ATA ENGINEERING, INC.

Vibrata

Advanced Modal Dynamic Analysis

User Guide

Version 5.2

TABLE OF CONTENTS

1. Overview 1
 1.1. Features 1
 1.2. Architecture 2
 1.3. Files 3
 1.4. Note on the "Global Coordinate System" 4
2. Preparing Nastran Input files 5
 2.1. Nastran Cards for Specific Vibrata Options 5
  2.1.1. General Requirements for Physical Response Recovery. 6
  2.1.2. Load Set Excitation. 6
  2.1.3. Enforced Motion. 8
  2.1.4. Coupled Damping Matrix. 10
  2.1.5. Residual Modes. 11
  2.1.6. Modal Effective Mass Output. 13
  2.1.7. Composite Failure Indices. 14
  2.1.8. Vibrata Nastran Deck Setup Graphical Utility. 15
  2.1.9. Creating the Vibrata Database (.vra5). 18
  2.1.10. Coordinate System Considerations. 20
  2.1.11. Supported Nastran Entities. 20
  2.1.12. Using MSC Nastran with Vibrata. 22
 2.2. Supported Nastran Versions and Solution Types 23
3. Vibrata User Guide 25
 3.1. Starting Vibrata 25
 3.2. Event Manager—Interactive User Interface 25
  3.2.1. General Characteristics of the User Interface. 25
  3.2.2. Event List Region. 25
  3.2.3. FEM/Input Database Selection Region. 25
  3.2.4. Solver Tab. 25
  3.2.5. Modeling Nonlinearities in the Transient Solver 25
  3.2.6. Excitations Tab. 25
  3.2.7. Modal Settings Tab. 25
  3.2.8. Defining Modal Damping. 25
  3.2.9. Output Request Tab. 25
  3.2.10. Contour Output Requests. 25
  3.2.11. Post Processing Tab. 25
 3.3. Event Manager—Batch Solution Mode. 25
  3.3.1. Batch Processing with MATLAB Scripts. 25
 3.4. Function Manager 25
  3.4.1. Functions Region. 25
  3.4.2. Function Sets Region. 25
  3.4.3. Plot Region. 25
  3.4.4. Math Region. 25
  3.4.5. Hints for Creating Complex Functions. 25
  3.4.6. Creating Force-Limited Input. 25
  3.4.7. Applying a Modal (Spatial) Filter. 25

3.5. XY-Plot Results Display 25
3.6. Contour Results Display 25
3.6.1. Viewing Transient Analysis Results. 26
3.6.2. Viewing Frequency Response Results. 26
3.6.3. Viewing Random Analysis Results. 26
3.6.4. Viewing Response Spectrum Results. 26
3.7. Event Definition File (EVT) 26
3.7.1. Event File Conventions. 26
3.7.2. Version Information Section. 26
3.7.3. Event Summary Section. 26
3.7.4. Solver Section. 26
3.7.5. Excitation Section. 26
3.7.6. Modal Settings Section. 26
3.7.7. Input Checksum. 26
3.7.8. Output Request Section. 26
3.7.9. Post-Processing Section. 26
3.7.10. Reusing EVT Files. 26
4. Example Problems 27
4.1. Descriptions of Example Models 27
4.1.1. Single DOF Spring-Mass Model. 27
4.1.2. Two DOF Spring-Mass Model 27
4.1.3. Two-Dimensional Frame. 28
4.1.4. Isat—Inner Planets Exploration Satellite. 29
4.1.5. Preparing the Models for Vibrata. 32
4.2. Steady-State Frequency Response Analysis 34
4.2.1. Analyze 1DOF Model Using a Constant Input. 34
4.2.2. 2DOF Frequency Response. 41
4.2.3. Frequency Response of a Frame. 47
4.3. Random Analysis 60
4.3.1. Create Common Acceleration PSD. 60
4.3.2. Isat with Rigid Base. 62
4.3.3. Isat with Uncorrelated Input at Each Adapter Leg. 73
4.3.4. Isat with Correlated Inputs. 80
4.3.5. RMS Contours Using FastRMS. 86
4.3.6. Deactivate Modes with Negligible Modal Effective Mass. 88
4.4. Response Spectrum Analysis 92
4.5. Transient Analysis 106
4.5.1. Isat Model with RCS Thruster Firing. 106
4.5.2. Frame Model Transient Animation. 122
5. CREATING CUSTOM FUNCTIONALITIES 134
5.1. Creating Custom Solvers 134
5.1.1. The Solver File 134
5.1.2. The SolverInfo File 134
5.1.3. The Custom Solver Folder 134
5.2. Creating a Custom Nonlinear Model 134
5.3. Creating Custom Postprocessing 134

6. Vibrata-MATLAB API For Custom SOLVERS and Functions 135
6.1. Directory Structure 135
6.2. Function Naming Convention 135
6.3. Utility Classes 135
6.3.1. vraParam—Vibrata Parameter Class. 135
6.3.2. vraReqmap—Vibrata Request Mapping Class. 135
6.3.3. *fcn*—Vibrata Function Class. 135
6.3.4. vraNonlinearParam—Vibrata Nonlinear Parameter Class. 135
6.3.5. vraEvent.Event—Vibrata Event Class. 135
6.4. Example Solver 135
6.4.1. Initialization. 135
6.4.2. Generate Modal Quantities. 135
6.4.3. Gather Event Setup for Generating Modal Quantities. 135
6.4.4. Calculate Modal Quantities. 135
6.4.5. Determine Output Requests. 135
6.4.6. Compute Output. 135
6.4.7. Cleanup and Return. 135
7. Theoretical Manual 136
7.1. Normal Modes Analysis 136
7.2. Viscous and Structural Damping 136
7.3. Steady-State Frequency Response Analysis 136
7.4. Random Response Analysis 136
7.5. Transient Analysis 136
7.5.1. Static and Dynamic Uncertainty Factors. 136
7.5.2. Nonlinear Forces 136
7.6. Enforced Motion Excitation 136
7.6.1. Seismic Mass Alternative. 136
7.7. Residual Vectors 136
7.8. Response Spectrum Analysis 136
7.8.1. Absolute Summation (ABS). 136
7.8.2. Square Root Sum Square Summation (SRSS). 136
7.8.3. Naval Research Lab summation (NRL). 136
7.8.4. Nuclear Regulatory Commission Rule (NRC). 136
8. Installation 137
8.1. Platform Requirements 137
8.2. Installing the License Server 137
8.2.1. Installing Sentinel RMS. 137
8.2.2. Environment Variables. 137
8.2.3. Checking Your License Status. 138
8.3. Installing Vibrata 138
8.3.1. Installing in Silent Mode. 139
8.4. Configuring Vibrata 140
8.4.1. Vibrata Launch Script. 140
8.5. Troubleshooting Vibrata Launch Issues 141
9. References 143

Appendix A. Additional Output2 Data for Specific Analyses 144

List of Tables

Table 2-1. Supported failure theories. 14
Table 2-2. Supported NX/Simcenter Nastran enforced motion datablocks. 21
Table 2-3. Nastran element types supported by the Vibrata translator. 22
Table 4-1. Maximum response level for each summation method. 106
Table 8-1. Vibrata environment variables and their meanings. 140

List of Figures

Figure 1-1. Overview of Vibrata components. 3
Figure 2-1. Nastran cards to enable load set excitation. 8
Figure 2-2. Include a U2 USET and special PARAMs to enable enforced motion excitation. 9
Figure 2-3. Alternate deck for enforced motion using DMAP alter. 10
Figure 2-4. Include PARAM,RSOPT,1 to get coupled modal damping matrices. 11
Figure 2-5. Add a RESVEC case control statement to get residual modes for applied forces. 12
Figure 2-6. You can get residual modes by defining a U6 USET, without any actual loads. 13
Figure 2-7. Add "PARAM,RESVINER" card to get residual modes for enforced motion. 13
Figure 2-8. Request for modal effective mass output relative to a specific node. 14
Figure 2-9. Access the *Analysis Setup* GUI through the Custom Tools menu in Femap. 16
Figure 2-10. *Analysis Setup* GUI. 17
Figure 2-11. Access the .vra5 translation utility from the Start menu. 19
Figure 2-12. Create the .vra5 database from OP2 using the command-line interface. 19
Figure 2-13. Create the .vra5 database from OP2 in MATLAB. 19
Figure 2-14. Select the OP2 file to translate using a graphical file dialog. 20
Figure 2-15. Use these parameters to write out the LAMA datablock in MSC Nastran. 23
Figure 2-16. Insert a DMAP alter to use MSC Nastran with Vibrata base excitation analysis. 23
Figure 4-1. Single DOF spring-mass model. 27
Figure 4-2. Two-DOF spring-mass model. 28
Figure 4-3. Two-dimensional frame model. 29
Figure 4-4. Isat model in launch configuration with four separate attachment points. 30
Figure 4-5. Isat model in launch configuration with single central connection to launcher. 31
Figure 4-6. Isat model in deployed configuration. 32
Figure 4-7. Read the bulk data into Femap using the *Import/Analysis Model* menu picks. 33
Figure 4-8. Define groups in Femap by importing a neutral file when available. 34
Figure 4-9. Create a new frequency response event and select the FEM on which to base it. 35
Figure 4-10. Create a new enforced motion excitation. 36
Figure 4-11. Use the Function Manager to create an acceleration function. 37
Figure 4-12. Save excitation function to new function file. 38
Figure 4-13. Assign 2.5% modal viscous damping to the mode directly in the modes table. 38
Figure 4-14. Request XY output for total and flexible X-accelerations at node 32. 39
Figure 4-15. Solve for the requested output. 40
Figure 4-16. Results are available for requests in green text. Display them. 40
Figure 4-17. Select and plot both requested acceleration responses at once. 41
Figure 4-18. Set the number of points at which you want to compute responses. 42
Figure 4-19. With only two modes, you can assign damping directly in the damping table. 43
Figure 4-20. Define a point force excitation on node 13 in the X-direction. 44
Figure 4-21. Create a new function with a constant 10 lbf amplitude from 0 to 30 Hz. 45
Figure 4-22. Request X-acceleration response plots for all three nodes. 46
Figure 4-23. Plot the acceleration response of all DOF. 47
Figure 4-24. Create a new model frequency event using the frame model. 48
Figure 4-25. Apply a forcing function to the available enforced motion DOF. 49

Figure 4-26. If necessary, reopen the file containing the excitation function from the 1dof example.......... 50
Figure 4-27. Change the copied function to have a range from 1 to 40 Hz. 51
Figure 4-28. Edit the new function's name and make it use log-log interpolation. 52
Figure 4-29. Make the function uneven and give it non-constant amplitude. 53
Figure 4-30. *Excitations* tab with resulting enforced motion definition. 54
Figure 4-31. Apply 2% damping to all modes via damping schedule, then set mode 1 to 5%. 55
Figure 4-32. Request both flexible and total nodal X-translation plots. 56
Figure 4-33. Use the **Copy** button to create a new event from an existing one.......... 57
Figure 4-34. For the new (copied) event, delete the old excitation and create a new one. 58
Figure 4-35. Load results from the constant-amplitude event into the output plotter. 59
Figure 4-36. Compare the responses of the top corner node for the two inputs. 60
Figure 4-37. Access Function Manager directly from Event Manager menu.......... 60
Figure 4-38. Create the acceleration PSD that will be used to excite the Isat. 61
Figure 4-39.Create and save the function, and finish by clicking **Done**.......... 62
Figure 4-40. Create a new random analysis event. 63
Figure 4-41. Select enforced motion, select Z-translation, and then bring up the Function Manager.......... 64
Figure 4-42. Select the modes above 400 Hz and set their status as residual vectors.......... 65
Figure 4-43. Assign damping via the schedule; export the schedule to a text file for later use. 66
Figure 4-44. Request *SVMS* for the BUS_SHELLS and *TAT* for the INST_PKG_FEM; only *RMS* output. 67
Figure 4-45. Select von Mises Stress for the first contour plot and turn off averaging.......... 68
Figure 4-46. RMS von Mises stress in satellite bus shell elements, rigid base. 68
Figure 4-47. RMS acceleration contours for the instrumentation package, rigid base.......... 69
Figure 4-48. Request Y- and Z-direction total accelerations for node 5577.......... 70
Figure 4-49. Node 5577 total Y-acceleration in g^2/Hz units, rigid base. 71
Figure 4-50. Request axial forces in the launcher adapter legs. 72
Figure 4-51. Axial forces in the launcher adapter legs, rigid base. 73
Figure 4-52. Use the **Copy** button to start a new event from the event just completed. 74
Figure 4-53. Apply enforced motion in the Z direction at all four base nodes.......... 75
Figure 4-54. Delete the automatic CSD entries.......... 76
Figure 4-55. RMS von Mises stress in satellite bus shell elements, four-point base, uncorrelated. 77
Figure 4-56. RMS acceleration contours for the instrumentation package, four-point base, uncorrelated. 78
Figure 4-57. Add the rigid-base results to those available for plotting. 79
Figure 4-58. The Y-acceleration responses for the two events are very different.......... 80
Figure 4-59. Copy the uncorrelated event; the new event will have correlated inputs. 80
Figure 4-60. Define 0-degree phase lag correlations from the first input PSD to the other three. 81
Figure 4-61. Finish defining 0-degree phase shift correlations among all the input PSDs. 82
Figure 4-62. The *Excitations* tab will show the defined CSDs but not their conjugates. 82
Figure 4-63. RMS von Mises stress in satellite bus shell elements, four-point-base, correlated. 83
Figure 4-64. RMS accelerations: rigid-base [a], four-point uncorrelated [b], four-point correlated [c]. 84
Figure 4-65. The correlated four-point results match the rigid-base results for node 5577 Y-acceleration. 85

Figure 4-66. The correlated four-point results also match the rigid-base results for launcher leg forces. 86
Figure 4-67. Copy the original rigid-base event; the new one will use FastRMS contour calculations. 87
Figure 4-68. Turn on the *Use FastRMS* toggle on the *Excitations* tab. 87
Figure 4-69. The FastRMS contours are indistinguishable from the original event. 88
Figure 4-70. Copy the original rigid-base event; the new one will exclude modes with negligible effective mass. 88
Figure 4-71. Select modes below 0.1% modal effective mass for all translations. 89
Figure 4-72. All but 51 of the 174 modes are turned off, but 98% of the effective mass remains. 90
Figure 4-73. The mass-filtered contours are visually identical to those from the original event. 91
Figure 4-74. The node 5577 response shows differences at high frequencies, but RMS values match. 91
Figure 4-75. The adapter-leg axial force again shows differences only at high frequencies. 92
Figure 4-76. New response spectrum event with absolute value summation. 93
Figure 4-77. Assign an enforced motion function to direction 1. 94
Figure 4-78. Define a Response Spectrum velocity function. 95
Figure 4-79. Store the new function in the frame excitations file. 95
Figure 4-80. The excitation uses the basic coordinate system. 96
Figure 4-81. Damping is not needed and the damping dialog button is disabled. 96
Figure 4-82. Request translational displacements and accelerations at all nodes. 97
Figure 4-83. Copy the event and change the summation method to SRSS. 98
Figure 4-84. Copy again to create a third event using NRL summation. 99
Figure 4-85. Copy again to create a fourth event using NRC summation. 100
Figure 4-86. Select *Criteria* and *Deformed* views from Femap's *Post* toolbar. 100
Figure 4-87. Displacements for [a] absolute value and [b] SRSS methods. 102
Figure 4-88. Displacements for [a] NRL and [b] NRC 10% methods. 103
Figure 4-89. Accelerations for [a] absolute value and [b] SRSS methods. 104
Figure 4-90. Accelerations for [a] NRL and [b] NRC methods 105
Figure 4-91. Transient event setup for deployed Isat model. 107
Figure 4-92. Nodes for RCS thruster loads. 108
Figure 4-93. Use the Point Force dialog to assign a Z-direction force to each thruster node. 109
Figure 4-94. Thruster force transient for Isat maneuver. 110
Figure 4-95. Save the Isat thruster forcing function to a file. 111
Figure 4-96. Scale the function by 20 and then create the final excitations. 111
Figure 4-97. Use the damping schedule to assign 1% damping to all flexible modes. 112
Figure 4-98. Request displacement and acceleration contours for all nodes at 0.05 second intervals. 113
Figure 4-99. Solve for the requested contours and then plot them. 114
Figure 4-100. Contours of acceleration on deformed mesh for output frame 14. 114
Figure 4-101. Request peak values of von Mises stress in the RCS mounting panels. 115
Figure 4-102. Select the RCS Panels contour request and plot its results. 116
Figure 4-103. Peak von Mises stress contours in RCS panels. 117
Figure 4-104. Von Mises results with only the RCS panels group displayed. 117
Figure 4-105. Select the BUS_SHELLS contour request and plot its results. 118
Figure 4-106. Von Mises results with only the BUS_SHELLS group displayed. 119
Figure 4-107. Request beam forces in elements that attach appendages to bus. 120
Figure 4-108. Solve the new Element XY requests and plot them. 121
Figure 4-109. Forces in IP boom connector; note effects of thruster start/stop. 121

Figure 4-110. Frame model showing nodes of special interest. 122
Figure 4-111. Solver setup for frame model transient analysis. 123
Figure 4-112. Forcing function will be applied at node 10 in a direction taken from the FEM geometry. 124
Figure 4-113. Screen-pick node 10, then node 9, to define the force direction. 125
Figure 4-114. Define a unit force triangular pulse function.. 126
Figure 4-115. Save the new function to the frame excitations file. 126
Figure 4-116. Apply the forcing function to the Vibrata event...................................... 127
Figure 4-117. Set the damping of the model to 10% for mode 1, 5% for mode 2, and 1% elsewhere. 128
Figure 4-118. Request displacement plots for node 15. 129
Figure 4-119. Request displacement (*UT*) contours for the first 3 seconds of the event. .. 131
Figure 4-120. Transient displacements for node 15.. 132
Figure 4-121. Frame 7 of the displacement animation.. 133
Figure 8-1. [a] The installer could not find the required MATLAB Runtime in the registry. [b] The installer could not find MATLAB Runtime or a supported MATLAB installation. 139
Figure 8-2. An error similar to this will be shown if the MATLAB Runtime-based solver was not properly registered. 141

Vibrata utilizes technology from the IMAT and IMAT+FEA MATLAB toolboxes, also developed by ATA Engineering. For more information, visit http://www.ata-e.com/software/ata-software/imat/.

1. OVERVIEW

Vibrata™ is a comprehensive, easy-to-use modal dynamics tool for predicting structural dynamic response to transient, harmonic, random, and response spectrum excitation. The initial release addresses structures represented by a single finite element model (FEM), but future releases will handle system-level analyses with multiple separate components whose representations may come from many sources. The program integrates design, analysis, and test activities for products for which dynamics is an important issue.

Vibrata employs a modal postprocessing approach, so you must first solve your FEM for normal modes. The program is best adapted for using results from Simcenter and NX Nastran, but MSC Nastran results can be used for many basic analyses. Note that unless otherwise specified, when Nastran is mentioned in this user guide, we are referring to Simcenter (formerly NX) Nastran. **Also note that Vibrata does impose certain requirements on the contents of your Nastran input files, so it is important that you review section 2 before you solve your model for normal modes**. Normal modes from other solvers can also be used; see Appendix A for more details.

Vibrata uses modal data to solve for the specified dynamic responses. The software makes it easy to define, solve, and display responses for both simple and advanced dynamic problems. Each dynamic problem is called an event, and its definition includes the type of analysis to perform (transient, frequency response, random, response spectrum), the FEM and modes to use, the excitations, modal damping, solution range and resolution, and the physical responses to compute. Interactive graphical processing lets you focus on the engineering rather than the data input formats and data transfer issues. Additionally, you can develop custom dynamics solvers and new solution methods using ordinary MATLAB scripting.

1.1. Features

The Vibrata user interface offers extensive capabilities for defining complicated dynamic analyses. At the same time, it provides useful defaults that will be acceptable in many cases, thus making it easy to define basic analyses as well. Vibrata features include:

- Interactively select and view the FEM and mode shapes, list natural frequency and modal effective mass, and assign modal damping values.
- Add enforced motion and static correction data (constraint modes, attachment modes, residual vectors) when available from the modes solve.
- Define type of analysis and solution range and resolution.
- Define excitation functions interactively, or import them from test results or other data sources.
- Solve for modal domain responses.
- Interactively select physical responses to recover.

- Recover and store physical responses at any or all physical degrees of freedom (DOF) as XY functions or field contours.
- Manage the input environments, event definitions, and physical responses.
- Run in interactive mode for defining and solving dynamic events. Run in batch mode for solving events that are already completely defined.

If you want to get started quickly, you may want to look at the example problems (section 4), before you delve into the User Guide (section 3), which covers everything in detail. However, you *must* familiarize yourself with Nastran input file requirements defined in section 2.1.

1.2. Architecture

Vibrata has been developed around Femap and MATLAB. Its primary user interface, called the Dynamic Event Manager, is a separate process that starts and drives its own Femap session and its own solver session (either a MATLAB Runtime compiled solver—the default—or directly in a MATLAB session) to act as its "FEM server" and "Solver server" respectively. If there is already a Femap session running on your computer, Vibrata connects to that instead of starting another one. If Vibrata starts its own Femap session, it will close that session when it exits, and any unsaved changes you have in that Femap session (even in other MODFEMs you opened) will be lost. If Vibrata connects to an already-running Femap session, when it closes it will close any MODFEMs that it opened, and it will leave the Femap session running.

Vibrata will always start its own solver session. By default, it will start a compiled MATLAB Runtime-based solver that does not require a MATLAB license, but it can also start its own MATLAB session. The latter is necessary if you want to use custom solvers. The solver session is the computational engine that solves the modal dynamics algorithms, and it is also the basis of the Function Manager and XY-Plot user interfaces that create, manage, and display excitation and response functions. Figure 1-1 shows a schematic of the Vibrata components. The separate processes communicate with each other through COM interfaces.

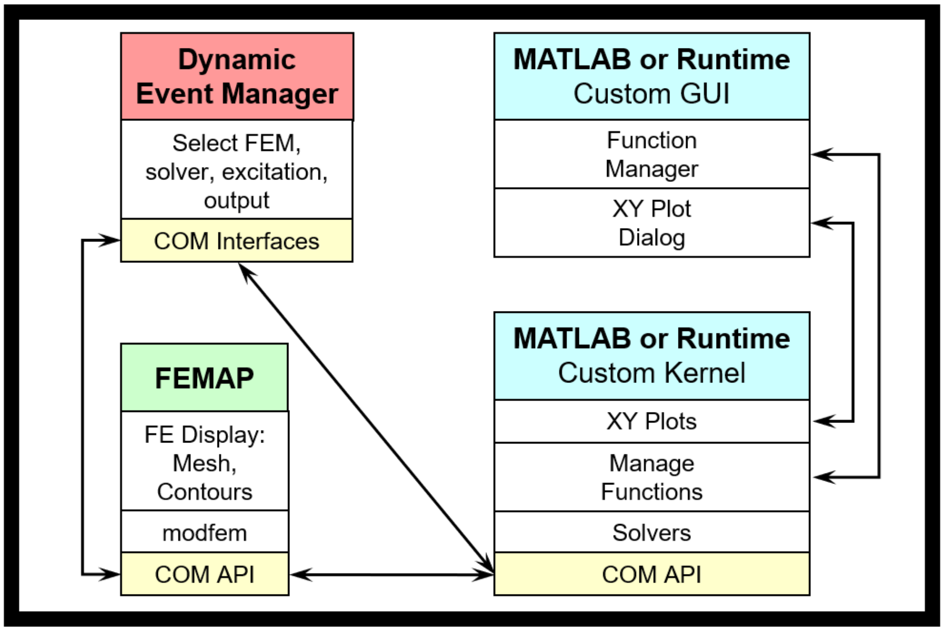


Figure 1-1. Overview of Vibrata components.

Vibrata includes a special version of IMAT, the "Interface between MATLAB, Analysis, and Test" MATLAB toolkit developed by ATA. IMAT enables much of the graphical, computational, and file import and export capabilities used by Vibrata. Femap and MATLAB, however, are not included with Vibrata; you must obtain them separately from Siemens Product Lifecycle Management Software (http://siemens.com/plm/femap) and The MathWorks, Inc. (http://www.mathworks.com) respectively. Please note that if you use the default MATLAB Runtime-based solver, you do not need MATLAB. However, you do need the appropriate MATLAB Runtime software, which is a separate free download available from ATA's software website (https://www.ata-e.com/software/ata-software/matlab-runtime/) or from the MathWorks website (https://www.mathworks.com/products/compiler/matlab-runtime.html).

1.3. Files

Model information and results are stored in five main files.

1. Femap Model file (.modfem). The FEM and its modal results reside in a Femap model file. Vibrata writes its contour results into that same file.
2. Function file (.fcn). Forcing functions reside in a Function file, which is a MATLAB file with a specific format created and managed by IMAT.
3. XY-plot results file (.vra_xyout). The XY-plot results (functions of time or frequency) are stored in a MATLAB .mat-formatted file whose format is understood by Vibrata's plotting and function management tools.

4. Event Definition file (.evt). All of the data that define a dynamic event, including excitations, modal damping parameters, requests for specific physical responses, which solver to use, and the names of the referenced .fcn, .modfem, and .vra_xyout files, are stored in Vibrata's Event Definition or EVT file. This is a text file, which means that you can edit it for re-use.
5. Vibrata HDF5 database (.vra5). The Nastran Output2 file contents needed by Vibrata are translated into this file and then used by the solvers.

1.4. Note on the "Global Coordinate System"

This manual, like Femap's documentation, uses the term "global coordinate system" interchangeably with "basic coordinate system." It refers to Femap's "Coordinate System 0," which is known as both the "Global Rectangular" system and the "Basic Rectangular" system. It is a single coordinate system, not an agglomeration of many separate systems. Nastran users are generally used to saying "basic coordinate system" when they mean "global coordinate system" and "global coordinate system" when they mean "the nodal displacement coordinate system at each node." In this manual, if we want to refer to the "nodal displacement coordinate system," we will *say* "the nodal displacement coordinate system"; the "global coordinate system" is *always* the same as Nastran's "basic coordinate system."

----

2. PREPARING NASTRAN INPUT FILES

Vibrata gets the data it needs from two sources: a Nastran OP2 file and a Femap MODFEM file. Many of the analyses Vibrata offers require the presence of specific data that Nastran does not output by default. This section describes how to prepare Nastran input files to ensure that they generate all the results Vibrata needs. If you want to use a different finite element (FE) solver (e.g., MSC Nastran), you will have to work out how to make it output the equivalent data to the OP2 file. See Appendix A for more details.

For the most part, Vibrata imposes very few and very easy-to-satisfy, but important, requirements on your Nastran decks. Section 2.1 defines the Case Control, Parameter, and other bulk data definitions you must include in your decks so that Vibrata can use those results. Following that, section 2.2 defines the Nastran versions and solution types that Vibrata supports.

Vibrata provides a useful GUI to facilitate setting up your Nastran deck within Femap. Its usage is discussed in section 2.1.8, but please read through section 2.1 first to make sure you understand what modifications this GUI makes.

2.1. Nastran Cards for Specific Vibrata Options

This section defines the executive control, case control, parameter, and other bulk data cards you must include in your Nastran decks to generate the data Vibrata needs in order to perform certain types of analysis. The following topics are covered:

- 2.1.1 General Requirements for Physical Response Recovery.
- 2.1.2 Load Set Excitation.
- 2.1.3 Enforced Motion.
- 2.1.4 Coupled Damping Matrix.
- 2.1.5 Residual Modes.
- 2.1.6 Modal Effective Mass Output.
- 2.1.8 Vibrata Nastran Deck Setup Graphical Utility.
- 2.1.9 Creating the Vibrata Database (.vra5).
- 2.1.10 Coordinate System Considerations.
- 2.1.11 Supported Nastran Entities.
- 2.1.12 Using MSC Nastran with Vibrata.

The referenced paragraphs show the actual bulk data file entries, not the details of how to make Femap write those entries into the decks.

2.1.1. General Requirements for Physical Response Recovery.

Since Vibrata uses a modal postprocessing approach, it needs mode shape data to process. As a result, Vibrata expects results generated by Nastran's normal modes analysis solution sequence (SOL 103, SEMODES).

By default, Femap will select a flag to have Nastran normalize the mode shapes to produce a unit modal mass for each mode. This is called "mass normalization." You must use this option. Vibrata can only use mass-normalized modes; it will not allow you to use modes that have been normalized any other way.

In addition, you must request output as part of the modes solution for any result types you will want Vibrata to produce. For example, if you plan to ask Vibrata to compute dynamic stress responses or bar, beam, or spring force responses, then you must request output to the OP2 file for them as part of your SOL 103 analysis. Of course, Vibrata can only recover physical responses for those nodes or elements that were included in your output requests. Thus, if your *DISPLACEMENT (PLOT)* output request specifies a subset of the nodes in your model rather than *ALL*, Vibrata will only be able to compute dynamic displacements, velocities, and accelerations for the nodes in the specified subset. This also has implications for applying point force excitations in your dynamic analyses. Your normal modes analysis must request displacement output for every node to which you intend to apply a point force; if the modal displacements are not present, Vibrata will not be able to transform the forces into the modal domain.

The POST parameter specifies the coordinate system(s) in which displacement results are written to the Nastran Output2 file. Having the displacement quantities in the displacement coordinate system is critical for enforced motion analyses. There are two options that cause Nastran to write displacement results in the displacement coordinate system. The first option is to specify PARAM,POST,-1, which is the option supported by Femap. If you are using MSC Nastran with PARAM,POST,-1, you must also add PARAM,POSTEXT,YES. The second option is to use PARAM,POST,-2, but if you do so, you must also use PARAM,OUGCORD,GLOBAL. Finally, any time Vibrata needs to perform coordinate transformations, both when converting OP2 results to the VRA5 file and also when computing output requests, coordinate system and node information must be present in the OP2 file. The PARAM,OGEOM,YES statement enforces this requirement. It is not necessary to include this PARAM, since "YES" is the default, but you must *not* use PARAM,OGEOM,NO.

2.1.2. Load Set Excitation.

If you want to use a time- or frequency-dependent scaling function to multiply the forces in a static load set, you must include the highlighted statements in Figure 2-1. You can use any combination of point forces, distributed loads, and body forces. Note the presence of the LSEQ entries. Each of these refers to FORCE*i*, PLOAD*i*, GRAV, or other static load cards whose set IDs (SIDs) correspond to the load ID (LID) in field 4 on an LSEQ card. The static load cards define the actual physical loads applied to the model, and obviously, they must also be present in the file. The LSEQ cards simply tell Nastran that these are the static loads to include in the generalized force matrix it writes to the OP2 file. You can see in this example that several LSEQ cards reference static load cards with different set IDs (SIDs). All the LSEQ cards that have a LOADSET ID (field 2) equal to the LOADSET ID specified in case control will be available as separate generalized forces in Vibrata. Generalized forces defined on LSEQ cards that reference a different LOADSET ID will not be available. We highly recommend that you use the same ID for the EXCITEID (field 3) and LID (field 4) on each LSEQ card.

It is important to understand how Vibrata interrogates the FEM and the database for the presence of load sets. Vibrata first checks the database for the presence of load sets. These load sets are identified by the load ID (LID), which is field 4 on the LSEQ card. If any load sets are present, Vibrata then queries Femap for the presence of all the LIDs in the Model, Loads definition. If any of these loads are not defined, Vibrata assumes that the load sets are not related to the current FEM and issues a warning message to the user in Femap's Message window and disables load set selection.

Note that in Figure 2-1, there are no dynamic excitation cards (e.g., TLOAD*i*, RLOAD*i*) in the deck. You must NOT include such dynamic excitation cards in your deck, as they may prevent Vibrata from using your generalized forces. These cards are not needed anyway, because the dynamic excitations of the static loads will be defined in the Vibrata environment.

```
$* EXECUTIVE CONTROL
$*
SOL 103
CEND
$*
$* CASE CONTROL
$*
TITLE = motor_rs_modes
SPC = 1
LOADSET = 101
METHOD = 10
DISPLACEMENT(PLOT,REAL) = ALL
$*
BEGIN BULK
$
PARAM,GRDPNT,0
PARAM,K6ROT,100.0
PARAM,POST,-1
$ Additional params for Vibrata load set excitation
PARAM,RSOPT,1
PARAM,OGEOM,YES
$
$             sid      v1      v2      nd
EIGRL          10                      10
. . .
$        -- LOADSET ID from LOADSET card in case control
$       /  -- Excitation ID (EXCITEID) for TLOADi, RLOADi, etc, cards (not shown)
$      /  /  -- Set ID (LID) for FORCEi, PLOADi, etc, cards (not shown)
$     /  /  /
LSEQ,101,1,1
LSEQ,101,2,2
LSEQ,101,3,3
```

Figure 2-1. Nastran cards to enable load set excitation.

2.1.3. Enforced Motion.

If you want to perform a frequency response, random, or transient analysis using enforced motion excitation, you must include the highlighted statements in Figure 2-2. These statements will also enable response spectrum analysis, although section 2.1.6 describes a more direct approach for that. Taken together, the specified cards tell Nastran to compute constraint modes for the degrees of freedom specified on the USET U2 card(s), and to write the U2 DOF to the OP2 file so Vibrata can find them. As usual, all DOF listed in the U2 set must also be constrained; that is, they must appear on an SPC1 card elsewhere in the bulk data. Vibrata requires geometry (specifically the GEOM4 datablock) in the OP2 file so that it can determine the U2 DOF when it first sets up the model. The PARAM,OGEOM,YES statement enforces this requirement. It is not necessary to include this PARAM, since “YES” is the default, but you must *not* use PARAM,OGEOM,NO. Finally, note that Nastran writes the Constraint modes, and Vibrata expects to read them, using the nodal displacement coordinate systems, *not* the basic system. Therefore, for enforced motion analyses that depend on Constraint modes, Vibrata always takes the prescribed motion to be in the driven node's displacement coordinate system. Response Spectrum analysis is handled differently, as described below.

The RSOPT and RSCON PARAM cards are not available in Femap; you must either export the deck and edit it or add these cards as text. These cards are recognized by Simcenter (NX) Nastran only. If you want to use some other Nastran, you will have to use a DMAP alter to generate the required output. See section 7.6 to read about the additional data required.

```
$
BEGIN BULK
$
PARAM,GRDPNT,0
PARAM,POST,-1
$ Additional/modified params for Vibrata enforced motion
PARAM,RSOPT,1
PARAM,RSCON,YES
PARAM,OGEOM,YES
$
$ DOF available for enforced motion
USET          U2      44     123      45     123      48     123
USET          U2      49     123
$
$ first 10 modes
$            sid      v1      v2      nd
EIGRL         10                      10
$
```

Figure 2-2. Include a U2 USET and special PARAMs to enable enforced motion excitation.

The example in Figure 2-2 enables enforced motion in all three translations at four nodes. For frequency response, random, and transient analyses, all twelve of these DOF will appear in the enforced motion definition dialog (paragraph 3.2.6.4), and you can apply independent excitations to all of them. Response spectrum analysis, however, is based on modal effective masses rather than constraint modes, so its motion is in the basic coordinate system relative to the single point at which the modal effective mass was calculated (see section 2.1.6).

The deck shown in Figure 2-2 does not require a DMAP alter, nor therefore a DMAP license. If you have a DMAP license and are running NX Nastran 7 or earlier, and you prefer to keep your decks as consistent as possible for Vibrata analyses, you can use the "vibrata.nx7" alter mentioned in paragraph 2.1.2, as shown in Figure 2-3. In that case, you can use the Femap standard PARAM,POST,-1 card, and leave out the PARAM,OUGCORD card. Otherwise, the decks are identical.

```
$* EXECUTIVE CONTROL
$*
SOL 103
INCLUDE c:\apps\Vibrata\vibrata.nx7
CEND
$
BEGIN BULK
$
PARAM,GRDPNT,0
PARAM,POST,-1
$ Additional/modified params for Vibrata enforced motion
PARAM,RSOPT,1
PARAM,RSCON,YES
$
$ DOF available for enforced motion
USET           U2      44     123      45      123      48      123
USET           U2      49     123
$
$ first 10 modes
$             sid      v1      v2      nd
EIGRL          10                      10
$
```

Figure 2-3. Alternate deck for enforced motion using DMAP alter.

2.1.4. Coupled Damping Matrix.

Nastran will write the full (coupled) modal damping matrix to the OP2 file if you include the PARAM,RSOPT card, as highlighted in Figure 2-4. This is the same alter used to enable load set excitation (paragraph 2.1.2). The alter does *not* cause Nastran to compute complex modes. It uses the undamped mode shapes to compute $[B_{HH}] = [\Phi]^T[B][\Phi]$, where [B] is the physical viscous damping matrix and $[B_{HH}]$ is the (nModes x nModes) modal viscous damping matrix. It will also write the modal structural damping matrix $[K_{HH}]$ or $[K4_{HH}]$ (the imaginary part of the modal stiffness matrix) if you have defined any material damping. Of course, if your model does not include any damping elements or materials with damping specified, there will be no $[B_{HH}]$, $[K_{HH}]$, or $[K4_{HH}]$ matrix to write. The matrices are stored in the OP2 file in datablocks BHH, KHH, and/or K4HH. Do *not* use the datablocks called RADAMPZ and RADAMPG; they do not contain the data that Vibrata requires.

```
$* EXECUTIVE CONTROL
$*
SOL 103
CEND
$*
BEGIN BULK
$
PARAM,GRDPNT,0
PARAM,K6ROT,100.0
PARAM,POST,-1
$ Additional param for coupled damping matrices
PARAM,RSOPT,1
$
```

Figure 2-4. Include PARAM,RSOPT,1 to get coupled modal damping matrices.

2.1.5. Residual Modes.

Vibrata does not support mode acceleration data recovery. Therefore, if you require static corrections for your modal dynamic analyses, you must tell Nastran to compute and store residual vectors.

When you have applied forces, the most effective approach is to use residual modes in conjunction with load set excitation. This means you need to define actual forces (even if they are unit forces) in the Nastran deck and compute the generalized force matrix for them, as shown in Figure 2-5. The only change from Figure 2-1 is the addition of the RESVEC case control statement.

```
$* EXECUTIVE CONTROL
$*
SOL 103
CEND
$*
$* CASE CONTROL
$*
TITLE = motor_rs_modes
SPC = 1
LOADSET = 101
METHOD = 10
DISPLACEMENT(PLOT,REAL) = ALL
$
RESVEC = YES
$
BEGIN BULK
$
PARAM,GRDPNT,0
PARAM,K6ROT,100.0
PARAM,POST,-1
$ Additional params for Vibrata load set excitation
PARAM,RSOPT,1
PARAM,OGEOM,YES
$
$             sid      v1      v2      nd
EIGRL          10                      10
. . .
$        -- LOADSET ID from LOADSET card in case control
$       /    -- Set ID for FORCEi, PLOADi, etc, cards (not shown)
$      /    /
LSEQ,101,1,1
LSEQ,101,2,2
LSEQ,101,3,3
```

Figure 2-5. Add a RESVEC case control statement to get residual modes for applied forces.

You can get residual modes for unit forces at designated DOF using USET U6 cards as shown in Figure 2-6, but these forces will not appear in the generalized force matrix and so cannot be used as a load set excitation.

```
$
RESVEC = YES
$
BEGIN BULK
$
PARAM,GRDPNT,0
PARAM,POST,-1
$
$ DOF for unit loads for residual modes
USET          U6    3693      12    3968      12    4243      12
USET          U6    4518      12
$
$ first 10 modes
$             sid      v1      v2      nd
```

Figure 2-6. You can get residual modes by defining a U6 USET, without any actual loads.

If you want residual modes for enforced motion excitation, you will need the inertial load residual vectors. These are computed by default when the RESVEC case control statement is used. If you are using PARAM,RESVEC instead of specifying this in case control, you will need a RESVINER card instead (or in addition to it, if you are applying loads in addition to base excitation) of RESVEC. This will create residual modes for unit accelerations of the model in all six directions. The required cards are shown in Figure 2-7. The only change from Figure 2-2 is the addition of the PARAM,RESVINER card.

```
$
BEGIN BULK
$
PARAM,GRDPNT,0
PARAM,POST,-1
$ Additional/modified params for Vibrata enforced motion
PARAM,RSOPT,1
PARAM,RSCON,YES
PARAM,RESVINER,YES
$
$ DOF available for enforced motion
USET          U2      44     123      45     123      48     123
USET          U2      49     123
$
$ first 10 modes
$             sid      v1      v2      nd
EIGRL          10                      10
$
```

Figure 2-7. Add “PARAM,RESVINER” card to get residual modes for enforced motion.

2.1.6. Modal Effective Mass Output.

Modal effective mass information is required for response spectrum analyses. Nastran will generate it automatically when you set up to recover the enforced motion data (section 2.1.3), but that may not be exactly what you want. Nastran will not generate the modal effective mass information on a restart. If you simply take those defaults, the effective mass will be computed relative to the origin of the basic coordinate system. While that will make no difference to the translational mass, it will affect the rotational mass, and that may be important if you mean to excite rotations in a response spectrum analysis. In that case, you may need to specify the node about which you want the rotational masses calculated, which will then be the rotational center of your response spectrum excitation. This can be done by including a MEFFMASS case control card, as in the highlighted statement in Figure 2-8. Modal effective mass is generally not required for analyses that do not use enforced motions, but you are welcome to request it if you want to see this data on the Modal Settings tab.

```
$*  CASE CONTROL
$*
TITLE    = Free Modes Small
ECHO     = NONE
$*
DISPLACEMENT(PLOT) = ALL
STRESS(SORT1,PLOT,FIBER,CORNER) = ALL
FORCE(SORT1,PLOT,CORNER) = ALL
MEFFMASS(NOPRINT,PLOT,GRID=6000,MEFFM) = YES
METHOD = 1
$*
```

Figure 2-8. Request for modal effective mass output relative to a specific node.

Vibrata supports Nastran models that have discarded modes using the MEFFMASS THRESH parameter.

2.1.7. Composite Failure Indices.

Vibrata can compute composite failure indices in modal transient solutions. The failure theories Vibrata supports are listed in Table 2-1. Other failure theories, such as maximum strain, can be implemented as user-defined postprocessing utilities (see section 3.2.11). Strength ratios can also be computed as a postprocessing step.

Table 2-1. Supported failure theories.

Failure Theory Type	**Supported Failure Theories**
Fiber failure theories	Hill (HILL), Hoffman (HOFF), Tsai-Wu (TSAI)
Bond failure theories	Interlaminar shear (SB), Interlaminar normal (NB)

Vibrata requires that material stress limits be present in the OP2 file for those elements on which failure indices will be calculated. This means that the stress limits must be set on the appropriate material and property cards in the Nastran deck, and that PARAM,OGEOM,NO must not be used. Additionally, the failure theory or theories to be used for the calculation must be specified in the Nastran deck. Please refer to the *Nastran Quick Reference Guide* and the *Nastran User's Guide* for details on supplying this data. When Vibrata creates a new VRA5 file from an OP2 file (see section 2.1.9), it looks for material stress limits and failure theories. If it finds them, it stores them in a lookup table in the VRA5 file for later use. If any stress limits are incorrectly or incompletely defined for the requested failure theory/theories, a warning message is sent to the Femap message window. If there are no correct material stress limits for fiber or bond failure theories in the OP2 file, Vibrata will disable the respective failure indices' output requests. Also, be aware that if Vibrata does not find any out-of-plane stress limits (limits in the zz, xz, and yz directions) for a material, it will perform the failure theory calculations for those elements as though they are 2D, even for 3D elements.

Note that Vibrata has one more requirement than Nastran does for calculating bond failure indices on solid laminate elements: stresses must be output at the top and bottom of each ply, not just at the mid-ply. This means that either the CPLYBT or CPLYBMT option must be chosen on the STRESS case control command that requests stresses on solid laminate elements. Vibrata always calculates bond failure using the top and bottom ply stresses for solid laminates. For shell laminates, Nastran always computes the in-plane stresses and approximates the out-of-plane shear stress components at the mid-ply. Thus, for shell laminates, Vibrata will always compute bond failure according to these mid-ply stress approximations. Note that such approximations can lead to inaccurate bond failure results for thick plies.

2.1.8. Vibrata Nastran Deck Setup Graphical Utility.

Even though the Nastran deck setup requirements for Vibrata are modest, it is always easier to have a utility that sets up the deck for you. Vibrata ships with a simple GUI that runs in Femap. It configures an analysis set with the settings described in the previous sections.

The GUI, *Analysis Setup*, is accessible from the Custom Tools menu in Femap, as seen in Figure 2-9. The installer places the executable file in the `api` folder in the installation directory of the version of Femap most recently used prior to running the Vibrata installer. You can make it accessible to other versions of Femap by copying it from this folder to the `api` folder in the other Femap installation.

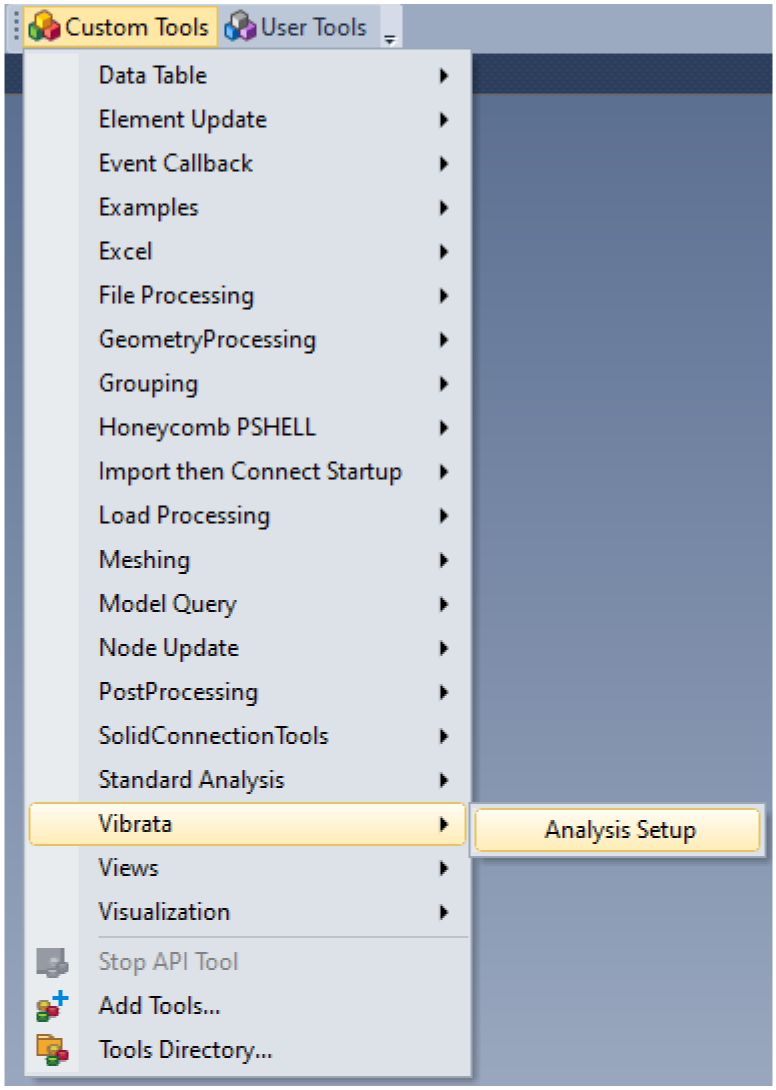


Figure 2-9. Access the *Analysis Setup* GUI through the Custom Tools menu in Femap.

Figure 2-10 shows the GUI when it is first launched. To use the GUI, first enter an analysis title. A pulldown list is available to select an existing analysis set, or you can create a new analysis by typing a name not already present. If you select an existing set, the **Create Analysis Set** button changes to **Modify Analysis Set,** and the checkboxes and pulldown list will autofill based on the values already present in that set.

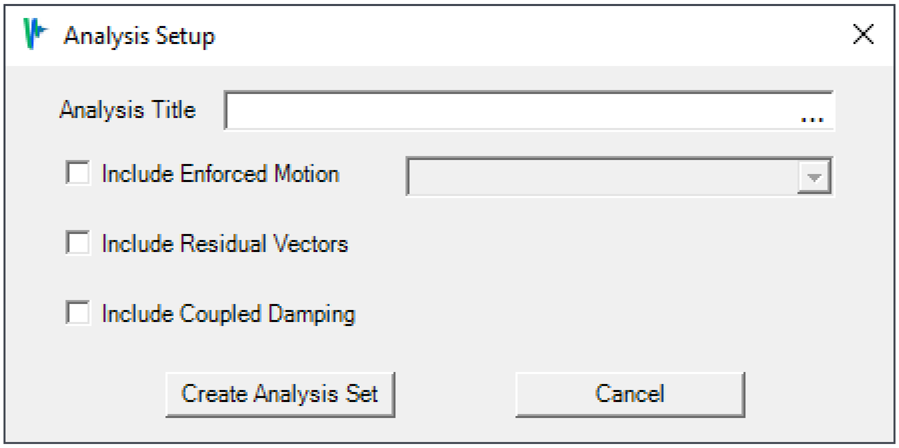


Figure 2-10. *Analysis Setup* GUI.

Regardless of the boxes checked, the GUI sets the solver to *NX Nastran*, the analysis type to *Normal Modes/Eigenvalue*, and the *Analysis Title* to what was entered in the GUI. It also sets the normalization method to *Mass Normalized* and turns on *Modal Effective Mass* output. It also toggles OGEOM on. Finally, it sets the output path of the analysis to the location of the MODFEM if it is known.

Toggle on **Include Enforced Motion** to configure Nastran to generate the output needed for base excitation analysis in Vibrata. After toggling it on, select an existing constraint set. This set specifies the DOF (SPC) to which base excitation may be applied. This option adds custom *Start Text* in *Manual Controls* to the Nastran Bulk Data Options. This custom text sets the parameters *RSOPT* to *1*, *RSCON* to *yes*, and *POSTEXT* to *yes*. It also sets the constraint set from the pulldown menu to be the active constraint and creates USET cards in the custom text.

Toggle on **Include Residual Vectors** to include residual vectors in the analysis. This option toggles on the *RESVEC* parameter. If the Enforced Motion toggle is also checked, the *RESVINER* parameter is also turned on.

Toggle on **Include Coupled Damping** to direct Nastran to create a fully populated damping matrix. The Nastran default is to generate a diagonal matrix, which can potentially neglect significant damping effects. This option also sets the parameter RSOPT to 1 in the custom *Start Text* in the *NASTRAN Bulk Data Options*.

After making your selections, press the **Create Analysis Set** (or **Modify Analysis Set**) button to apply these changes. If you press **Cancel**, none of the selections will be applied. If the GUI modifies an existing analysis set, it is careful to retain any existing custom start text. However, any custom text that could conflict with the text being added is commented out and left in the custom text section. All new text is placed between comment lines indicating that it was created by the GUI for Vibrata. Text between these comment lines is deleted if the GUI modifies this analysis set in the future, so you should not manually add anything between these lines.

Once you are finished with the GUI, you may still need to edit the analysis set to complete your output requests and any other settings you need that are not Vibrata-specific.

2.1.9. Creating the Vibrata Database (.vra5).

To achieve better performance and reduce memory usage, Vibrata uses its own database file with .vra5 extension to import mode shapes and other necessary quantities during a solve. It translates the Nastran OP2 file to this database. Vibrata will create this database file automatically when you create a new event (see section 3.2.3), but in some cases you may wish to create the .vra5 ahead of time. For instance, you may want to translate OP2 files in a batch process in cases where the translation may take some time because the OP2 file is large.

Vibrata offers four ways to create the .vra5 file explicitly.

1. In the Vibrata GUI menu, select File → Convert OP2 to run the translation utility.
2. Access the translation utility from the Start menu as shown in Figure 2-11.
3. Invoke the translation utility without any graphical interaction by launching it from a command prompt and optionally supplying the OP2 name as an input argument, as shown in Figure 2-12.
4. Invoke the translation utility via MATLAB by launching it from the MATLAB prompt (assuming the Vibrata path is set correctly in your MATLAB path), again optionally supplying the OP2 name as an input argument, as shown in Figure 2-13.

In all cases, if no OP2 file is supplied, you will be prompted to select an OP2 file using the graphical file dialog shown in Figure 2-14. The conversion process will then proceed, and the resulting .vra5 file will be located in the same directory as the OP2 file with the same base name.

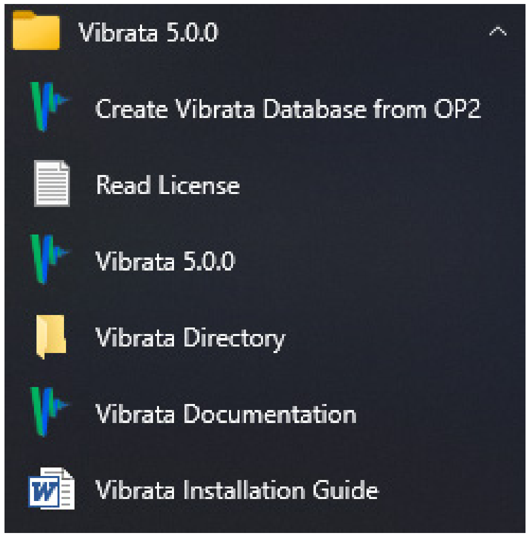


Figure 2-11. Access the .vra5 translation utility from the Start menu.

```
S:\> vraBuildHDF5Database filename.op2
```

Figure 2-12. Create the .vra5 database from OP2 using the command-line interface.

```
>> addpath C:\Apps\Vibrata_v5.2.0\matlab_vra_api
>> addpath C:\Apps\Vibrata_v5.2.0\imat
>> vraBuildHDF5Database                       % Prompt for filename
>> vraBuildHDF5Database('filename.op2')
```

Figure 2-13. Create the .vra5 database from OP2 in MATLAB.

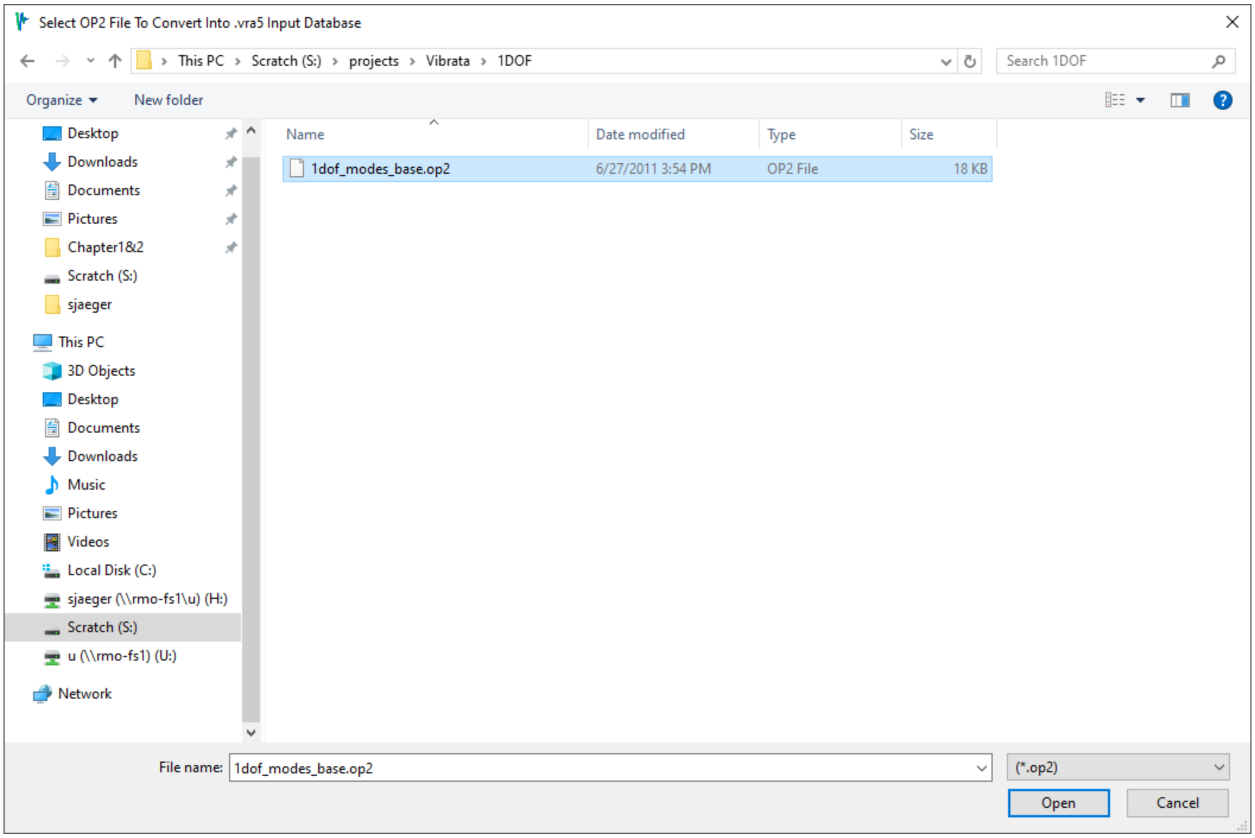


Figure 2-14. Select the OP2 file to translate using a graphical file dialog.

2.1.10. Coordinate System Considerations.

Vibrata does not perform any coordinate system transformations, so all input degrees of freedom and output results are defined/reported as they are provided in the Nastran OP2 file. This also means that enforced motion excitations are always defined in the drive node's displacement coordinate system.

2.1.11. Supported Nastran Entities.

Table 2-2 summarizes the Nastran enforced motion datablocks that Vibrata supports. Table 2-3 contains a summary of the supported element types along with any noteworthy details of their translation.

Table 2-2. Supported NX/Simcenter Nastran enforced motion datablocks.

Result Type	**Nastran Datablocks**	**Data Type**
Constraint Mode	RADCONS	Displacement
Constraint Mode	RARCONS	Reaction Force
Constraint Mode	RAFCONS	Beam Force and Shell Stress Resultants
Constraint Mode	RASCONS	Stress
Constraint Mode	RAECONS	Strain
Constraint Mode	RANCONS	Strain Energy and Strain Energy Density
Constraint Mode	RAGCONS	Grid Point Force
Modal Effective Inertia Matrix	RADEFMP	Modal Matrix
Load Set Modal Forces	RAFGEN	Modal generalized force vectors

Table 2-3. Nastran element types supported by the Vibrata translator.

Element Type	**Relevant Notes**
CBAR	Forces and moments; stress and strain at CL and stress recovery points C, D, E, F. Please see the *Nastran Quick Reference Guide* for more details, as Nastran writes out different stress components for CBAR and CBEAM.
CBEAM CBEND	* See CBAR.
CBUSH CELAS	Forces and moments.
CTETRA CPENTA CHEXA	Stress/strain results are stored in the coordinate system specified by the PSOLID card.
CQUAD4	Nodal stress/strain results are only available for CQUAD4 elements if the CORNER option is used in the output request. If the CORNER option is specified, both nodal and centroidal results are available. Stress/strain results are left in the coordinate system(s) in which the OP2 results were written (usually the element coordinate system). See PCOMP notes for details about composite element translation. Shell stress resultants (Nastran Element forces, Femap Section Forces) are transformed to the material orientation angle specified for that element.
CQUAD8	* See CQUAD4.
CQUADR CTRIAR	* See CQUAD4. Composite ply results are not supported (Nastran limitation).
CROD CONROD CTUBE	Stress/strain is stored in the Femap S11 component.
CTRIA3 CTRIA6	* See CQUAD4.
PCOMP	Complex results are not supported (Nastran limitation).

2.1.12. Using MSC Nastran with Vibrata.

MSC Nastran requires some special setup to generate the datablocks necessary for Vibrata. If you are using PARAM,POST,-1, you must also use PARAM,POSTEXT,YES, as shown in Figure 2-15. These parameters are necessary for MSC Nastran to write out the datablocks needed by Vibrata.

```
BEGIN BULK
$
PARAM,GRDPNT,0
PARAM,K6ROT,100.0
PARAM,POST,-1
PARAM,POSTEXT,YES
$
```

Figure 2-15. Use these parameters to write out the LAMA datablock in MSC Nastran.

MSC Nastran does not natively provide functionality to generate the constraint modes required by Vibrata for base excitation analysis. To enable this capability, a Nastran alter that provides that functionality is available in the root directory of the Vibrata installation. After you set up your deck for base excitation from the instructions provided in section 2.1.3, simply include the alter in the executive control of your MSC Nastran bulk data deck, as shown in Figure 2-16. You will need a DMAP license to use this alter. The alter has been tested with MSC Nastran 2021.3 through 2024.1 and may work with newer versions of Nastran.

```
$* EXECUTIVE CONTROL
$*
SOL 103
INCLUDE c:\apps\Vibrata_v5.2.0\vibrata_msc.dmap
CEND
$
```

Figure 2-16. Insert a DMAP alter to use MSC Nastran with Vibrata base excitation analysis.

Please note that MSC Nastran support has some limitations: solid composites are not supported, and other limitations may also exist.

2.2. Supported Nastran Versions and Solution Types

Vibrata supports both MSC and Simcenter (NX) Nastran, although Simcenter Nastran is the preferred solver. In most cases, any version of Nastran is acceptable. As discussed in the preceding sections, however, some capabilities such as enforced motion response (and thus response spectrum analysis) and load set excitation may require a custom DMAP for some Nastran versions.

