



User Guide

Version 5.2

The content of this document is ATA proprietary and confidential information. This document is copyright © ATA Engineering, Inc., 2004–2025. All rights reserved. No part of this work may be reproduced or used in any form or by any means, graphic, electronic or mechanical (including photocopying, recording, taping, or information storage and retrieval systems) without the express written permission of ATA. All copies of this document must include the copyright notice as noted above and the other information contained in the paragraph.

TABLE OF CONTENTS

1. (Over	view				
-	1.1.	Featur	es1			
-	1.2.	Archite	ecture			
	1.3.	Files				
-	1.4.	Note o	n the "Global Coordinate System"			
2. 1	Prepa	arina Na	astran Input files			
	2.1.	Nastra	n Cards for Specific Vibrata Options			
-		2.1.1.	General Requirements for Physical Response Recovery			
		2.1.2.	Load Set Excitation			
		2.1.3.	Enforced Motion8			
		2.1.4.	Coupled Damping Matrix 10			
		2.1.5.	Residual Modes 11			
		2.1.6.	Modal Effective Mass Output			
		2.1.7.	Composite Failure Indices			
		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			
4	2.2.	Suppo	rted Nastran Versions and Solution Types			
3. \	Vibra	ita User	r Guide 25			
	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 lab			
		3.2.5.	Modeling Nonlinearities in the Transient Solver			
		3.2.0. 2.2.7	EXCILICIONS TOD			
		328	Defining Modal Damping			
		329	Output Request Tab 25			
		3.2.10.	Contour Output Requests			
		3.2.11.	Post Processing Tab			
	3.3.	Event	Manager—Batch Solution Mode			
	0.01	3.3.1.	Batch Processing with MATLAB Scripts			
	3.4.	Functio	on Manager			
	0111	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 25			

	3.5.	XY-Plo	t Results Display	. 25
	3.6.	Contou	ur 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.	Exam	ple Pro	blems	. 27
	4.1.	Descri	ptions 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	y-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.	Rando	m 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.	Respo	nse Spectrum Analysis	. 92
	4.5.	Transi	ent Analysis	106
		4.5.1.	Isat Model with RCS Thruster Firing	106
		4.5.2.	Frame Model Transient Animation.	122
5.	CREA	TING C	CUSTOM FUNCTIONALITIES	134
	5.1.	Creati	ng 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.	Creati	ng a Custom Nonlinear Model	134
	5.3	Creati	ng Custom Postprocessing	134
		2. 20.01		

6.	Vibra	ata-MATLAB API For Custom SOLVERS and Functions 13				
	6.1.	. Directory Structure				
	6.2.	Function Naming Convention				
	6.3.	Utility Classes				
		6.3.1. v	raParam—Vibrata Parameter Class	135		
		6.3.2. v	raReqmap—Vibrata Request Mapping Class	135		
		6.3.3. fo	cn—Vibrata Function Class.	135		
		raNonlinearParam—Vibrata Nonlinear Parameter Class.	135			
		6.3.5. vraEvent.Event—Vibrata Event Class				
	6.4.	Example Solver				
		6.4.1. II		135		
		6.4.2. G	Seherate Modal Quantities.	135		
		6.4.3. G	Sather Event Setup for Generating Modal Quantities.	135		
		0.4.4. C	alculate Moual Qualitities.	125		
		646 C	Compute Output	135		
		647 C	Pleanup and Return	135		
_				100		
7.	Theo	retical Ma	anual	136		
	/.1.	Normal I	Modes Analysis	136		
	7.2.	Viscous	and Structural Damping	136		
	7.3.	Steady-S	State Frequency Response Analysis	136		
	7.4.	Random	Response Analysis	136		
	7.5.	Transien	it Analysis	136		
		7.5.1. S	Static and Dynamic Uncertainty Factors	136		
		7.5.2. N	Ionlinear Forces	136		
	7.6.	Enforced	Motion Excitation	136		
		7.6.1. S	Seismic Mass Alternative	136		
	7.7.	Residual	Vectors	136		
	7.8.	Respons	e Spectrum Analysis	136		
		7.8.1. A	bsolute Summation (ABS).	136		
		7.8.2. S	Square Root Sum Square Summation (SRSS)	136		
		7.8.3. N	laval Research Lab summation (NRL).	136		
		7.8.4. N	iuclear Regulatory Commission Rule (NRC).	136		
8.	Insta	llation		137		
	8.1.	Platform	Requirements	137		
	8.2.	Installing	g the License Server	137		
		8.2.1. Ir	nstalling Sentinel RMS	137		
		8.2.2. E	nvironment Variables	137		
		8.2.3. C	Checking Your License Status.	138		
	8.3.	Installing	g Vibrata	138		
		8.3.1. Ir	nstalling in Silent Mode	139		
	8.4.	Configur	ing Vibrata	140		
		8.4.1. V	/ibrata Launch Script	140		
	8.5.	Troubles	shooting Vibrata Launch Issues	141		
9.	Refe	rences		143		

Appendix A. Additional Output2 Data for Specific Analyses 1	Data for Specific Analyses 1	44
---	------------------------------	----

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
excitation 9
Figure 2-3. Alternate deck for enforced motion using DMAP alter
Figure 2-4. Include PARAM, RSOPT, 1 to get coupled modal damping matrices
Figure 2-5. Add a RESVEC case control statement to get residual modes for applied
forces
Figure 2-6. You can get residual modes by defining a U6 USET, without any actual
loads
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
Figure 2-9. Access the Analysis Setup GOT through the Custom roots menu in Femap 10 Figure 2-10. Analysis Setup GUT
Figure 2-10. Analysis Secup Continuation utility from the Start menu 19
Figure 2-12. Create the .vra5 database from OP2 using the command-line interface
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
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
Figure 4-1. Single DOF spring-mass model
Figure 4-2. Two-DOF spring-mass model
Figure 4-3. Two-dimensional frame model
Figure 4-4. Isat model in launch configuration with four separate attachment points 30
Figure 4-5. Isat model in idunch configuration with single central conflection to
Figure 4-6 Isat model in deployed configuration 32
Figure 4-7. Read the bulk data into Femap using the <i>Import/Analysis Model</i> menu
picks
Figure 4-8. Define groups in Femap by importing a neutral file when available
Figure 4-9. Create a new frequency response event and select the FEM on which to
base it
Figure 4-10. Create a new enforced motion excitation
Figure 4-11. Use the Function Manager to create an acceleration function
Figure 4-12. Save excitation function to new function file
Figure 4-13. Assign 2.5% modal viscous damping to the mode directly in the modes
Educe A-14 Request XV output for total and flexible X-accelerations at node 32 30
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
Figure 4-18. Set the number of points at which you want to compute responses
Figure 4-19. With only two modes, you can assign damping directly in the damping
table
Figure 4-20. Define a point force excitation on node 13 in the X-direction
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
Figure 4-23. Plot the acceleration response of all DOF
Figure 4-24. Create a new model frequency event using the frame model
righter i 25. Apply a foreing function to the available enforced motion Dor

Figure 4-26. If necessary, reopen the file containing the excitation function from the
Figure 4-27. Change the copied function to have a range from 1 to 40 Hz
Figure 4-28. Edit the new function's name and make it use log-log interpolation
Figure 4-29. Make the function uneven and give it non-constant amplitude
Figure 4-30. <i>Excitations</i> tab with resulting enforced motion definition
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
Figure 4-33. Use the Copy button to create a new event from an existing one57
Figure 4-34. For the new (copied) event, delete the old excitation and create a new
one
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
Figure 4-37. Access Function Manager directly from Event Manager menu
Figure 4-38. Create the acceleration PSD that will be used to excite the Isat
Figure 4-39.Create and save the function, and finish by clicking Done
Figure 4-40. Create a new random analysis event
Figure 4-41. Select enforced motion, select Z-translation, and then bring up the
Function Manager
Figure 4-42. Select the modes above 400 Hz and set their status as residual vectors65
Figure 4-43. Assign damping via the schedule; export the schedule to a text file for
later use
Figure 4-44. Request SVMS for the BUS_SHELLS and TAT for the INST_PKG_FEM;
only <i>RMS</i> output67
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
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 557770
Figure 4-49. Node 5577 total Y-acceleration in g ² /Hz units, rigid base
Figure 4-50. Request axial forces in the launcher adapter legs
Figure 4-51. Axial forces in the launcher adapter legs, rigid base
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 nodes75
Figure 4-54. Delete the automatic CSD entries76
Figure 4-55. RMS von Mises stress in satellite bus shell elements, four-point base,
uncorrelated77
Figure 4-56. RMS acceleration contours for the instrumentation package, four-point
base, uncorrelated
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
Figure 4-59 Convite 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 PMS yon Mises stress in satellite bus shell elements, four-point-base
correlated
Figure 4-64 DMS accelerations, rigid-bace [a] four-point uncorrelated [b] four point
rigure 4-04. Kino accelerations. Tigurbase [a], Tour-point uncorrelated [b], Tour-point correlated [c]
Figure 4-65. The correlated four-point results match the rigid base results for pade
5577 Y-acceleration
5577 T acceleration

Figure 4-66.	The correlated four-point results also match the rigid-base results for launcher leg forces
Figure 4-67.	Copy the original rigid-base event; the new one will use FastRMS contour
Figure 1-68	Turn on the Use FastPMS togale on the Excitations tab
Figure 4-00.	The East DMC contaurs are indictinguishable from the original event
Figure 4-09.	Convisional rigid base events the new one will evolve medee with
Figure 4-70.	Copy the original rigid-base event; the new one will exclude modes with
	negligible effective mass
Figure 4-71.	Select modes below 0.1% modal effective mass for all translations
Figure 4-72.	All but 51 of the 1/4 modes are turned off, but 98% of the effective mass remains
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
2	values match
Figure 4-75.	The adapter-leg axial force again shows differences only at high
J	frequencies
Figure 4-76.	New response spectrum event with absolute value summation
Figure 4-77.	Assign an enforced motion function to direction 1
Figure 4-78	Define a Response Spectrum velocity function 95
Figure $4-70$	Store the new function in the frame excitations file
Figure 4-90	The excitation uses the basic coordinate system
Figure 4-80.	Demping is not needed and the damping dialog button is disabled
Figure 4-01.	Bequest translational displacements and accelerations at all nodes
Figure 4-62.	Converte a suggest and abange the suggestion method to CDCC
Figure 4-83.	Copy the event and change the summation method to SRSS
Figure 4-84.	Copy again to create a third event using NRL summation
Figure 4-85.	Copy again to create a rourth event using NRC summation
Figure 4-86.	Select Criteria and Deformed views from Femap's Post toolbar
Figure 4-87.	Displacements for [a] absolute value and [b] SRSS methods
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
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
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
5	second intervals
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
Figure 4-101	Request neak values of yon Mises stress in the RCS mounting nanels 115
Figure 4-102	2. Select the RCS Panels contour request and plot its results 116
Figure 4-102	Peak von Mises stress contours in RCS nanels
Figure $4-102$	1 Von Mises results with only the RCS namels aroun disnlaved
Figure $4-10^{\circ}$	5 Select the RUS SHELLS contour request and not its results 112
Figure A_{-100}	5. Von Mices results with only the BUS SHELLS aroun displayed 110
Figure 4 10	7. Poquest beam forces in elements that attach appendages to bus 120
Figure 4-107	R Solve the new Element XV requests and plot them
Figure 4-100	b. Solve the new Element Λ requests and plot them
rigure 4-109	7. Forces in 19 boom connector; note enects of thruster start/stop

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

Vibrata is a trademark of ATA Engineering, Inc. \odot 2004–2025 ATA Engineering, Inc.

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

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 <u>not</u> included with Vibrata; you must obtain them separately from Siemens Product Lifecycle Management Software (<u>http://siemens.com/plm/femap</u>) and The MathWorks, Inc. (<u>http://www.mathworks.com</u>) 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 (<u>https://www.ata-e.com/software/ata-software/matlab-runtime/</u>) or from the MathWorks website (<u>https://www.mathworks.com/products/compiler/matlabruntime.html</u>).

1.3. <u>Files</u>

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," we will *say* "the nodal displacement 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. <u>General Requirements for Physical Response Recovery.</u>

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 <u>must</u> 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 <u>as part of the modes solution</u> 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 <u>must</u> 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.

