3</td>
</tr>
</tbody>
</table>

A function should be associated with each such load. An example follows.

Loads // example for FRF analysis

    nodeset 1
        force=1 0 0     // the real part of the load
        function=11
    nodeset 2
        iforce=1 0 0    // the imaginary part of the load
        scale .707
        function=12

END

6.3.17. Modal Force Loading

Modal force loading can be used to directly load specific modes in the modaltransient solution case. An example input follows.

ECHO
    modalvars
END
Loads
    body
        ModalForce

358
The format of Qforce.txt is identical to the output from a traditionally loaded modal transient solution with `echo modalvars`. `echo modalvars` does not print out a load at time zero, so either the load at time zero should be added to the datafile, or `origin` should be used to specify the first time in the file. Loads can be linearly interpolated between time values in the datafile. With redundant modes, some modes can change between subsequent Sierra/SD runs. Care should be taken to ensure the load are applied correctly.

Full details of table input are discussed in Section 2.8.19. Only two-dimensional tables are supported for modal force loading, with rows representing equally spaced time steps and columns representing each mode. Modal force loading is the only load type that supports two-dimensional table inputs.

### 6.3.18. Rotational frames

Often when analyzing rotating structures, it is convenient to perform the analysis in the rotating frame where the structure is not undergoing large displacement. Analysis in that frame introduces “fictional” or “pseudo” forces with centrifugal, Coriolis and Euler contributions. These are termed “forces”, but the contributions are introduced from operating in a non-inertial coordinate frame as described in the Theory Manual section Loads and Materials subsection Analysis of Rotating Structures.

The associated keywords are found in Table 6-123.

The Galerkin framework used for finite elements, introduces matrices associated with these pseudo forces. In addition to the standard mass and stiffness matrices that arise in linear structural dynamics, force-based matrices are also common. These include follower stiffness matrices from applied pressures, and Coriolis/centrifugal matrices in rotating structures.

---

2There is often confusion about the description of the “centrifugal” or “centripetal” term. The *centripetal* force is a real force applied in the inertial coordinate frame which causes an object to travel in a circular path. The *centrifugal force* is the pseudo-force that appears from inertial terms in a rotating coordinate frame.
Option Parameters

<table>
<thead>
<tr>
<th>angular_velocity</th>
<th>vel1 vel2 vel3</th>
</tr>
</thead>
<tbody>
<tr>
<td>angular_acceleration</td>
<td>accel1 accel2 accel3</td>
</tr>
<tr>
<td>coordinate</td>
<td>coordinate name</td>
</tr>
</tbody>
</table>

Table 6-123. – Rotating Frame Parameters.

Input 6.13 provides the corresponding Sierra/SD input for a rotational load applied to a body. The centrifugal stiffness and Coriolis coupling matrices are both derived from the rotational velocity of the structure, which uses the keyword `angular_velocity`. The vector angular velocity components are specified after the `angular_velocity` keyword.

An angular acceleration, $\dot{\Omega}$, may also occur, as when an aircraft carrying a weapon makes a rapid course correction. This angular acceleration results in a pseudo-force, called the Euler force, that is tangent to the angular acceleration vector. Application of angular acceleration is restricted to linear and nonlinear statics analysis in Sierra/SD.

For static loads analysis angular acceleration and angular velocity are applied independently. A similar static loads analysis of a rocket provides envelope survivability information during launch.

```
LOADS
body
  angular_velocity = 0.0 2.0 0.0
  coordinate = 1
body
  angular_acceleration = 0.0 0.0 3.0
  coordinate = 3
END
```

Input 6.13. Application of centrifugal and Euler forces. The loads above apply an angular acceleration of 3 radians/s$^2$ in the Z-direction of coordinate frame 3, and an angular velocity of 2 radians/s in the Y-direction of coordinate frame 1. Angular acceleration is applicable only in statics.

**Left Hand Side contribution**

Angular velocity introduces both left-hand side matrices and right-hand side force vectors. The algebraic expression for dynamics can be written as follows.

$$ (K_m + K_g + K_{cen}) u + (C + C_{cor}) \ddot{u} + M \dddot{u} = f_{extern} + f_{cen} \quad (6.9) $$

where,
$K_m$ is a material matrix,
$K_g$ is the geometric stiffness matrix correction,
$K_{cc}$ is the centrifugal softening term,
$C$ is the damping/coupling matrix,
$C_{cor}$ is the Coriolis coupling matrix,
$M$ is the mass matrix,
$f_{ext}$ is the external force vector,
$f_{cen}$ is the centrifugal force, and
$u$ is the displacement.

Except for $K_g$, every matrix term is constant, depending only on the geometry and the elements. The Coriolis and centrifugal terms are also independent of displacement, $u$, though they depend on $\Omega$.

For linear analysis (both linear statics and linear transient dynamics), the geometric stiffness terms is zero. However, since this term depends on stress, which is proportional to displacement, the geometric stiffening is typically proportional to the square of the angular velocity. As the geometric stiffening is typically of the same magnitude as centrifugal softening (also proportional to $\Omega$), confusion can arise.

In multicase analyses, the matrices are typically generated only once; exceptions occur for nonlinear solutions and for the tangent method. It is recommended that linear solution cases include an update to the tangent stiffness matrix as part of a multicase solution. An example is shown in input 6.14.

```plaintext
Solution
case s1
  statics
    load=1
case up
tangent
case s2
  statics
    load=1
End
```

**Input 6.14. Example using Tangent Update**

**Limitations**

There are a number of limitations for the rotational frames implementation.
1. Static analysis appropriately applies the centrifugal and Euler forces. The left-hand side matrix for geometric stiffness is only properly updated if the `tangent` step is applied.

2. In a single case solution, quadratic eigen solutions will include Coriolis and centrifugal terms if angular velocity is specified in the `loads` section. This is the case even though there is no true load for Q EVP.

3. Currently, Q EVP solutions can be computed for rotating structures only if there are no rigid body modes in the structure. An example is shown in input 6.15. In this case, a static preload with a rotational load is computed, followed by a tangent update, and then followed by a Q EVP analysis. This type of analysis would be useful for examining the effect of rotational loading on the modes of a structure. However, this will only work correctly if there are no rigid body modes in the structure.

4. The user is limited to one rotational frame per analysis. In other words, the whole body must be rotate together. One could not model a helicopter in a fixed frame and the associated rotor in another.

5. Rotational loads applied in the rotating frame are linear loads, and do not require a follower keyword.

6. For transient dynamics, the time varying function must be 1.0.

7. Angular acceleration is only applicable to statics analyses.

8. Superelements do not retain full accuracy. It is recommended that interface dofs for superelements retain either 3 or 6 degrees of freedom.

9. The Boundary section applies to all cases in a multicase solution.

```
Solution
case 'statics'
  statics
  load=1
case 'up'
  tangent
case 'qevp'
  qevp
    method=projection_eigen
    nmodes=100
    load=1
End
```

**Input 6.15. Example of using qevp for Tangent Update**
NOTE:
A time varying function with magnitude 1.0 for the full time span should be used for time varying solution cases. Additional work would be required to apply general loading patterns.

Finally, for optimal accuracy, the user can update the tangent matrix at the expense of greater computational expense.

6.3.19. Rigid Body Filter for Input

For some analyses, it is advantageous to remove rigid body components of a solution. The input forces may be filtered so that only self-equilibrated forces are applied. This process of removing the rigid body component from the solution is sometimes referred to as 'Inertia Relief' or 'Inertial Relief'.

The rigid body filter is applied using input in the parameters section (2.3) and is illustrated in the example of input 6.16. The filtered force values can be output by requesting the force output option (section 7.1.34).

While the filter can ensure equilibrated loads, additional parameters may be required to help the linear solver address the singularity generated by floating structures. Typical input is provided here, with details in the appropriate sections. The constrain_rbs solver option must be used in conjunction with FilterRbmLoad only when the rigid body modes are in the null space of the system matrix. For example, constrain_rbs should be used for statics where the system matrix is the stiffness matrix, but constrain_rbs should not be used for transient where the system matrix includes inertial terms.

Currently, only GDSW supports selectively constraining rigid body modes. The FilterRbmLoad parameter is supported for transient and static solution cases. For other solution cases this parameter will have no effect on the solution. The similar capability for modal solutions is presented in Section 3.33.

Solution
  statics
  solver=gdswn
  solver_options=gdswn_options
End

Parameters

---

3The FilterRbm option is only compatible with Newmark-Beta time integration. The generalized-alpha time integrator may be used, but the coefficient alpha_m=(2 rho-1)/(1+rho) will be set to zero, where rho is the integration parameter. The resulting integration scheme is still 2nd-order accurate. The rigid body motion equations will be integrated using the same scheme as the flexible body equations. This is reviewed in subsection Damping of Flexible Modes Only section Solution Procedures of the Theory Manual.

4Modal solutions, such as modaltransient, do not use FilterRbmLoad. However, see for means of accomplishing the same process by direct use of the geometry rigid body modes.
Input 6.16. Rigid Body Filter Example Input

The names of the rigid body modes to constrain are the same as those setting boundary conditions 6.1. The ‘p’ keyword refers to the constant pressure in a structural acoustics problems, and the other six are the typical rigid body modes of a structure in Cartesian coordinates.  

If the rigid body filter is activated, Sierra/SD calculates the corresponding rigid body modes, checks the residuals, and report a fatal error if the residual norms,

\[ \frac{\| K\Phi_r \|_2}{\| Kd\|_\infty\| \Phi_r \|_2} \]

are larger than RbmTolerance. The default RbmTolerance is \(10^{-10}\). Each module that needs the rigid body modes will recalculate them.

6.3.20. Ran Loads

The RanLoads section is used to provide input information for spectral input to a random vibration analysis. In a random analysis, the output response relates to the input, as follows. \(H_{ij}\) is the transfer function from input \(i\) to dof \(j\). \(S_{jk}\) is the input power spectrum. Typically, this is in units of (force)\(^2\)/Hz. It is dimensioned to the number of independent inputs.

The relationship is

\[ a_i(\omega) = \sum_{j,k} H_{ji}^T(\omega)S_{jk}(\omega)H_{km}(\omega). \]  

(6.10)

The Ran Loads section provides a specification for \(S_{jk}(\omega)\). Note that this input will contain both a spatial and spectral component. In Sierra/SD, we require that each matrix element in the input power spectrum be expressible as a product of spectral and

\[ ^5 \text{The keywords are “x”, “y”, “z”, “RotX”, “RotY”, “RotZ” and “p”. The set of these keywords must be enclosed in quotation marks.} \]
spatial components. $Y_i$ is a spatial loading term associated with the $i^{th}$ row and column of $S$, and $F_{ij}$ is a spectral only matrix function.

$$S_{ij}(\omega, x) = Y_i(x)Y_j(x)F_{ij}(\omega)$$  \hspace{1cm} (6.11)

It typically has units of $1/\text{Hz}$.

The **RanLoads** section contains the following required keywords.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Argument</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>matrix</td>
<td>Int/String</td>
<td>matrix-function identifier</td>
</tr>
<tr>
<td>load</td>
<td>Integer</td>
<td>row/column identifier</td>
</tr>
</tbody>
</table>

The **matrix** keyword identifies the appropriate **matrix-function** (see Section 2.8.17). The matrix-function determines the dimensionality of the input (using the **dimension** keyword). It also determines the spectral characteristics of the load.

The spatial characteristics (which correspond to $Y_i$ in equation 6.11) are determined in **load** sections within the **RanLoads** definition. There must be exactly as many **load** sections as the dimensionality of input. For example, if the $S_{FF}$ matrix is $3 \times 3$, then there should be 3 separate load sections. Each load section within the **RanLoads** block must be followed by an integer indicating to which row/column it corresponds. The details of each **load** section are identical to the over all **loads** section (see 6.3) except that no time/frequency function is allowed. Note that only one load is required per row of the $S_{FF}$ matrix, but each entry of the matrix may have a spectral definition (identified by a real and/or imaginary function).

The following example illustrates the definition of a single input specification. The loading is scaled so that a 1000 lb (or 454 kg) mass located on the input point (in nodeset 12 here) is scaled to produce a unit $g^2/\text{Hz}$ loading.

```
RanLoads
  matrix=1
  load=1
  nodeset 12
    force=0 1 0
    scale 1.00e3 // needed to convert to g
    // loads input in lbs. The PSD is in g^2/Hz.
    // F = accel * mass
    //   = accel * (scale_factor)
    //   = accel * ((1000*.00259)*384.6)
END
```

Scaling the input force for a random vibration analysis can be confusing. 6 This is

---

6Note that we are scaling the spatial forces, $Y_i$, which are combined as a product in equation 6.11. Thus, the scale factor is linear in the load. The resulting input power spectrum, $S_{ij}$, will contain the square of the scale factor.
especially true since enforced acceleration cannot be used to apply the force. The example above uses United States customary units. The \texttt{wtmass} parameter has been applied. In SI units, \texttt{wtmass}= 1, and the force would need to be multiplied by $g$ to apply the input as acceleration in $g$'s.

The input acceleration may be examined by evaluating the output PSD at the input degree of freedom. This is done by putting the applied load set into the \texttt{frequency} section (7.4), and adding the \texttt{acceleration} keyword. The output is in the native units of analysis. For the example above, the output will be in $(\text{in}\cdot\text{lbm}/s^2)^2/\text{Hz}$, and must be divided by $(386.4)^2$ to convert to $g^2/\text{Hz}$.

### 6.4. Initial Conditions

Initial conditions are specified via the \texttt{initial-conditions} section. The initial conditions are used as the initial state for transient analysis. Both linear and nonlinear transient are supported. The variables supported for initial conditions are given in Table 6.4.

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Field Names</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>displacement</td>
<td>dispX, dispY, dispZ, displacement_x,</td>
<td>Translational displacement</td>
</tr>
<tr>
<td></td>
<td>displacement_y, displacement_z</td>
<td></td>
</tr>
<tr>
<td>velocity</td>
<td>velX, velY, velZ, velocity_x,</td>
<td>Translational velocity</td>
</tr>
<tr>
<td></td>
<td>velocity_y, velocity_z</td>
<td></td>
</tr>
<tr>
<td>rotation</td>
<td>rotX, rotY, rotZ, rot_displacement_x,</td>
<td>Rotations for beams, shells, etc.</td>
</tr>
<tr>
<td></td>
<td>rot_displacement_y, rot_displacement_z</td>
<td></td>
</tr>
<tr>
<td>rotational_velocity</td>
<td>velRX, velRY, velRZ, rotational_velocity_x,</td>
<td>Rotational velocity for beams, shells,</td>
</tr>
<tr>
<td></td>
<td>rot_velocity_y, rot_velocity_z</td>
<td>etc.</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>acoustics</td>
<td>acoustics</td>
<td>Primary acoustic variable, usually pressure</td>
</tr>
<tr>
<td>acousticsdot</td>
<td>acousticsdot</td>
<td>Derivative of primary acoustic variable, usually particle velocity</td>
</tr>
</tbody>
</table>

Each term may be initialized in one of three ways. Initial conditions may be either

1. Read in from the \texttt{Exodus} file,
2. specified globally in the \texttt{initial-conditions} section,
3. or specified on a block-by-block basis in the input deck.
6.4.1. Reading Initial Conditions from the Mesh File

Initial conditions can be read from fields present on the mesh file. This method would usually be used when handing of initial conditions from the end state of a previous Sierra/SD or Sierra/SM run. The time command can be used to set which time step from the exodus file the initial conditions are read from. Valid options for time include reading from a specific time, start time of the transient simulation, or from the first or last step of the Exodus database. By default, time is set to the first step of the input Exodus database.

```
INITIAL-CONDITIONS
  velocity=from_file
  rotational_velocity=from_file
  displacement=from_file
  rotation=from_file
  time = last
END
```

An example of initial conditions input from Exodus file is given below. In this case, Sierra/SD will read both velocity and displacement initial conditions from the last time step of the Exodus file using the field names in Table 6.4.

6.4.2. Setting Initial Conditions in the Input Deck

An example of setting initial conditions globally is given below.

```
INITIAL-CONDITIONS
  velocity=1 0 0
END
```

In this case, the entire model is given an initial velocity of 1 in the $x$ direction, and 0 for the $y$ and $z$ directions.

An example of the third option (block-by-block specification) follows.

```
INITIAL-CONDITIONS
  velocity=by_block
END

BLOCK 1
  velocity = 1 0 0
```
In this case, the velocity is read in from the input deck on a block-by-block basis. If two blocks share nodes and are given different initial conditions, then the results may be unpredictable, since the common nodes on the blocks would have conflicting initial conditions. Thus, it is recommended to verify that blocks are disjoint before specifying different initial conditions on a block-by-block basis.

Initial conditions are currently only implemented for transient analysis. They can also be used in multicase solutions, but they will only have an effect on the transient analysis that are in the multicase solution. For multiple transient analysis in a multicase, only the first transient analysis will use the initial conditions. The subsequent transient cases will get initial conditions from the previous case.

**Usage Guidelines** Care must be taken when setting initial conditions. If initial conditions are set from a Sierra/SM analysis, then that analysis must in general have small deformations. For example transferring an initial condition displacement from Sierra/SM with significant rigid body rotation would yield large stress in a linear Sierra/SD analysis due to the incompatible linear and nonlinear deformation state. Additionally, for elements like shells the nodal rotations must be compatible with the overall translational displacement.

### 6.5. Use cases for initial acceleration

All these cases can be applied as a spring mass system.

1. Apply force, displacement and velocity as an initial condition for transient dynamics. Compute,

\[ A_n = \left( F_{\text{applied}} + Cv_n + Kd_n \right) / M \]

With a constant force, we should get a constant acceleration.

2. Apply a static load to the mass on the spring. Then apply (in a second case) the same load as a transient load. This should also result in a constant displacement.

3. TSR read. Read a static stress to the body. This is usually followed by a subsequent transient run.

Ring down is expected. Internal stresses are not treated the same as applied forces. Applied external forces are not carried forward. Internal stresses are carried forward.
in the solution. When you start the transient run, you have an internal force only. Compute

\[ A_n = \frac{F_{\text{internal}}}{M} \]

4. **receive_sierra_data.** Read undeformed coordinates, \( X_o \), and a nodal displacement, \( U_o \). Compute initial coordinate, \( X_1 = X_o + U_o \). Following this, no load is required to maintain the \( X_1 \) configuration. Initial stresses may cause a deformation from that initial system. We may get velocity, force, and acceleration from file.

5. Prescribed acceleration. \( A_o = a(t) \). This is applied only on a surface or node set. We convert this to a prescribed force, and treat it identically to an applied force.

6. Restart. \( A(0), v(0) \) and \( d(0) \) are prescribed at all locations. No solve needed.

7. There is a jump in force at some time other than \( t=0 \). Do nothing. We assume that the analyst wants it to ring.

### 6.5.0.1. Summary

- Use **velocity, displacement** from state change (whether read _sierra, user input, or multicase). Eigen may store displacement on the database. It is never used in a subsequent transient or statics analysis. It is always used in modal superposition cases. We get this by calling them PHI.

- Retain stresses across state change (multicase or read). Some cases retain stresses (statics, TSR, transient). Eigen may compute stresses, but does not store them on the database.

- Do NOT retain accelerations or forces across state change. We should use the forces calculated at that time (in the next state) for computing the balance.
7. OUTPUTS

7.1. Exodus

The outputs section identifies the data to write to the corresponding output files. Geometry-based finite element results are written to an output Exodus file. The name of this file is generated by taking the base name of the input Exodus geometry file, and inserting -out before the file extension. For example, if the input Exodus file specification is example.exo, output will be written to example-out.exo. When using a multicase solution (Section 3.2), the case identifier is used in place of out. More details are available in the file section (2.2).

Non-geometry-based finite element data, including system matrices and tables may be output in a format that is compatible with MATLAB. These text files have the extension m. The base file names describe the nature of the output. These files are generated in the current working directory.

<table>
<thead>
<tr>
<th>Option</th>
<th>Description</th>
<th>Section</th>
</tr>
</thead>
<tbody>
<tr>
<td>acceleration</td>
<td>acceleration at nodes</td>
<td>7.1.15</td>
</tr>
<tr>
<td>acoustic_lighthill</td>
<td>acoustic Lighthill source term</td>
<td>N/A</td>
</tr>
<tr>
<td>aforce</td>
<td>acoustic forces</td>
<td>N/A</td>
</tr>
<tr>
<td>block_energies</td>
<td>block kinetic and strain energies</td>
<td>7.1.31</td>
</tr>
<tr>
<td>constraint_info</td>
<td>nodal constraint information</td>
<td>7.1.48</td>
</tr>
<tr>
<td>disp</td>
<td>displacements at nodes</td>
<td>7.1.13</td>
</tr>
<tr>
<td>EForce</td>
<td>element forces for beams</td>
<td>7.1.38</td>
</tr>
<tr>
<td>ElemEigChecks</td>
<td>first eigenvalue, seventh eigenvalue, and the largest eigenvalue</td>
<td>7.1.11</td>
</tr>
<tr>
<td></td>
<td>on</td>
<td></td>
</tr>
<tr>
<td>energy</td>
<td>element strain energy and strain energy density</td>
<td>7.1.28</td>
</tr>
<tr>
<td>eorient</td>
<td>element orientation vectors</td>
<td>7.1.43</td>
</tr>
<tr>
<td>faa</td>
<td>force vector in the a-set</td>
<td>7.1.8</td>
</tr>
<tr>
<td>fatigue</td>
<td>Fatigue Damage Estimates</td>
<td>7.1.23</td>
</tr>
<tr>
<td>force</td>
<td>the applied force</td>
<td>7.1.34</td>
</tr>
<tr>
<td>GEnergies</td>
<td>global sum of energies</td>
<td>7.1.29</td>
</tr>
<tr>
<td>globals</td>
<td>ensure global output</td>
<td>7.1.30</td>
</tr>
<tr>
<td>Kaa</td>
<td>stiffness matrix in the a-set</td>
<td>7.1.7</td>
</tr>
<tr>
<td>line_weld</td>
<td>line weld elements</td>
<td>7.1.39</td>
</tr>
<tr>
<td>Maa</td>
<td>mass matrix in the a-set</td>
<td>7.1.4</td>
</tr>
<tr>
<td>material</td>
<td>material parameter element output</td>
<td>7.1.5</td>
</tr>
<tr>
<td>material_direction_1</td>
<td>Local element coordinate system R vector</td>
<td>7.1.6</td>
</tr>
<tr>
<td>material_direction_2</td>
<td>Local element coordinate system S vector</td>
<td>7.1.6</td>
</tr>
<tr>
<td>material_direction_3</td>
<td>Local element coordinate system T vector</td>
<td>7.1.6</td>
</tr>
<tr>
<td>MLumped</td>
<td>nodal lumped mass in eigen solution</td>
<td>7.1.9</td>
</tr>
</tbody>
</table>
In the following example, the mass and stiffness matrices will be output in MATLAB format, but the displacement variables, stresses and strains will not be output. All the various options of the outputs section are shown in Table 7-124. The next sections describe each of the options and their results assuming an input file named example.inp and a geometry file named example.exo.