If you are using a newer version of Nastran and you encounter problems with Vibrata, please let ATA know via the TIER system (https://tier.ata-e.com). ATA will not provide support for enforced motion and load set excitation for MSC Nastran unless requested to do so by users. If you would like to make that request, you can also do that through the TIER system. Of course, a skilled DMAP programmer can make MSC Nastran produce the required data just as ATA has done for Simcenter (NX) Nastran, and Vibrata will use it as long as it is stored in the OP2 file according to Vibrata’s expectations.

The current version of Vibrata does not support upstream data recovery for superelements.

----

3. VIBRATA USER GUIDE

3.1. Starting Vibrata

3.2. Event Manager—Interactive User Interface

3.2.1. General Characteristics of the User Interface.

3.2.2. Event List Region.

3.2.3. FEM/Input Database Selection Region.

3.2.4. Solver Tab.

3.2.5. Modeling Nonlinearities in the Transient Solver

3.2.6. Excitations Tab.

3.2.7. Modal Settings Tab.

3.2.8. Defining Modal Damping.

3.2.9. Output Request Tab.

3.2.10. Contour Output Requests.

3.2.11. Post Processing Tab.

3.3. Event Manager—Batch Solution Mode.

3.3.1. Batch Processing with MATLAB Scripts.

3.4. Function Manager

3.4.1. Functions Region.

3.4.2. Function Sets Region.

3.4.3. Plot Region.

3.4.4. Math Region.

3.4.5. Hints for Creating Complex Functions.

3.4.6. Creating Force-Limited Input.

3.4.7. Applying a Modal (Spatial) Filter.

3.5. XY-Plot Results Display

3.6. Contour Results Display

3.6.1. Viewing Transient Analysis Results.

3.6.2. Viewing Frequency Response Results.

3.6.3. Viewing Random Analysis Results.

3.6.4. Viewing Response Spectrum Results.

3.7. Event Definition File (EVT)

3.7.1. Event File Conventions.

3.7.2. Version Information Section.

3.7.3. Event Summary Section.

3.7.4. Solver Section.

3.7.5. Excitation Section.

3.7.6. Modal Settings Section.

3.7.7. Input Checksum.

3.7.8. Output Request Section.

3.7.9. Post-Processing Section.

3.7.10. Reusing EVT Files.

----

4. EXAMPLE PROBLEMS

The easiest way to learn the basics of Vibrata is to run some analyses using the example models provided. This section introduces four different models, shows how to load the models and their modal results in Femap, and then guides you through a number of analyses using them.

4.1. Descriptions of Example Models

Each of the example models is provided as a Nastran bulk data file. They are found in the *examples* subdirectory of your Vibrata installation. To prepare the models for use with Vibrata, follow the instructions in section 4.1.5.

4.1.1. Single DOF Spring-Mass Model.

The single DOF spring-mass model (Figure 4-1) consists of a spring grounded at one end with a lumped mass attached to the other. SPCs allow the mass to move only in X-direction translation.

The model is located in the *examples\1dof* directory. (In fact, there are two models in that folder, called "1dof_modes_base.dat" and "1dof_modes_fixed.dat." You must always use "1dof_modes_base.dat" with Vibrata, as it meets all the requirements described in section 2.1. The other file is perfectly valid for a typical Nastran SOL 103 analysis, but it will not produce all the data needed for Vibrata; it is provided to illustrate these differences.)

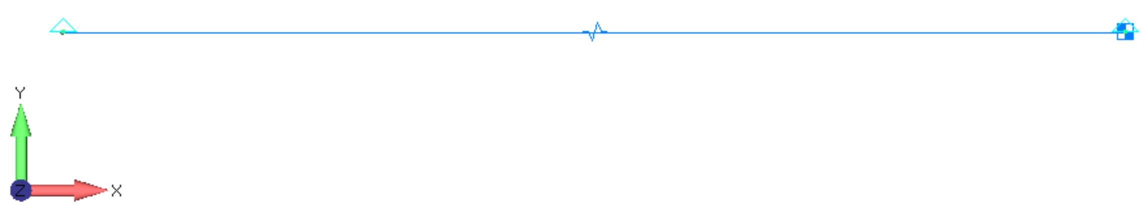


Figure 4-1. Single DOF spring-mass model.

4.1.2. Two DOF Spring-Mass Model.

The two-DOF spring-mass model (Figure 4-2) consists of three nodes connected by two springs in series, with the left node fixed to ground and lumped masses on the other two nodes. The masses are only allowed to translate in the X direction. The two natural frequencies of the system are about 5% apart.

The model is located in the *examples\2dof* directory. (As with the single-DOF example, there are two models present here, called "2dof_modes_base.dat" and

"2dof_modes_fixed.dat." You must always use "2dof_modes_base.dat" with Vibrata, as it meets all the requirements described in section 2.1. The other file is provided to illustrate the differences between a deck that is Vibrata-compatible and one that is not.)

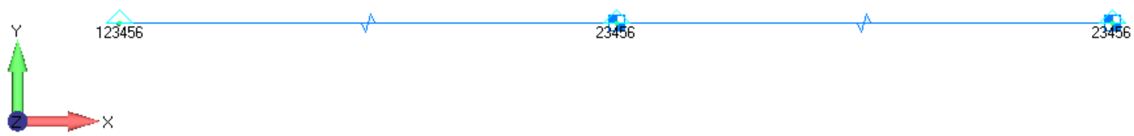


Figure 4-2. Two-DOF spring-mass model.

4.1.3. Two-Dimensional Frame.

The two-dimensional frame model (Figure 4-3) consists of an array of bar elements three columns wide and four rows high. Concentrated mass elements are located at each intersection of bar elements while the bars themselves are massless, thus making this a lumped mass approximation. The bottom node of each outer column is connected by a rigid element to the bottom node of the center column, which is fixed in all six DOF. Enforced motion in X translation is enabled at this node. Out-of-plane (Z) translation and X and Y rotation are constrained at all other nodes.

The Nastran run for this FEM requests residual vectors. Since we are applying a force at node 10, an RVDOF card is defined for the X and Y DOF on this node.

The model is located in the *examples\frame* directory. This folder includes a file with sample excitation functions.

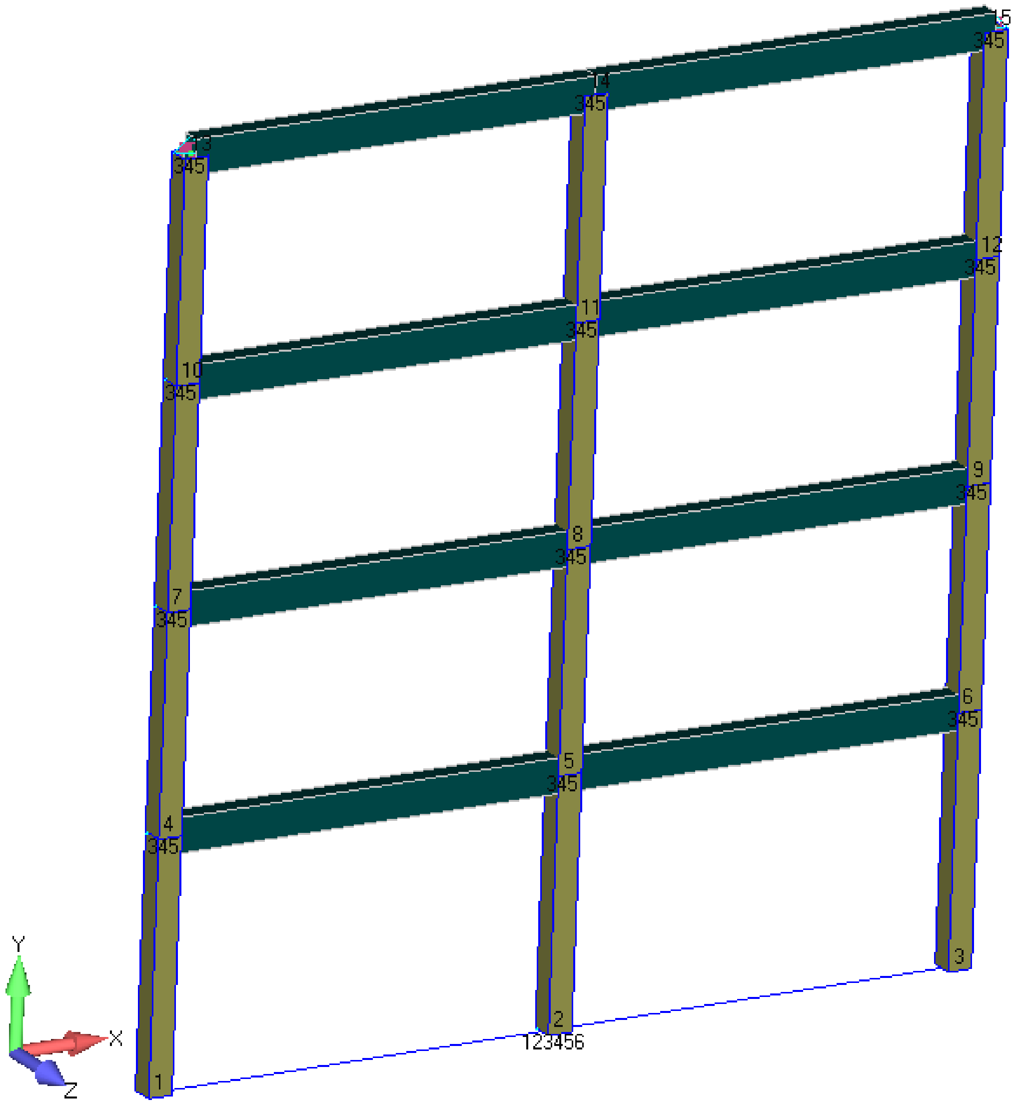


Figure 4-3. Two-dimensional frame model.

4.1.4. <u>Isat—Inner Planets Exploration Satellite.</u>

The Isat, or Inner Planets Exploration Satellite, is a fully developed model of an aerospace structure for which response analysis is a critical part of the qualification phase. The model comes in both “launch” and “deployed” configurations.

There are two launch configurations. In the standard model “Isat_Launch_Sm_4pt.dat” (Figure 4-4), the bottom apex of each launcher adapter leg has all translational DOF restrained, representing a ball-joint connection for each leg. Enforced motion is also enabled for all twelve of these DOF. The other launch configuration has these four points connected by a rigid element to a single central node at which all six DOF are restrained “Isat_Launch_Sm_Rgd.dat” (Figure 4-5). Enforced motion is enabled for the three translations at this node.

The deployed configuration “Isat_Dploy_Sm.dat” (Figure 4-6) is a free-free model with the launcher adapter removed and the solar panels, antenna dishes, and instrument package deployed. This model includes residual vectors that can improve stress calculations for loads applied at the reaction control system (RCS) thrusters.

The models are located in the *examples\Isat* directory. They are shipped in a Zip file called Isat.zip.

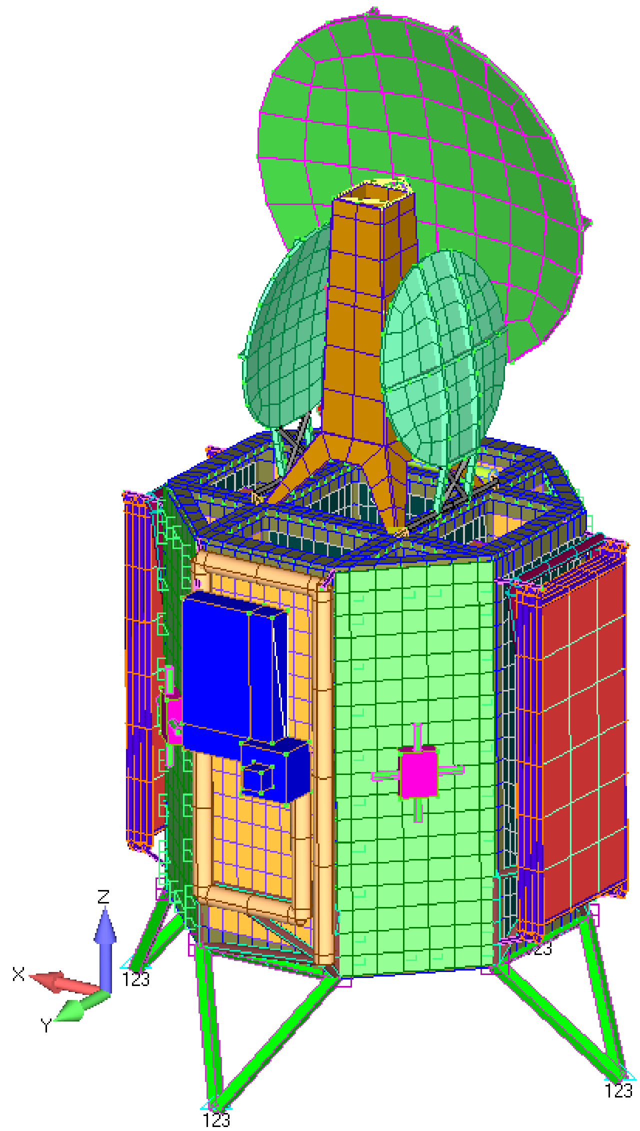


Figure 4-4. Isat model in launch configuration with four separate attachment points.

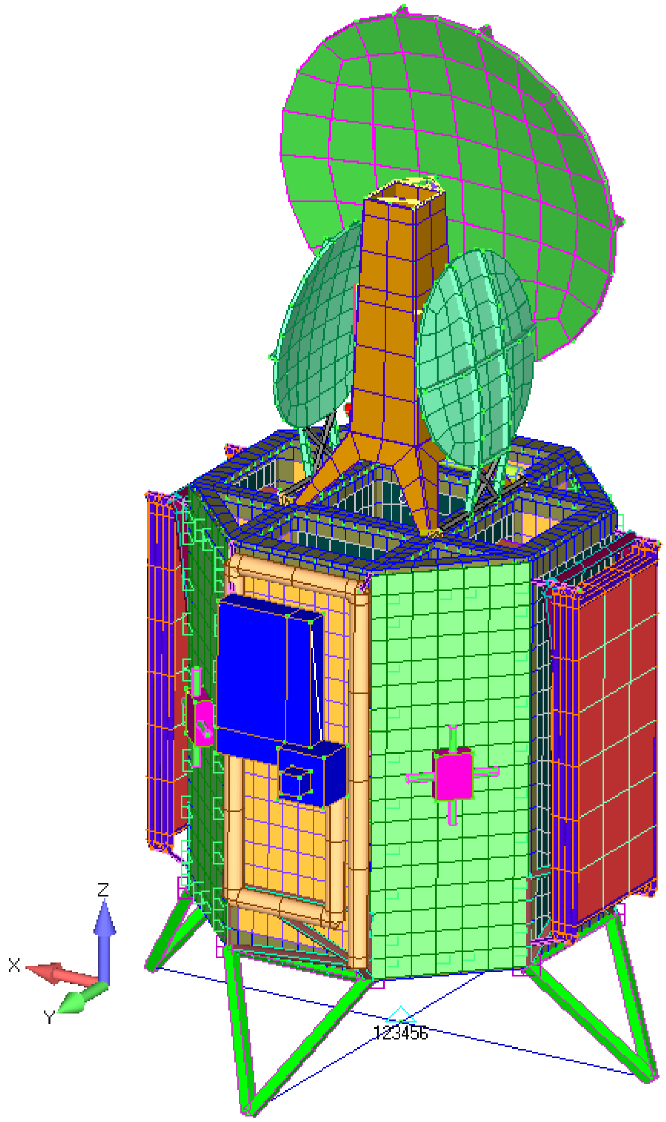


Figure 4-5. Isat model in launch configuration with single central connection to launcher.

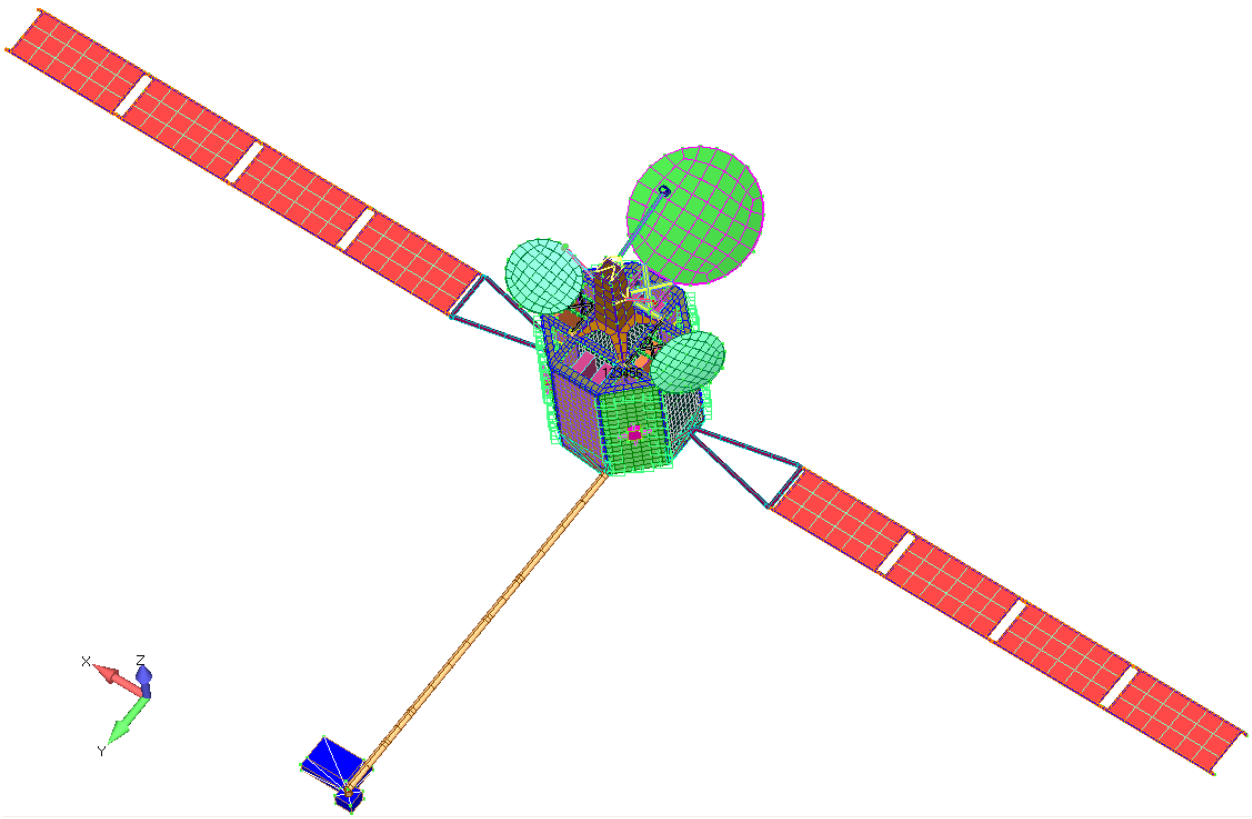


Figure 4-6. Isat model in deployed configuration.

4.1.5. Preparing the Models for Vibrata.

To use the models described in the previous section, copy the contents of the Vibrata *examples* folder to your own local directory and thereafter work only with these local copies. You will note that we have included a results file (Nastran Output2) containing the normal modes solution for each model. You are welcome to use these instead of solving the models yourself, or you may run the models and use the provided Output2 files for comparison to your own results. Either way, it is instructive to examine the input files to see how they satisfy the requirements described in section 2.1.

Import the bulk data into Femap, as shown in Figure 4-7. Where the example folder includes a Femap neutral file with the same name as the .dat and .op2 files, import the neutral file into Femap as well, as shown in Figure 4-8. Finally, save the model as a Femap MODFEM file. You will import the .op2 file when setting up your event in Vibrata.

You may wish to adjust the view settings in Femap at this point. For example, in the figures showing the frame and Isat models (Figure 4-3 through Figure 4-6), we set the background color to solid white; set the color of all labels and postprocessing titles to black; had Femap assign a different color to each physical property in the model and then set the element color mode to use physical property colors; and set the element orientation/shape option to show cross sections, which also shows shell element thickness.

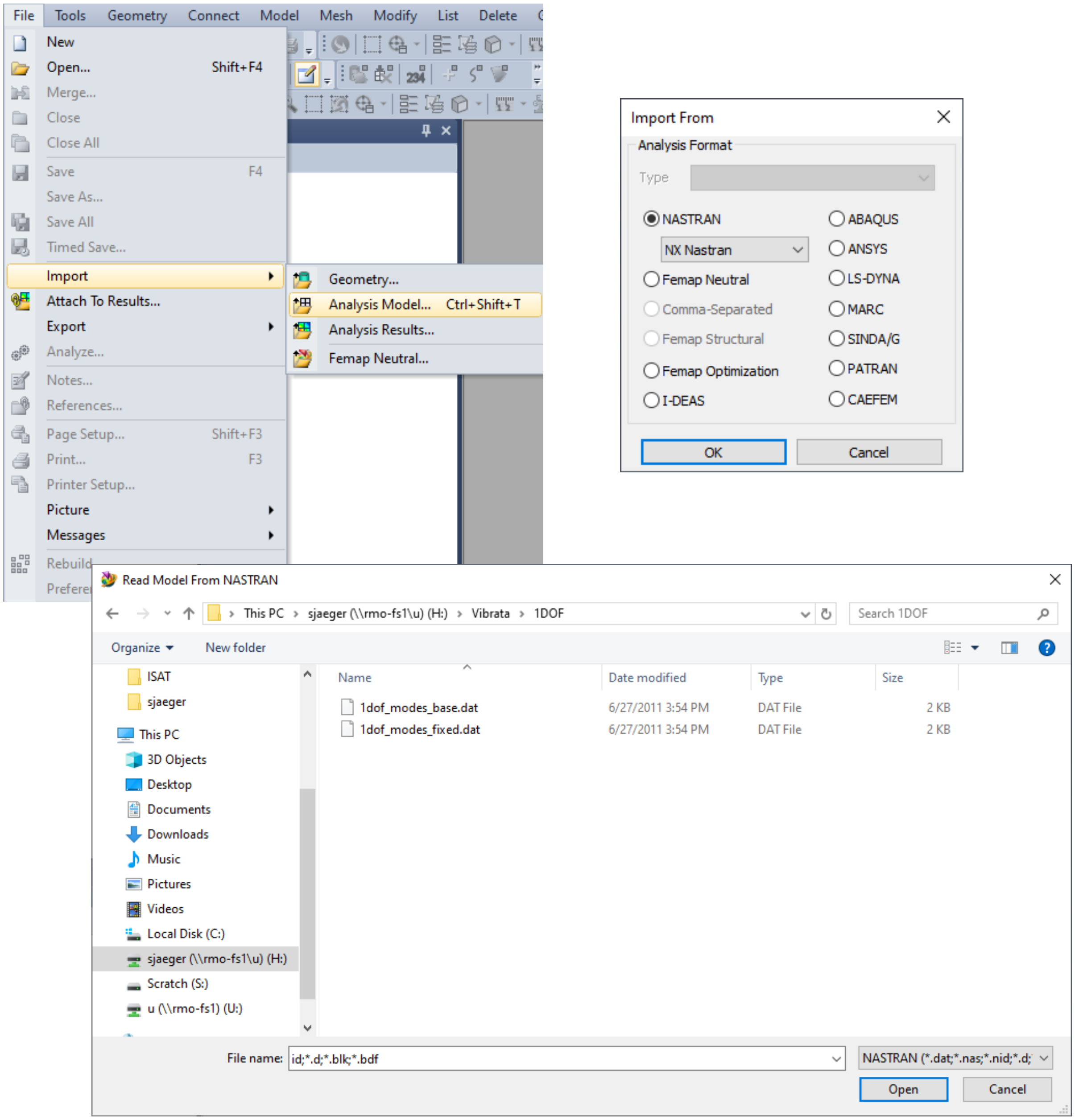


Figure 4-7. Read the bulk data into Femap using the *Import/Analysis Model* menu picks.

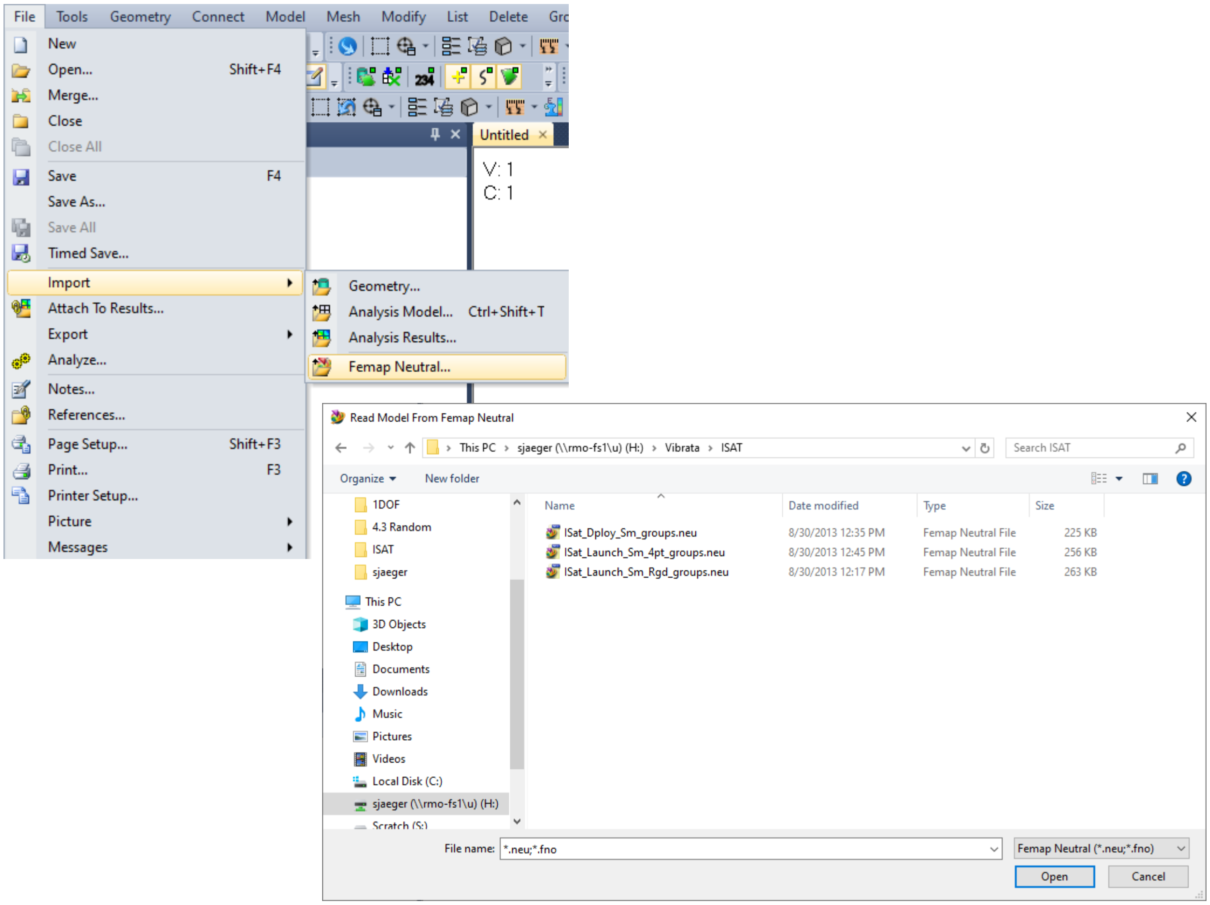


Figure 4-8. Define groups in Femap by importing a neutral file when available.

4.2. Steady-State Frequency Response Analysis

The following examples demonstrate Vibrata's frequency response analysis capabilities.

4.2.1. Analyze 1DOF Model Using a Constant Input.

1. Import the 1DOF model "1dof_modes_base.dat" (Figure 4-1). Save the file as "1dof.modfem."
2. Start Vibrata and create a new modal frequency event using that modfem file (Figure 4-9) and select the OP2 named "1dof_modes_base.op2" for the results. You can assign an XY Prefix here, or you will be asked for one when you solve.

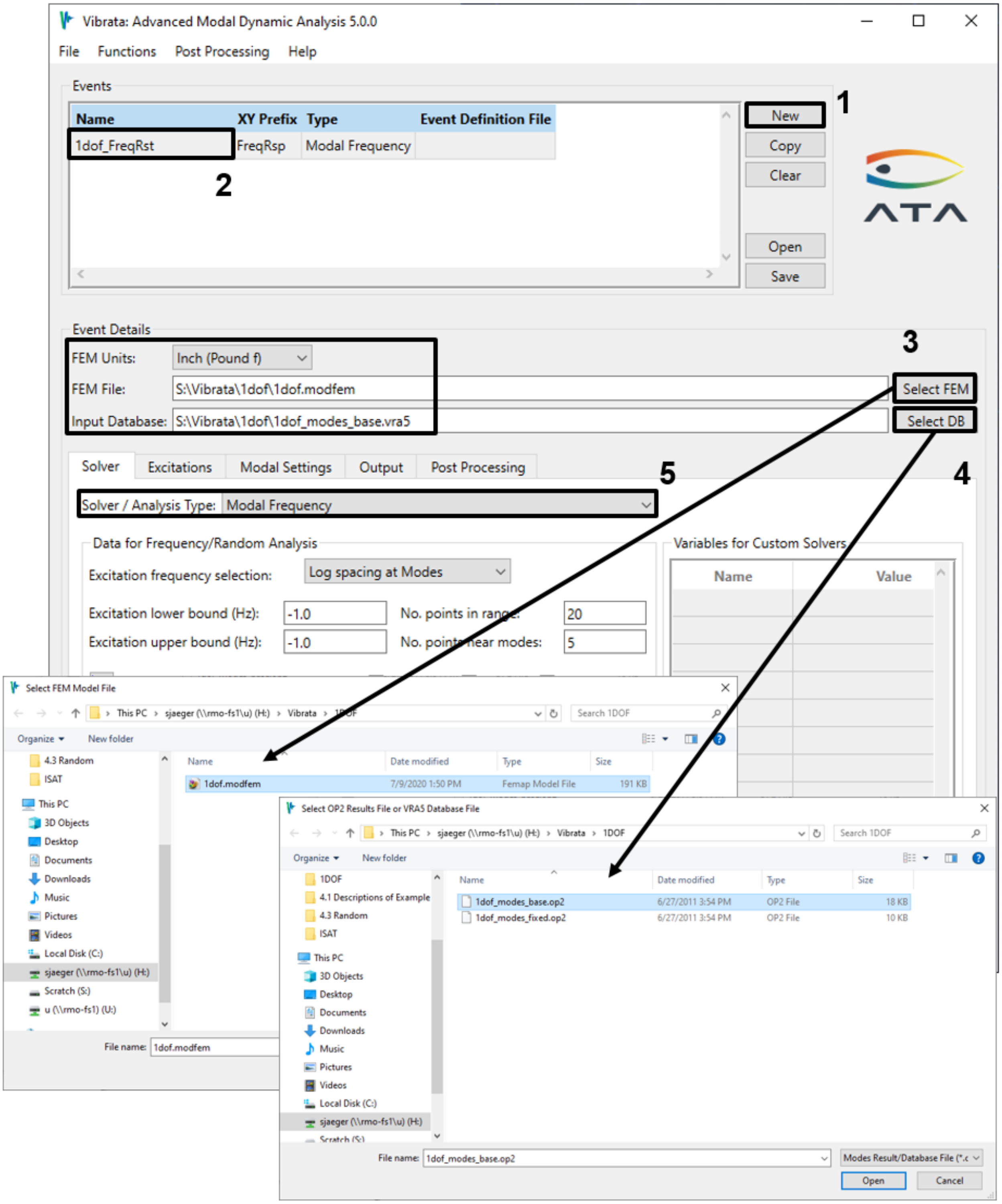
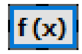


Figure 4-9. Create a new frequency response event and select the FEM on which to base it.

3. Create a constant acceleration excitation on the enforced DOF with a frequency range that includes the natural frequency of the system.

   a. Select the *Excitations* tab and click **Enforced Motion**.

b. On the *Enforced Motion* dialog, select the one available DOF and assign an excitation function to it by clicking the 

button to bring up the Function Manager (Figure 4-10).

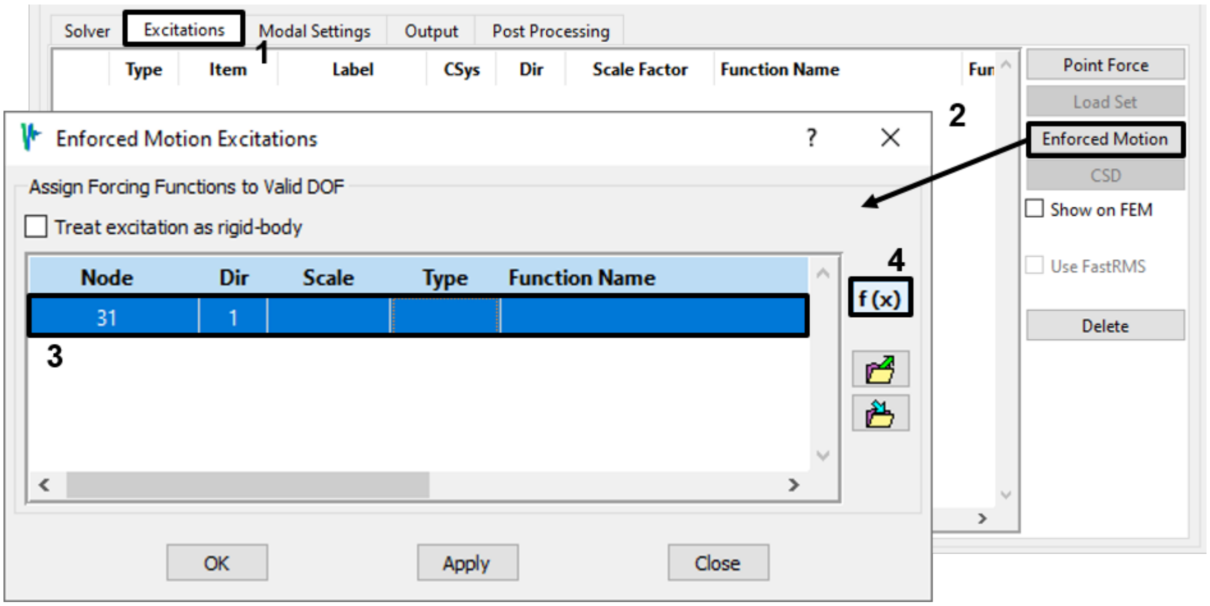


Figure 4-10. Create a new enforced motion excitation.

c. In the Function Manager (Figure 4-11), create a new function. Define it as a frequency function with acceleration in G. Give it a constant magnitude of 10G from 1 to 20 Hz.

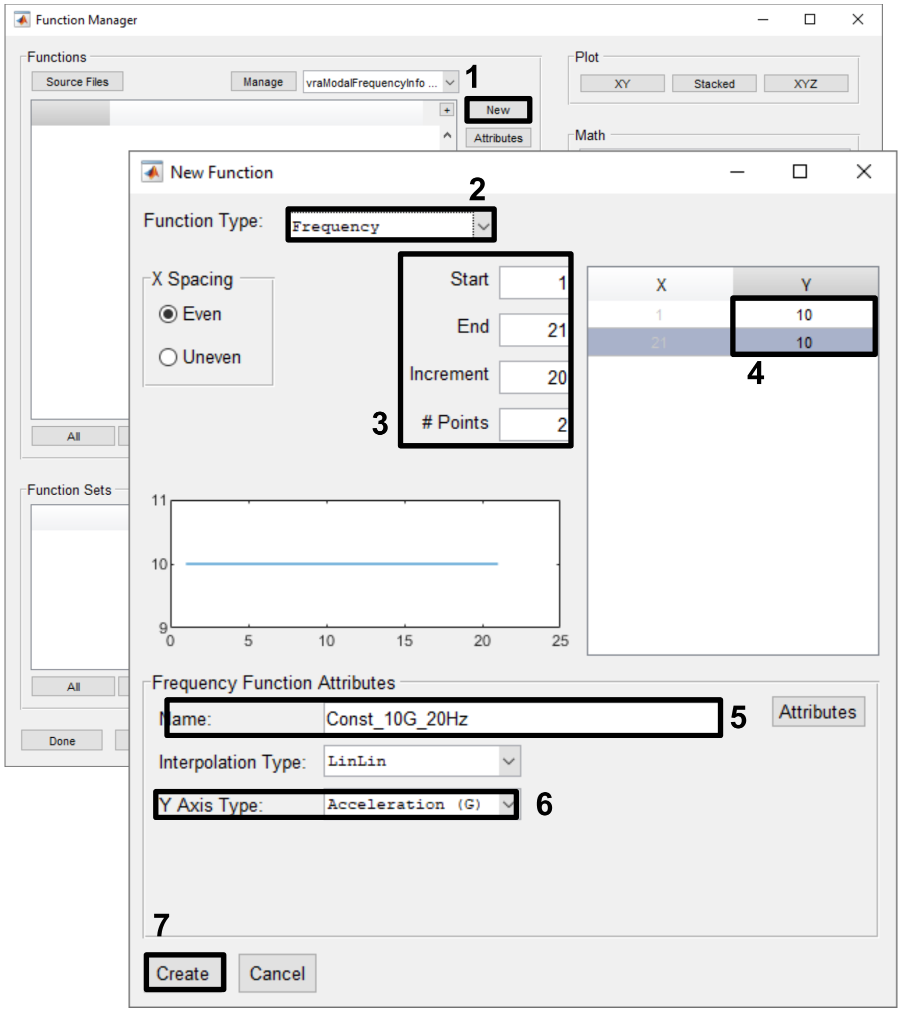


Figure 4-11. Use the Function Manager to create an acceleration function.

d. Click the **Create** button to save the new function into an *fcn* file. Call this file "1dof_examples_functions.fcn" and place it in the 1dof folder (Figure 4-12).

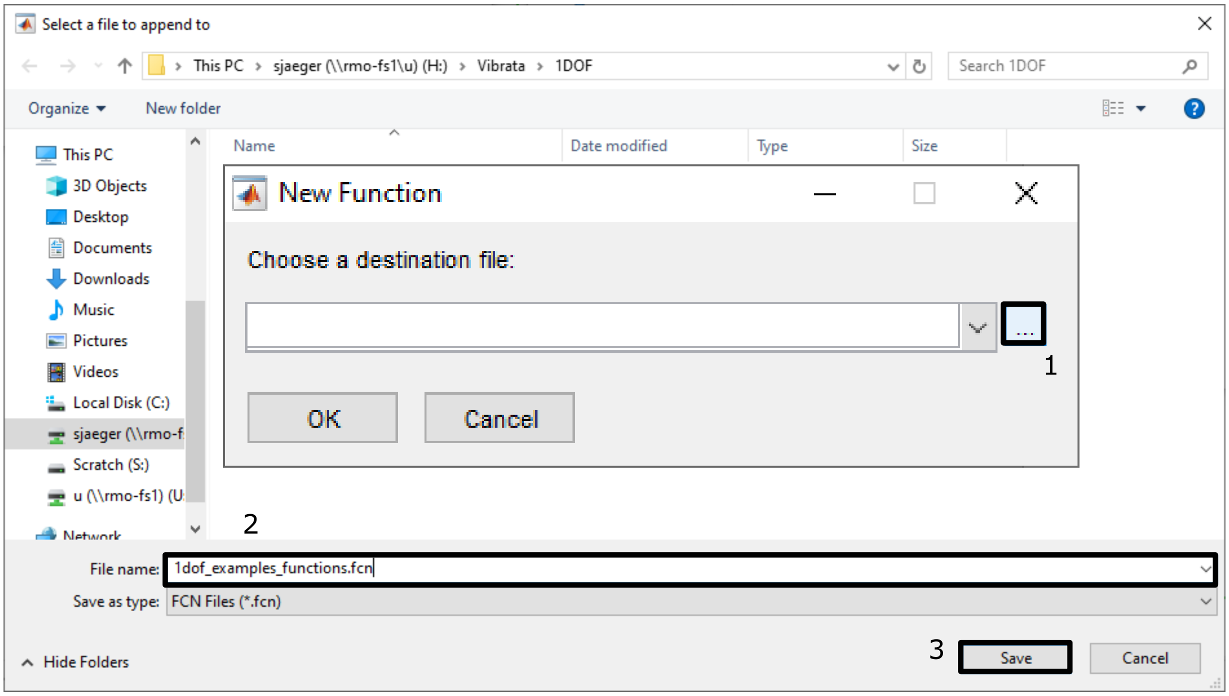


Figure 4-12. Save excitation function to new function file.

e. With your new function selected in the Function Manager, click **Done** to return to Event Manager, then click **OK** to close the *Enforced Motion* dialog and create the new excitation. It will be listed in the table on the *Excitations* tab.

4. Assign 2.5% damping to the single mode. Select the *Modal Settings* tab, double-click in the *Viscous* cell under *Modal Damping* for the mode, and enter a value of 2.5 (Figure 4-13).

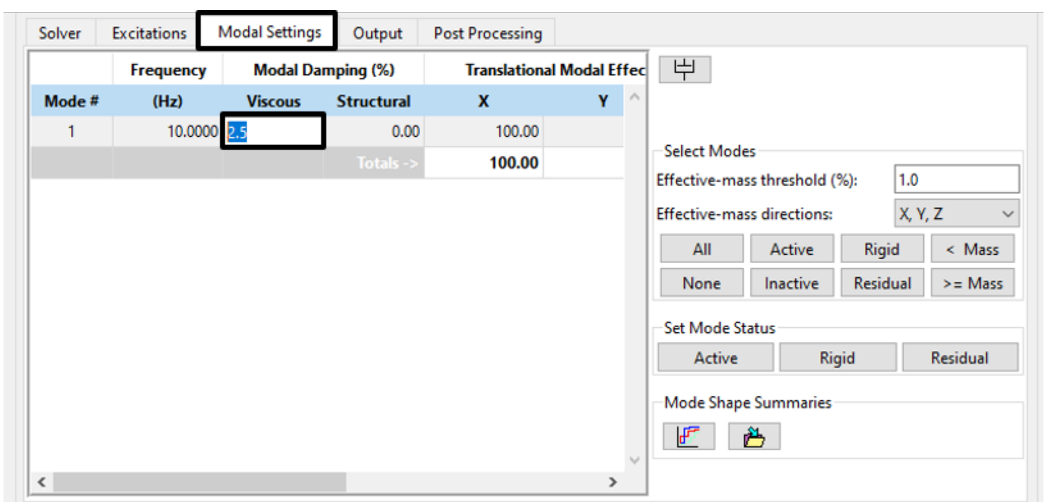


Figure 4-13. Assign 2.5% modal viscous damping to the mode directly in the modes table.

5. Create a Node XY output request for the free DOF. Select the *Output* tab. Request X-direction output at node 32 for Total Accelerations (*TATx*) and Relative Accelerations (*Atx*), as shown in Figure 4-14. When you click **OK** or **Apply** on the *Nodal XY-Plot* dialog, the request appears in the output request table as shown in Figure 4-15.

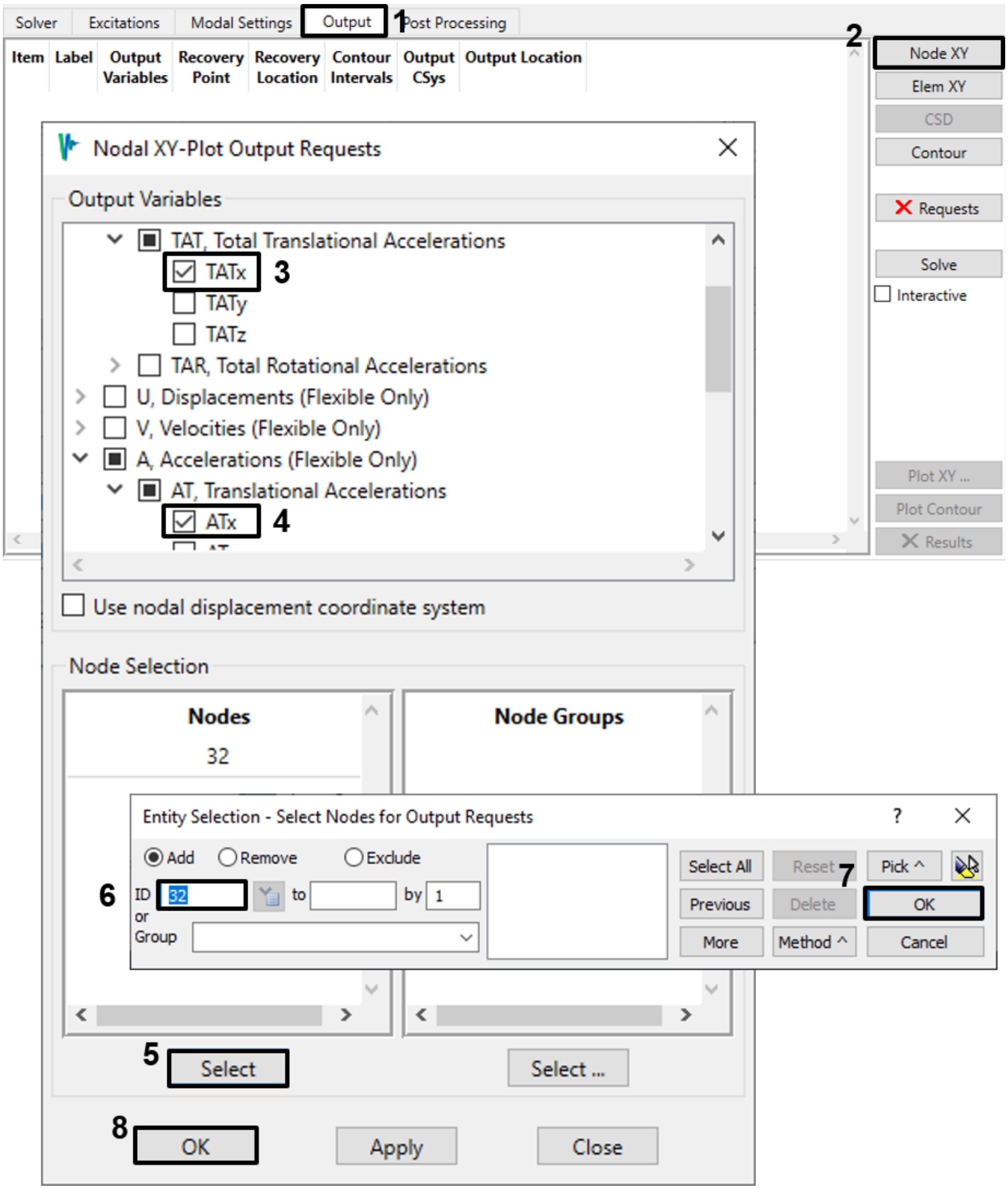


Figure 4-14. Request XY output for total and flexible X-accelerations at node 32.

6. Solve, plot, and compare the two output results.

   a. On the Output tab, click Solve (Figure 4-15) to compute the frequency response.

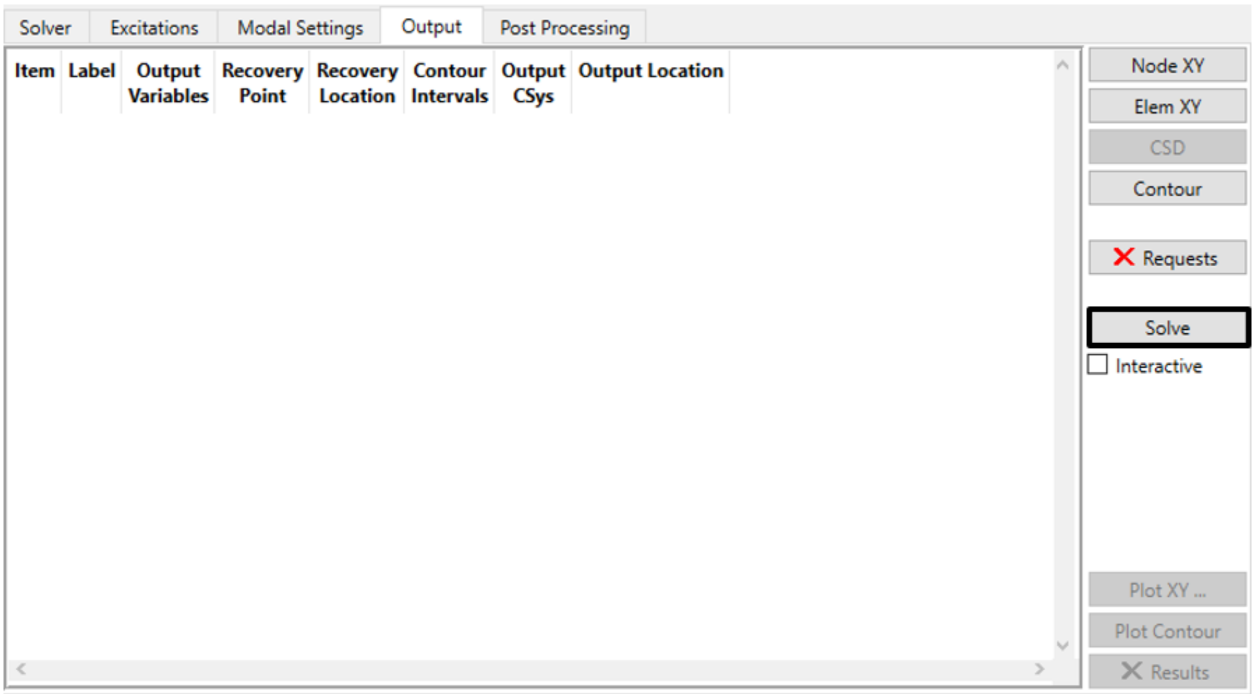


Figure 4-15. Solve for the requested output.

   b. The text of the request turns green when the solver has finished and its results are available (Figure 4-16). Plot the results by clicking the **Plot XY...** button.

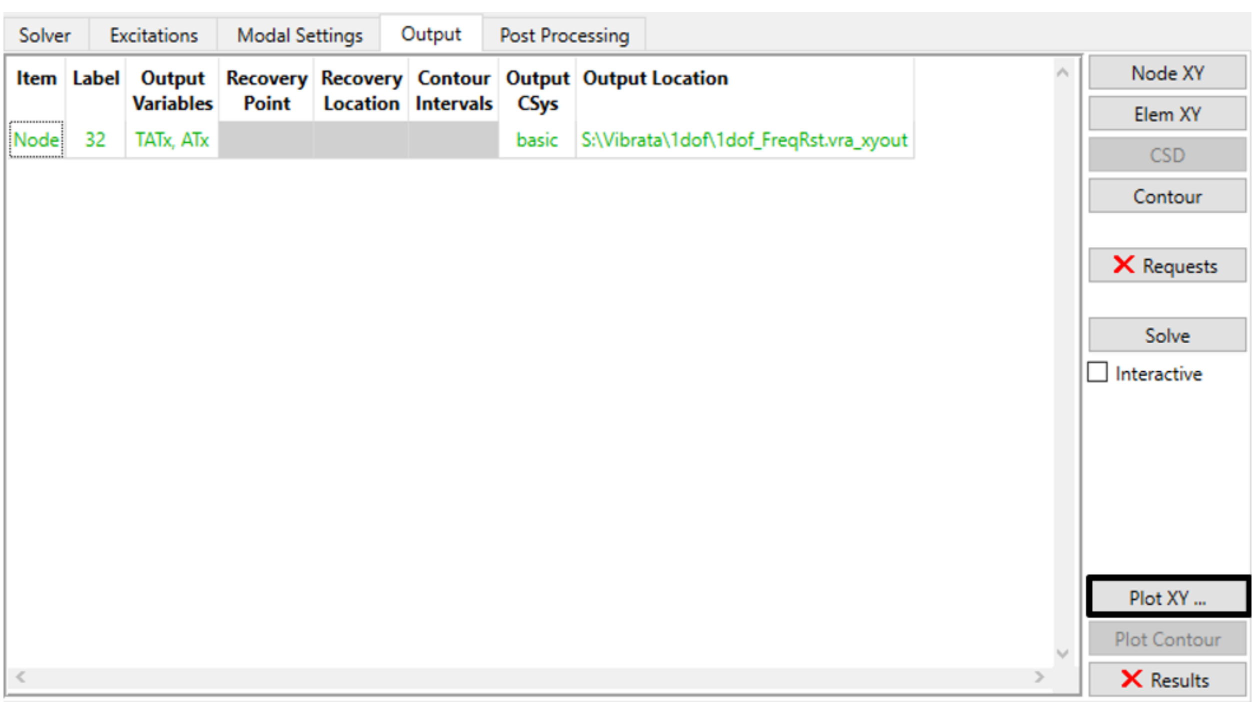


Figure 4-16. Results are available for requests in green text. Display them.

c. In the *UIPLOT* dialog, select both functions and click **Plot** to display the results (Figure 4-17). Notice the dynamic amplification and phase change of the responses at the natural frequency.

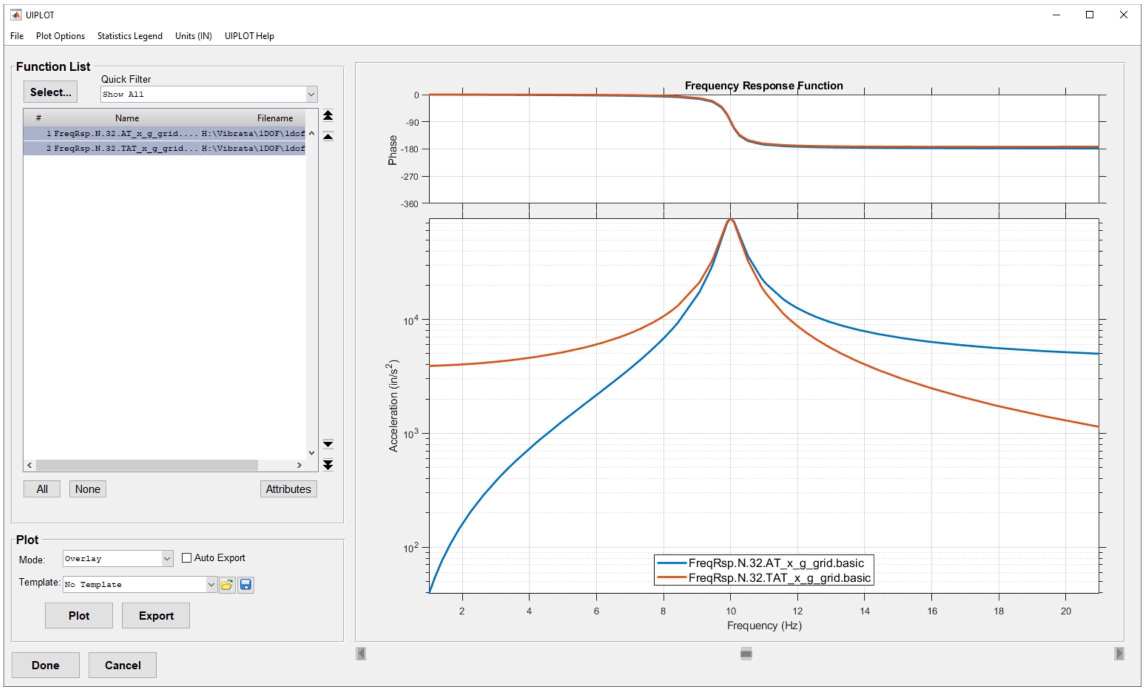


Figure 4-17. Select and plot both requested acceleration responses at once.

4.2.2. <u>2DOF Frequency Response.</u>

1. You will now perform a frequency response analysis on the 2 DOF model. Import the Nastran model "2dof_modes_base.dat" (Figure 4-2) into Femap and save as "2dof.modfem."

2. Create a frequency response event with this model. *Always* define a useful, recognizable name for the event. Select the previously created modfem and "2dof_modes_base.op2".

3. Change the number of points in the range and near modes on the *Solver* tab, as in Figure 4-18. This will refine the discretization in the XY plot. (In real models with perhaps hundreds of modes, this will be far more refinement than you will actually want. Five points on either side of each mode will be plenty, and you may want only two points in the range.)

   a. Set the number of points in the range to 100.

   b. Set the number of points near modes to 10.

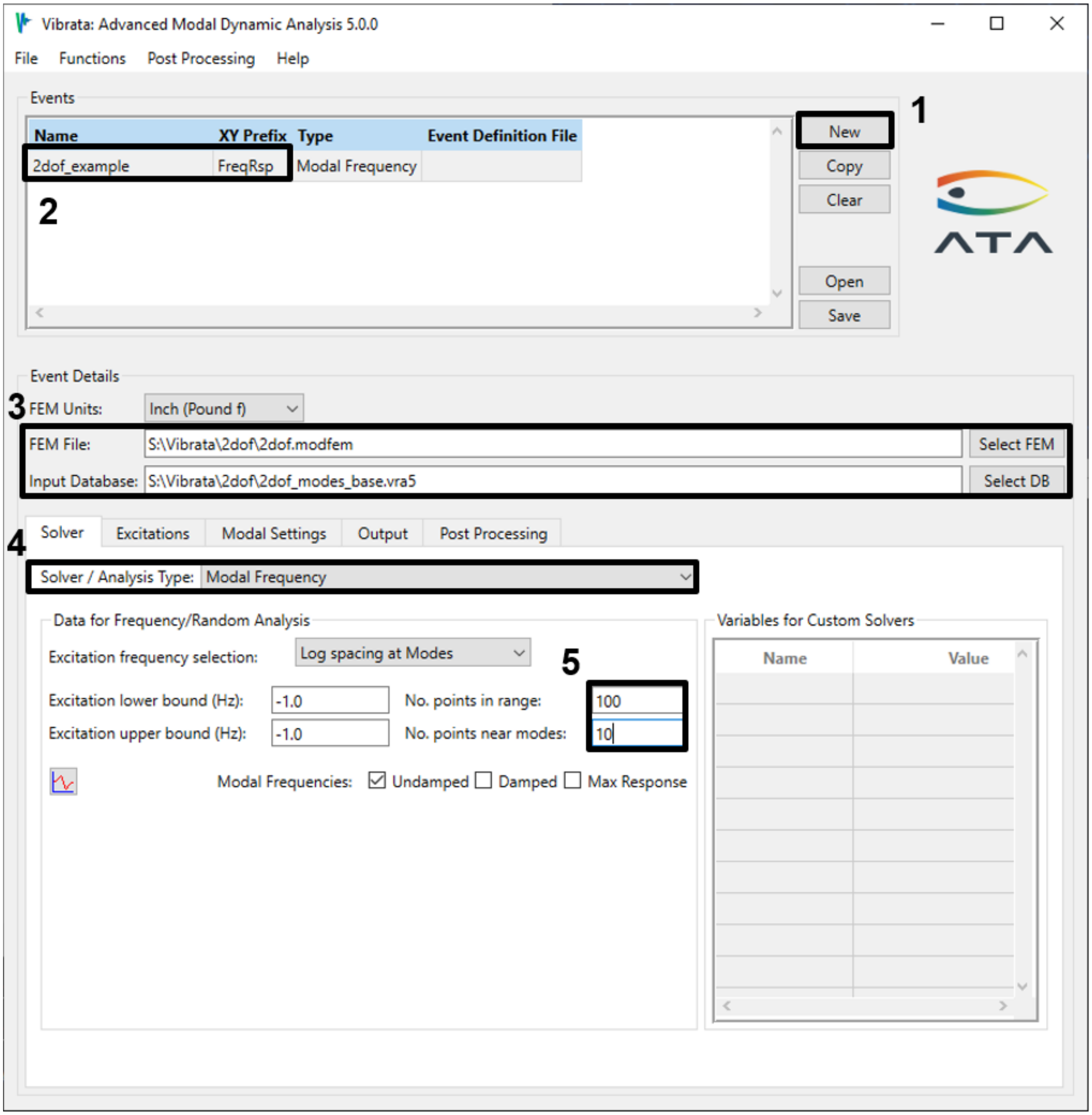


Figure 4-18. Set the number of points at which you want to compute responses.

4. On the *Modal Settings* tab, assign 1.5% damping to mode 1 and 3.5% damping to mode 2. Double-click in the *Viscous* cell under *Modal Damping (%)* for each mode and entering the proper value (Figure 4-19).