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 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.

```
$ 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
```



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
Ś*
INCLUDE c:\apps\Vibrata\vibrata.nx7
$ Additional/modified params for Vibrata enforced motion
PARAM, RSOPT, 1
PARAM, RSCON, YES
Ś
$ DOF available for enforced motion
                                        45
                                                                123
USET
              U2
                       44
                              123
                                               123
                                                         48
              U2
                              123
USET
                       49
```

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 <u>undamped</u> 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
$*
$*
$* CASE CONTROL
$*
LOADSET = 101
METHOD = 10
DISPLACEMENT (PLOT, REAL) = ALL
RESVEC = YES
$ Additional params for Vibrata load set excitation
PARAM, RSOPT, 1
PARAM, OGEOM, YES
       -- 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
$
s 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.

```
$ Additional/modified params for Vibrata enforced motion
PARAM, RSOPT, 1
PARAM, RSCON, YES
PARAM, RESVINER, YES
Ś
$ DOF available for enforced motion
USET
              U2
                       44
                              123
                                        45
                                                123
                                                         48
                                                                123
USET
              U2
                       49
                              123
```

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.

Vibrata Documentation: Preparing Nastran Input files

🕨 Analysis Setup	\times						
Analysis Title							
Include Enforced Motion	[
Include Residual Vectors							
Include Coupled Damping							
Create Analysis Set Cancel							

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 \rightarrow 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.

Vibrata Documentation: Preparing Nastran Input files



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.

🕐 Select OP2 File To Convert Into .vra5 Input Database								×	
← → < ↑ 📑 > This PC > Scratch (S:) > projects > Vibrata > 1DOF 🗸 🖏 Search 1DOF							P		
Organize 👻 New folder									•
📃 Desktop	* ^	Name	Date modified	Туре	Size				
🖶 Downloads	*	1dof_modes_base.op2	6/27/2011 3:54 PM	OP2 File	18 KB				
Documents	*								
Pictures	*								
Chapter1&2	*								
👝 Scratch (S:)									
, sjaeger									
💻 This PC									
🧊 3D Objects									
📃 Desktop									
🔮 Documents									
🖊 Downloads									
👌 Music									
Pictures									
📑 Videos									
🏪 Local Disk (C:)									
🛖 sjaeger (\\rmo-fs1\u) (H	:)								
Scratch (S:)									
🛖 u (\\rmo-fs1) (U:)									
🔿 Network									
-									
	~								
File name	: 1dof_r	modes_base.op2			~	(*.op2)			\sim
						Open	(Cancel	
									:

Vibrata Documentation: Preparing Nastran Input files

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 <u>always</u> 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.

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

Element Type	Relevant Notes	
CBAR	Forces and moments; stress and strain at CL and stress recovery points C, D, E, F. Please see the <i>Nastran Quick Reference Guide</i> 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).	

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

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 (<u>https://tier.ata-e.com</u>). 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. <u>General Characteristics of the User Interface.</u>
- 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. <u>Hints for Creating Complex Functions.</u>
- 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. <u>Viewing Frequency Response Results.</u>
- 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*\2*dof* 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. <u>Two-Dimensional Frame.</u>

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. Isat—Inner Planets Exploration Satellite.

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.

Vibrata Documentation: Example Problems



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.
 - Import the 1DOF model "1dof_modes_base.dat" (Figure 4-1). Save the file as "1dof.modfem."
 - 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.

File Functions							
Events						1	
Name 1dof_FreqRst	XY Pre FreqRs 2	r fix Type p Modal Frequency	Event Definition File	^	New Copy Clear	$\overline{\mathbf{I}}$	>
<	-			> ~	Open Save	ЛТ	~
Event Details						•	
FEM Units:	Inch (Pound f)	~				3	
FEM File:	S:\Vibrata\1dof\1	dof.modfem				Sele	ect FEN
Input Database	e: S:\Vibrata\1dof\1	dof_modes_base.vra5	; ;			Sel	ect DB
							/
Solver Ex	citations Modal	Settings Output	Post Processing	5		/	4
Solver / Anal	ysis Type: Modal Fi	requency		~			
Data (as fa	· · · · · · · · · · · · · · · · · · ·						
Data for Fre	equency/kandom A				C . C .		
	equency/ tandoin A	nalysis		Variables f	or Custom Solv	ers	
Excitation fr	requency selection:	Log spacing at N	lodes V	Variables f	or Custom Solv	Value	^
Excitation fr Excitation lo	requency selection:	Log spacing at N	Nodes v Io. points in range: 20	Variables f	or Custom Solv	Value	^
Excitation fr Excitation lo Excitation u	requency selection: ower bound (Hz): pper bound (Hz):	Log spacing at M -1.0 N -1.0 N	Nodes V Io. points in range: 20 Io. points near modes: 5	Variables f	ne	Value	^
Excitation fr Excitation lo Excitation u	requency selection: ower bound (Hz): pper bound (Hz):	Log spacing at N -1.0 N -1.0 N	Nodes V Io. points in range: 20 Io. point near modes: 5	Variables f	ne	value	^
Excitation fr Excitation lo Excitation u	requency selection: ower bound (Hz): pper bound (Hz):	Log spacing at N -1.0 N -1.0 N -1.0 N -1.0 S Vibrata > 100	Addes v lo. points in range: 20 lo. points near modes: 5	variables f	ne	Value	^
Excitation fr Excitation lo Excitation u EXCitation u FEM Model File This PC > New folder	requency selection: ower bound (Hz): pper bound (Hz): sjæger (\\rmo-fs1\u) (Hz)	Log spacing at M -1.0 N -1.0 N -1.0 N Vibrata > 1975	Nodes V Io. points in range: 20 Io. points near modes: 5 V V O Sear	ch 1DOF	ne	Value	
Excitation fr Excitation lo Excitation u FEM Model File This PC > New folder France	requency selection: ower bound (Hz): pper bound (Hz): sjæger (\\rmo-fs1\u) (H;)	Log spacing at M -1.0 N -1.0 N vibrata > 100F	Nodes V Io. points in range: 20 Io. point near modes: 5 V V Sear modified Type	ch 1DOF	ne	Value	^
Excitation fr Excitation lo Excitation u EXCITATION U EXCITATION U EXCITATION Random AT	requency selection: ower bound (Hz): pper bound (Hz): sjæger (\\rmo-fs1\u) (H)	Log spacing at N -1.0 N -1.0 N Vibrata > 1977 Date 7/9/2	Nodes lo. points in range: 20 lo. points in range: 5 lo. points in range: 5 lo. points in range: 20 lo. points in range: 5 lo. points	ch 1DOF	ne Solve	Value	
Excitation fr Excitation lo Excitation u EXM Model File TM Model File TM Model File TM Model File TM Model File TM Model File TM File This PC > New folder Random TP PC Objects	requency selection: ower bound (Hz): pper bound (Hz): sjæger (\rmo-fs1\u) (Hz) Name	Log spacing at N -1.0 N -1.0 Vibrata > 1007 Vibrata	Nodes V Io. points in rance: 20 Io. points mear modes: 5 V V O Sear modified Type 2020 1:50 PM Femap Model File RAS Database File	Ch IDOF	ne	Value	
Excitation fr Excitation lo Excitation u Excitation u Excitation u TeM Model File This PC New folder Random AT PC Objects sktop	requency selection: ower bound (Hz): pper bound (Hz): sjæger (\\rmo-fs1\u) (Hz) Name	Log spacing at N -1.0 N -1.0 N Vibrata → 190F Date 7/9/7 V Select OP2 Results File or V ← → ← ↑ → This:	Nodes lo. points in range: 20 lo. points file rodes: 5 w 0 Sear modified Type 2020 1:50 PM Femap Model File RAS Database File PC > sjæger (\rmo-fs1\u) (H; > Vibrata >	ch 1DOF	v C	Value Value Search 100F	
Excitation fr Excitation lo Excitation u Excitation u Exc	requency selection: ower bound (Hz): pper bound (Hz): sjæger (\rmo-fs1\u) (Hz) Name	Log spacing at N -1.0 N -1.0 N -1.0 N • Vibrata > 100F Vibrata > 100F Vibrata > 100F Vibrata > 100F Vibrata > 100F	Aodes lo, points in range: 20 lo, point near modes: 5 modified Type 2020 1:50 PM Femap Model File RA5 Database File PC + sjøeger (\\rmo-fs1\\u) (H;) + Vibrata +	ch 1DOF	re Custom Solv	Value Value Search 1DOF	
Excitation fr Excitation lo Excitation u Excitation u FEM Model File This PC > New folder Random W PC Objects sktop usuments winnloads usic	requency selection: ower bound (Hz): pper bound (Hz): sjæger (\\rmo-fs1\u) (H)	Log spacing at N -1.0 N -1.0 N -1.0 N • Vibrata → 18-F Date 7/9/2 V Select OP2 Results File or V ← → ~ ↑ ▲ → This Organize ▼ New folder ↓ 1DOF ↓ 4.1 Descriptions of Exa	Aodes lo. points in rance: 20 lo. points in rance: 20 lo. point mear modes: 5 v 0 Sear modified Type 2020 1:50 PM Femap Model File RAS Database File PC > sjæger (\\rmo-fs1\u) (H;) > Vibrata > mple 1 dof_modes_base.op2	Variables f Nar Nar k	Type OP2 File	Search 1DOF	
Excitation fr Excitation lo Excitation u EXModel File This PC > New folder Random V PC Objects sktop womoads usic tures	requency selection: ower bound (Hz): pper bound (Hz): sjæger (\\rmo-fs1\u) (H)	Log spacing at N -1.0 N -1.0 N -1.0 N • Vibrata > 190F • Vibrata > 190F • Select OP2 Results File or V ← → ← ↑ → This Organize → New folder 100F 4.1 Descriptions of Exa 4.3 Random 6.4 T	Nodes 20 Io. points in range: 20 Io. points in range: 5 Io. points in range: 6 Name 6 Inder_modes_fixed.op2 1dof_modes_fixed.op2	× ch 1DOF 191 KB Date modified 6/27/2011 3:54 PM 6/27/2011 3:54 PM	or Custom Solv ne	Value Value Search 1D0F Size 18 KB 10 KB	