### Table 7-124. – OUTPUT Section Options.

<table>
<thead>
<tr>
<th>Variable</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>MPhi</td>
<td>Mass × Displacement in eigen solution</td>
<td>7.1.10</td>
</tr>
<tr>
<td>mesh_error</td>
<td>mesh discretization errors</td>
<td>7.1.32</td>
</tr>
<tr>
<td>MFile</td>
<td>MATLAB MFiles</td>
<td>7.1.33</td>
</tr>
<tr>
<td>nodal_charge</td>
<td>applied charge at nodes</td>
<td>N/A</td>
</tr>
<tr>
<td>pressure</td>
<td>pressure load vector</td>
<td>7.1.44</td>
</tr>
<tr>
<td>rainfall</td>
<td>Rainflow cycle counter output</td>
<td>7.1.22</td>
</tr>
<tr>
<td>reaction_force</td>
<td>the Dirichlet boundary reaction force</td>
<td>7.1.36</td>
</tr>
<tr>
<td>relative_disp</td>
<td>1D element nodal separation</td>
<td>7.1.40</td>
</tr>
<tr>
<td>rhs</td>
<td>RHS of system of equations to be solved</td>
<td>7.1.37</td>
</tr>
<tr>
<td>statistics</td>
<td>Mean and Std deviation of some variables</td>
<td>7.1.49</td>
</tr>
<tr>
<td>strain</td>
<td>element strain</td>
<td>7.1.16</td>
</tr>
<tr>
<td>stress</td>
<td>element stress</td>
<td>7.1.18</td>
</tr>
<tr>
<td>stress = gp</td>
<td>element stresses at Gauss points</td>
<td>7.1.17</td>
</tr>
<tr>
<td>principal_stresses</td>
<td>element principal stress vectors and magnitudes</td>
<td>7.1.19</td>
</tr>
<tr>
<td>signed_vonmises</td>
<td>element signed von Mises</td>
<td>7.1.21</td>
</tr>
<tr>
<td>temperature</td>
<td>element centroid temperature</td>
<td>7.1.53</td>
</tr>
<tr>
<td>velocity</td>
<td>velocity at nodes</td>
<td>7.1.14</td>
</tr>
<tr>
<td>voltage</td>
<td>voltage at nodes</td>
<td>7.1.54</td>
</tr>
<tr>
<td>vonMises</td>
<td>element von Mises stress</td>
<td>7.1.20</td>
</tr>
<tr>
<td>vrms</td>
<td>RMS quantities (random vibration only)</td>
<td>7.1.25</td>
</tr>
<tr>
<td>WarningLevel</td>
<td>Control of warning messages</td>
<td>N/A</td>
</tr>
</tbody>
</table>

7.1.1. **Surface Projection of Element Variables**

Element output such as stress, strain or temperature is evaluated at the element centroid. Hexahedra, tetrahedra and prisms support the projection of some element quantities to
Figure 7-60. – Convergence of maximum stress at element centroids and surfaces.

element faces. We do this by sampling the gradients of the element shape functions at the local coordinates associated with that face’s centroid.

Notes:

1. Surface Projection is triggered by the output_sideset_data parameters flag.
2. Surface Projection results are stored as sideset variables on the Exodus mesh.
3. Only some outputs have been enabled for surface projection, most notably the strain tensor, stress tensor, and von Mises stress.
4. Due to the nature of bubble shape functions when sampled away from the element centroid, the default Hex8 element (Hex8b) will report skewed surface projection results, and is not expected to give the same benefit as other volumetric elements, given the same displacement field.
5. Even when using the bubble hex element, surface projection is expected to give a more accurate representation of maximum stress, given that the maximum stress occurs on a sideset. See figure 7-60.
7.1.2. Database Name

Option **database name** in the **outputs** section allows the user to specify an output file name instead of the default behavior as described above. As before, `-out` (or the case name for multi-case solutions) will be inserted before the file extension.

For instance, where the geometry file name `example.exo` would previously result in the output file `example-out.exo`, the user could instead specify `database name = new_example.exo`, which would result in output written to `new_example-out.exo`.

7.1.3. Properties

**Exodus** Output Properties is currently BETA release.

Enable with the “--beta” command-line option.

A number of options are available to control the contents of **Exodus** output files. These options can be used to shrink the output size or change the format of the output file. All of these properties are preceded by the **property** keyword in any input deck. Available properties are given in Table 7-125.

<table>
<thead>
<tr>
<th>Property Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>integer_size_db</td>
<td>Number of bytes for integers in <strong>Exodus</strong> output. Valid sizes are 4 or 8. By default, the size is the same as used in the input mesh file. 8 byte integers are required if node/element IDs greater than about two billion are used.</td>
</tr>
<tr>
<td>real_size_db</td>
<td>Number of bytes for real numbers in <strong>Exodus</strong> output. Valid sizes are 4 or 8. Default is 8, double precision. Use of 4 byte single precision output will roughly halve the output size but reduce output accuracy. Of particular note, Do not use four byte reals for restart.</td>
</tr>
<tr>
<td>compression_level</td>
<td>Compression level can be given values from zero to nine. Zero is no compression and nine is maximum compression. The default is generally zero compression. The compression level provides only a suggestion to the <strong>Exodus</strong> library, the actual amount of compression will vary. Potentially compression can reduce the size of the file by a factor of three or more. Compressed files may be incompatible with some post-processors.</td>
</tr>
</tbody>
</table>

The below example parameters would provide minimal output file size for a run.
7.1.4. Maa

Option Maa selects the analysis-set mass matrix (if it exists) for output to the file named Maa.m. These are the file names for the serial version of Sierra/SD. In the parallel version, an underscore and the processor number will precede the .m, and a separate file will be written for each processor.

7.1.5. Material

The material keyword will output material properties for each element. Currently, this capability is only enabled for elasticity calculations, and is also not enabled for Lamé materials or isotropic_viscoelastic_complex materials.

7.1.6. material_direction

Local coordinate systems may be defined to orient directional materials (sections 2.7 and 4.7.1 and section 4.7.2.3). The material_direction_1, material_direction_2, and material_direction_3 outputs provide the element local coordinate system (\(\hat{r}, \hat{s}, \hat{t}\)) vectors at each element. Visualizing these vectors can help inform if material coordinate systems have been setup as intended. Additionally, these coordinate vectors are the rows of a 3 \times 3 rotation matrix for transforming quantities between the global (X, Y, Z) and local (\(\hat{r}, \hat{s}, \hat{t}\)) coordinate systems within the element.

7.1.7. Kaa

Option Kaa will output the analysis-set stiffness matrix to a file named Kaa.m. These are the file names for the serial version of Sierra/SD. In the parallel version, an underscore and the processor number will precede the .m, and a separate file will be written for each processor.
7.1.8. Faa

Option faa will output the analysis-set force vector (if it exists) to a file named Faa.m. These are the file names for the serial version of Sierra/SD. In the parallel version, an underscore and the processor number will precede the .m, and a separate file will be written for each processor.

7.1.9. MLumped

Option MLumped will output the lumped mass matrix to the Exodus mesh as a nodal variable. MLumped is only implemented for the Eigen solution case. The lumped mass output is based on the analysis-set reduced mass matrix. Thus, mass on fixed degrees of freedom will be zero.

<table>
<thead>
<tr>
<th>MLumped Variable Names</th>
</tr>
</thead>
<tbody>
<tr>
<td>M_X (x translation)</td>
</tr>
<tr>
<td>M_Y (y translation)</td>
</tr>
<tr>
<td>M_Z (z translation)</td>
</tr>
<tr>
<td>M_RX (x rotation)</td>
</tr>
<tr>
<td>M_RY (y rotation)</td>
</tr>
<tr>
<td>M_RZ (z rotation)</td>
</tr>
<tr>
<td>M_A (acoustic)</td>
</tr>
<tr>
<td>M_V (voltage)</td>
</tr>
<tr>
<td>M_T (temperature)</td>
</tr>
</tbody>
</table>

7.1.10. MPhi

Option MPhi triggers computation and output of the mass matrix product $M\Phi$ to the Exodus file for the mode shapes, $\Phi$, computed in the previous eigen solution. MPhi is only implemented for the Eigen solution case. A consistent mass matrix is used. Mass matrix output is explained in Section 7.1.4. Like the mode shapes, the mass matrix is defined only on the analysis-set dofs. An ASetMap is also provided. The $M\Phi$ vanishes on fixed dofs. The names of the different dofs at a node are specified in Table 7.1.10.
7.1.11. **ElemEigChecks**

Option **ElemEigChecks** will turn on the element output of the lowest eigenvalue, the 7th eigenvalue (commonly the first flexible eigenvalue), and the largest eigenvalue for the element. The output will be stored in the *Exodus* output file. The element variable names for the 1st eigenvalue, the 7th eigenvalue, and the maximum eigenvalue are `ElemEig_1st`, `ElemEig_7th`, and `ElemEig_max`, respectively. Note: All 3-d and 2-d elements have this capability. The **Beam2**, **TiBeam**, **Spring**, **Truss**, **Spring3**, and **RSpring** elements are also supported. All remaining elements will output values of zero. Finally, if `ElemEig_1st < -1e-12 ElemEig_max`, a negative eigenvalue warning will be printed.

7.1.12. **ElemQualchecks**

Option **ElemQualchecks** takes a boolean, e.g. the default on or off. Unless this option is off, all the elements in the input file are checked for quality using the **Sierra** element quality metrics. The following table shows the elements currently checked and their acceptable ranges. The element quality reporting may also be modified by the `condition_limit` parameter specified in the *Parameters* section (2.3).

<table>
<thead>
<tr>
<th>Element Type</th>
<th>Full Range</th>
<th>Acceptable Range</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hex8</td>
<td>1 – ∞</td>
<td>1 – 8</td>
</tr>
<tr>
<td>Tet4</td>
<td>1 – ∞</td>
<td>1 – 3</td>
</tr>
<tr>
<td>Tria3</td>
<td>1 – ∞</td>
<td>1 – 1.3</td>
</tr>
<tr>
<td>TriaShell</td>
<td>1 – ∞</td>
<td>1 – 1.3</td>
</tr>
<tr>
<td>Quad4</td>
<td>1 – ∞</td>
<td>1 – 4</td>
</tr>
<tr>
<td>Wedge6</td>
<td>1 – ∞</td>
<td>1 – 5</td>
</tr>
</tbody>
</table>

If the option on is selected and the element’s condition numbers falls outside the acceptable range, a warning message is printed. The value output with the warning is normalized by the maximum number of the acceptable range for that element. A summary is also printed, reporting the maximum condition number of all elements in the mesh.
In addition to these checks, solid elements are checked for negative volumes. This can occur if the node ordering for the element establishes a “height” vector using the right-hand rule that is in the opposite direction of the actual element height. In other words, the nodes should normally be ordered in a counter clockwise direction on the bottom surface of the element. Some codes such as NASTRAN, are insensitive to this ordering. If element checks are run, then Sierra/SD will correct (and report) any solid elements found to have negative volumes. Without these corrections, the code will continue, but results that depend on these elements are suspect.

It is strongly recommended that any Exodus file with negative volumes be corrected.

### 7.1.13. Displacement

Option disp will output the displacements calculated at the nodes to the output Exodus file. The output file has the following nodal variables.

<table>
<thead>
<tr>
<th>Variable</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>dispX</td>
<td>X component of displacement</td>
</tr>
<tr>
<td>dispY</td>
<td>Y component of displacement</td>
</tr>
<tr>
<td>dispZ</td>
<td>Z component of displacement</td>
</tr>
<tr>
<td>rotX</td>
<td>Rotation about X</td>
</tr>
<tr>
<td>rotY</td>
<td>Rotation about Y</td>
</tr>
<tr>
<td>rotZ</td>
<td>Rotation about Z</td>
</tr>
</tbody>
</table>

In addition, if the analysis involves complex variables (ceigen Section 3.21.2, frequency response analysis such as ModalFrF or sa_eigen), then the imaginary vectors are also included. The imaginary component of the vector has “imag” prefixed to the name. For example, the imaginary component in the X direction is “imagDispX”.

### 7.1.14. Velocity

Option velocity will output the velocities at the nodes to the output Exodus file.

### 7.1.15. Acceleration

Option acceleration will output the accelerations at the nodes to the output Exodus file.
7.1.16. Strain

Option **strain** will output the strains for all the elements to the output **Exodus** file. The total strain $\epsilon$, elastic strain $\epsilon^e$, and thermal strain $\epsilon^\theta$ are defined by:

$$\epsilon = \frac{1}{2} (\nabla u + \nabla u^T) \quad (7.1)$$

$$\epsilon^e = \epsilon - \epsilon^\theta \quad (7.2)$$

$$\epsilon^\theta_{ii} = \alpha \Delta T \quad (7.3)$$

They can be requested separately by using the options

- **strain**: $\epsilon$
- **elastic_strain**: $\epsilon^e$
- **thermal_strain**: $\epsilon^\theta$

Strains will be output for shell elements. The output variable names start **SStrain**. Shell elements output engineering strain. E.g. shells output $\epsilon_{xx}$, $\epsilon_{yy}$, $\tau_{xy}$ where $\tau_{xy} = 2\epsilon_{xy}$. This is distinct from solid elements which output strain as $\epsilon_{xx}$, $\epsilon_{yy}$, $\epsilon_{zz}$, $\epsilon_{xy}$, $\epsilon_{yz}$, $\epsilon_{xz}$. This difference in strain definition means that for an equivalent deformation a shell element will report twice the numerical shear strain that a solid element will report even though the two shear states are identical.

**SStrainX1, SStrainY1, SStrainXY1** - *top layer of the shell*

**SStrainX2, SStrainY2, SStrainXY2** - *mid-plane of the shell*

**SStrainX3, SStrainY3, SStrainXY3** - *bottom layer of the shell*

For elastic strain,

**SElasticStrainX1, SElasticStrainY1, SElasticStrainXY1** - *top layer of the shell*

**SElasticStrainX2, SElasticStrainY2, SElasticStrainXY2** - *mid-plane of the shell*

**SElasticStrainX3, SElasticStrainY3, SElasticStrainXY3** - *bottom layer of the shell*

and for thermal strains we have

**SThermalStrainX1, SThermalStrainY1, SThermalStrainXY1** - *top layer of the shell*

**SThermalStrainX2, SThermalStrainY2, SThermalStrainXY2** - *mid-plane of the shell*

**SThermalStrainX3, SThermalStrainY3, SThermalStrainXY3** - *bottom layer of the shell*

Strains are evaluated in the local element coordinate system. The local element coordinate system and also the ordering of the layers both depend on the ordering of the element’s nodes.

The following strains will be output for volume elements:

**VStrainX, VStrainY, VStrainZ, VStrainYZ, VStrainXZ, VStrainXY**
Likewise, for elastic and thermal strains, we have

\[ \text{VElasticStrain}_X, \text{VElasticStrain}_Y, \text{VElasticStrain}_Z, \]
\[ \text{VElasticStrain}_YZ, \text{VElasticStrain}_XZ, \text{VElasticStrain}_XY, \]
\[ \text{VThermalStrain}_X, \text{VThermalStrain}_Y, \text{VThermalStrain}_Z, \]
\[ \text{VThermalStrain}_YZ, \text{VThermalStrain}_XZ, \text{VThermalStrain}_XY. \]

Note: These strains are in the global coordinate system, not the local coordinate system.

For more information on stress/strain recovery, see Section 7.6.

7.1.17. \textbf{Strain = GP}

An output specification of \texttt{strain = GP} reports strain at the Gauss points of volumetric elements. For more information, see section 7.1.24.

7.1.18. \textbf{Stress}

Option \texttt{stress} will output the stresses for all supported elements to the output Exodus file.

This is the Cauchy stress \( \sigma^C = C : \varepsilon^e \); see equation 7.2.

7.1.18.1. \textbf{Shell Stresses}

The following stresses will be output for shell elements.

\[ \text{SSStress}_X1, \text{SSStress}_Y1, \text{SSStress}_XY1, \text{SvonMises}_1 - \text{top layer of the shell} \]
\[ \text{SSStress}_X2, \text{SSStress}_Y2, \text{SSStress}_XY2, \text{SvonMises}_2 - \text{mid-plane of the shell} \]
\[ \text{SSStress}_X3, \text{SSStress}_Y3, \text{SSStress}_XY3, \text{SvonMises}_3 - \text{bottom layer of the shell} \]

\textit{Note: the top layer of the shell is determined by the ordering of the nodes of the shell, and can be output by using the eorient output options (see Section 7.1.43). Also, the stresses are in the local element coordinate system defined by the ordering of the nodes.}
7.1.18.2. Volume Stresses

For volume elements, the stress is always output in the global coordinate system, not the local coordinate system. The following stresses will be output for volume elements:

<table>
<thead>
<tr>
<th>Variable</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>VStressX</td>
<td>$\sigma_{xx}$</td>
</tr>
<tr>
<td>VStressY</td>
<td>$\sigma_{yy}$</td>
</tr>
<tr>
<td>VStressZ</td>
<td>$\sigma_{zz}$</td>
</tr>
<tr>
<td>VStressYZ</td>
<td>$\sigma_{yz}$</td>
</tr>
<tr>
<td>VStressXZ</td>
<td>$\sigma_{xz}$</td>
</tr>
<tr>
<td>VStressXY</td>
<td>$\sigma_{xy}$</td>
</tr>
<tr>
<td>VonMises</td>
<td>von Mises stress</td>
</tr>
</tbody>
</table>

For more information on stress/strain recovery, see Section 7.6.

7.1.19. Principal Stresses

Option `principal_stresses` will output the three principal stress vectors and magnitudes in order of descending signed value. `Principal_stresses` are only supported for volume elements.

The magnitudes are output as:

Max_Principal_Stress, Intermediate_Principal_Stress, Min_Principal_Stress

The eigenvectors are output as:

Max_Principal_Stress_x, Intermediate_Principal_Stress_x, Min_Principal_Stress_x,
Max_Principal_Stress_y, Intermediate_Principal_Stress_y, Min_Principal_Stress_y,
Max_Principal_Stress_z, Intermediate_Principal_Stress_z, Min_Principal_Stress_z

7.1.20. von Mises stress

Option `vonMises` will output the von Mises stress for all the elements to the output Exodus file. For volume elements, the output will be the von Mises stress of the element as VonMises. Surface elements define stresses on the top, center and bottom layers. The von Mises stress is reported in the output exodus file at each of the 3 layers (SVonMises#) as well as the maximum over all layers (VonMises). See section 7.6.3 for more information on shell stress recovery. For beam elements, the von Mises stress is reported at each stress recovery point (VonMises_SRP#), as well as a maximum over all stress recovery points (VonMises). See section 7.6.4 for more information on beam stress recovery.
Note that the von Mises stress is computed and output as a portion of the output if full stress recovery is requested. This option provides a mechanism for reducing output. Thus, if full stress output is requested, then the vonMises will provide no additional output. In other words, specifying both vonMises and stress in the outputs section is redundant, but does not result in an error.

7.1.21. Signed von Mises Stress

Option Signed_VonMises will output the magnitude of von Mises stress, given the sign of the principal stress with the largest magnitude. Signed_VonMises is only supported for volume elements.

The output variable is:

Signed_VonMises

7.1.22. Rainflow Cycle Counting

Option rainflow triggers the rainflow cycle counter to track stress cycles encountered by each element over time. Rainflow relies on Signed_VonMises to convert the stress tensor to a scalar signal, and is intended as a preprocessing step to time domain fatigue calculations. Rainflow is only supported for volume elements, and only in transient analyses.

The output variables are:

NumCycles, LastCycleAmplitude, LastCyclePeak

These outputs contain insufficient information to be useful on their own except in simple verification exercises. Rainflow is intended to be a silent dependency of fatigue, rather than a standalone output option.

7.1.23. Fatigue Damage

Option fatigue will output a damage estimate for each element using the stress history of that element. This process is supported for transient solutions and for modal random vibration, but in very different ways.

In a transient analysis, fatigue damage is calculated using stress cycles identified by the rainflow algorithm, and applying those cycles to the Walker damage function:

$$ log_{10}(N) = A_1 + A_2 * log_{10}(S_{max}*(1 - R)^{A_3} - A_4) $$
Where $S_{\text{max}}$ is the peak stress of the cycle, $S_{\text{min}}$ is the minimum Stress of the cycle, and
$R = S_{\text{min}}/S_{\text{max}}$. The number of cycles to failure $N$ is then related to damage $D$ by:

$$D = 1/N$$

$A_1, A_2, A_3,$ and $A_4$ are material constants. Note that a cycle is ignored if $S_{\text{max}} <= 0$ or
$S_{\text{max}}*(1-R)^{A_3} <= A_4$, because purely compressive cycles are assumed to cause no
damage, and because cycles below the endurance limit causes no damage.