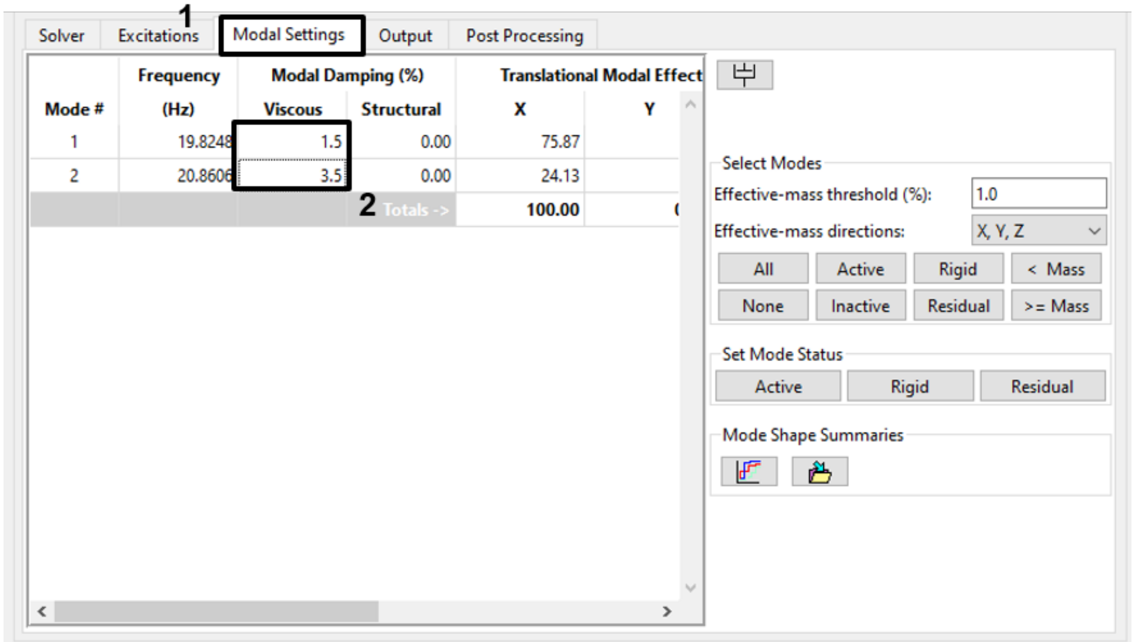


Figure 4-19. With only two modes, you can assign damping directly in the damping table.

5. For this event, we will apply a point force rather than imposing an enforced acceleration.

   a. Select the *Excitations* tab and click **Point Force**.

   b. On the *Point Force Excitations* dialog, first select node 13 to receive the applied force. Next, select the X-direction in the *Forcing Functions* table, then click the  button to bring up the Function Manager (Figure 4-20).

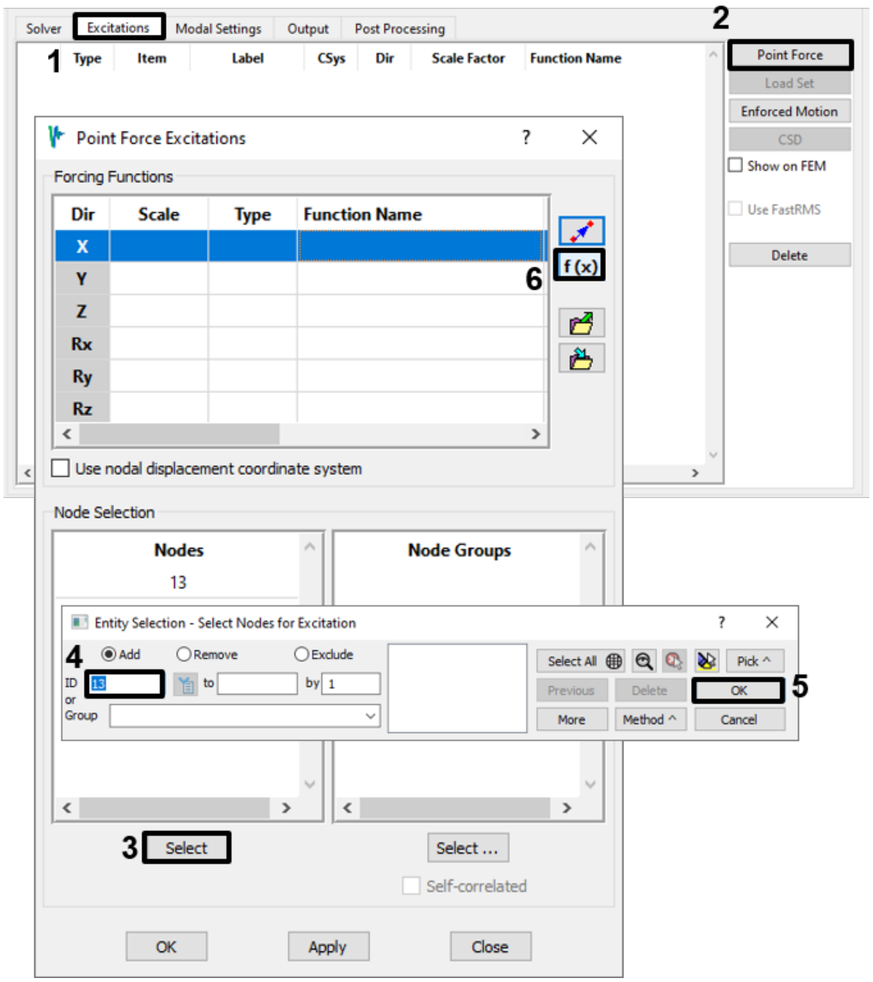


Figure 4-20. Define a point force excitation on node 13 in the X-direction.

c. Create a new frequency function from 0 to 30 Hz with an amplitude of 10. Make sure the *Y Axis Type* is *Force*, and give the function a name that you will be able to recognize later (Figure 4-21).

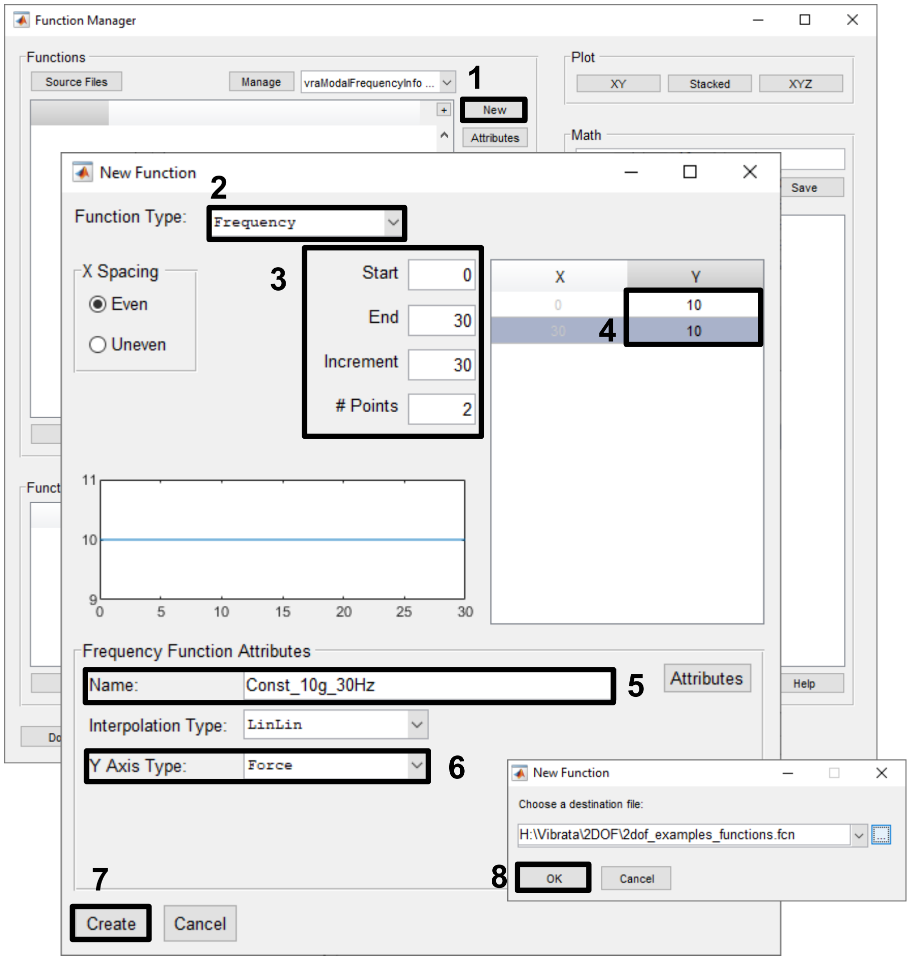


Figure 4-21. Create a new function with a constant 10 lbf amplitude from 0 to 30 Hz.

6. Request nodal output data at each node. On the *Output* tab, click the **Node XY** button. On the *Nodal XY-Plot Output Requests* dialog, select acceleration in the X-direction (Atx) and select all three nodes of the model in Femap.

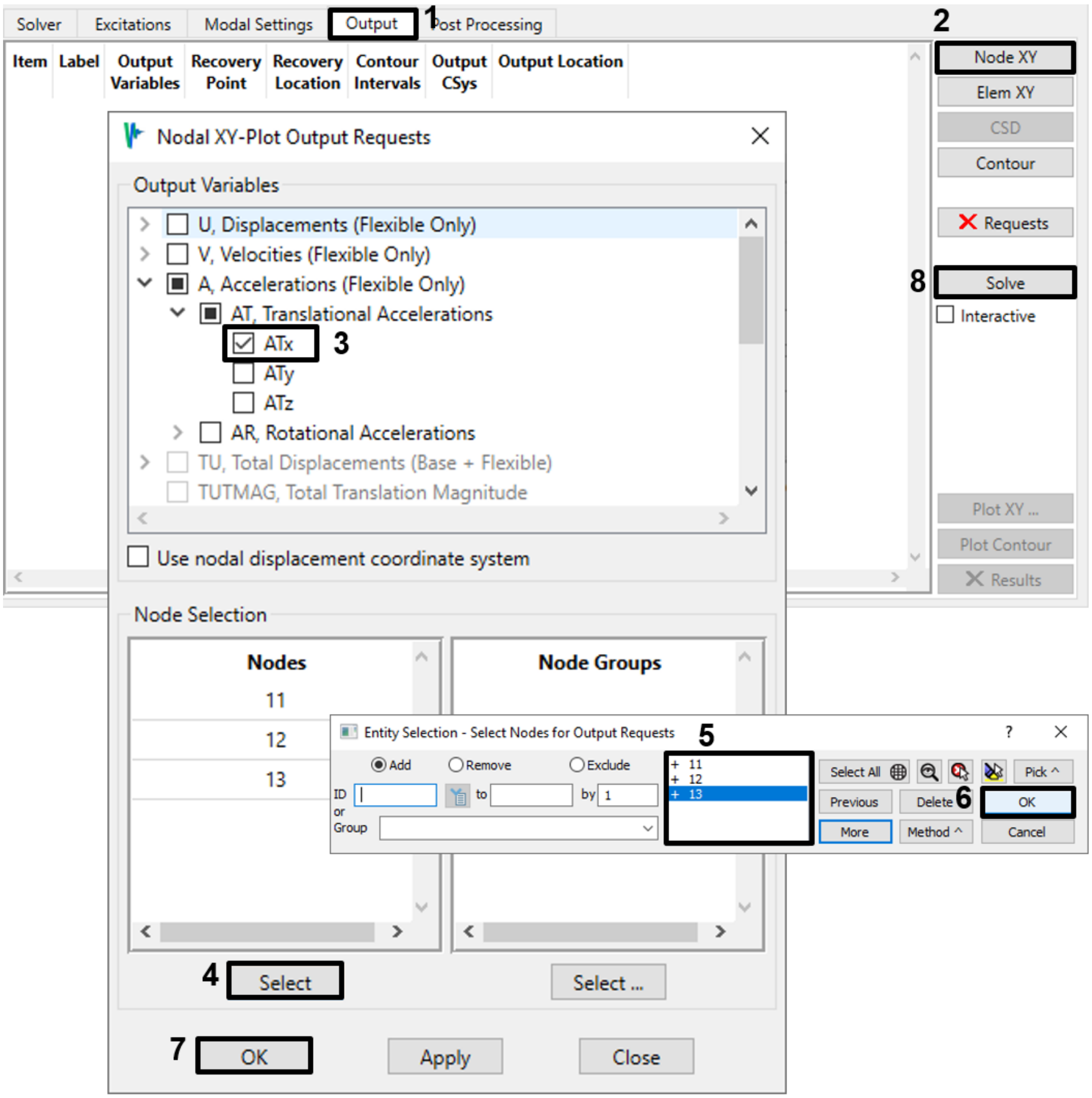


Figure 4-22. Request X-acceleration response plots for all three nodes.

7. Click **Solve** on the *Output* tab to solve for the requested data.
8. Plot the results.
    d. Select **Plot XY…** on the *Output* tab.
    e. On the *UIPLOT* dialog, select **All** and then **Plot**. By right-clicking on the chart and selecting Y-axis → Linear, you can change the y-axis scale. Notice the dynamic amplification and phase change around each natural frequency. Also note how, by right-clicking on the legend, you can reposition and reorient it.

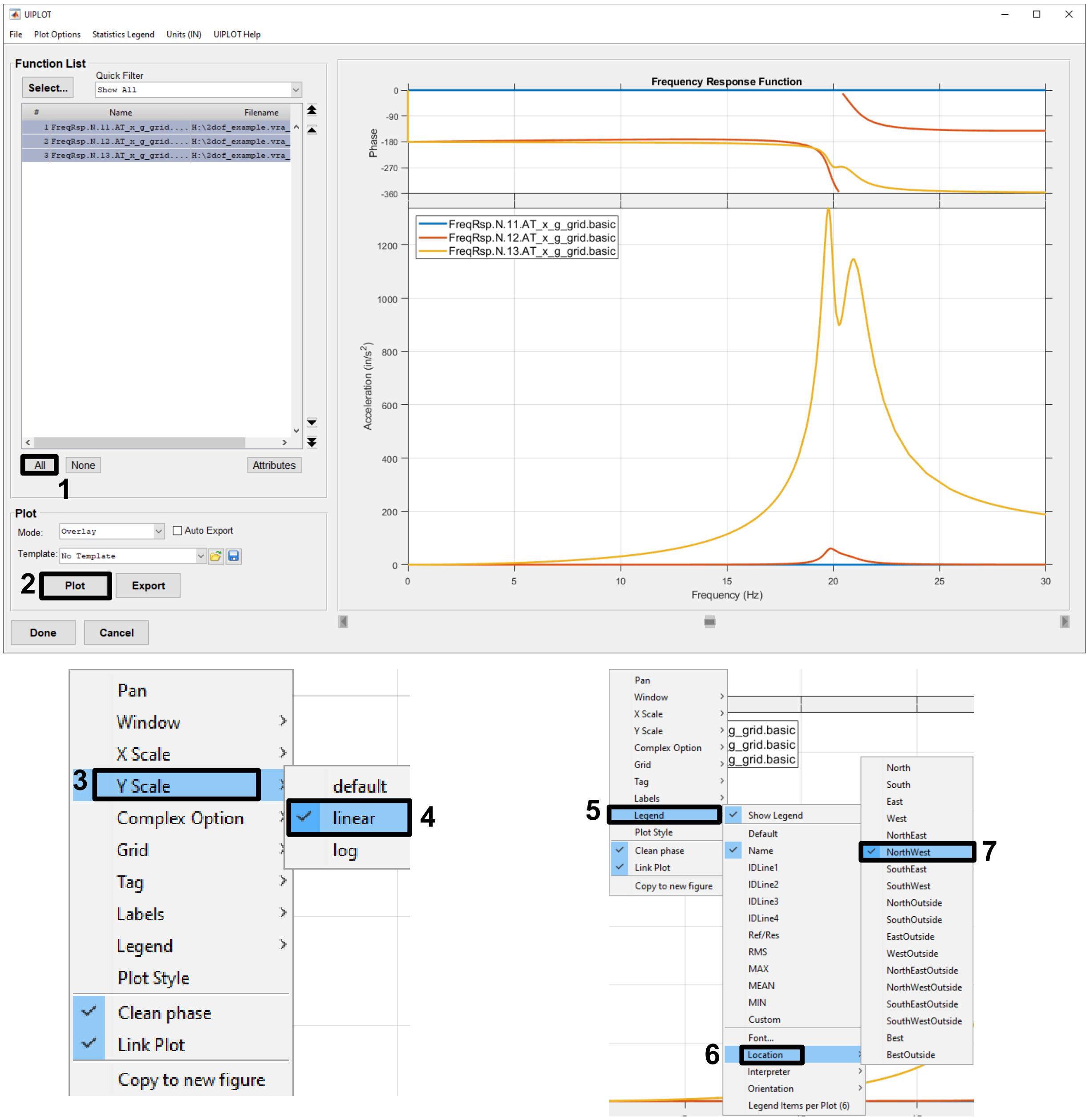


Figure 4-23. Plot the acceleration response of all DOF.

4.2.3. <u>Frequency Response of a Frame.</u>

In this example you will run two events with the frame model, one with a constant input and one with a non-constant input, to see how the different inputs affect the response.

1. In Femap, import the frame model "frame01_modes.dat" (Figure 4-3). Save the Femap model file as "frame.modfem."
2. In the Event Manager, start a new modal frequency event and select "frame.modfem" for the FEM and "frame01_modes.op2" for the results. Give the event a recognizable name, such as the one shown in Figure 4-24.

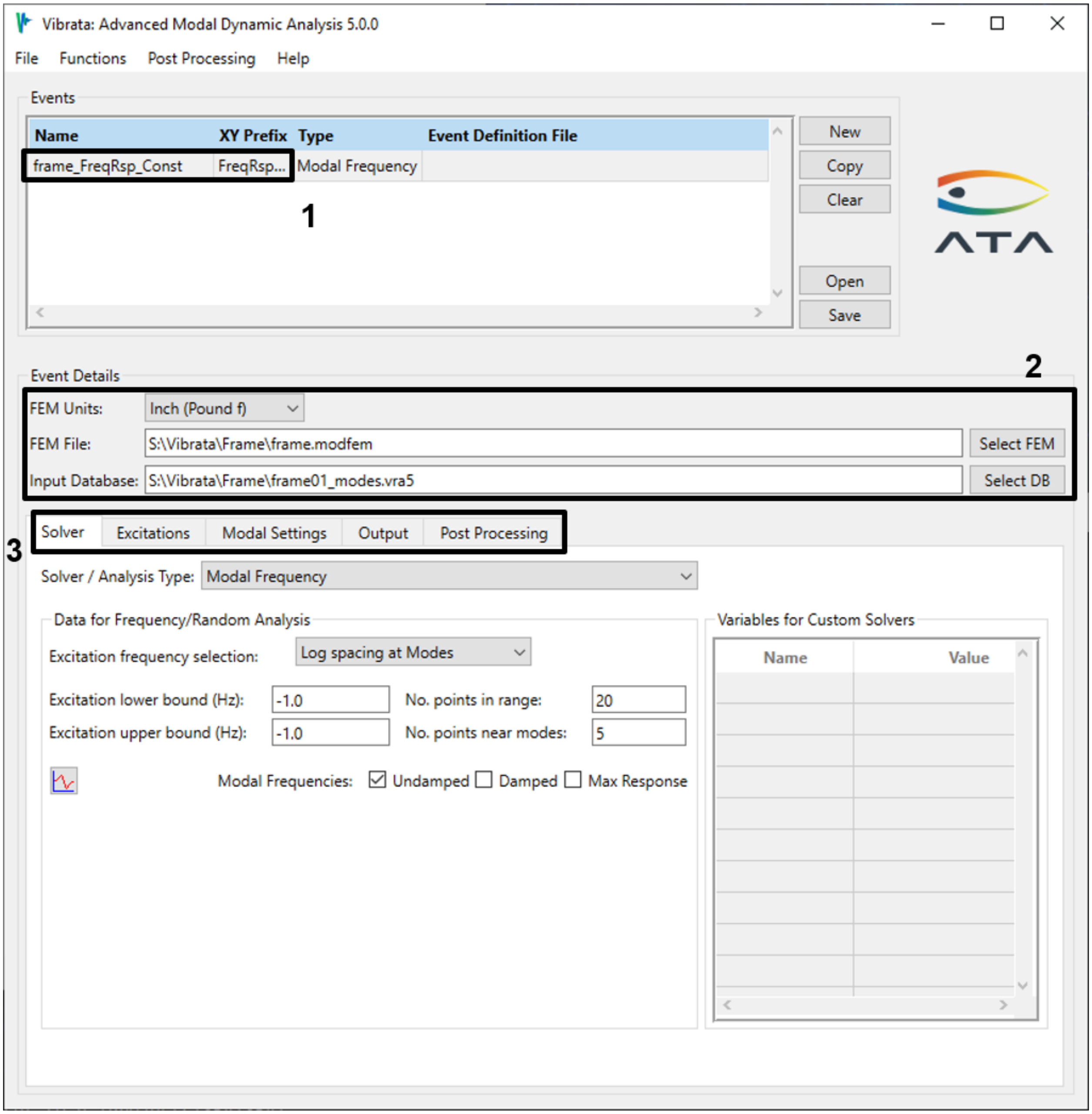


Figure 4-24. Create a new model frequency event using the frame model.

3. On the *Excitations* tab, select **Enforced Motion**, select the one available DOF, and then click the f(x) button (Figure 4-25).

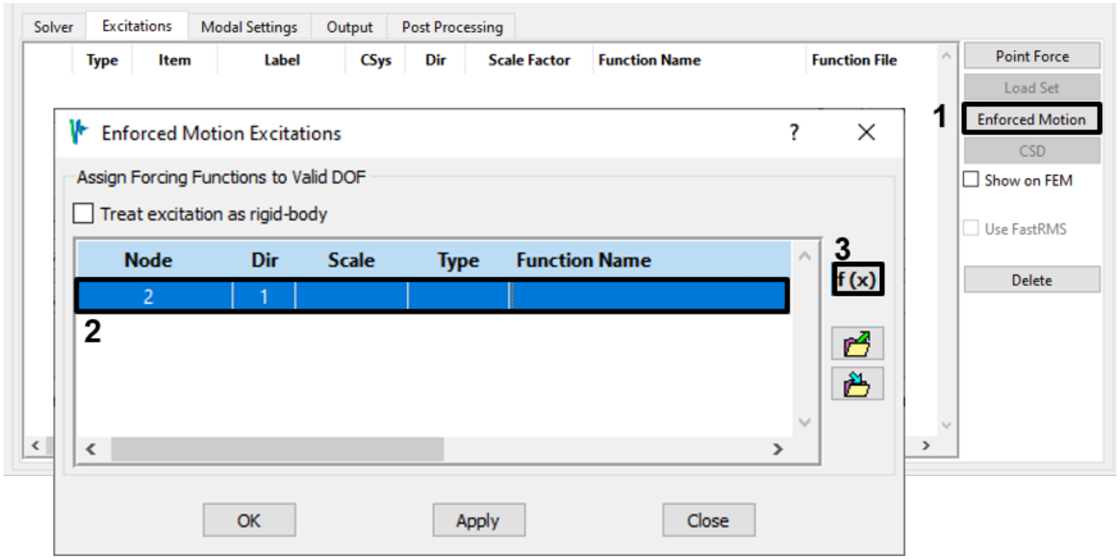


Figure 4-25. Apply a forcing function to the available enforced motion DOF.

4. Define both functions for the analyses.

   a. In the Function Manager dialog, click the **Source Files** button and open the function file from the 1DOF example (Figure 4-26).

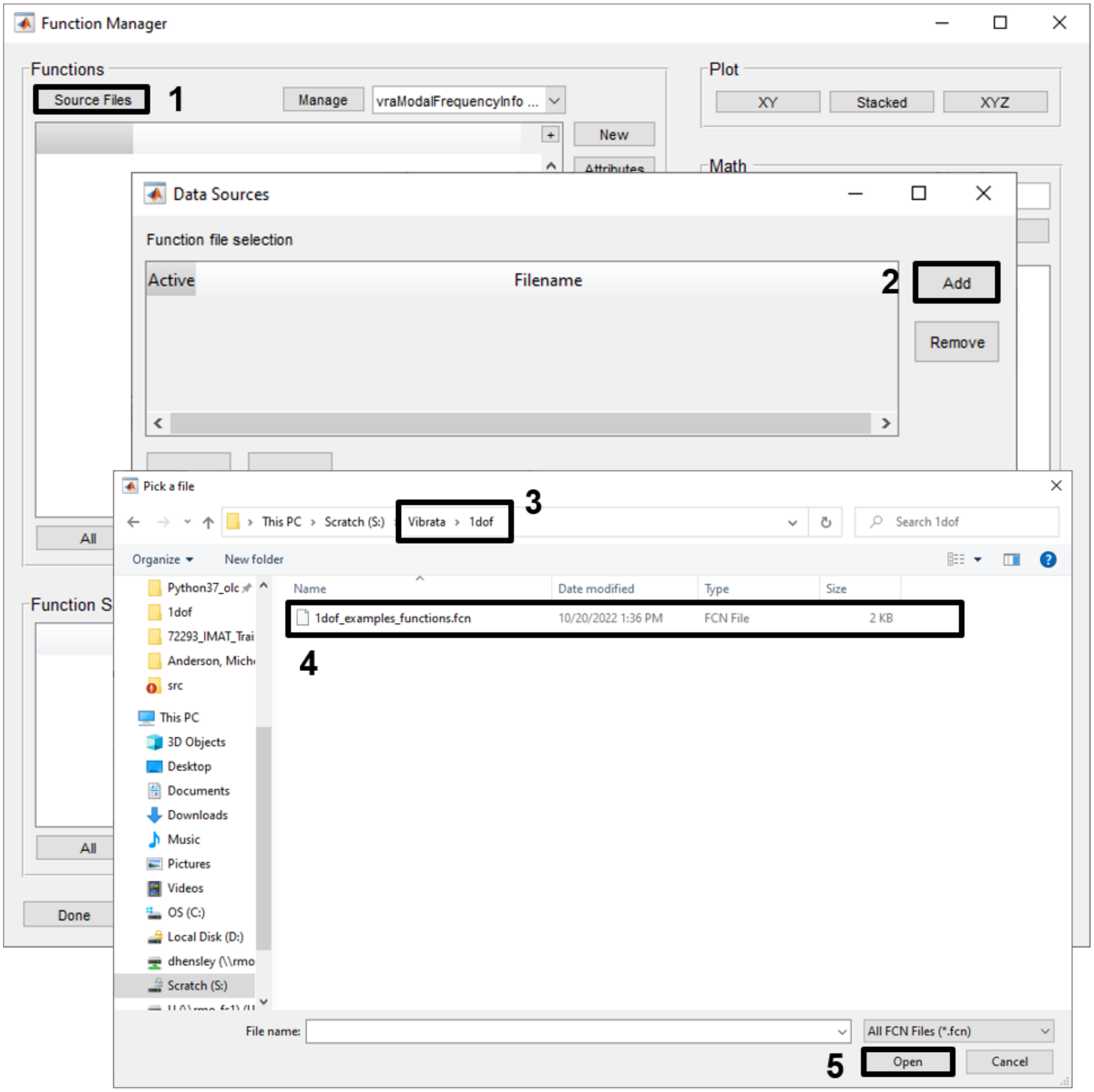


Figure 4-26. If necessary, reopen the file containing the excitation function from the 1dof example.

b. Select the original function and click **Copy**. Store the new function to a new function file in the *frame* folder; call it "frame_excitations.fcn." Select the new function and click the **Edit** button. On the *Edit Function* dialog, change the second X-value from 30 to 40 and click **Save** (Figure 4-27).

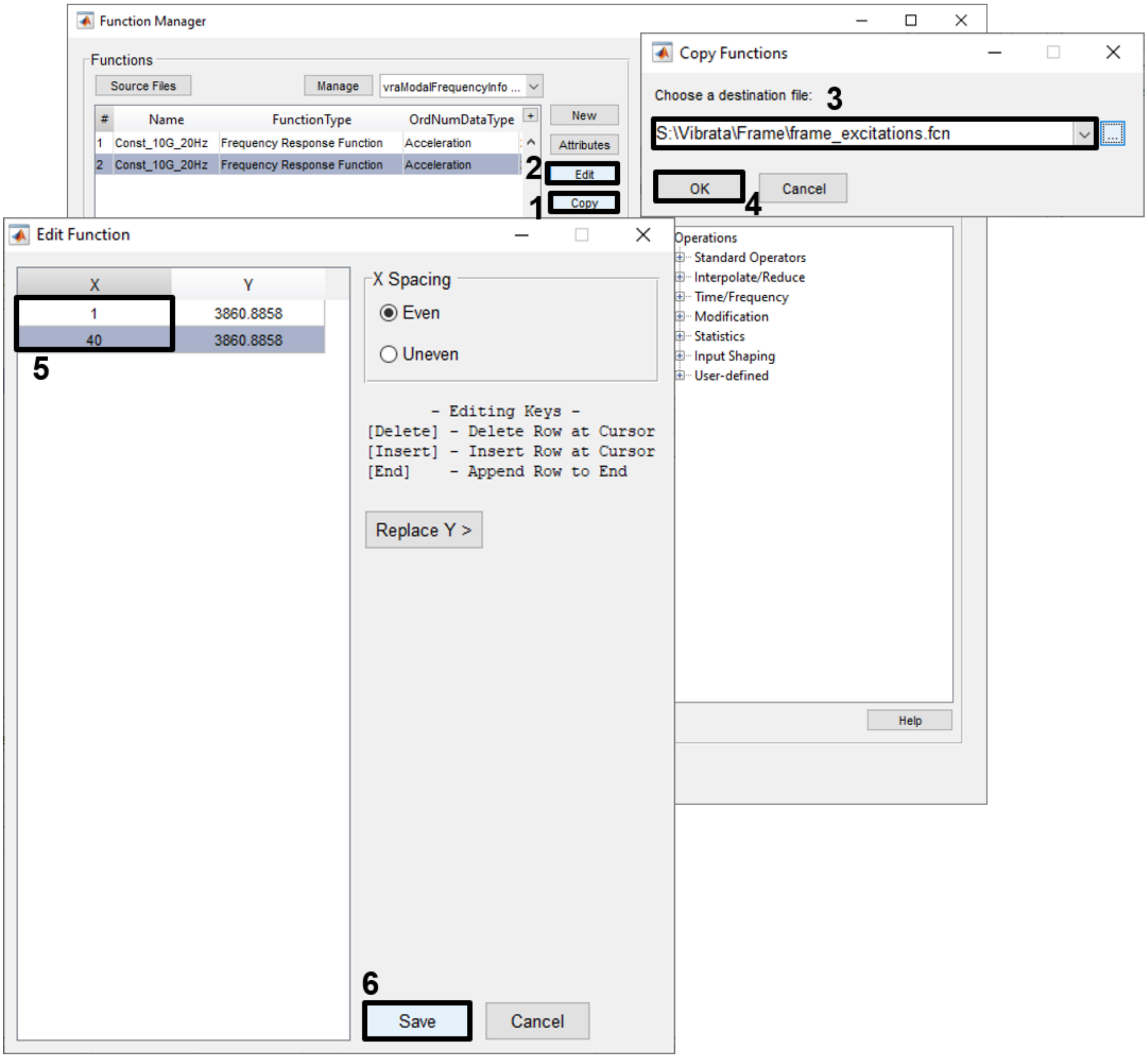


Figure 4-27. Change the copied function to have a range from 1 to 40 Hz.

c. With the new function still selected, click the **Attributes** button to open the *Edit Function Attributes* dialog; change the interpolation method to *LogLog* and its name to “Const_10g_1to40Hz” (Figure 4-28). Click **Done** to close the dialog and save the changes.

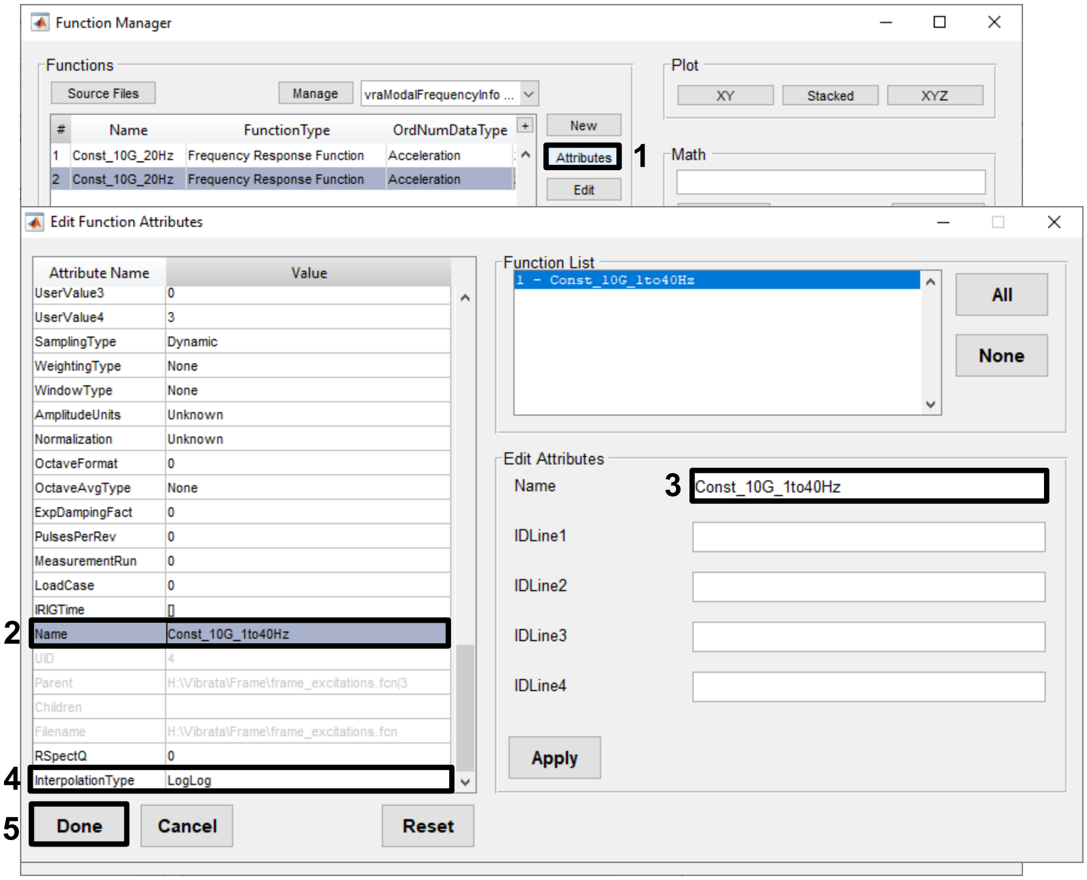


Figure 4-28. Edit the new function's name and make it use log-log interpolation.

d. With the Function Manager still open, create a new acceleration function that is not constant. Use the values shown in Figure 4-29. Be sure to name it as shown and set its interpolation method to *LogLog*. We will use this function in the second part of this example.

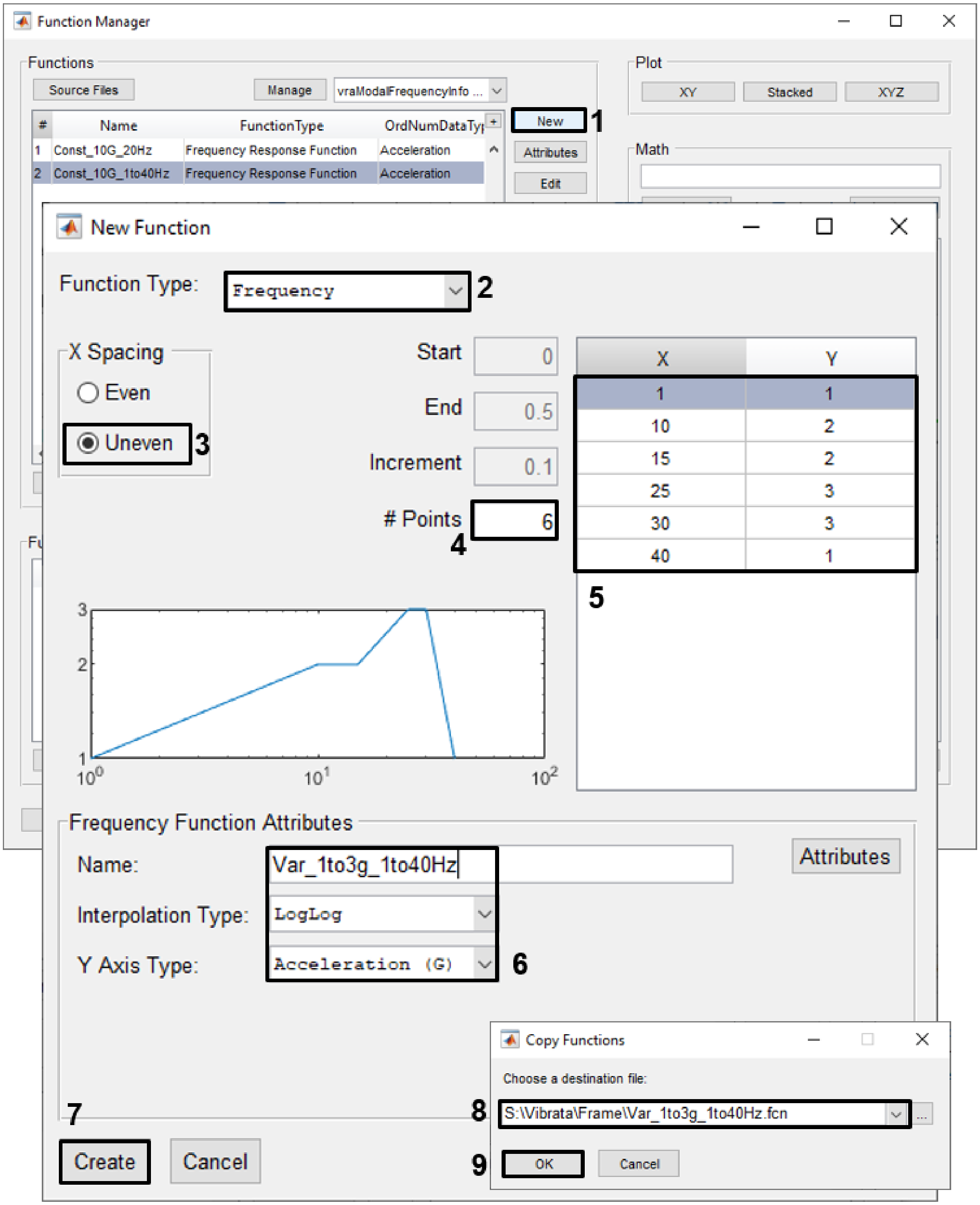


Figure 4-29. Make the function uneven and give it non-constant amplitude.

e. Select the constant forcing function for the first event. Your *Excitations* tab should look like Figure 4-30. Note that the excitation, because it is an enforced motion, is taken to be in the driven node's displacement coordinate system.

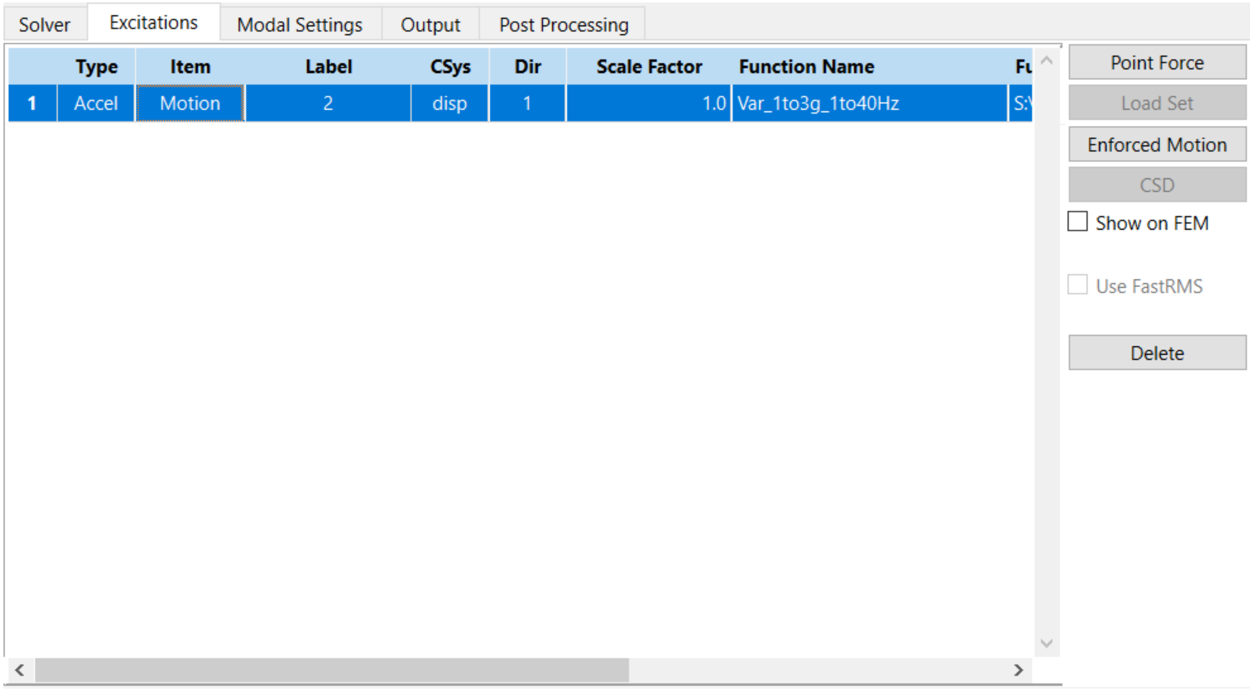
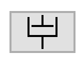
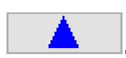


Figure 4-30. *Excitations* tab with resulting enforced motion definition.

5. Assign viscous damping of 5% to the first mode and 2% to all other modes. Set the Residual modes.

   a. On the *Modal Settings* tab, click the Damping button  to bring up the *Modal Damping Definition* dialog.

   b. In the *Viscous Damping Schedule* table, enter a value of 2 in the *Damping (%)* column and leave the frequency column blank.

   c. Click the Apply Schedule button . In the *Damping Summary* table, each mode now has 2% damping. Change the first mode's damping to 5% by double-clicking on that cell in the *Damping Summary* table and typing in the value, as shown in Figure 4-31. Click **OK** to apply the damping changes and close the dialog.

   d. On the Modal Settings tab, select modes 11–15 and set the mode status to Residual.

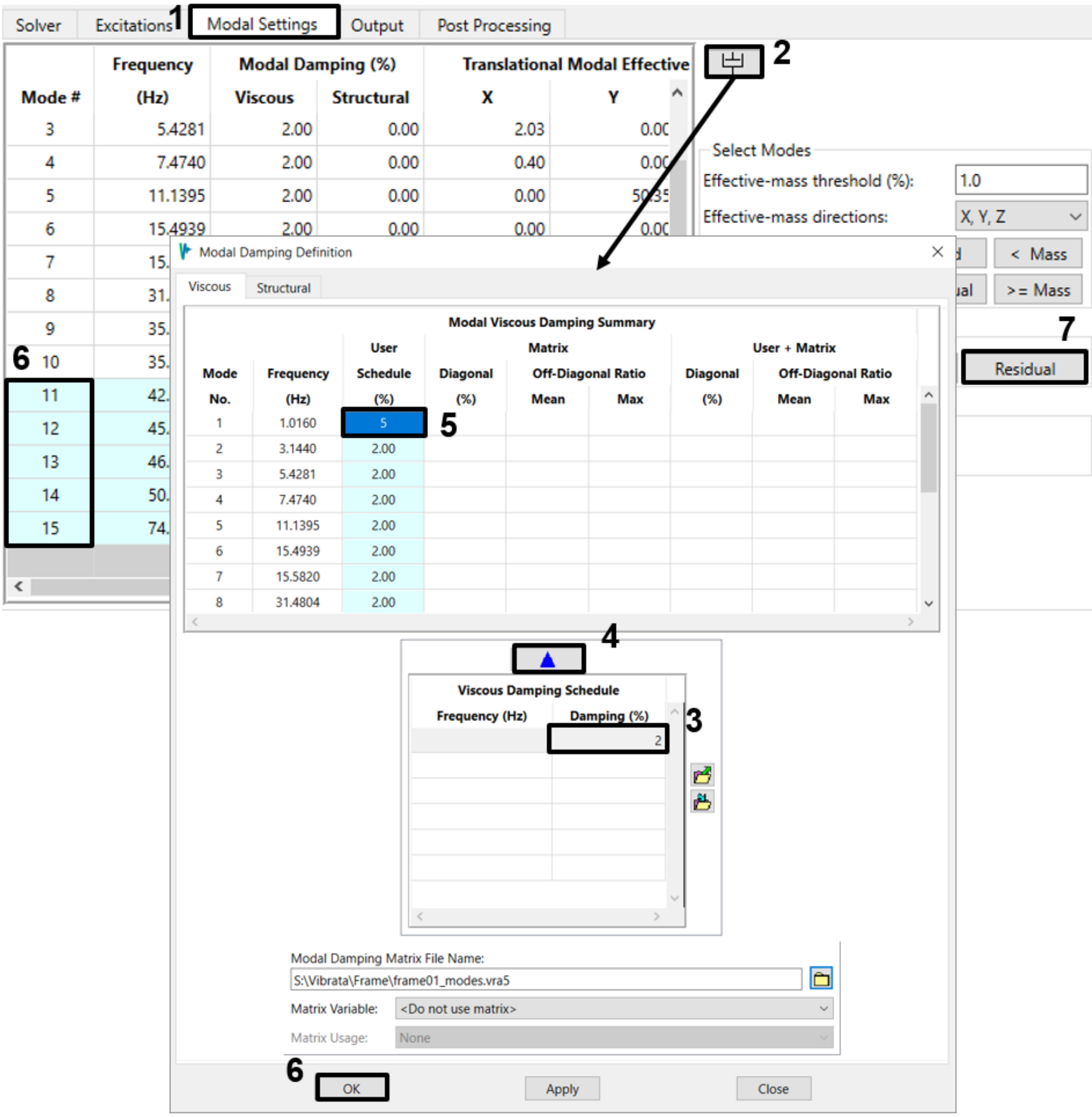


Figure 4-31. Apply 2% damping to all modes via damping schedule, then set mode 1 to 5%.

6. On the *Output* tab, create a Node XY output request for flexible and total X-translations (*TUTx* and *Utx*) for the center base (node 2), center second floor (node 8) and upper right corner (node 15), as shown in Figure 4-32.

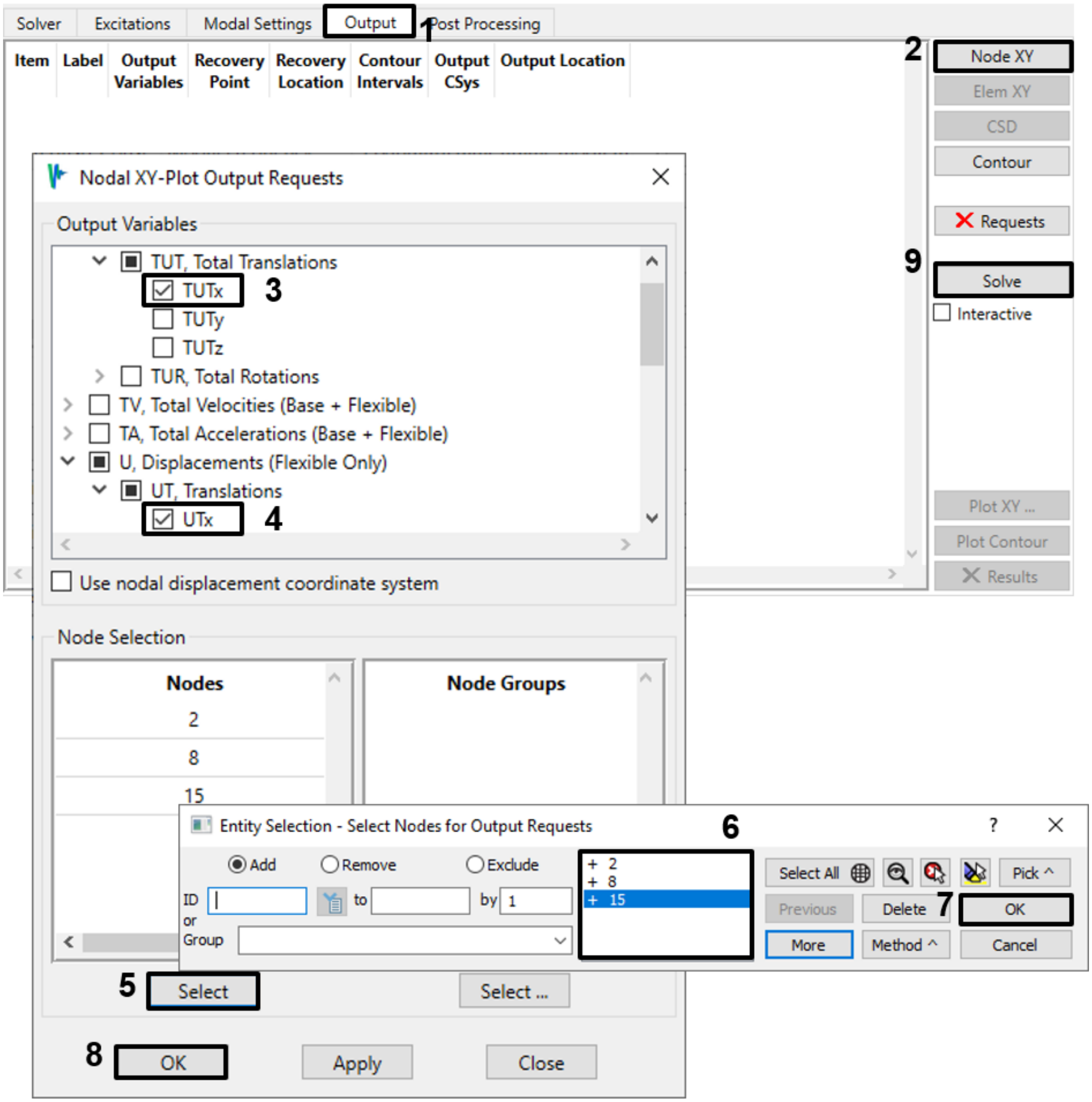


Figure 4-32. Request both flexible and total nodal X-translation plots.

7. Solve for the requested output by clicking **Solve** on the *Output* tab. We will review these results later.

8. Copy the current event and modify it to use a different forcing function.

   a. In the *Events* list, select the “frame_FreqRsp_Const” event and click **Copy**.

   b. Rename the new event to “frame_FreqRsp_Var” and set the XY Prefix accordingly. Note that its output requests are again in black text; these have not yet been solved (Figure 4-33).

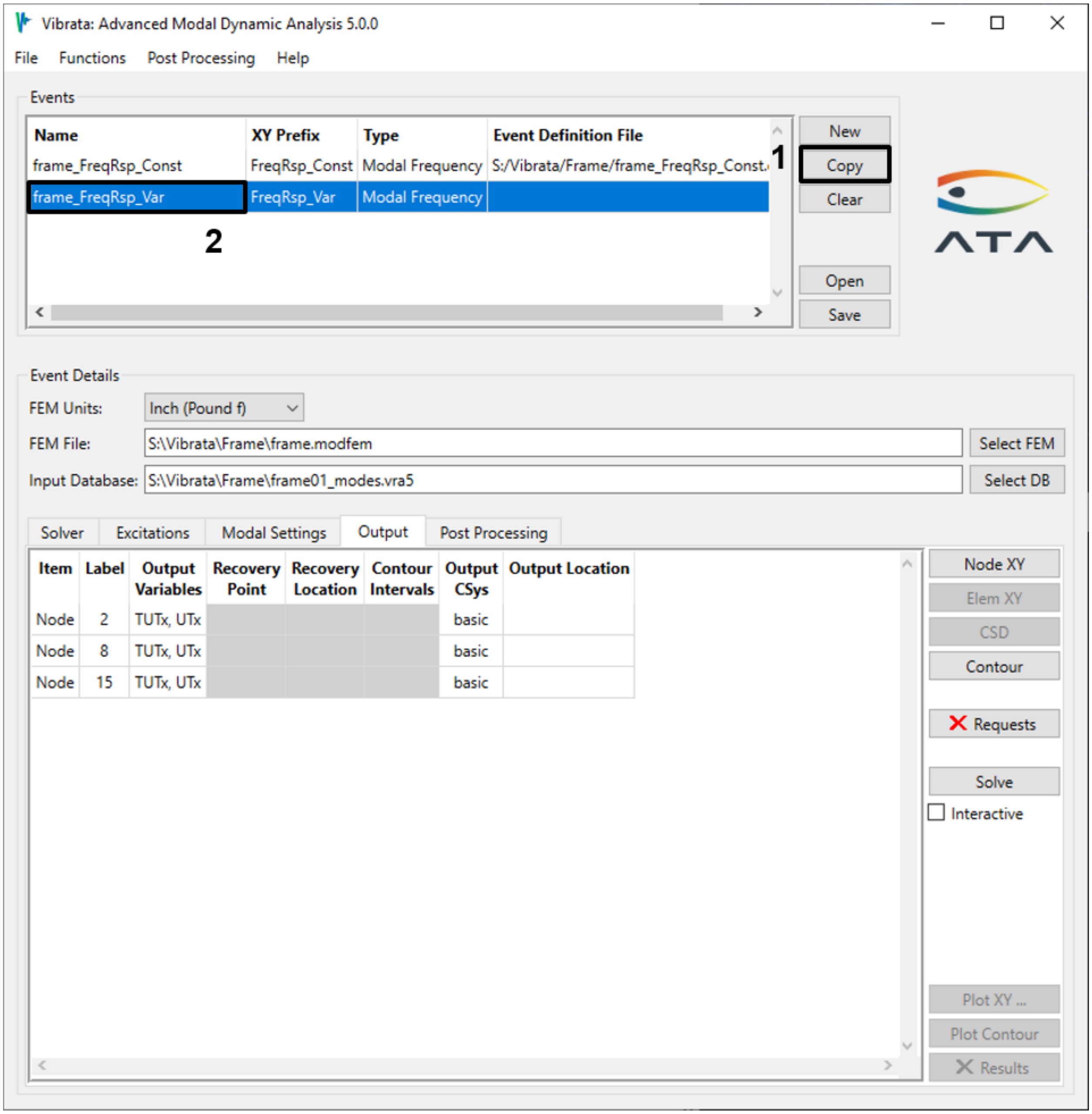


Figure 4-33. Use the **Copy** button to create a new event from an existing one.

c. With the new event selected, go to the *Excitations* tab, select the existing excitation, and edit it by clicking on **Enforced Motion**. Select the variable-amplitude forcing function (Figure 4-34).

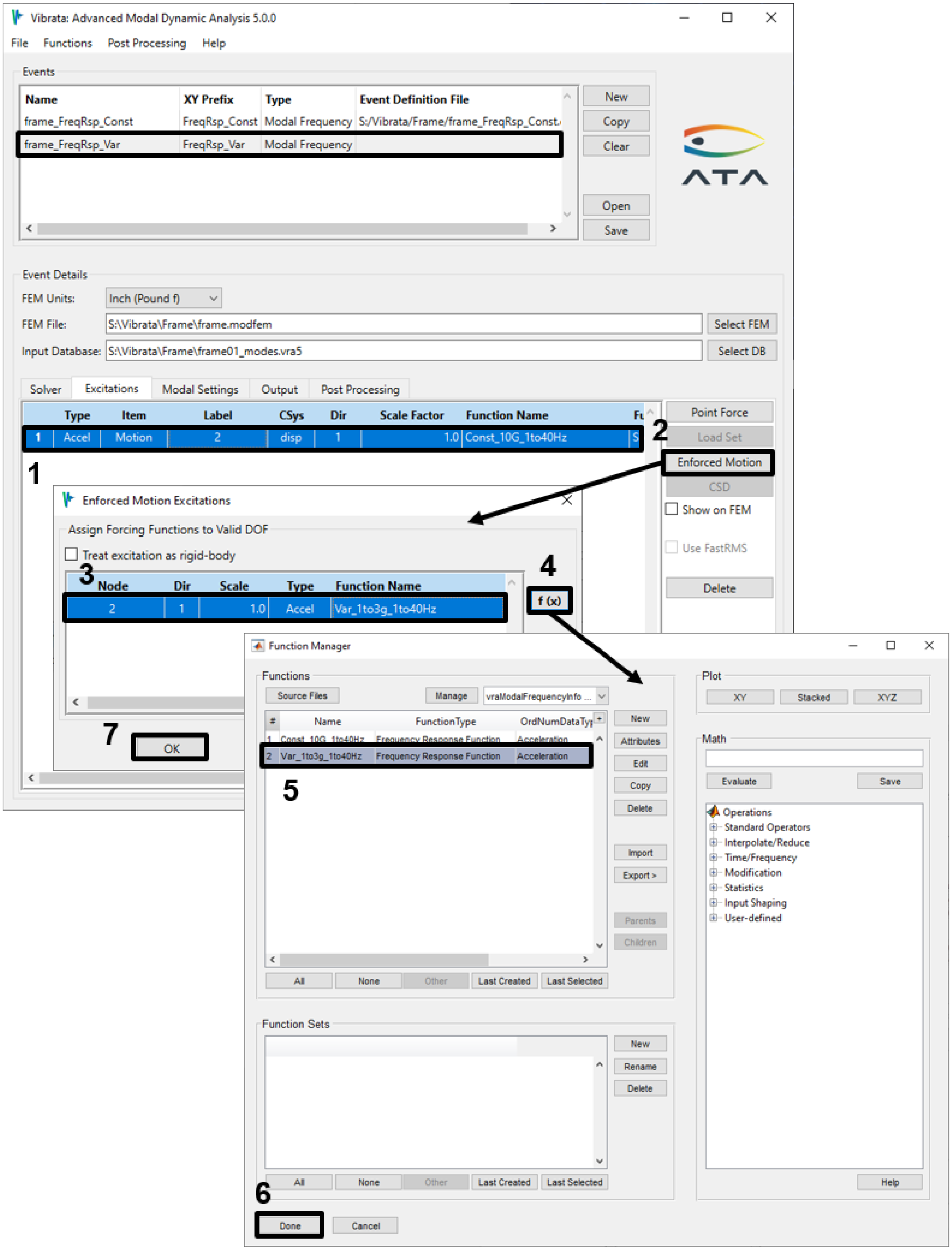


Figure 4-34. For the new (copied) event, delete the old excitation and create a new one.

9. Solve the new event by clicking **Solve** on the *Output* tab.
10. Compare the results of the two events.
    a. On the *Output* tab, click **Plot XY...** after the solution has finished.
    b. In the *UIPlot* dialog, load the results from the first frame event (Figure 4-35).

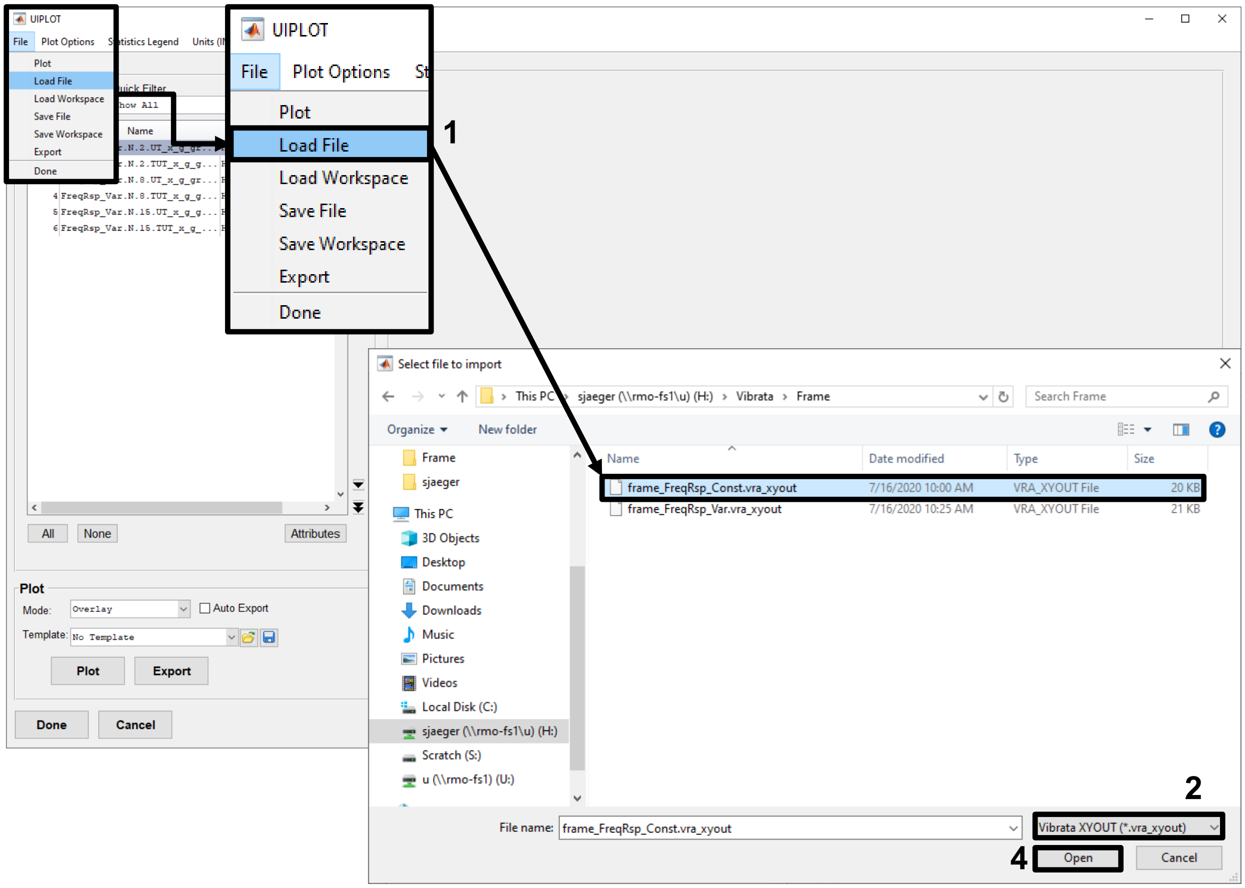


Figure 4-35. Load results from the constant-amplitude event into the output plotter.

    c. Plot and compare the results. For example, Figure 4-36 shows the total X-displacement of node 15 from both events. Although the amplitudes differ due to the different excitation levels, the phase changes are identical for both curves because these depend only on the modal frequencies. Use the Plot Options menu to change the data displayed in the legend (inset). Use the right-mouse-button menu in the plot window to show the X-axis with a log scale.