Excitation fr Excitation lo Excitation u Excitation u EXM Model File The Mode	requency selection: ower bound (Hz): pper bound (Hz): sjæger (\\rmo-fs1\u) (H)	Log spacing at N -1.0 N -1.0 N -1.0 N · Vibrata > 10 F · Vi	Nodes 20 Io. points in range: 20 Io. points in range: 5 Interview 5 Interview 1	Variables f Nar Nar k 1DOF 191 KB 100F 191 KB 100F 191 KB 100F 100F 100F 100F 100F 100F 100F 100	or Custom Solv ne V V V V V V V V V V V V V	Search 1DOF	
Excitation fr Excitation lo Excitation lo Excitation u Excitation u TeM Model File T New folder New	requency selection: ower bound (Hz): pper bound (Hz): sjæger (\rmo-fs1\u) (Hz) Name	Log spacing at N -1.0 N -1.0 N -1.0 N -1.0 N · Vibrata > 10-F · Vibrata > 10	Aodes 20 Io. points in range: 20 Io. points in range: 5 Intervention: 5	Ariables f Nar Nar Nar Size 191 KB Date modified 6/27/2011 3:54 PM 6/27/2011 3:54 PM	or Custom Solv ne	Search 1DOF	
Excitation fr Excitation lo Excitation u Excitation u EXModel File The Model	requency selection: ower bound (Hz): pper bound (Hz): sjæger (\\rmo-fs1\u) (Hz): Name Name 1dof.modfem	Iog spacing at N -1.0 N Other N -1.0 N -1	Aodes 20 Io. points in range: 20 Io. points in range: 5 Interview 5 Interview 5 Interview 1	Ariables f Nar Nar Nar Nar Size 191 KB 100F Date modified 6/27/2011 3:54 PM 6/27/2011 3:54 PM	or Custom Solv ne V N V V V V V V V V V V V V V	Value Value	
Excitation fr Excitation lo Excitation u Excitation u Exc	requency selection: ower bound (Hz): pper bound (Hz): sjæger (\\rmo-fs1\u) (Hz) Name Name 1dof.modfem	Iog spacing at N -1.0 N -1.	Aodes lo, points in range: 20 lo, points in range: 20 lo, points in range: 5 lo, points in range:	Ariables f Nar Nar Nar Size 191 KB Date modified 6/27/2011 3:54 PM 6/27/2011 3:54 PM	Type OP2 File	Value Value	
Excitation fr Excitation lo Excitation u Excitation u Exc	requency selection: pper bound (Hz): pper bound (Hz): sjæger (\\rmo-fs1\u) (Hz) Name Idof.modfem	Log spacing at N -1.0 N -1.0 N -1.0 N -1.0 N Vibrata → 100F Vibrata → 100F V	Modes	Ariables f Nar Nar Size 191 KB Dof Date modified 6/27/2011 3:54 PM 6/27/2011 3:54 PM	or Custom Solv ne V OP2 File OP2 File	Value Value	
Excitation fr Excitation lo Excitation u Excitation u Exc	requency selection: power bound (Hz): pper bound (Hz): sjæger (\\rmo-fs1\u) (Hz) Name Idof.modfem def.modfem	Log spacing at N -1.0 N -1.0 N -1.0 N -1.0 N Vibrata → 100F Vibrata → 100F V	Aodes	Ariables f Nar Nar Size 191 KB Date modified 6/27/2011 3:54 PM 6/27/2011 3:54 PM	or Custom Solv ne V Ne V	Search 1DOF	
Excitation fr Excitation lo Excitation u Excitation u Excitation u Excitation u Excitation u Excitation u The PC Objects Sktop PC Objects Sktop Incuments Incuments Incu	requency selection: power bound (Hz): pper bound (Hz): sjæger (\\rmo-fs1\u) (Hz) Name Idof.modfem dof.modfem	Log spacing at N -1.0 N -1.0 N -1.0 N -1.0 N Vibrata → 100F Vibrata → 100F Vibrata → 100F Vibrata → 100F -4.1 Descriptions of Exa -4.3 Rendom ISAT This PC 30 Objects Desktop Documents Documents Documents Video	Aodes lo, points in range: 20 lo, point near modes: 5 modified Type 2020 1:50 PM Femap Model File RA5 Database File PC > sjæger (\rmo-fs1\u) (H;) > Vibrata > mple 140f_modes_base.op2 140f_modes_fixed.op2	Arriables f	or Custom Solv ne v b v v b	Search 1DOF	
Excitation fr Excitation lo Excitation u Excitation u Excitation u Excitation u Excitation u Excitation u The PC Objects Skitop PC Objects Skitop Scuments Wenloads Susic Stures Sees cal Disk (C:) eger (\rmo-fs1\u) (H) File name: 1	requency selection: pwer bound (Hz): pper bound (Hz): sjæger (\\rmo-fs1\u) (Hz) Name Idof.modfem def.modfem	Log spacing at N -1.0 N -	Aodes Io. points in range: Io. points in range: Io. point mear modes: Io. point mear modes: Io. point mear modes: Io. point mear mo	Variables f Nar Nar (Nar (Nar (Nar (Nar)) (Nar (Nar)) (Nar) (Na	or Custom Solv ne	Value Value Search 1DOF Size 18 KB 10 KB	

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

Solver	Excita	tions M	odal Settings	Output I	Post Proc	essing				
	Туре	Item	Label	CSys	Dir	Scale Factor	Function Name		Fun ^	Point Force
									2	Load Set
👫 Enfor	ced Moti	on Excitat	tions				?	\times		Enforced Motion
Assign Fo	rcing Fun	ctions to V	alid DOF							CSD
Treat e	excitation	as rigid-bo	ody							Show on FEM
Na	da	Dir	Scala	Tuno	Eunct	ion Nomo	~	4		Use FastRMS
ONI O	ae		Scale	туре	runct	ion Name		f (x)		
3										Delete
3								7		
										
							~			
<							>		~	
									>	
		ОК		Apply		C	lose			
		ОК		Apply		C	lose		>	

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.

承 Function Manager		-		\times
Functions	Plot			
Source Files	Manage vraModalFrequencyInfo V XY Stacked		XYZ	
	+ New			
_	Attributes Math			
	New Function –	-		×
	Function Type: Frequency			
	X Spacing Start 1 X		γ	
	Even End 21		10 10	
	O Uneven Increment 20	4		
All	3 # Points 2			
Function Sets	11 10 9 0 5 10 15 20 25			
	Frequency Function Attributes			
	Name: Const_10G_20Hz	5 /	Attribut	es
Done	Interpolation Type: LinLin ~			
	Y Axis Type: Acceleration (G) v 6			
	7 Create Cancel			

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).

Select a file to append to							×
\leftrightarrow \rightarrow \land \uparrow \square \rightarrow This	s PC > sjaeger (\\rmo-fs1\u) (H:) > Vibrata	> 1DOF		v Ö ≤	Search 1DOF		٩
Organize 🔻 New folde	r					•== •	?
This PC	Name	Date modified	Туре	Size			
3D Objects Desktop	承 New Function		-		×		
DocumentsDownloads	Choose a destination file:						
Music Pictures					~		
Videos Local Disk (C:) r sjaeger (\\rmo-f	OK Cance	1			1		
■ Scratch (S:) ▼ u (\\rmo-fs1) (U)	2						
File name: 1dof_e	examples_functions.fcn						\sim
Save as type: FCN Fi	iles (*.fcn)						~
∧ Hide Folders				3	Save	Cancel	

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.
- 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).

Solver	Excitations	Modal Settings	Output	Post Processing		
	Frequency	Modal Dar	mping (%)	Translationa	l Modal Effec	
Mode #	(Hz)	Viscous	Structural	x	Y ^	
1	10.000	00 2.5	0.00	100.00		
			Totals ->	100.00		Select Modes
						Effective-mass threshold (%):
						Effective-mass directions: X, Y, Z
						All Active Rigid < Mass
						None Inactive Residual >= Mas
						Set Mode Status
						Active Rigid Residual
						Mode Shape Summaries
					×.	

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.

Solv	er Excitations Modal Settings Output Post Processing	2
Item	Label Output Recovery Recovery Contour Output Output Location	A Node XY
	Variables Point Location Intervals CSys	Elem XY
r		CSD
	🕐 Nodal XY-Plot Output Requests 🛛 🕹 🗙	Contour
	Output Variables	
		× Requests
	TAT. 2	Solve
		□ Interactive
	> TAR. Total Rotational Accelerations	
	> U, Displacements (Flexible Only)	
	> V, Velocities (Flexible Only)	
	 A, Accelerations (Flexible Only) 	DI-+ VV
	✓ ■ AT, Translational Accelerations	Plot AT
<	ATx 4	Plot Contour Plot Contour
	Use nodal displacement coordinate system	
	Node Selection	
	Nadas Al Nada Casura	
	Nodes Node Groups	
	32	
	Entity Selection - Select Nodes for Output Requests	? X
	Select All	Reset 7 Pick ^
	6 ID 32 to by 1 Previous	Delete OK
	Group V More	Method ^ Cancel
	5	
	Select Select	
	8	
	OK Apply Close	
L		

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.

Solve	r E	xcitations	Modal Se	ettings	Output	Post Pro	essing			
Item	Label	Output	Recovery	Recovery	Contour	Output	Output	Location	^	Node XY
		Variables	Point	Location	Intervals	CSys				Elem XY
										CSD
										Contour
										× Requests
										Solve
										Interactive
										Plot XY
									~	Plot Contour
<									>	➤ Results

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. 2DOF Frequency Response.

- 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."
- 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.

Vibrata: Advanced Modal Dynamic Analysis 5.0.0		– 🗆 X
File Functions Post Processing Help		
Events		1
Name XY Prefix Type Event Definition File	^ New	-
2dof_example FreqRsp Modal Frequency	Сору	
2	Clear	•
		ΛΤΛ
	Open	
<	> Save	
Event Details		
3 FEM Units: Inch (Pound f) V		
FEM File: S:\Vibrata\2dof\2dof.modfem		Select FEM
Input Database: S:\Vibrata\2dof\2dof_modes_base.vra5		Select DB
Solver Excitations Modal Settings Output Post Processing		
Solver / Analysis Type: Modal Frequency		
bolici / mayso ijper modul requerty		
Data for Frequency/Random Analysis	Variables for Custom Solve	rs
Excitation frequency selection: Log spacing at Modes 5	Name	Value
Excitation lower bound (Hz): -1.0 No. points in range: 100		
Excitation upper bound (Hz): -1.0 No. points near modes: 10		
Modal Frequencies: 🗹 Undamped 🗌 Damped 🗌 Max Response		
	<	× >
	<	>

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).

Solver	Excitations	Modal Settings	Output	Post Processing			
	Frequency	Modal Dan	nping (%)	Translationa	l Modal Effe	ect 낙	
Mode #	(Hz)	Viscous	Structural	x	Y	^	
1	19.824	8 1.5	0.00	75.87			
2	20.860	6 3.5	0.00	24.13		Select Modes	
			2 Totals ->	100.00	(Effective-mass threshold (%): 1.0	
						None Inactive Residual >= I Set Mode Status Active Rigid Residual Mode Shape Summaries Image: Control of the state	Mass

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 f(x) button to bring up the Function Manager (Figure 4-20).

Type	Item	Label	CSvs I	Dir Scale Factor	Function Name		A Point For
type		Cuber	CJys 1	Scale Factor	Tunction Hume		Load Se
							Enforced M
Point	Force Excita	tions			? ×		CSD
Forcing F	unctions						Show on Fi
Dir	Scale	Туре	Function	Name			Use FastRM
X							Delete
Y			land and a second s		6 ^{f(x)}		Derete
z							
Rx							
Rv					<u></u>		
Rz							
<					>		
Use n	odal displacem ection	ient coordin	ate system			>	
Use n	odal displacem ection Nodes	nent coordin	ate system	Node Groups	~	>	
Use n	odal displacem ection Nodes 13	nent coordin	ate system	Node Groups	^	>	