The output variable is:

**Damage**

In modal random vibration, *fatigue* is its own solution case. See section 3.13 for more
details. The output variables are:

| NarrowBandDamageRate, WirschingDamageRate, ZeroCrossingRate, PeakFrequency, Damage, Vrms |

### 7.1.24. Stress = GP

An output specification of Stress = GP reports stress at the Gauss points of volumetric
elements. It is currently only available for Hex20, Tet10, and Wedge15 elements. Note that
for a Hex20 there are 27 Gauss points with 6 stresses, for a total of 162 outputs per
element.

The Gauss point ordering follows the description in the paper by Thompson. For the
convenience of the reader, that order is reproduced here in Table 7-126.
<table>
<thead>
<tr>
<th>number</th>
<th>label suffix</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>111</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>112</td>
<td>0</td>
<td>0</td>
<td>A</td>
</tr>
<tr>
<td>3</td>
<td>110</td>
<td>0</td>
<td>0</td>
<td>-A</td>
</tr>
<tr>
<td>4</td>
<td>121</td>
<td>0</td>
<td>A</td>
<td>0</td>
</tr>
<tr>
<td>5</td>
<td>122</td>
<td>0</td>
<td>A</td>
<td>A</td>
</tr>
<tr>
<td>6</td>
<td>120</td>
<td>0</td>
<td>A</td>
<td>-A</td>
</tr>
<tr>
<td>7</td>
<td>101</td>
<td>0</td>
<td>-A</td>
<td>0</td>
</tr>
<tr>
<td>8</td>
<td>102</td>
<td>0</td>
<td>-A</td>
<td>A</td>
</tr>
<tr>
<td>9</td>
<td>100</td>
<td>0</td>
<td>-A</td>
<td>-A</td>
</tr>
<tr>
<td>10</td>
<td>211</td>
<td>A</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>11</td>
<td>212</td>
<td>A</td>
<td>0</td>
<td>A</td>
</tr>
<tr>
<td>12</td>
<td>210</td>
<td>A</td>
<td>0</td>
<td>-A</td>
</tr>
<tr>
<td>13</td>
<td>221</td>
<td>A</td>
<td>A</td>
<td>0</td>
</tr>
<tr>
<td>14</td>
<td>222</td>
<td>A</td>
<td>A</td>
<td>A</td>
</tr>
<tr>
<td>15</td>
<td>220</td>
<td>A</td>
<td>A</td>
<td>-A</td>
</tr>
<tr>
<td>16</td>
<td>201</td>
<td>A</td>
<td>-A</td>
<td>0</td>
</tr>
<tr>
<td>17</td>
<td>202</td>
<td>A</td>
<td>-A</td>
<td>A</td>
</tr>
<tr>
<td>18</td>
<td>200</td>
<td>A</td>
<td>-A</td>
<td>-A</td>
</tr>
<tr>
<td>19</td>
<td>011</td>
<td>-A</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>20</td>
<td>012</td>
<td>-A</td>
<td>0</td>
<td>A</td>
</tr>
<tr>
<td>21</td>
<td>010</td>
<td>-A</td>
<td>0</td>
<td>-A</td>
</tr>
<tr>
<td>22</td>
<td>021</td>
<td>-A</td>
<td>A</td>
<td>0</td>
</tr>
<tr>
<td>23</td>
<td>022</td>
<td>-A</td>
<td>A</td>
<td>A</td>
</tr>
<tr>
<td>24</td>
<td>020</td>
<td>-A</td>
<td>A</td>
<td>-A</td>
</tr>
<tr>
<td>25</td>
<td>001</td>
<td>-A</td>
<td>-A</td>
<td>0</td>
</tr>
<tr>
<td>26</td>
<td>002</td>
<td>-A</td>
<td>-A</td>
<td>A</td>
</tr>
<tr>
<td>27</td>
<td>000</td>
<td>-A</td>
<td>-A</td>
<td>-A</td>
</tr>
</tbody>
</table>

Table 7-126. – Hex20 Gauss Point Locations. The constant $A=0.7745966924148$. The unit element is 2x2x2, with a volume of 8 cubic units.
7.1.25.  **Vrms**

Option *vrms* will output computed root mean squared (RMS) quantities from a random vibration analysis. These quantities are written to a separate output file. Quantities output include the RMS displacement, acceleration and von Mises stress. With the SVD option, the $D$ matrix terms\textsuperscript{48} which contribute to the von Mises stress are also output (see Section 3.18).

7.1.26.  **Rotational_displacement**

Option *rotational_displacement* will output computed root mean squared (RMS) quantities for rotational displacement in a random vibration analysis to the output file. In the frequency section, it will output the corresponding Power Spectral Densities to the frequency file. (See section 3.18).

7.1.27.  **Rotational_acceleration**

Option *rotational_acceleration* will output computed root mean squared (RMS) quantities for rotational acceleration in a random vibration analysis to the output file. In the frequency section, it will output the corresponding Power Spectral Densities to the frequency file. (See section 3.18).

7.1.28.  **Energy**

Option *energy* will place strain energies and strain energy density in the output *Exodus* file. Note that the current implementation of strain energies requires updating the element stiffness matrix, which can be expensive.

7.1.29.  **GEnergies**

Option *GEnergies* in either the echo or outputs will trigger computation of global energy sums for the results or output *Exodus* file, respectively.

**strain energy** The strain energy is computed from $u^T Ku/2$ where $u$ is the displacement and $K$ is the current estimate of the tangent stiffness matrix.

**kinetic energy** Computed as $v^T Mv/2$. Here $v$ is the velocity and $M$ is the mass matrix.
work As a particle moves along $x(t)$ in the force field $F$, it does work

$$W(t) = \int_{x(0)}^{x(t)} F(x)dx$$

$$= \int_0^t F(\tau)\dot{x}(\tau)d\tau$$

The simplest possible approximation is used,

$$W_n \sim \sum_{i=0}^n F_i\dot{x}_i\Delta t.$$  

Integral approximation errors may introduce inconsistencies with the other energies. For the outputs case, the total energy is written out at each time step.

Known Issues:

- The strain energy may not be complete for nonlinear solutions. Linear viscoelastic materials have contributions that will not be included in this sum.
- The strain energy and work are currently not computed correctly in models with non-zero displacement, velocity, or acceleration boundary conditions.
- When used in statics, GEnergies only works with echo, not results output.

7.1.30. Globals

Option Globals in the outputs section will ensure that any global data is output, even if no other outputs are requested. Without this option (and with an empty output block), no output file would be written. If no global data is defined for a given solution type, or if any other outputs are requested, this option will have no effect. This option is also available for history, frequency, and statistics output, and behaves similarly.

7.1.31. Block_Energies

Option block_energies in the echo or outputs will trigger block-wise energy sums for the results or output Exodus file, respectively. The energy computations are done as described in 7.1.29, where the displacement and velocity vectors have been restricted to the element block. Kinetic and strain energies are computed. Global variables in the Exodus file are “KineticEnergy_” and “StrainEnergy_” with the block name appended.
7.1.32. Mesh_Error

The mesh_error keyword causes mesh discretization error metrics to be computed. These are computed as output quantities, but the overhead associated with the metrics is not negligible. Mesh discretization quantities depend upon the solution type, and are not available for all solutions. Output is typically available as element quantities (usually in the mesherr field). For some mesh discretization errors, a global quantity is also output. See [62] for a detailed description of the MeshErr calculation.

<table>
<thead>
<tr>
<th>Output</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ErrExplicitLambda</td>
<td>Relative error in $\lambda$.</td>
</tr>
<tr>
<td>ErrExplicitFreq</td>
<td>Frequency error estimate (Hz)</td>
</tr>
</tbody>
</table>

We note that for eigenvalue analysis, relative errors are reported for the eigenvalue when using the mesh_error keyword. Thus, for a given eigenvalue $\lambda$, the reported error is

$$\text{ErrExplicitLambda} = \frac{\lambda_h - \lambda}{\lambda}$$  \hspace{1cm} (7.4)

This is more convenient since the analyst does not have to divide by the eigenvalues to see the percent error. The global variable ErrExplicitFreq provides an absolute estimate (useful in plots for example).

The error ($R$) for a single element ($K$) is given as

$$R_K(u_h, \theta_h) = \nabla \cdot \sigma(u_h) + \theta_h \rho u_h + \sum_f \frac{1}{2} R_F$$ \hspace{1cm} (7.5)

where $\theta$ is the eigenvalue, $\rho$ is the density, and $u$ is the displacement for mode $h$. $R_F$ is the error on a single face

$$R_F(u_h) = J_F(N_F \cdot \sigma(u_h))$$ \hspace{1cm} (7.6)

where $J_F$ is the jump of the stress across the element boundary in the direction of the element normal. Then, the global error estimate is written as:

$$\text{MeshErr} = \sqrt{\sum_e \frac{h_K^2}{p^2 d_{K,\text{min}}} ||R_K||^2 + \sum_f \frac{h_F}{pd_{F,\text{min}}} ||R_f||^2}$$ \hspace{1cm} (7.7)

where $h$ is the element length, $p$ is the element order, and $d$ is the maximum eigenvalue of the element or face.

7.1.33. MFile

Option MFile will cause Sierra/SD to output various MFiles like Ksrr.m, Mssr.m, etc. Using MFile with a model with a large numbers of elements is challenging due to the large size of the corresponding MATLAB files. A partial index of the files written using this option is provided in Table 7-127.
Table 7-127. – Data Files Written Using the MFile Option.

<table>
<thead>
<tr>
<th>Filename</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Stiff.m</td>
<td>Unreduced stiffness matrix including all active dofs</td>
</tr>
<tr>
<td>Kssr.m</td>
<td>Reduced stiffness matrix</td>
</tr>
<tr>
<td>Mass.m</td>
<td>Unreduced mass matrix</td>
</tr>
<tr>
<td>Mssr.m</td>
<td>Reduced mass matrix</td>
</tr>
<tr>
<td>LumpedMass.m</td>
<td>unreduced lumped mass matrix</td>
</tr>
<tr>
<td>xxx_gid.m</td>
<td>global IDs of the nodes</td>
</tr>
<tr>
<td>ASetmap_a.m</td>
<td>Map to convert from G-set to A-set</td>
</tr>
<tr>
<td></td>
<td>The right-hand side is the equation number.</td>
</tr>
<tr>
<td></td>
<td>The left-hand index is 9*(node index)+coordinate</td>
</tr>
<tr>
<td>Dampr.m</td>
<td>unreduced damping matrix (real components)</td>
</tr>
<tr>
<td>Dampi.m</td>
<td>unreduced damping matrix (imaginary components)</td>
</tr>
<tr>
<td>xxx_accelN.m</td>
<td>G-set acceleration output of step N</td>
</tr>
<tr>
<td>xxx_accel_aN.m</td>
<td>A-set acceleration output of step N</td>
</tr>
<tr>
<td>xxx_afN.m</td>
<td>G-set applied force output of step N</td>
</tr>
<tr>
<td>xxx_af_aN.m</td>
<td>A-set applied force output of step N</td>
</tr>
<tr>
<td>xxx_dispNN.m</td>
<td>G-set displacement output of step N</td>
</tr>
<tr>
<td>xxx_disp_aN.m</td>
<td>A-set displacement output of step N</td>
</tr>
<tr>
<td>xxx_presN.m</td>
<td>G-set nodal applied pressure of step N</td>
</tr>
<tr>
<td>xxx_pres_aN.m</td>
<td>A-set nodal applied pressure of step N</td>
</tr>
<tr>
<td>xxx_velocN.m</td>
<td>G-set velocity output of step N</td>
</tr>
<tr>
<td>xxx_veloc_aN.m</td>
<td>A-set velocity output of step N</td>
</tr>
<tr>
<td>modal_amp.m</td>
<td>modaltransient output of mode amplitude vs time</td>
</tr>
</tbody>
</table>

- Above the xxx refers to the input file name root.
- G-set output has dimension 9 (number of nodes).
- The file names above are for the serial version of Sierra/SD. In the parallel version, an underscore and the processor number will precede the “.m”. For example, the reduced stiffness matrix becomes Kssr_0.m. Matrices are never globally assembled, and so cannot be output.
- Some solution methods will not write all files. For example, there are no mass matrices output in the solution of statics. Generally, matrices are output in sparse symmetric row format.
- The 1 to N node ordering of the input Exodus file defines the ASetMap. Output file ordering may be different if there is a node order map.
7.1.33.1. Example of Using Matlab Outputs

As an example, consider obtaining the Y component stiffness matrix diagonal entry of global node 77. The maps may be used as follows.

- Search through xxx_gid.m to find the global node number 77. Call index at which the node is found is \textit{inode}.

- Calculate the GSet index, which in this case is \textit{igset} = (\textit{inode} \times 9) + 2. The 9 is for 9-DOFs per node. The 2 is for Y being the second DOF of the nine (X, Y, Z, RotX, RotY, RotZ, Acoustic, Voltage, Temperature.)

- Lookup the Aset index, \textit{iaset} = \textit{ASetmap(igset)}

- If \textit{iaset} is zero, then there is no defined stiffness matrix diagonal (could be an undefined DOF, or a fixed DOF.) Otherwise the stiffness diagonal is \( kaa(iaset,iaset) \).

7.1.34. Force

Option \textbf{force} will output the applied force vector to the output Exodus file, and the net force applied to active degrees of freedom, and the net moment about the origin of forces applied to active degrees of freedom. The net force is calculated from the right-hand side given to the solver, so implementations that modify the right-hand side (cavitation) may display non-physical net forces and moments. Net forces are available for static and direct transient solution cases.

If rigid body filtering is requested via the FilterRbmLoad option (section 6.3.19), this option will also output the filtered force to the output Exodus file, and the net force and moment about the origin. The naming convention for both nodal and net (global) values are \textit{force_inertia_relief} for forces, and \textit{moment_inertia_relief} for moments.

7.1.35. Constraint force

Option \textbf{constraint force} will output the forces required to tie blocks together due to multipoint constraints, RBE3s, Rbars, or rigid elements. Additionally, it will output the net constraint force on the active degrees of freedom, and the net moment about the origin of constraint forces on the active degrees of freedom. This output is currently only active for the static and transient solution cases and for the GDSW solver.

7.1.36. Reaction Force

Option \textbf{reaction force} will output the Dirichlet boundary reaction force vector to the output Exodus file. Additionally, it will output the net reaction force on the active degrees of freedom, and the net moment about the origin of reaction forces on the active degrees of freedom. This output is currently only active for the static and transient solution cases and for the GDSW solver.
7.1.37. Right-hand side

This output is useful primarily for verification and debugging purposes. Option **RHS** selects the right-hand side vector for the analysis type. For statics and dynamics, the **RHS** vector is the applied forces, pressures, inertial forces, or any pseudo forces introduced in preload (say by TSR).

7.1.38. EForce

Option **eforce** will output the element forces for line elements (such as beams and springs) to the output **Exodus** file. Each two node, 1-dimensional element will have 6 force entries for each node, for a total of 12 element forces per element, and an additional 3 variables describing the difference in displacement across the element.

The element force is not a stress or a strain, and should not be used as such. If you want beam stresses, you may want to mesh that portion of the structure either as a shell or a solid. Only limited stress output is available for beams. EForce is used primarily to help understand the behavior of nonlinear line elements such as the Joint2g element (see Section 5.21). The output is the direct output of our internal force routine (which is a nonlinear routine). It can be confusing to output these nonlinear forces in a linear analysis. 7

Eforce variable names are different for each solution case. When requested in typical analyses, such as **statics**, **transient**, or **eigen**, **eforce** produces the variables in table 7-128. For **FRF** solutions, “Imag” is prepended to the existing names 7-129. For **modalranvib**, only translational forces are output, and are different between **outputs** and **frequency** files. In an **outputs** result file, you will see the RMS of the force in the translation directions 7-130. In **frequency**, the Spectral Densities of the element force are represented by a Hermitian tensor at each frequency 7-131.

---

7 Confusion arises because of the transformation to the element coordinate frame. For finite length elements, we perform a transformation of the element coordinate frame based on the displacements. After the coordinate frame is transformed, we express the element force in the new coordinate frame. This is done for both linear and nonlinear analyses. The resulting element force is no longer linear in displacement. Zero length elements do not have a rotated coordinate frame by default. Forces for zero length elements are linear in the displacement.
<table>
<thead>
<tr>
<th>Variable Names</th>
</tr>
</thead>
<tbody>
<tr>
<td>eforce1_x</td>
</tr>
<tr>
<td>eforce1_y</td>
</tr>
<tr>
<td>eforce1_z</td>
</tr>
<tr>
<td>emoment1_x</td>
</tr>
<tr>
<td>emoment1_y</td>
</tr>
<tr>
<td>emoment1_z</td>
</tr>
<tr>
<td>eforce2_x</td>
</tr>
<tr>
<td>eforce2_y</td>
</tr>
<tr>
<td>eforce2_z</td>
</tr>
<tr>
<td>emoment2_x</td>
</tr>
<tr>
<td>emoment2_y</td>
</tr>
<tr>
<td>emoment2_z</td>
</tr>
<tr>
<td>e_dx</td>
</tr>
<tr>
<td>e_dy</td>
</tr>
<tr>
<td>e_dz</td>
</tr>
</tbody>
</table>

Table 7-128. – Typical Output.

<table>
<thead>
<tr>
<th>Real Names</th>
<th>Imaginary Names</th>
</tr>
</thead>
<tbody>
<tr>
<td>eforce1_x</td>
<td>Imageforce1_x</td>
</tr>
<tr>
<td>eforce1_y</td>
<td>Imageforce1_y</td>
</tr>
<tr>
<td>eforce1_z</td>
<td>Imageforce1_z</td>
</tr>
<tr>
<td>emoment1_x</td>
<td>Imagemoment1_x</td>
</tr>
<tr>
<td>emoment1_y</td>
<td>Imagemoment1_y</td>
</tr>
<tr>
<td>emoment1_z</td>
<td>Imagemoment1_z</td>
</tr>
<tr>
<td>eforce2_x</td>
<td>Imageforce2_x</td>
</tr>
<tr>
<td>eforce2_y</td>
<td>Imageforce2_y</td>
</tr>
<tr>
<td>eforce2_z</td>
<td>Imageforce2_z</td>
</tr>
<tr>
<td>emoment2_x</td>
<td>Imagemoment2_x</td>
</tr>
<tr>
<td>emoment2_y</td>
<td>Imagemoment2_y</td>
</tr>
<tr>
<td>emoment2_z</td>
<td>Imagemoment2_z</td>
</tr>
<tr>
<td>e_dx</td>
<td>Image_dx</td>
</tr>
<tr>
<td>e_dy</td>
<td>Image_dy</td>
</tr>
<tr>
<td>e_dz</td>
<td>Image_dz</td>
</tr>
</tbody>
</table>

Table 7-129. – FRF Output.

<table>
<thead>
<tr>
<th>Real Names</th>
<th>Imaginary Names</th>
</tr>
</thead>
<tbody>
<tr>
<td>eforceRMS_x</td>
<td>ImageforceRMS_x</td>
</tr>
<tr>
<td>eforceRMS_y</td>
<td>ImageforceRMS_y</td>
</tr>
<tr>
<td>eforceRMS_z</td>
<td>ImageforceRMS_z</td>
</tr>
</tbody>
</table>

Table 7-130. – ModalRanVib Exodus Output.
### Table 7-131. ModalRanVib Frequency Output.

<table>
<thead>
<tr>
<th>Real Names</th>
<th>Imaginary Names</th>
</tr>
</thead>
<tbody>
<tr>
<td>eforce_xx</td>
<td>Imageforce_xx</td>
</tr>
<tr>
<td>eforce_yy</td>
<td>Imageforce_yy</td>
</tr>
<tr>
<td>eforce_zz</td>
<td>Imageforce_zz</td>
</tr>
<tr>
<td>eforce_xy</td>
<td>Imageforce_xy</td>
</tr>
<tr>
<td>eforce_yz</td>
<td>Imageforce_yz</td>
</tr>
<tr>
<td>eforce_xz</td>
<td>Imageforce_xz</td>
</tr>
</tbody>
</table>

7.1.39. **Line_Weld**

Option Line_Weld outputs line-weld-specific outputs as described in Section 5.22. The outputs include whether a given beam is in an active weld, and the force per unit length produced by the weld.

NOTE: The force returned is in the element (not global) coordinate frame.

7.1.40. **Relative_Disp**

Option relative_disp in the outputs, history, and/or frequency section(s) will output the relative displacement between the two nodes of 1-D elements in the model. That is, it will output the difference in displacements across the 1-D element. Currently, relative_disp is only supported for the Joint2G element type, and is not supported for FRF solutions. Relative rotations are not available. For solutions other than modalranvib, including FRF, this data is also accessible through eforce.

relative_disp is output in the element-local coordinate system if a coordinate system has been defined for the Joint2G block, and in the global coordinate system otherwise. This is consistent with eforce outputs, but differs from displacement outputs. Using the coordinate keyword in the outputs, history, or frequency will not effect this output.

relative_disp is particularly useful for the modalranvib solution case, where Sierra/SD does not typically calculate or output enough information to post-process the difference in displacement between two points. The names of relative_disp outputs also change in modalranvib results. See tables 7-132, 7-133, and 7-134.

relative_disp outputs are available in PSD form if requested in the frequency section during modalranvib (Table 7-133). Note that the diagonal relativeDisp terms are output for both real and imaginary components. Hermitian symmetry is assumed in the calculation of relative Disp outputs, and incorrect results will be reported if this assumption is broken. Neither the input load, nor the relativeDisp output should have non-zero imaginary terms along the diagonal. That is, ImagRelDispGxx.
### Table 7-132. – Typical Output.

<table>
<thead>
<tr>
<th>Real Names</th>
<th>Imaginary Names</th>
</tr>
</thead>
<tbody>
<tr>
<td>RelDispGxx</td>
<td>ImagRelDispGxx</td>
</tr>
<tr>
<td>RelDispGyy</td>
<td>ImagRelDispGyy</td>
</tr>
<tr>
<td>RelDispGzz</td>
<td>ImagRelDispGzz</td>
</tr>
<tr>
<td>RelDispGxy</td>
<td>ImagRelDispGxy</td>
</tr>
<tr>
<td>RelDispGyz</td>
<td>ImagRelDispGyz</td>
</tr>
<tr>
<td>RelDispGxz</td>
<td>ImagRelDispGxz</td>
</tr>
</tbody>
</table>

### Table 7-133. – ModalRanVib Frequency Output.

ImagRelDispGyy, and ImagRelDispGzz should all be zero. The warning message “Correlation Matrix is negative” indicates you may have this problem.