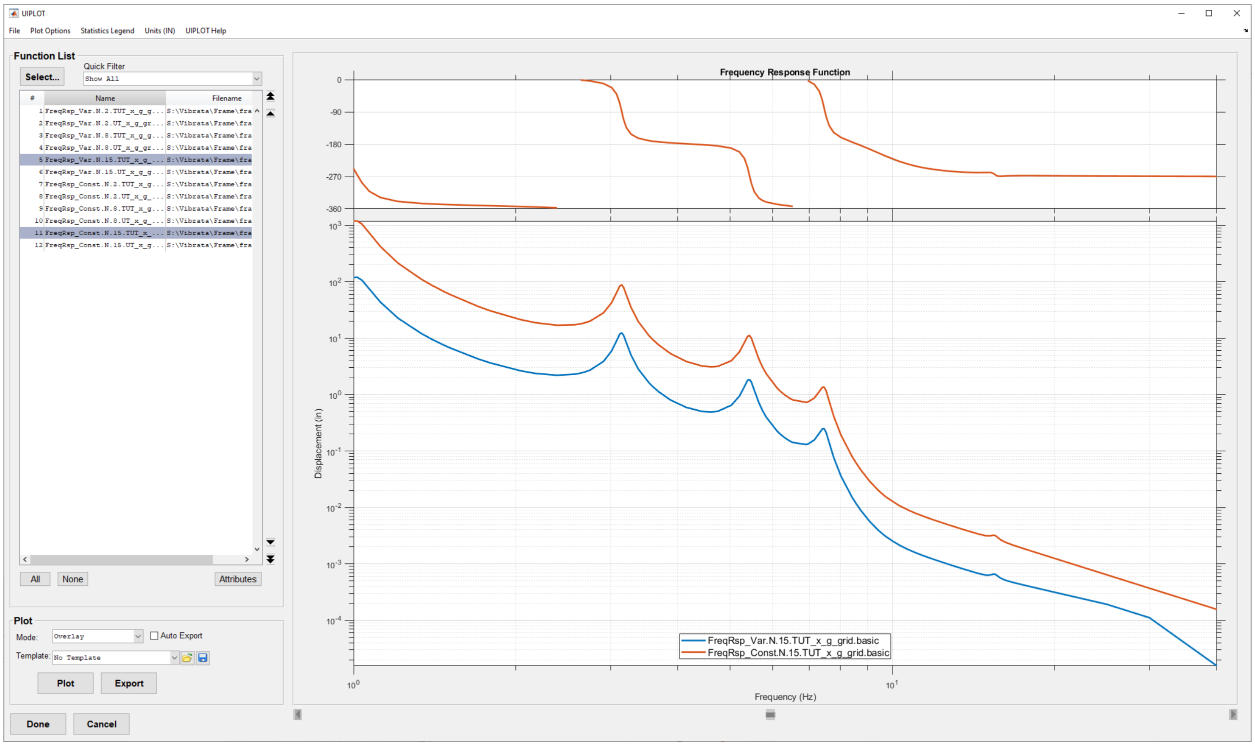


Figure 4-36. Compare the responses of the top corner node for the two inputs.

4.3. <u>Random Analysis</u>

For this example, you will use both of the Isat launch models. You will start with the single-point base model and then repeat the analysis with the four-point base. With the four-point base, you will use both correlated and uncorrelated input to see the effects of correlation, and to illustrate the correlation definition dialog. Finally, you will return to the single-point model and repeat the analysis, having deactivated any modes with no significant modal effective mass in the excitation direction.

4.3.1. <u>Create Common Acceleration PSD.</u>

All examples in this section will use the same excitation function. You can create it before you create any events.

1. In the Event Manager, access the Function Manager via the *Functions* menu.

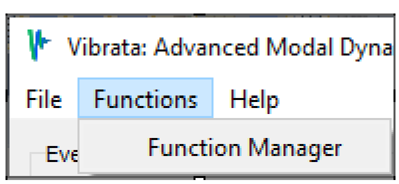


Figure 4-37. Access Function Manager directly from Event Manager menu.

2. In the Function Manager, click **New** to bring up the *New Function* dialog. As shown in Figure 4-38, set the *Function Type* to *PSD*, the *X Spacing* to *Uneven*, and the number

of points to 6. Key in the frequencies and amplitudes as shown. Set the *Y-Axis Type* to *Acceleration (EU)* so the Y-values will be in engineering units, in this case $(in/s^2)^2/Hz$, rather than g^2/Hz. The *Interpolation Type* must be *LogLog*. As always, give the function a name that will help you recognize it when you want it for later use.

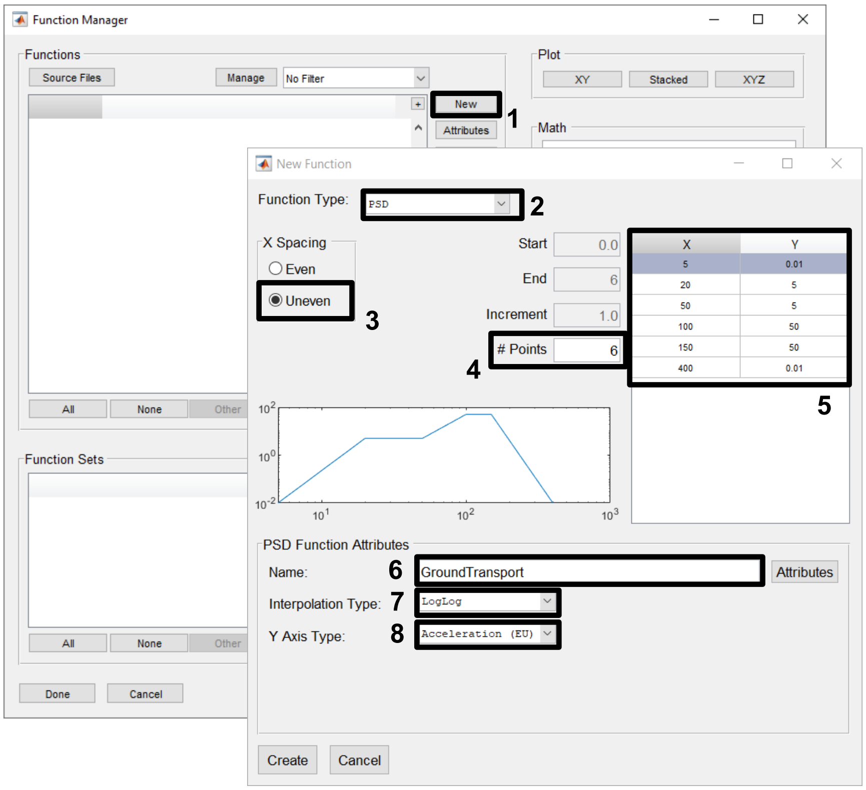


Figure 4-38. Create the acceleration PSD that will be used to excite the Isat.

3. When all the data and attributes are set correctly, click **Create** on the *New Function* dialog and save the function as shown in Figure 4-39. The new function will appear in the main Function Manager dialog, ready for use in the Vibrata events you will create in the following examples. Click **Done** to return to the Event Manager.

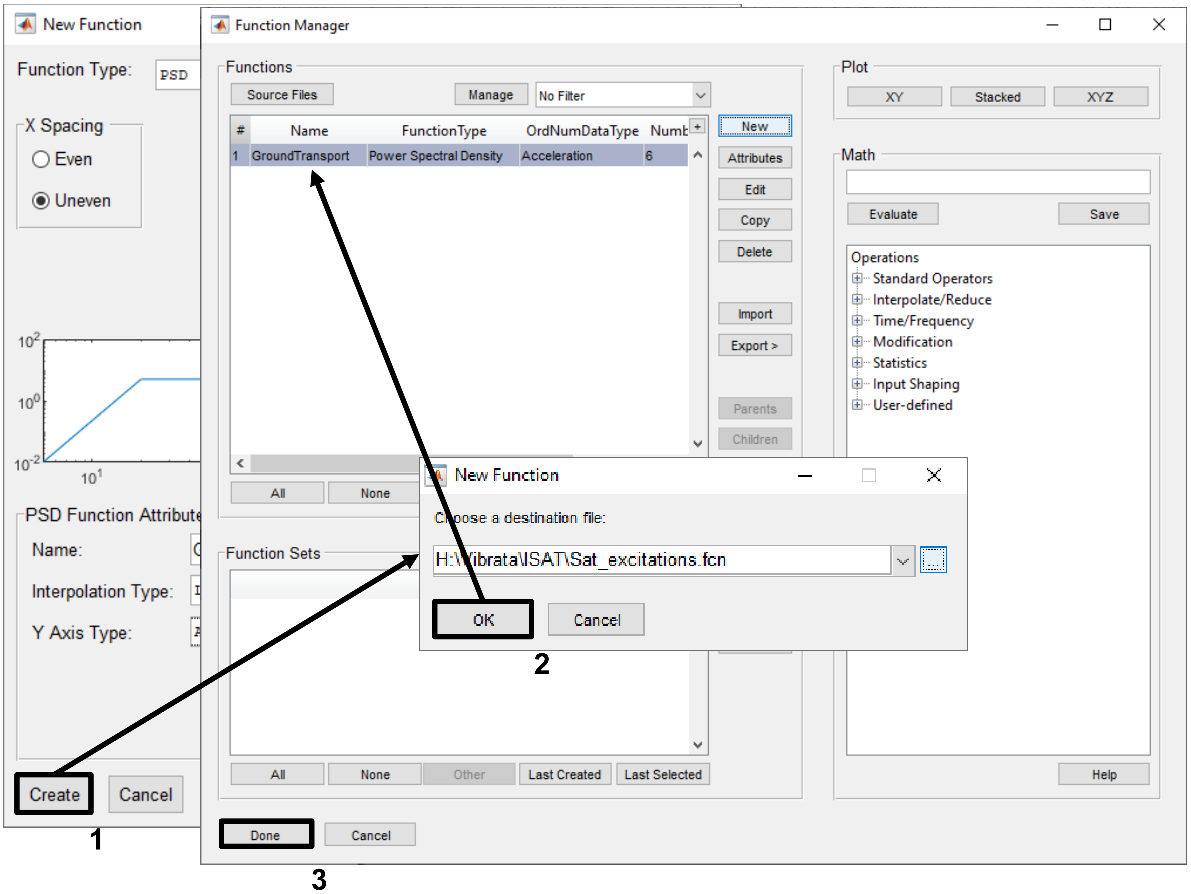


Figure 4-39.Create and save the function, and finish by clicking **Done**.

4.3.2. Isat with Rigid Base.

1. In Femap, import model "Isat_Launch_Sm_Rgd.dat" (Figure 4-5), and the neutral file containing its groups "Isat_Launch_Sm_Rgd_groups.neu". Save as "Isat_sm_Launch_Rgd.modfem."
2. Start a new event using the previously created modfem for the FEM and "isat_launch_sm_rgd.op2" for the results. Set the *Solver/Analysis Type* to *Modal Random*. Be sure to enter a recognizable event name (Figure 4-40).

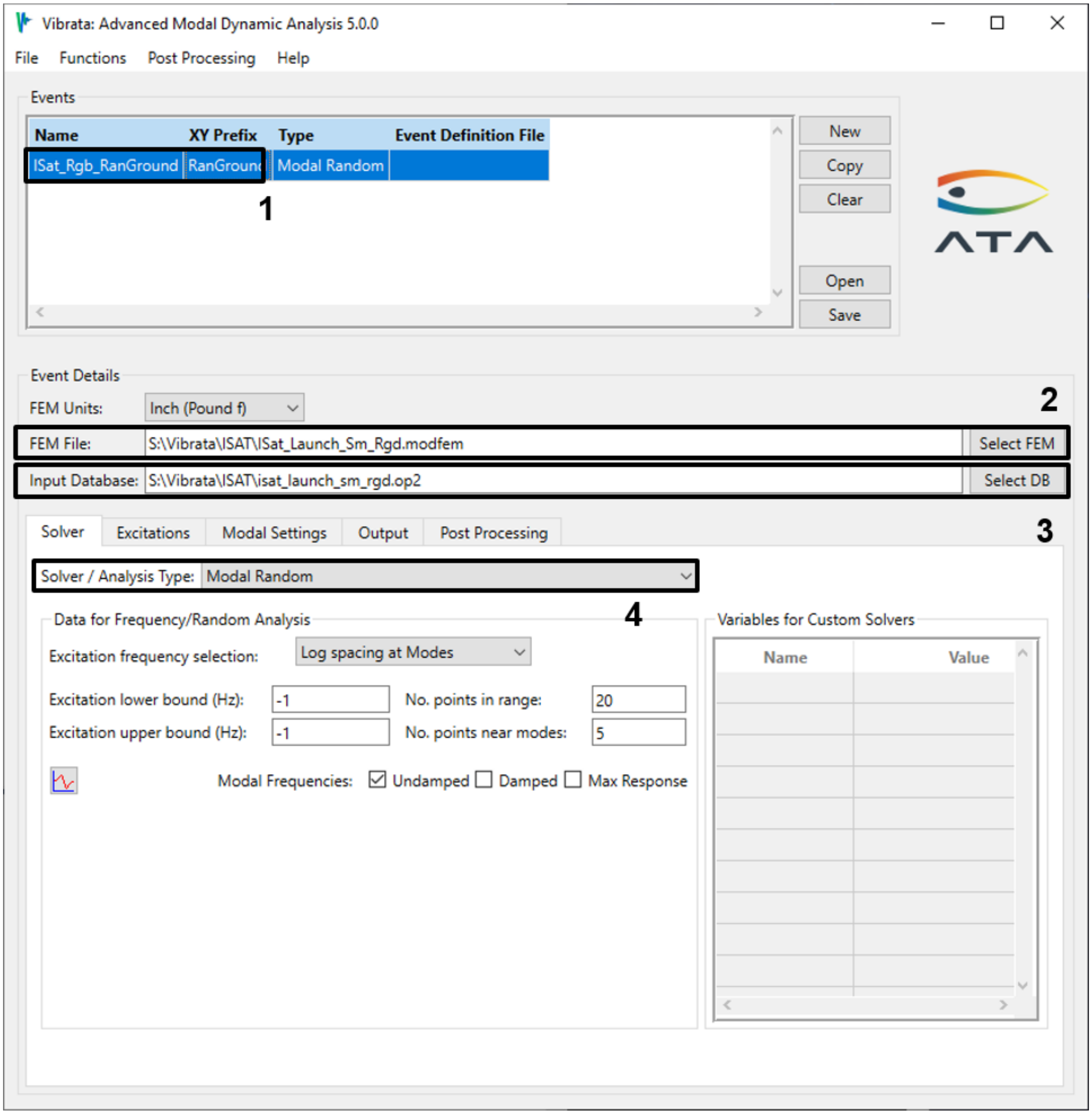
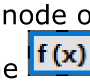


Figure 4-40. Create a new random analysis event.

3. The excitation for this event is enforced acceleration in the Z-direction, applied at the base of the satellite. Define it as follows.

   a. From the *Excitations* tab, bring up the *Enforced Motion* dialog (Figure 4-41). The only DOF available are those requested on the USET U2 card in the Nastran input file, which in this case are the translations of node 6000, the independent node of the base rigid element. Select Z-translation for excitation and then click the 

   button to bring up the Function Manager.

b. In the Function Manager, select the PSD function created above and click **Done**.

c. Uncheck the “Use FastRMS” checkbox for this example.

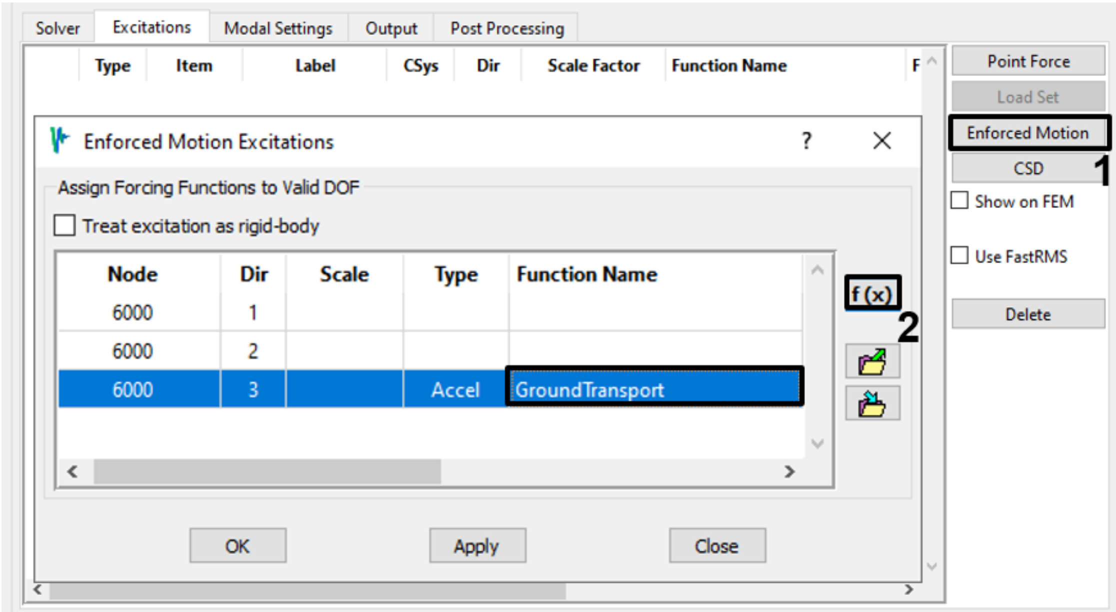


Figure 4-41. Select enforced motion, select Z-translation, and then bring up the Function Manager.

4. Since the Nastran solution requested residual vectors, make sure the modes above 400 Hz are marked appropriately. From the *Modal Settings* tab (Figure 4-42), select the modes above 400 Hz and then click on the **Residual** button. These modes will change to have a light-blue background, indicating that they are marked as residual vectors.

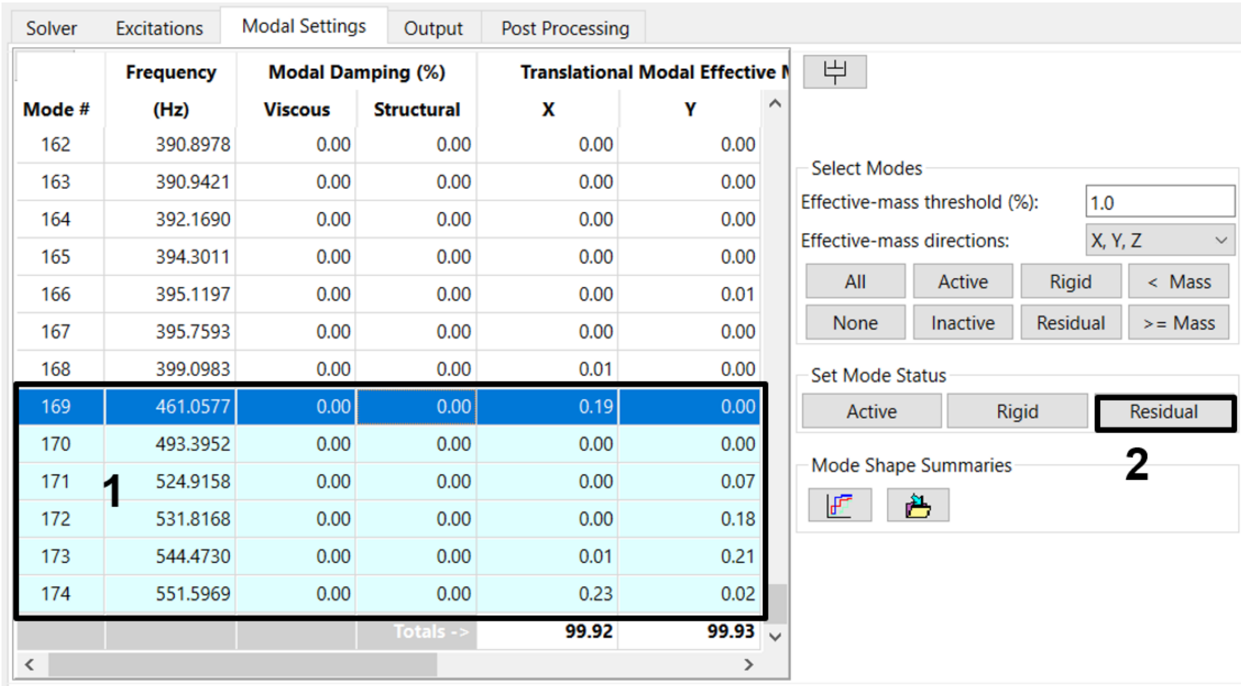


Figure 4-42. Select the modes above 400 Hz and set their status as residual vectors.

5. Using the *Viscous Damping Schedule* on the *Modal Damping Definition* dialog, assign 1% damping for modes up to 100 Hz and 2% for modes above 100 Hz.

   a. From the *Modal Settings* tab, bring up the *Modal Damping Definition* dialog (Figure 4-43).

   b. In the Viscous Damping Schedule, enter a frequency of 100.0 and a damping value of 1. Next, enter a frequency of 100.01 and a damping value of 2.

   c. Click on the blue triangle to apply this damping schedule.

   d. In the Function Manager, select the PSD function created above and click **Done**.

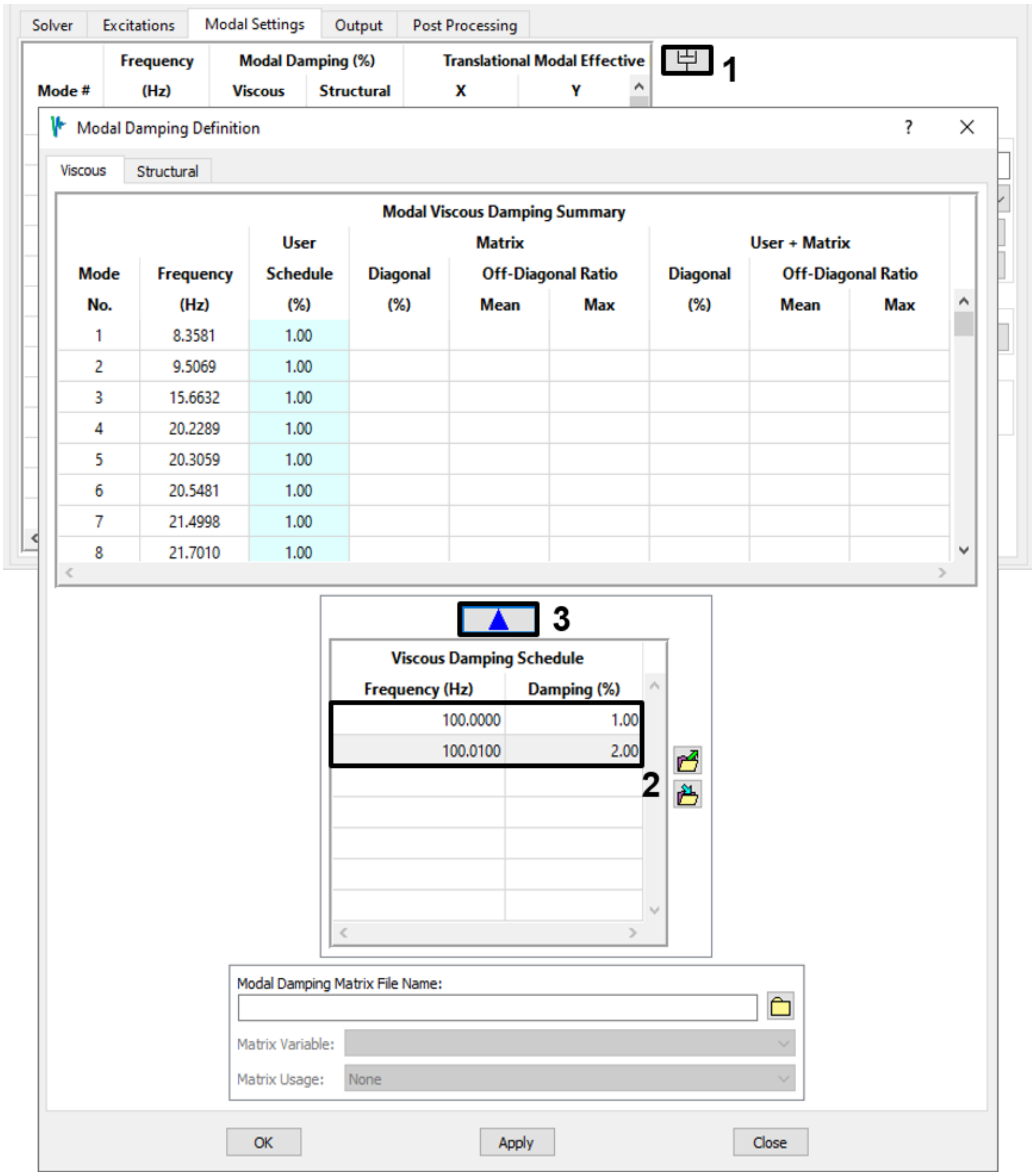


Figure 4-43. Assign damping via the schedule; export the schedule to a text file for later use.

The Instrumentation Package contains scientific instruments that may be damaged by excessive vibration, and we want to be sure the satellite bus will not be overstressed. Click the **Contour** button on the *Output* tab to open the *Contour Requests* dialog. Request Von Mises stress (*SVMS*) for the BUS_SHELLS group, click **OK** to close the group selection dialog. In the *Output Intervals* panel, make sure that only *RMS* output is checked. Click **Apply** to create the output request. Next, turn on the *Select nodal variables* toggle and request total translational acceleration (*TAT*) for the INST_PKG_FEM group (Figure 4-44). Click **OK** to close the output request form. Back on the *Output* tab, click the **Solve** button. Even with FastRMS turned off, the solution does not take long.

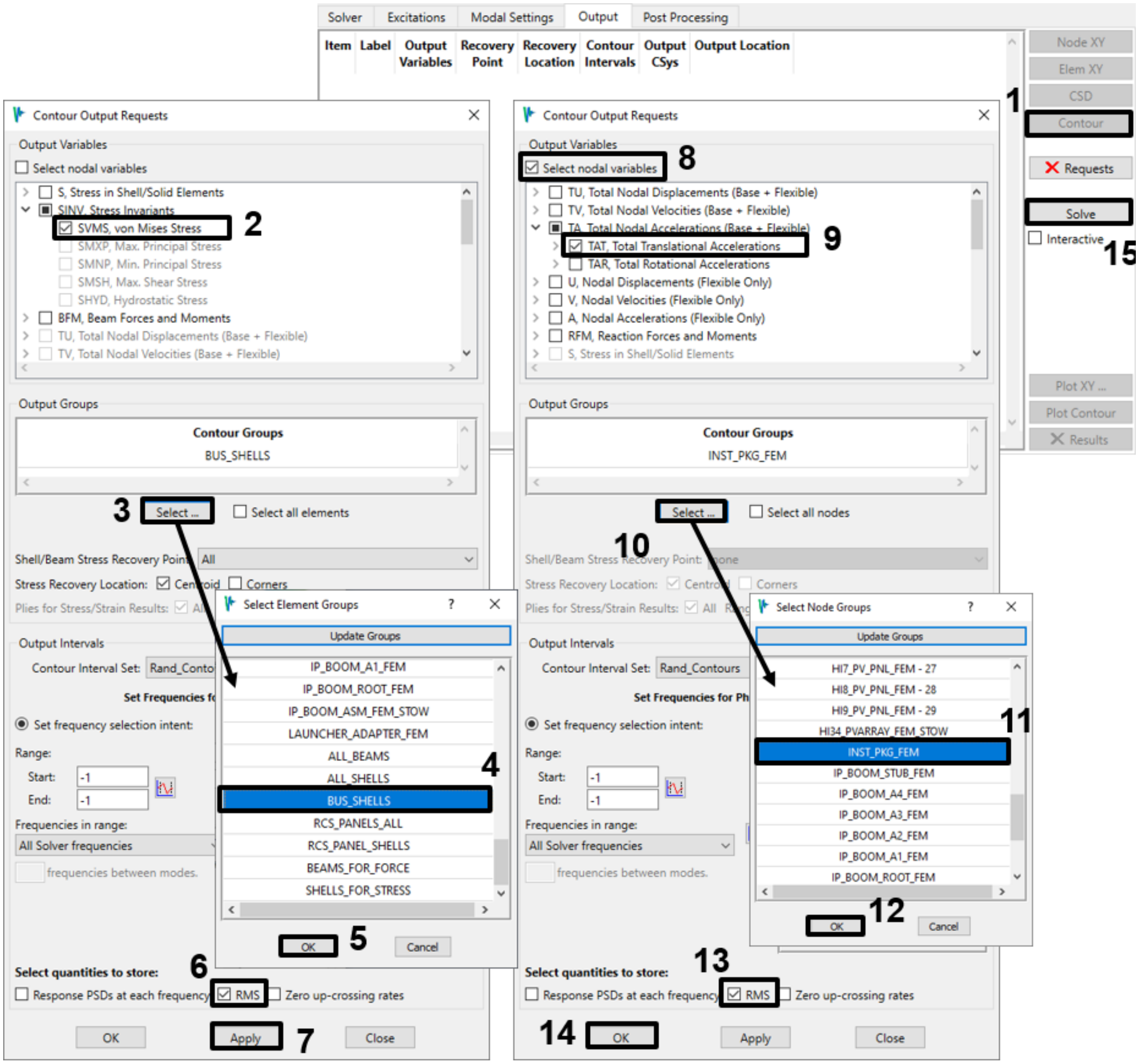


Figure 4-44. Request *SVMS* for the BUS_SHELLS and *TAT* for the INST_PKG_FEM; only *RMS* output.

6. When the solver has finished, click the *Output* tab's **Plot Contour** button to examine the results in Femap. Use Femap's  toolbar button to open the *Select PostProcessing Data* dialog (Figure 4-45). Select *Plate Top von Mises* from the *Contour* pulldown. Click **OK** to display the contours.

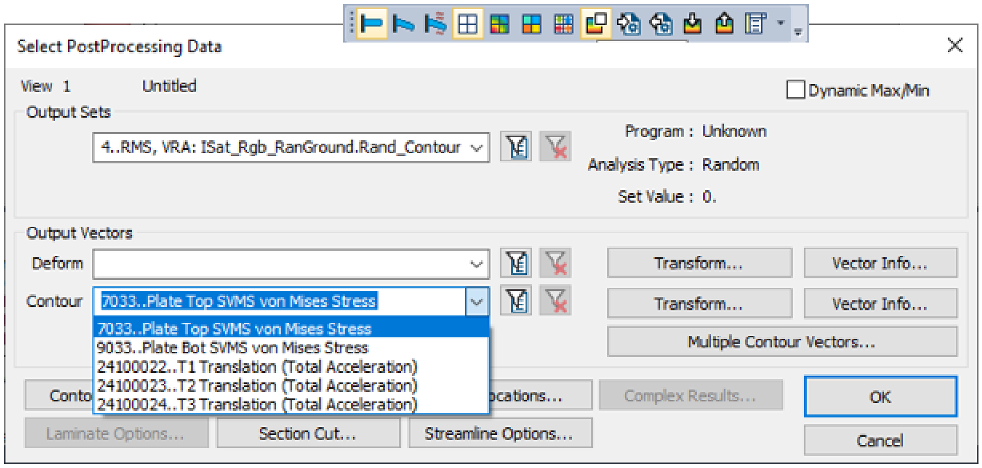


Figure 4-45. Select von Mises Stress for the first contour plot and turn off averaging.

7. Since you only calculated stresses for the BUS_SHELLS group, the best way to view them is to activate that group and set Femap to display only the active group (Figure 4-46). With all the desired settings now defined for contour plots, you can use Femap's  toolbar buttons to select the next or previous data component for display. When you get to the translational accelerations, you should activate the INST_PKG_FEM group (Figure 4-47).

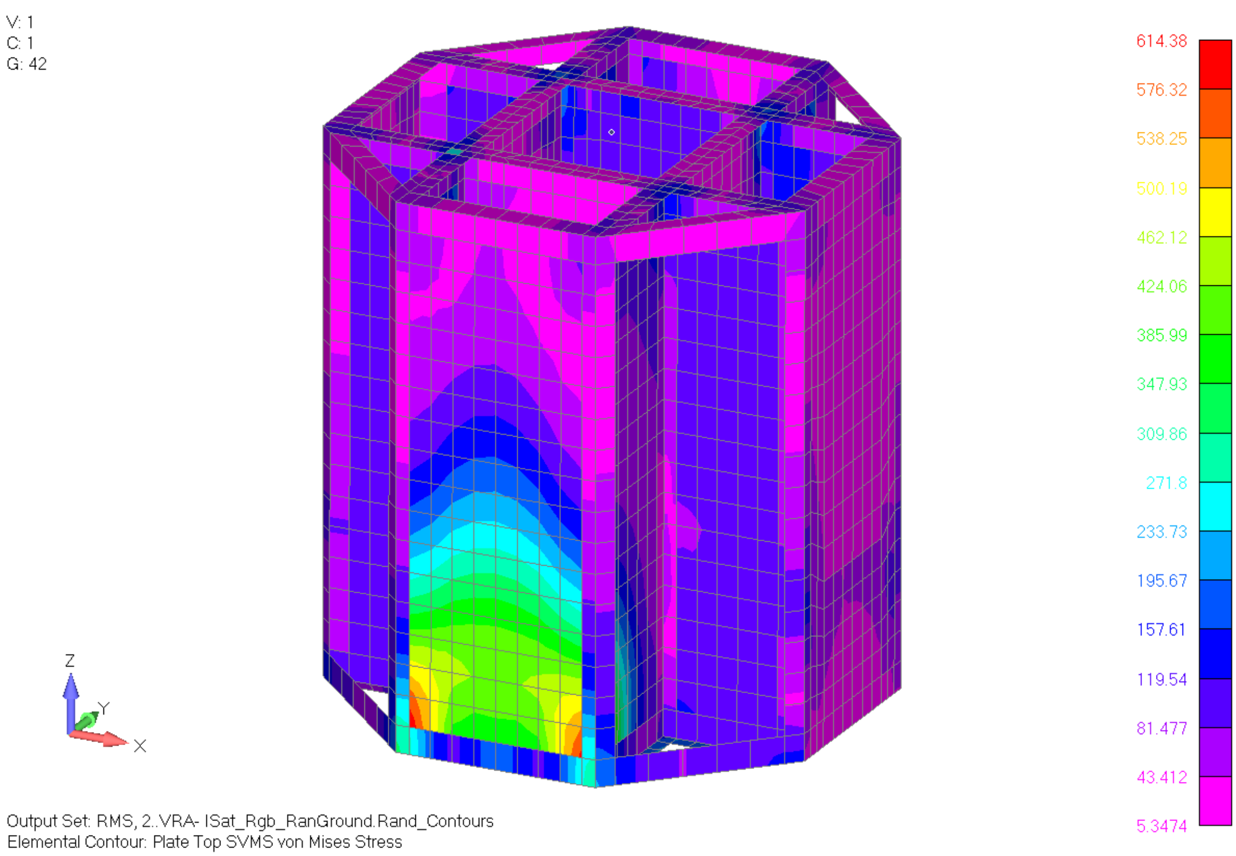


Figure 4-46. RMS von Mises stress in satellite bus shell elements, rigid base.

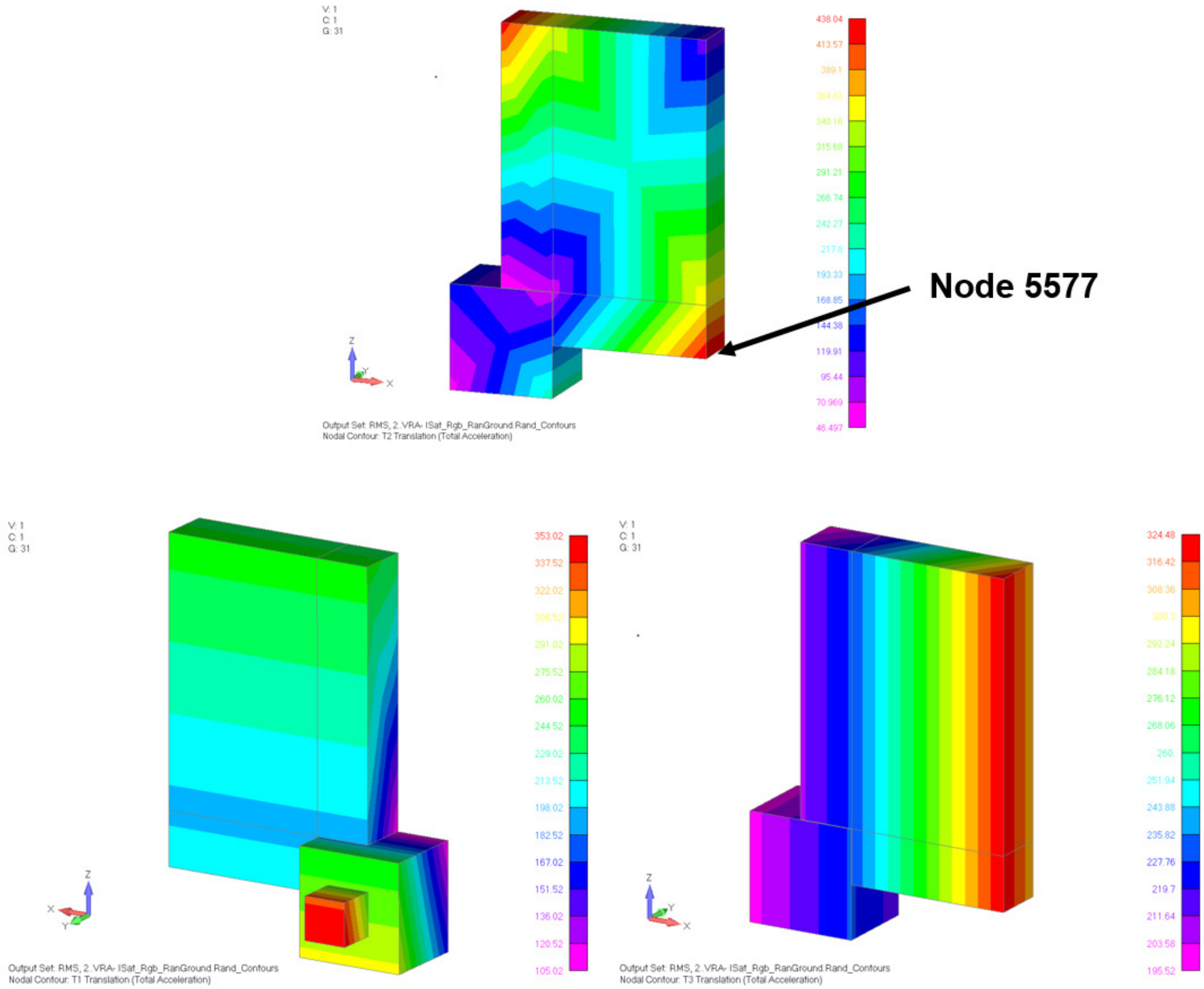


Figure 4-47. RMS acceleration contours for the instrumentation package, rigid base.

8. The highest RMS acceleration occurs at node 5577 for Y-acceleration. That node also shares the highest Z-acceleration, although it is lower than the Y-direction. You would like to know the frequencies at which the largest responses occur, so create a Node XY output request for that node and those directions (Figure 4-48). (You can request X-acceleration if you wish, but it will clutter the plotting displays.) Solve this new request and plot the Y-acceleration function (Figure 4-49). Set the display units to “IN (G’s)” to see the responses in g^2/Hz rather than $(in/s^2)^2$/Hz.

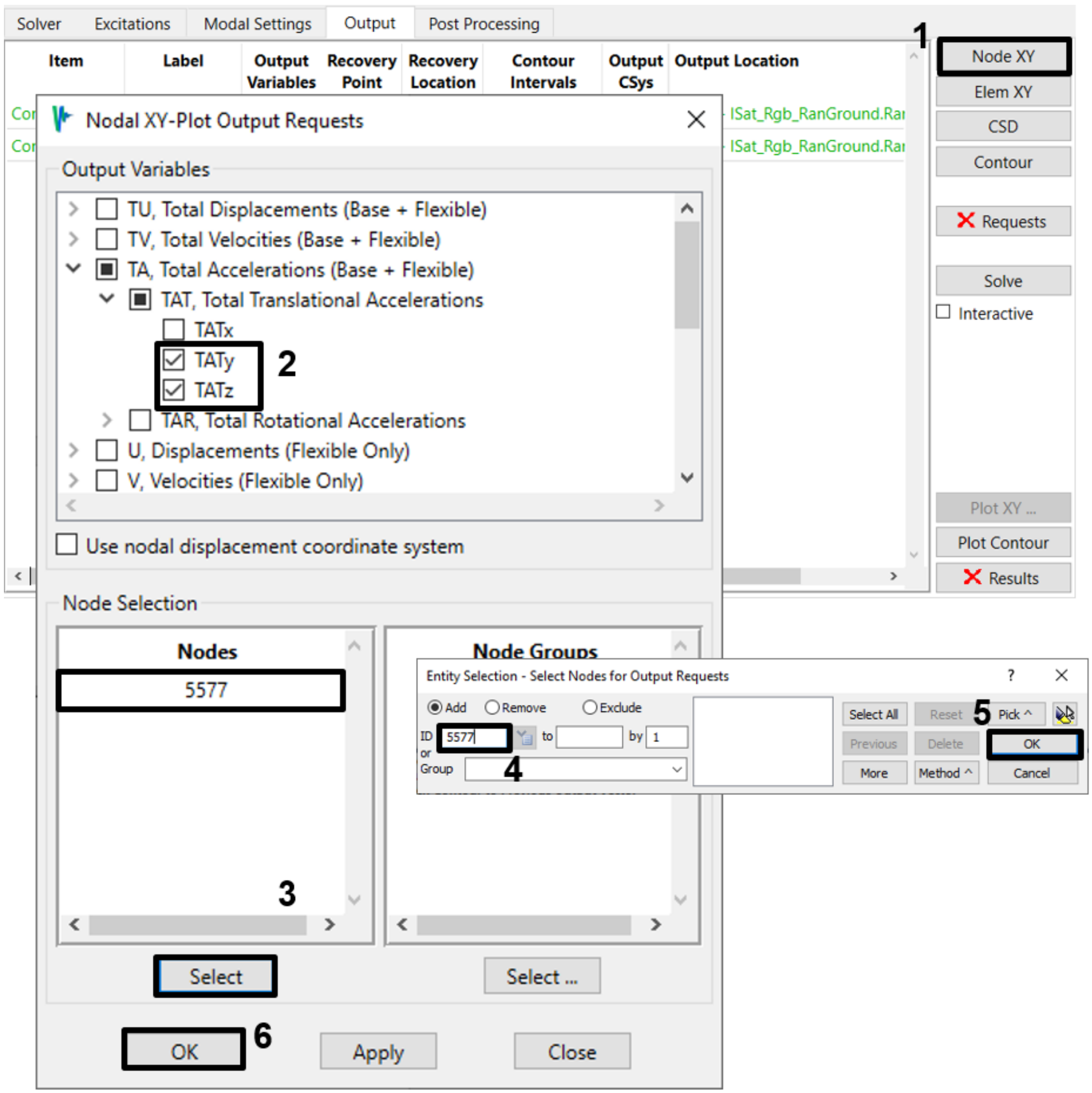


Figure 4-48. Request Y- and Z-direction total accelerations for node 5577.

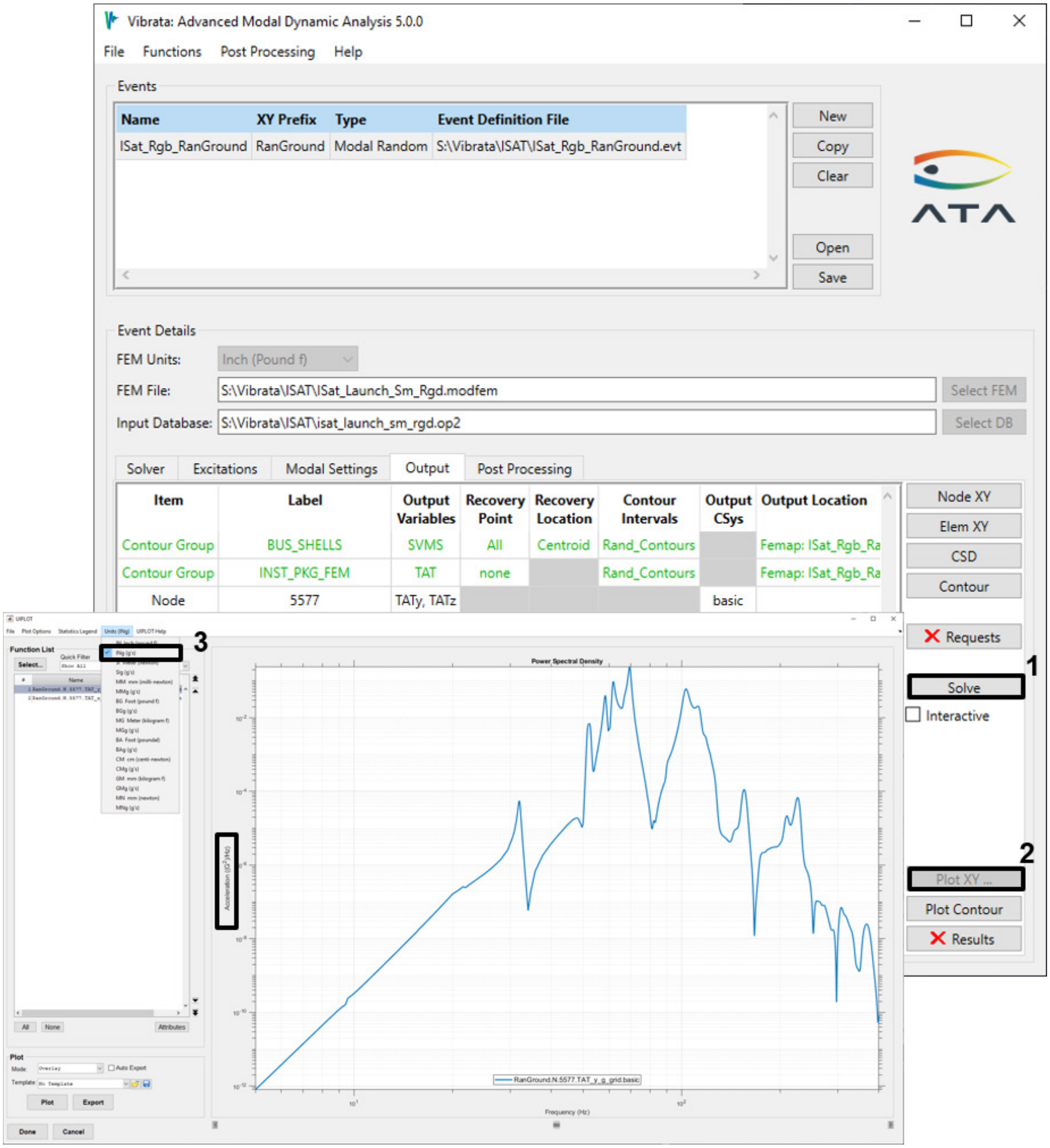


Figure 4-49. Node 5577 total Y-acceleration in g^2/Hz units, rigid base.

9. It is also important that we not exceed allowable loads in the launcher adapter legs. Request axial force responses for the elements in group "LAUNCHER_ADAPTER_FEM" (Figure 4-50). Solve and plot (Figure 4-51). The RMS values, along with other statistics about the functions, can be displayed using the *UIPLOT* dialog's *Statistics Legend* menu. Note that element 5632 has the largest RMS value.

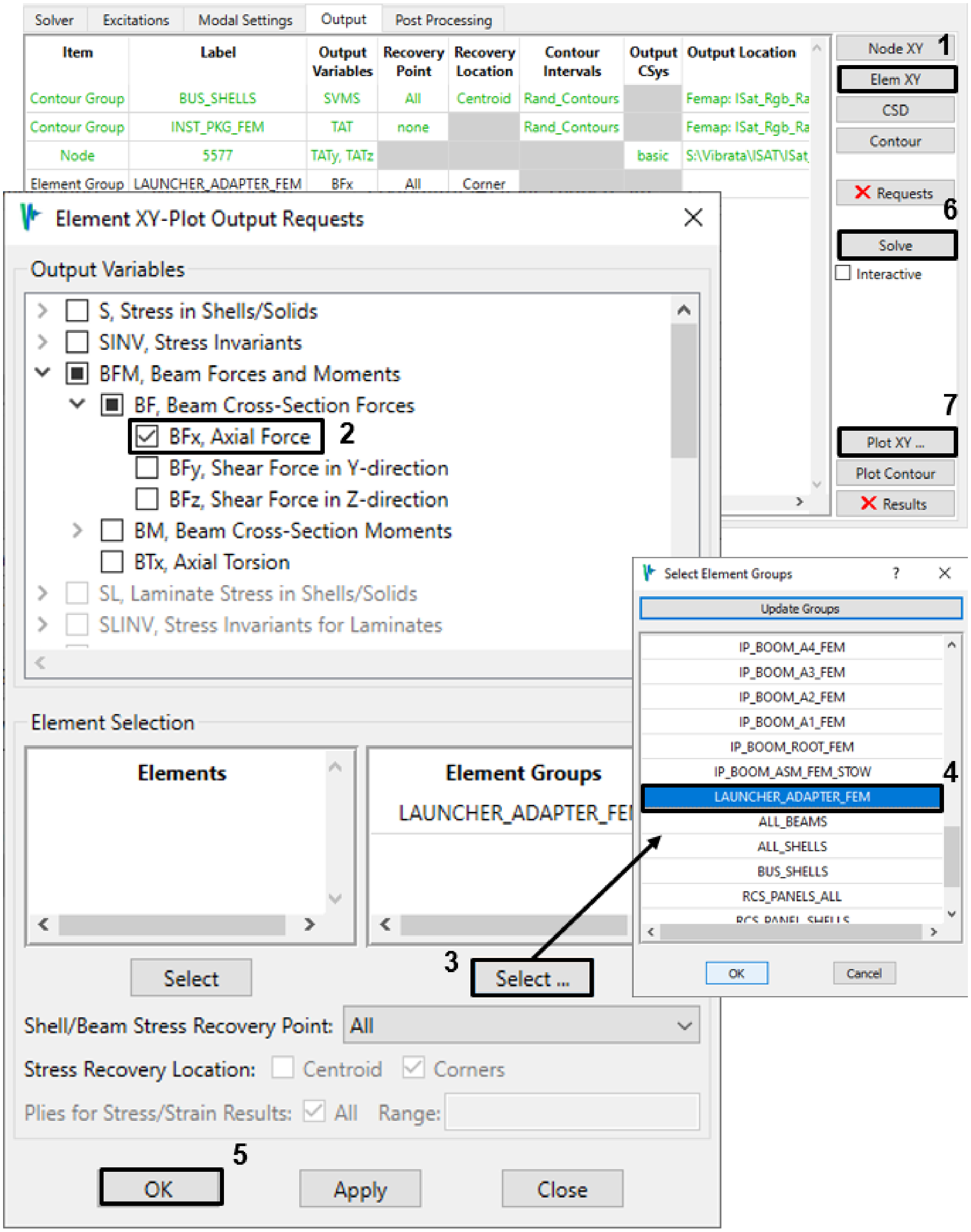


Figure 4-50. Request axial forces in the launcher adapter legs.

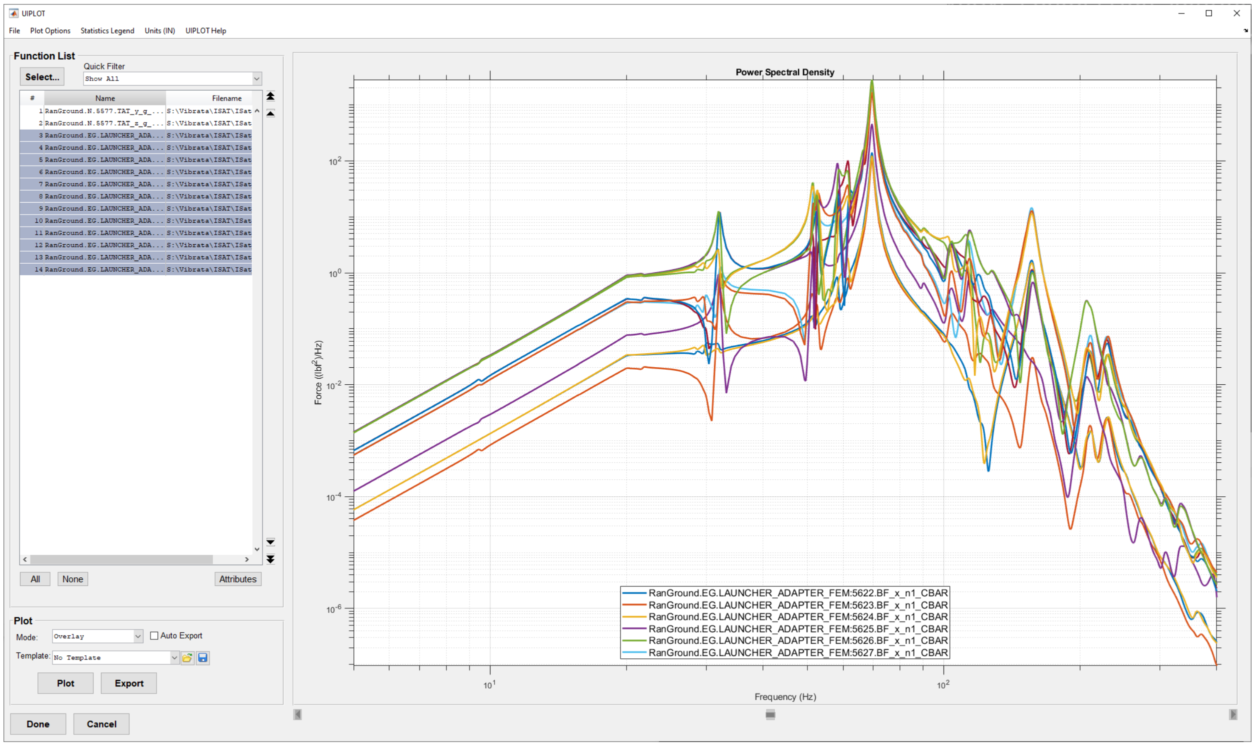


Figure 4-51. Axial forces in the launcher adapter legs, rigid base.

4.3.3. Isat with Uncorrelated Input at Each Adapter Leg.

1. In Femap, import the launch model with four independent base nodes "Isat_Launch_Sm_4pt.dat" (Figure 4-4), and groups "Isat_Launch_Sm_4pt_groups.neu". Save the model as "Isat_sm_Launch_4pt.modfem".
2. In the Event Manager, copy the event from section 4.3.2 using the **Copy** button and then assign a new name to the new event (Figure 4-52). Set the XY-Plot Prefix to "RanGround_uncorrelated" and change the event name to "Isat_Rgd_RanGround_uncorrelated". Use the **Select FEM** button to select the modfem file for the four-point model. Use the **Select OP2** button to select the OP2 with the same name. You will see a dialog warning of possible changes to excitations and output requests for the new FEM; confirm that you want to make the change. The two events will be identical in their modal settings and output requests, although the new event will have no results yet. The excitation function will also be the same as before, but we now have four input locations instead of only one, so we will have to redefine the excitations.

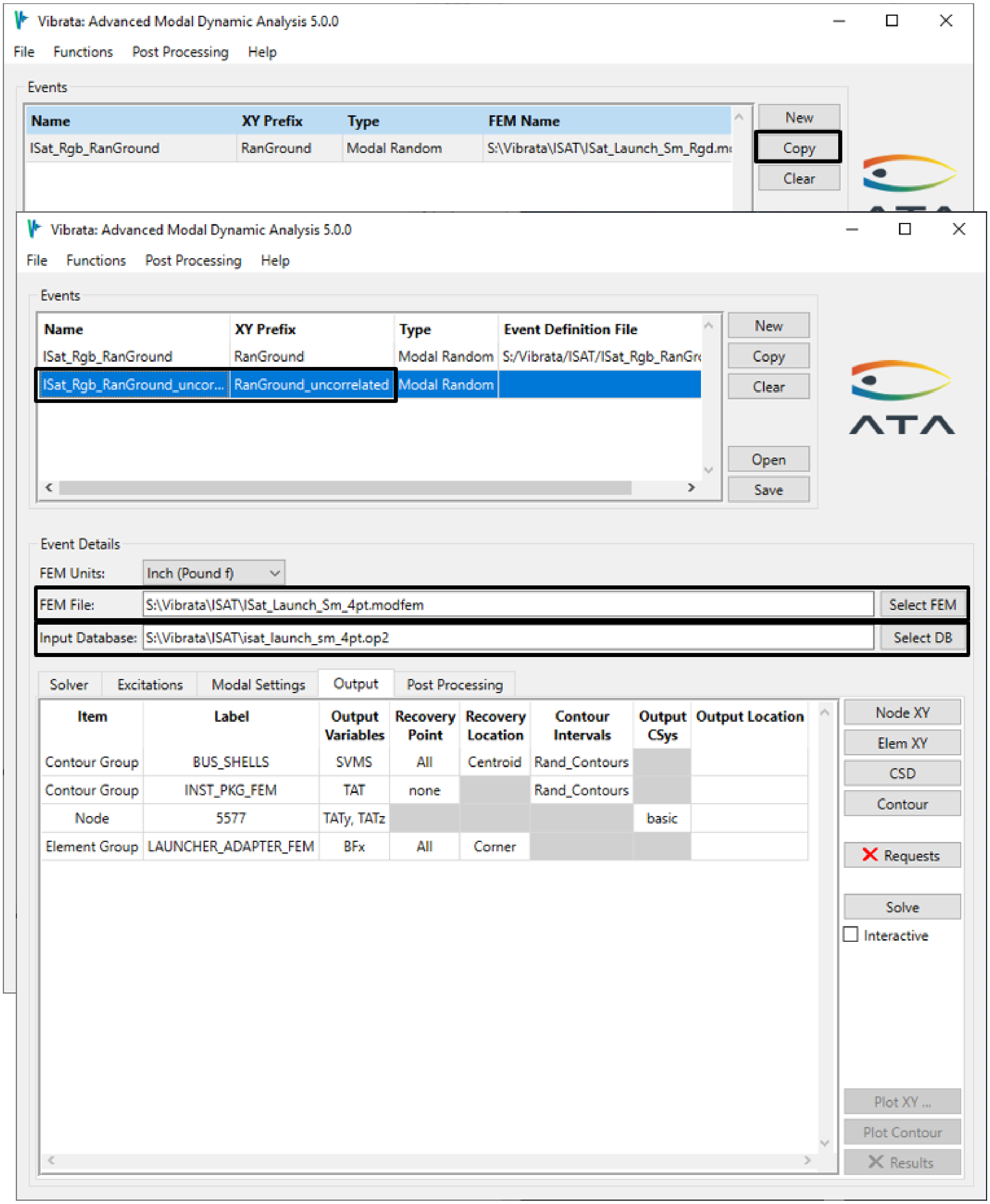


Figure 4-52. Use the **Copy** button to start a new event from the event just completed.

3. Go to the *Excitations* tab. First, select the existing excitation and **delete it**; the node at which it is applied does not exist in this model. Next, open the *Enforced Motion* dialog and select the Z-direction DOF for all four nodes in the list. Use the Function

Manager to assign the ground transport PSD function created in section 4.3.1 to these DOF (Figure 4-53).