Use n Node Sel	odal displacem ection Nodes 13 ity Selection - Se	elect Nodes fo	or Excitation	Node Groups		>	? ×
Use n Node Sel	odal displacem ection 13 ity Selection - Se 0 Add O Re	elect Nodes for move	Attended to the system Attended to the system OExcitation OExclude	Node Groups	Select All	, e	? ×
Vode Sel	odal displacem ection Nodes 13 ity Selection - Se 0) Add ORe	elect Nodes fo move o	Ate system A	Node Groups	Select All) Q Q	? × Pick^ 5
Use ni Node Sel	odal displacem ection Nodes 13 ity Selection - Se 0 Add ORe 12 t	elect Nodes fo move o	or Excitation O Exclude by 1 v	Node Groups	Select All @ Previous More	> C C C C	? × Pick ^ Cancel
Use ni Node Sel	odal displacem ection Nodes 13 ity Selection - Se 0.Add ORe 13 tu	elect Nodes fo	or Excitation	Node Groups	Select All	> @ @ @ @	? × Pick ^ OK 5 Cancel
Use ni Node Sel	odal displacem ection Nodes 13 ity Selection - Se O Add O Re M T t	elect Nodes for move o	or Excitation O Exclude by 1	Node Groups	Select All (# Previous More	> C C C	? × Pick ^ OK 5
Use ni Node Sel	odal displacem ection Nodes 13 ity Selection - Se O Add O Re 10 10 10 10 10 10	elect Nodes fo	or Excitation O Exclude by 1 V	Node Groups	Select All (# Previous More	> Constant of the second secon	? × Pick^ Cancel
Use ni Node Sel	odal displacem ection Nodes 13 ity Selection - Se OAdd ORe Mode The transformation The tr	elect Nodes fo	or Excitation O Exclude by 1 V	Node Groups	Select All @ Previous More	> Contraction of the second se	? × Pick^ OK 5
Use ni Node Sel	odal displacem ection Nodes 13 ity Selection - Se)Add ORe 14 15 10 10 10 10 10 10 10 10 10 10 10 10 10	elect Nodes for move	recitation CExclude by 1 v v c	Node Groups	Select All () Previous More	> C C C	? × Pick ^ OK 5
Use ni Node Sel	odal displacem ection Nodes 13 ity Selection - Se OAd ORe Mod Re To tu	elect Nodes for move o	ate system	Node Groups	Select All (# Previous More	> C C C	? × Pick^ OK 5

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).

Function Manager			S 1 -	-	- 🗆 X
Source Files	Manage vraModalFrequencyInt	• V 1 + New Attributes	Math	Stacked	XYZ
New Function	2 Frequency V		_	X	Save
X Spacing Even Uneven	3 Start End Increment # Points	0 30 30 2	x 0 0 4	Y 10 10	
Funct 11	10 15 20 25	30			
Frequency Func	tion Attributes Const_10g_30Hz		5	Attributes	Help
Provide the second seco	pe: LinLin	- 6 (m) Cha	New Function Hose a destination file: Vibrata\2DOF\2dof_e	xamples functio	
7 Create Can	cel	8	OK Cancel		



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.

Solve	er E	xcitations	Modal Se	ettings	Output	Post Process	sing			2
ltem	Label	Output	Recovery	Recovery	Contour	Output Ou	utput Location		^	Node XY
		Variables	Point	Location	Intervals	CSys			,	Elem XY
		🕨 No	dal XY-Plo	ot Outpu	t Requests	s		×		CSD
		Outpu	ut Variable	25						Contour
			U. Displa	acement	(Flexible	Only)		^		× Requests
			V, Veloc	ities (Flex	ible Only)				
			A, Accel	erations	(Flexible (Dnly)			8	Solve
		~	· 🔳 AT, 1	Translatio			Interactive			
			\checkmark	ATx 3	3					
				АТу						
				ATz						
				Kotationa	al Accelera	ations	ible)			
				G Total T	ranslation	Magnitud	ibie)	_		
		<		o, iotai ii	ansideon	rmagnitaa	C.	>		Plot XY
			e nodal di	splaceme	ent coordi	inate system	m			Plot Contour
<			- nodur un	spincerine		note system			>	➤ Results
		Node	Selection							
			No	odes	^		Node Group	5 ^		
				11	_					
				12	Entity Sele	ection - Select N	lodes for Output Reque	ests 5	_	? ×
				13	Add	Remove	O Exclude	+ 11 + 12	Select All 🌐 🤄	🔍 🕵 😣 Pick ^
				or			by 1	+ 13	Previous De	elete OK
				G	oup		~		More Met	thod ^ Cancel
					~	·		× 1		
		<			>	<		>		
			4 s	elect			Select			
		7	OK		1	Apply	Close	2		

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. Frequency Response of a Frame.

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."
- 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.

₩ Fi	Vibrata: Advan le Functions	nced Modal Dynamic Analysis 5.0.0 Post Processing Help	-		×
L.	Events				
ſ	Name frame_FreqRsp	XY Prefix Type Event Definition File New Const FreqBsp Modal Frequency Conv			
	name_rreqrop_	1	2		;
	<	> Open Save			•
	Event Details			2	
	FEM Units:	Inch (Pound f) v			
	FEM File:	S:\Vibrata\Frame\frame.modfem		Select FEN	N
	Input Database:	S:\Vibrata\Frame\frame01_modes.vra5		Select DB	3
2	Solver Exci	itations Modal Settings Output Post Processing			
3	Solver / Analys	sis Type: Modal Frequency ~			
	Data for Free	quency/Random Analysis Variables for Custom Solver	s		
	Excitation free	quency selection: Log spacing at Modes V Name	Val	ue ^	
	Excitation low	ver bound (Hz): -1.0 No. points in range: 20			
	Excitation upp	per bound (Hz): -1.0 No. points near modes: 5			
	<u>k</u>	Modal Frequencies: 🗹 Undamped 🗌 Damped 🗌 Max Response			
				~	
		<		>	

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

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

Type Item	Label	CSys	Dir	Scale Factor	Function Name		Function File	\sim	Point Force
									Load Set
Forced M	ntion Excitat	tions				?	×	1	Enforced Mot
Linorecum						•	~		CSD
Assign Forcing F	unctions to V	alid DOF						[Show on FE
Treat excitati	on as rigid-bo	ody						r	Lice FactPM
Node	Dir	Scale	Туре	Function	n Name	/	3		
2	1		-77-2				f (x)		Delete
<u> </u>						_			
2							6		
							- <u>21</u>		
						~	/	\sim	
<						>		>	
J									

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.

Vibrata Documentation:	Example Problem	ns
------------------------	-----------------	----

Functions Source Files	Manage vraModalFr	quencyInfo 🗸	Plot XY Stacked	XYZ	
# Name 1 Const_10G_20 2 Const_10G_20	FunctionType OrdNi Hz Frequency Response Function Acceler Hz Frequency Response Function Acceler	mDataType + New ation Attributes 1 Edit	Math		
Edit Function Att	ributes			_	
Attribute Name	Value	Function List	40Hz	<u>^</u>	All
Iser\/alue4	3				All
SamplingType	o Dynamic				
VeightingType	None				None
VindowType	None				
molitudel loite	llakaowa	_		~	
Iormalization	linknown				
OctaveEormat	0	Edit Attributes			
	None	Name	3 Const 10G 1to40Hz		
vnDamningEact	0				
PulsesPerRev	0	IDI ine1			
AeasurementRun	0				
oadCase	0	IDI ine2			
RIGTime	Π				
lame	Const 10G 1to40Hz	IDI ine3			
JID	4				
Parent	H:\Vibrata\Frame\frame_excitations.fcnl3	IDI ine4			
Children		IDE NOT			
ilename	H:\Vibrata\Frame\frame excitations.fcn				
RSpectQ	0	Apply			
nterpolationType	LogLog	V VPPIJ			

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.

Function Manager				-	
Functions Source Files # Name Fun	Manage vraModalFrequencyInfo ctionType OrdNumDataTyp	 New 	Plot XY 1	Stacked	XYZ
1 Const_10G_20Hz Frequency Re 2 Const_10G_1to40Hz Frequency Re	sponse Function Acceleration sponse Function Acceleration	Attributes Edit	Math		
承 New Function				- 0	×
Function Type: Fre	quency ~	2			
X Spacing	Start	0	Х	Y	
⊖ Even	End	0.5	1	1	
Uneven 3	Increment	0.1	10	2	
	Increment	0.1	25	3	
	# Points	6	30	3	
FL	4-		40	1	
	101	10 ²	5		
Frequency Function	Attributes				
Name:	Var_1to3g_1to40Hz			Attribut	es
Interpolation Type:	LogLog	/			
Y Axis Type:	Acceleration (G)	6			
7		Copy Fund Choose a desti	ctions ination file: rame\Var_1to3g_1to40	— 🗆	· · · ·
Create Cancel	9	ОК	Cancel		

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.

Solve	er Exc	itations	Modal Settings	Output	Post Processing				
	Туре	Item	Label	CSys	Dir Scale	e Factor	Function Name	FL ^	Point Force
1	Accel	Motion	2	disp	1	1.0	Var_1to3g_1to40Hz	S:\	Load Set
									Enforced Motion
									CSD
									Show on FEM
									Use FastRMS
									Delete
									Delete
								~	
<								>	

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 \ddagger 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 A. 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.

Frequency Modal Damping (%) Translational Modal Effective X P 2 3 5.4281 2.00 0.00 2.03 0.00 5 5 5 5 5 5 5 5 9 5 2.00 0.00 0.00 5 5 11.1395 2.00 0.00 0.00 5 5 10 5 2.00 0.00 0.00 5 5 10 5 10 3 5 5 10 3 5 10 3 5 10 <th>Solver</th> <th>Excitations</th> <th>Mod</th> <th>lal Settings</th> <th>Output</th> <th>Post Process</th> <th>sing</th> <th></th> <th></th> <th>-</th> <th></th> <th></th> <th></th> <th></th> <th></th>	Solver	Excitations	Mod	lal Settings	Output	Post Process	sing			-					
Mode # (Hz) Viscous Structural X Y 3 5.4281 2.00 0.00 2.03 0.00 5 111.395 2.00 0.00 0.00 500 6 15.4939 2.00 0.00 0.00 500 7 15 Model Damping Definition X X X Z 9 35 6 10 35 11 42 4 7 15 Model Damping Definition X X Z III 42 III III 42 III 42 III 42 III III 42 III III 42 IIII 43 IIII 44 IIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIII		Frequer	ncy	Modal Dam	ping (%)	Translat	ional Moda	l Effective	•]2					
3 5.4281 2.00 0.00 2.03 0.00 4 7.4740 2.00 0.00 0.40 0.01 5 11.1395 2.00 0.00 0.00 505 Effective-mass threshold (%): 1.0 6 15.49330 2.00 0.00 0.00 505 Effective-mass threshold (%): 1.0 7 15 Modal Damping Definition X,Y,Z Image: Construction of the constructio	Mode #	(Hz)	1	Viscous	Structural	х		Y ^							
4 7.4740 2.00 0.00 0.40 0.00 5 11.1395 2.00 0.00 0.00 5025 Effective-mass threshold (%): 1.0 6 154.4393 2.00 0.00 0.00 5025 0.00 Effective-mass threshold (%): 1.0 7 15 Moda Damping Definition Image: Statutal Status Image: Status Imag	3	5.	4281	2.00	0.00	2	.03	0.00	/						
5 11.1395 2.00 0.00 0.00 5535 Effective-mass threshold (%): 1.0 6 15.4939 2.00 0.00 0.00 0.00 Effective-mass directions: X, Y, Z 7 15 Modal Damping Definition X X, Y, Z Image: Constraint of the constraint	4	7.	4740	2.00	0.00	0	.40	0.00	Selec	t Modes					
6 15.4939 2.00 0.00 0.00 0.00 Effective-mass directions: X, Y, Z 7 15 8 31 9 35 10 35 11 42 4 10 0500 Structural Modal Viscous Damping Summary User + Matrix Diagonal Off-Diagonal Ratio 0% Mean Max 1 >= 12 45 2 3.1440 2.00 1 10166 5 5 1 102 3 5.4281 2.00 1 10166 5 11 1016 5 1 10166 5 1 10166 5 1 1016 1 10166 5 1 10166 5 1 10166 5 1 10166 5 1 10166 5 1 10166 5 1 10166 5 1 10167 1 10167 1 10167 1 10167 1 10167 1 10167 1 10167 1 10167 1 10167 1 10167 1 101395 1 101393 <	5	11.	1395	2.00	0.00	0	.00	50.35	Effecti	ve-mass th	reshold (%	s):	1.0		