<table>
<thead>
<tr>
<th>Real Names</th>
<th>Imaginary Names</th>
</tr>
</thead>
<tbody>
<tr>
<td>RelDispRMSX</td>
<td>ImagRelDispRMSX</td>
</tr>
<tr>
<td>RelDispRMSY</td>
<td>ImagRelDispRMSY</td>
</tr>
<tr>
<td>RelDispRMSZ</td>
<td>ImagRelDispRMSZ</td>
</tr>
</tbody>
</table>

### Table 7-134. – ModalRanVib Exodus Output.

relative _disp_ outputs are available in RMS form if requested in the outputs section during modalranvib (Table 7-134). Only diagonal terms are reported. The imaginary diagonal terms are set to 0.

#### 7.1.41. Residuals

For most solution types, a linear solver is used to compute systems of the form $Ax = b$. For direct serial solvers, these systems are typically solved to numerical precision. However, with iterative solvers the solution is only approximate. Sometimes it is advantageous to evaluate the performance of the solver. For example, regions with large residuals may be candidate areas for mesh refinement, or may point to other mesh problems.

**Eigen.** For eigen analysis, the residual is $(K - \lambda_i M)\phi$. The vector is not normalized by the norm of $\phi$, or any other quantity. A nodal residual work is also output. This is the product $\phi^T(K - \lambda_i M)\phi$ summed to the nodes, i.e. on a given node we sum the contributing degrees of freedom. Again, the value is not normalized. With mass normalized eigenvectors (which do not have units of length), the units of the residual work are not energy, and the term may well be negative. The residual is output for each mode.
**Transient Dynamics.** For transient analysis the residual reported is $Au - b$, where $A$ is the dynamics stiffness matrix defined in subsection Linear transient analysis section Solution Procedures of the Theory Manual.

With a displacement based Newmark-Beta integrator the dynamic stiffness is $K + \frac{2}{\Delta t}C + \frac{1}{\Delta t^2}M$. The residual is output at each time step.

In addition to the residual vector, the norm of the residual is output as a global variable.

### 7.1.42. TIndex

It is occasionally useful to examine the residual after each iteration or solve. In the cases of nonlinear transient or nonlinear statics analysis, there may be many solves per output. Because of limitations in the output database format, it is difficult (or impossible) to intersperse the residuals from each solve with the usual solution output. However, it is possible to select between the standard time step and an “iteration time step”. Note that the Exodus database writes output for each “time step”. It uses the step number as an index to the data, and only one such index is supported. When we substitute the iteration number for the time step we can write the data properly, but once iteration has completed, we may not write data using the other index (time step, or mode number). Should that occur, we would have residuals from one iteration sharing the same time axis index with transient data. The parameters for the option are listed in Table 7-135.

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Application</th>
</tr>
</thead>
<tbody>
<tr>
<td>standard</td>
<td>use time step or mode number as index</td>
</tr>
<tr>
<td>iteration</td>
<td>use the iteration count as index</td>
</tr>
</tbody>
</table>

**Table 7-135.** – TIndex parameters.

```plaintext
OUTPUT
  disp
  residuals
  Tindex=iteration // output on each iteration
END
```

**Input 7.1. TIndex example**

TIndex makes sense only in solutions that require multiple iterations per solve, such as nonlinear solutions. In other solutions, it is ignored, and output is provided at the standard time step.

**NOTE:** TIndex is a debugging function. As such, we do minimal checks. In some solutions, it might be possible to output data using both steps.
7.1.43. **EOrient**

Option **EOrient** in the outputs will output the element orientation vectors for all elements. The element orientation is a design quantity that normally does not change significantly through the course of an analysis. This output is provided to help in model construction and debugging.

The orientation vectors are output as nine variables that collectively make up the three vectors required for element orientation. The output variables and the associated meanings for various elements are shown in tables 7-136 and 7-137 respectively.

<table>
<thead>
<tr>
<th>Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>EOrient1_X</td>
<td>first orientation vector</td>
</tr>
<tr>
<td>EOrient1_Y</td>
<td></td>
</tr>
<tr>
<td>EOrient1_Z</td>
<td></td>
</tr>
<tr>
<td>EOrient2_X</td>
<td>second orientation vector</td>
</tr>
<tr>
<td>EOrient2_Y</td>
<td></td>
</tr>
<tr>
<td>EOrient2_Z</td>
<td></td>
</tr>
<tr>
<td>EOrient3_X</td>
<td>third orientation vector</td>
</tr>
<tr>
<td>EOrient3_Y</td>
<td></td>
</tr>
<tr>
<td>EOrient3_Z</td>
<td></td>
</tr>
</tbody>
</table>

Table 7-137. – Element Orientation Interpretation.

<table>
<thead>
<tr>
<th>Element</th>
<th>EOrient1</th>
<th>EOrient2</th>
<th>EOrient3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Beam2</td>
<td>axial</td>
<td>first bending (I1)</td>
<td>2nd bending (I2)</td>
</tr>
<tr>
<td>Shells</td>
<td>Element X</td>
<td>Element Y</td>
<td>Normal</td>
</tr>
<tr>
<td>Solids</td>
<td>Element X</td>
<td>Element Y</td>
<td>Element Z</td>
</tr>
<tr>
<td>Hexshell</td>
<td>Element X</td>
<td>Element Y</td>
<td>thickness</td>
</tr>
<tr>
<td>ConMass</td>
<td>null</td>
<td>null</td>
<td>null</td>
</tr>
</tbody>
</table>

7.1.44. **Pressure**

Option **pressure** in the outputs selects applied pressure output to the Exodus file as both a sideset variable and a new nodeset variable. The addition of nodeset pressure output enables restarts using the output pressure as an input load. For most applications this also provides a useful tool for checking input loads.
7.1.45. NPressure

Option npressure in the outputs will output the nodal pressure to the output Exodus file as a nodal variable. This output is only available for solutions that introduce nodal pressure (currently only the random pressure loading).

7.1.46. APressure

Option APRESSure in the outputs will output the acoustic pressure to the output Exodus file as a nodal variable. For purely acoustic elements, this will result in one degree of freedom per node. At the wetted interface, the nodes of node-face interactions inherit the degrees of freedom from the face, this will result in four degrees of freedom per node in the output Exodus file.

7.1.47. APartVel

Option APartVel in the outputs will output the acoustic particle velocity to the output Exodus file as an element variable. This is the velocity of the fluid particles. It is computed in Sierra/SD as the gradient of the velocity potential. For purely acoustic elements, this will result in three degrees of freedom per element.

7.1.48. Constraint_Info

Linear system solvers are sensitive to redundant or inconsistent constraints. Keyword constraint_info selects constraint information on the nodes. The information is useful in preparing models that are less sensitive to redundant constraints. Redundancy can be generated in any of the constraint types or intersections between types.

Constraint information has several fields.

MPC_Status Indicates if a given node is involved in any MPC equation. A value of '1' indicates the node is used in at least one equation, '0' otherwise.

MPC_Touched Indicates how many times a given node shows in MPC equations.

Node_Face_MPC_Count For only node-face contact constraints, this indicates how many times that a node is used as the node of a node on face constraint. A value greater than one can be problematic as a given node can only be correctly tied to a single face without introducing over-constraint.
**Node_Face_MPC_Redundancy** For only node-face contact constraints this highlights nodes that may be over-constrained. Pay attention to values greater than one. The **Node_Face_MPC_Redundancy** is typically one less than **Node_Face_MPC_Count** unless there are more than 3 independent constraints for a specific sideset pair, as may occur if a sideset pair is used in both a tied constraint and a slip contact, or Tied Joint.

**Node_Face_MPC_Gap** indicates the distance the node of a node-face constraint must be moved to be placed on the face. Many problems with constraints stem from surfaces that do not properly match up geometrically.

**Node_Face_MPC_Both_Node_and_Face** For node face constraints only this indicates if a particular node acts as a node in one node-face constraint and also is attached to the face of separate node-face constraint. A ’1’ means such situation occurs. In some cases such nodes may be part of a problematic cyclic constraint. However, in other cases this situation may be expected and cause no problems.

**MPC_Origin** Indicates what capability created the MPC. For example sliding contact vs. tied contact. The index number output matches a table in the rslt log file. Currently, this output is only available from node face contact constraints.

### 7.1.49. Statistics

For transient dynamics solutions only, summary statistical information may gathered and output for the time history of variables listed in Table 7-138. We gather information about the mean, the min/max, and the standard deviation. Data is gathered at each time step, independent of the frequency of output (e.g. \texttt{nskip} is ignored).

Because this is summary data, it is not convenient to append this data to the file used for output of the time history. Another file is generated with a \texttt{.stat} extension to store that data.

Statistical data requires \textit{two} keywords for output. Both “statistics” and the keyword associated with that output quantity must be selected. For example, to output statistics of the force, the following output section is required.

```
  Outputs
    statistics
    force
  end
```

As with output in general (section 7.1.2), users can define a custom statistical output file name via the \texttt{statistics} block:

```
  Statistics
    database name = <string>
  end
```
<table>
<thead>
<tr>
<th>Keyword</th>
<th>Section</th>
<th>Comment</th>
</tr>
</thead>
<tbody>
<tr>
<td>Displacement</td>
<td>7.1.13</td>
<td></td>
</tr>
<tr>
<td>Velocity</td>
<td>7.1.14</td>
<td></td>
</tr>
<tr>
<td>Acceleration</td>
<td>7.1.15</td>
<td></td>
</tr>
<tr>
<td>Force</td>
<td>7.1.34</td>
<td>applied force</td>
</tr>
<tr>
<td>RHS</td>
<td>7.1.37</td>
<td>Right-Hand Side vector at each load.</td>
</tr>
<tr>
<td>vonMises</td>
<td>7.1.18</td>
<td>von Mises stress, supported for echo output only.</td>
</tr>
</tbody>
</table>

Table 7-138. – Supported Statistical Data types for Transient Dynamics. Selection of these quantities along with “statistics” results in an addition Exodus file containing mean, min/max and standard deviation data.

7.1.50. KDiag

Option kdiag in the outputs will output the maximum and minimum values of the diagonal of the stiffness matrix as nodal variables KDiagMax and KDiagMin. These are the max and min of the 7 variables associated with the 3 translational, 3 rotational and 1 acoustic degree of freedom on each node. These values are primarily useful for diagnostics purposes, where they may help identify stiff regions of a model. All 7 terms may be seen by outputting KDiag in the echo section.

Figure 7-61 illustrates the use of this option. Note how the center sections of the model are highlighted by their stiffness terms. This tool is especially important for analyzing some collections of beams. Beam stiffness is proportional to $1/L^3$. A common mistake is to generate stiff beams, which can ruin the numerical solution. See Section 7.1.51 for a related diagnostic on the dynamics matrix. ¹

¹The stiffness diagonal and dynamic matrix diagonal depend to some extent on the linear solver used. Domain decomposition solvers generally use Lagrange multipliers to eliminate constraints, while some sparse solvers remove constraints through reductions of rows and columns of the matrices. Because the matrices to be solved are different, the diagonals and conditioning of the matrices are also different.
7.1.51. ADiag

Option ADiag in the outputs will output the maximum and minimum values of the diagonal of the dynamics matrix as nodal variables ADiagMax and ADiagMin. Refer to the KDiag section, (7.1.50), for format information.

The “dynamic matrix” is the matrix which is solved by the linear solver. The “ADiag” diagnostic can help identify regions of the model that may contribute to poor matrix conditioning. Summary of a few of the dynamics matrix terms are listed in Table 7-139. Refer to the theory manual for details of the matrix to be solved. Dynamics matrix output is available for most solvers (including GDSW), and for some solution methods.

<table>
<thead>
<tr>
<th>Solution</th>
<th>Matrix</th>
<th>Comment</th>
</tr>
</thead>
<tbody>
<tr>
<td>eigen</td>
<td>$K - \sigma M$</td>
<td>real eigen problem</td>
</tr>
<tr>
<td>transient</td>
<td>$K + \frac{4}{\Delta T^2} M + \frac{2}{\Delta T} C$</td>
<td>standard Newmark-Beta</td>
</tr>
<tr>
<td>statics</td>
<td>N/A</td>
<td>dynamics matrix is stiffness matrix</td>
</tr>
<tr>
<td>qevp</td>
<td>N/A</td>
<td>unimplemented</td>
</tr>
</tbody>
</table>

Table 7-139. – Selected Dynamic Matrix Definitions.

7.1.52. ddamout

Table 7-140 lists the nodal and element variables that are output when the ddamout keyword is selected in the OUTPUTS or HISTORY sections. Element variables will be skipped when writing to the history file.
In Ddam analysis some of this data is also written to text files.

**Table 7-140.** Variables that are output from ddam analysis.

<table>
<thead>
<tr>
<th>Option</th>
<th>data type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DDAM_MDISP</td>
<td>nodal</td>
<td>modal displacements</td>
</tr>
<tr>
<td>DDAM_MVEL</td>
<td>nodal</td>
<td>modal velocities</td>
</tr>
<tr>
<td>DDAM_MACC</td>
<td>nodal</td>
<td>modal accelerations</td>
</tr>
<tr>
<td>DDAM_MFOR</td>
<td>nodal</td>
<td>modal forces</td>
</tr>
<tr>
<td>DDAM_NRL_SDISP</td>
<td>nodal</td>
<td>nrl-summed displacements</td>
</tr>
<tr>
<td>DDAM_NRL_SVEL</td>
<td>nodal</td>
<td>nrl-summed velocities</td>
</tr>
<tr>
<td>DDAM_NRL_SACC</td>
<td>nodal</td>
<td>nrl-summed accelerations</td>
</tr>
<tr>
<td>DDAM_NRL_SFOR</td>
<td>nodal</td>
<td>nrl-summed forces</td>
</tr>
<tr>
<td>DDAM_VStress</td>
<td>element</td>
<td>modal volumetric stresses (tensor values)</td>
</tr>
<tr>
<td>DDAM_NRL_SUM_VStress*</td>
<td>element</td>
<td>nrl-summed volumetric stresses</td>
</tr>
<tr>
<td>DDAM_SStress*</td>
<td>element</td>
<td>modal surface stresses (tensor values)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>at each of 3 shell layers</td>
</tr>
<tr>
<td>DDAM_NRL_SUM_SStress*</td>
<td>element</td>
<td>nrl-summed surface stresses</td>
</tr>
<tr>
<td>DDAM_axialStress</td>
<td>element</td>
<td>modal axial stress (1D only)</td>
</tr>
<tr>
<td>DDAM_NRL_SUM_axialStress</td>
<td>element</td>
<td>nrl-summed axial stress (1D only)</td>
</tr>
<tr>
<td>DDAM_bendingStress*</td>
<td>element</td>
<td>modal bending stress (1D only)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>at each stress recovery point</td>
</tr>
<tr>
<td>DDAM_NRL_SUM_bendingStress*</td>
<td>element</td>
<td>nrl-summed bending stress</td>
</tr>
<tr>
<td>DDAM_shear<em>Stress</em></td>
<td>element</td>
<td>modal shear stress (1D only)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>2 shear directions at each stress recovery point</td>
</tr>
<tr>
<td>DDAM_NRL_SUM_shear<em>Stress</em></td>
<td>element</td>
<td>nrl-summed shear stress</td>
</tr>
<tr>
<td>DDAM_MVMSTR</td>
<td>element</td>
<td>modal von Mises stress</td>
</tr>
<tr>
<td>DDAM_NRL_SVMSTR</td>
<td>element</td>
<td>nrl-summed von Mises stress</td>
</tr>
<tr>
<td>DDAM_HYDROSTATIC</td>
<td>element</td>
<td>modal hydrostatic stress</td>
</tr>
<tr>
<td>DDAM_NRL_SUM_HYDROSTATIC</td>
<td>element</td>
<td>nrl-summed hydrostatic stress</td>
</tr>
<tr>
<td>DDAM_MaxShear</td>
<td>element</td>
<td>modal max shear stress (3D only)</td>
</tr>
<tr>
<td>DDAM_NRL_SUM_DDAM_MaxShear</td>
<td>element</td>
<td>nrl-summed max shear stress</td>
</tr>
</tbody>
</table>

NRL-Summed variables are calculated across modes with the equation:

\[
R_{ia} + \sqrt{\sum_{b=1}^{N} R_{ib}^2 - R_{ia}^2}
\]

Where:

- \( R_{ia} \) is the maximum absolute value at location \( i \) for all modes.
$R_{ib}$ is the value at location $i$ and mode $b$.

$N$ is the number of modes.

Finally, we note a couple of additional details for output of ddam data.

- In parallel runs, the text file output will not include nodal variables, since that data would not be usable in that form. Instead, that data could be written to the Exodus file with the `ddamout` keyword.

- History output of ddam data will only write the nodal variables, not the element variables. Element variable history output for ddam analysis is currently not in place.

### 7.1.53. Temperature

The **Temperature** keyword is a single keyword that can be used to trigger Exodus output of element temperature at the centroid. Element temperature can either be read in as input temperature or calculated from energy deposition. The element centroid temperature can originate from the reference or block temperature, and the centroid, nodal or Gauss point Exodus data.

### 7.1.54. Voltage and Charge

The **voltage** keyword is a single keyword that can be used to trigger Exodus output of the nodal voltages for electro-mechanical coupled models. The **nodal_charge** keyword is a single keyword that can be used to trigger Exodus output of the applied nodal charges.

**Volume** Keyword **Volume** selects Exodus output of element volume in the undeformed state. Volume is the integral of shape functions for solid elements. For shell elements the volume is the area times the thickness of the elements. For bar and beam elements the volume is length times the cross-sectional area of the element.

### 7.2. User Output

#### 7.2.1. Block Statistics

```
User Output
block <list> (block names/ids)

    compute global <string>Name1 as average|avg
    \rightarrow of element <string>Variable1 (weighted by volume)

    compute global <string>Name2 as max|min|maxabs|minabs
    \rightarrow of element <string>Variable1
```
The **user output** block selects the volume-weighted average of any element variable within a user specified list of blocks for output file or history file as a global variable. Multiple blocks are defined in the same way as the subdomain selection in the echo block (Section 7.7.4).

Each user output can then be included by name in the `outputs` or `history` block to turn on that user output in the corresponding output file, e.g.

```plaintext
History
  Name1
  Name2

  nodeset tail
disp
  ...
end
```

The format of the global output will then be `NameVariable`, i.e. for `Name="stress_gauge"`, and `Variable="stress"`, your outputs for a solid element would be of the form `stress_gaugeVStressX, stress_gaugeVStressXY, stress_gaugeVStressXZ, ....`

### 7.2.2. Closest Distance

```plaintext
User Output
  block <list>("A" block names/ids)
 (OR) surface <list>("A" sideset names/ids)
 (OR) nodeset <list>("A" nodeset names/ids)

  compute global <string>name_1 as closest distance
     → to block <list>("B" block names/ids)
     → (search node_face|node_node|mixed)

  compute global <string>name_2 as closest distance
     → to surface <list>("B" sideset names/ids)
     → (search node_face|node_node|mixed)

  compute global <string>name_3 as closest distance
     → to nodeset <list>("B" nodeset names/ids)

  compute nodal <string>name_4 as closest distance
     → to surface <list>("B" sideset names/ids)
```
The global option computes the closest distance between the “A” set blocks, sidesets, nodesets and the “B” set blocks, sidesets, or nodesets as a single global variable. This is the minimum separation distance between these two sets anywhere in the model in the current displaced shape. The optional search command applies to the block/sideset from the "compute global" line, the default (mixed) is node_face. The node face search calculates the distance from the nodes in set “A” to the closest face in set “B”. The node-node search option computes the closest distance from the nodes in set “A” to the nodes in set “B”. If the “B” set contains only nodesets only the node_node search can be used, so any search option there will be ignored (with a warning).

The nodal option computes the closest distance between each node in the “A” set to the faces in the “B” set. This output is a full field result for the closest distance calculation.

As with block statistics output (section 7.2.1), the results of the user output calculation will be output as global or nodal variables, and are enabled by including the variable name in the outputs or history section, e.g.

```
History
  nodeset tail

    Name1
    Name2

  disp
  ...
end
```

### 7.2.3. Derived Variables

Derived user output variables support the same use case as the Statistics file, with more granular control over variable selection, and broader support for variables to be derived. Derived user output variables are intended as a replacement for the Statistics file.

Statistical user output values do not match values in the Statistics file. The cause of this is the inclusion of the model’s initial state in user output calculations, and its exclusion from Statistics file calculations. The OutputInitialTime parameter has no effect on either calculation.
This user output block computes two new derived variables \texttt{name1} and \texttt{name2} by applying an operator over time to the built-in variables \texttt{var1} and \texttt{var2}. These derived outputs require time histories, and are only enabled for transient solution cases.

Supported operators include minimum(\texttt{min}), maximum(\texttt{max}), mean estimate(\texttt{average}), the Root-Mean-Square (\texttt{rms}), and the standard deviation. Note that the standard deviation operator uses Bessel’s correction in its calculations. That is, we define the variance as:

\[
\frac{1}{N-1} \sum_{i=1}^{N} (x_i - \bar{x})^2
\]

not

\[
\frac{1}{N} \sum_{i=1}^{N} (x_i - \bar{x})^2
\]

The source variables \texttt{var1} and \texttt{var2} must be built-in variable request strings; "chaining" derived variables together is not currently supported. Additionally, these variable request strings must match variable requests in other contexts; \texttt{Stress} and \texttt{Displacement} are valid, but \texttt{StressXX} and \texttt{DispX} are not.

Most built-in variable requests represent several variable fields in the results file. Therefore, derived variable fields are defined by prepending the user string \texttt{name1} to the built-in field names \texttt{StressXX}, \texttt{StressYY}, \texttt{StressZZ}, etc. The result in this case would be \texttt{name1StressXX}, \texttt{name1StressYY}, \texttt{name1StressZZ}, etc.