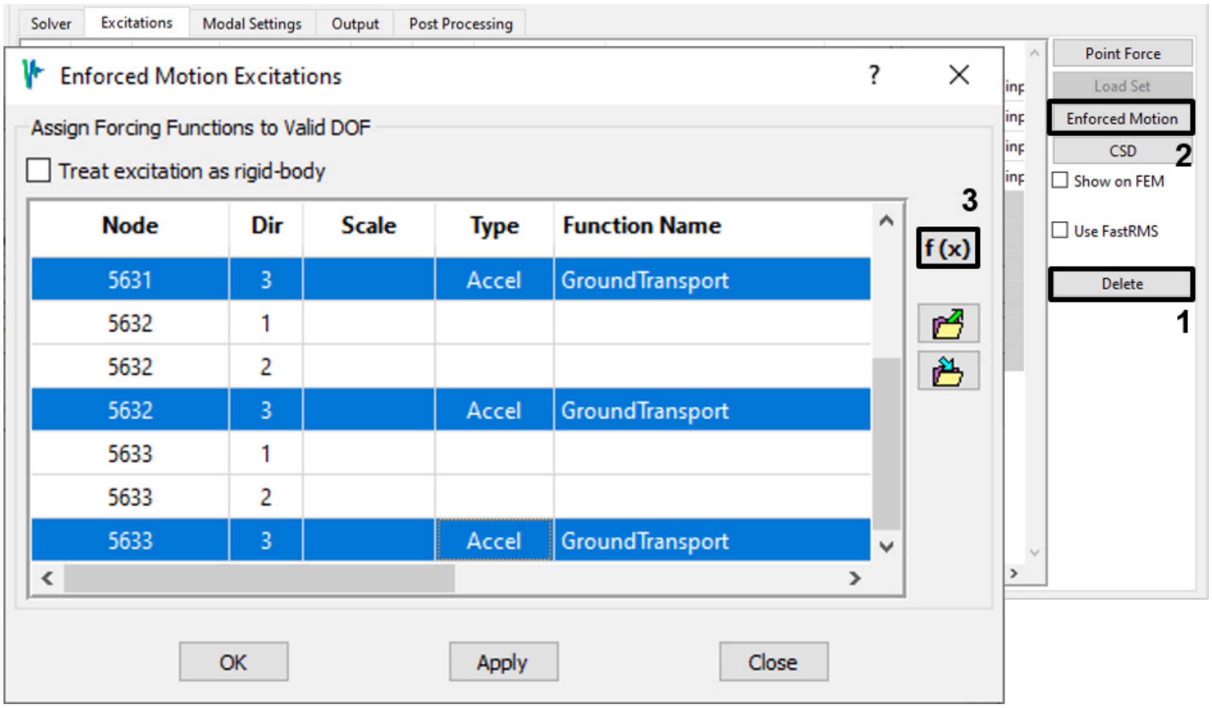


Figure 4-53. Apply enforced motion in the Z direction at all four base nodes.

4. Note that 0-phase CSD functions have been defined automatically to correlate these excitations. Ordinarily that is exactly what we want, but here we want to compare correlated and uncorrelated results, and we also want to illustrate how to use the CSD definition dialog. Therefore, delete these CSD functions (Figure 4-54). We will recreate them manually for the event in section 4.3.4.

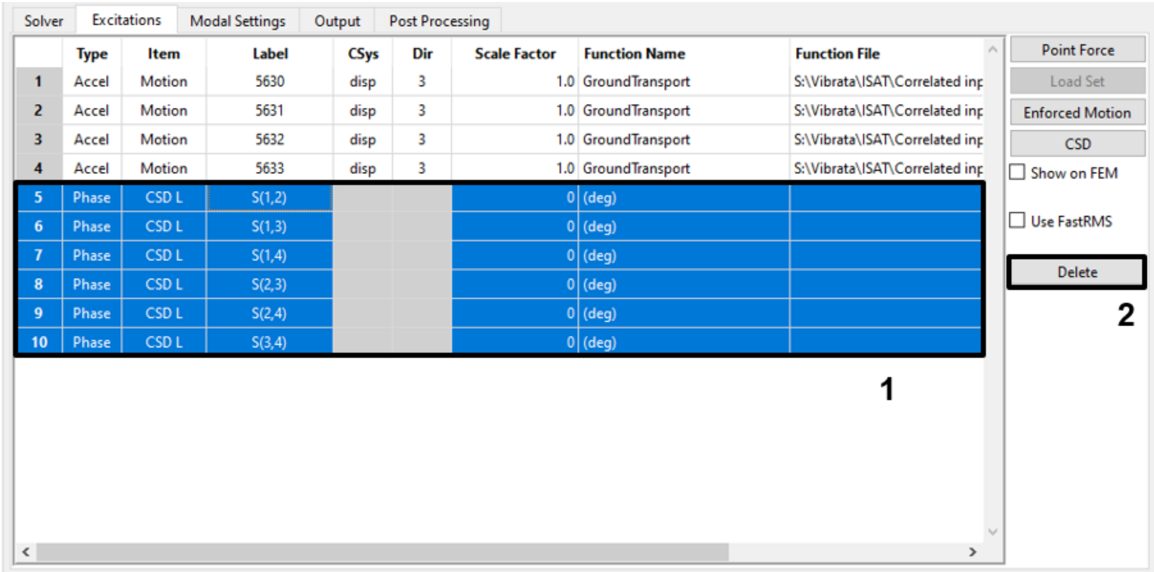


Figure 4-54. Delete the automatic CSD entries.

5. Go to the *Modal Settings* tab and verify that the modes still have damping per the damping schedule.
6. All other settings from the rigid-base model may be left as they are. Go to the *Output* tab and click **Solve**.
7. When the solver finishes, examine the RMS von Mises stresses in the bus (Figure 4-55) and the RMS accelerations in the instrument package (Figure 4-56) as before. The color bars in these figures are set to the same levels as in Figure 4-46 and Figure 4-47 respectively. The results are very different.

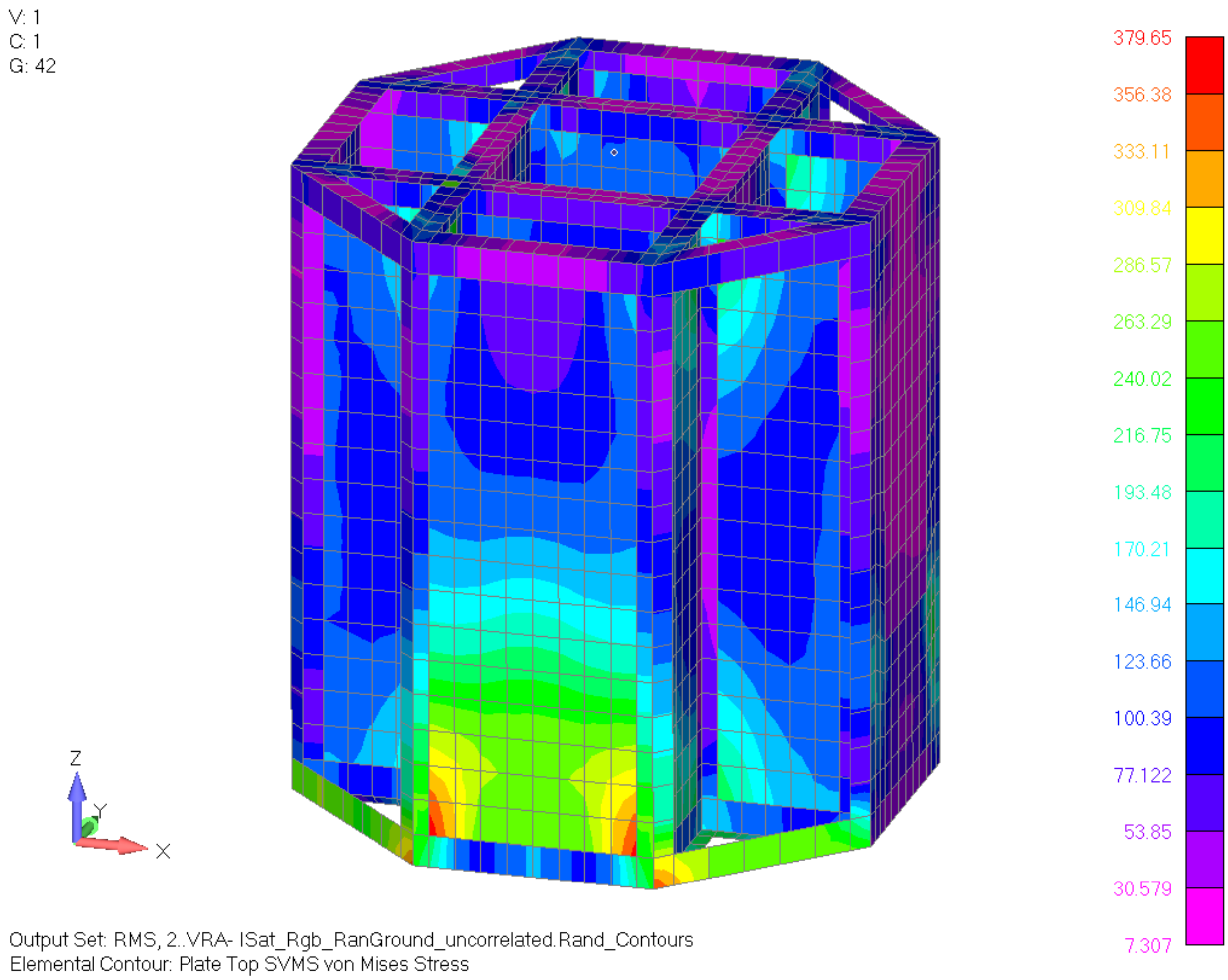


Figure 4-55. RMS von Mises stress in satellite bus shell elements, four-point base, uncorrelated.

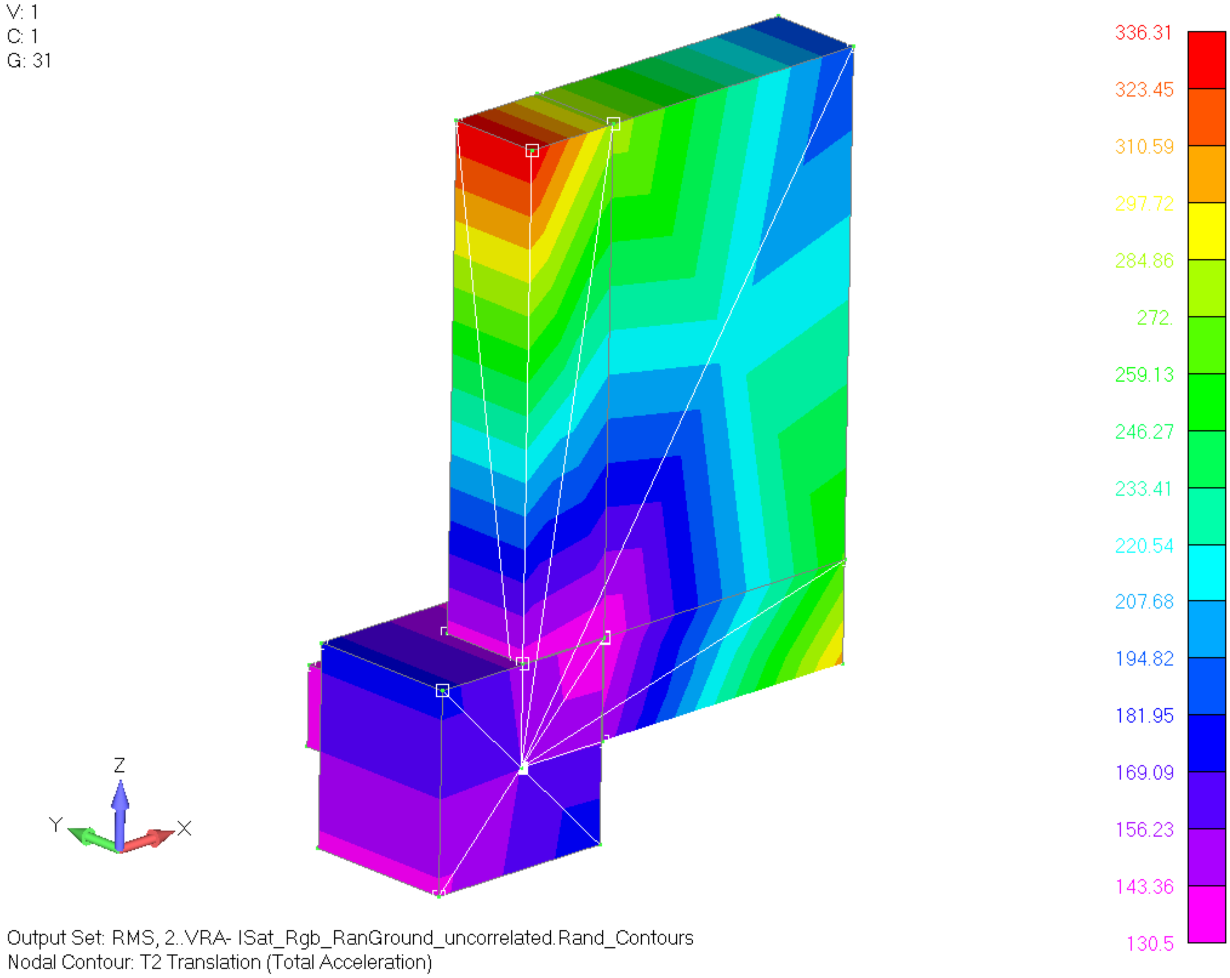


Figure 4-56. RMS acceleration contours for the instrumentation package, four-point base, uncorrelated.

8. Compare the Y-acceleration response PSD from this case to the one from the rigid-base event. On the *Output* tab, click the **PLOT XY...** button to bring up the *UIPLOT* dialog. It will show the functions for the current event, which is the four-point model with uncorrelated inputs. Use the *File / Load File* menu to make the results of the rigid-base event available (Figure 4-57). Plot the *TATy* function from both events (Figure 4-58). Again, they are clearly quite different.

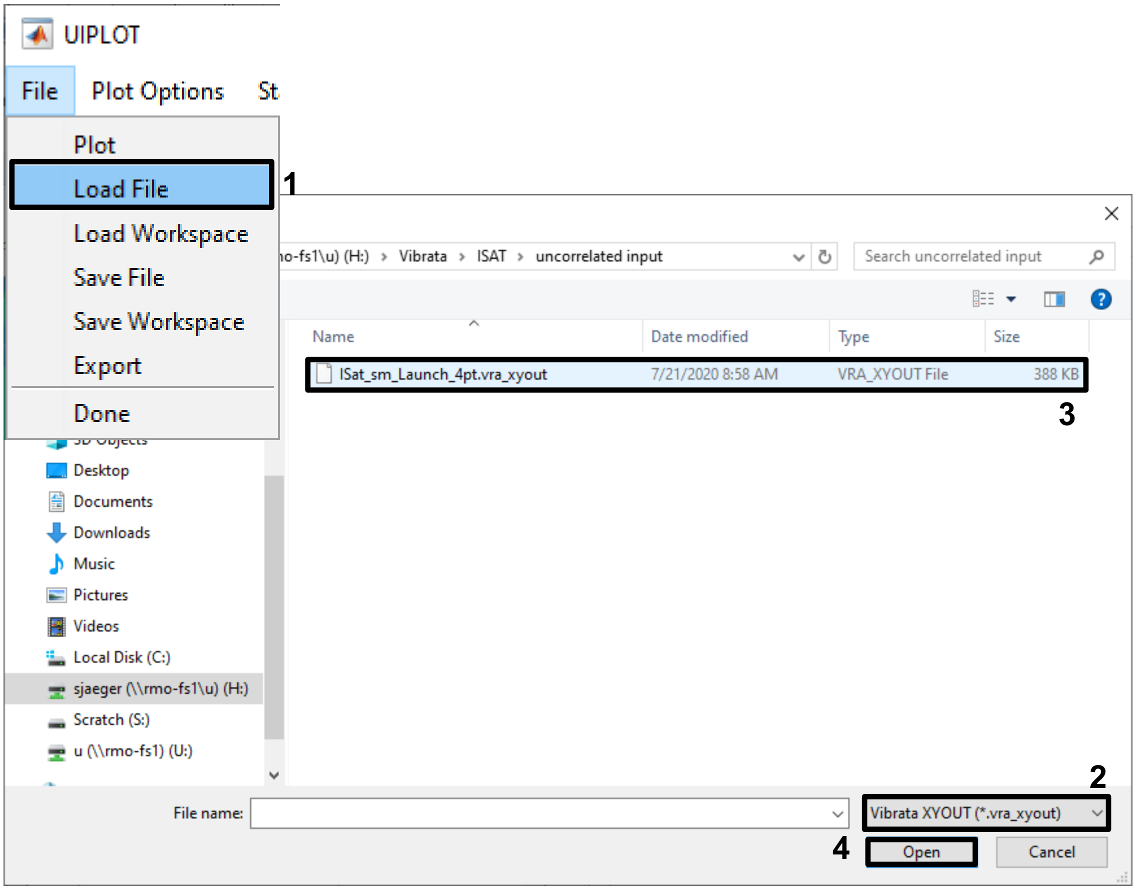


Figure 4-57. Add the rigid-base results to those available for plotting.

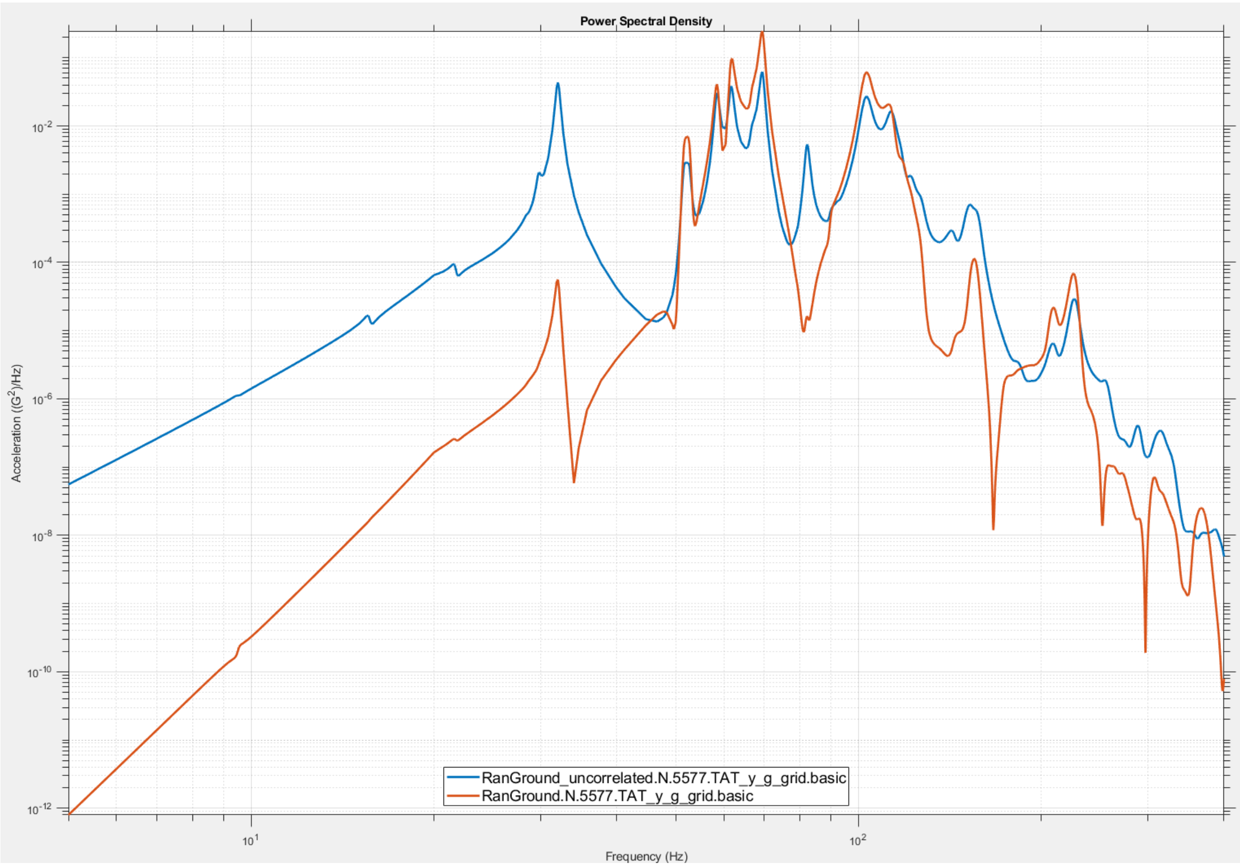


Figure 4-58. The Y-acceleration responses for the two events are very different.

4.3.4. Isat with Correlated Inputs.

1. Copy the uncorrelated event for the "4pt" model from example 4.3.3 and assign the still use the "4pt" model.

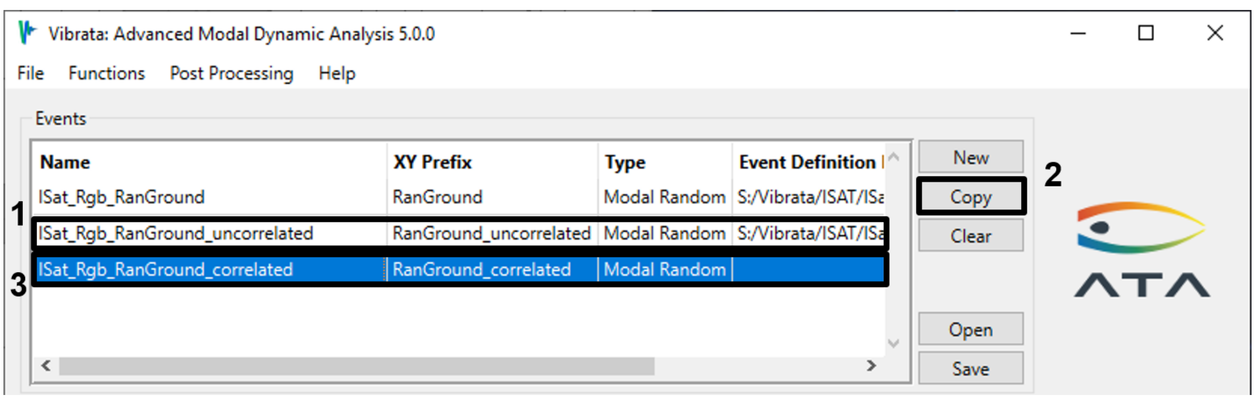


Figure 4-59. Copy the uncorrelated event; the new event will have correlated inputs.

2. On the *Excitations* tab, bring up the CSD dialog. Select the entire first row of the *CSD Matrix* table and click the **Assign Selected CSD** button  (Figure 4-60). This prepares the dialog to create correlations from the first input PSD to the other three PSDs. Make sure the **CSD Type** pulldown is set to *Phase Lag* and the **Phase/Time**

lag field is set to 0 degrees. When you click the **Apply** button, the defined CSDs are created and the *CSD Matrix* table is updated. Continue selecting rows, clicking the  button, and clicking **Apply** until the entire matrix is filled (Figure 4-61). When you click **OK** to close the CSD dialog, the CSDs appear as additional excitations on the *Excitations* tab (Figure 4-62). The new event is now ready to solve.

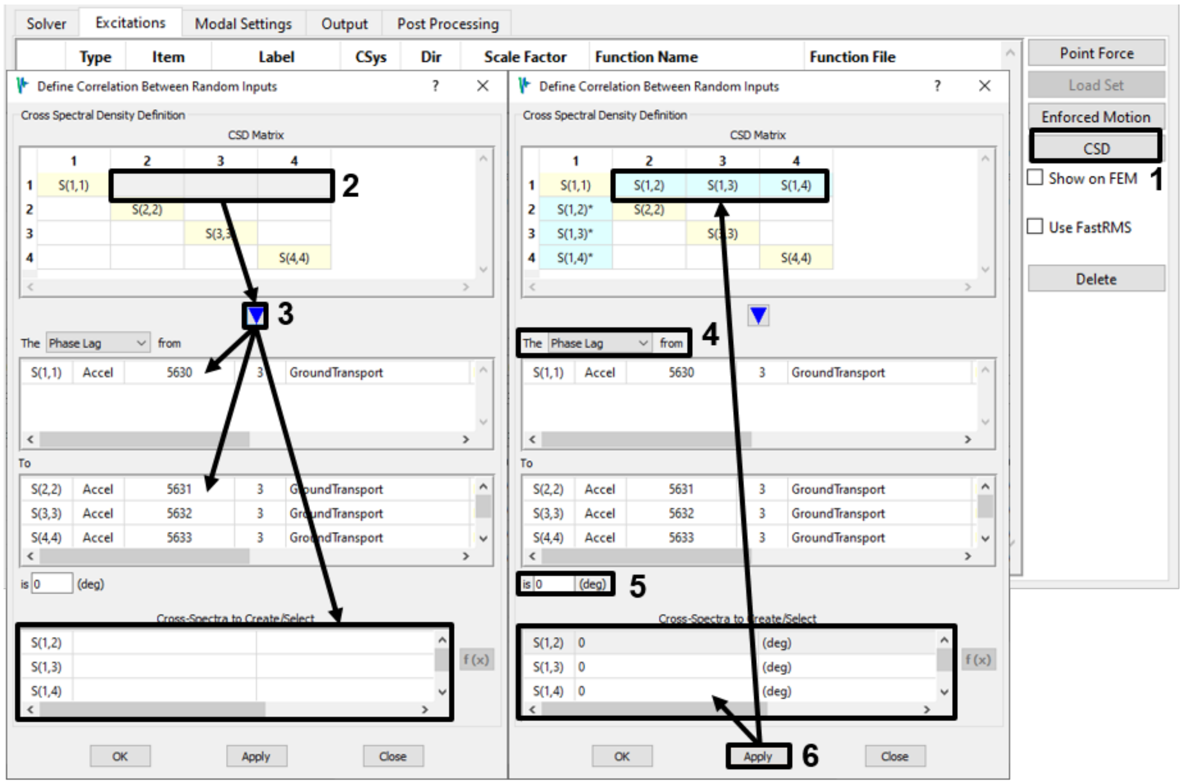


Figure 4-60. Define 0-degree phase lag correlations from the first input PSD to the other three.

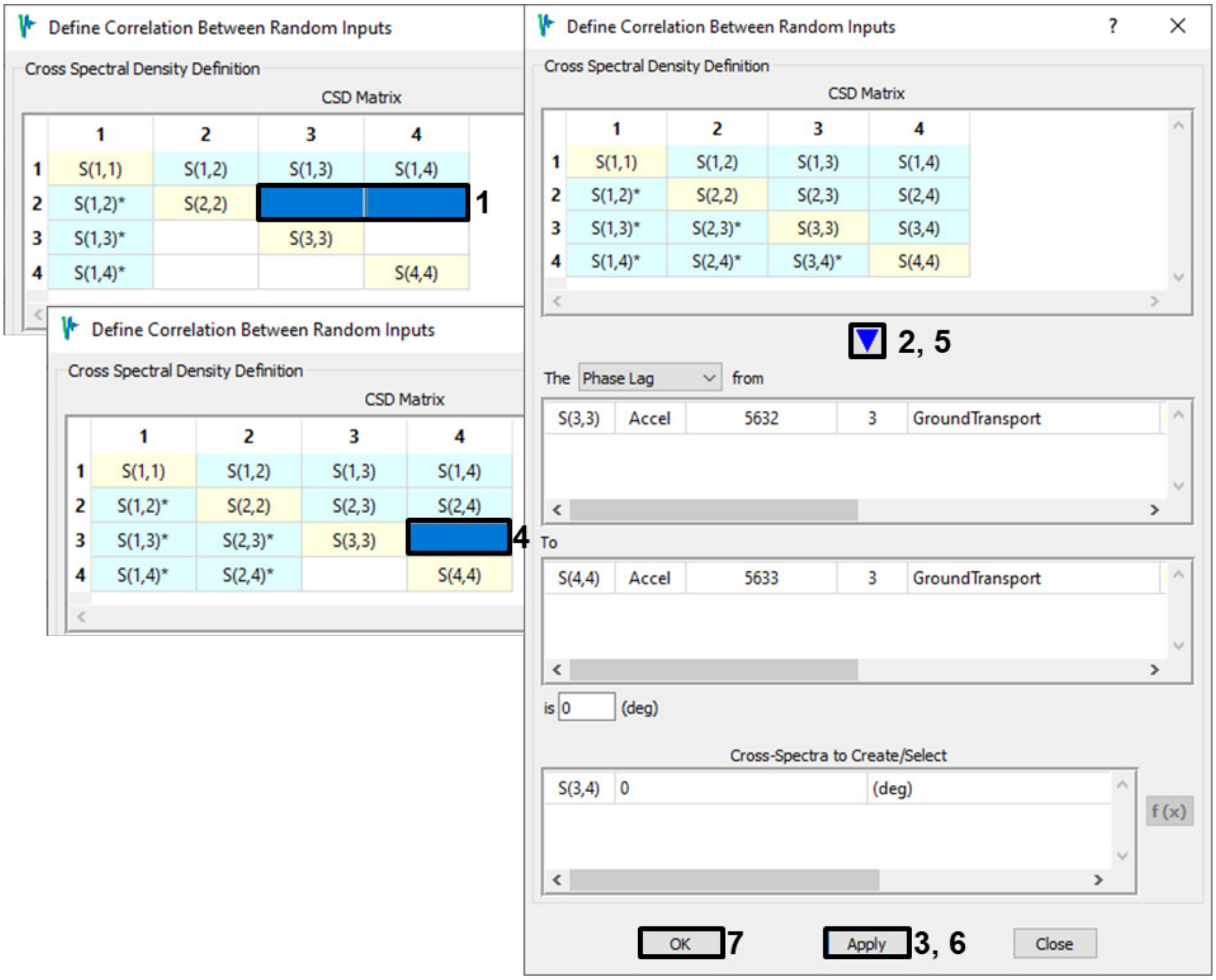


Figure 4-61. Finish defining 0-degree phase shift correlations among all the input PSDs.

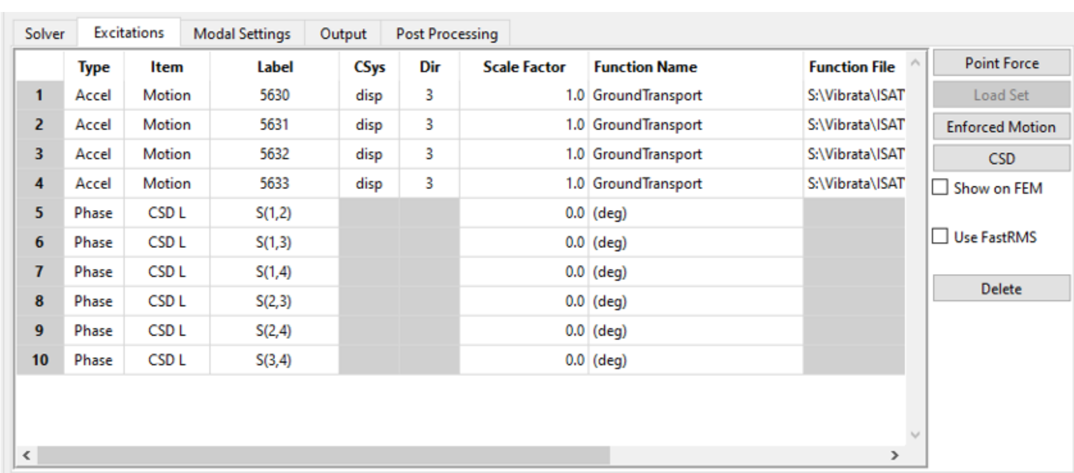


Figure 4-62. The *Excitations* tab will show the defined CSDs but not their conjugates.

3. Go to the *Output* tab and solve this new event. The RMS von Mises contours here (Figure 4-63) are identical to those from the rigid-base model (Figure 4-46), not

those from the uncorrelated four-point-base event (Figure 4-55). The same is true for the instrumentation package acceleration contours (Figure 4-64). Plotting the Y-acceleration response PSDs for all three events together confirms this: the blue curve for the four-point-base with correlated input is completed covered by the red curve from the rigid-base model, and those curves have identical RMS values (Figure 4-65). This holds for the launcher leg axial forces as well (Figure 4-66).

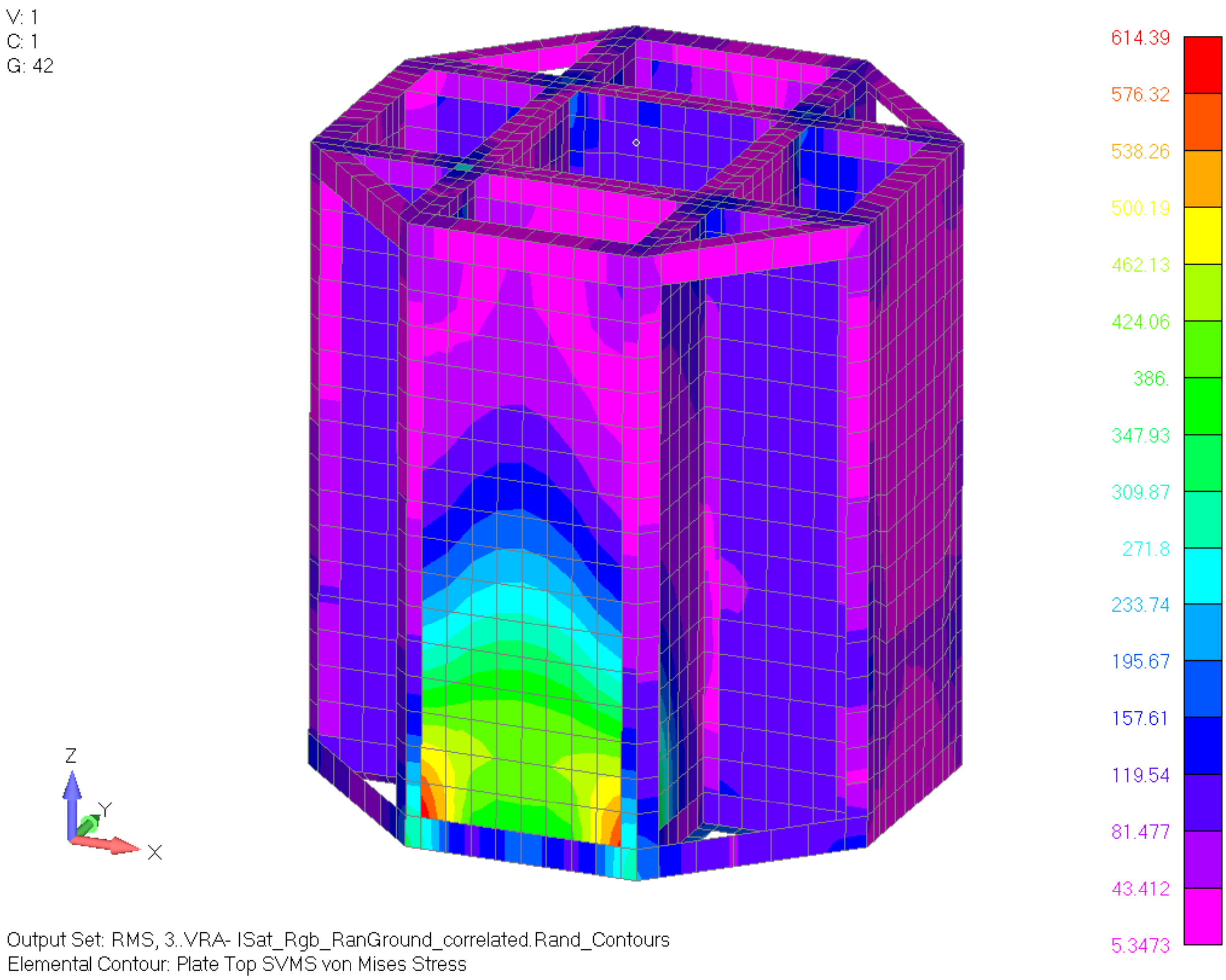


Figure 4-63. RMS von Mises stress in satellite bus shell elements, four-point-base, correlated.

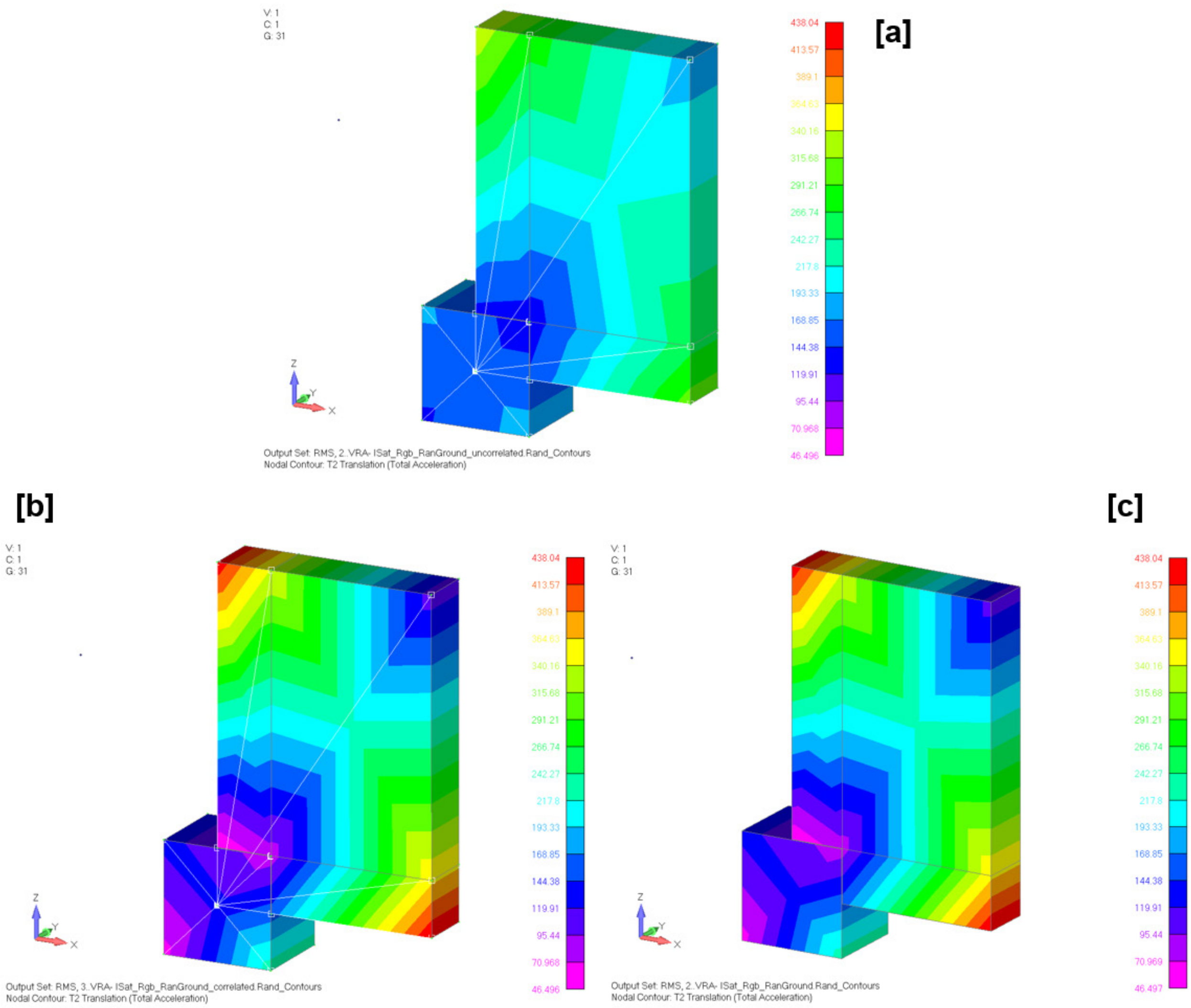


Figure 4-64. RMS accelerations: rigid-base [a], four-point uncorrelated [b], four-point correlated [c].

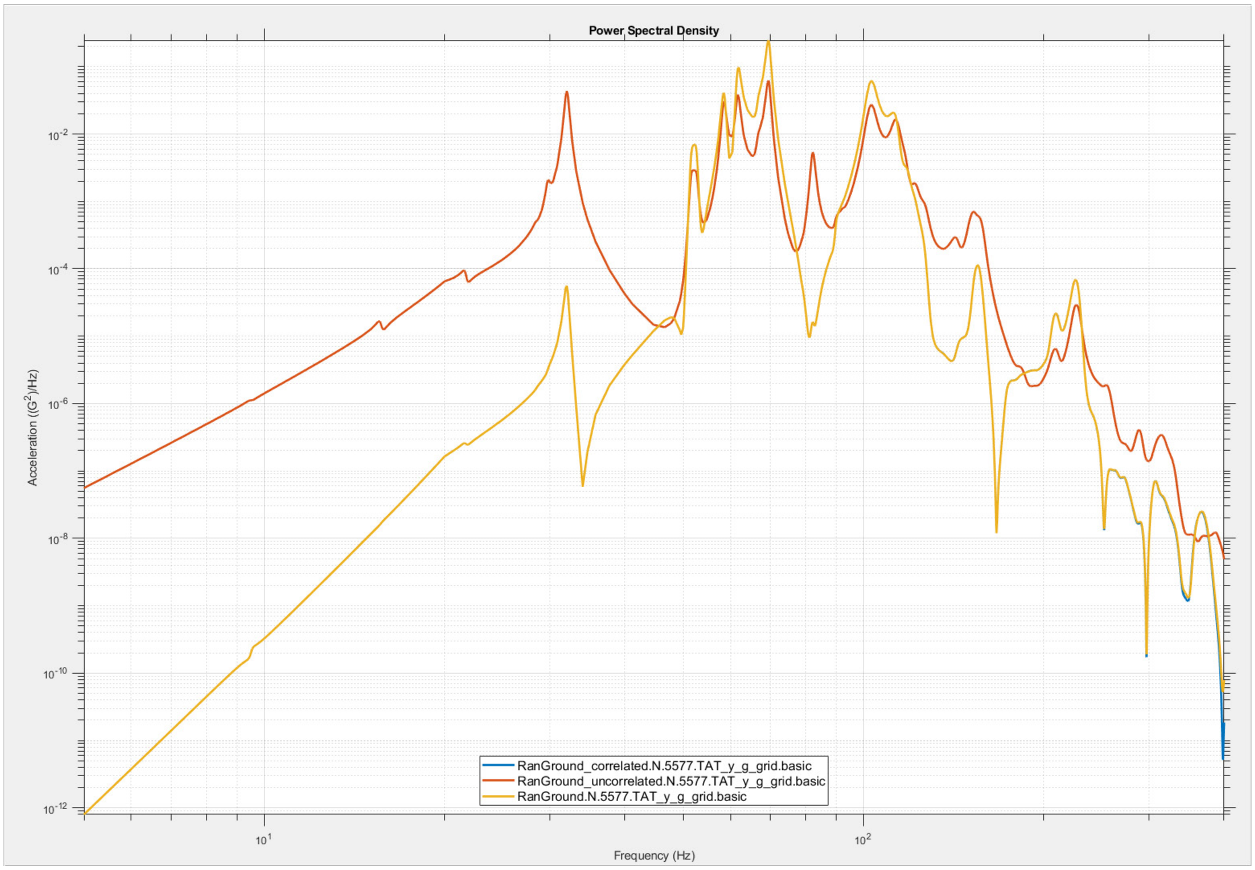


Figure 4-65. The correlated four-point results match the rigid-base results for node 5577 Y-acceleration.

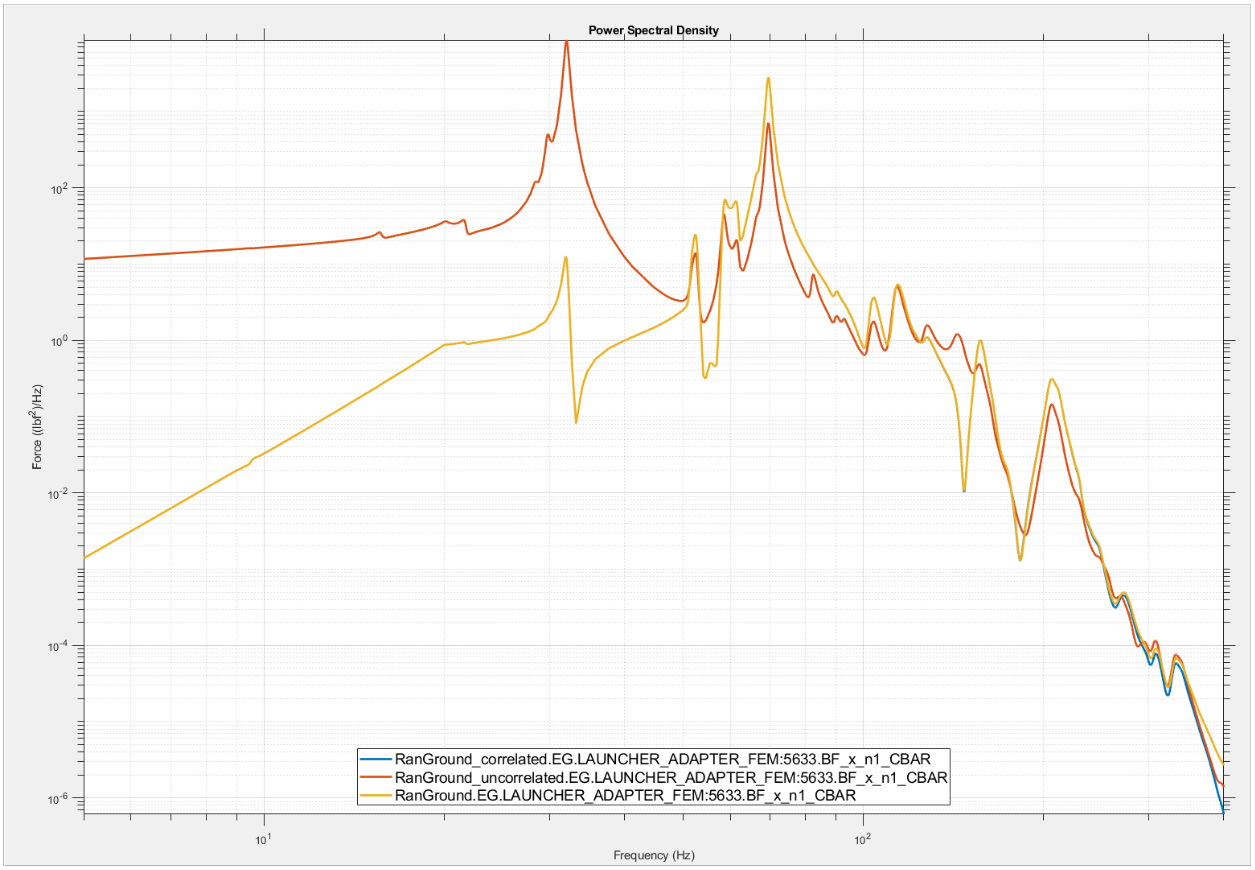


Figure 4-66. The correlated four-point results also match the rigid-base results for launcher leg forces.

4.3.5. RMS Contours Using FastRMS.

The Isat model is small by today's standards and the frequency range under study is not very broad, so the RMS contour calculations in the examples above do not take very long. More realistic analyses, where the model may have more than 500,000 elements and the frequency range of interest may be up to 2000 Hz, can take many hours using standard approaches to computing RMS values. Vibrata has an alternative method that is many times faster and has equivalent accuracy.

1. In the Event Manager, copy the original rigid-base event from section 4.3.2 using the **Copy** button and then assign a new name to the new event (Figure 4-67).

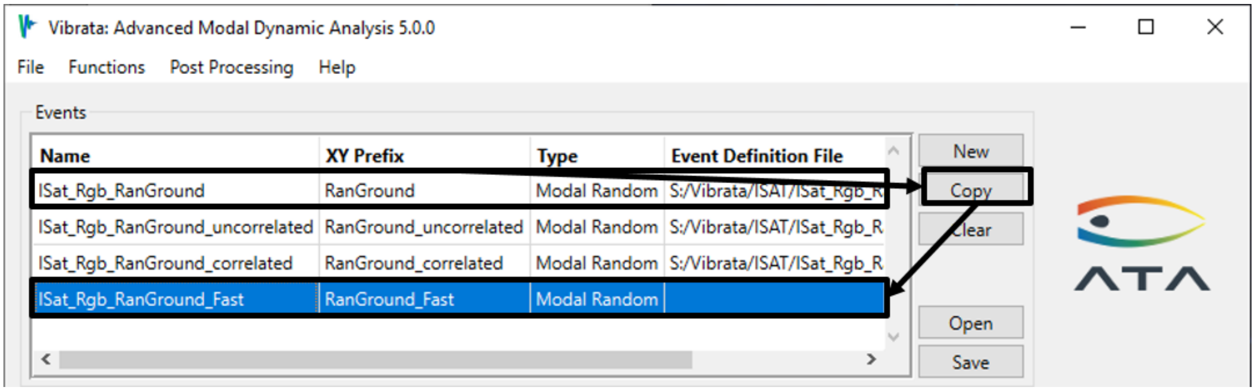


Figure 4-67. Copy the original rigid-base event; the new one will use FastRMS contour calculations.

2. On the *Excitations* tab, turn on the *Use FastRMS* toggle (Figure 4-68). That is the only change needed for the new event.

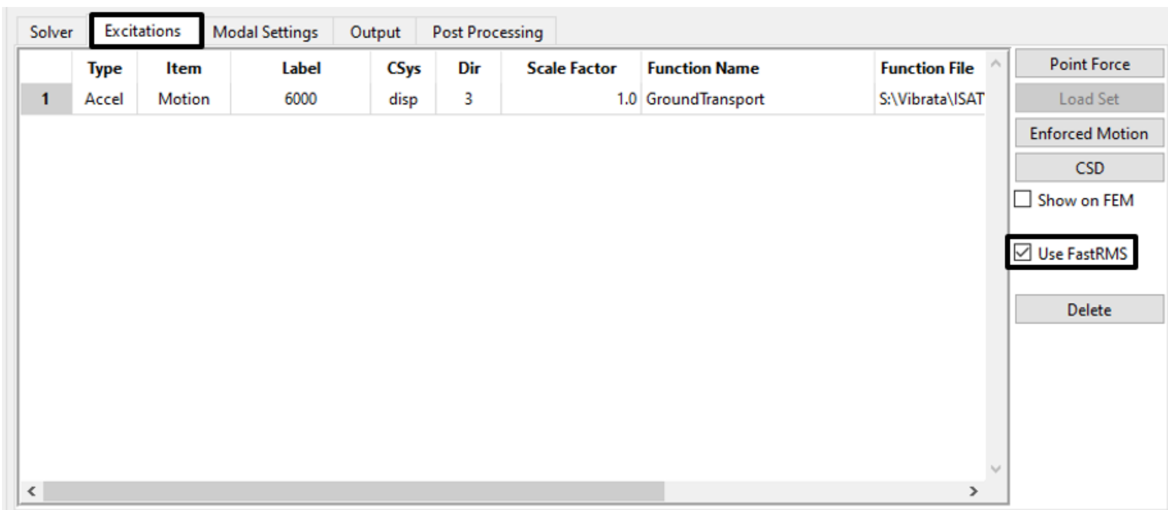


Figure 4-68. Turn on the *Use FastRMS* toggle on the *Excitations* tab.

3. Go to the *Output* tab and click **Solve**. The FastRMS approach has no effect on computing response XY PSDs, so we need only look at the contour results (Figure 4-69). Visually they are indistinguishable from the results of the standard computations (Figure 4-46, Figure 4-47), although interrogation will show very slight numerical differences that can be attributed to the interpolation scheme used for integrating the spectra.

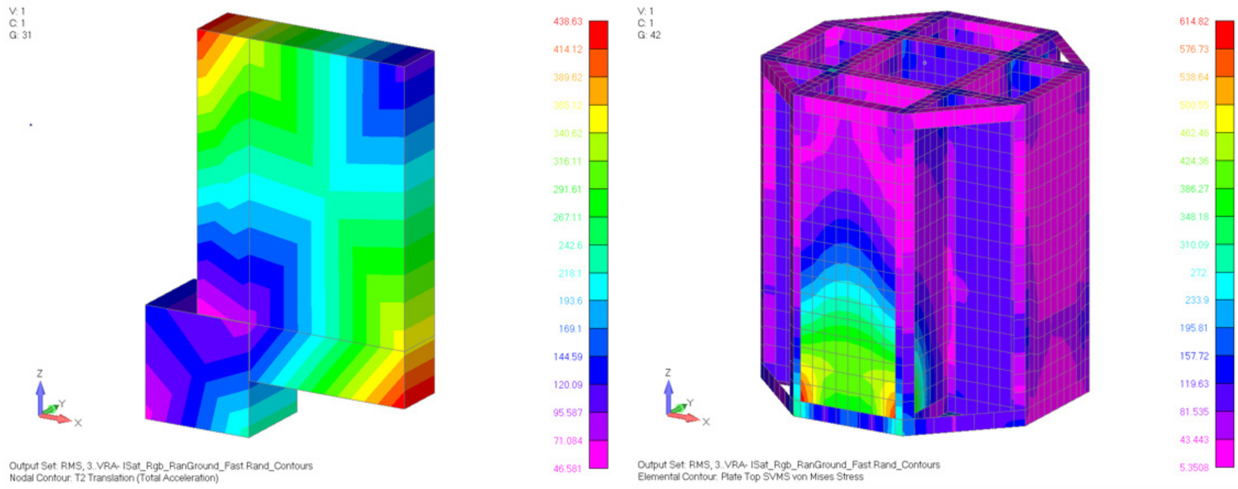


Figure 4-69. The FastRMS contours are indistinguishable from the original event.

4.3.6. Deactivate Modes with Negligible Modal Effective Mass.

Another way to speed up your analyses is by deactivating (excluding from the dynamic solution) modes that will not be excited by the environment you are analyzing. For enforced motion excitations (but not for any other kind), you can determine such modes by examining their modal effective mass. Modes with no significant effective mass will simply not respond to enforced motion excitation, so we need not include them in our calculations. (See also section 7.6.) In this example, we will deactivate all modes that do not have at least 0.1% of the modal effective mass in at least one of the translational DOF.

1. In the Event Manager, copy the event from section 4.3.2 using the **Copy** button and then assign a new name to the new event (Figure 4-70).

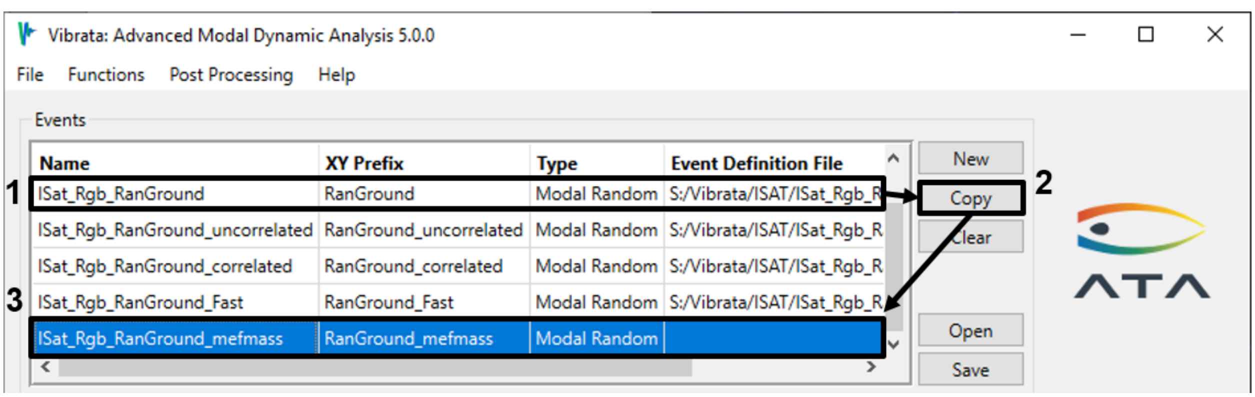


Figure 4-70. Copy the original rigid-base event; the new one will exclude modes with negligible effective mass.

2. On the *Modal Settings* tab, in the *Select Modes* panel, set the *Effective Mass Threshold* to 0.1 and the *Effective Mass Directions* to *X, Y, Z*. Click the **< Mass** button to select the modes whose effective mass is less than that threshold for all the translations (Figure 4-71).

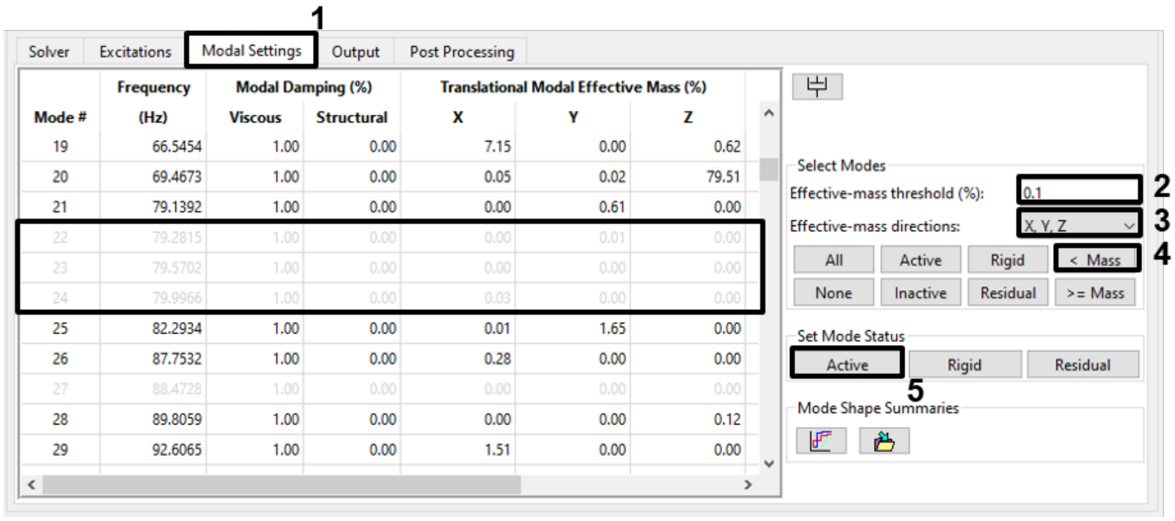


Figure 4-71. Select modes below 0.1% modal effective mass for all translations.

3. With the low-mass modes selected, click the **Active** button in the *Set Mode Status* panel to deactivate the selected modes (Figure 4-72). This leaves only 51 of the 174 modes active, so the solver will compute responses at many fewer frequencies. Note, however, that we have deactivated less than about 1% of the effective mass for any of the translations.

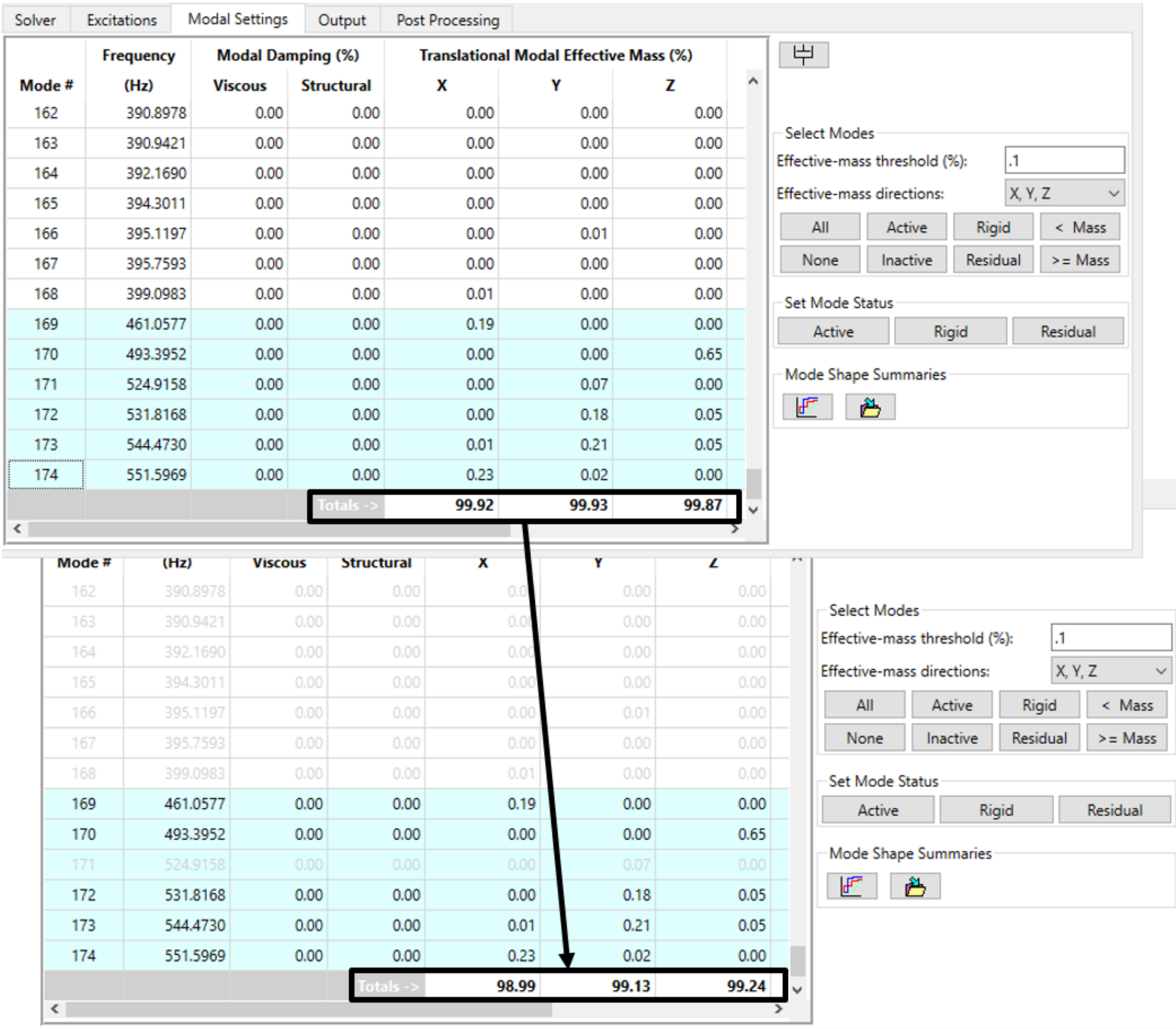


Figure 4-72. All but 51 of the 174 modes are turned off, but 98% of the effective mass remains.