7 15 * Modal Damping Definition * < 1	6	15,	4939	2.00	0.00	0	.00	0.00	Effecti	ve-mass di	rections:		Х, У	, Z	~
8 31 9 35 10 35 11 42 12 45 13 46 14 5 15 74 0 5 11 42 14 20 15 74 0 5 14 200 15 74 0 5 14 200 15 74 0 5 13 46 14 7400 200 200 7 15320 11 135 12 31.404 200	7	15	🕨 Modal	Damping Defini	tion		4	/				×	ł	< 1	Mass
0 3 it 9 35.5 10 35.6 11 42.2 12 45.1 13 46.6 14 50.0 15 74.1 6 15.4221 2.00 13 46.6 15.4231 2.00 14 50.0 5 1.1395 2.00 7 15.5820 2.00 1 1.1395 2.00 8 31.4004 2.00 1 1.1395 2.00 7 15.5820 2.00 1 1.1395 2.00 8 31.4004 2.00 1 1.1395 2.00 8 31.4004 2.00 1 1.1395 <th< td=""><td>•</td><td>21</td><td>Viscous</td><td>Structural</td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td>Jal</td><td>>=</td><td>Mass</td></th<>	•	21	Viscous	Structural									Jal	>=	Mass
9 33 6 10 11 42 12 45 13 46 14 50 15 74 6 11.1395 20 11.1395 3 5.4281 200 1.1395 5 11.1395 6 15.4282 7 15.8202 8 31.4804 200 1.4204 8 31.4804 9 0.6 15.4281 2.00 7 15.8202 8 31.4804 14 50 15 11.395 14 50 15 11.395 16 15.4339 200 1.404 17 15.8202 8 31.4804 10 1.4004 10 1.4004 10 1.4004 10 1.4004 10 1.4004 10 1.4004	0	517				Modal Visco	us Damping Su	ummary							7
0 10 35 11 42 12 45 13 46 14 50 15 74 6 11.1395 7 15.820 7 15.820 8 31.4804 2.00	°	30.			User		Matrix			User + Matri	x				
11 42 12 45 13 46 14 50 15 74 6 15.432 7 15.5820 2.00	D 10	35.	Mode	Frequency	Schedule	Diagonal	Off-Diagona	Ratio	Diagonal	Off-Diag	onal Ratio			Resid	ual
12 45, 13 46, 14 50, 15 74, 6 15,4939 7 15,5820 7 15,5820 8 3,14804 2.00	11	42.	No.	(Hz)	(%)	(%)	Mean	Max	(%)	Mean	Max	^			
13 46. 14 50. 15 74. 6 15 74. 6 13 46. 14 50. 15 74. 6 15 74. 6 15 74. 6 15 74. 6 15.920 2.00 8 31.4804 2.00 8 3.14804 2.00 Viscous Damping Schedule Frequency (Hz) Damping (%) 3 8 Modal Damping Matrix File Name: St/Ubrata/Fame01_modes.vra5 Matrix Variable: Oo not use matrix> Matrix Usage: None	12	45.	1	1.0160	5	5						- 11			
14 50 15 74 5 11.1395 2.00 6 15.4939 2.00 7 15.5820 2.00 8 31.4804 2.00 Viscous Damping Schedule Frequency (Hz) Damping (%) 3 3 Modal Damping Matrix File Name: SV/Wbatak/Frame@framef	13	46.	2	5.4281	2.00							- 11			
15 74. 5 11.1395 2.00 6 15.4939 2.00 7 15.5820 2.00 8 31.4804 2.00 Use of the second sec	14	50.	4	7.4740	2.00										
6 15.4939 2.00 7 15.5820 2.00 8 31.4804 2.00 Viscous Damping Schedule Frequency (Hz) Damping (%) 3 Modal Damping Matrix File Name: Stylibrata/Frame/frame01_modes.vra5 Matrix Variable: 	15	74.	5	11.1395	2.00										
7 15.5820 2.00 8 31.4804 2.00 Viscous Damping Schedule Frequency (Hz) Damping (%) 3 3 Modal Damping Matrix File Name: S:\Vibrata\Frame\frame\frame\tram\trane\tram\trame\tram\trame\trame\tram\trame\trame\trame\tram\tr			6	15.4939	2.00										
8 31.4804 2.00 Viscous Damping Schedule Frequency (Hz) Damping (%) 2 <td><</td> <td></td> <td>7</td> <td>15.5820</td> <td>2.00</td> <td></td> <td></td> <td></td> <td></td> <td></td> <td></td> <td>- 1</td> <td></td> <td></td> <td></td>	<		7	15.5820	2.00							- 1			
Modal Damping Matrix File Name: Si\Vibrata\Frame\frame01_modes.vra5 Matrix Variable: Matrix Variable: None			8	31.4804	2.00			_				>	⊢		
Matrix Usage: None				Modal S:\Vibr Matrix	Damping Matrix rata\Frame\frame Variable: <00	Viscous Da Frequency (Hz File Name: -01_modes.vra5 not use matrix>	mping Schedu) Dampi	le ng (%) 2	3						
				Matrix	Usage: Non	e				~					

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.

Sol	ver E	xcitations	Modal Se	ettings	Dutput 🖌	Post Pro	cessing					
Iter	n Label	Output	Recovery	Recovery	Contour	Output	Output L	ocation			2	Node XY
		Variables	Point	Location	Intervals	CSys						Elem XY
												CSD
Ιſ	M No	dal XV-Plo	ot Output	Requests				×				Contour
			orouput	nequests				~				
	Outpu	ut Variable	25						1			Kequests
	~		, Total Trai	nslations				^			9	Solve
			TUTy	3							ĺ	Interactive
			TUTz									
	>		, Total Rot	ations								
] TV, Tota	l Velocitie	s (Base +	Flexible)							
] IA, Iota] II Displ	l Accelera acements	tions (Base (Flevible (e + Flexib Doly)	le)						
		UT,	Translation	ns	,							
		\checkmark	UTx	4				~				Plot XY
	<							>			× .	Plot Contour
	Use	e nodal di	splacemer	nt coordin	ate syste	m					2	X Results
	Node	Selection										
		N	odec	~	1	Node	Groups	<u>^</u>				
			2			Nou	coroups					
			0									
			•									
	<u> </u>		Entity	Selection - S	Select Nod	es for Ou	itput Reque	ests	6			? ×
				d Op	emove	0	Exclude	+ 2				
					to	h	v 1	+ 8		Select All	₩ ⊄ ٩	North Pick A
			or				7 1			Previous	Delete	ОК
			Group				~			More	Method ^	Cancel
		5 5	Select			S	elect					
	0											
	0	OK		A	oply		Close					

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).

Vibrat	ta: Adva	anced Mod	al Dynamic	Analysis 5.	0.0							-		×
File Ful	icuons	POST PIO	cessing r	leib										
Events					-	-				A	New			
Name		Carat	XYP	refix	Type		vent De	finition Fi	le Trans	·	Conv			
frame_	FreqRs	p_Const	Freq	(sp_Const	Modal Fred	quency 5	:/ VIDrata	/Frame/fra	ime_rreqksp	_Const.	Сору	-		5
name_	irequa	p_vai	Treq	(sp_var	Modarried	fuency					Clear			
			2										T/	
										~	Open			
<										>	Save			
Event D	etails –			_										
FEM Ur	nits:	Inch (Po	und f)	~										
FEM Fil	e:	S:\Vibrat	ta\Frame\fra	me.modfe	m								Select F	EM
Input D	atabase	e: S:\Vibrat	ta\Frame\fra	me01_mo	des.vra5								Select	DB
Solve	r Ex	citations	Modal Se	ttings	Output	Post Pro	cessina							
Item	Label	Output	Recovery	Recovery	Contour	Output	Output	Location				^ N	lode XY	
		Variables	Point	Location	Intervals	CSys						E	lem XY	
Node	2	TUTx, UTx				basic							CSD	
Node	8	TUTx, UTx				basic						(Contour	
Node	15	TUTx, UTx				basic								_
												×	Request	s
													Solve	
												Internet	eractive	
														-1
												P	IOT XY	
												V Plo	t Contou	r
<											>		. Results	

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).

ne	XY Prefix	Туре	Event Definition File		New		
ne_FreqRsp_Const	FreqRsp_Cons	t Modal Frequency	S:/Vibrata/Frame/frame_Fr	reqRsp_Const	Сору		
ie_FreqRsp_Var	FreqRsp_Var	Modal Frequency	1		lear	-	
				> C	Open Save		
t Details							
Units: Inch (Pour	nd f) 🗸 🗸						
File: S:\Vibrata	\Frame\frame.modf	fem				Select FEM	
Database: S:\Vibrata	\Frame\frame01_m	odes.vra5				Select DB	
ver Excitations	Modal Settings	Output Post P	rocessing				
Type Item	Label	CSys Dir	Scale Factor Fund	ction Name	FL ^	Point Force	
Accel Motion	2	disp 1	1.0 Cons	st_10G_1to40Hz	s 4	Load Set	
					En	forced Motion	
						CSD	
Enforced Motion	on Excitations			×		how on FEM	
Assign Forcing Fu	unctions to Valid D	OF	~				
Treat excitation	n as rigid-body				U 🗌 U	se FastRMS	
3				4			
2	1 1	Accel Var Function Manager	nction Name _1to3g_1to40Hz	T FW		Delete	- 0
2 < 7	1 1 ОК	Function Manager Function Manager Functions Source Fies # Name 1 Const 100 the0 2 Var_tho3g the0 5	Ito3g_1to40Hz	rraModaFrequencyInfo V OrdNumDataTyr	New Attributes Edit Copy Delete import Export > Parents Children	Plot XY Sta Math Evaluate Operations Standard Operators Interpolate/Reduce Time/Frequency Modification Statistics Input Shaping User-defined	acked XYZ
< 7	1 1	Control Contr	Ito3g_1to40Hz Manage FunctionType Hz Frequency Response Function Hz Frequency Response Function	f (x)	New Attributes Edit Copy Delete Import Export > Parents Children	Plot XY Sta Math Evaluate Operations Standard Operators Interpolate/Reduce Time/Frequency Modification Statistics Interpolate/Reduce Time/Frequency User-defined	acked XYZ
2 < 7	1 1	All Function Sets	Ito3g_1to40Hz Manage FunctionType Hz Frequency Response Funct Hz Frequency Response Funct	f (x)	New Attributes Edit Copy Delete Import Export > Parents Children	Plot XY Sta Math Evaluate Operations Standard Operators Interpolate/Reduce Time/Frequency Modification Statistics Input Shaping User-defined	acked XYZ
2 < 7	1 1	Contemporation of the second	Ito3g_1to40Hz Manage FunctionType Hz Frequency Response Funct Hz Frequency Response Funct Late	rraModaFrequencyInfo V OrdNumDataTyr H ion Acceleration ion Acceleration	New Attributes Edit Copy Delete import Export > Parents Children New New	Plot XY Sta Math Evaluate Operations Standard Operators Interpolate/Reduce Time/Frequency Modification Statistics Input Shaping Input Shaping Input Shaping	acked XYZ
2 < 7	1 1	Function Manager Function Manager Functions Source Fies # Name 1 Const 100 the0 2 Var_tio3g_tto40t 5 C Function Sets	Ito3g_1to40Hz Manage FunctionType Hz Frequency Response Funct Kz Frequency Response Funct Ltage None Other	rraModaFrequencyInfo V OrdNumDataTyr H ion Acceleration * ion Acceleration * st Created Last Selected	New Attributes Edit Copy Delete Import Export> Parents Children	Plot XY Sta Math Evaluate Operations - Standard Operators - Interpolate/Reduce - Time/Frequency - Modification - Statistics - Input Shaping - User-defined	acked XYZ
2 < 7	1 1 ОК	Control Contr	Ito3g_1to40Hz Manage FunctionType Hz Frequency Response Function Hz Frequency Response Function	rraModalFrequencyInfo OrdNumDataTyr ion Acceleration ion Acceleration at Created Last Selected	New Attributes Edit Copy Delete Import Export> Parents Children	Plot Plot XY Sta Evaluate Operations Standard Operators Interpolate/Reduce Time/Frequency Modification Statistics Input Shaping User-defined	acked XYZ

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 Xdisplacement 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. Random Analysis

For this example, you will use both of the Isat launch models. You will start with the singlepoint 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. Create Common Acceleration PSD.

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.
 - 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."
 - 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).