Every built-in element request and most built-in nodal requests are available for use in derived outputs. Some very specialized nodal requests are not supported for derived outputs, due to the unusual nature of their calculations. For example, variables derived from \texttt{AcousticLighthill} will always report 0. This is a known deficiency which can be addressed as needed.

7.3. History

All the results from the “OUTPUT” section can be output to a limited portion of the model using a history file. Only those outputs described in Table 7-124 are supported. If the output is also specified in the OUTPUT section, there is little need to write the data in
the history file. The following output section options are ignored in the history section because all history file output will be in the Exodus format.

- MFile
- Kaa, Maa, Faa
- vrms

In addition to the outputs selection options, the history file section contains information about the regions of output. The default is NO output selection. Selection may be for node sets, side sets, a node list file (see Section 6.1.5), or element blocks. Virtual blocks can be included in this section. For example, one could output the element force in a virtual Joint2g element. If side sets are selected, the side set selection is for the nodes associated with that side set, not for the elements themselves. All nodal variables selected in the history file will be output for all selected nodes. Selecting an element block automatically selects the associated nodes in that block. The format for the selection of multiple regions uses a series of MATLAB concatenated ranges, as with subdomain selection in the echo block (Section 7.7.4). Additionally, the keyword all can be used to select all nodesets, sidesets, or blocks for output. For example,

```plaintext
History
  nodeset tail,1:10,17
  sideset all
  nodeset '8,15' coordinate 4
  block '5,6,nose'
  stress
disp
nskip 10
  element centroid stress nearest
      location 0 1 2 as stress_gauge
  element strain nearest location 1 2 1 as strain_gauge
  element stress at element 1 as element_1_stress_gauge
  element strain nearest center of
      sideset wing as sideset_1_strain_gauge
  node acceleration at node 7 as node_7_accel_gauge
  node displacement nearest location 2 2 10 as disp_gauge
end
```

Any number of nodeset, block, and/or sideset selections can be specified in the history section. Nodeset and sideset specifications may be followed by an optional coordinate entry. If a coordinate is specified (see section 2.7), all nodal results for the nodes in the nodeset/sideset are transformed to the specified coordinate system before output to the file. If a particular node is identified in more than one specification, the last specification is used for the output. Note that the coordinate keyword for history section output will only work with nodesets and sidesets: it is not supported for blocks or node list files.
The coordinate *name* of nodes in the history file may be printed out in the echo file by specifying *nodes* in the echo section of the input. The coordinate *ID* will also be written to the history file (as a nodal variable *CID*) provided any nonzero coordinate frames have been specified. Note that if the coordinate name is not convertible to an integer id (i.e. if it is a string), the *CID* output will be -1.

Note that for output specifications, the corresponding nodeset, block, and sideset ids/names should be specified as shown in the above example, with either a list of ids/names, or the keyword “all”. The rules for specifying these lists are the same as for integer lists, and are detailed in Section 2.1.

In transient dynamics solutions, user control of output step interval “nskip” and output buffer “flush” operations are provided to increase efficiency of output. See Section 3.30 for examples. The history file respects the *nskip* and *flush* parameters set in the solution block, but additional user control is provided for history files by inserting the *nskip* and *flush* parameters in the history block. In that case, history files for all multicase solutions will have output and buffer flushing at the intervals specified in the history section, and the entries in the solution section will be ignored for history files.

Unlike subdomains, node set and side set IDs need not be contiguous in the Exodus file. If the selection criteria is a range, it may identify nonexistent sets. These will be silently ignored. In example 7.3 above in which nodesets *tail*, 1:10 and 17 are selected, the Exodus mesh must have nodeset number 10. Nodeset, sideset, and block names/IDs in the history file must be consistent with the corresponding Exodus input file.

As with output in general (section 7.1.2), users can define a custom history output file name via the database name keyword. Additionally, Exodus properties can be defined as shown in Section 7.1.3.

Only one history file will be written per analysis. The name of the history file is derived from the name of the Exodus output file, except that the extension is “.h”.

While the history file provides a convenient means for transforming coordinates, its applicability may be somewhat limited when output in many coordinate frames is desired. In particular, only a single history file is written in each analysis, and only one coordinate frame may be output per node. The history file will display variables as Cartesian regardless of coordinate choice. Table 2-23 shows the corresponding values for cylindrical and spherical coordinates.

### 7.3.1. Global History Output Near a Location

The history data of a mesh entity near a location can be written to the output file as a global variable using any of the following syntax:
Outputs at multiple locations can be requested from a history block. Each output location needs a unique `history_variable_name`. This capability is currently only supported for outputting data at the closest location for 3D elements or the closest centroid for 2D and 3D elements (using the optional `CENTROID` flag). If the closest location to the centroid of a specified sideset is requested, the data is obtained at the nearest location on a 3D element `surface` that touches the sideset. If the data at a specified element is requested, the data is obtained from that element centroid for 2D and 3D elements. Trying to specify a 1D element is not supported. If a 1D element is specified directly, or if no elements can be found that satisfy the search conditions, the .rslt output summary will contain a statement that no element was found.

**Note:** User-specified element, nodal, and sideset IDs should be base-1 to match the Exodus convention.

All element output cases are currently only supported for stress, strain, and von Mises stress output. Likewise, displacement, velocity, acceleration, and force outputs are the currently supported nodal outputs. A similar capability and syntax is used in Sierra/SM.

A summary of the global history data output will be written to the .rslt file in the following format:

```
------------------------------- GLOBAL HISTORY OUTPUT -----------------------------
<table>
<thead>
<tr>
<th>Prefix</th>
<th>Block</th>
<th>Entity, GId(: Type)</th>
<th>Variable</th>
<th>Near</th>
<th>At</th>
</tr>
</thead>
<tbody>
<tr>
<td>stress_gauge</td>
<td>1</td>
<td>element 1: Hex8b</td>
<td>stress</td>
<td>(0,1,2)</td>
<td>(0.5,0.5,0.5)</td>
</tr>
<tr>
<td>strain_gauge</td>
<td>1</td>
<td>element 5: Hex8b</td>
<td>strain</td>
<td>(1,2,1)</td>
<td>(1,2,1)</td>
</tr>
<tr>
<td>stress_gauge</td>
<td>1</td>
<td>element 1: Hex8b</td>
<td>stress</td>
<td>N/A</td>
<td>(2.5,1.5,0.5)</td>
</tr>
<tr>
<td>strain_gauge</td>
<td>1</td>
<td>element 6: Hex8b</td>
<td>strain</td>
<td>(2,3,1)</td>
<td>(2,2,1)</td>
</tr>
<tr>
<td>accel_gauge</td>
<td>N/A</td>
<td>node 7</td>
<td>accel</td>
<td>N/A</td>
<td>(1,1,0)</td>
</tr>
<tr>
<td>disp_gauge</td>
<td>N/A</td>
<td>node 38</td>
<td>disp</td>
<td>(2,2,10)</td>
<td>(2,2,1)</td>
</tr>
</tbody>
</table>
-------------------------------
```
In the **Exodus** history file output, the global variable’s prefix or `history_variable_name` will be prepended onto the element variables output name (see sections 7.1.18 and 7.1.16 for stress and strain variable names). For example, if you had two requested outputs near a shell centroid and a solid, with prefixed names “shell_strain” and “solid_stress” respectively, the global history output would produce the following global variables in the history file:

```
shell_strainSStrainX1, shell_strainSStrainY1, shell_strainSStrainXY1,
shell_strainSStrainX2, shell_strainSStrainY2, shell_strainSStrainXY2,
shell_strainSStrainX3, shell_strainSStrainY3, shell_strainSStrainXY3,
solid_stressVStressX, solid_stressVStressY, solid_stressVStressZ,
solid_stressVStressXY, solid_stressVStressXZ, solid_stressVStressYZ
```

Variables requested for global history output will be output on every other element block in the history block.

### 7.4. Frequency

The frequency section provides information for data output from the ModalFrf, directFRF, shock, modalshock, and random vibration solution methods. One frequency file is written per analysis. The name of the frequency file is derived from the name of the **Exodus** output file, except that the extension is “.frq”. The section format follows that of the history section. As in the case of the history section, data can be written to a nodeset, noderset, node_list_file, or a block. In the case of output to a block, the block can be a virtual block. Thus, one could output element force on a Joint2g element. Solution methods that do not write frequency domain output silently ignore the Frequency section.

The frequency section also includes the definitions of the frequency values for calculation. A frequency section (with some output selection region) must be selected for any solution method requiring frequency output. To fail to do so is an error, since the solution would be computed and no output provided.

As with output in general (section 7.1.2), users can define a custom frequency output file name via the `database name` keyword. Additionally, **Exodus** properties can be defined as shown in Section 7.1.3.

The frequency values may be specified using the methods specified in Table 7-141. The methods are mutually exclusive, i.e. do not mix keywords from the “linear” method with those of the “table” method. An example follows.

```
FREQUENCY
   nodeset '1:10,17'
sideset '3:88'
block 5,6,3
disp
```
Table 7-141. – Frequency Value Specification Methods.

<table>
<thead>
<tr>
<th>Method</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>method=linear</td>
<td>freq_min</td>
<td>minimum frequency (typically in Hz)</td>
</tr>
<tr>
<td></td>
<td>freq_max</td>
<td>stop frequency</td>
</tr>
<tr>
<td></td>
<td>NF</td>
<td>number of frequency intervals.</td>
</tr>
<tr>
<td></td>
<td>freq_step</td>
<td>frequency increment (or use NF)</td>
</tr>
<tr>
<td>method=log</td>
<td>freq_min</td>
<td>minimum frequency (typically in Hz)</td>
</tr>
<tr>
<td></td>
<td>freq_max</td>
<td>stop frequency</td>
</tr>
<tr>
<td></td>
<td>NF</td>
<td>number of frequency intervals.</td>
</tr>
<tr>
<td>method=table</td>
<td>table</td>
<td>name of a 1D table (see Section 2.8.19)</td>
</tr>
</tbody>
</table>

acceleration
freq_min=10 // starting frequency in HZ
freq_step=10 // frequency increment
freq_max=2000 // stop freq. Outputting 201 freq points.
END

For the “linear” and “log” methods, the frequencies are obtained by the following equation.

\[
F_k = \begin{cases} 
F_{min} + k \cdot F_{step} & \text{for method=linear} \\
F_{min} \exp(kD) & \text{for method=log}
\end{cases}
\]

Here \( D = 1/N_F \log\left(\frac{F_{max}}{F_{min}}\right) \). If both \( \text{freq}_\text{step} \) and \( \text{NF} \) are specified, \( \text{NF} \) is used.

Output Region:

The controls in the frequency section also affect data written to the results (or echo) file. In particular, the echo file contains data only for those nodes in the selection region of the frequency section. Selection of a specific output (such as displacement or acceleration) is independent. For example, you may echo only displacements, but write displacements and accelerations to the Exodus frequency output file. The history section (7.3) has more information on specification of the output region.

The Sierra translator exo2mat may be used to translate the output into MATLAB format for further manipulation and plotting.

7.5. Linesample

The line sample (LineSample) section of the input file provides a means of evaluating and outputting fields or internal variables at sampling points within a structure. These sampling points are defined on a series of lines.
Section 3.31 discusses the primary application of line sample, verification of stress field input to Sierra/SD from TSR. Line sample is used for energy deposition (see Two Element Exponential Decay Variation Hex20 in the Verification manual). Energy deposition is interchangeable with supplying an applied temperature. Line sample is also used for far-field processing in acoustics problems (see 6.1.9.1 or How To), for example with infinite elements.

Keywords for the line sample input are listed in the table below. An example follows.

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Arguments</th>
</tr>
</thead>
<tbody>
<tr>
<td>samples per line</td>
<td>integer</td>
</tr>
<tr>
<td>endpoint</td>
<td>6 real numbers</td>
</tr>
<tr>
<td>format</td>
<td>string</td>
</tr>
<tr>
<td>nskip</td>
<td>integer</td>
</tr>
</tbody>
</table>

**samples per line**  The number of sample points on each line. All lines will have the same number of samples.

**endpoint**  The endpoints of the line. There should be 3 real numbers for the XYZ location of the beginning of the line, followed by 3 real numbers at the end. There can be any number of endpoint entries.

**format**  The format of the output file. Two output formats are supported: Exodus and MATLAB MFile. The default is MFile.

**nskip**  Results output frequency; defaults to the value specified in the solution section; see Section 3.30.0.4.

Output from the line sample is written to linedata.m for MFile output, or to linedata.exo for Exodus format. There is no need to join this data for parallel runs. In those output files, a nodal variable called Displacement will be created. The entries in this array correspond to 3 displacement variables, 3 rotation variables, acoustic pressure, voltage, and temperature. For transient data, the time values are also output for each of these arrays.

```
LineSample
  samples per line 5
  endpoint 0. 0. 0. 1. 1. 1.
  endpoint 0.0 0.5 0.5 1. 0.5 0.5
  format exodus
END
```

### 7.6. Stresses and Strains

Stress and strain values at element centroids are available. Solid and beam element stress is evaluated in the global or basic frame. However, shell element stress and strain are evaluated in element space.
The total strain (7.1), elastic strain (7.2), and thermal strain (7.3) can all be requested separately; see Section 7.1.16. The stress reported is always the Cauchy stress $\sigma^e = C : \epsilon^e$; see Section 7.1.18.

7.6.1. Stress/Strain Truth Table

The Exodus data format provides an element truth table. Element variables are defined globally (for all element blocks), but output data is stored only for those blocks that have entries in the truth table. Thus, in Sierra/SD if stress output is requested (see Section 7.1.18), then stress variables are defined for solids and shells. Space is allocated in the output Exodus file, and data is written only if it is applicable. Table 7-142 illustrates this for stresses. A similar table can be generated for strains. Note that volume stresses always start with “V” and surface stresses start with “S”. Note that “vonMises” is the only entry that applies to both solids and shells.

\footnote{The variables are defined for solids and shells even if only one or the other occurs in the model}
Table 7-142. – Element Stress Truth Table.

<table>
<thead>
<tr>
<th>Variable Name</th>
<th>Solid</th>
<th>Shell</th>
<th>Beam</th>
</tr>
</thead>
<tbody>
<tr>
<td>SStressX1</td>
<td>$\sigma_{xx}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>SStressY1</td>
<td>$\sigma_{yy}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>SStressXY1</td>
<td>$\tau_{xy}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>SvonMises1</td>
<td>$\sigma_{vm}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>SStressX2</td>
<td>$\sigma_{xx}^{mid}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>SStressY2</td>
<td>$\sigma_{yy}^{mid}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>SStressXY2</td>
<td>$\tau_{xy}^{mid}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>SvonMises2</td>
<td>$\sigma_{vm}^{mid}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>SStressX3</td>
<td>$\sigma_{xx}^{bottom}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>SStressY3</td>
<td>$\sigma_{yy}^{bottom}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>SStressXY3</td>
<td>$\tau_{xy}^{bottom}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>SvonMises3</td>
<td>$\sigma_{vm}^{bottom}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>VStressX</td>
<td>$\sigma_{xx}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>VStressY</td>
<td>$\sigma_{yy}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>VStressZ</td>
<td>$\sigma_{zz}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>VStressYZ</td>
<td>$\sigma_{yz}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>VStressXZ</td>
<td>$\sigma_{xz}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>VStressXY</td>
<td>$\sigma_{xy}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>VonMises</td>
<td>$\sigma_{vm}$</td>
<td>$\max(\sigma_{vm})$</td>
<td></td>
</tr>
<tr>
<td>Signed_VonMises</td>
<td>$\pm\sigma_{vm}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Max_Principal_Stress</td>
<td>$</td>
<td></td>
<td>\sigma_1</td>
</tr>
<tr>
<td>Intermediate_Principal_Stress</td>
<td>$</td>
<td></td>
<td>\sigma_2</td>
</tr>
<tr>
<td>Min_Principal_Stress</td>
<td>$</td>
<td></td>
<td>\sigma_3</td>
</tr>
<tr>
<td>Max_Principal_Stress_x</td>
<td>$\sigma_{1x}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Max_Principal_Stress_y</td>
<td>$\sigma_{1y}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Max_Principal_Stress_z</td>
<td>$\sigma_{1z}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Intermediate_Principal_Stress_x</td>
<td>$\sigma_{2x}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Intermediate_Principal_Stress_y</td>
<td>$\sigma_{2y}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Intermediate_Principal_Stress_z</td>
<td>$\sigma_{2z}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Min_Principal_Stress_x</td>
<td>$\sigma_{3x}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Min_Principal_Stress_y</td>
<td>$\sigma_{3y}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Min_Principal_Stress_z</td>
<td>$\sigma_{3z}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>ElemForce</td>
<td>forces</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
7.6.2. Solid Elements

If stresses are requested, solid elements will output the values of stress at the element centroid. The values reported are the engineering stresses in the global coordinate system.

If principal stresses are requested, solid elements will output the three principal stress vectors and magnitudes in order of descending signed value. The vectors are of unit length, defined in the global coordinate system.

7.6.3. Shell Elements

Shell elements introduce two complexities to stress/strain recovery. First, it is often important to recover data from the virtual surfaces of the elements (where the stresses are highest). This requires data recovery at the top, mid-plane and bottom surfaces. Second, there are no stresses or strains normal to the surface. Thus, stresses are naturally reported in the surface of the element. This can also introduce confusion about the in-plane coordinate frames. As shown in Figure 7-62, the stresses and strains are recovered in the physical space \( x_1, x_2 \) coordinate frame, which has been mapped from the \( \eta_1, \eta_2 \) frame in element space. Note that the direction of the \( x_1 \) vector depends on the ordering of the mesh, and may vary from element to element in the same surface mesh. Element orientation vectors are selected with the \textbf{eorient} keyword described in Section 7.1.43. von Mises stress, an invariant, is independent of the element orientation.

Stress recovery for the \textit{TriaShell} is interesting. A \textit{TriaShell} is a shell element created by combining Allman’s triangle\(^2\) with a DKT element.\(^8\) Its stress vector \( \vec{\sigma} = (\sigma_x, \sigma_y, \sigma_{xy})^T \) is the sum of stresses \( \vec{\sigma}_{at} \) for Allman’s triangle and \( \vec{\sigma}_{dkt} \) for the DKT element,

\[
\vec{\sigma} = \vec{\sigma}_{at} + \vec{\sigma}_{dkt}. \tag{7.8}
\]

Allman’s triangle represents the membrane dof, i.e., \((u, v, \theta_z)\). If the element lies in the \( x-y \) plane, then \( \beta_x \) and \( \beta_y \) are rotations of the normal to the undeformed middle surface in the \( x-z \) and \( y-z \) planes, respectively.

\[
\{\kappa\} = \begin{bmatrix} \beta_{x,x} \\ \beta_{y,y} \\ \beta_{x,y} + \beta_{y,x} \end{bmatrix} \tag{7.9}
\]

\(\{\epsilon\}\) is the strain vector, and \([D]\) is the elasticity matrix for Allman’s triangle. The stresses through the three surfaces of the shell element are the same. Therefore,

\[
\vec{\sigma}_{at} = [D]\{\epsilon\}. \tag{7.10}
\]

For the DKT element, \( z \) is the coordinate direction normal to the element, with \( z = 0 \) representing the mid-plane. \([D]\) is the elasticity matrix.

\[
\vec{\sigma}_{dkt} = z[D]\{\kappa\} \tag{7.11}
\]
\( \bar{\sigma}_{\text{dkt}} \) does vary with the thickness of the element. Note, the above stress equations are written with respect to a local element coordinate system as shown in Figure 7-62.

Combining the stress vectors from Allman’s triangle and the DKT element above yields the stress vector for the element which is output in the local element frame.

For composite elements (such as QuadT, Quad8T and Tria6), the stresses are computed from the underlying Tria3 element and then transformed to the element orientation of the composite element. For the quad elements, the stress of the two central triangles is averaged. Figures 5-26, 5-27 and 5-28 describe these composite elements.

7.6.4. Beam Elements

Reporting stresses for line type elements (Beams, Rods, Springs, etc) is even more problematic than it is for shells. For many of these elements an axial stress could be reported. But, for beam elements that stress could not include the effects of beam bending unless details of the beam cross section were available. For some elements (such as a spring) no concept of stress is even correct. As a consequence, we do not report stresses for most line type elements. However, some recovery may be obtained using the element force output (see Section 7.1.38).