4. Solve the event. Here again, the contour results (Figure 4-73) are visually indistinguishable from those of the original event (Figure 4-46, Figure 4-47). The XY Plot results, however, do show some differences. We deactivated all the modes from 240 Hz to 400 Hz, so neither the Y-acceleration response PSD (Figure 4-74) nor the axial force response PSD (Figure 4-75) has any dynamic content in that range, while the original event does. However, the RMS values for the two cases match to 3 significant digits for both acceleration and axial force. In these plots, the orange curve is for the current (mass-filtered) event, and the blue curve is for the original (all-modes-active) event.

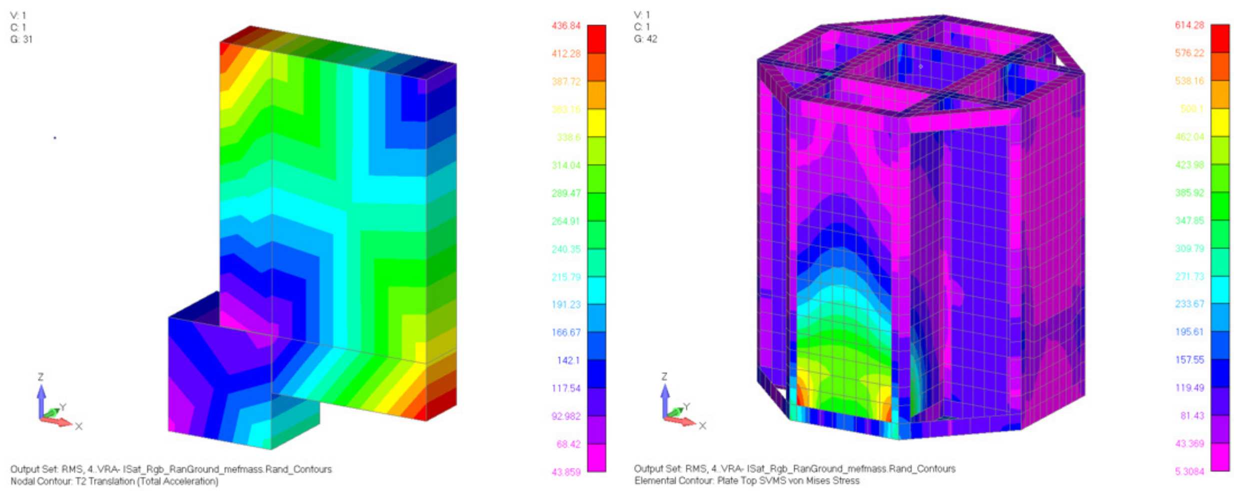


Figure 4-73. The mass-filtered contours are visually identical to those from the original event.

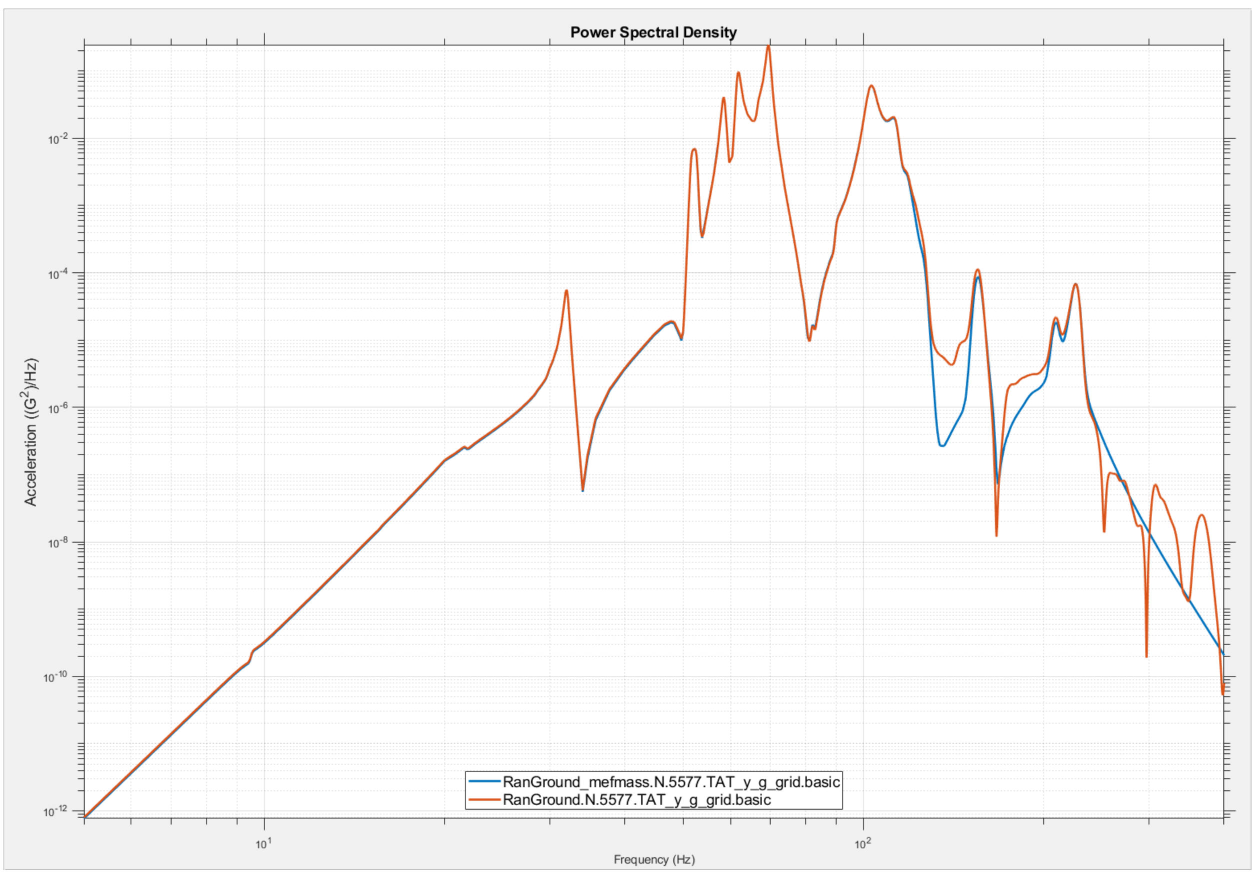


Figure 4-74. The node 5577 response shows differences at high frequencies, but RMS values match.

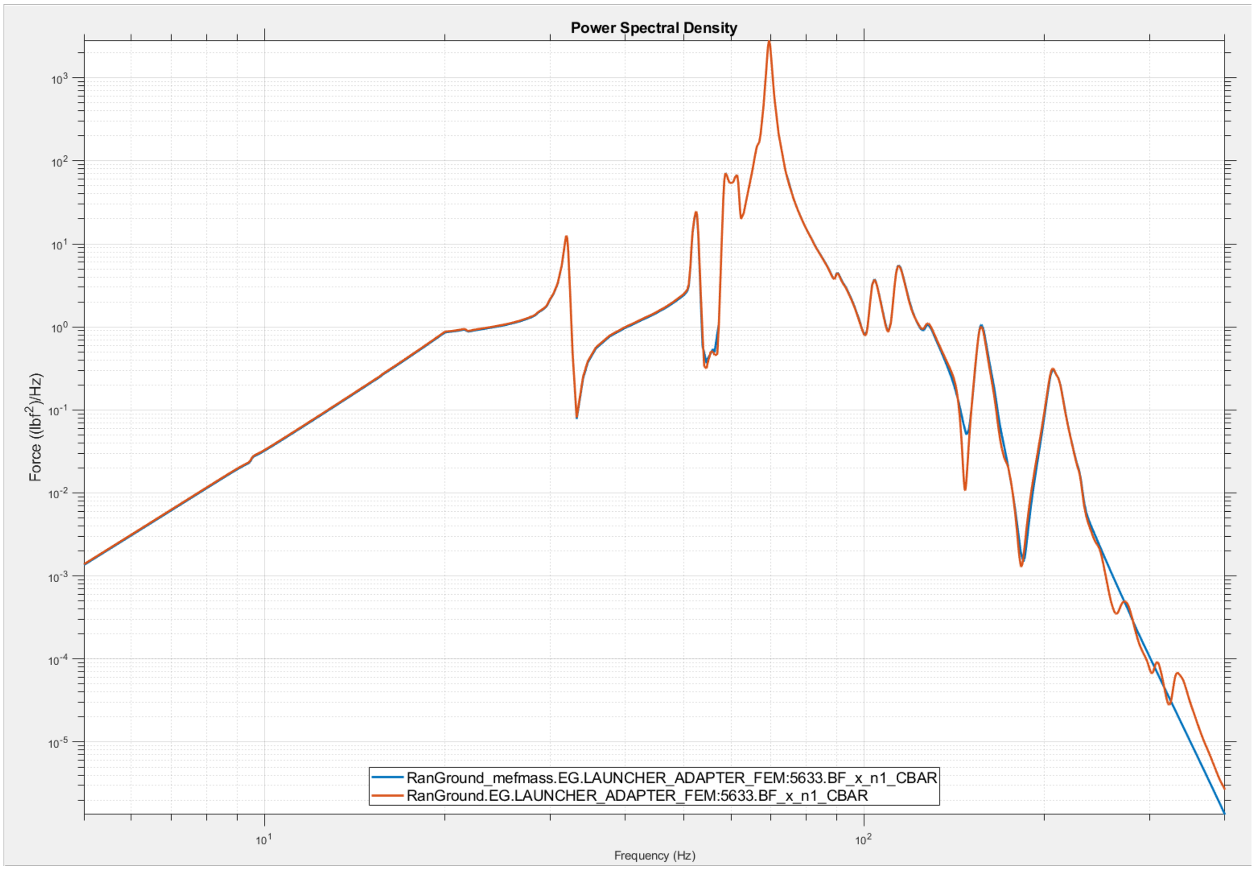


Figure 4-75. The adapter-leg axial force again shows differences only at high frequencies.

4.4. <u>Response Spectrum Analysis</u>

The 2DOF model (Figure 4-2) is used to show the difference between different summation techniques (ABS, SRSS, and other) for response spectrum analysis.

1. If you have not already loaded the frame example model (Figure 4-3) into Femap, do so now. In Femap, import the frame model "frame01_modes.dat". Save the Femap model file as "frame.modfem."
2. Create a new Vibrata event, select the "frame.modfem" file for the FEM and "frame01_modes.op2" for the results, and create a Modal Response Spectrum event with the **Summation Method** set to *Absolute Value* (Figure 4-76).
3. Save the *.evt using a descriptive name.

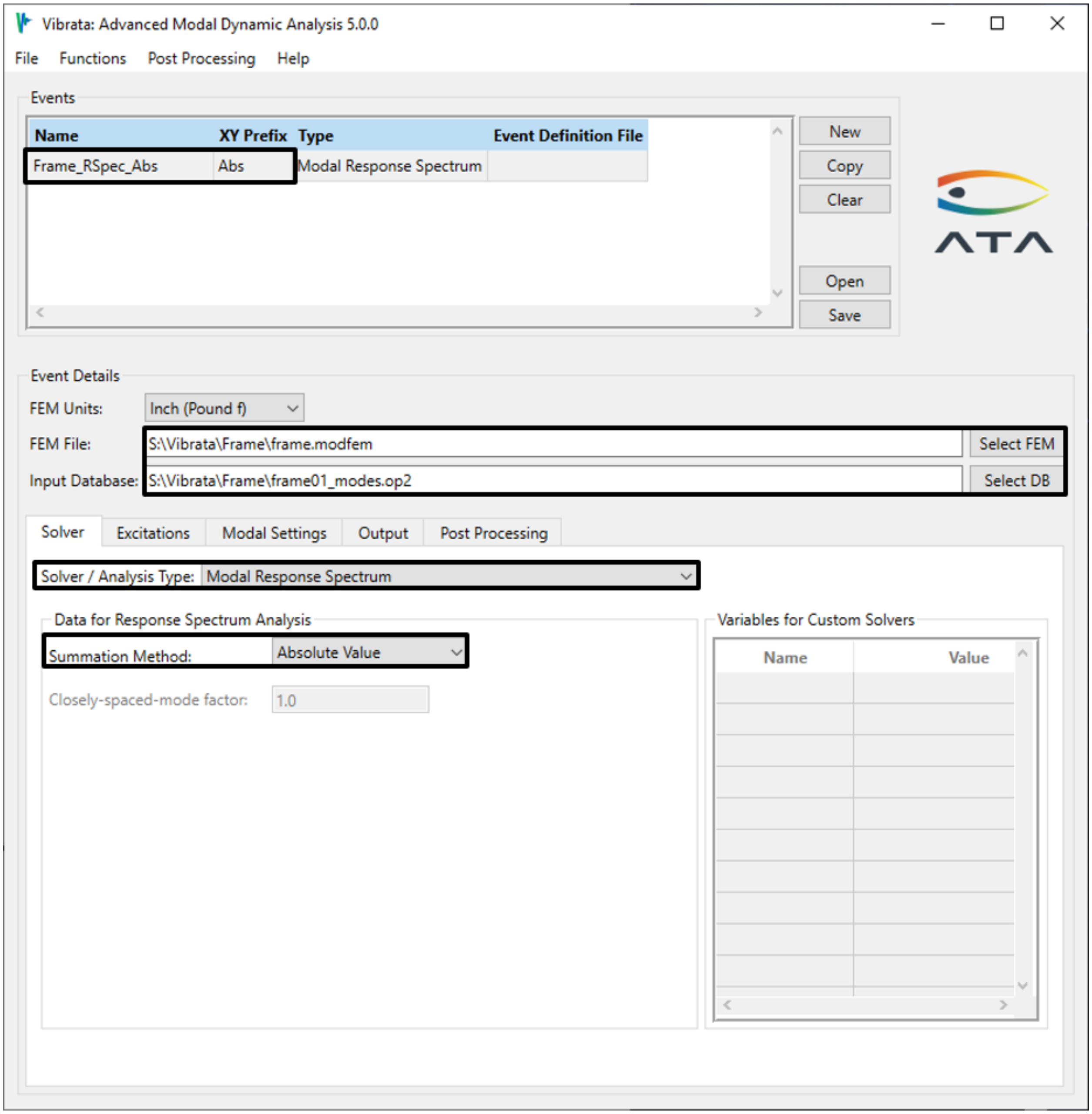
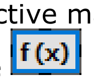


Figure 4-76. New response spectrum event with absolute value summation.

4. Response spectra are applied as enforced motions, so go to the *Excitations* tab and bring up the *Enforced Motion Excitations* dialog (Figure 4-77). Note that no motion out of the XY plane is offered; those degrees of freedom were removed from the model, so no modes have any effective mass in those directions. Select the X direction (direction 1) and click the  button to bring up the Function Manager.

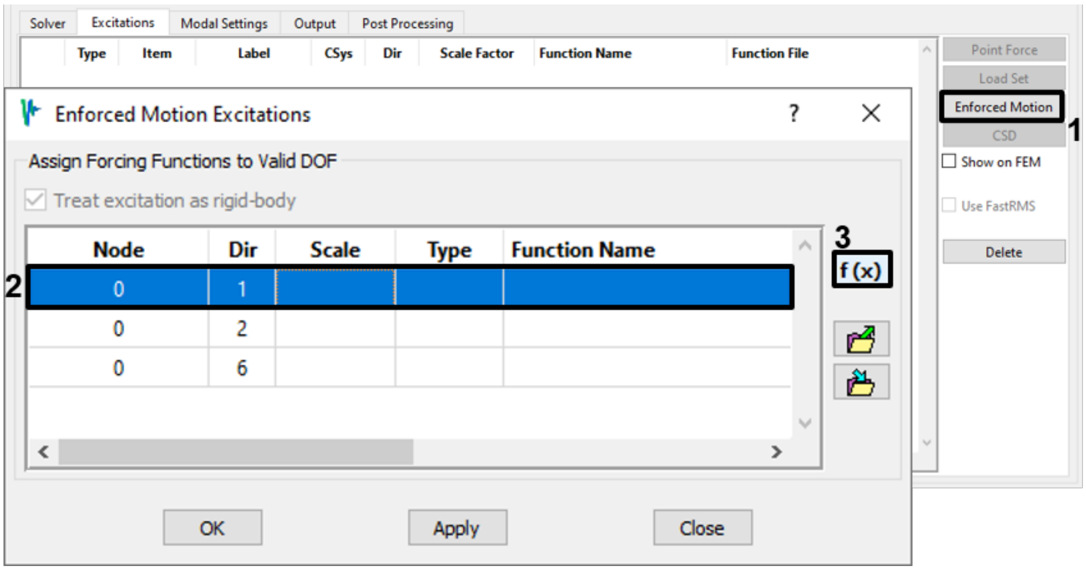


Figure 4-77. Assign an enforced motion function to direction 1.

5. In the Function Manager, use the Source Files button to open the "frame_excitations.fcn" file from section 4.2.3. (If you have not created that file yet, you will do so here when you finish defining the response spectrum function.) Define a new velocity response spectrum function as shown in Figure 4-78. Click the **Create** button and store the function to "frame_excitations.fcn" (Figure 4-79). Click **Done** on the Function Manager main dialog to return to the Event Manager.

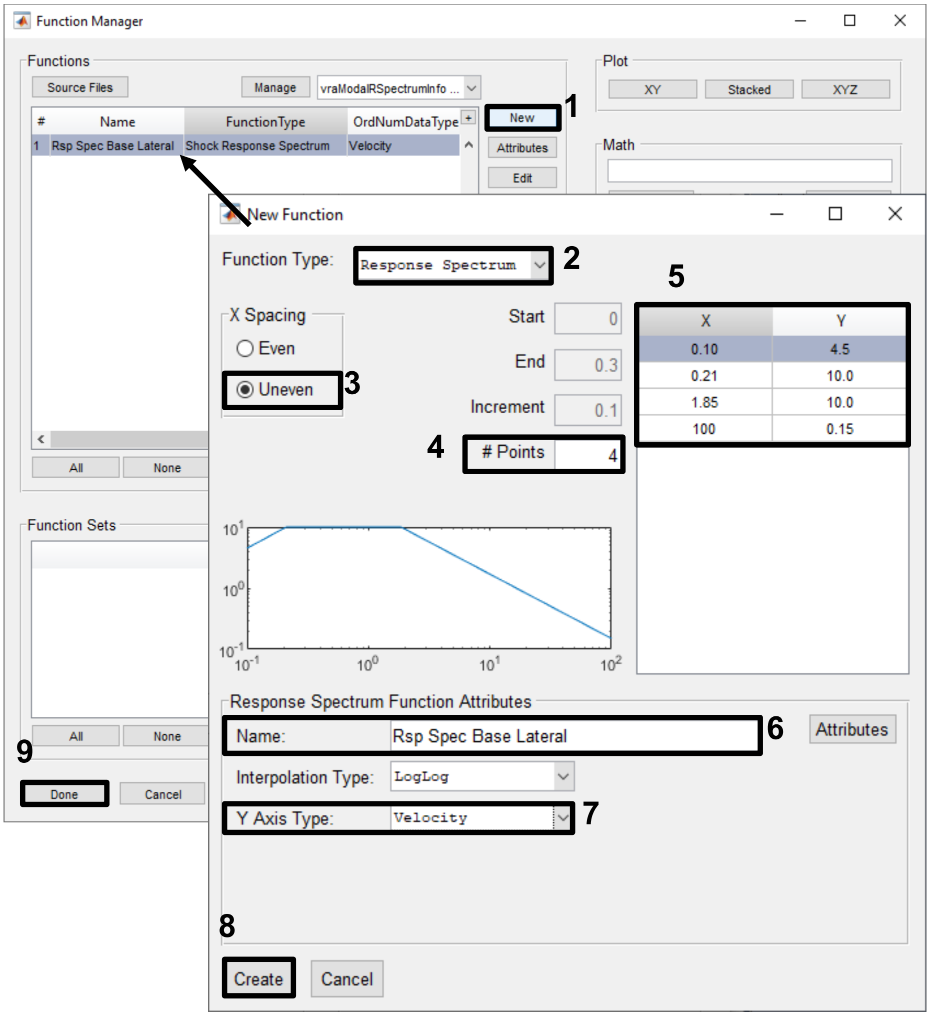


Figure 4-78. Define a Response Spectrum velocity function.

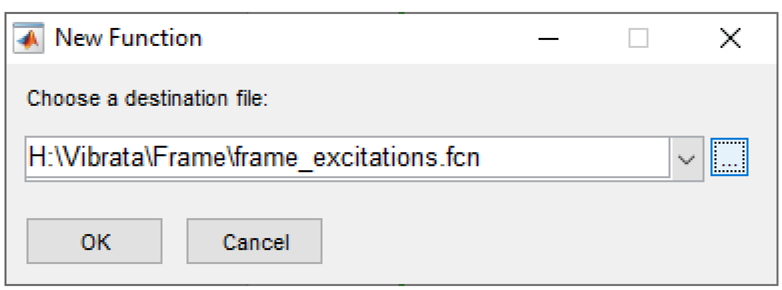


Figure 4-79. Store the new function in the frame excitations file.

6. Click **OK** on the enforced motion dialog to finalize the excitation definition (Figure 4-80). Note that it uses the basic coordinate system. Also note that it is considered part of rigid body motion set; this makes no difference here, but it would be important if more than one direction were being excited.

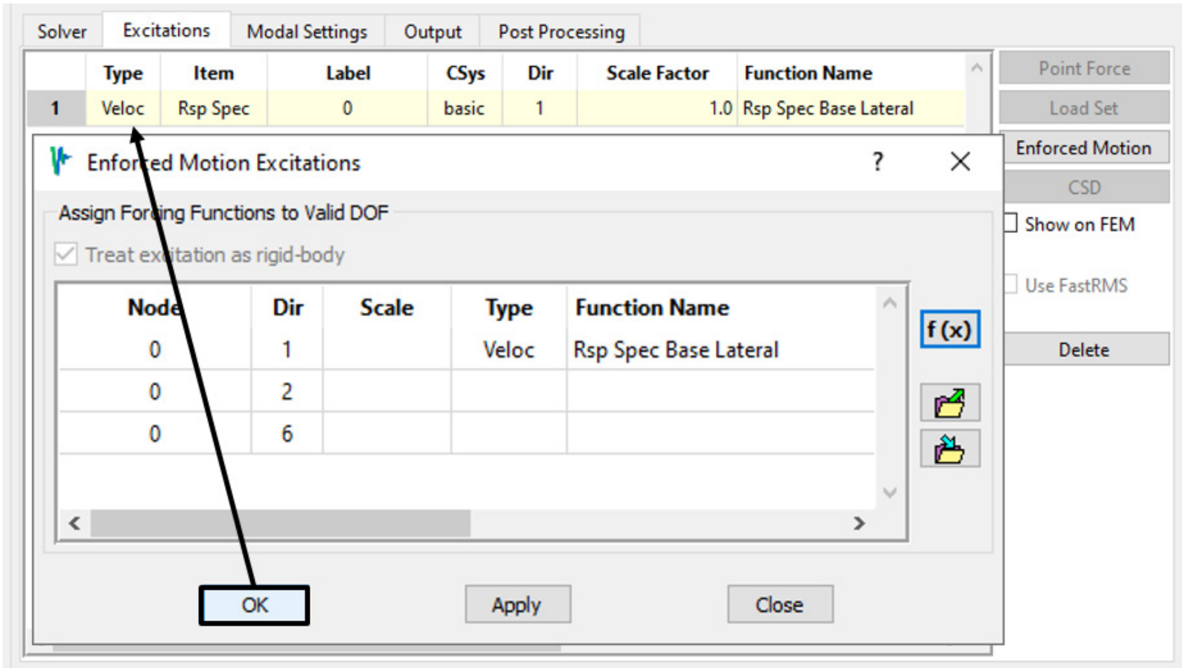


Figure 4-80. The excitation uses the basic coordinate system.

7. No damping is needed for response spectrum analysis, as the damping level is already accounted for in the response spectrum function. In fact, you cannot even open the damping dialog (Figure 4-81).

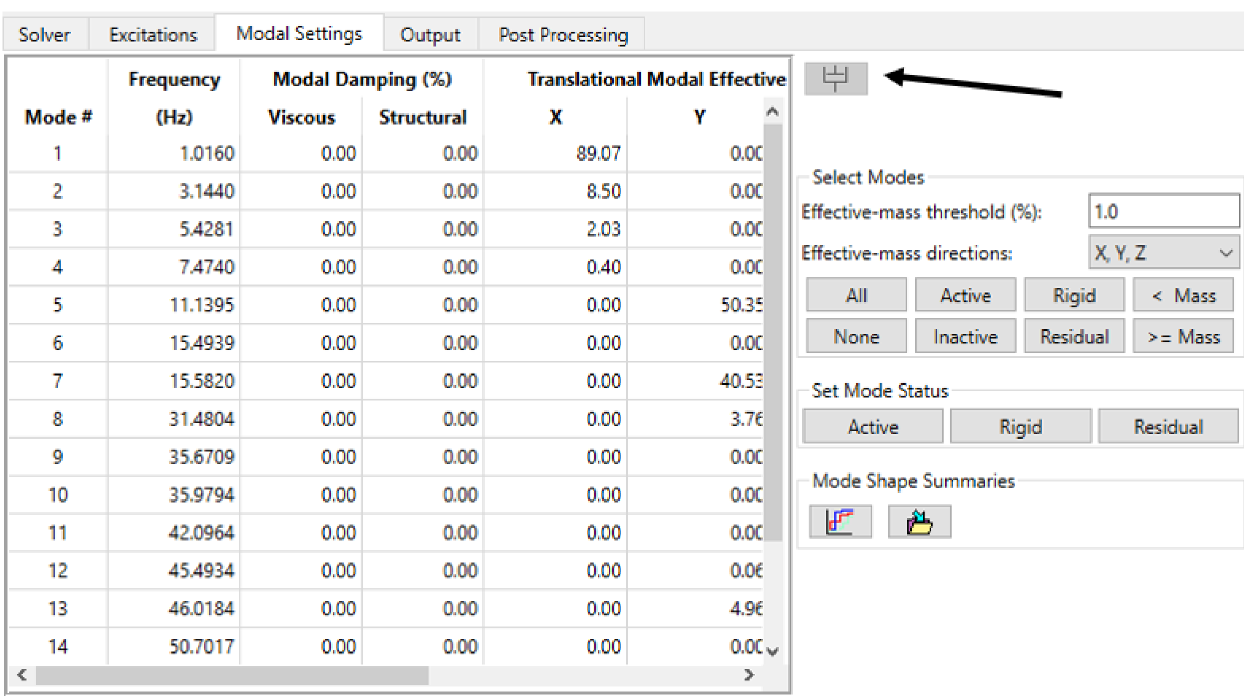


Figure 4-81. Damping is not needed and the damping dialog button is disabled.

8. Go to the *Output* tab. Only contour output is available for response spectrum analysis, so create a contour request for displacement and acceleration (*UT*, *AT*) at all nodes (Figure 4-82). Note that the **Select nodal variables** toggle is checked and inactive; the normal modes solve did not request any elemental data. Solve the event but do not plot the contours at this time.

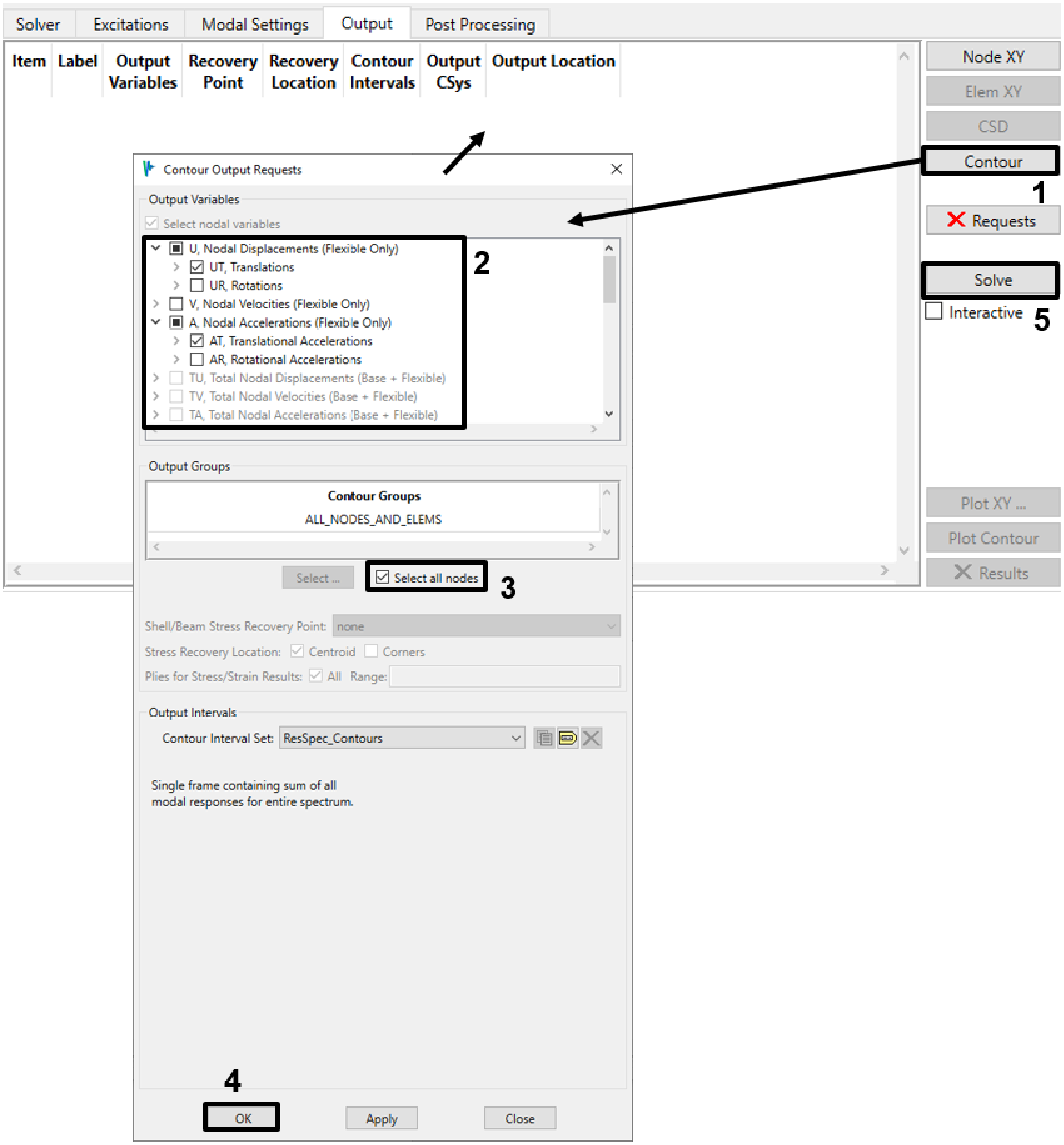


Figure 4-82. Request translational displacements and accelerations at all nodes.

9. Copy this event. Change the summation method to *SRSS* and change the event name to reflect this (Figure 4-83). Solve the SRSS event.

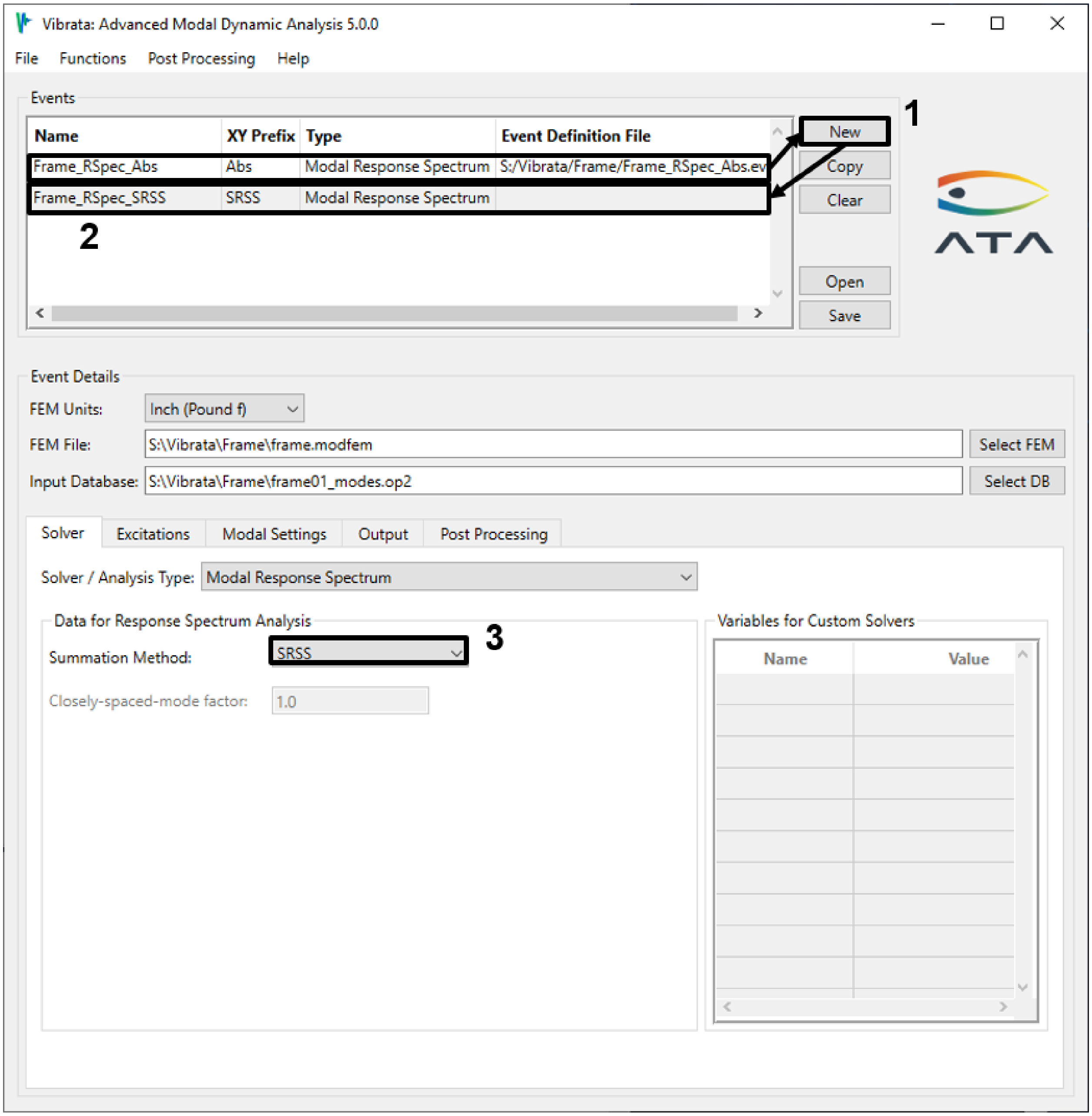


Figure 4-83. Copy the event and change the summation method to SRSS.

10. Copy the first event again. Change the summation method to *NRL* and set the Closely-spaced-mode factor to 1.1 (Figure 4-84). From Figure 4-81 you can see that the frequencies of modes 6 and 7 and modes 9 and 10 are within 10% of each other, so they will be considered closely spaced. Solve the event.

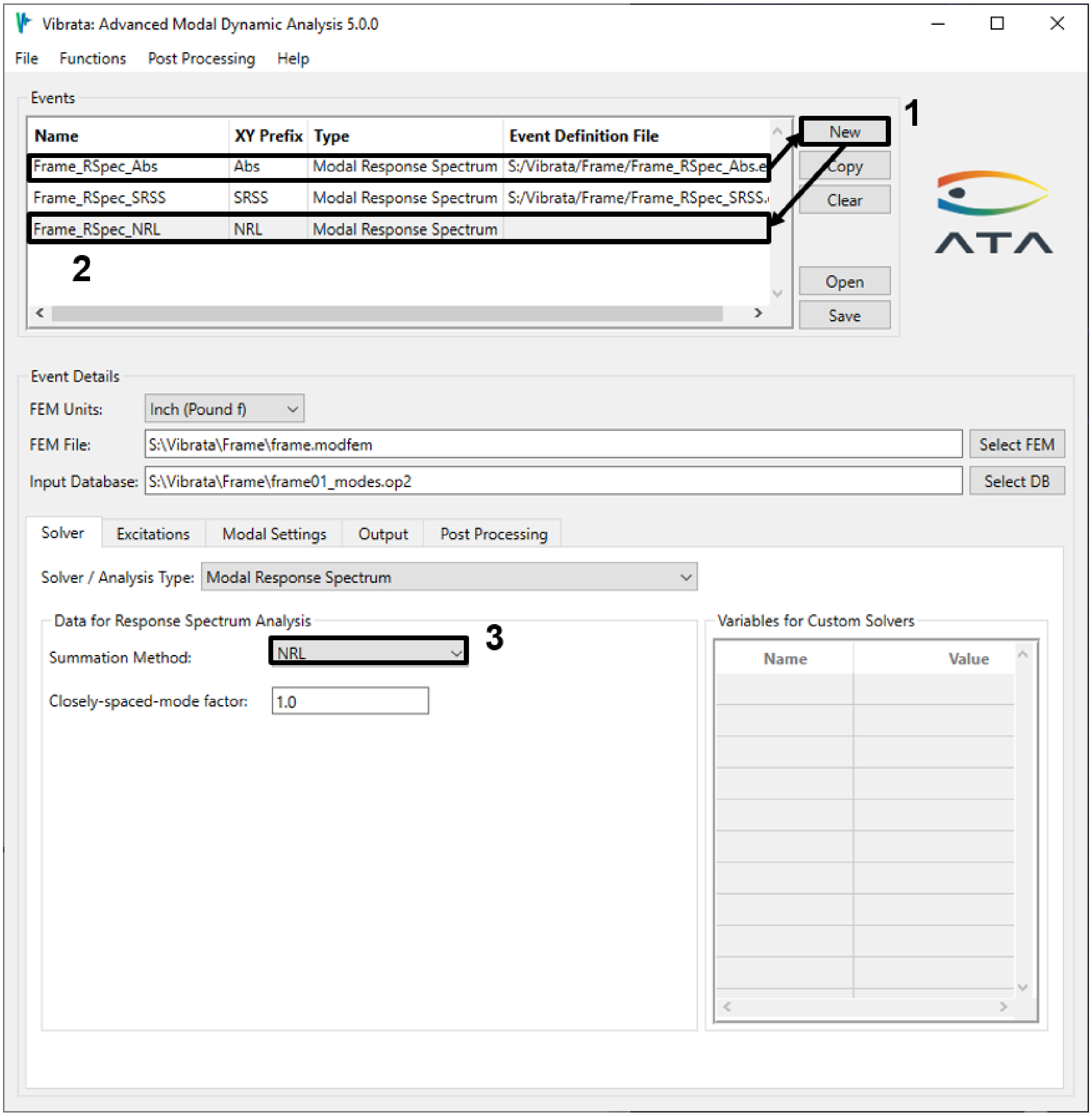


Figure 4-84. Copy again to create a third event using NRL summation.

11. Copy the first event again and change the summation type to *NRC 10%* (Figure 4-85). This should identify the same modes as closely spaced as in the NRL example above. Solve this event.

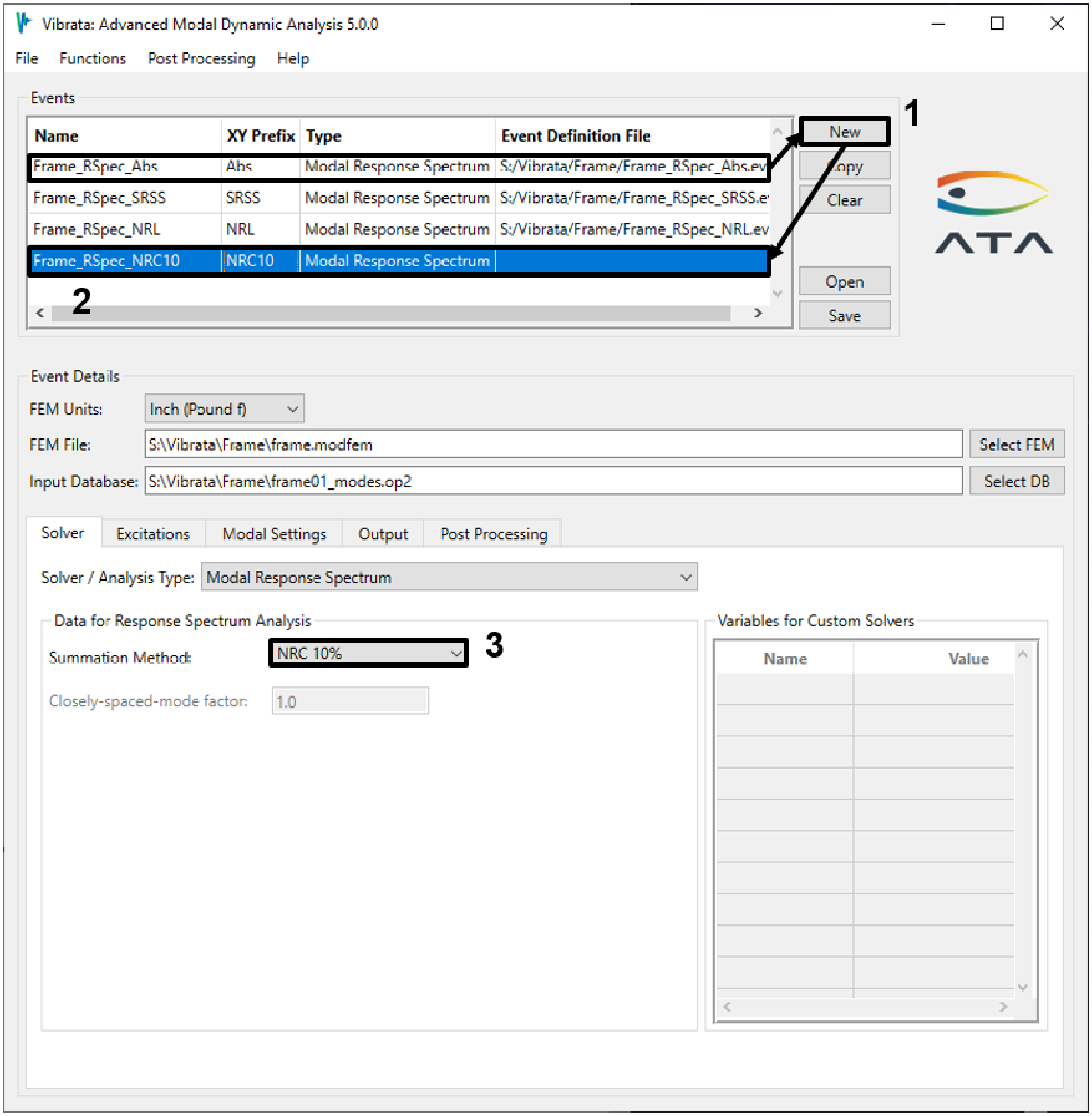


Figure 4-85. Copy again to create a fourth event using NRC summation.

12. For each event, plot the results in Femap. Note that Femap does not actually display contours on beam elements; you will have to use criterion plots. You can select that display option, and also turn deformed mesh display, using Femap's *Post* toolbar (Figure 4-86). The deformation will be X translation rather than translation magnitude because the latter is not computed in response spectrum analysis.

Figure 4-86. Select *Criteria* and *Deformed* views from Femap's *Post* toolbar.

13. The displacement results are shown in Figure 4-87 and Figure 4-88. For all of the displacement plots, the color bar has been set to 10 intervals with the range from 0.34 to 2.1 inches. The acceleration results are in Figure 4-89 and Figure 4-90. The color bar range in these plots is from 20 to 140 in/s^2.

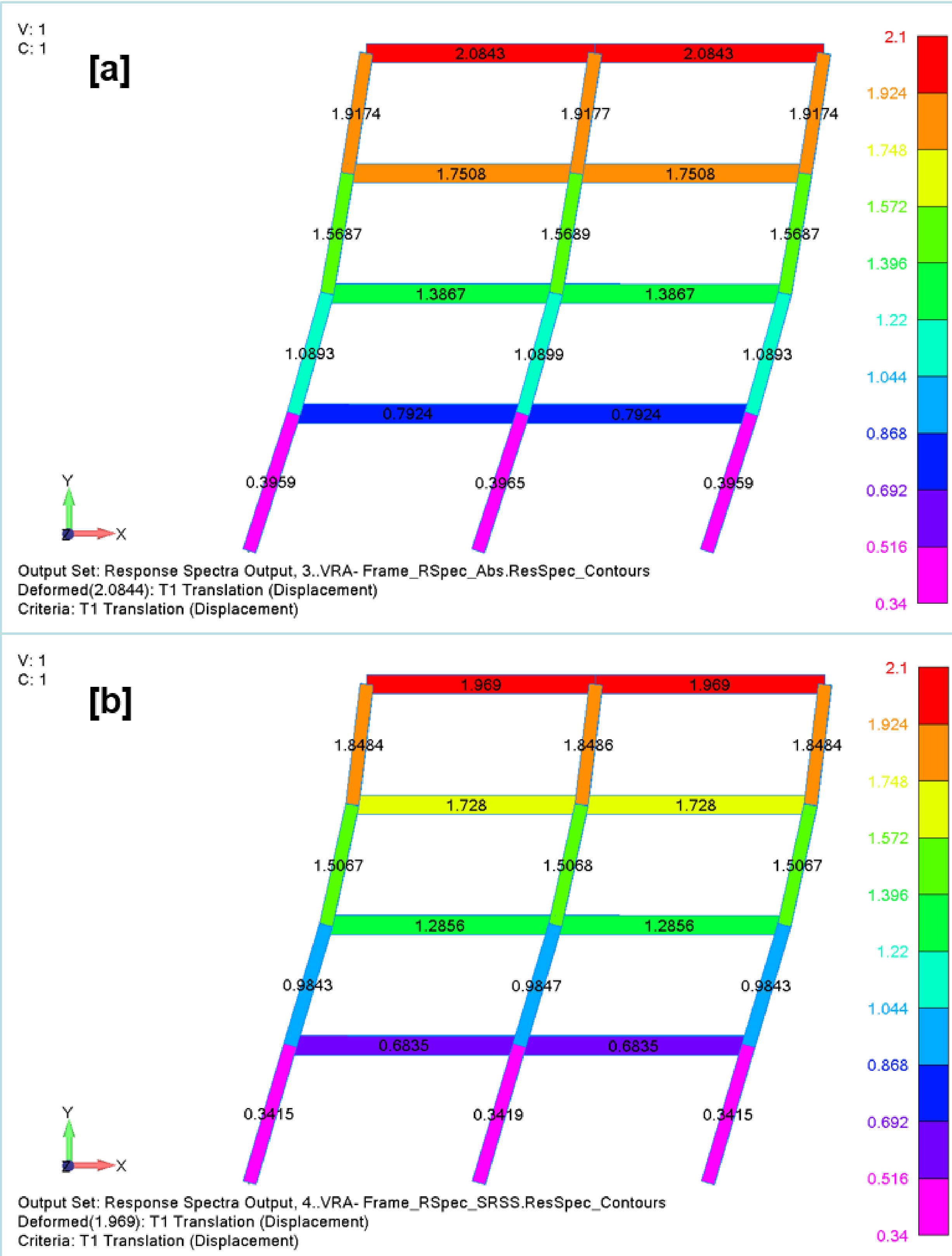


Figure 4-87. Displacements for [a] absolute value and [b] SRSS methods.

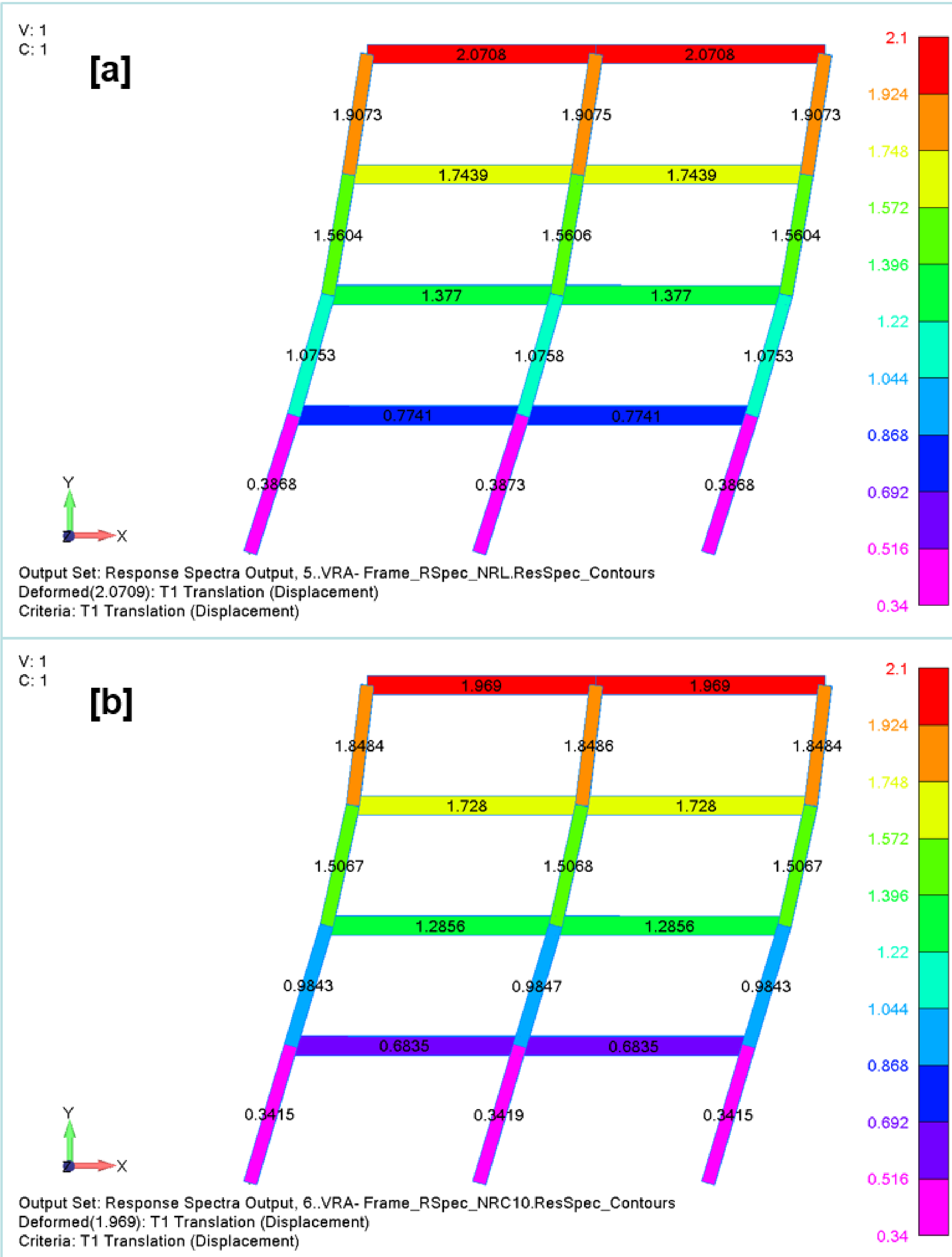


Figure 4-88. Displacements for [a] NRL and [b] NRC 10% methods.

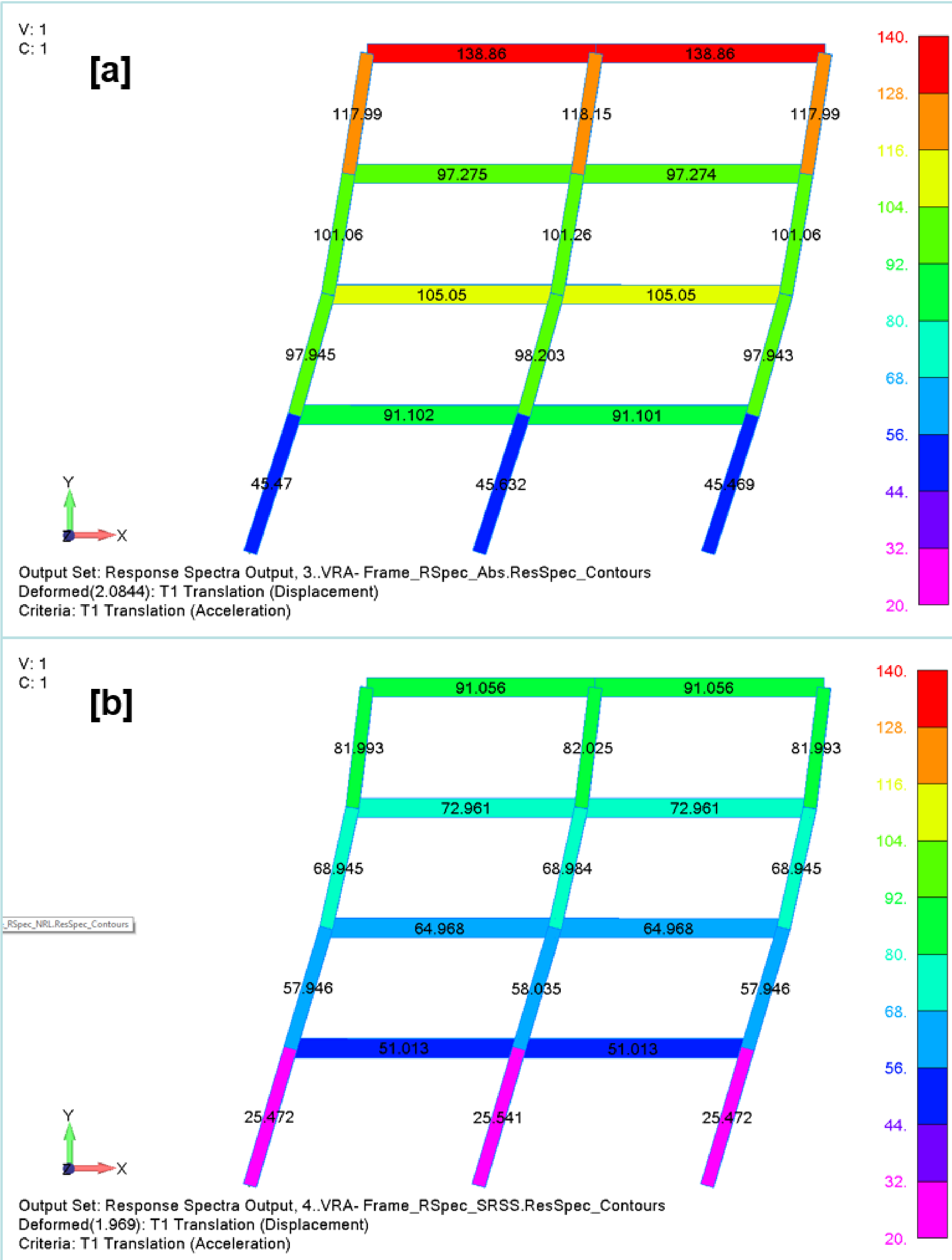


Figure 4-89. Accelerations for [a] absolute value and [b] SRSS methods.

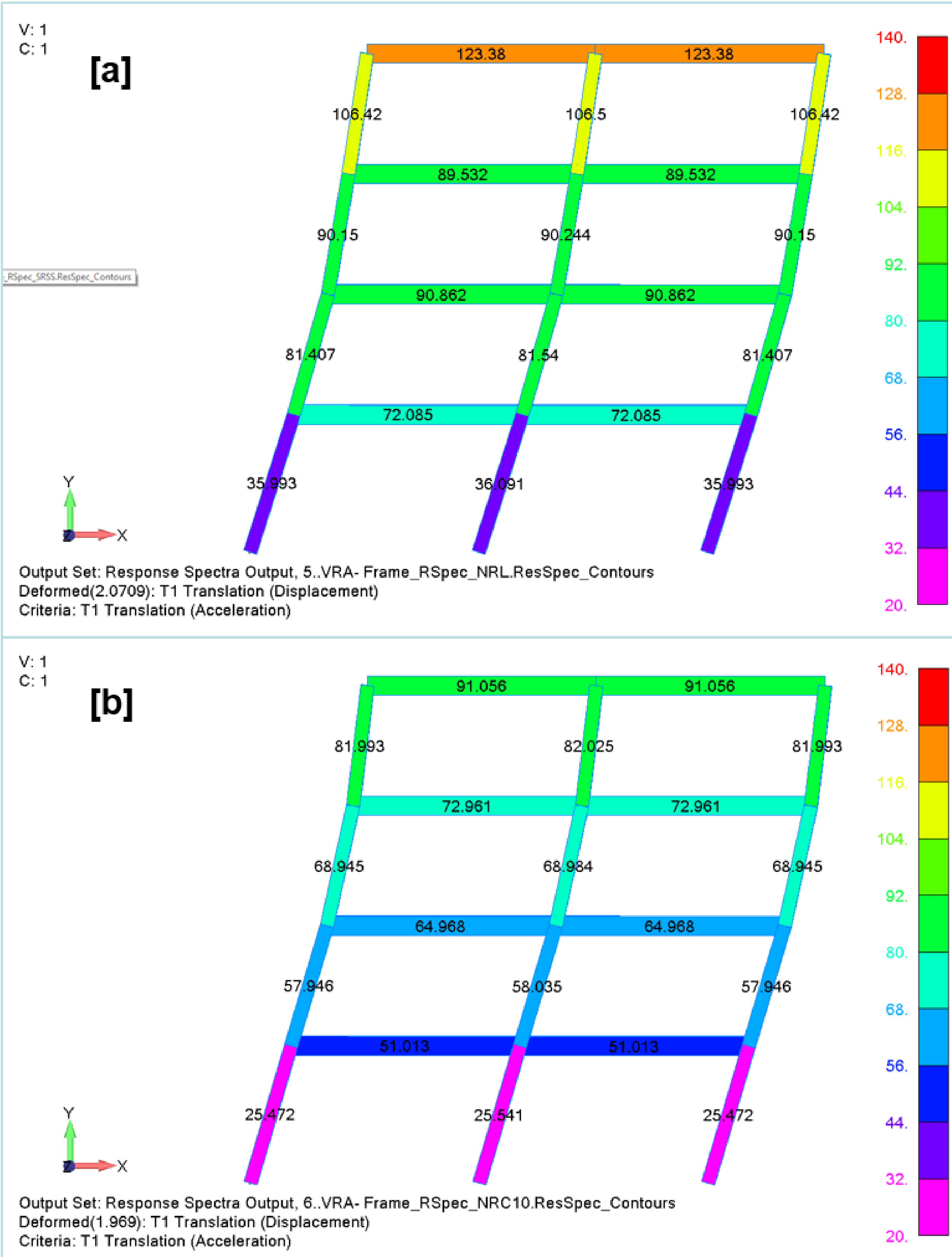


Figure 4-90. Accelerations for [a] NRL and [b] NRC methods

14. The results are summarized in Table 4-1. The SRSS method produces the lowest responses since it sums the modal responses as if they are all *out* of phase, while the absolute value method produces the highest responses since it sums the modal responses as if they are all *in* phase. The NRL results in this example are between these two because it sums modal responses as in phase if it considers them closely spaced and out of phase if not closely spaced. This makes the NRC10 results somewhat surprising. Since it uses the same closely-spaced-mode parameter as the one we selected for the NRL case, we might have expected its results to match the NRL results, yet they are indistinguishable from the SRSS results. There is a significant difference between the NRL and NRC10 methods that accounts for this. While both treat closely spaced modes as in phase, NRL also treats the maximum-response mode as in phase with all of the closely spaced modes. Looking at the effective masses in Figure 4-81, this will clearly be mode 1. It is not "close" to any other mode, so it will not be part of the in-phase summation for NRC10, but it will be for NRL.

Table 4-1. Maximum response level for each summation method.

Response	Summation Method			
	ABS	NRL	NRC10	SRSS
Displacement, in	2.084	2.071	1.969	1.969
Acceleration, in/s^2	138.86	123.38	91.06	91.06

4.5. <u>Transient Analysis</u>

The next two examples illustrate Vibrata's modal transient analysis capabilities. In the first example we will simulate a thruster firing on the Isat model, and in the second example we will simulate a hammer impacting the frame model.

4.5.1. <u>Isat Model with RCS Thruster Firing.</u>

In this example, we will use the Isat in its deployed configuration (Figure 4-6). Note that this model includes residual modes to give us improved stress results for loads applied at the RCS thrusters.

1. In Femap, import the Isat deployed model "Isat_Dploy_Sm.dat", and the Neutral file that defines many useful groups for it "Isat_Dploy_Sm_groups.neu". Save the model as Isat_Dploy_Sm.modfem.
2. In the Event Manager, set up a new transient event and select the Femap file you just created and the OP2 "isat_deploy_sm.op2". Set the event end time to 4.0 seconds and its initial conditions to Zero. Assign it an informative name. See Figure 4-91.
3. Save the *.evt using a descriptive name.