🜵 Vibrata: Advanced Modal Dynamic Analysis 5.0.0		-		\times
File Functions Post Processing Help				
Events				
Name XY Prefix Type Event Definition File ISat_Rgb_RanGround RanGrount Modal Random 1 1	New Copy Clear Open Save	√ <	T	~
FEM Units: Inch (Pound f)				2
FEM File: S:\Vibrata\ISAT\ISat_Launch_Sm_Rgd.modfem			Select F	FEM
Input Database: S:\Vibrata\ISAT\isat_launch_sm_rgd.op2			Select	DB
Solver / Analysis Type: Modal Random V Data for Frequency/Random Analysis	Variables for Custom Solver	s		
Excitation frequency selection: Log spacing at Modes Excitation lower bound (Hz): -1 No. points in range: 20 Excitation upper bound (Hz): -1 No. points near modes: 5 Image: State of the state of t	Name	Val	ue ^	

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 f(x) 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.

Solver	Excitations	Modal Set	tings Outp	ut Post Pr	ocessing				
	Type Iter	m I	label	CSys Dir	Scale Factor	Function Name		F ^	Point Force
									Load Set
₩ E	Enforced Mot	tion Excitat	ions				? >	×	Enforced Motion
- A a a	ion Earcing Eur	actions to Vi	did DOE						CSD
Assi	ign Forcing Fur	ICUOIS to Va							Show on FEM
	reat excitation	n as rigid-bo	dy						
	Node	Dir	Scale	Туре	Function Name		^ 6	3	
	6000	1						2	Delete
	6000	2					1	5	
	6000	3		Accel	GroundTransport	t			
								7	
	_					>	× I		
		OK		Analy		Class			
		UK		Apply		Close		\sim	
<								>	

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.

Solver	Excitations	Modal Settings	Output	Post Processing						
_	Frequency	Modal Dam	ping (%)	Translational I	Modal Effectiv	e N	부			
Mode #	(Hz)	Viscous	Structural	x	Y	^				
162	390.8978	0.00	0.00	0.00	0.00					
163	390.9421	0.00	0.00	0.00	0.00		Select Mode	es		
164	392.1690	0.00	0.00	0.00	0.00		Effective-mas	s threshold (%): 1.0	
165	394.3011	0.00	0.00	0.00	0.00		Effective-mas	s directions:	X, Y	, Z ~
166	395.1197	0.00	0.00	0.00	0.01		All	Active	Rigid	< Mass
167	395.7593	0.00	0.00	0.00	0.00		None	Inactive	Residual	>= Mass
168	399.0983	0.00	0.00	0.01	0.00		-Set Mode St	tatus		
169	461.0577	0.00	0.00	0.19	0.00		Active	Rig	gid	Residual
170	493.3952	0.00	0.00	0.00	0.00					<u> </u>
171	524.9158	0.00	0.00	0.00	0.07		Mode Shap	e Summaries		2
172	531.8168	0.00	0.00	0.00	0.18		<u> </u>	≞		
173	544.4730	0.00	0.00	0.01	0.21					
174	551.5969	0.00	0.00	0.23	0.02					
			Totals ->	99.92	99.93	~				
<					>					

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

- 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.

Select Post	Processing Data	- - • š	8 👪 👪 📖 🕻	<mark>- 12 12 12 12 12 12 12 12 12 12 12 12 12 </mark>	* -	×
View 1	Untitled				Dynamic Max/Min	
Output Set	s 4RMS, VRA: ISat_Rgb_RanG	round.Rand_Contour	- ¥ ¥ ,	Program : Unknown Analysis Type : Random Set Value : 0.		
Output Vec	tors					
Deform			~ 射 🏹	Transform	Vector Info	
Contour	7033Plate Top SVMS von Mis	es Stress	~ X ¥	Transform	Vector Info	
9	033Plate Top SVMS von Mise 033Plate Bot SVMS von Mise 24100022T1 Translation (Tota	s Stress s Stress d Acceleration)		Multiple Cont	tour Vectors	
Conto 2	4100023T2 Translation (Tota 4100024T3 Translation (Tota	al Acceleration) al Acceleration)	ocations	Complex Results	ОК	
Laminat	e Options Section	Cut Stream	nline Options		Cancel	

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 g²/Hz rather than (in/s²)²/Hz.



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



Figure 4-49. Node 5577 total Y-acceleration in g²/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.
 - 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.

/ibrata: Advance	d Modal Dynai	mic Analysis 5.	0.0									~
Functions Po	ost Processing	Help										
ents												
ame		XY Prefix	Туре		FEM	Name		^	New			
at_Rgb_RanGrou	nd	RanGround	Modal	Random	S:\Vib	rata\ISAT\ISat_Lau	unch_Sm_	Rgd.m	Сору	,		
						_			Clear	r	•	
Vibrata: Advanc	ced Modal Dyr	amic Analysis	5.0.0							-	-	
e Functions	Post Processin	g Help										
Events												
Name		XY Prefix		Туре	Fw	ent Definition File	•	^ N	ew			
ISat Rob RanGro	ound	RanGround		Modal Ra	ndom S:/\	/ibrata/ISAT/ISat I	- Rob RanG	ire Co	vov			
ISat Rob RanGro	ound uncor	RanGround u	ncorrelated	Modal Ra	ndom		5-2	C	Par		•	
									cui			
												ГЛ
								Op	ben			
<								> Sa	ive			
Event Details FEM Units:	Inch (Pound f)	↓ ✓	Sm Ant ma	dfem							5	alact EEN
Event Details FEM Units: FEM File: Input Database:	Inch (Pound f) S:\Vibrata\ISA S:\Vibrata\ISA) ∨ T\lSat_Launch, T\isat_launch ;	_Sm_4pt.mc	odfem							S	elect FEN
Event Details FEM Units: FEM File: Input Database: Solver Excit	Inch (Pound f) S:\Vibrata\ISA S:\Vibrata\ISA	T\ISat_Launch_ T\isat_launch_ dal Settinos	_Sm_4pt.mc sm_4pt.op2 Output	odfem Post Pro	cessing						S	elect FEN Select DB
Event Details FEM Units: FEM File: Input Database: Solver Excit	Inch (Pound f) S:\Vibrata\ISA S:\Vibrata\ISA ations Mo Lat	T\ISat_Launch_ T\isat_launch_ dal Settings bel	Sm_4pt.mc sm_4pt.op2 Output Output	Post Pro	cessing Recovery	y Contour	Output	Output Loc	cation	~ [S	elect FEN Gelect DB
Event Details FEM Units: [FEM File: [Input Database:] Solver Excit Item	Inch (Pound f) S:\Vibrata\ISA S:\Vibrata\ISA ations Mo Lal	T\JSat_Launch_ T\isat_launch_ dal Settings bel	Sm_4pt.mc sm_4pt.op2 Output Output Variables	Post Prov Recovery Point	cessing Recovery Location	Contour Intervals	Output CSys	Output Loc	cation	^	Si S	elect FEN Select DB Se XY m XY
Event Details FEM Units: FEM File: Input Database: Solver Excit Item Contour Group	Inch (Pound f) S:\Vibrata\ISA S:\Vibrata\ISA ations Mo Lal BUS_S	TVISat_Launch_ TVisat_launch_s dal Settings bel HELLS	Sm_4pt.mc sm_4pt.op2 Output Output Variables SVMS	Post Prov Recovery Point All	cessing Recovery Location Centroid	Contour Intervals Rand_Contours	Output CSys	Output Loc	ation	^	Si S	elect FEN Select DB de XY n XY SD
Event Details FEM Units: FEM File: Input Database: Solver Excit Item Contour Group Contour Group	Inch (Pound f) S:\Vibrata\ISA S:\Vibrata\ISA ations Mo Lal BUS_S INST_P	T\ISat_Launch_ T\isat_launch_ dal Settings bel HELLS KG_FEM	Sm_4pt.mc sm_4pt.op2 Output Output Variables SVMS TAT	Post Prov Recovery Point All none	cessing Recovery Location Centroid	Contour Intervals Rand_Contours Rand_Contours	Output CSys	Output Loc	cation		Si Si Noc Eler Cor	elect FEN Gelect DE de XY m XY SD htour
Event Details FEM Units: EEM File: Input Database: Solver Excit Item Contour Group Contour Group Node	Inch (Pound f) S:\Vibrata\ISA S:\Vibrata\ISA tations Mo Lal BUS_S INST_Pi 55	T\ISat_Launch_ T\isat_launch_ dal Settings bel HELLS KG_FEM 77	Sm_4pt.mc sm_4pt.op2 Output Output Variables SVMS TAT TATy, TATz	Post Prov Recovery Point All none	cessing Recovery Location Centroid	Contour Intervals Rand_Contours Rand_Contours	Output CSys basic	Output Loc	cation		Noc Eler Cor	elect FEN Gelect DB de XY m XY SD Mour
Event Details FEM Units: [FEM File: [Input Database: [Solver Excit Item Contour Group Contour Group Node Element Group	Inch (Pound f) S:\Vibrata\ISA S:\Vibrata\ISA Mo Lal BUS_S INST_Pł 55 LAUNCHER_A	TVISat_Launch_ TVIsat_launch_ dal Settings bel HELLS KG_FEM 77 DAPTER_FEM	Sm_4pt.op2 output Output Variables SVMS TAT TATy, TATz BFx	Post Prov Recovery Point All none	cessing Recovery Location Centroid Corner	Contour Intervals Rand_Contours Rand_Contours	Output CSys basic	Output Loc	cation		Noc Eler Cor	elect FEN Gelect DB de XY m XY SD Mtour quests
Event Details FEM Units: FEM File: Input Database: Solver Excit Item Contour Group Contour Group Node Element Group	Inch (Pound f) S:\Vibrata\ISA S:\Vibrata\ISA ations Mo Lal BUS_S INST_P} 55 LAUNCHER_A	T\ISat_Launch_ T\isat_launch_ dal Settings bel HELLS KG_FEM 77 DAPTER_FEM	Sm_4pt.mc sm_4pt.op2 Output Output Variables SVMS TAT TATy, TATz BFx	Post Prov Recovery Point All All	cessing Recovery Location Centroid Corner	Contour Intervals Rand_Contours Rand_Contours	Output CSys basic	Output Loc	cation		Noc Eler C Cor X Re	elect FEN Gelect DB de XY m XY SD atour quests
Event Details FEM Units: [FEM File: [Input Database: [Solver Excit Item Contour Group Contour Group Node Element Group	Inch (Pound f) S:\Vibrata\ISA S:\Vibrata\ISA tations Mo Lal BUS_S INST_P S5 LAUNCHER_A	T\ISat_Launch_ T\isat_launch_ dal Settings bel HELLS KG_FEM 77 DAPTER_FEM	Sm_4pt.mc sm_4pt.op2 Output Output Variables SVMS TAT TATy, TATz BFx	Post Prov Recovery Point All none	cessing Recovery Location Centroid	Contour Intervals Rand_Contours Rand_Contours	Output CSys basic	Output Loc	cation		Si S	elect FEN Gelect DB de XY m XY SD atour quests
Event Details FEM Units: FEM File: [Input Database:] Solver Excit Item Contour Group Contour Group Node Element Group	Inch (Pound f) S:\/Vibrata\ISA S:\/Vibrata\ISA ations Mo Lal BUS_S INST_P} 55 LAUNCHER_A	T\ISat_Launch_ T\isat_launch_ dal Settings bel HELLS KG_FEM 77 DAPTER_FEM	Sm_4pt.mc sm_4pt.op2 Output Output Variables SVMS TAT TATy, TATz BFx	Post Provery Point All All	cessing Recovery Location Centroid Corner	Contour Intervals Rand_Contours Rand_Contours	Output CSys basic	Output Loc	cation		Noc Eler Cor X Re Sc	elect FEN Gelect DB de XY m XY SD ntour quests olve ctive
Event Details FEM Units: [FEM File: [Input Database: [Solver Excit Item Contour Group Contour Group Node Element Group	Inch (Pound f) S:\Vibrata\ISA ations Mo Lal BUS_S INST_P} 55 LAUNCHER_A	TVISat_Launch_ TVIsat_launch_ dal Settings bel HELLS KG_FEM 77 DAPTER_FEM	Sm_4pt.mc sm_4pt.op2 Output Output Variables SVMS TAT TATy, TATz BFx	Post Prov Recovery Point All none	cessing Recovery Location Centroid Corner	Contour Intervals Rand_Contours Rand_Contours	Output CSys basic	Output Loc	cation		Noc Eler C Cor X Re Sc Intera	elect FEN Gelect DB de XY m XY SD otour quests olve ctive