For Beam2, Nbeam, and TiBeam elements (sections 5.9 to 5.11), axial stress will be reported if stress is requested in the outputs section. Additionally, stress recovery points may be requested using the stress recovery point keyword. For each stress recovery point, the bending and 2 shear stresses will be reported (see input 7.2). Each stress recovery point represents a point on the beam cross-section and is defined by a pair of offsets in physical coordinates from the neutral axis of the beam. The von Mises stress at
each stress recovery point is reported, as well as the maximum von Mises stress over all the
stress recovery points. If no stress recovery points are requested, von Mises is taken to be
absolute value of the axial stress.

```
BLOCK beam_block
  material=1
  beam2
    Area=8
    I1=2
    I2=10
    J=7
    stress recovery point  0.5  0.5
                          -0.5 -0.5
END
```

Input 7.2. Beam Stress Recovery Points Example

7.7. Echo

Results, in ASCII format, from the various intermediate calculations may be output to a
results file, e.g. example.rslt, where the file name is generated by taking the base name of
the text input file (without the extension) and adding the extension rslt. Output to the
results file is selected in the Sierra/SD input file using the echo section. An example is
given below, and the interpretation of these keywords is shown in Table 7-143.

```
echo
  materials
  elements
  Jacobian
  mesh
  input on
  nodes
  MPC
end
```

We remark that for virtual blocks, element variables such as element force are also written
to the results file. Since only Joint2g elements are currently supported as virtual blocks,
the only element variable that can be written at this time is the element force, eforce.
<table>
<thead>
<tr>
<th>Option</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ADiag</td>
<td>diagonal of dynamics matrix</td>
</tr>
<tr>
<td>acceleration</td>
<td>nodal accelerations (better in output section)</td>
</tr>
<tr>
<td>block</td>
<td>block wise mass properties (used only following mass)</td>
</tr>
<tr>
<td>debug</td>
<td>debug output</td>
</tr>
<tr>
<td>displacement</td>
<td>nodal displacements (better in output section)</td>
</tr>
<tr>
<td>EForce</td>
<td>element force for beams</td>
</tr>
<tr>
<td>ElemEigChecks</td>
<td>element eigenvalues</td>
</tr>
<tr>
<td>elements</td>
<td>element block info, i.e. what material, element type, etc</td>
</tr>
<tr>
<td>Elmat</td>
<td>element material properties</td>
</tr>
<tr>
<td>energy</td>
<td>element strain energy and strain energy density</td>
</tr>
<tr>
<td>eorient</td>
<td>element orientation</td>
</tr>
<tr>
<td>fatigue</td>
<td>fatigue related parameters</td>
</tr>
<tr>
<td>force</td>
<td>applied forces (better in output section)</td>
</tr>
<tr>
<td>GEnergies</td>
<td>global kinetic and strain energy sums</td>
</tr>
<tr>
<td>block_energies</td>
<td>block kinetic and strain energies</td>
</tr>
<tr>
<td>input (&lt;bool&gt;)</td>
<td>echo of post-Aprepro input <em>(for parse errors)</em> – default = input on</td>
</tr>
<tr>
<td>input_summary</td>
<td>summaries of many sections</td>
</tr>
<tr>
<td>Jacobian</td>
<td>block summary of Jacobians</td>
</tr>
<tr>
<td>KDiag</td>
<td>diagonal of stiffness matrix</td>
</tr>
<tr>
<td>line_weld</td>
<td>Line-weld-specific output variables</td>
</tr>
<tr>
<td>mass</td>
<td>mass properties in the basic coordinate system</td>
</tr>
<tr>
<td>materials</td>
<td>material property info, e.g. E, G</td>
</tr>
<tr>
<td>memusage</td>
<td>prints per processor per task memory use</td>
</tr>
<tr>
<td>mesh</td>
<td>summary of data from the input Exodus file</td>
</tr>
<tr>
<td>mesh_error</td>
<td>mesh discretization error metrics</td>
</tr>
<tr>
<td>ModalVars</td>
<td>modal force and amplitude for modal solutions (echo section)</td>
</tr>
<tr>
<td>MPC</td>
<td>MPC equations</td>
</tr>
<tr>
<td>NLresiduals</td>
<td>turns on residual output per iteration of the Newton loop for nonlinear solution methods</td>
</tr>
<tr>
<td>nodes</td>
<td>nodal summary</td>
</tr>
<tr>
<td>residuals</td>
<td>residual vectors</td>
</tr>
<tr>
<td>rhs</td>
<td>Right Hand Side vector (better in output section)</td>
</tr>
<tr>
<td>strain</td>
<td>element strains at centroids</td>
</tr>
<tr>
<td>stress</td>
<td>element stresses at centroids</td>
</tr>
<tr>
<td>subdomains “0:3:6,10”</td>
<td>Controls which processor will output results file</td>
</tr>
<tr>
<td>threading_summary</td>
<td>threading summary table</td>
</tr>
<tr>
<td>timing_summary</td>
<td>timing and threading summary tables</td>
</tr>
<tr>
<td>velocity</td>
<td>nodal velocities (better in output section)</td>
</tr>
<tr>
<td>vonMises</td>
<td>von Mises stress only</td>
</tr>
<tr>
<td>vrms</td>
<td>RMS quantities (random vibration only)</td>
</tr>
</tbody>
</table>
7.7.1. **Mass Properties**

The mass properties may only be reported in the Echo section (i.e. at this time there is no mass property report in the outputs section). The mass properties are the total mass, the center of gravity and the moments of inertia of the system. All are reported in the basic coordinate system. Note that moments are about the origins, not about the center of gravity. Masses are reported in a unit system consistent with the input, with or without the wtmass parameter (see Section 2.3).

An additional option of block may be used in the echo section to output the block wise mass properties to the results file. Please note that the block wise mass properties, though summed for all processors (if running on a parallel machine), are only output to the result file from the first processor (processor 0). The block wise mass properties option, called block, reports the number of blocks, the mass of each block, and the center of gravity of each block along the x, y, and z axis. Please note that block may only be used in the echo section following the mass option as shown below.

```
echo
  materials
  elements
  mass=block
  nodes
end
```

If the keyword mass does not directly precede block in the echo section, then Sierra/SD will abort with the following error.

```
Unrecognized "echo" option "block".
Aborting.
```

Requesting both massblock and mass=block causes output to the result file of only the global mass properties. If only block-level mass properties are desired, then specifying mass=block will suffice, as follows

```
echo
  mass=block
end
```

7.7.2. **Multipoint constraints**

Text descriptions of the MPC equations are output to the result file using MPC. This is a check on the input deck. An example of the output format is as follows
In this case, the MPC equation is constraining the acoustic pressure in nodes 25 and 106 to be equal in the global (default) coordinate system.

### 7.7.3. ModalVars

The text file `modalvars` contains modal forces and modal amplitudes for modal based superposition solutions including “modaltransient” and “ModalFrf”. Two text files are written: `Qdisp.txt` and `Qforce.txt`. Each line of the file contains data for a solution increment (a time or frequency step). For transient solutions, each column corresponds to a mode in the solution. Because FRF solutions are complex, two adjacent columns describe the complex modal amplitude (or force) for a mode. In terms of the physical force at time $t_n$, $F(t_n)$, and the $i^{th}$ eigenvector $\phi_i$, the modal force and displacements are

$$f_{qi}(t_n) = \phi_i^T F(t_n), \quad u(t_n) = \sum_i \phi_i q_i(t_n).$$

The expressions in the frequency domain are similar.

The text files are readable by either MATLAB or MS/excel.

### 7.7.4. Subdomains

In parallel calculations, one results file is written per subdomain. Only data associated with that subdomain are written to the file. By default, results are only written to subdomain 0 (the root processor), and in the results for subdomain 0 will always be output. Use the “subdomains” option to specify additional subdomains for which data will be written. The `subdomains` specification is made using a MATLAB like string, as detailed in Section 2.1. For example,

```
subdomains '0:2:8'
```

selects subdomains 0, 2, 4, 6 and 8 (again, subdomain 0 will always be output, and so it is redundant here). The following selects subdomains 0, 2, 3, 4, 6, 8, 9 and 15.

```
subdomains 0:2:8,3:3:9,15
```

In addition, the keyword “all” selects all subdomains.
7.7.5. Memusage

The **Memusage** keyword selects memory usage information output for the results file. In addition, it also requests that a per processor, per task break down be written to a separate text file. The memory in the text file is shown in Mbytes. The output for the text file will be formatted as follows:

```
  1  2  3  4
40 41 41 42 "initialization"
23 12 15 15 "assembling matrices"
45 56 65 54 "initializing solver"
10 25 12 52 "writing output"
```

The primary use of the memory usage printing is to diagnose where or why memory is being exhausted by an analysis.

**Elmat** The bulk modulus, mass density and shear modulus of each element are reported if the **Elmat** keyword is added to the echo section.
8. CONTACT

8.1. Tied Surfaces

Tied surfaces provide a mechanism to connect surfaces in a mesh that will always be in contact. Because the surfaces are always tied, the constraints may be represented by a set of linear multipoint constraints. Tied surfaces are also known in the literature as glued surfaces or as tied contact. They are used almost exclusively to combine two surfaces of a mesh that have not been meshed consistently.

There are a number of ways of combining surfaces that have not been consistently meshed. The simplest method constrains the nodes of one surface (node-surface) to lie on the faces of another surface (face-surface). In this method, the constraint is called inconsistent because the mesh does not ensure that linear stress will be maintained across the boundary. The stress and strain in the region of the constraint will be wrong. However, loads are properly transferred across the boundaries, so a few element diameters away from the boundary, the stresses and strains should be approximately correct.

Tied surfaces can currently be specified for structural-structural interfaces, acoustic-acoustic interfaces, and structural-acoustic interfaces (i.e. wet interfaces). The syntax in the tied data section is the same. For structural-structural interfaces, the nodal displacements on the node-surface are constrained to lie on the faces of the face-surface. In the last case, the nodal acoustic pressures on the node-surface are constrained to match the interpolated value of pressure on the face-surface.

For tied structural-acoustic interfaces, it is necessary to ensure a weak continuity of both stress and displacement (velocity) across the wet interface. Also, we recommend that the acoustic surface be defined as the face-surface (and hence should have its sideset number listed first in the input deck). Defining the structural surface as the face-surface sometimes causes an error related to singular subdomain matrices.

A simulation may have multiple tied surfaces as long as certain requirements are met. For example acoustic-acoustic and tied structural-acoustic data blocks in the same input file are supported. However, it is necessary that each sideset be exclusively attached to either structural elements or acoustic elements. A sideset containing both acoustic and structural elements is not supported. This does not restrict the possible types of analysis. It can increase the number of tied data blocks. However, this extra input will reduce confusion and likely also reduce potential modeling errors.

The keyword transverse controls the constraints on transverse displacements. Transverse displacements can be tied or slip. Its default is tied. The tied option is the standard inconsistent tied surface approach. The slip option only constrains normal degrees of freedom between the surfaces. In this option, the tangential degrees of freedom are free to slide. This would be the case if there was no friction between the surfaces. The friction option for specifying a simple friction model is not currently supported.
In a tied data block, if the tied contact is inconsistent, i.e., the method is not mortar or other options, then by default the gap is removed. Gap removal only works with tied data. In the tied data section, the order of the surface ids is important. The first surface id becomes the face surface, and the second becomes the node-surface. Gap removal moves node-surface nodes to the face surface. Set gap removal to off to skip gap removal. ACME’s gap and push back vector quantities provide the gap. Updated coordinates instead of the original coordinates appear in the output Exodus file, and are also used by the system matrices.

Debugging tied contact is easier using the gap removal solution case 3.35.

8.1.1. Contact Normal Vectors

For all the contact type interactions, including tied surfaces, tied joints, and contact definition the algorithms used restrict the search to matching faces that have opposing normal vectors. For solids, this is seldom an issue. The normal vectors for a solid are always outward from the solid, so two interacting solids (unless they occupy the same volume), will naturally have opposing normal vectors. However, the situation for shell-shell or shell-solid interactions can be more complicated.

Sidesets may be created from the top or bottom surfaces of the shells. Thus, the shell surface has a natural normal direction determined by its connectivity, and the sidesets generated from the shells have a direction too. The sideset direction may align or oppose the direction normal of the shell itself. If the shell normal does not oppose the normal of the mating surface, no interactions will be found, and the surfaces cannot be tied. See Figure 8-63.
Normal Opposed. Interactions are Possible

Normals are aligned. No Interactions are Possible

Normal Opposed. Interactions are Possible

Figure 8-63. – Shell Normal in Contact or Tied Interactions.
8.1.2. Mortar Methods

Mortar methods may also be used to tie the surfaces. This is currently under development, but some capability is available. Large tied surfaces using the mortar methods may have many fully coupled constraints which can overwhelm most parallel solvers. The cost in computing the mortar contribution is higher than the inconsistent method, but the solution will typically be much better in the region of the constraint.

Two different mortar methods are available. Both constrain surfaces together in an integral (or weak) sense. Standard mortar methods are somewhat simpler, but can result in a constrained system which fully couples all the nodes of both surfaces together in a single constraint. Dual mortar methods are much more friendly to the linear solver, as the constraint system decouples the constraints similarly to what is seen in node-face contact. The dual mortar method is the default.

Mortar methods are specified by adding mortar to the tied data block. To select the type of method, standard or dual, in the parameters block specify MortarMethod=standard or MortarMethod=dual respectively. 3

8.1.3. Node to Face

Tied surfaces are specified by a listing of face-surface and node-surface side sets. Any number of tied data blocks may be specified in the input. Each tied data block represents a single logical pairing of constraint side sets.

TIED DATA
  tied faces 12
  tied nodes 18
  name "tying_12-18"
  transverse slip
  search tolerance = 1e-7
  edge tolerance = 1e-8
  gap removal = on
END

In the example above, sideset 12 is the face-surface. Side set 18 is the node-surface. Each node in the node-surface may be tied to the nearest face in the face-surface by a constraint equation. The transverse degrees of freedom are allowed to slip in this example. If the transverse keyword were omitted, standard tied surfaces would be used.

Tied surfaces use a node-to-face search algorithm. In this algorithm, the “search tolerance” represents the normal distance from a node on one surface to a corresponding face on the other. Thus, the search tolerance will typically be small and represents the amount the two

---

3There is no means of applying standard mortar methods to some interactions, and dual mortar methods elsewhere.
For node-to-node searches the *search tolerance*, must be large enough to capture nearby nodes. For node-to-face searches (as used in tied surfaces), it should only capture the nearby surface.

surfaces may not be coincident. This is in contrast to a node-to-node search, where the “search tolerance” represents a search radius. See Figure 8-64.

Special care should be used when using the “edge tolerance”. If this tolerance is too large, non-intuitive interactions can be created.

**Note:** The current implementation ties a face-surface that consists of the two-dimensional faces of shell or solid elements. It is not possible to tie a node to the one dimensional edges of shell elements.

The relevant parameters for tied surfaces are shown in Table 8-144.

Smoothing parameters may be needed to control smoothing of the normal. Figure 8-65 illustrates the normal definitions on a faceted surface. The discontinuity in normal vectors can be an important consideration on curved surfaces where faceting affects tangential sliding. Smoothing parameters are illustrated in Figure 8-66 and include the following :

- **Smooth angle** If an angle between two faces exceeds this value (in degrees), then the angle is considered to be “sharp”, and no smoothing is done. Default is 30°.

- **Smoothing resolution** The resolution method can be either node based, or edge based. This may be needed to control smoothing on edges that include both a sharp and a non-sharp edge. Default=node.

**Figure 8-65.** – Normal Definitions on Faceted Geometry. When low order elements are used to describe a curved boundary, the normal is poorly defined at the edge of the facets.
Table 8-144. – Tied Surface Parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>type</th>
<th>description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Name</td>
<td>String</td>
<td>name of tied data block, useful for diagnosing error messages, defaults to tied</td>
</tr>
<tr>
<td>Surface</td>
<td>integer pair</td>
<td>face-surface and node-surface sidesets separated by comma or space</td>
</tr>
<tr>
<td>Tied Faces</td>
<td>&lt;sideset&gt;</td>
<td>face-surface sideset</td>
</tr>
<tr>
<td>Tied Nodes</td>
<td>&lt;sideset&gt;</td>
<td>node-surface sideset</td>
</tr>
<tr>
<td>Search Tolerance</td>
<td>Real</td>
<td>face normal of search tolerance defaults to 1e-8</td>
</tr>
<tr>
<td>Edge Tolerance</td>
<td>Real</td>
<td>search tolerance beyond an edge facet defaults to 1/10 search tolerance.</td>
</tr>
<tr>
<td>Method</td>
<td>String</td>
<td>inconsistent (default most solvers) mortar</td>
</tr>
<tr>
<td>Transverse</td>
<td>String</td>
<td>tied (default) slip (transverse displacements can slip)</td>
</tr>
<tr>
<td>Gap Removal</td>
<td>String</td>
<td>Yes (default: for inconsistent only ) No</td>
</tr>
<tr>
<td>smooth angle</td>
<td>Real</td>
<td>maximum angle for smoothing (def=30)</td>
</tr>
<tr>
<td>smoothing resolution</td>
<td>String</td>
<td>“node” or “edge” based</td>
</tr>
</tbody>
</table>

Figure 8-66. – Smoothing Parameters for Surface Normal Vectors. No smoothing occurs for faces that are misaligned by more than the specified “smooth angle”. Within the “smooth distance”, normal vectors vary linearly with relative distance from the node.
8.2. Contact Definition

The contact definition block provides flexible syntax to define tied MPCs. The contact definition provided a similar capability to Tied Data (see Section 8.1) but with a more powerful and flexible ways to define contact surfaces. The contact definition capability leverages the same Dash contact algorithms used in Sierra SM. Both the syntax and capability of the Dash contact definition is compatible with SM recommending Dash contact for both SM/SD hand-off analyses and analyses with a lot of general contact.

A brief description of contact definition commands is given here. More detailed descriptions of how contact surface and interaction definition commands function can be found in the Sierra Solid Mechanics User Manual.58

The full set of available contact commands are as follows:

```
Begin Contact Definition <name>
  contact [surface|nodeset] <surf_name> contains <strings>
  skin all blocks = on [exclude <names>]
  gap removal = off|on(on)

Begin Interaction Defaults
  general contact = off|on(off)
  self contact = off|on(off)
  normal tolerance = <Real>
  tangential tolerance = <Real>
  constraint formulation = Node_Face|Face_Face(Node_Face)
  friction model = tied|frictionless(tied)
End Interaction Defaults

Begin Interaction
  normal tolerance = <Real>
  tangential tolerance = <Real>
  side a = <string_list>
  side b = <string_list>
  surfaces = <string>
  interaction behavior = No_Interaction
  constraint formulation = Node_Face|Face_Face
  friction model = tied|frictionless
End Interaction

Begin Dash Options
  separate disconnected mesh components = true|false(false)
  ignore_shells = true|false(false)
  cutoff variable <nodal_variable> >|< <Real>
End Dash Options
End Contact Definition
```
Information is presented in the format command = option (default) format. For example gap removal is on by default. Debugging contact is easier using the gap removal solution case 3.35. Gap removal is another name for initial overlap removal.

For a surface in the begin interaction section to define interacting surfaces, it must first be assigned a local name. To define a surface named surf_name, use

\[ \text{contact surface } \text{surf_name} \text{ contains } \text{entity_strings} \]

Here ‘entity_strings’ may reference blocks, sidesets or nodesets in the input mesh. It may reference a block, surface, or node set by its name. Or mesh entities can be accessed using the Sierra/SM convention, by index using ’block_##’, ’surface_##’ or ’nodelist_##’.

General contact is a far-reaching capability. If there are 2 (i.e. more than one) interaction, and general contact is on, then the contact module automatically detects all interactions between any pair of surfaces (except self interactions, and assuming that self contact is off).

8.2.1. Defining Contact Surfaces

Each contact definition block is a self-contained description of contact surfaces and how those contact surfaces interact. Several options are available for defining contact surfaces as shown in the following examples.

```plaintext
// Create contact surface from exterior skin of blocks
contact surface fixture contains block_1 block_9 block_12

// Create contact surface from union of sidesets
contact surface bolt_flange contains surface_1 surface_10

// Short cut to create contact surface based on the exodus part names.
contact surface block_7
contact surface surface_10
contact surface bearing contains bearing1 bearing2

// Create contact nodeset from union of node sets
contact nodeset ns1 contains nodelist_7 nodelist_11

// Create contact nodeset containing all nodes in a set of blocks
contact nodeset ns2 contains block_15 block_19

// Generate exterior skins for all blocks
skin all blocks = on
```
// Generate exterior skins for most blocks
skin all blocks = on exclude block_7 block_11

The skin of a finite element block contains all the exterior faces. When using 'skin all blocks' one contact surface is created for each block in the mesh, the contact surface is given the same name as the block. The block skinning algorithm is described in the Sierra SM Users Manual. For a model using only solid elements block skinning relieves the user from having to set an extensive number of sidesets in the input mesh.

Note the commands that create contact surfaces from blocks only work on solid and shell elements (hexes, tetrahedra, wedges, quads, triangles, etc.). If “ignore_shells” is set to true in the Dash options block, then all-to-all contact will ignore shells. Sidesets can be used to define contact surfaces on either solid or shell blocks. Contact nodesets can be defined on any element type, solid, shell or beam. When defining interactions a contact nodeset must be paired with a sideset or block skin in order to find node/face constraints. Two contact nodesets cannot directly interact.

In rare situations poorly posed cyclic/self contact constraints are created that are problematic for the linear solver. First the model is divided into disjoint components (without shared nodes). Pairs of disjoint components then tie to each other with one sided node-face interactions. Multipoint constraints tie nodes on one side of the interface to faces on the other side of the interface. Parts sharing nodes may have no unique node-to-face pairing. A conformally meshed part that contacts itself has no unique pairing.

If no unique pairing exists, nodes on side A of the interface generate MPCs with faces on side B and also nodes on side B generate MPCs with the faces of side A. The constraints may be over-determined. Some redundant constraints are removed. Determining unique constraints is an open problem.

The Interaction Weight Matrix shows the potential node-face interactions specified in the input deck. It is in the log file. A pair of surfaces must be in proximity to actually interact.

\[
\begin{array}{ll}
\text{disjoint} & \text{if} IJ = JI = 0 \\
\text{dependent node} I & \text{if} IJ = 1 \text{and} JI = 0 \\
\text{symmetric} & \text{if} IJ = JI = H \\
\text{error} & \text{otherwise}
\end{array}
\]  

(8.1)

Ideally the weight is either 1 or 0. This indicates one-way node face pair where nodes on one side of the interface interact with faces on the other side. Self contact is denoted by $H$ for half. For self contact nodes on side A of the interface interact with faces on side B and also nodes on side B interact with faces one side A. Such self contact constraints are often redundant and cyclic. The attempt to remove and make the contact constraints uniquely determined creates messages and warnings about removed and redundant constraints.

Even after the redundant constraint removal step the self contact constraints can cause solver robustness and accuracy issues. Thus, self contact should be avoided.
The disconnected component finder is available for setting up contact surfaces. Say a mesh contained a flashlight with four batteries, and the four batteries were all in the same element block in the mesh. The disconnected component finder would split this battery block into four separate contact surfaces. The disconnected component finder is useful for setting up interactions in a way that avoids self contact. See the Sierra/SM User Manual for more details on use of the disconnected component finder.

```
Begin Dash Options
    separate disconnected mesh components = TRUE|FALSE(FALSE)
End Dash Options
```

Surfaces may be defined to be in or out of contact solely based on proximity. The cutoff variable Dash option enables users to further filter contact based on any nodal input variable. Fine-grained contact information determined by Sierra/SM may be passed to Sierra/SD to refine its contact constraints. Including the following lines in a contact definition will ignore contact where the Sierra/SM celement field is below 0.55 on the nodes of the node-face constraints.

```
Begin Dash Options
    cutoff variable celement < 0.55
End Dash Options
```

Multiple such lines may be defined in a single Dash Options block. This would enable the filtering of constraints based on the union of multiple nodal variables, or only retaining contact where a variable is inside a range \((a, b)\) (i.e. cutoff variable var_name < a and cutoff variable var_name > b).

cutoff variable is currently BETA release.
Enable with the “--beta” command-line option.