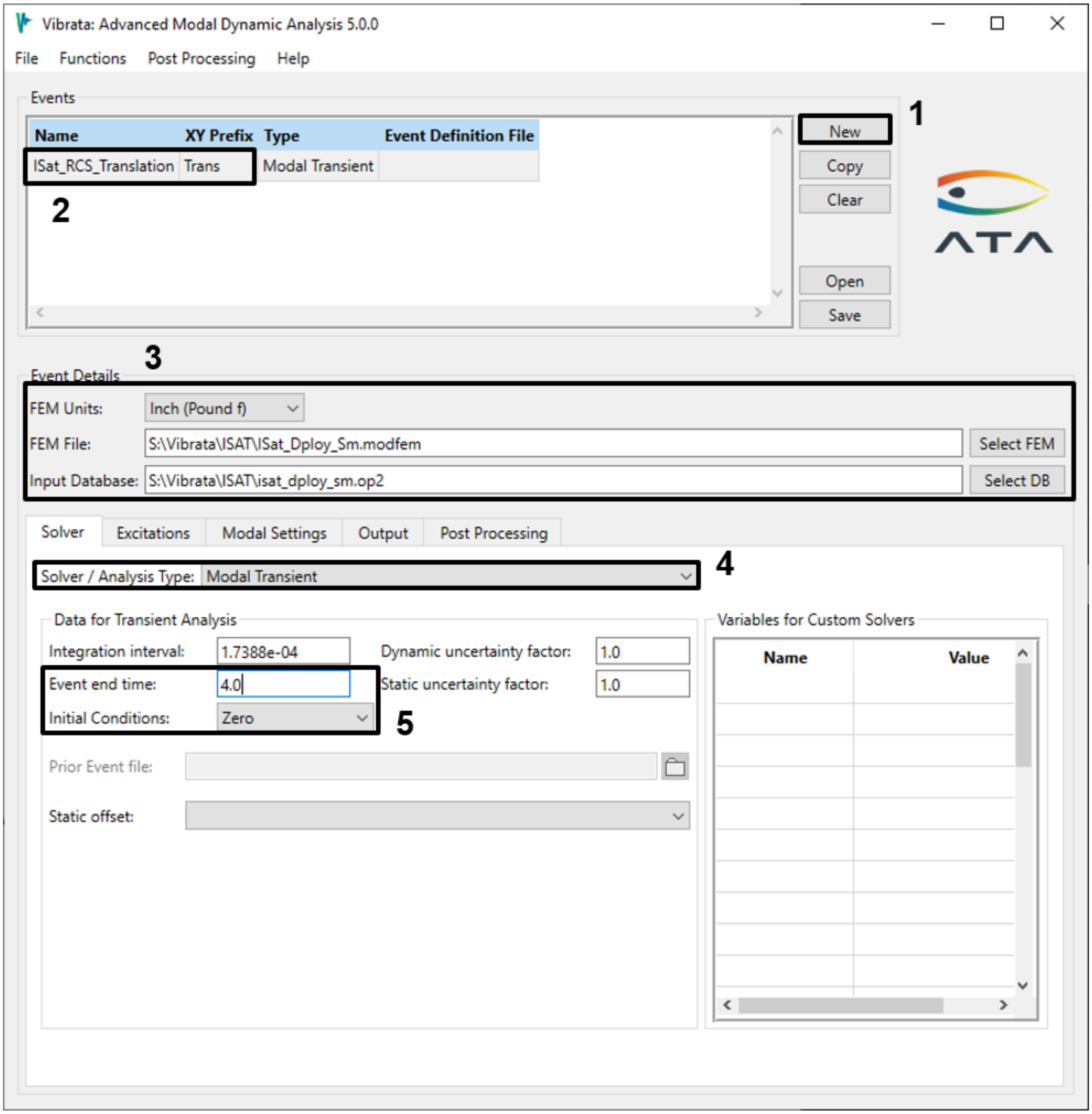
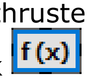


Figure 4-91. Transient event setup for deployed Isat model.

4. The RCS system will fire a pulse to translate the spacecraft along its Z-axis, let it drift for half a second, then fire a reverse pulse to stop its translation. We will simulate this by applying a point-force excitation at each RCS thruster node (Figure 4-92). On the Excitations tab, bring up the Point Force dialog (Figure 4-93). Select the individual thruster nodes as shown, highlight the Z-direction row in the Functions table, and click 

to bring up the Function Manager.

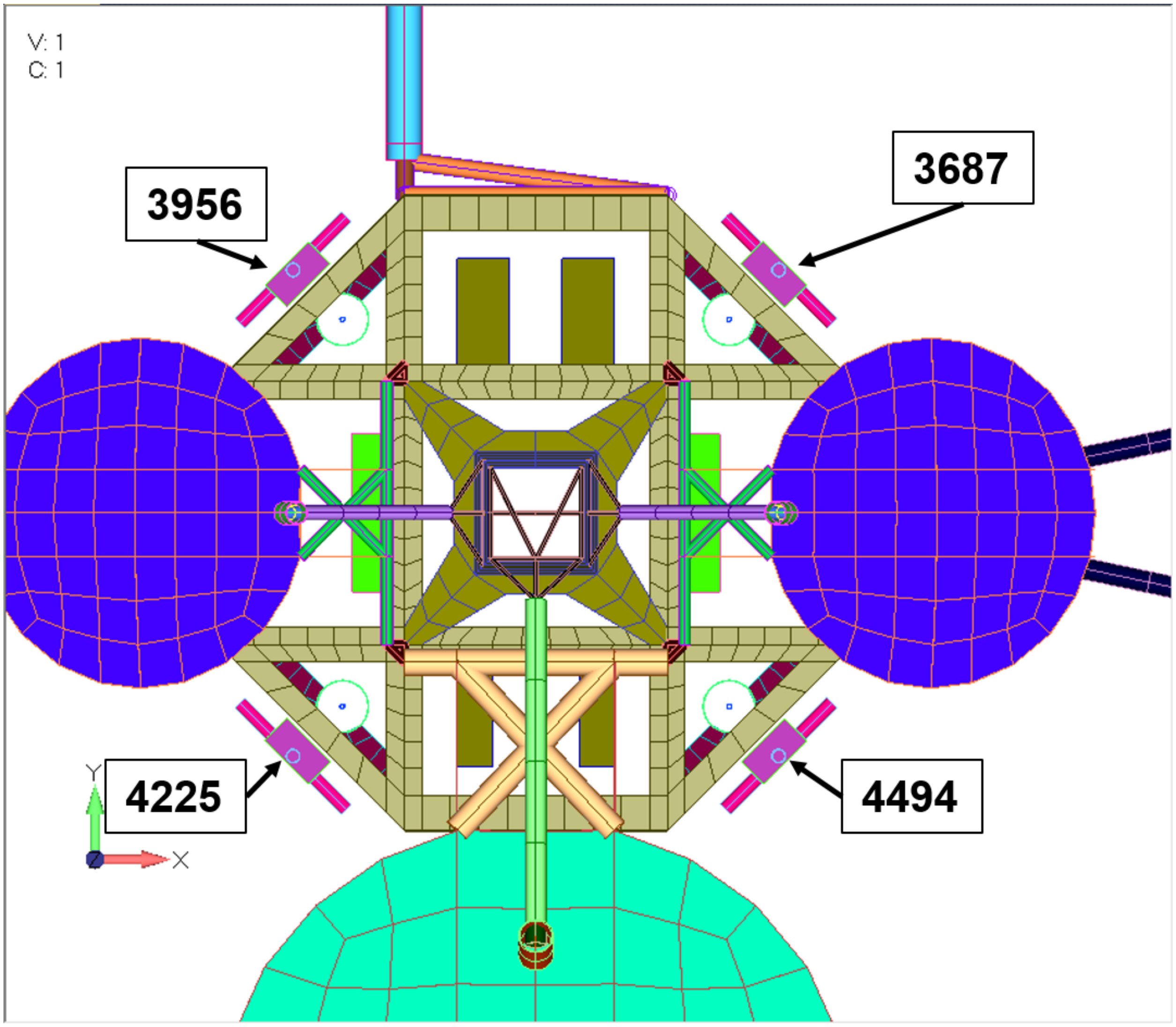


Figure 4-92. Nodes for RCS thruster loads.

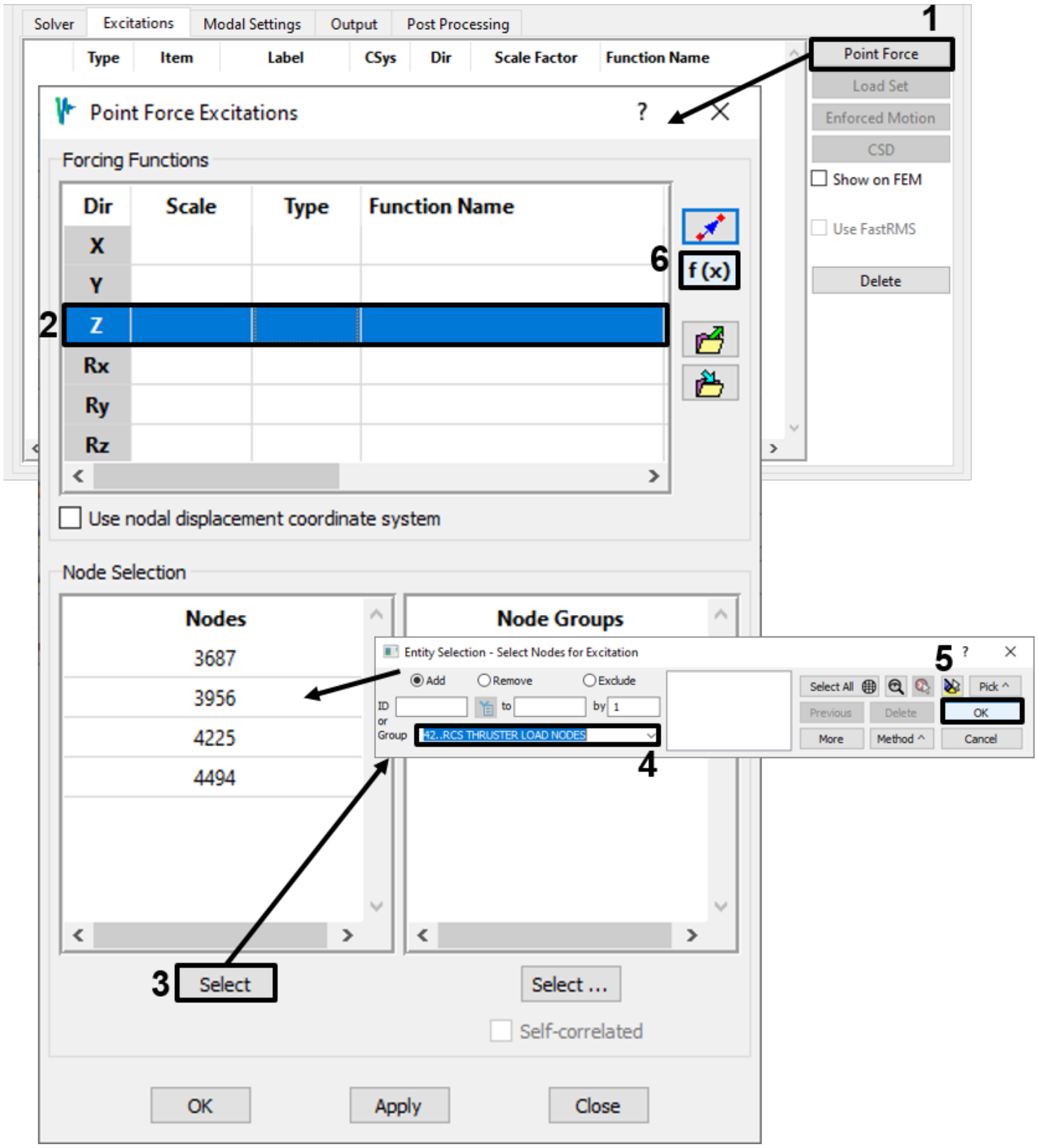


Figure 4-93. Use the Point Force dialog to assign a Z-direction force to each thruster node.

5. In the Function Manager, you will create a time function to excite the Isat transient event. If you have already run the random examples in section 4.3.1, open the “Isat_excitations.fcn” file used there; if that file does not currently exist, you will create it when you have finished defining the function for the current example. As shown in Figure 4-94, create a new transient function, set its Y-axis to *Force*, give it uneven X-spacing, set the number of points to 9, and assign it the X and Y values shown. The pulses have unit amplitude; this will let us size the RCS thrusters just by scaling the excitations. Remember to give the function a recognizable name. Click **Create** to store the function to an *fcn* file (Figure 4-95). Click **Done** on the Function Manager to return this function to the Point Force dialog. Assign it a scale factor of

20, and then click **OK** on the Point Force dialog to create the excitations (Figure 4-96).

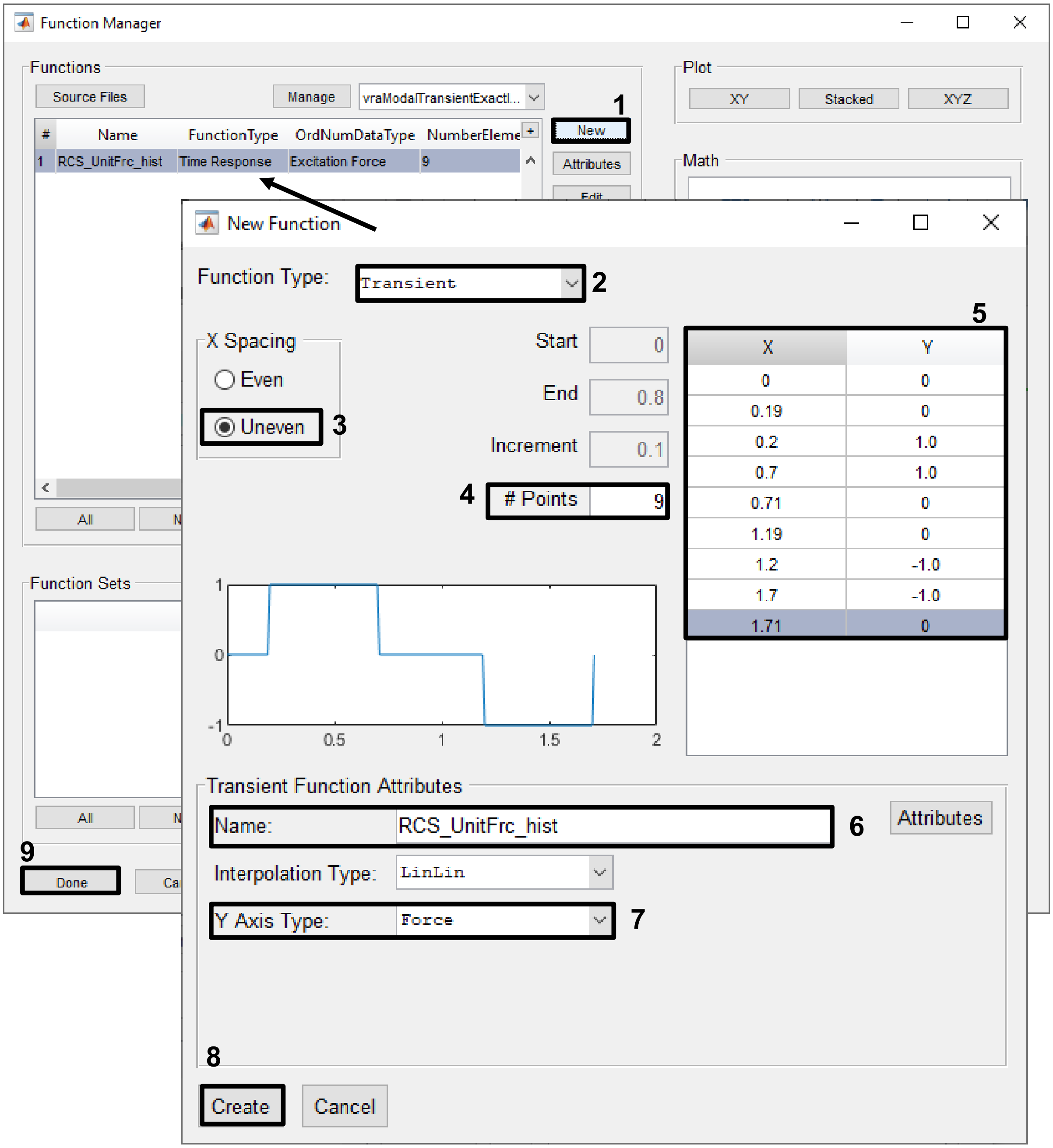


Figure 4-94. Thruster force transient for Isat maneuver.

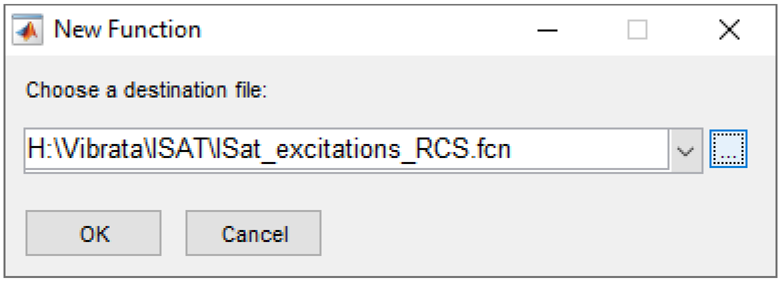


Figure 4-95. Save the Isat thruster forcing function to a file.

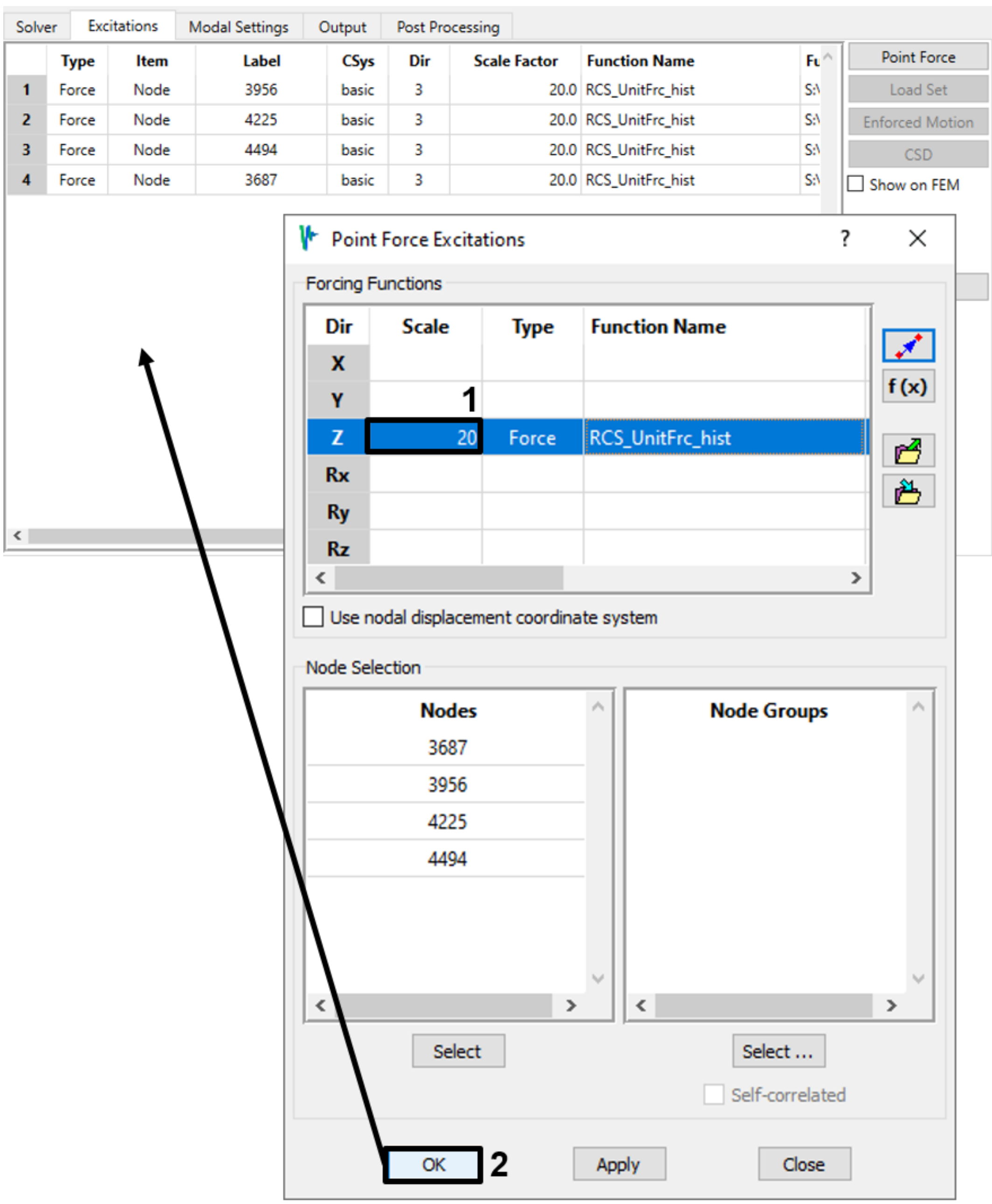


Figure 4-96. Scale the function by 20 and then create the final excitations.

6. Go to the *Modal Settings* tab. Make sure to designate the modes above 400 Hz as residual vectors and then set the damping. This model has too many modes to set individually, so bring up the damping dialog and use its damping schedule to set the damping of all modes to 1% (Figure 4-97). Rigid body modes will remain at 0% damping.

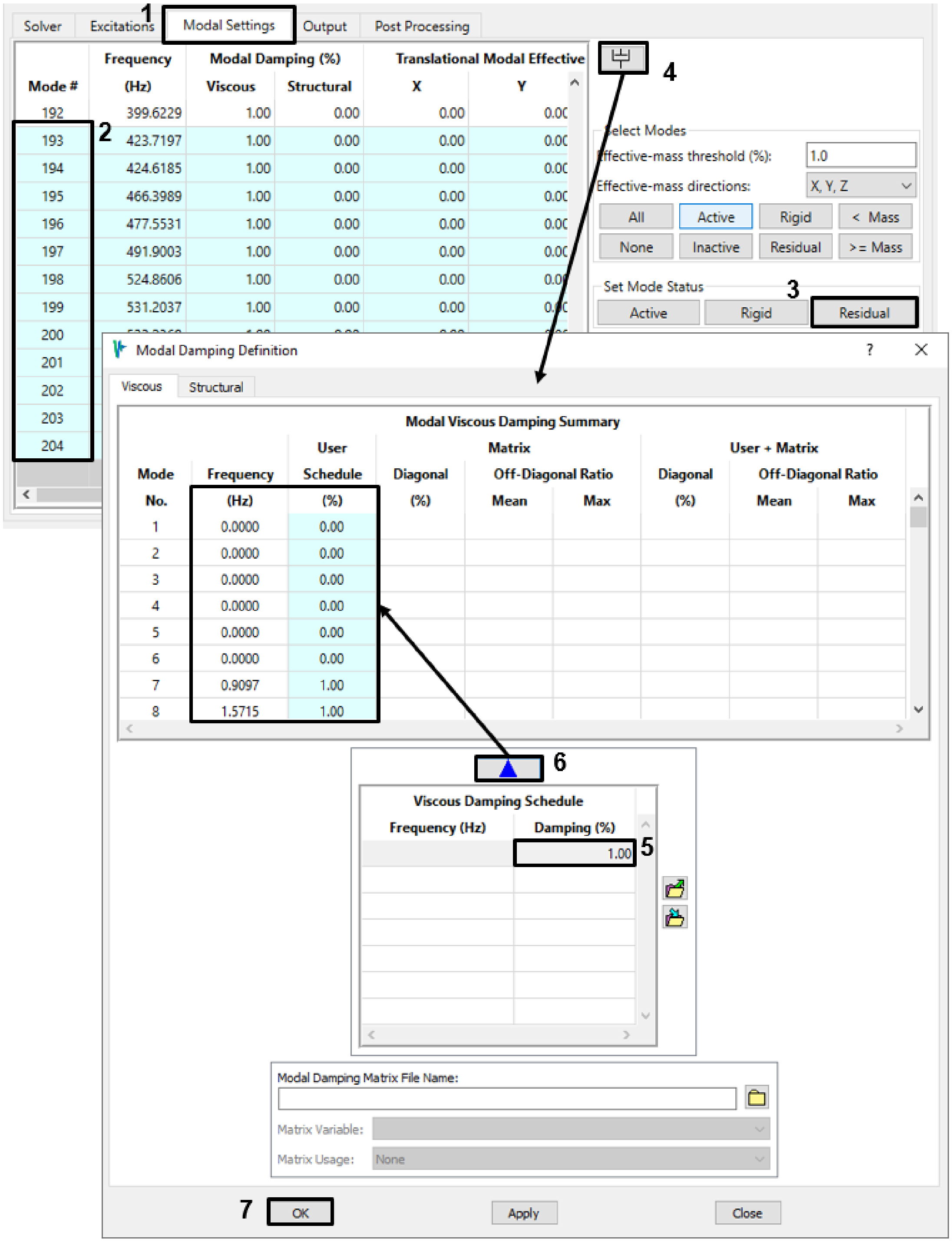


Figure 4-97. Use the damping schedule to assign 1% damping to all flexible modes.

7. On the Output tab, create a contour request for translational displacements and accelerations (UT and AT) for all nodes in the model. Request output at 0.05 second intervals to span the defined start and end times of the event (Figure 4-98); for a 4.0 second time span, this will produce 81 output time steps. Note that Vibrata will change your definition to its preferred form (inset) when you click **Apply** or **OK**; this will produce the same results as settings you entered. Click **OK** to create the request.

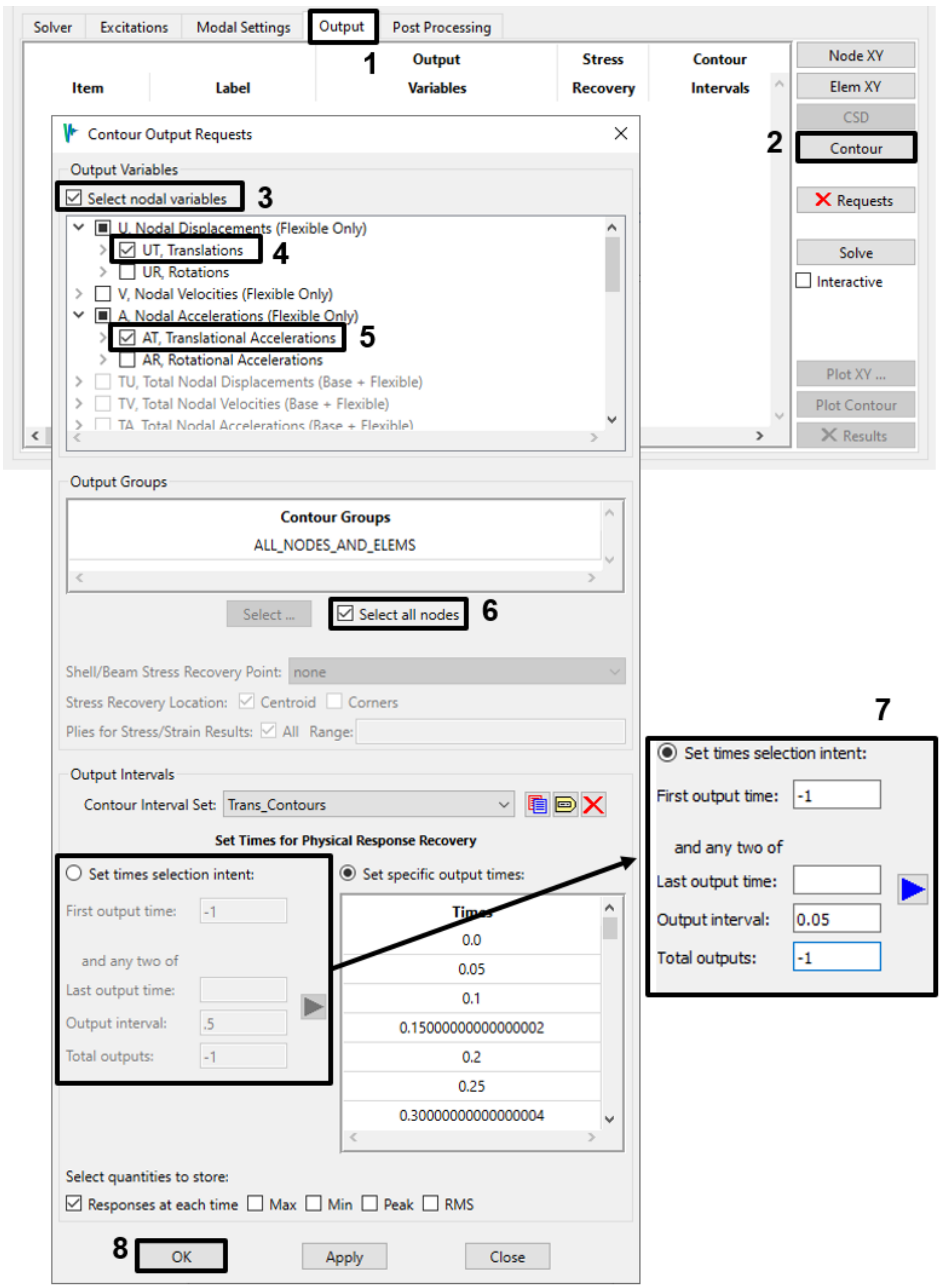


Figure 4-98. Request displacement and acceleration contours for all nodes at 0.05 second intervals.

8. On the Output tab again, click **Solve** to compute and store the requested contours (Figure 4-99). When the solver finishes, click **Plot Contour** to view the results in Femap. You can simply double-click in Femap's graphics window to start the animation. You can also do any of the usual postprocessing operations such as showing contours of total acceleration on a deformed mesh for a specific time step, as in Figure 4-100.

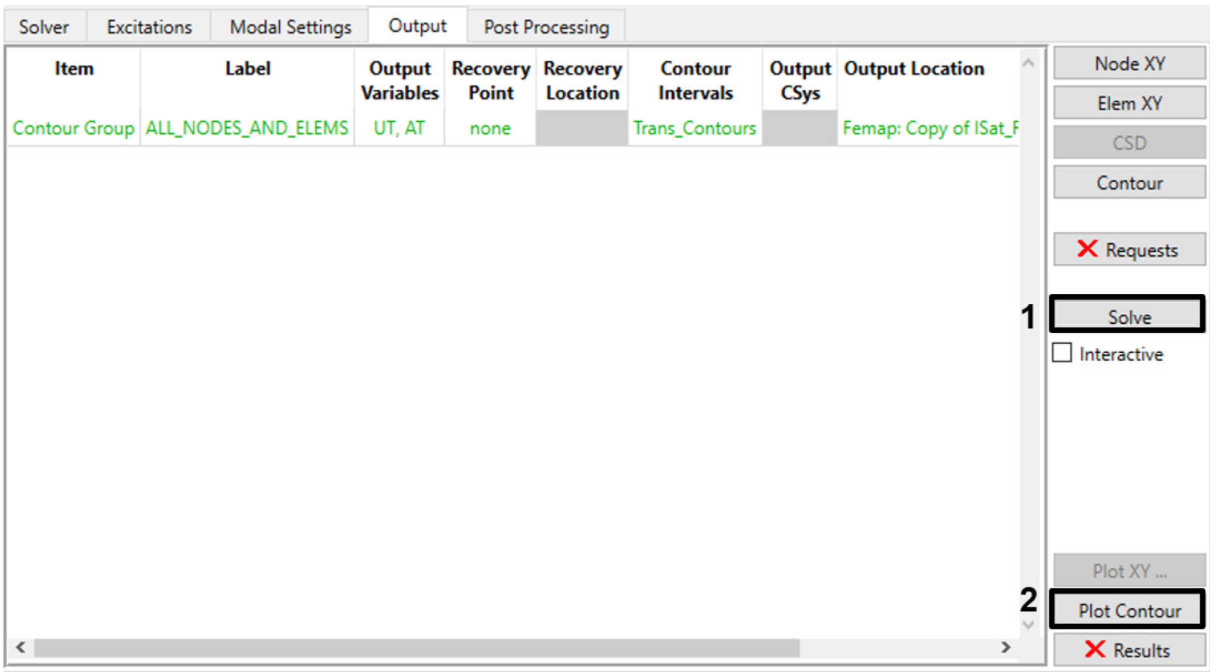


Figure 4-99. Solve for the requested contours and then plot them.

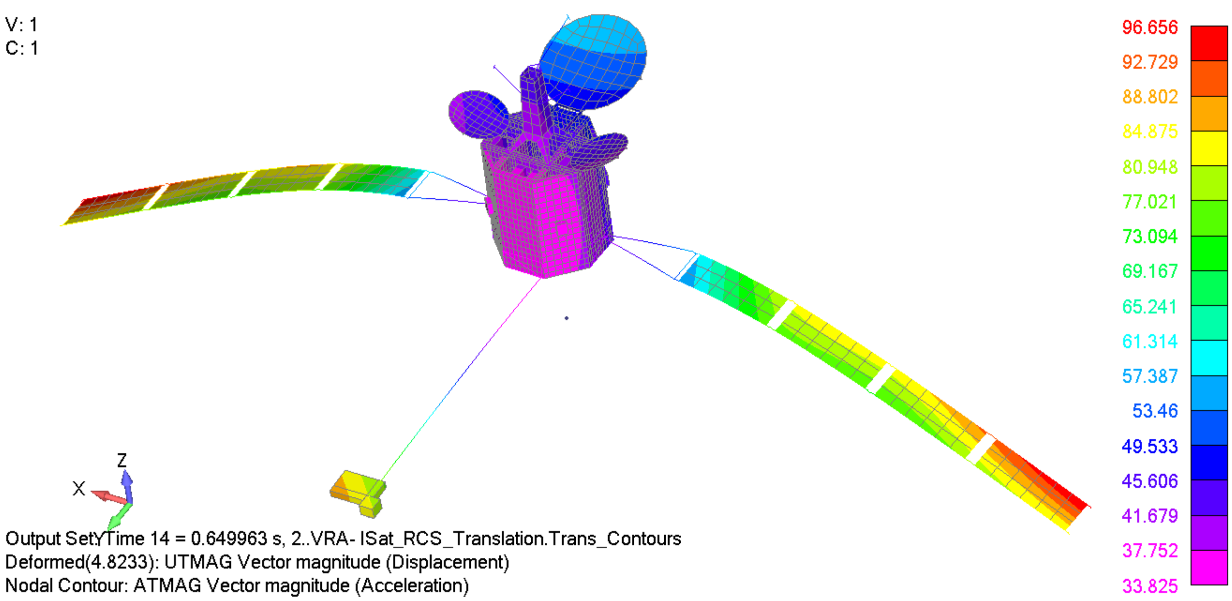


Figure 4-100. Contours of acceleration on deformed mesh for output frame 14.

9. Create another contour request. This time ask for von Mises stress in the RCS mounting panels, but request only the peak value found in each element over the same time steps used for the displacements. You can create the new interval definition by copying the first one and then changing the settings of the **Select quantities to store** toggles (Figure 4-101). Click "Yes" to close the "Output Requests Verification of Intent" Window.

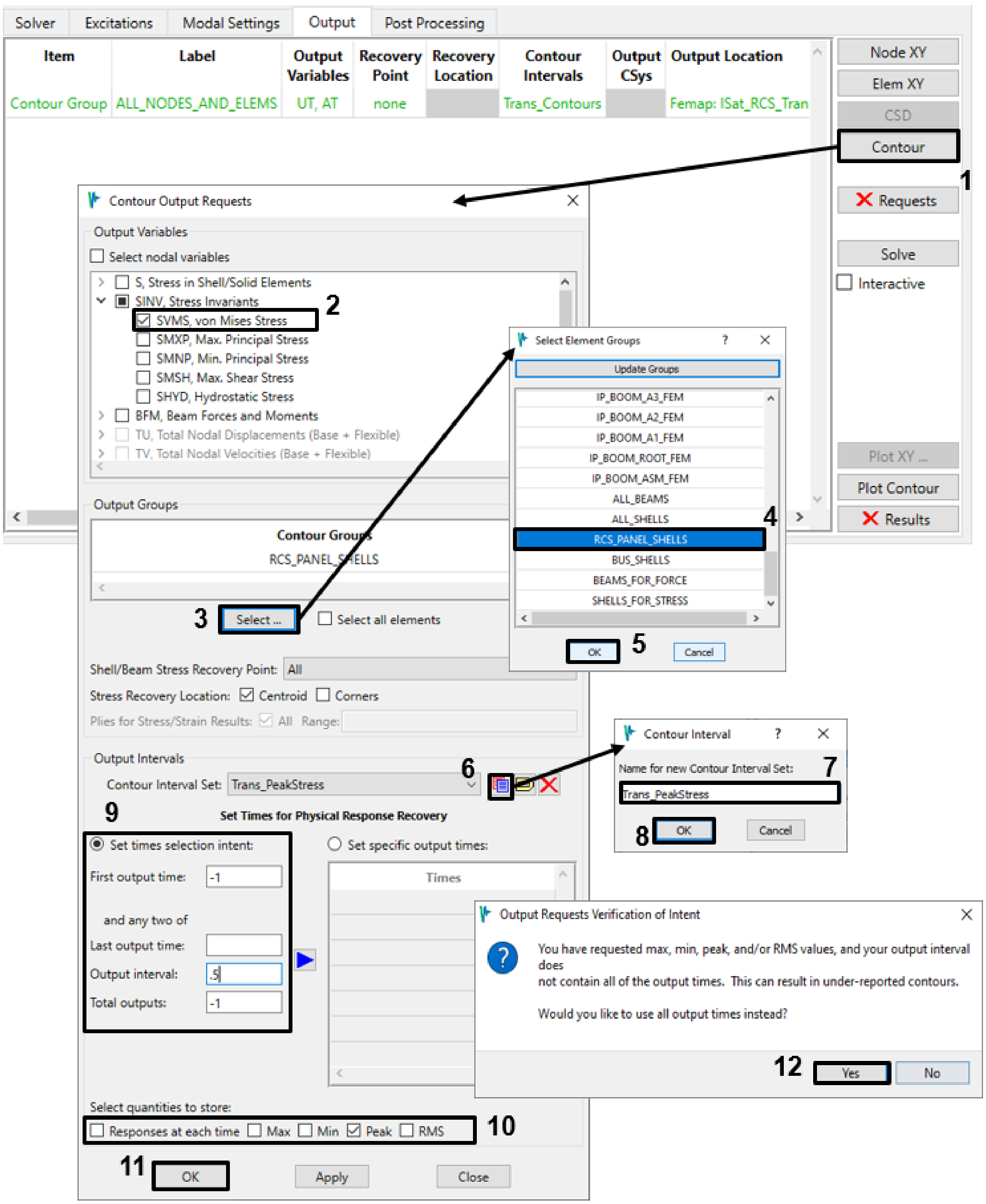


Figure 4-101. Request peak values of von Mises stress in the RCS mounting panels.

10. Solve the new contour request. When the solver finishes, select the new contour request in the table, then click **Plot Contour** (Figure 4-102). Vibrata will set Femap to display the results from the selected request (Figure 4-103). Since we only requested stresses for the RCS_PANEL_SHELLS group, you may want to display only that group (Figure 4-104).

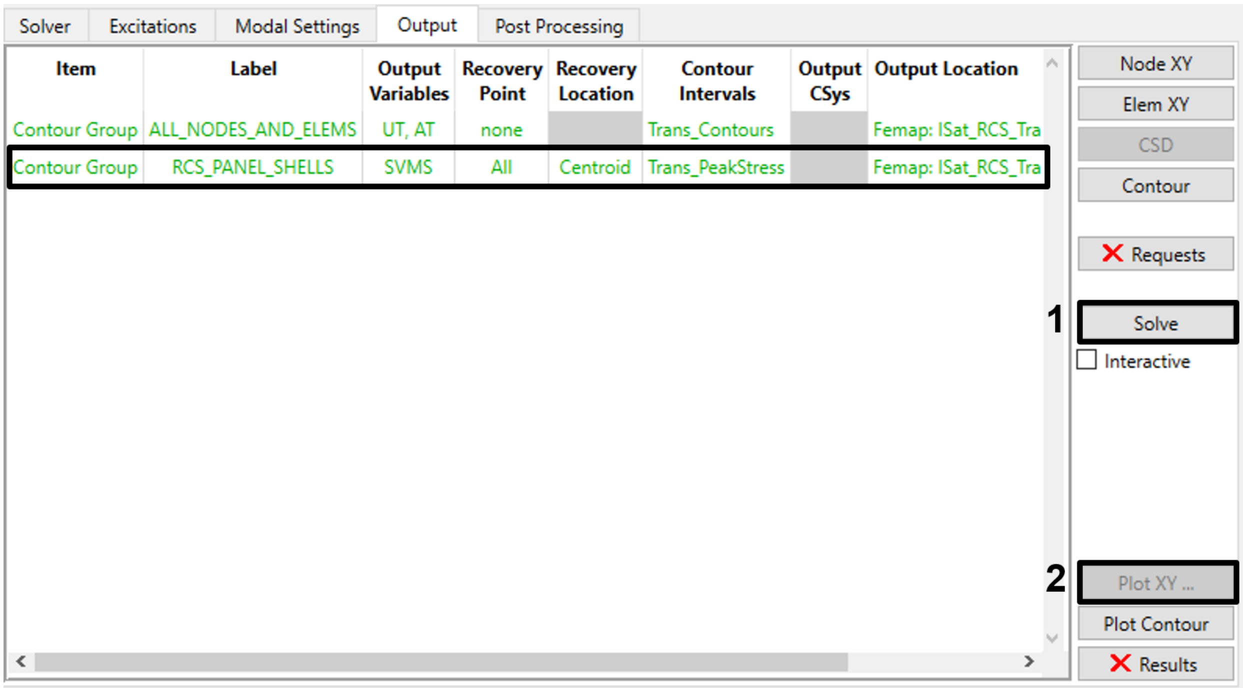


Figure 4-102. Select the RCS Panels contour request and plot its results.

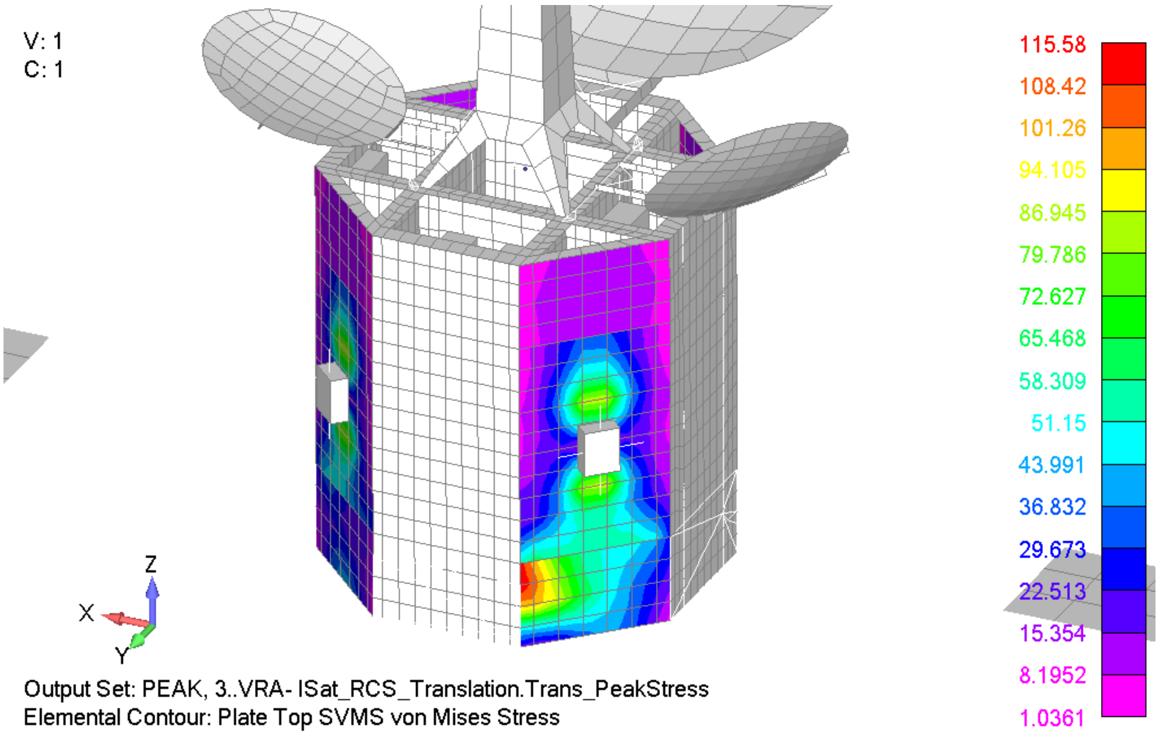


Figure 4-103. Peak von Mises stress contours in RCS panels.

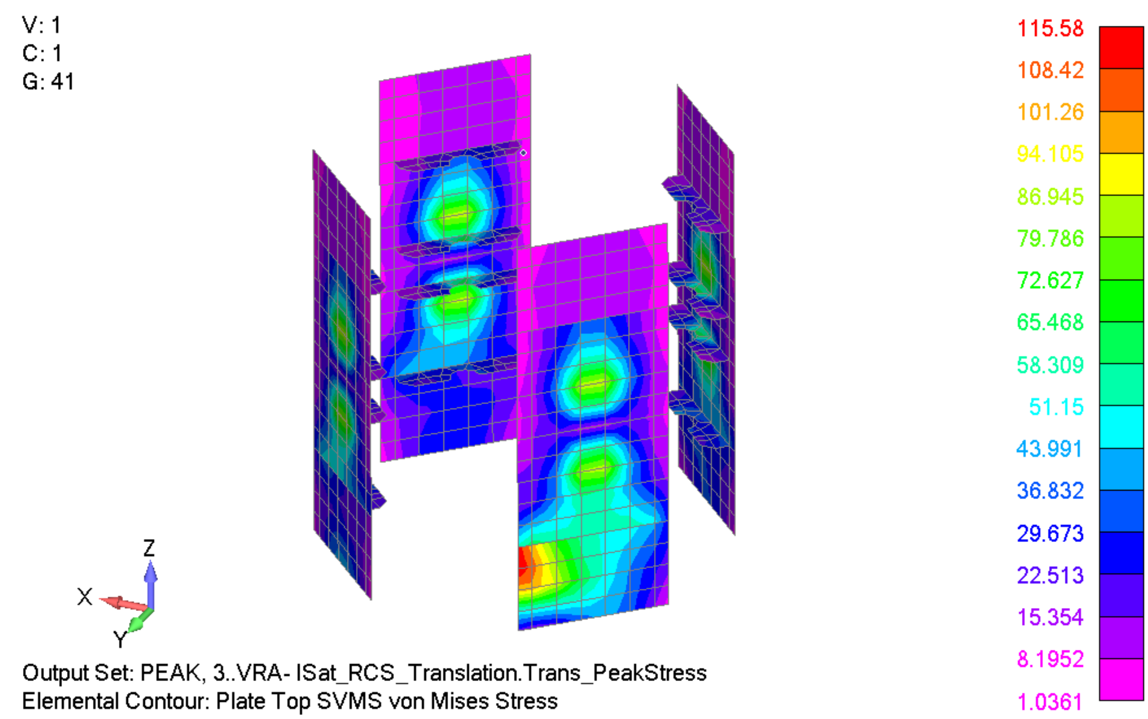


Figure 4-104. Von Mises results with only the RCS panels group displayed.

11. There appears to be a significant stress concentration where the instrumentation package support attaches to the bus. Create a contour request for the peak stresses in the BUS_SHELLS group with the same settings used for the RCS panels. Solve this new request, then select it in the table and click **Plot Contour** again (Figure 4-105). This time, display only the BUS_SHELLS group in the contour plot (Figure 4-106). There is indeed a stress concentration, and the stress is nearly seven times higher than in the RCS panels.

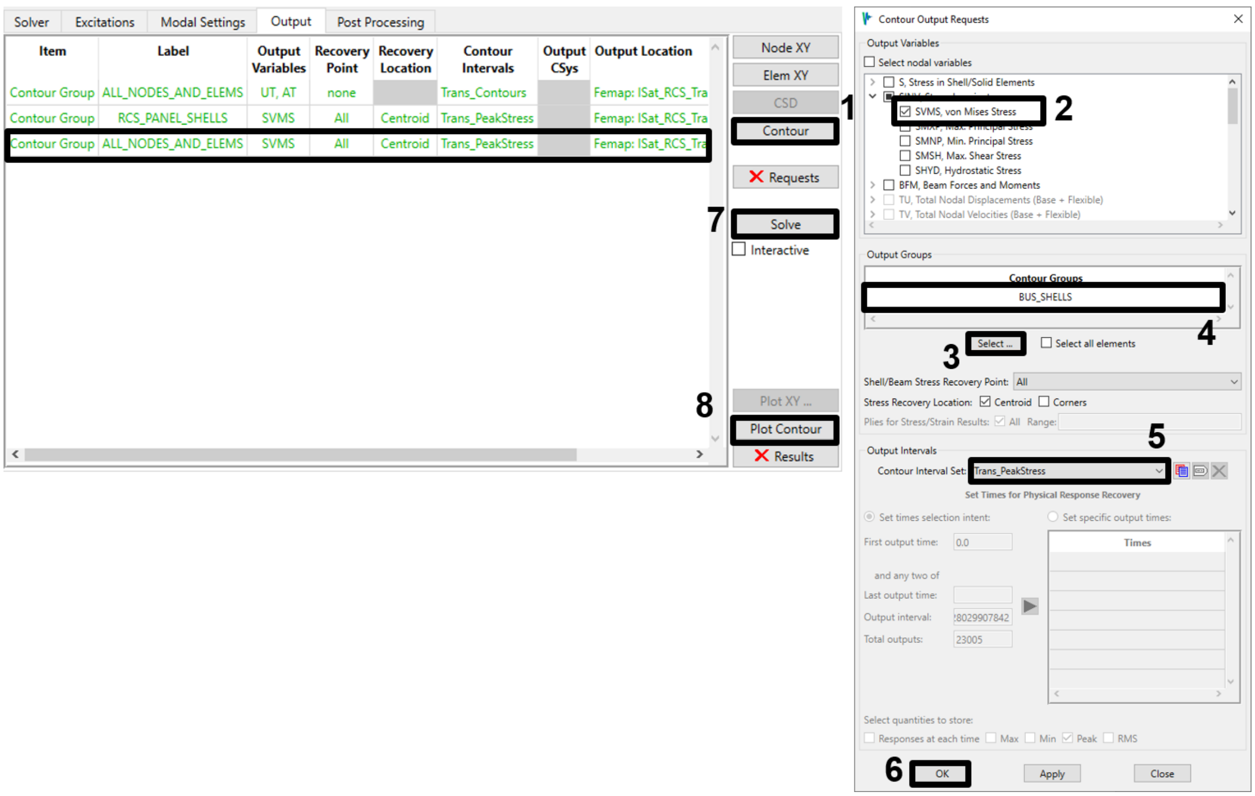


Figure 4-105. Select the BUS_SHELLS contour request and plot its results.

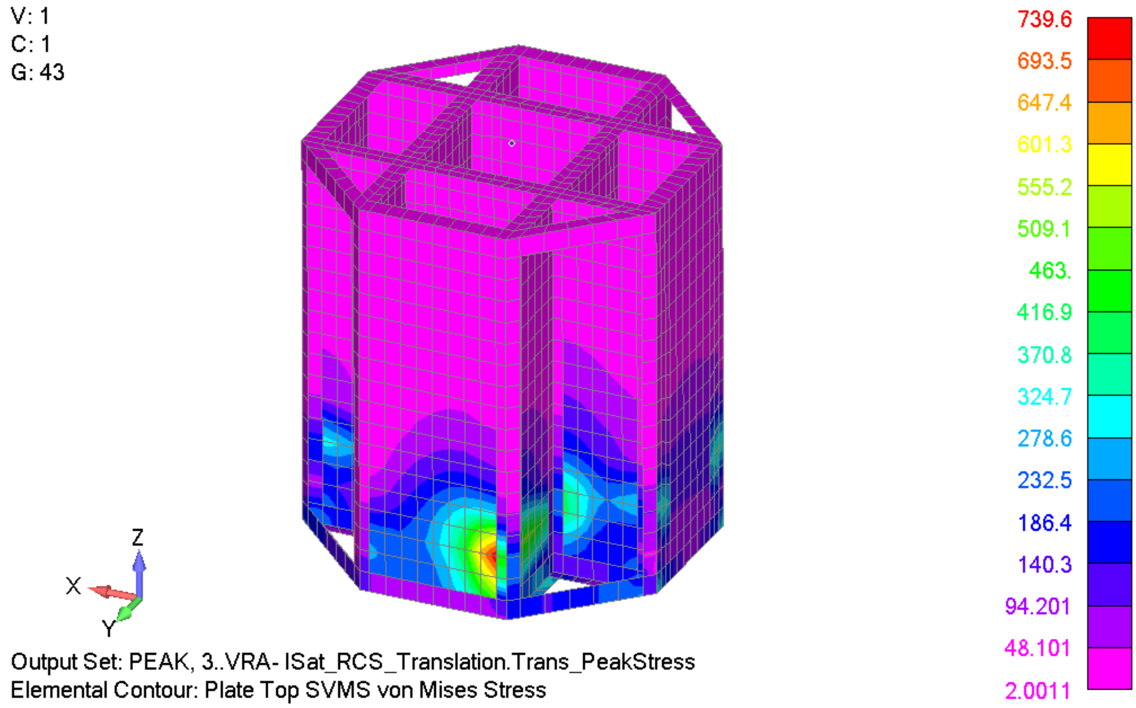


Figure 4-106. Von Mises results with only the BUS_SHELLS group displayed.

12. Now we want to be sure that the solar arrays and instrument package do not fall off due to these thruster firings, so create Element XY output requests for beam forces and moments in the elements that attach them to the spacecraft bus. Note that the Stress Recovery Location text is active, and Corners are checked but grayed out. For BFM results, you must request Corner output since BFM results are not available on the Centroid. (Figure 4-107).

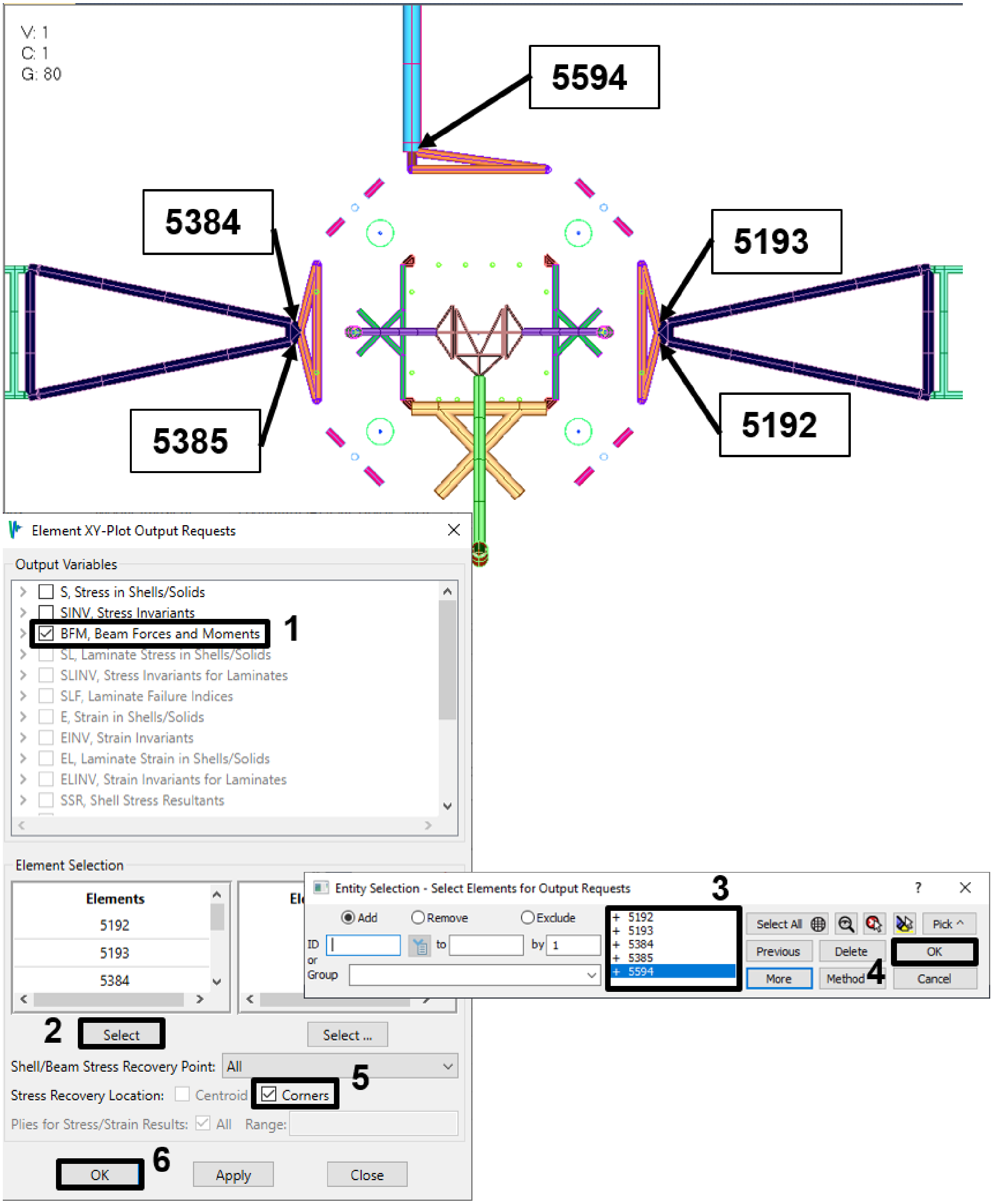


Figure 4-107. Request beam forces in elements that attach appendages to bus.

Solve again and then click **Plot XY** (Figure 4-108). The forces in the instrument package boom element are shown in Figure 4-109, which also shows when the thrusters are firing. Note that we have displayed the forcing function on the plot to make clear when thruster firing starts and stops. Display the other element responses as you wish.

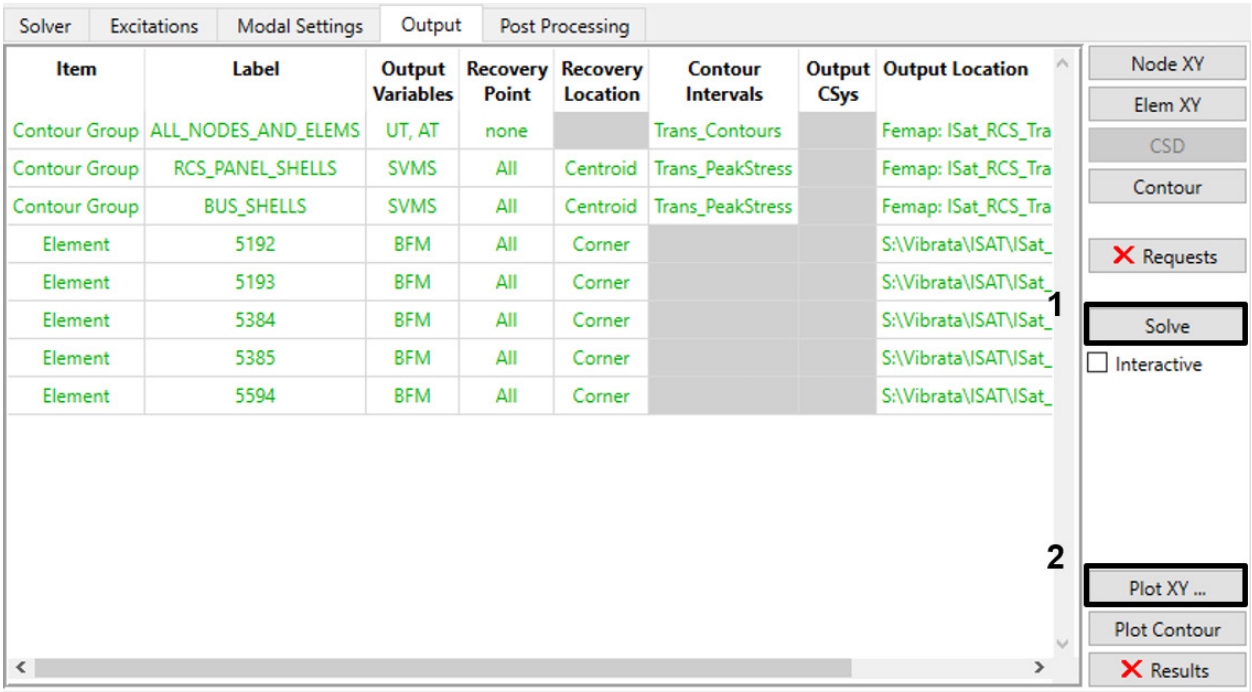


Figure 4-108. Solve the new Element XY requests and plot them.