Event Details FEM Units: [FEM File: [Solver Excit Item Contour Group Contour Group Node Element Group	Inch (Pound f) S:\Vibrata\ISA S:\Vibrata\ISA ations Mo Lal BUS_S INST_P} 55 LAUNCHER_A	T\ISat_Launch_ T\isat_launch_ dal Settings bel HELLS KG_FEM 77 DAPTER_FEM	Sm_4pt.op2 output Output Variables SVMS TAT TATy, TATz BFx	Post Prov Recovery Point All All	cessing Recovery Location Centroid Corner	Contour Intervals Rand_Contours Rand_Contours	Output CSys basic	Output Loc	cation		Solution Sol	elect FEN ielect DB de XY m XY SD itour quests olve ctive
Event Details FEM Units: [FEM File: [Input Database: [Solver Excit Item Contour Group Contour Group Node Element Group	Inch (Pound f) S:\Vibrata\ISA S:\Vibrata\ISA tations Mo Lal BUS_S INST_P S5 LAUNCHER_A	T\ISat_Launch_ T\isat_launch_ dal Settings bel HELLS KG_FEM 77 DAPTER_FEM	Sm_4pt.mc sm_4pt.op2 Output Output Variables SVMS TAT TATy, TATz BFx	Post Prov Recovery Point All All	cessing Recovery Location Centroid	Contour Intervals Rand_Contours Rand_Contours	Output CSys basic	Output Loc	cation		Noc Eler Cor Re Sc Intera	elect FEN Gelect DB de XY m XY SD atour quests olve ctive
Event Details FEM Units: FEM File:	Inch (Pound f) S:\/Vibrata\ISA S:\/Vibrata\ISA tations Mo Lal BUS_S INST_P} 55 LAUNCHER_A	T\ISat_Launch_ T\isat_launch_ dal Settings bel HELLS KG_FEM 77 DAPTER_FEM	Sm_4pt.mc sm_4pt.op2 Output Output Variables SVMS TAT TATy, TATz BFx	Post Provery Point All All	cessing Recovery Location Centroid	Contour Intervals Rand_Contours Rand_Contours	Output CSys basic	Output Loc	cation		Noc Eler C Cor X Re Sc	elect FEN Gelect DB de XY n XY SD ntour quests olve ctive
Event Details FEM Units: [Input Database: [Solver Excit Item Contour Group Contour Group Node Element Group	Inch (Pound f) S:\Vibrata\ISA S:\Vibrata\ISA ations Mo Lal BUS_S INST_P) 55 LAUNCHER_A	TVISat_Launch_ TVIsat_launch_ dal Settings bel HELLS KG_FEM 77 DAPTER_FEM	Sm_4pt.mc sm_4pt.op2 Output Output Variables SVMS TAT TATy, TATz BFx	Post Prov Recovery Point All All	cessing Recovery Location Centroid Corner	Contour Intervals Rand_Contours Rand_Contours	Output CSys basic	Output Loc	cation		Noc Eler C Cor X Re Sc Intera	elect FEN select DB de XY m XY SD atour quests slive ctive
Event Details FEM Units: [FEM File: [Solver Excit Item Contour Group Contour Group Node Element Group	Inch (Pound f) S:\Vibrata\ISA S:\Vibrata\ISA tations Mo Lal BUS_S INST_P+ S5 LAUNCHER_A	T\ISat_Launch_ T\isat_launch_ dal Settings bel HELLS KG_FEM 77 DAPTER_FEM	Sm_4pt.op2 output Output Variables SVMS TAT TATy, TATz BFx	Post Prov Recovery Point All All	cessing Recovery Location Centroid Corner	Contour Intervals Rand_Contours Rand_Contours	Output CSys basic	Output Loc	cation		Solution Solutitaa Solution Solution Solution Solution Solution Solution So	elect FEN ielect DB de XY m XY SD atour quests olve ctive
Event Details FEM Units: [FEM File: [Input Database: [Solver Excit Item Contour Group Contour Group Contour Group Element Group	Inch (Pound f) S:\Vibrata\ISA S:\Vibrata\ISA tations Mo Lal BUS_S INST_Pi LAUNCHER_A	T\ISat_Launch_ T\isat_launch_ dal Settings bel HELLS KG_FEM 77 DAPTER_FEM	Sm_4pt.mc sm_4pt.op2 Output Output Variables SVMS TAT TATy, TATz BFx	Post Prov Recovery Point All All	cessing Recovery Location Centroid	Contour Intervals Rand_Contours Rand_Contours	Output CSys basic	Output Loc	cation		Noc Eler Cor X Re Sc Intera	elect FEN Gelect DB de XY m XY SD atour quests olive ctive

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.

Solver	Excit	ations Mod	dal Settings	Output F	Post Proc	essing			
	Туре	Item	Label	CSys	Dir	Scale Factor	Function Name	Function File	Point Force
1	Accel	Motion	5630	disp	3	1.0	GroundTransport	S:\Vibrata\ISAT\Correlated inp	Load Set
2	Accel	Motion	5631	disp	3	1.0	GroundTransport	S:\Vibrata\ISAT\Correlated inp	Enforced Motion
3	Accel	Motion	5632	disp	3	1.0	GroundTransport	S:\Vibrata\ISAT\Correlated inp	CSD
4	Accel	Motion	5633	disp	3	1.0	GroundTransport	S:\Vibrata\ISAT\Correlated inp	Show on FEM
5	Phase	CSD L	S(1,2)			0	(deg)		
6	Phase	CSD L	S(1,3)			0	(deg)		Use FastRMS
7	Phase	CSD L	S(1,4)			0	(deg)		
8	Phase	CSD L	S(2,3)			0	(deg)		Delete
9	Phase	CSD L	S(2,4)			0	(deg)		2
10	Phase	CSD L	S(3,4)			0	(deg)		
								1	
								>	

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**.
- 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.

-	UIPLOT			
File	Plot Options	t		
	Plot]		
	Load File	1		
	Load Workspace			×
	Save File	ho-fs1\u) (H:) > Vibrata > ISAT > uncorrelated input	✓ O Search uncorrelated input	
	Save Workspace	^)
	Export	Name Date modified	d lype Size	
	Done	isat_sm_tauncn_4pt.vra_xyout //21/2020 8:58	0 AIVI VKA_XYOUT FILE 500 KD	
	DUTE DUTE	_	5	
	Desktop			
	Documents			
-	Downloads			
<mark>ل</mark> ا	Music			
	Pictures			
	Videos			
-	Local Disk (C:)			
T	sjaeger (\\rmo-fs1\u) (H:)			
-	Scratch (S:)			
=	u (\\rmo-fs1) (U:)		0	
~	Г	×	Z	1
	File name:		Vibrata XYOUT (*.vra_xyout) V	
			4 Open Cancel]

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.

F	 Vibrata: Advanced Modal Dynamic Analys ile Functions Post Processing Help 		_		×			
Γ	Events							
	Name	XY Prefix	Туре	Event Definition	New	2		
1	ISat_Rgb_RanGround	RanGround	Modal Random	S:/Vibrata/ISAT/ISa	Сору	-		
ľ	ISat_Rgb_RanGround_uncorrelated	RanGround_uncorrelated	Modal Random	S:/Vibrata/ISAT/ISa	Clear			>
3	ISat_Rgb_RanGround_correlated	RanGround_correlated	Modal Random			~	т/	\
Ĭ					Open			
	<			× >	Save			

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 **b**utton, 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.

Solver	Excit	ations	Modal Settings	Output	Post Proc	essing			
	Туре	Item	Label	CSys	Dir	Scale Factor	Function Name	Function File ^	Point Force
1	Accel	Motio	n 5630	disp	3	1.0	GroundTransport	S:\Vibrata\ISAT	Load Set
2	Accel	Motio	n 5631	disp	3	1.0	GroundTransport	S:\Vibrata\ISAT	Enforced Motion
3	Accel	Motio	n 5632	disp	3	1.0	GroundTransport	S:\Vibrata\ISAT	CSD
4	Accel	Motio	n 5633	disp	3	1.0	GroundTransport	S:\Vibrata\ISAT	Show on FEM
5	Phase	CSD L	S(1,2)			0.0	(deg)		
6	Phase	CSD L	S(1,3)			0.0	(deg)		Use FastRMS
7	Phase	CSD L	. S(1,4)			0.0	(deg)		
8	Phase	CSD L	S(2,3)			0.0	(deg)		Delete
9	Phase	CSD L	S(2,4)			0.0	(deg)		
10	Phase	CSD L	S(3,4)			0.0	(deg)		
c								>	

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 4-64. 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 Yacceleration.



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.

 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).

🕨 Vibrata: Advanced Modal Dynami	_		×				
File Functions Post Processing	Help						
Events							
Name	XY Prefix	Туре	Event Definition File	New			
ISat_Rgb_RanGround	RanGround	Modal Random	S:/Vibrata/ISA1/ISat_Rgb_R	Сору			
ISat_Rgb_RanGround_uncorrelated	RanGround_uncorrelated	Modal Random	S:/Vibrata/ISAT/ISat_Rgb_R	clear	•		>
ISat_Rgb_RanGround_correlated	RanGround_correlated	Modal Random	S:/Vibrata/ISAT/ISat_Rgb_R		~	т/	~
ISat_Rgb_RanGround_Fast	RanGround_Fast	Modal Random					
			~	Open			
<			>	Save			

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.

Solve	r Excit	ations	Modal Settings	Output	Post Process	ing			
	Туре	ltem	Label	CSys	Dir	Scale Factor	Function Name	Function File	Point Force
1	Accel	Motion	n 6000	disp	3	1.0	GroundTransport	S:\Vibrata\ISAT	Load Set
									Enforced Motion
									CSD
									Show on FEM
									Use FastRMS
									Delete
								\sim	
<								>	

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 <u>not</u> 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).

V	Vibrata: Advanced Modal Dynami	c Analysis 5.0.0					_	-		\times
F	ile Functions Post Processing	Help								
Γ	Events									
	Name	XY Prefix	Туре	Event Definition File	^	New	~			
1	ISat_Rgb_RanGround	RanGround	Modal Random	S:/Vibrata/ISAT/ISat_Rgb_R	H	Сору	2			
	ISat_Rgb_RanGround_uncorrelated	RanGround_uncorrelated	Modal Random	S:/Vibrata/ISAT/ISat_Rgb_R	2	Clear		•		>
	ISat_Rgb_RanGround_correlated	RanGround_correlated	Modal Random	S:/Vibrata/ISAT/ISat_Rgb_R						
3	ISat_Rgb_RanGround_Fast	RanGround_Fast	Modal Random	S:/Vibrata/ISAT/ISat_Rgb_R			-		. /	
	ISat_Rgb_RanGround_mefmass	RanGround_mefmass	Modal Random		v I	Open				
	<			>		Save				

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

 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).

olver	Excitations	Modal Settings	Output	Post Processing				
	Frequency	Modal Dar	nping (%)	Translational M	Nodal Effective N	lass (%)		
Mode #	(Hz)	Viscous	Structural	x	Y	z	^	
19	66.5454	4 1.00	0.00	7.15	0.00	0.62		
20	69.4673	3 1.00	0.00	0.05	0.02	79.51		Select Modes
21	79.1392	2 1.00	0.00	0.00	0.61	0.00		Effective-mass threshold (%): 0.1
22	79.281	5 1.00	0.00	0.00	0.01	0.00	1	Effective-mass directions: X, Y, Z
								All Active Rigid < Mass
24	79.996	6 1.00	0.00	0.03	0.00	0.00		None Inactive Residual >= Mass
25	82.2934	4 1.00	0.00	0.01	1.65	0.00		Set Mode Status
26	87.7532	2 1.00	0.00	0.28	0.00	0.00		Active Rigid Residual
27	88.4728	8 1.00						5
28	89.8059	9 1.00	0.00	0.00	0.00	0.12		Mode Shape Summaries
29	92.606	5 1.00	0.00	1.51	0.00	0.00		

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.

lver	Excitations	Modal Setting	s Output	Post Processing								
	Frequency	Modal D	amping (%)	Translation	al Modal Effectiv	ve Mass (%)		中				
lode #	(Hz)	Viscous	Structura	I X	Y	Z	^					
162	390.8978	3 0.00	0.0	0.00	0.00	0.00						
163	390.942	1 0.00	0.0	0.00	0.00	0.00		Select Mode	:S	(9/).	1	
164	392.1690	0.0	0.0	0.00	0.00	0.00		Effective-mas	s threshold	(70):	.1	
165	394.301	1 0.00	0.0	0.00	0.00	0.00		Effective-mas	s directions		X, Y, Z	~
166	395.1197	7 0.00	0.0	0.00	0.01	0.00		All	Active	Rig	jid	< Mass
167	395.7593	3 0.00	0.0	0.00	0.00	0.00		None	Inactive	Resid	dual	= Mass
168	399.0983	3 0.00	0.0	0.01	0.00	0.00		-Set Mode St	atus			
169	461.057	7 0.00	0.0	0.19	0.00	0.00		Active	R	igid	Re	sidual
170	493.3952	2 0.00	0.0	0.00	0.00	0.65				-		
171	524.9158	3 0.00	0.0	0.00	0.07	0.00		Mode Shape	e Summarie	5		
172	531.8168	3 0.00	0.0	0.00	0.18	0.05		E i	<u>➡</u>			
173	544.4730	0.00	0.0	0.01	0.21	0.05						