### 8.2.2. Setting up Contact Interactions

Once the contact surfaces are defined, the next step is to set up the interactions between those contact surfaces. The interaction defaults block can be used to define both which surfaces will interact with each other and how those surfaces interact. One and only one interaction defaults block may be present in a contact definition. The interaction block can enforce contact between specific surface pairs and set the interaction parameters for that pair (overriding the interaction defaults behavior for the surface pair.) Any number of interaction command blocks may be present in the contact definition.

Contact is used to tie structures together that are in adjacent. The normal and tangential tolerance define how far away from a face a node can be and still find contact. By default, a reasonable normal and tangential tolerance is automatically computed in the Dash contact library based on a small fraction of the characteristic element size. Thickness of shell elements are also considered in the default tolerance. For shell blocks the default
search tolerance is set to at least sixty percent of the maximum element thickness of the elements in the block.

BEGIN CONTACT DEFINITION <name>
BEGIN INTERACTION DEFAULTS
   GENERAL CONTACT = OFF|ON(OFF)
   SELF CONTACT = OFF|ON(OFF)
   NORMAL TOLERANCE = <real>
   TANGENTIAL TOLERANCE = <real>
   CONSTRAINT FORMULATION = NODE_FACE|FACE_FACE(NODE_FACE)
   FRICTION MODEL = TIED|FRICITIONLESS(TIED)
END INTERACTION DEFAULTS
BEGIN INTERACTION
   SIDE A = <string_list>
   SIDE B = <string_list>
   SURFACES = <string_list>
   NORMAL TOLERANCE = <real>
   TANGENTIAL TOLERANCE = <real>
   CONSTRAINT FORMULATION = NODE_FACE|FACE_FACE
   FRICTION MODEL = TIED|FRICITIONLESS
   INTERACTION BEHAVIOR = NO_INTERACTION
END INTERACTION
END CONTACT DEFINITION

Command options:

- **general contact**: On means that every surface will contact every other surface. By default, the Dash contact library will pick which surface is used for nodes and which for faces in each surface-to-surface pairing automatically. Generally finer meshed surface will provide the nodes and the coarser the faces. Additional considerations may also be taken into account for selection of node and face surfaces such as avoiding cyclic constraints.

- **self contact**: Self contact on indicates a surface may contact itself, this could occur if a structure folds over on itself. Self contact should generally be avoided in SD as it leads to over-constraint problems.

- **normal tolerance** and **Tangential Tolerance**: The default face-sized based search distance can be overridden by manually specifying the normal and tangential tolerances. These tolerances have units of length.

- **constraint formulation**: The constraint formulation line defines the constraint type to be used. Only the node_face option should be used with Sierra/SD. The face_face option is experimental.
• **friction model**: The friction model line selects the type of contact constraint, either tied or frictionless. Tied contact ties all translational DOFs together at the interface preventing any normal or tangential motion. For structural problems each of the three translational degrees are constrained together. Rotational DOFs are never tied. For acoustic-acoustic contact the acoustic degree of freedom is tied together. For structure-acoustic contact the normal-motion of the structure is tied to the acoustic degree of freedom. The frictionless keyword selects sliding contact that is tied in the normal direction only. For frictionless structural contact only the surface-normal motion of the contacting surfaces are constrained together, the surfaces are free to slip in the tangential directions. The normal direction for the frictionless constraint is taken from the normal of the contacting face. For acoustic-acoustic or structural-acoustic contact frictionless contact is equivalent to tied.

• **SIDE A SIDE B**: For node/face contact the nodes are defined by the B surface and the faces by the A surface. If multiple surfaces are given for side A or side B, then the faces of each side A surface will be constrained to then nodes of each side B surfaces.

• **surfaces** defines a set of surfaces in contact. The Dash library picks the node face pairing automatically based on relative mesh density and other considerations such as avoiding cyclic constraints. If more than two surfaces are given a contact interaction will be formed between each surface in the list and each other surface in the list. Using SIDE A or SIDE B and SURFACES commands in the same interaction section will cause an error.

• **interaction behavior** The special interaction behavior command allows turning off contact between specific surface pairs. The no_interaction option would generally be paired with interaction defaults general contact on and used to turn off specific pairings where contact should not occur, such as slide lines.

### 8.2.3. Gap removal and element quality

As with tied surfaces 8.1, 3.35 the Contact Definition supports gap removal.

\[
\text{GAP REMOVAL} = \text{OFF|ON(ON)}
\]

The rewards and risks of gap removal are described in Section 3.35.

**Gap removal** will trigger output of an element quality table listing the worst element in each block before and after gap removal. The element quality metric used in this table is not consistent with Sierra Solid Mechanics. Element condition number is used instead. This maintains consistency with other features within Sierra Structural Dynamics, and with other Sierra applications such as Cubit.

The table will begin with the header shown below:
A value of 1.0 is an ideal element. As the value approaches INF the quality is decreasing. Values less than or equal to zero indicate the element cannot compute condition number. For example a beam, conmass, Rbar, etc. A value of N/A also means no shape metric exists for that element topology. Element quality can be plotted on the mesh with the ElementQuality variable, triggered by the gap\_removal solution case.

The table contents below the header are too wide to fit in this document.

**8.2.4. Examples**

```plaintext
// Most basic contact definition to tie everything that touches
begin contact definition
  skin all blocks on
  begin interaction defaults
    general contact = on
  end interaction defaults
end contact definition

// Tie a few specific surfaces, like in 'tied data' block
begin contact definition
  contact surface s2 contains surface_2
  contact surface s3 contains surface_3
  contact surface s4 contains surface_4
  begin interaction
    side A = s2
    side B = s3 s4
    normal tolerance 0.25
  end interaction
end contact definition

// All-to-all contact with some custom tolerances
begin contact definition
  skin all blocks = on
  begin interaction defaults
    general contact = on
    normal tolerance 1e-3
    tangential tolerance 1e-6
  end interaction defaults
end contact definition
```
// Tie the nodes of the two bolt flanges to a fixture block
begin contact definition
    contact nodeset boltFlange1 contains nodeset 901
    contact nodeset boltFlange2 contains nodeset 902
    contact surface fixture contains blocks_1 block_3 block_4
begin interaction
    side A = fixture
    side B = boltFlange1 boltFlange2
end interaction
end contact definition

// All-to-all contact, but turning off tying of a specific
// surface pair and using sliding contact for different
// surface pair.
begin contact definition
    skin all blocks = on
    begin interaction defaults
        general contact = on
    end
    begin interaction
        surfaces = piston piston_housing
        friction model = frictionless
    end-interaction
    begin interaction
        surfaces = drive_shaft drive_bearing
        interaction behavior = no_interaction
    end interaction
end contact definition

8.2.5. Notes and Usage Guidelines

- Multiple 'begin contact definition' blocks may be included in a single analysis. However, as with Sierra/SM using multiple contact definition blocks is generally discouraged. If multiple contact definitions are used, then the surfaces used in the contact definitions can not overlap. If the same surfaces are used in multiple contact definitions duplicate and/or incompatible constraints may be found between the contact definition blocks causing solver difficulty.

- Contact constraints are ultimately enforced by translational MPCs. The MPCs generated by contact tie only translational degrees of freedom.

- Self contact (contact constraints generated from a surface folding over on itself) are often over-determined and cannot be enforced accurately. Though the contact
definition can find such self contact constraints, the use of self contact should be avoided.

- If contact constraints are defined at a significant gap those constraints will artificially impede the rotation of the model. Gap removal can help with this issue, but it is recommended to only use contact to tie objects that are in close physical proximity.

- Use the 'outputs' option [constraint_info] to visualize more information about the generated contact constraints. The nodes of a node/face contact constraint will have a '1' for [node_face_mpc_count] output. The [mpc_status] nodal field will be painted with '1' for any node involved in a contact constraint on either the node or face side. If any nodes have the [node_face_mpc_both_node_and_face] flag this could indicate an issue of over-constraint.

8.2.6. Differences Between SM and SD Defaults

- By default, SD uses node/face constraints. SM uses face/face constraints by default. Face/face constraints may be optionally used within SD. However, face/face constraints are considered experimental for SD at this time and not recommended.

- By default, SD will try to remove the gap from contact constraints. SM does not have a gap removal option at this time.

- In SD the friction model defaults to 'TIED'. In SM the friction model defaults to 'FRICTIONLESS'.

- The SD friction model is a linearization of the SM frictionless model. A key difference is that in SD the frictionless constraint is free to slide on the face but can have no motion in the normal direction of the face. In SM the sliding along the face is also unconstrained, the node is prevented from penetrating the face, but differing from SD the frictionless constraint can open gap and separate the surfaces. This different behavior for the positive and negative normal directions is a fundamentally nonlinear behavior not applicable to linear structural dynamic analysis.

8.3. Lofted Surfaces and Gap Removal

Lofted surfaces are important because analysts often build meshes with an initial gap between the surfaces. If standard methods are used to tie the surfaces, but the separation (or lofting) is not taken into account, then the constraints are no longer consistent with rigid body motion. Generally this means that the rotational rigid body motion introduces strain into the system.

There is a gap removal 3.35 solution.
8.3.1. Example

This example illustrated in Figure 8-67 uses the coordinates listed in Table 8-145. In this figure, the face (represented by nodes 1-4) constrains each of the 3 nodes (nodes 5, 6, and 7.) Node “6” is on the face. It will be clear that standard methods apply the proper constraints. However, nodes “5” and “7” are offset from the face. As a consequence, constraint equations written for a node constrained to the face introduce errors when applied to the lofted node.

![Lofted Constraint Example](image)

Figure 8-67. – Lofted Constraint Example.

<table>
<thead>
<tr>
<th>Node</th>
<th>Coordinates</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0 4 0</td>
</tr>
<tr>
<td>2</td>
<td>0 2 0</td>
</tr>
<tr>
<td>3</td>
<td>2 0 0</td>
</tr>
<tr>
<td>4</td>
<td>4 0 0</td>
</tr>
<tr>
<td>5</td>
<td>1 3 0</td>
</tr>
<tr>
<td>6</td>
<td>1 1 0</td>
</tr>
<tr>
<td>7</td>
<td>3 1 0</td>
</tr>
</tbody>
</table>

Table 8-145. – Coordinates of Face (red) and Nodes (blue).

8.3.2. Projection Approach

The constraint equations from a conventional approach (meaning that the constraints are written by projecting the node location to the plane of the face, but not adjusting for the lofting) are shown in Table 8-146. These equations are not orthogonal to rigid body modes, and as a consequence, there are only two zero energy modes for this system rather than the 3 we anticipate. ¹

¹All motion out of the $xy$ plane has been eliminated.
\[\begin{align*}
u_x(1) + u_x(2) - 2u_x(5) &= 0 \\
v_y(1) + u_y(2) - 2u_y(5) &= 0 \\
u_x(2) + u_x(3) - 2u_x(6) &= 0 \\
v_y(2) + u_y(3) - 2u_y(6) &= 0 \\
u_x(3) + u_x(4) - 2u_x(7) &= 0 \\
v_y(3) + u_y(4) - 2u_y(7) &= 0
\end{align*}\]

**Table 8-146.** – Conventional Constraint Equations.

These constraints are represented by the matrix \(C\), where the \(x\) and \(y\) dofs are grouped together.

\[
C = \begin{bmatrix}
1 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & -2 & 0 & 0 & 0 & 0 \\
0 & 1 & 0 & 1 & 0 & 0 & 0 & 0 & -2 & 0 & 0 & 0 & 0 \\
0 & 0 & 1 & 0 & 1 & 0 & 0 & 0 & -2 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 1 & 0 & 1 & 0 & 0 & 0 & 0 & -2 & 0 & 0 \\
0 & 0 & 0 & 0 & 1 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & -2 \\
0 & 0 & 0 & 0 & 0 & 1 & 0 & 1 & 0 & 0 & 0 & 0 & -2
\end{bmatrix}
\]

The three rigid body vectors (in this 2D frame) are,

\[
R = \begin{bmatrix}
1 & 0 & 1 & 0 & 1 & 0 & 1 & 0 & 1 & 0 & 1 & 0 & 1 & 0 \\
0 & 1 & 0 & 1 & 0 & 1 & 0 & 1 & 0 & 1 & 0 & 1 & 0 & 1 \\
-3 & -1 & -1 & -1 & 1 & 1 & 1 & 3 & -1 & 0 & 0 & 0 & 0 & 1
\end{bmatrix}
\]

where the first two vectors represent translations, and the last is a rigid body rotation about point “6”. The product of \(C \ast R^T\) can be computed.

\[
C \ast R^T = \begin{bmatrix}
0 & 0 & -2 \\
0 & 0 & -2 \\
0 & 0 & 0 \\
0 & 0 & 0 \\
0 & 0 & 2 \\
0 & 0 & 2
\end{bmatrix}
\]

Each row of this matrix corresponds to a constraint equation from Table 8-146. Each column is associated with one of the three rigid body vectors. The translational rigid body vectors are orthogonal to the constraint matrix; the products are zero and no strain is induced. However, rotation about node 6 induces strain. The constraints are not invariant to rotation.

In a transient dynamic analysis with modest rotations and small gaps the effects of these constraint errors are often imperceptible. However, for large rotations or large gaps they may become apparent. They are always observable in modal analysis where they manifest as nonzero rigid body modes.
Mitigation Strategies As demonstrated in the previous section, constraint errors can introduce resistance to either translational or rotational rigid body motion. There are several strategies to mitigate these issues.

1. The analyst building the model ensures there are no projection errors.
2. Correct the initial geometry (using gap removal) so there are no projection errors.
3. Modify the constraints through algebraic means to ensure that they are orthogonal to rigid body motion.
4. Use the constraints appropriate to the lofted geometry. There is no software to do this currently.

If these methods cannot be applied, then the analysis must absorb the errors due to the constraint errors introduced by projection.

8.4. Moving MPCs

Sierra/SD supports moving contact through the use of moving MPCs. This can be enabled using the solution parameter `nUpdateConstraints = 1` as shown in 2.3. There are several other recommended parameters to be used with moving contact. These will be described in this section.

The example input blocks below show how the moving MPCs may be enabled. The `predictorCorrector` is set to 0 within the `transient` block. In the `Parameters` block, the solver should be set to update constraints without updating the matrices by setting `solverReset = constraints`. There is an inexpensive preconditioner for acoustic problems with moving constraints. To select this preconditioner, set the `preconditioner_type`, `krylov_method`, `orthog`, and `max_numterm_C1` as shown in the example input file below. The final two parameters in the `GDSW` block enable the block diagonal preconditioner with block size defined by the element condition number. To make a more powerful, but also more expensive, preconditioner increase the `max_element_condition`.

```plaintext
solution
    solver GDSW
    transient
        time_step 1.e-4
        nsteps 100
        nUpdateConstraints = 1
        predictorCorrector = 0
    end

parameters
    solverReset = constraints
end
```
GDSW
  preconditioner_type DIAG
  krylov_method PCG
  orthog 0
  max_numterm_C1 0
  identify_low_quality_elements true
  max_element_condition 5
end
9. EXAMPLE INPUT FILES

Example input files are shown for several types of analyses. The input file is case-insensitive except for special cases such as file names.

9.1. Eigenvalue problem

The following input file will output the first four mode shapes to an Exodus output file name hexplate-out.exo. A results file, hexplate.rslt, will not be created since no results have been selected for output in the echo section.

```
Solution
  eigen
  nmodes 4
  title 'Mode Shapes of Lowest Frequency Modes'
end

file // finite element mesh
  geometry_file hexplate.exo
end

Boundary
  nodeset 77
  fixed
end

outputs
  deform
end

block 44 // The default is the Hex8b
  material 3
  hex8
end

Material 3
  name "steel"
  E 30e6 +/- 1%
  nu .3
  density 0.288
end

sensitivity
```
values all
end

9.2. **Anisotropic Material**

The following input file is an example of a hexahedron mesh with anisotropic properties.

Solution
eigen
title 'Anisotropic Format'
end

file
  geometry_file anisogump.exo
end

Boundary
  nodeset 4 y = 0
  nodeset 5 x = 0
  nodeset 6 z = 0
end

load
  // sum of forces on surface should be equal to area
  // imposed forces are additive
  nodeset 1 force = 0.0 0.083333 0.0
  nodeset 2 force = 0.0 -0.041666 0.0
  nodeset 3 force = 0.0 -0.020833 0.0
end

outputs
  deform
end

block 1
  hex8
  material 1
end

Material 1
  name "anisotropic gump"
  anisotropic
  Cij
  1.346  0.5769  0.5769  0  0  0
  1.346  0.5769  0  0  0  0
9.3. Multiple materials

The next example shows the input for an Exodus model with many element blocks and materials. Keyword lumped in the Solution section selects a lumped (nearly diagonal) mass matrix.

Solution
eigen
nmodes 1
title 'Dozen blocks and six materials'
lumped
end
file
geometry_file multi.exo
end
Boundary
nodeset 1
fixed
nodeset 3
x = 0
y = 0
z = 0
RotY = 0
RotZ = 0
end
outputs
deform
end
// A block is required for each element block
block 1 //in the input Exodus (Genesis) mesh database.
material 2
Beam2
end
block 101
integration full
wedge6
material 1
end
block 2
  material 2
end
block 102
  integration full
  wedge6
  material 2
end
block 3
  material 3
end
block 103
  integration full
  wedge6
  material 3
end
block 4
  material 4
end
block 104
  integration full
  wedge6
  material 4
end
block 5  // Tip. Not capitalizing "material" here
  material 5  // helps to distinguish it
end // from a Material section.
block 105
  wedge6
  integration full
  material 5
end
block 6
  material 6
end
block 106
  wedge6
  integration full
  material 6
end     // Each material referenced in a necessary block
Material 1 // must be defined here. Extra materials are ignored.
  name "Phenolic"
  E 10.5E5
  nu .3
Material 2
  name 'Aluminum'
  E 10.0E6
  nu 0.33
  density 253.82e-6
end

Material 3
  name 'foam'
  E 100.
  nu 0.3
  density 18.13e-6
end

Material 4
  name 'HE'
  E 5E5
  nu 0.45
  density 129.5e-6
end

Material 5 // Tip. Capitalizing material helps to
distinguish it from a material in a block.
  name 'Uranium'
  E 30e6
  nu 0.3
  density 1768.97e-6
end

Material 6
  name 'wood'
  E 200.e3
  nu .3
  density 77.7e-6
end

9.4. Modaltransient

The next example shows the input for a modaltransient analysis. Accelerations are output to an Exodus file bar-out.exo. This example has damping, polynomial and linear functions. Also, sensitivities are calculated.

Solution
  modaltransient
    nmodes 10
    time_step .000005
    nsteps 100
nskip 1

title 'Test modal transient on prismatic bar'
end

file
    geometry_file bar.exo
end

outputs
    acceleration
end

Boundary
    nodeset 1
        fixed
end

damping
    gamma 0.001
end

block 1
    material 1
end

Material 1
    name "aluminum"
    E 10e6
    nu .33
    density 2.59e-4
end

load
    nodeset 3
        force = 1. 1. 1.
        function = 3
end

function 1
    type linear
    name "test_func1"
    data 0.0 0.0
    data 0.0150 0.0
    data 0.0152 1.0
    data 0.030 0.0
9.5. **ModalFrf**

In this ModalFrf analysis, accelerations are output to an Exodus file *bar-out.frq*.

Solution

    ModalFrf
    nmodes 10
       title 'Test ModalFrf on prismatic bar'
    end

    file
       geometry_file bar.exo
    end

    frequency
       freq_min 0
       freq_step=10
       freq_max=3000
    end

    outputs
       acceleration
    end

Boundary

    nodeset 1
       fixed
    end

damping
gamma 0.001
end

block 1
  material 1
end

Material 1
  name "aluminum"
  E 1e6
  nu .33
  density 2.59e-4
end

load
  nodeset 3
    force = 1. 1. 1.
    function = 3
end

function 2
// a smooth pulse of duration .05 sec
// peaking near t=.02 sec at 0.945
  type polynomial
  name "poly_fun"
  data 0. 0.
  data 2.0 -8.0e2
  data 0.5 8.9443
end

function 3
  type linear
  name "white noise"
  data 0.0 1.0
  data 10000. 1.0
end

9.6. Directfrf

A directfrf is run with displacements output to the Exodus file bar-out.frq.