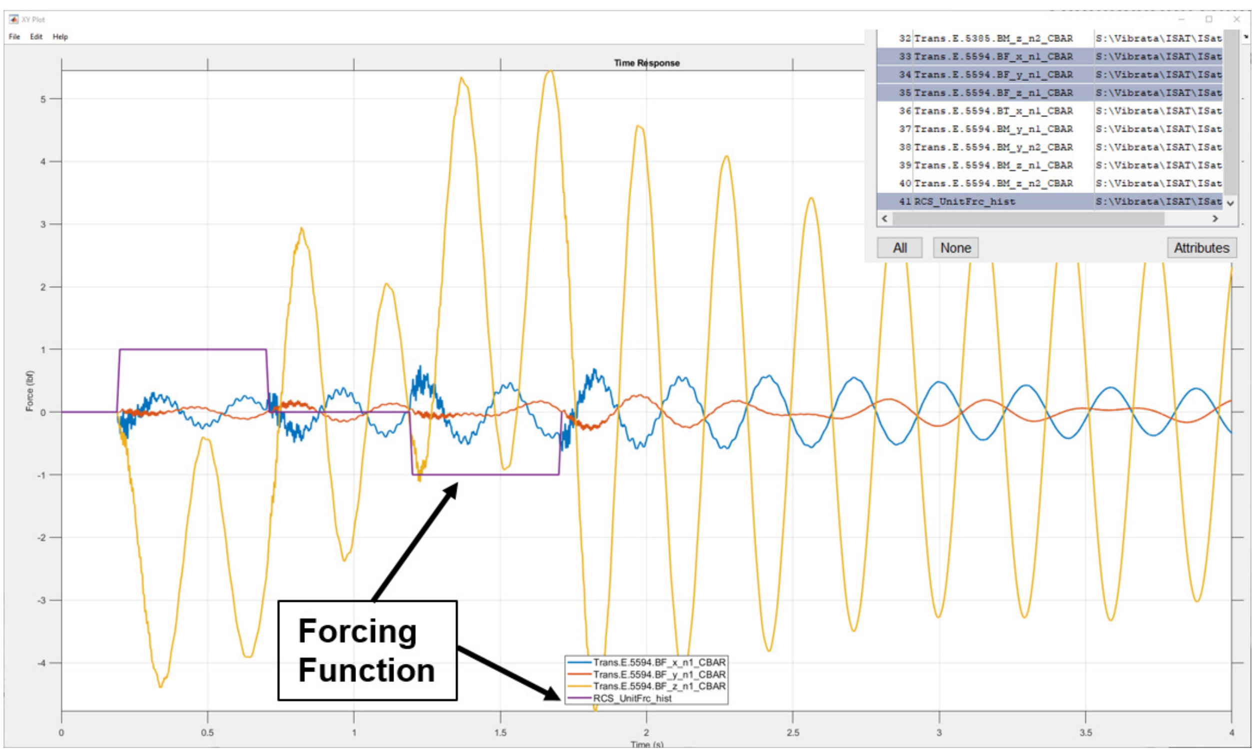


Figure 4-109. Forces in IP boom connector; note effects of thruster start/stop.

4.5.2. <u>Frame Model Transient Animation.</u>

1. If you have not already done so for a previous example, import the frame model "frame01_modes.dat" into Femap. Save the model file as "frame.modfem." Figure 4-110 shows the model with notes about nodes that will be of special interest for this analysis, either for defining input or for recovering physical responses.

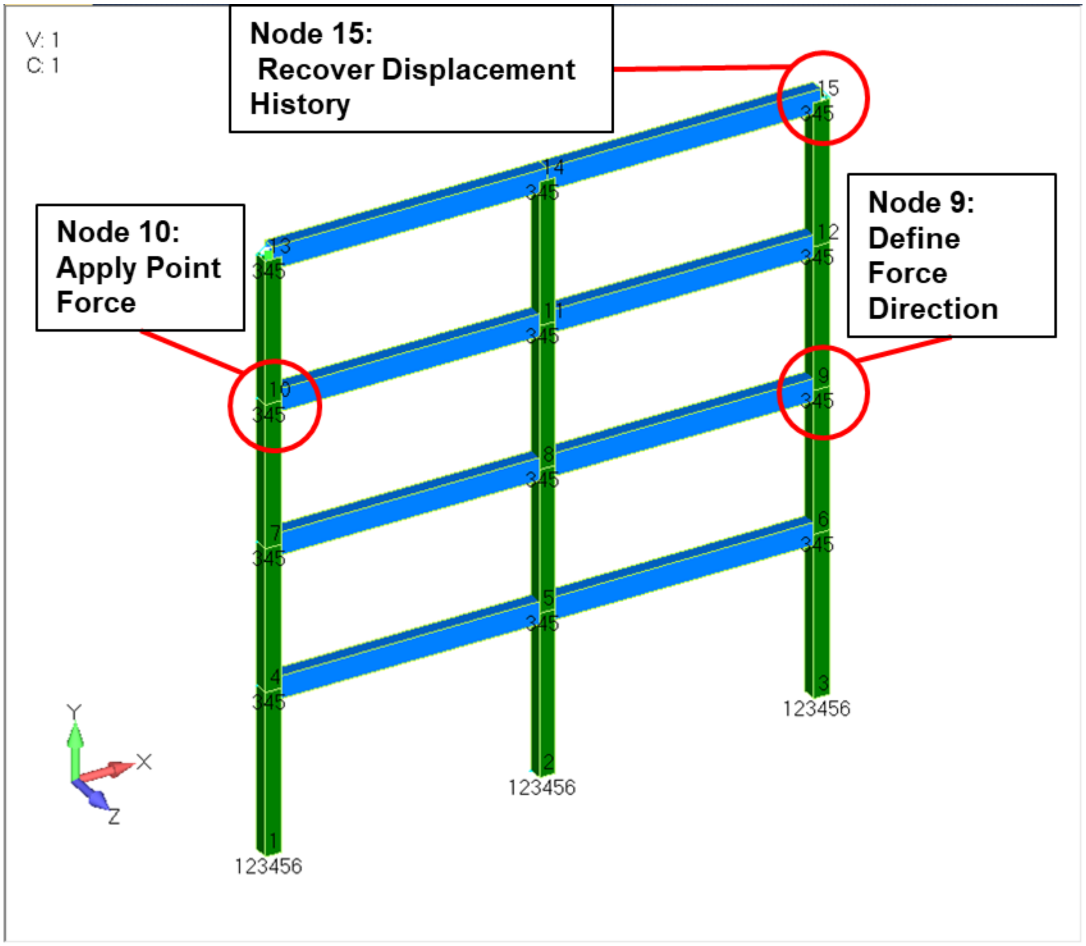


Figure 4-110. Frame model showing nodes of special interest.

2. In the Event Manager, set up a new transient event and select the frame's modfem file and the OP2 file "frame01_modes.op2". Set the event end time to 10 seconds and its initial conditions to Zero, as shown in Figure 4-111. As always, be sure to assign a recognizable event name.
3. Save the *.evt file using a recognizable event name.

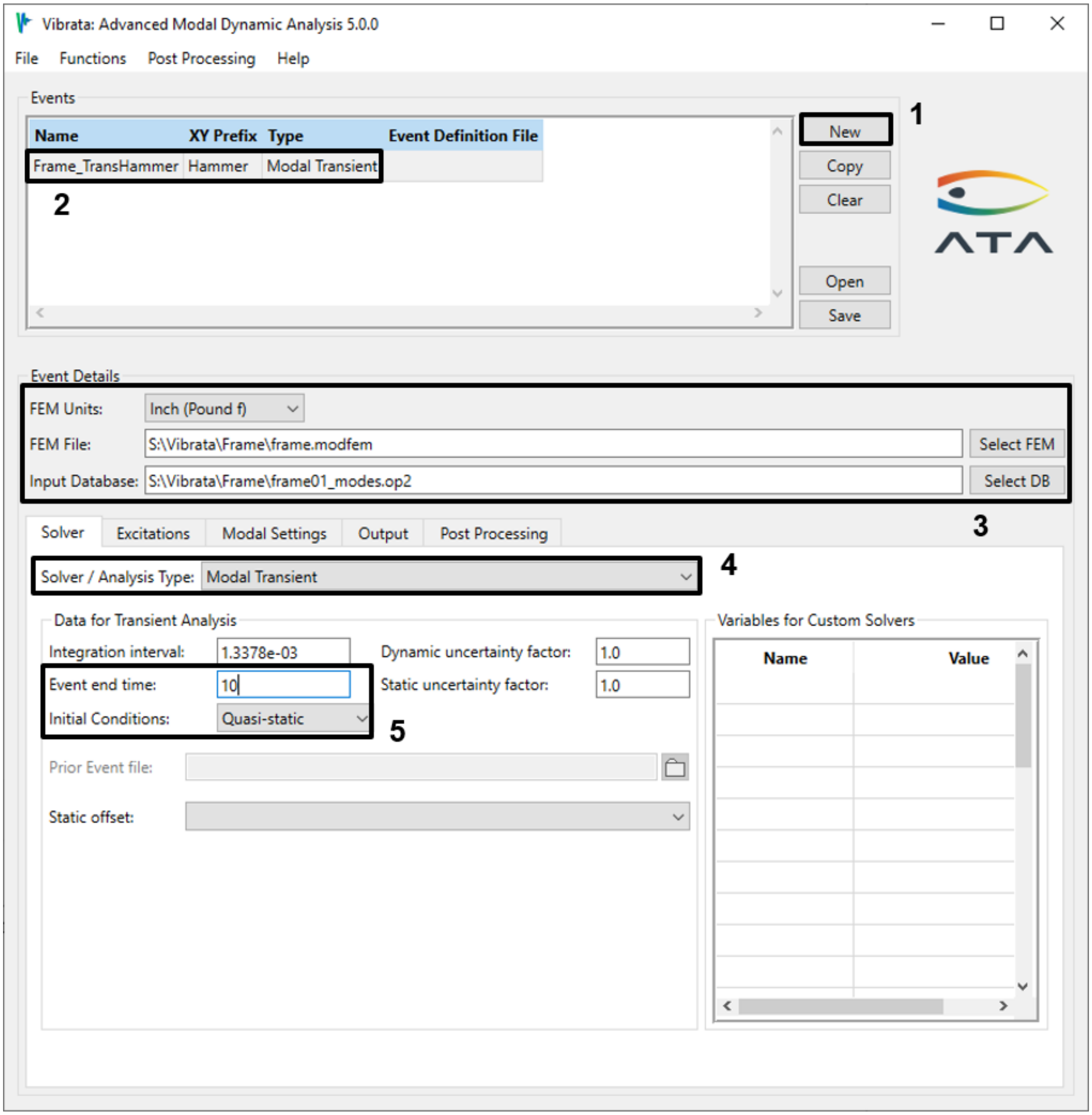
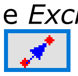


Figure 4-111. Solver setup for frame model transient analysis.

4. Go to the *Excitations* tab and create a point force load. Apply the force at node 10. Use the  button to set the force direction (Figure 4-112). You can screen-pick the nodes (from node 10 to node 9) or you can key in the labels using Femap's standard node selection dialog. When you click **OK** on that dialog, Vibrata will ask you to confirm that the displayed direction is correct (Figure 4-113).

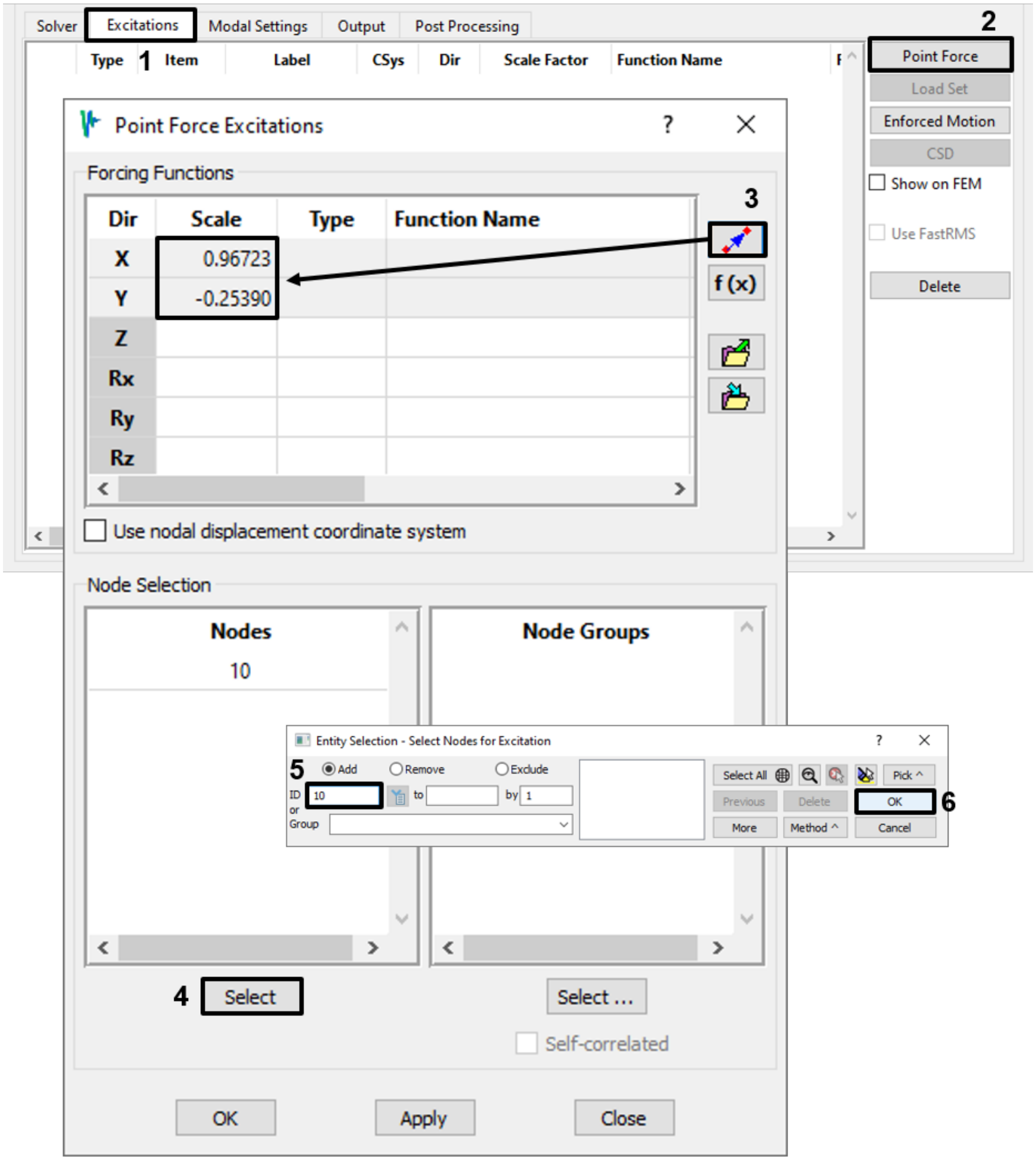


Figure 4-112. Forcing function will be applied at node 10 in a direction taken from the FEM geometry.

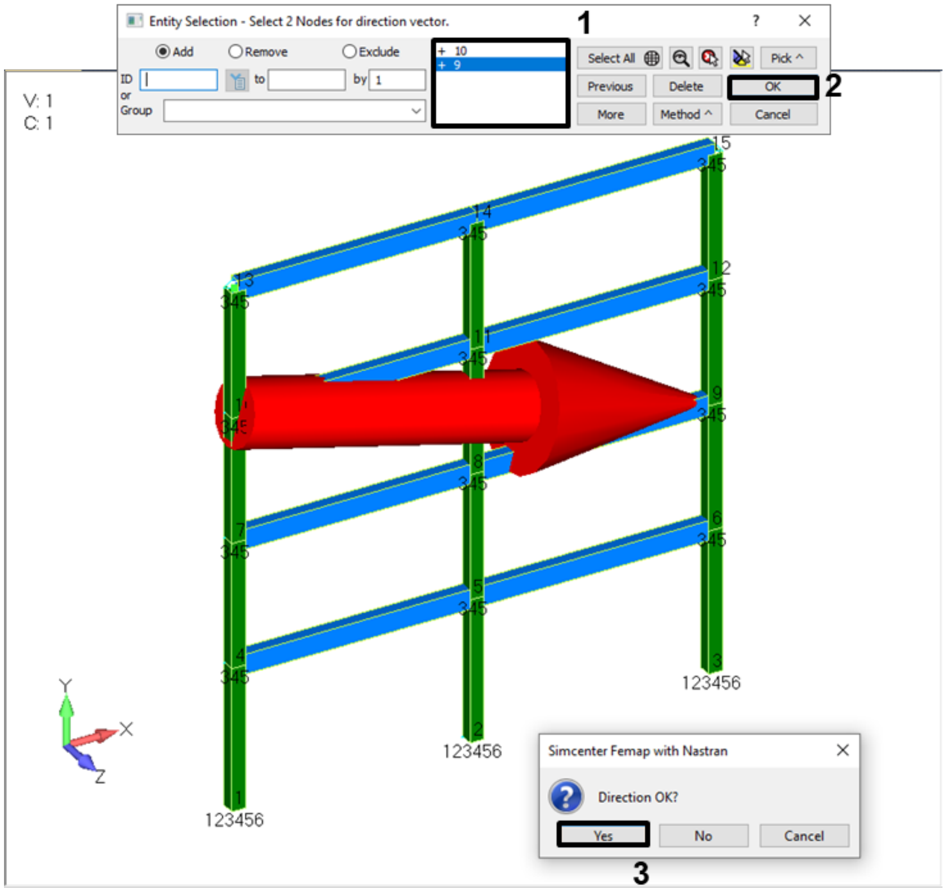


Figure 4-113. Screen-pick node 10, then node 9, to define the force direction.

5. With the *Point Force Excitations* dialog set per Figure 4-112, use the  button to bring up the Function Manager. If you have already run the examples in section 4.2.3, open the "frame_excitations.fcn" file used there; if that file does not currently exist, you will create it when you have finished defining the current function. Define a triangular pulse as shown in Figure 4-114. Note that you enter the pulse parameters (*Start, Width, Amplitude, End*), not the specific X and Y data points; when you click **Apply**, the Function Manager automatically makes the appropriate data point entries. This force roughly models a hammer blow to the structure. When all the definitions are complete on the *New Function* dialog, click the **Create** button. Store the function to "frame_excitations.fcn" (Figure 4-115). Click **Done** on the Function Manager main dialog to return to *Point Force Excitations* dialog, and click **OK** there to finalize the excitations (Figure 4-116).

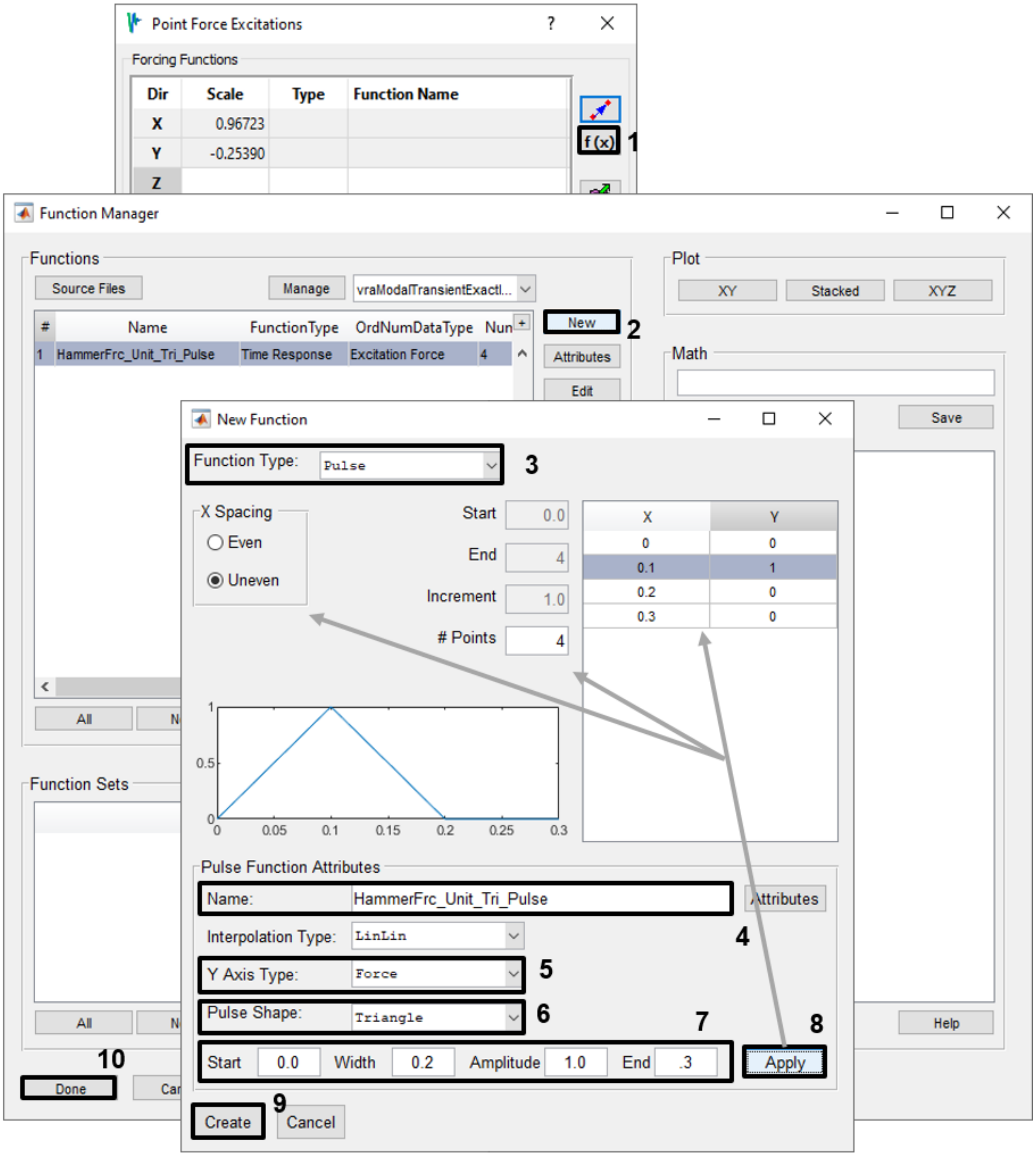


Figure 4-114. Define a unit force triangular pulse function.

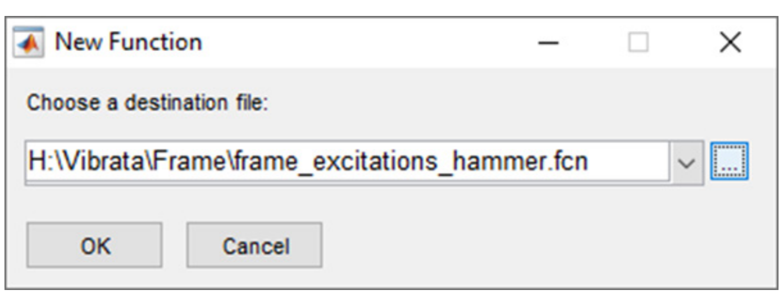


Figure 4-115. Save the new function to the frame excitations file.

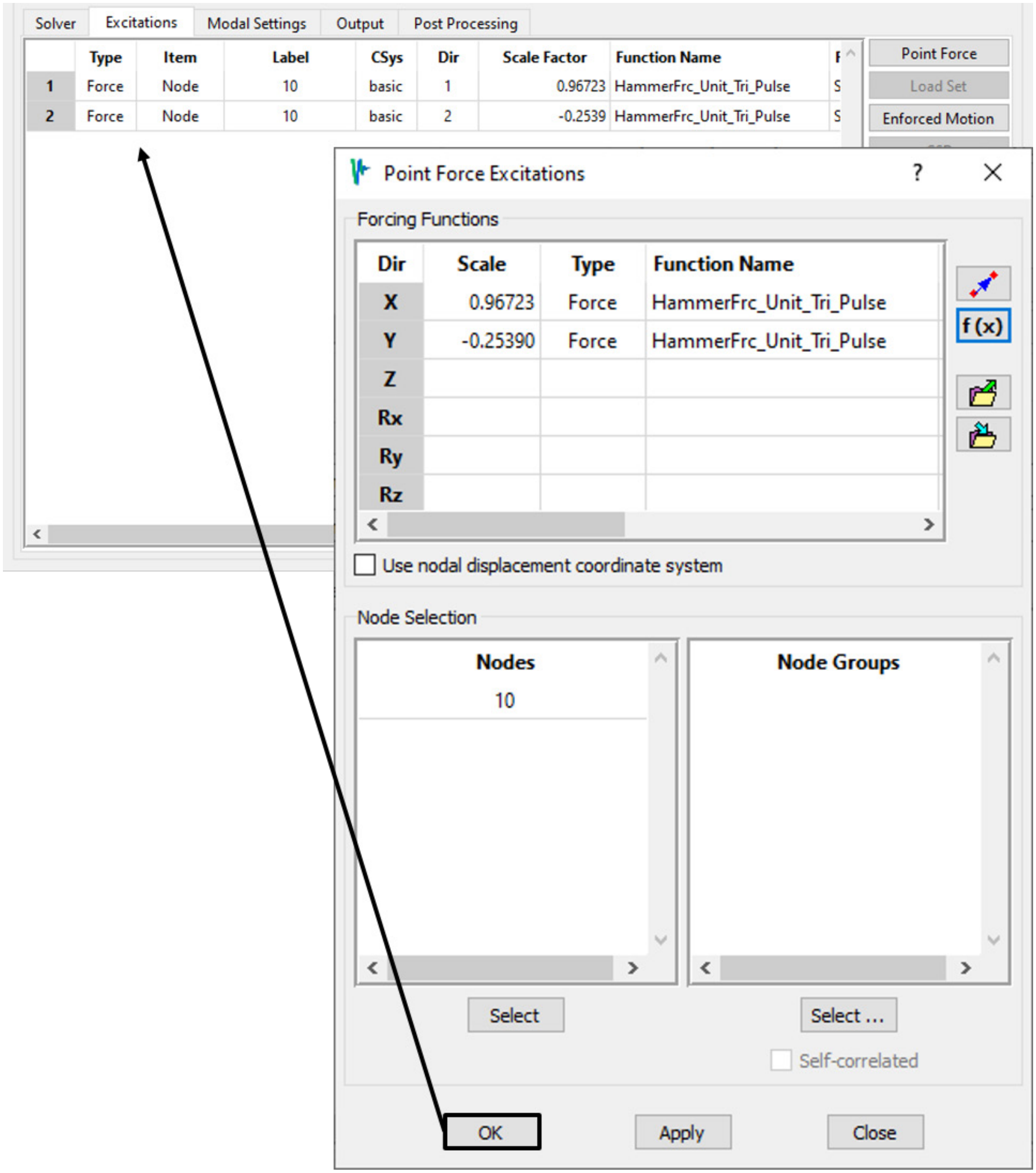


Figure 4-116. Apply the forcing function to the Vibrata event.

6. Go to the *Modal Settings* tab, select modes 11–15, and set their status as Residual modes. Next, open the damping dialog. Use the damping schedule to set the viscous damping to 1% for all modes, then key in damping of 10% for mode 1 and 5% for mode 2, as in Figure 4-117.

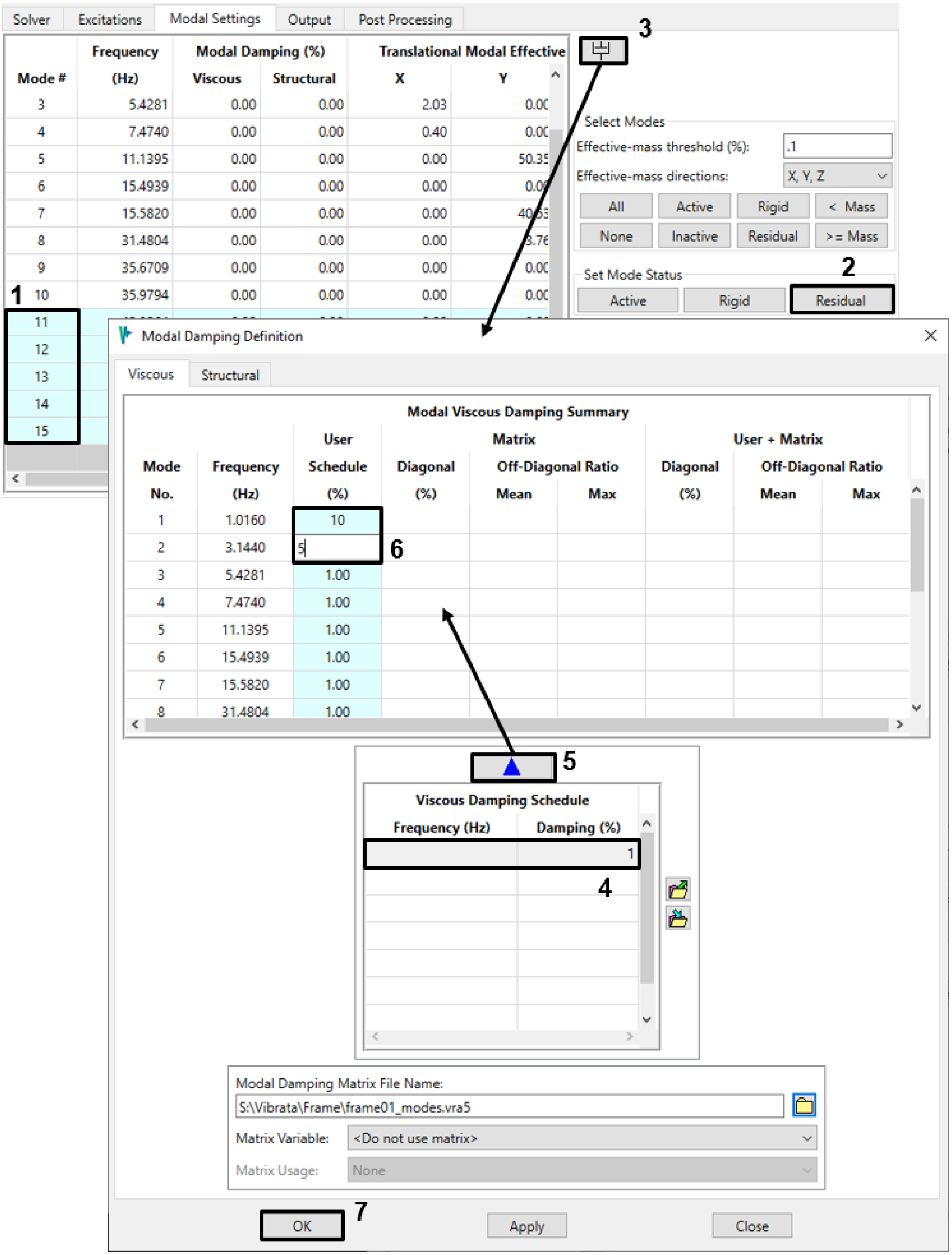


Figure 4-117. Set the damping of the model to 10% for mode 1, 5% for mode 2, and 1% elsewhere.

7. Go to the *Output* tab. To get a measurement of the structure's movement over time, create Nodal XY Plot output request for X and Y displacements (*Utx*, *Uty*) and displacement magnitude (*UTMAG*) for node 15, as in Figure 4-118.

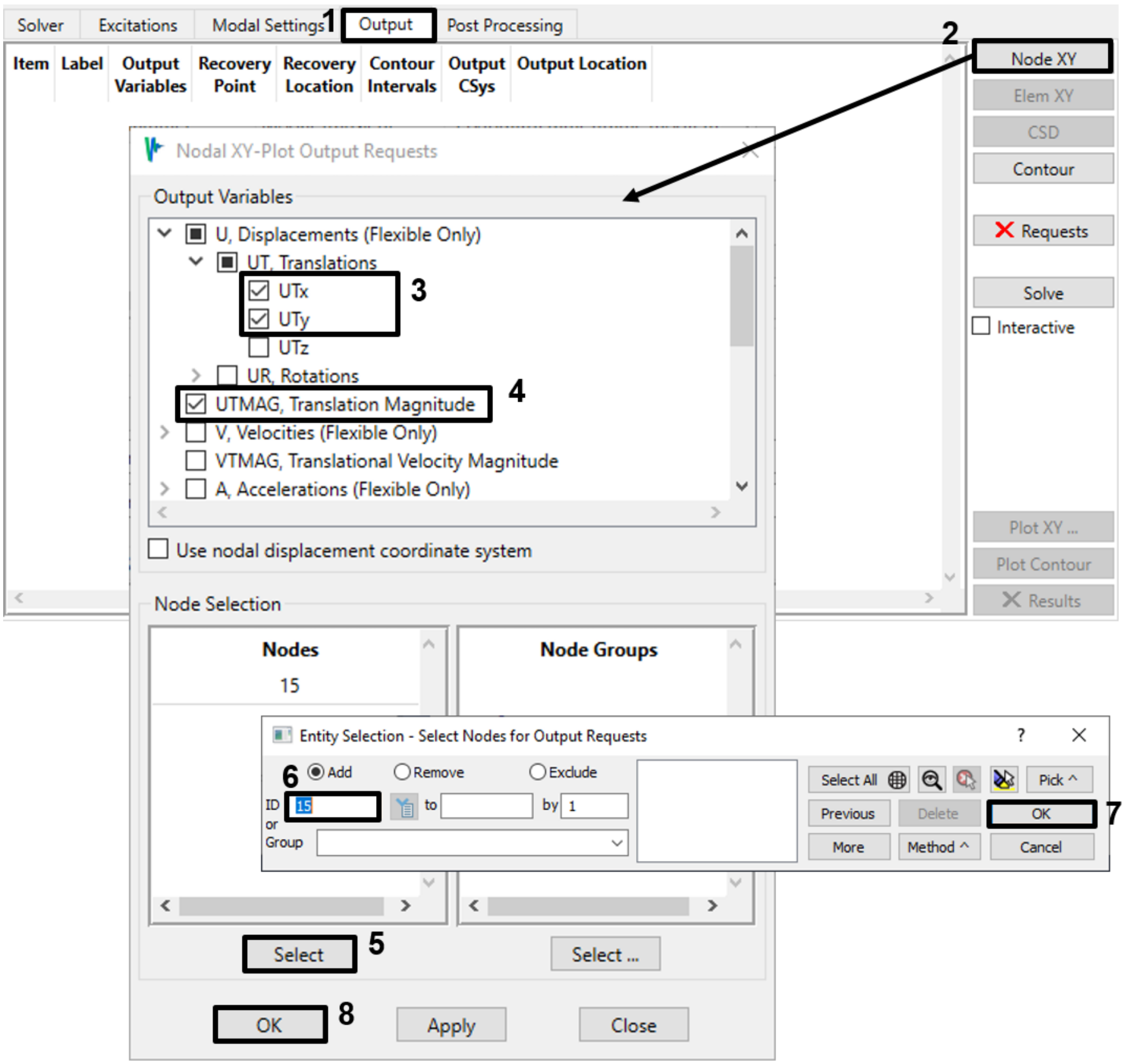


Figure 4-118. Request displacement plots for node 15.

8. In order to create an animation of the structure in motion, create a contour request for displacement (UT) at all nodes (Figure 4-119). Enter 0.0 for the *First output time* and 3.0 for the *Last output time* to create contours from 0.0 to 3.0 seconds. We have selected 3 seconds as the end time because the structure's response will decay to about 30% of its maximum by then (see Figure 4-120). Set the *Output interval* to 0.025 (seconds) and leave *Total outputs* blank. This will produce 121 time frames for

the animation. (Alternatively, you can specify how many animation frames you want using *Total outputs*, and let Vibrata determine the time step needed to fill the output time span with that many frames.) Select *Responses at each time* to store the results from each time step as an animation frame. Click **OK** to create the request and return to the *Output* tab.

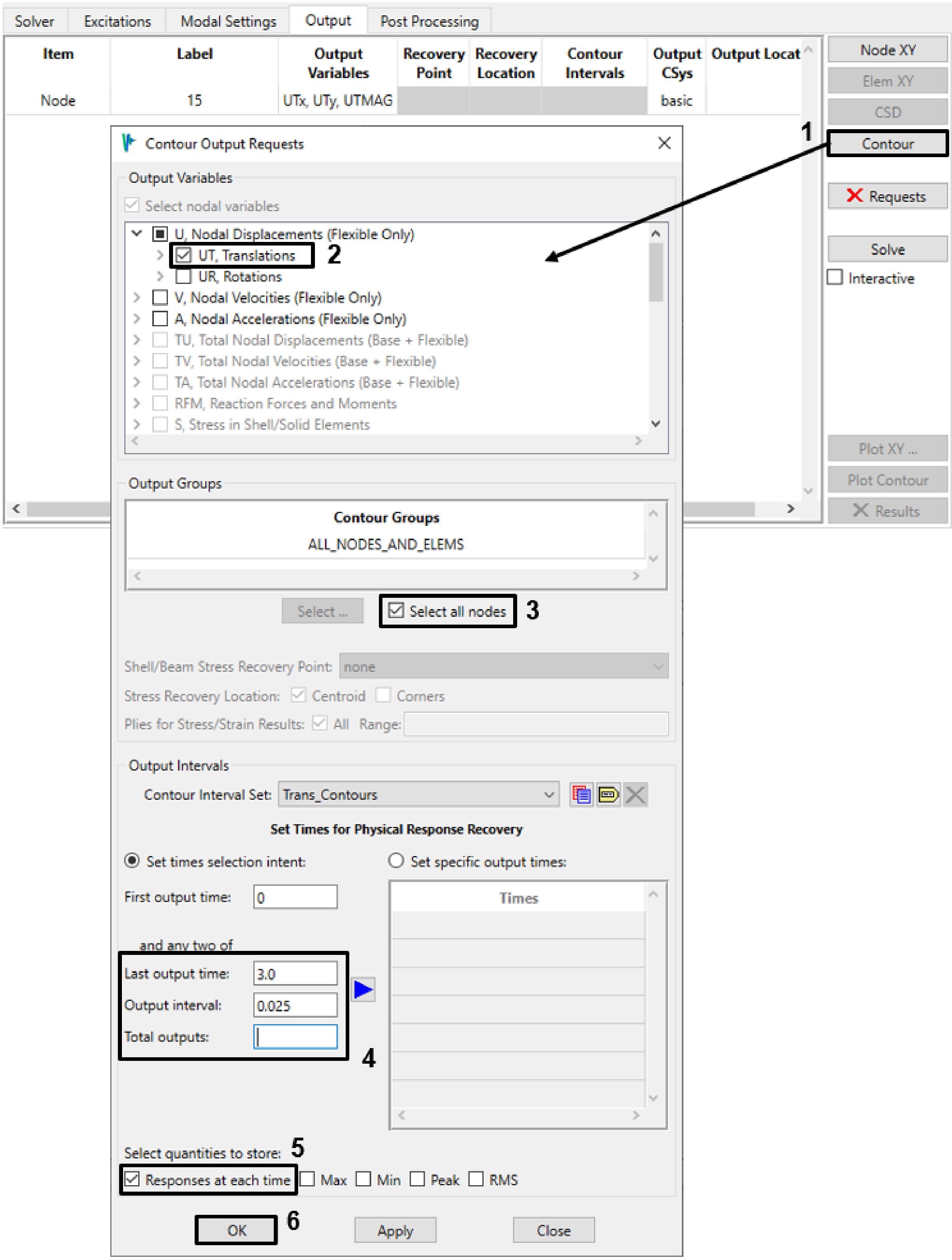


Figure 4-119. Request displacement (*UT*) contours for the first 3 seconds of the event.

9. Solve the event. Click the **Plot XY** button to view the requested nodal displacement histories. These are shown in Figure 4-120 using the *stacked* plot mode. Note the different orders of magnitude on the Y-axis labels.

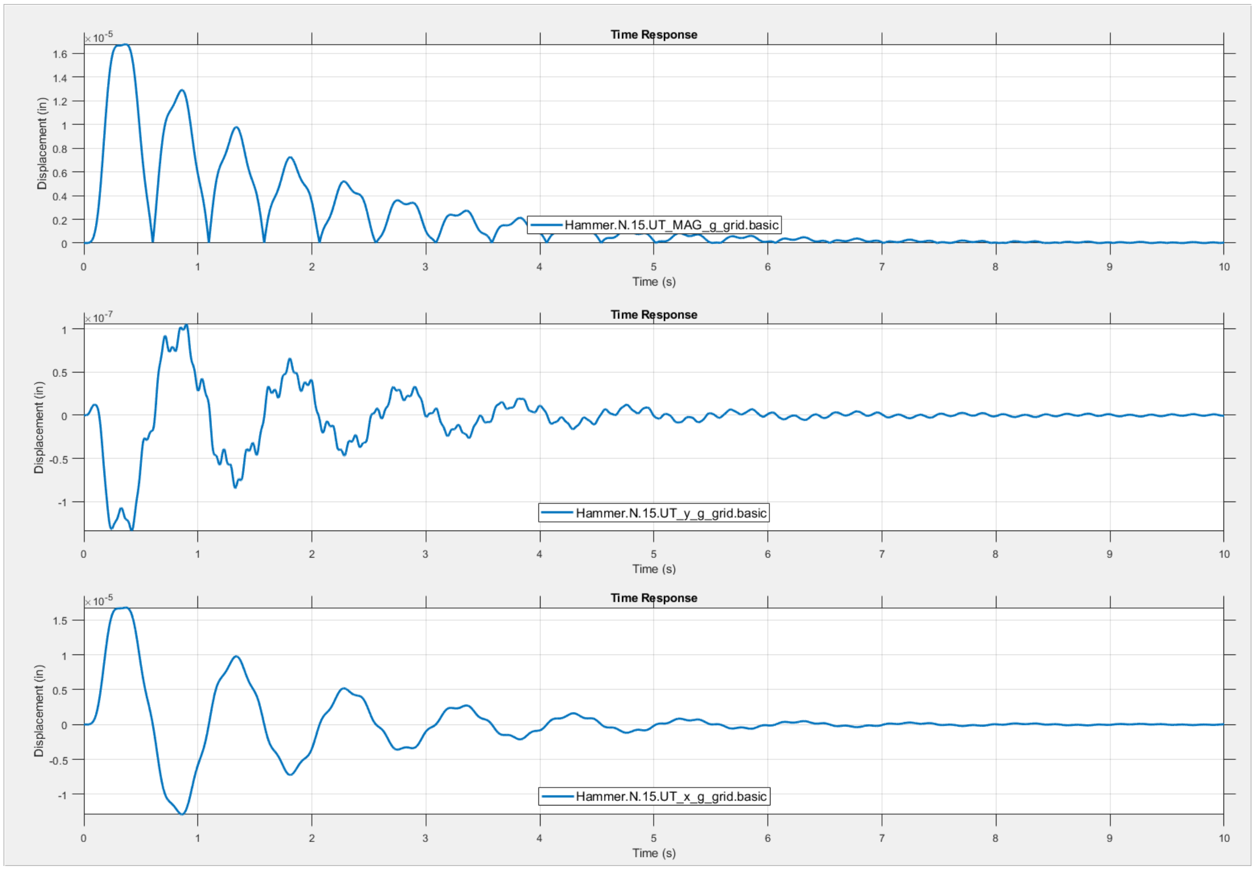


Figure 4-120. Transient displacements for node 15.

10. Plot and animate the contours by clicking the **Plot Contour** button. Double-click anywhere in Femap's graphics window to start the animation. Adjust Femap's display parameters in any way you find helpful.

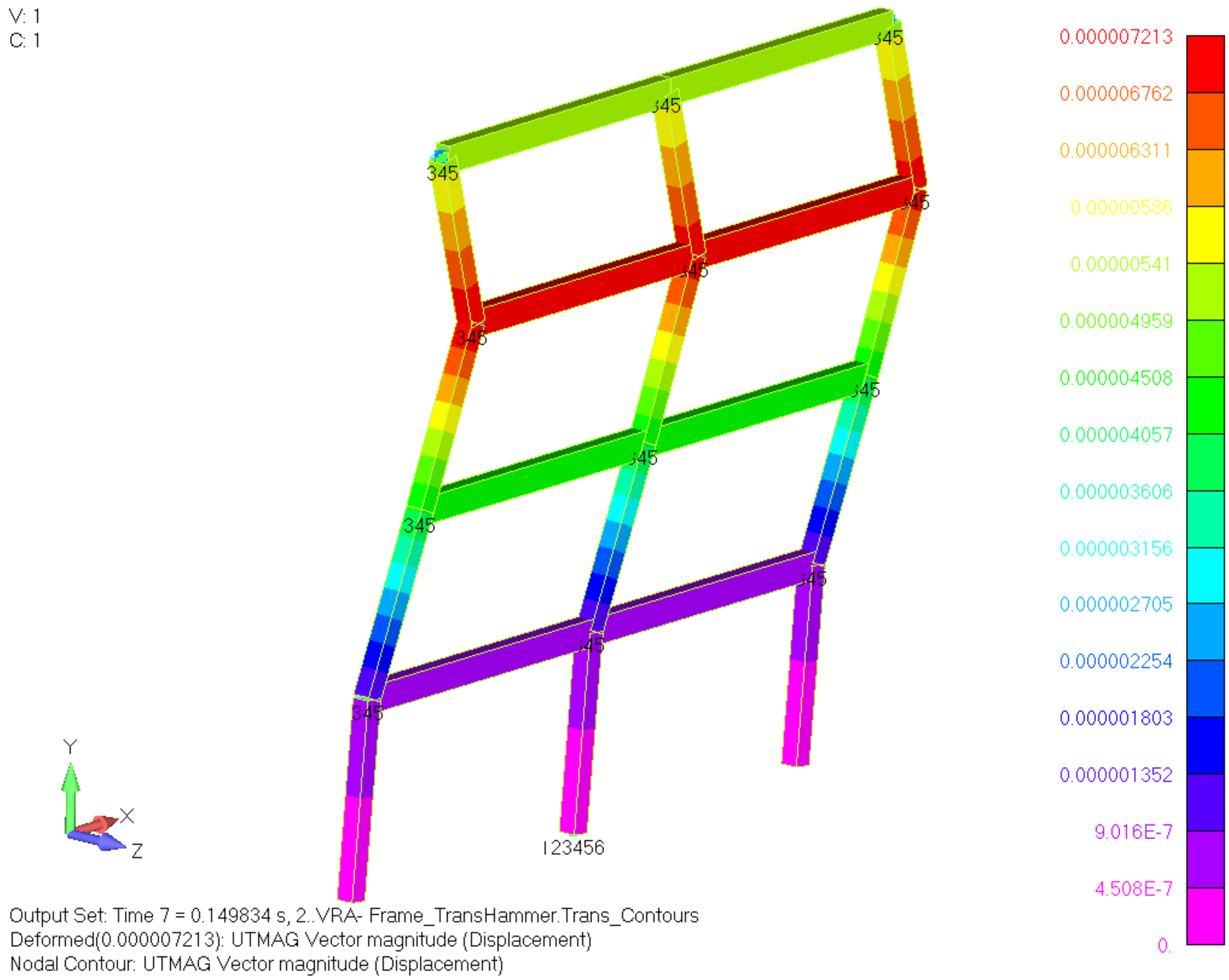


Figure 4-121. Frame 7 of the displacement animation.

5. CREATING CUSTOM FUNCTIONALITIES

5.1. Creating Custom Solvers

5.1.1. The Solver File

5.1.2. The SolverInfo File

5.1.3. The Custom Solver Folder

5.2. Creating a Custom Nonlinear Model

5.3. Creating Custom Postprocessing

----

6. VIBRATA-MATLAB API FOR CUSTOM SOLVERS AND FUNCTIONS

6.1. Directory Structure

6.2. Function Naming Convention

6.3. Utility Classes

6.3.1. vraParam—Vibrata Parameter Class.

6.3.2. vraReqmap—Vibrata Request Mapping Class.

6.3.3. *fcn*—Vibrata Function Class.

6.3.4. vraNonlinearParam—Vibrata Nonlinear Parameter Class.

6.3.5. vraEvent.Event—Vibrata Event Class.

6.4. Example Solver

6.4.1. Initialization.

6.4.2. Generate Modal Quantities.

6.4.3. Gather Event Setup for Generating Modal Quantities.

6.4.4. Calculate Modal Quantities.

6.4.5. Determine Output Requests.

6.4.6. Compute Output.

6.4.7. Cleanup and Return.

----

7. THEORETICAL MANUAL

7.1. Normal Modes Analysis

7.2. Viscous and Structural Damping

7.3. Steady-State Frequency Response Analysis

7.4. Random Response Analysis

7.5. Transient Analysis

7.5.1. Static and Dynamic Uncertainty Factors.

7.5.2. Nonlinear Forces

7.6. Enforced Motion Excitation

7.6.1. Seismic Mass Alternative.

7.7. Residual Vectors

7.8. Response Spectrum Analysis

7.8.1. Absolute Summation (ABS).

7.8.2. Square Root Sum Square Summation (SRSS).

7.8.3. Naval Research Lab summation (NRL).

7.8.4. Nuclear Regulatory Commission Rule (NRC).

----

8. INSTALLATION

Vibrata uses a client-server licensing scheme. You must therefore install and configure the Sentinel RMS license server (section 8.2 below) separately from Vibrata itself (section 8.3). If you have other ATA software, the license server is already installed and you will only need to obtain a specific license file for Vibrata to make it run.

8.1. Platform Requirements

Vibrata™ supports 64-bit Windows platforms; any platforms supported by both Femap and MATLAB will be supported by Vibrata.

Please see the Installation Guide for the supported and required versions of Femap, MATLAB, and MATLAB Runtime.

To run the default mode, which uses the compiled Runtime-based solver, Vibrata requires MATLAB Runtime. Please see the Installation Guide for the MATLAB Runtime version required.

To run custom solvers, Vibrata requires MATLAB.

Vibrata is compatible with all versions of Siemens Simcenter and NX Nastran and with MSC.Nastran v2001 or later.

8.2. Installing the License Server

Sentinel RMS is a robust, commercial client-server-based licensing system from SafeNet that can serve multiple licenses for multiple software products simultaneously. The server typically resides on a central computer while client software such as Vibrata resides on computers that will be utilizing the software. The client computer may or may not be the same as the license server. It can also be on a different software platform than the client.

Sentinel RMS installation is straightforward. Detailed installation instructions are shipped with the Sentinel RMS package available from ATA's website at http://www.ata-e.com/software/rmsserver/. This document highlights the basics of the installation process.

8.2.1. Installing Sentinel RMS.

On Windows, the Sentinel RMS package is an InstallShield application. As such, it must be installed by someone with administrator privileges. ATA recommends that the default selections be used during the installation.

8.2.2. Environment Variables.

All Sentinel RMS clients need a way to determine where the server is running. By default, the client will scan the subnet your client system is connected to and will identify any Sentinel RMS servers running on your subnet. It will then contact each until the license request is satisfied or it has run out of servers. You can control the order in which the servers are contacted through an environment variable called `LSHOST`. Set the environment variable to the hostname of the server or servers. Separate each license server name from the next with a tilde (~). The server name must be prepended by the name `'no-net'`. For example, set it to

```
no-net~server1~server2
```

where `server1` is the name of the first license server, `server2` is the name of the second license server, and so on.

If you only have one server and wish to bypass the network scan by the client, you can use the environment variable `LSFORCEHOST`. You can only specify a single server with this environment variable. For example, you can set it to

```
server
```

By default, the license server and client communicate on port 5093. If your Sentinel RMS server is running on a nondefault port, you can configure the client to communicate on that port by setting the LSPORT environment variable to the correct port number to use.

On Windows, the environment variable can be set in the Control Panel, typically in System. Please ask your system administrator or refer to your operating system documentation for more details on how to set environment variables.

8.2.3. Checking Your License Status.

You can use the RMS License Administration application `WlmAdmin`, included in the top-level Vibrata directory, to check the status of the licenses. When the GUI opens, open up Subnet Servers in the tree on the left. Under each license server, the individual licenses will be listed. Clicking on a license will display statistics about that license, including who is using it and how many are available. You can also install licenses using this application.

Prior to using Vibrata, you must contact ATA to obtain a valid license file for it. To obtain a permanent license, you will need to run the `echoid.cmd` batch file that comes with the Sentinel RMS download and send the resulting `echoid.txt` file to software@ata-e.com. If you have permissions issues running this file from the installation location, you can copy the `echoid.*` files to a directory where you have write access and run it from there.

8.3. Installing Vibrata

To install Vibrata, run `Vibrata_v*_Setup.exe`. Vibrata will install itself into the `C:\Apps\Vibrata_v###` (where ### is the Vibrata version number) directory by default.

Vibrata's Runtime-based solver is a MATLAB Runtime server that requires a specific version of MATLAB Runtime. The installer goes to great lengths to make sure you have the required MATLAB Runtime version installed. It first checks the registry for the MATLAB Runtime version. If it is not found, you will see a warning message similar to dialog [a] shown in

Figure 8-1. If you elect to proceed with the installation, the installer will not be able to register the Runtime-based solver, and you will need to modify `vibrata.bat` or the Vibrata launcher in the Start menu to force Vibrata to launch the MATLAB-based solvers when it starts (see section 3). Please consult your system administrator if you do not know how to do this.

If the installer cannot find a supported version of MATLAB but it did find MATLAB Runtime, no warning will appear. However, you will not be able to run custom solvers if a supported version of MATLAB is not present. If the installer cannot find MATLAB Runtime or a supported version of MATLAB, the warning in dialog [b] of Figure 8-1 will appear.

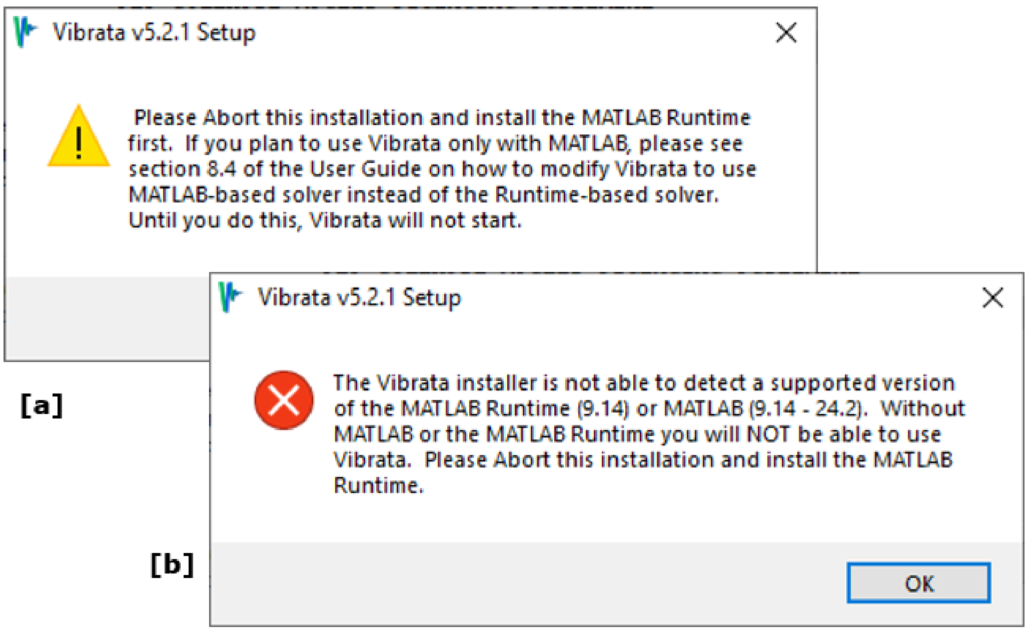


Figure 8-1. [a] The installer could not find the required MATLAB Runtime in the registry. [b] The installer could not find MATLAB Runtime or a supported MATLAB installation.

8.3.1. Installing in Silent Mode.

You can install Vibrata in silent mode, which means that the installer does not open a graphical installer, and it does not prompt you for anything. To install in silent mode, run the installer with the `/S` flag. Note that you must use a capital “S”.

If you would like to override the default installation directory, use the /D=<install_directory> flag, where <install_directory> is the directory where you wish to install Vibrata.

Please note that in silent mode, the installer will not inform you if it was unable to find a supported Femap version or if it was unable to register the Runtime solver. Care must be taken to make sure these prerequisites are satisfied before running the installer, or Vibrata may not run.

8.4. Configuring Vibrata

The default configuration that installs with Vibrata should be sufficient for most people, so you should not have to do any additional configuration. However, if you have a nonstandard environment or wish to fine-tune how Vibrata works, you may need to configure it. All possible changes will be made in the Vibrata launch script.

8.4.1. Vibrata Launch Script.

The Vibrata launch script (`vibrata.bat`) is located in the top-level directory of your Vibrata installation. It also supports several command-line arguments. To see what arguments it supports, type '`vibrata --help`' in a command prompt (Windows).

By default, Vibrata will attempt to use the Runtime-based solver. To force Vibrata to use the MATLAB-based solver, pass in the input argument '--`matlab`' (see section 3.1). If you wish to modify your launcher to launch the MATLAB-based solver by default, you will need to edit `vibrata.bat`. Look for the section of the script that sets the VIBRATA_SOLVER_TYPE environment variable, and follow the directions outlined in the comments. The Vibrata launch script performs several operations. First, it sets several environment variables that Vibrata needs. These are described in Table 8-1. Finally, it will launch the Event Manager, which will launch Femap and possibly MATLAB. The VIBRATA_ROOT variable in the launch script must NOT be modified unless you are sure you know what you are doing. Table 8-1 describes the environment variables that you may need to configure.

Table 8-1. Vibrata environment variables and their meanings.

Environment Variable	Description
VIBRATA_SOLVER_TYPE	Specify whether to use the MATLAB-based or Runtime-based solvers.
VIBRATA_ROOT	Defines the Vibrata installation location. Do not modify this variable unless you know what you are doing.
VIBRATA_LOGDIR	Directory where the log files are written. If you launch Vibrata from a command prompt, the log file will be written to the directory from where you launched Vibrata. Otherwise, the log file will be written to the `Vibrata_log` subdirectory of your home directory.
VIBRATA_CUSTOM_PATH	Contains a list of directories for Vibrata to search for custom solvers. The directories are separated by a semicolon (;). You can set this environment variable prior to calling the launch script.
VIBRATA_DEBUG_ON	Specifies whether to write debug messages to the solver log file. The default is 0 (no). Set to 1 to write debug messages when you are developing a custom solver.

8.5. Troubleshooting Vibrata Launch Issues

Since Vibrata connects three different components together via the Microsoft Component Object Model (COM), these components must be registered with the operating system. Although Vibrata is set up to register the components seamlessly when it launches, occasionally these steps may not complete successfully, causing Vibrata to fail, in which case a detailed error message will be displayed showing the suspected cause of failure and the suggested remedy. The remainder of this section describes these situations.

Vibrata registers the MATLAB Runtime-based solver when it launches in default mode. The registration can fail for several reasons. One reason is that the MATLAB Runtime is not installed. In this case, either install the required version of MATLAB Runtime or launch Vibrata so that it uses MATLAB instead of MATLAB Runtime (see section 8.4.1). Another reason the Runtime-based solver registration may fail is that MATLAB Runtime is installed but your system path does not have the runtime directory of MATLAB Runtime in its definition. This situation can occur if the length of the contents of the PATH variable is close to the character limit of this environment variable; the MATLAB Runtime installer thus cannot add the runtime directory to the path when it installs. Please consult with your system administrator if this appears to be the situation. You will need to remove something from your system path so that you can add the MATLAB Runtime directory.

To test the Runtime-based solver registration manually, open a command prompt with administrator privileges in the Vibrata installation directory and then enter the following command:

```
mwregsvr mcr_solver\Vibrata_MCR_solver.dll
```

Please consult your system administrator if you need help with any of the steps discussed in this section.

If Vibrata is unable to determine why the Runtime-based solver registration fails, you will see an error similar to the message shown in Figure 8-2. In this case, please contact Vibrata technical support at vibrata@ata-e.com for further assistance.

```
*** Pythoncom exception in SRASolverClientMCR.__init__.
Failed with code -2147221005: Invalid class string
while attempting connection to server "Vibrata_5_2_0.Solver"
```

Figure 8-2. An error similar to this will be shown if the MATLAB Runtime-based solver was not properly registered.

The second COM server that must be registered is the Vibrata GUI. This server is called VRA.EventServer. Vibrata attempts to register this COM server as well when it starts up. If the registration fails, Vibrata will display an error message with the suspected cause. To register the GUI manually, run `comRegisterSraEventServer.exe`, found in the *vra_gui* subdirectory of your Vibrata installation. It should run very briefly and does not produce any output.

Finally, the last component that may sometimes fail to launch is Femap. Each time Femap launches, if it has administrator privileges, it will register itself as a COM server. Therefore, to make sure that Femap is correctly configured as a COM server, launch it once with administrator privileges. This is also the way to change which version of Femap launches when you launch Vibrata. Alternatively, if you do not have administrator privileges or Femap otherwise cannot register itself as a COM server, you can open Femap first and then launch Vibrata.

Note that if Femap cannot find a license, it typically produces an error message to that effect. However, if Femap has launched, attempted to retrieve a license, and been unable to do so, it may leave an instance of Femap running in the background. Therefore, you should also double-check that you do not have any instances of Femap running in the background by opening the Windows Task Manager and looking for it in the list of running programs. If Femap is still running, it may be necessary to force-quit it by selecting it from the list and pushing the “End Task” button.

----

9. REFERENCES

[1] National Aeronautics and Space Administration. *Force Limited Vibration Testing*. Revision C. NASA Technical Handbook NASA-HDBK-7004C. November 2012.

[2] Wirsching, P.H., T.L. Paez, and K. Ortiz. *Random Vibrations: Theory and Practice*. Mineola, New York: Dover Publications, 1995.

[3] Segalman, D.J., et al. "An Efficient Method for Calculating RMS von Mises Stress in a Random Vibration Environment." *Journal of Sound and Vibration* 230, No. 2 (2000): 393–410.

[4] Chapman, M.J. "Incorporating a Full Damping Matrix in the Transient Analyses of Nonlinear Structures." Damping '93 Conference, February 1993.

Appendix A. **Additional Output2 Data for Specific Analyses**