Solution
directfrf
frequency
  freq_min = 1000.0
  freq_step = 7000
  freq_max = 5.0e4
  disp
  block 1
end

file
  geometry_file bar.exo
end

outputs
  disp
end

Boundary
  nodeset 1
    fixed
end

block 1
  material 1
end

Material 1
  name "aluminum"
  G 0.8E+9
  K 4.8E+9
  density 2.59e-4
end

load
  sideset 1
  pressure = -1.0
  function=3
end

function 3
  type linear
  name "white noise"
  data 0.0 1.0
  data 10000. 1.0
9.7. **Statics**

The following example is a *statics* analysis which will output stresses to the *Exodus* output file *quadt-out.exo*.

```
Solution
    statics
    title '10x1 beam of quadt'
end
file
    geometry_file quadt.exo
end
Boundary
    nodeset 1
        fixed
end
loads
    nodeset 2
        force = 1000.0 1000.0 0.0
end
outputs
    stress
end
block 1
    material 1
        QuadT
end
Material 1
    name "steel"
    E 30.0e6
    nu 0.25e0
    density 0.7324e-3
end
```
This page intentionally left blank.
BIBLIOGRAPHY


INDEX

acoustic
  dispersion, 54
  p0, 323
  Pdot, 323
  point source, 345
  scale, 346
acoustic_accel, 342, 345–347
acoustic_vel, 342, 345–347
AcousticFraction, 177
acoustic_light_hill, 346
adiag, 398
aeigen, 145
  anblocksize, 146
  anverbosity, 146
  eig_tol, 145
  shift, 145
AllowExodusDistFacts, 342
AllStructural, 45
analysis_direction, 137
Anasazi, 145, 160, 171
angular_acceleration, 360
angular_velocity, 360
anisotropic, 209–211
  example, 439
APartVel, 395
APressure, 395
ARPACK, 143
average, 314

Beam2, 261, 263, 264, 267, 268, 376
begin-periodic, 331
bending_factor, 250
block, 28, 213, 214, 228, 290, 298, 306, 341, 416
  blkalpha, 232–235
  blksbeta, 232–235
  coordinate, 232
density_scale_factor, 232

non-structural mass, 233
nonlinear, 232
NSM, 233
nsm, 232
parameters, 231
stiffness_scale_factor, 232
block_energies, 385
blockwise density scaling, 234
boundary, 164, 318, 330, 331, 344
  fixed, 319
infinite element, 327
  displacement1, 329
FieldTime, 329
P, 319
RotX, 319
RotY, 319
RotZ, 319
slosh, 326
V, 322
X, 319
Y, 319
Z, 319
buckling, 150
  nmodes, 150
  shift, 150
case, 121
CBModel, 125, 127, 128
  file, 129
finite_difference, 133
format, 129, 307
GlobalSolution, 129
inertia_matrix, 129
netcdf, 128
nodeset, 129
OTM, 131
OTME, 131
OutElemMap, 131
OutMap, 131
sensitivity_method, 133
sideset, 129
spoint_offset, 129
CBR see Craig-Bampton reduction, 125
ceigen, 160, 171, 175, 216, 377
centrifugal, 359
centripetal, 359
checkout, 123
Cij, 211, 226
CMS see component mode synthesis, 125
command line
Aprepro, 16
beta, 16
complex load, 358
component mode synthesis, 125
cont_tolerance, 27
condition_limit, 40
ElemQualChecks, 40
ConMass, 270
ConMassA, 270
consistent loads, 352
constrain_rbm, 363
constraint formulation, 429
constraint_info, 208, 395
ConstraintCorrectionFrequency, 195
ConstraintErrorDiagnostics, 195
constraintmethod
Lagrange, 54
contact
normal, 420
contact definition, 12, 207, 425
Dash, 425
element quality, 430
friction model, 430
gap removal, 430
gap_removal, 207
initial overlap removal, 207
interaction, 428
interaction defaults, 428
side, 430
surfaces, 430
coordinate, 78, 228, 234, 250, 253, 256, 341, 360, 404
copy, 42
Coriolis, 359
Craig-Bampton reduction, 125, 125, 126, 306
correction=vectors, 126
inertia tensor, 127
mass inertia matrix, 127
null space correction, 126
RbmDof, 126
cutoff variable, 428
damper, 238, 274, 287
cubic, 287
viscous, 274
damping, 164, 235
block, 232
CJdamp, 124, 225
CJetaFunction, 124, 225
Conor Johnson, 124
ratiofun, 237
dashpot, 274, 287
database name, 373, 405, 407
dd_solver_output_file, 62
DDAM
analysis_direction, 136
athwartship, 136
fore_and_aft, 136
vertical, 136
preddam, 134
load, 134
modalfilter, 134
ddamout, 136, 398, 400
dead, 308
defaultSpecificHeat, 43
density, 217
blockwise scaling, 234
density_scale_factor, 234
diagnostics, 22
adiag, 398
beams, 397
cubit, 23
epu, 24
explore, 23
kdiag, 25, 397
stk_balance, 23
dielectric, 227
directfrf, 138, 158, 328
  example, 445
displacement, 329, 391
  prescribed, 321
DMIG, 303, 305, 307
echo, 34, 164, 169, 208, 254, 306, 384, 385,
  397, 405, 414, 416, 438
Elmat, 418
GEnergies, 384
mass=block, 416
massblock, 416
material, 418
memusage, 418
modal amplitude, 417
MPC, 208, 416
NLresiduals, 415
subdomains, 34
eforce, 290, 389, 391, 414
eig_tol, 24, 140, 173
eigen, 38, 121, 140, 143, 145, 146, 151,
  158, 163, 164, 169, 389
  example, 438
fluidloading, 143
nmodes all, 140
restart, 51
untilfreq, 143
eigenvalue problem
  comparison, 170
eigen_norm, 43
element checks, 376
Largest_Ev, 147
normalization, 43
quadratic, 175, 178
structural acoustics, 175
structural acoustics (modal basis), 178
tolerance, 38
elastic-plastic see elas element, 286
elastic_strain, 378
ElemEigChecks, 376
element
  Beam2, 261
  ConMass, 270
dashpot, 274
deal, 308
eigenvalue checks, 376
eplas, 286
force, 389
Ftruss, 268
Gap, 291
  Joint2G, 285
Gap2D, 294
GasDmp, 295
Hex20, 242
Hex8, 241
HexShell, 256
Hys, 277
dmax, 277
fmax, 277
kmax, 277
kmin, 277
Hysteresis element, 277
Joint2G, 279
  Iwan, 279
  Property, 279, 286, 287
line_weld, 288, 290
  gap removal, 288
line_weld_force_rst, 290
line_weld_moment_rst, 290
Nbeam, 264
Nmount, 296
nquad, 247
ntria, 247
offset shell, 254
orientation, 394
Quad8T, 243
QuadM, 245
QuadT, 243
Rigid
  Rbar, 299
  RBE2, 300
  RBE3, 300
  Rrod, 298
RSpring, 272
Spring, 271
Spring3, 273
SpringDashpot, 275
Stress/Strain, 410
Superelement, 302
Tet10, 243
Tet4, 243
TiBeam, 268
Tria3, 249
Tria6, 243
TriaShell, 249
Truss, 268
Truth Table, 410
Wedge15, 242
Wedge6, 242

ElemQualChecks
  condition_limit, 376
  limitations, 24
energy_exo_var, 41, 350, 352
energy_load, 343, 352
energy_time_step, 41, 352
enforced acceleration
  random vibration, 366
engineering units, 37
eorient, 257, 379, 394, 412
Euler force, 359
evaluate expression
  in Function
    examples of, 96
    rules and options for composing, 95
Exodus
  assemblies, 32
  creation, 33
  entity types, 32
  naming
    limitations, 33
Exodus Read Functions, 354

Farhat, Charbel, 1
fatigue, 148, 163
  material, 217
  S-N curve, 218
Fatigue Solution, 147
Felippa, Carlos, 1
fiber_orientation, 252
FilterRbm, 237
FilterRBMLoad, 56
FilterRbmLoad, 13, 42, 45, 202, 363
flush, 47
force, 205
  constraint force, 388
  reaction force, 388
frequency, 384, 389, 391, 408
FRF, 389, 391
Ftruss, 268, 269
fuego, 42
function, 86, 87, 116, 138, 164, 204,
  322, 337, 346, 365
  blended, 109
  exo_var, 335, 337, 338
ExodusRead, 338
linear, 88
loglog, 91
offset, 87
Piecewise Linear, 89
planar step wave, 104
plane wave, 101
  K0, 101, 104
  tied data, 101
plane_wave_freq, 103
polynomial, 91
random, 93, 335, 337
  interp, 93
ReadNodal, 335, 348
  interp, 335
ReadNodalSet, 336
ReadSurface, 337, 348
  velX, 337
  velY, 337
  velZ, 337
SamplingRandom, 92
shift, 87
SpatialBC, 334
spherical_wave, 105
table, 90, 114
  datafile, 115
  dataline, 115
delta, 115, 359
dimension, 115
origin, 115, 359
size, 115
tablename, 91
undex loads, 106
wet surface pressure, 108

Gap, 286
  ellipsoidal, 294
Gap element, 285, 291, 294
Gap2D, 294, 294
GasDmp, 295
GDSW, 55, 62, 363
  constrain_rbms, 56
    p, 56
    rotx, 56
    roty, 56
    rotz, 56
    x, 56
    y, 56
    z, 56
  options, 52
Parameters, 55
prt_debug, 67, 69
solver_options, 61
solver_tol, 46
SuperLUDist, 55, 61, 66
general contact, 429
Generalized Alpha, 195
  rho, 195
gometry_file, 31, 31, 189
global, 402
global variables, 393
globalHist, 405
grepos, 204

hand-off, 182, 188, 211, 333, 338, 425
Hex20, 242
Hex8, 241, 242
Hex8b, 241
Hex8F, 242
Hex8u, 241
HexShell, 256, 257
  autolayer, 256
  Mass, 260
history, 169
hydrostatic balance, 203
I1, 261
I2, 261
iforce, 358
ignore_gap_inversion, 45
igravity, 358
ImagRelDispGxx, 391
ImagRelDispGyy, 392
ImagRelDispGzz, 392
imoment, 358
impedance_pressure, 326
impedance_shear, 326
initial overlap, 207
initial-conditions, 366
  time, 367
  velX, 366
  velY, 366
  velZ, 366
input
  acceleration, 322
  Aprepro, 16, 30
  comment
    entire section, 29
    comments, 28
  end, 28
input see hand-off, 28
interaction behavior, 430
interpolation, 42
ipressure, 358
isotropic, 209
isotropic_viscoelastic, 209
isotropic_viscoelastic_complex, 217
iteration, 358
Iwan, 238, 282
Joint2G, 286, 287, 302, 313, 317
  relative_disp, 391
keepmodes, 163
krylov_solver_output_file, 62
layer, 252
left_stretch, 211
lf_cutoff, 163
Lighthill, 345, 346
linesample, 197, 329, 408
load, 122, 164, 194, 204, 341, 341, 354, 365
  body, 341
  complex, 358
consistent, 352
electrostatic, 353
follower, 342
follower stiffness, 344
function, 354
random pressure, 354
scale, 354
spatially-dependent, 334–337
statics, 353
transient, 354
loads, 102, 103, 122, 150, 164, 322, 324, 340, 340, 341, 344, 362, 365
Lumped see mass matrix lumping, 53
mass, 416
  blockwise properties, 416
  properties, 416
mass matrix lumping, 53, 249, 255, 440
material, 208, 212
Lamé, 211
  begin-lame-material, 211
  end-lame-material, 211
  lame_state_hyperfoam, 211
acoustic, 213
anisotropic, 211
complex Viscoelastic, 217
  Gim, 217
  Greal, 217
  Kim, 217
  Kreal, 217
density, 217
E, 209
example, 440
exodus mesh properties, 223
G, 209
isotropic, 209
K, 209
layered, 249
nu, 209
orthotropic, 210
specific heat, 224
temperature dependent, 222
temperature function, 222
viscoelastic, 214
material_direction_1, 374
material_direction_2, 374
material_direction_3, 374
matrix, 365
  file names, 387
  output in MFile format, 387
  Ran Loads parameter, 365
  matrix-function, 163, 164, 365
  nominal, 112
table, 112
MatrixFloor, 42
max_newton_iterations, 182
MaxmpcEntries, 43
maxRatioFlexibleRbm, 238
membrane_factor, 250
memory diagnostics, 24
Memusage, 418
mesh discretization error, 386
mesh_error, 4, 386
MFile, 386
mfile, 4
MinimumNodalSpacing, 357
mk super, 302, 304, 305
modal acceleration, 158
modal effective mass, 154, 155
  meff, 154
  MPF, 154
modal_amp, 387
modalfilter, 143
modalfiltercase, 143
ModalFraction, 177
ModalFr f, 138, 157–159, 236, 444
  example, 444
  nrbms, 159
  use modalaccel, 159
modalranvib, 163, 163, 164, 236, 389, 391, 392
  acceleration, 163
  noSVD, 163
  RMS von Mises stress, 163
modalshock, 167
modaltransient, 50, 51, 121, 168–170, 233, 236, 358, 442
  example, 442
modalvars, 169, 359, 417
mortar method, 422
dual, 422
standard, 422
MPC see multipoint constraint, 43
mpmd_transfer_sidesets, 42
mpmd_transfer_type, 42
mpmd_transfer_version, 3, 42
multipoint constraint, 40, 116
  constraint force, 388
  constraint_correction, 43, 195
  constraint_info, 395
nUpdateConstraints, 195
orthogonalization, 43
problematic, 27

NASTRAN
  auto spc, 40
  output4 in CBR, 129
Nbeam, 261, 264, 264–267
NegEigen, 140
neglect_mass, 328
netcdf, 134, 305–307
new, 42
Newmark beta, 195, 196
Ng, Esmond, 2
NLstatics, 179, 181, 182
NLtransient, 181, 182
nmodes, 126, 143, 152, 159, 163, 236
Nmount, 296, 296
  stability, 296
nodal, 402
nodal_charge, 400
NodeListFile, 324, 404
nodeset, 128, 336, 341
nodesets_with_disp, 46, 195
nonlinear, 231, 234
nonlinear_default, 38, 231
normal tolerance, 429
npressure, 205, 395
nquad, 243, 247
nquad_eps_max, 248
NSC, 42
ntria, 247
num_newton_load_steps, 179, 182
num_procs, 47
num_rigid_mode, 45
nUpdateConstraints, 46
nUpdateDynamicMatrices, 223, 350
nUpdateTemperature, 223, 350, 351
offset shell, 254, 255
orthotropic, 209, 210, 249
orthotropic_layer, 210
orthotropic_piezoelectric, 226
orthotropic_prop, 209, 210
output
  block statistics, 400
    Name, 400
    Variable, 400
  element force, 389
  error metrics, 386
  history, 403
    coordinate, 391
    nodes, 403
    relative_disp, 391
  Matlab, 371, 409
    exo2mat, 408
    relative_disp, 391
  user output, 400
    closest distance, 401
    derived variables, 402
output_sideset_data, 35
outputs, 40, 49, 129, 138, 148, 159, 164,
  169, 208, 290, 306, 346, 370, 370,
  371, 373, 384, 385, 389, 391, 392,
  394, 395, 397, 398, 404
  acceleration, 377
constraint_force, 388
constraint_info, 395
coordinate, 391
disp, 377
energy, 384
fia, 375
force, 388
GEneregies, 384
  work, 384
Globals, 385
history, 130
Kaa, 374
line_weld, 391
Maa, 374

461
material, 374
Matlab, 19, 130, 134, 154, 169, 198, 329, 417
MFile_Format, 44
MLumped, 375
MPhi, 375
reaction_force, 388
RHS, 389
RMS, 384
rotational_acceleration, 384
rotational_displacement, 384
strain, 378
stress, 379
velocity, 377
von Mises stress, 380
overlap removal see Gap Removal, 207

Padé, 139
parallel computing, 20
epu, 19, 20
exodus_file, 19
parameters, 35, 35, 56, 57, 197, 299, 350, 363, 422
info, 38
negeigen, 35
reorder_Rbar, 40
reserved_keywords, 39
syntax_checking, 39
wtmass, 35
performance, 17, 22, 55, 116
periodic boundary conditions, 331
permittivity_ij, 226
piezoelectric, 208
e_ij, 226
plane_wave, 104
point_volume_accel, 345–347
point_volume_acceleration, 9, 338
point_volume_vel, 345–347
point_volume_velocity, 338
power spectral density, 165, 366
acceleration, 165
coordinate, 165
displacement, 165
scaling, 365
preddam, 143
PredictorCorrector, 195
prescribed acceleration, 322
accelX, 322
accelY, 322
accelZ, 322
disp0, 322
RotaccelX, 322
RotaccelY, 322
RotaccelZ, 322
vel0, 322
prescribed frequency, 324
DispX, 324
DispY, 324
DispZ, 324
FreqP, 324
FreqV, 324
RotDispX, 324
RotDispY, 324
RotDispZ, 324
pressure, 342, 345, 394
depth dependent, 352
nodal, 395
prescribed, 321
pressure_z, 352, 353
Problematic Elements, 25
Problematic Subdomains, 24
projection_eigen, 171, 172, 178
property, 279, 286, 287, 373
PSD see power spectral density, 165
qevp, 158, 160, 170–173, 175, 177
Quad8T, 243, 243, 244
QuadM, 245, 245
quadm, 243
QuadT, 243, 243, 244
quadt, 243
Random Number Generator, 44
RandomPressure, 355
RanLoads, 164, 364, 364, 365
acceleration, 364
dimension, 364
rational function, 139
Rbar, 299, 300
reorder_Rbar, 40
sa_eigen, 160, 171, 172, 175
ErrorNorm, 177
limitations, 175
Salinas, 15
scattering, 54
sd_factor, 241, 242, 246
Section Commands
Loads Rigid Body Filter, 363
block, 228
block parameters, 228
boundary, 318
coordinate, 78
damping, 235
echo, 414
file, 31, 370
frequency, 103, 407
function, 86
history, 403
Load, 341
loads, 340
material, 208
matrix-function, 112
outputs, 370
parameters, 35
periodic boundary conditions, 331
RanLoads, 364
Rrodset, 310
sensitivity, 73
solution, 119
solver_options, 61
table, 114
tied surfaces, 419
user output, 400
block statistics, 400
closest distance, 401
derived variables, 402
self contact, 429
sensitivity, 73, 76, 133, 442
+/-, 76
Attune, 74
iterations, 73
tolerance, 73
values, 73
vectors, 73
sensitivity_param, 306
shear_axis, 280
shells
    offset, 254
shift, 142, 151
sideset, 128, 256, 337, 341, 342
SkipmpcTouch, 40
smoothing parameters, 423
Solution, 28, 46, 53, 54, 119, 119, 121,
        142, 151, 164, 180, 188, 204, 440
    AEigen, 145
    Buckling, 150
cceigen, 173
    Checkout, 123
    CJdamp, 124
complex eigen, 173
Craig-Bampton reduction, 125
DDAM, 135
DirectFRF, 138
Dump, 139
Eigen, 140
Fatigue, 147
Gap Removal, 207
    elementInversionFlag, 208
    elementQuality, 208
GeometricRigidBodyModes, 13, 201
Largest_Ev, 147
Modal Participation Factor, 153
ModalFilter, 151
    write_files, 151, 153
ModalFrf, 157
ModalRanVib, 162
ModalShock, 167
ModalTransient, 168
MPF, 154
NLStatics, 179
NLTransient, 181
Options, 46
    constraint method, 54
    GDSW, 52
    mass matrix lumping, 53
    restart, 47
    scattering, 54
    symmetrize_struc_acous, 54
    Table, 122
preddam, 134
QEVP, 170
Receive_Sierra_Data, 182
receive_sierra_data, 121, 182, 188,
        210, 217
    include_internal_force, 183
    no_geom_stiff, 184
Residual Vectors, 199
Statics, 188
Superposition, 188
Tangent, 190
TranShock, 191
    srs_damp, 191
Transient, 193
TSR_preload, 197
Waterline, 203
solver
    parameters, 55
solver_options, 62
sparc, 42
spherical_wave, 105
Spring, 271, 271, 272, 376
    cubic, 273
    Linear, 271
    Parameter Values, 272
    Rotational, 272
Spring3, 273, 273, 376
SpringDashpot, 275, 275
statics, 188, 348, 389
    example, 447
statistics, 396
    min/max, 396
    standard deviation, 396
stiffness_scale_factor, 235
strain, 378
    Gauss point, 379
stress, 3, 164, 211, 413
    Gauss point, 382
Stress and Strain Recovery, 409
stress recovery point, 413
Structural Acoustics
eigen, 175, 178
StructuralFraction, 177
subdomain
    output, 417
Superelement, 132
SuperLU, 2
superposition, 189
surface
description of, 32
surface_charge, 353
symmetrize_struc_acous, 55
syntax_checking, 39
tangent, 121, 122, 190, 361, 362
Tangential Tolerance, 429
TangentMethod, 38
tcoord, 256, 257
Temperature, 400
Tet10, 243, 243
Tet4, 243, 243
thermal_exo_var, 41, 350, 352
thermal_load, 41, 343, 348, 350, 352
thermal_strain, 378
thermal_time_step, 41, 223, 350–352
TiBeam, 268, 376
tied data, 207, 419
gap_removal, 27, 207
initial_overlap_removal, 207
transverse, 419
tied joint
normal, 312
shear_axis, 313
side, 313
slip, 313
surface, 312
tied_surface, 419
gap_removal, 419
TIndex, 393
tolerance, 179, 181
traction, 342
transhock, 191
transient, 51, 191, 193, 348, 389
nskip, 193, 405
nsteps, 193
start_time, 193
time_step, 193
Tria3, 244, 249, 249
Tria6, 243, 244
TriaShell, 244, 249, 249, 412
troubleshooting, 22
cubit, 23
explore, 23
TruncationMethod, 163
Truss, 268, 268, 376
units_of_measure, 35, 37
update_tangent, 179, 182, 291
user subroutine file, 31, 296
viscofreq, 173, 174
viscous_damper, 274
voltage, 400
accelV, 323
transV, 323
Volume, 400
volume_acceleration, 335
vonMises, 3
vrms, 384
waterline, 203
point_a, 203
point_b, 203
point_c, 203
Wedge15, 242
Wedge6, 242, 242
write_files, 151, 169
wtmass, 35, 366, 416
XML, 16
This page intentionally left blank.
### DISTRIBUTION

**Hardcopy—Internal**

<table>
<thead>
<tr>
<th>Number of Copies</th>
<th>Name</th>
<th>Org.</th>
<th>Mailstop</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>K. H. Pierson</td>
<td>1542</td>
<td>0845</td>
</tr>
</tbody>
</table>

**Email—Internal (encrypt for OUO)**

<table>
<thead>
<tr>
<th>Name</th>
<th>Org.</th>
<th>Sandia Email Address</th>
</tr>
</thead>
<tbody>
<tr>
<td>Technical Library</td>
<td>1911</td>
<td><a href="mailto:sanddocs@sandia.gov">sanddocs@sandia.gov</a></td>
</tr>
</tbody>
</table>