174	551.596	9 0.0	0.0	0 0.23	0.02	0.00						
			Totals -	> 99.92	99.93	99.87	~					
							>					
Mo	de# (H	z) Viso	ous Stru	ictural X	Y	۷						
- 10			0.00	0.00	0.0	0.00		- Solori	Madar			
- 10		0.9421			0.00	0.00		Effectiv	e-mass thr	shold (×).	1
16	54 39				0.00			- File all	e moss em		,	V V 7
- 16		4.3011			0.00			Effectiv	/e-mass dire	ections:		Α, Υ, Ζ
16		5.1197			0.00			A		ctive	Rigid	< M
10	57 39				0.00			No	ne In	active	Residu	al >= N
16					0.01			Set M	lode Status			
16	59 46	1.0577	0.00	0.00	0.19	0.00	0.00	A	ctive	Rig	gid	Residu
17	70 49	3.3952	0.00	0.00	0.00	0.00	0.65		ci . c			
17					0.00			Mode	shape Sun	maries		
17	72 53	1.8168	0.00	0.00	0.00	0.18	0.05		<u></u>			
17	73 54	4.4730	0.00	0.00	0.01	0.21	0.05					
	74 55	1.5969	0.00	0.00	0.23	0.02	0.00					
17												

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.

- 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."
- 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.

Vibrata: Advanced Modal Dynamic Analysis 5.0.0	- 🗆 ×
File Functions Post Processing Help	
Events	
Name XY Prefix Type Event Definition File New Frame_RSpec_Abs Abs Modal Response Spectrum Copy Clear Open Open Save Save Save Event Details FEM Units: Inch (Pound f) FEM File: S:\Vibrata\Frame\frame.modfem Input Database: S:\Vibrata\Frame\frame01_modes.op2	Select FEM Select DB
Solver Excitations Modal Settings Output Post Processing	
Solver / Analysis Type: Modal Response Spectrum	
Data for Response Spectrum Analysis	
Summation Method: Absolute Value ~ Name	Value ^
Closely-spaced-mode factor: 1.0	×

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 f(x) button to bring up the Function Manager.

S	olver Excitations N	lodal Settings	Output Post	Processing	function Name	function film		A Poi	nt Force
	type item	Laber	Ciys 1	JII Stale Fact		Function File		Lo	ad Set
Y	Enforced Motio	on Excitat	ions			?	×	Enfor	ced Motion
-4	Assign Forcing Fund	tions to Va	alid DOF					Shov	on FEM
	Treat excitation	as rigid-bo	dy					Use F	astRMS
	Node	Dir	Scale	Туре	Function Name	\sim	3		Delete
2	0	1					f(x)		
	0	2					P ²		
	0	6					<u> </u>		
	<					>		~	
		ОК		Apply		Close			

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 <u>velocity</u> 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.

承 Function Manager					- 🗆	×
Functions Source Files	Manage vraModa	IRSpectrumin fo 🗸	Plot	Stacked	XYZ	
# Name 1 Rsp Spec Base Lateral Shoo	FunctionType Or k Response Spectrum Ve	rdNumDataType + New locity ^ Attributes Edit	Math			
	New Function	· ·		-	- 🗆	×
	Function Type:	Response Spectrum V]2	5		
	X Spacing	Start	0	Х	Y	
	⊖ Even	End	0.3	0.10	4.5	
	Our Uneven	Increment	0.1	1.85	10.0	
<			0.1	100	0.15	
All None		4 # Points	4			
Function Sets	10 ¹ 10 ⁰					
	10 ⁻¹ 1	0 ⁰ 10 ¹	10 ²			
	Response Spectru	um Function Attributes		<i>ı</i>	Attributo	
All None	Name:	Rsp Spec Base Late	eral	'	Attributes	5
Done Cancel	Interpolation Type	e: LogLog	~			
	Y Axis Type:	Velocity	7			
	Create Cance	əl				

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

New Function	_		×
Choose a destination file:			
H:\Vibrata\Frame\frame_excitations.fcn		~	
OK Cancel			

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.

Solver	Excit	ations	Modal S	ettings	Output	Post Pro	ocessing			
	Туре	Iten	n	Label	CSys	Dir	Scale Factor	Function Name	^	Point Force
1	Veloc	Rsp Sp	bec	0	basic	1	1.0	Rsp Spec Base Lateral		Load Set
1	Enforte	d Moti	on Excitat	tions				?	×	Enforced Motion
										CSD
Assi	ign Ford	ing Fun	ctions to V	alid DOF						Show on FEM
	Treat ex	ditation	as rigid-bo	ody						
	No	Ja	Dir	Sca	le	Туре	Function Name	~		Use FastRMS
	0		1			Veloc	Rsp Spec Base La	teral	f(x)	Delete
	0		2						₽	
	0		6						<u></u>	
							1			
			1		CSys Dir Scale Factor Function Name basic 1 1.0 Rsp Spec Base Lateral Load Set ? × Enforced Mot CSD Show on FEN Use FastRMS Use FastRMS Delete Apply Close					
			1						Point F Iteral Enforced ? X f(x) Show on Image: Comparison of the second	
		_						-		
			OK			Apply		Close		

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).

Solver	Excitations	Modal Settings	Output	Post Processing					
	Frequency	Modal Dar	mping (%)	Translational	Modal Effective	4 ←			
Mode #	(Hz)	Viscous	Structural	x	Y ^				
1	1.016	0.00	0.00	89.07	0.00				
2	3.144	0.00	0.00	8.50	0.00	Select Mode	es		
3	5.428	1 0.00	0.00	2.03	0.00	Effective-mas	s threshold (%): 1.0	
4	7.474	0.00	0.00	0.40	0.00	Effective-mas	s directions:	Х, Ч	Y, Z ∨
5	11.139	5 0.00	0.00	0.00	50.35	All	Active	Rigid	< Mass
6	15.493	9 0.00	0.00	0.00	0.00	None	Inactive	Residual	>= Mass
7	15.582	0.00	0.00	0.00	40.53	-Set Mode St	tatus		
8	31.480	4 0.00	0.00	0.00	3.76	Active	Ri	qid	Residual
9	35.670	9 0.00	0.00	0.00	0.00				
10	35.979	4 0.00	0.00	0.00	0.00	Mode Shap	e Summaries		
11	42.096	4 0.00	0.00	0.00	0.00	E 1	<u> </u>		
12	45.493	4 0.00	0.00	0.00	0.06				
13	46.018	4 0.00	0.00	0.00	4.96				
14	50.701	7 0.00	0.00	0.00	0.0C 🗸				
<					>				

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.

olver E	xcitations	Modal S	ettings	Output	Post Pro	cessing	 	
m Label	Output	Recovery	Recovery	Contour	Output	Output Location	∧ No	de X
	Variables	TOIL	Location	intervals	Coys		Ele	em XY
					1		C	CSD
	🖡 Co	ntour Output R	equests			×	Co	ntou
	Outp	ut Variables						
	Sel	ect nodal varia	bles	with Oak A			× R	eque
	>	UT, Trans	lations	xible Only)	2	2		olve
		UR, Rotat V, Nodal Velo	ions ocities (Flexible	Only)				orve
	~	A, Nodal Acc	elerations (Flex ational Acceler	tible Only)				active
	>	AR, Rotat	ional Accelerat	ions	(h) a			
	>	TV, Total Noc	dal Velocities (B	ase + Flexible)	xible			
	>] TA, Total Nod	lal Acceleration	ns (Base + Flexi	ble)	>		
	Outp	ut Groups						
			C	ontour Group	5	^	Plo	t XY
			ALL_N	ODES_AND_EL	EMS	~	Plot	Cont
	<					>		Donit
	_		Select	Sele	ct all nodes	3		Kesuli
	Shell/I	Beam Stress Re	covery Point:	none		~		
	Stress	Recovery Locat	tion: 🗹 Centr	oid 🗌 Corne	rs			
	Plies f	or Stress/Strain	Results: 🗹 Al	I Range:				
	Outp	ut Intervals						
	Co	ntour Interval S	et: ResSpec_C	Contours		✓ III ■ X		
	Singl	e frame contair	ning sum of all					
	mod	al responses fo	r entire spectru	im.				
		4						
		4	_					
		OK		Apply		Close		

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.

ame	XY Prefix	Туре	Event Definition File	^ New	ינ
ame_RSpec_Abs	Abs	Modal Response Spectrum	S:/Vibrata/Frame/Frame_RSpe	c_Abs.ev Copy	
ame_RSpec_SRSS	SRSS	Modal Response Spectrum		Clear	
2					
-					
				Open	
				Save	
nt Details					
1 Units: Inch (F	ound f) v				
File: SAVibr	ata\Frame\fram				Select F
5.(vibi					Selection
it Database: S:\Vibr	ata\Frame\fram	ne01_modes.op2			Select I
olver / Analysis Type:	Modal Sett	ings Output Post Pro	ocessing ~		
olver / Analysis Type: Data for Response S	Modal Sett Modal Respo pectrum Analys	ings Output Post Pro	v v	-Variables for Custom Sc	olvers
Diver / Analysis Type: Data for Response S Jummation Method:	Modal Sett Modal Respo pectrum Analys	ings Output Post Pro	∝ S	Variables for Custom Sc Name	olvers Value
Diver / Analysis Type: Data for Response S Summation Method:	Modal Sett Modal Respo pectrum Analys	ings Output Post Pro	~ 3	Variables for Custom Sc	Value
Data for Response S Summation Method:	Modal Sett Modal Respo pectrum Analys SR e factor: 1.0	ings Output Post Pro	v S	Variables for Custom Sc	Value
Diver / Analysis Type: Data for Response S Summation Method: Closely-spaced-mode	Modal Sett Modal Respo pectrum Analys SR: e factor: 1.0	ings Output Post Pro	∝ 3	Variables for Custom Sc	Value ^
Data for Response S Summation Method:	Modal Sett Modal Respo pectrum Analys SR: e factor: 1.0	ings Output Post Pro	∼ 3	Variables for Custom So	Value ^
Data for Response S Data for Response S Summation Method:	Modal Sett Modal Respo pectrum Analys SR: e factor: 1.0	ings Output Post Pro	∼ 3	Variables for Custom Sc	Value ^
Diver / Analysis Type: Data for Response S Summation Method: Closely-spaced-mode	Modal Sett Modal Respo pectrum Analys SR: e factor: 1.0	ings Output Post Pro	∝ 3	Variables for Custom Sc	Value
Diver / Analysis Type: Data for Response S Summation Method: Closely-spaced-mode	Modal Sett Modal Respo pectrum Analys SR: e factor: 1.0	ings Output Post Pro	Socessing ✓	Variables for Custom Sc	Value
Diver / Analysis Type: Data for Response S Summation Method: Closely-spaced-mode	Modal Sett Modal Respo pectrum Analys SR e factor: 1.0	ings Output Post Pro	Socessing ✓	Variables for Custom Sc	Value ^
Diver / Analysis Type: Data for Response S Summation Method: Closely-spaced-mod	Modal Sett Modal Respo pectrum Analys SR: e factor: 1.0	ings Output Post Pro	Socessing ✓	Variables for Custom Sc	Value
Data for Response S Data for Response S Summation Method:	Modal Sett Modal Respo pectrum Analys SR: e factor: 1.0	ings Output Post Pro	Socessing ✓	Variables for Custom So	Value
Diver / Analysis Type: Data for Response S Summation Method: Closely-spaced-mod	Modal Sett Modal Respo pectrum Analys SR: e factor: 1.0	ings Output Post Pro	Socessing ✓	Variables for Custom So	Value
Diver / Analysis Type: Data for Response S Summation Method: Closely-spaced-mod	Modal Sett Modal Respo pectrum Analys SR: e factor: 1.0	ings Output Post Pro	Socessing ✓	Variables for Custom Sc	Value



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.

					1	
lame	XY Prefix	Туре	Event Definition File	^ New	-	
rame_RSpec_Abs	Abs	Modal Response Spectrum	S:/Vibrata/Frame/Frame_RSpec_A	bs.e Copy		
rame_RSpec_SRSS	SRSS	Modal Response Spectrum	S:/Vibrata/Frame/Frame_RSpec_SI	RSS. Clear	•	
2				> Open Save		-
ent Details M Units: Inch (Po M File: S:\Vibra	ound f) ~	e.modfem			Sele	ect FE
		-01 modes and			Sel.	
Solver Excitations	Modal Settir Modal Respon	ngs Output Post Prod	cessing v			
Solver Excitations Solver / Analysis Type: Data for Response Sp	Modal Settir Modal Respon	ngs Output Post Prod se Spectrum s 3	verssing verson vers	iables for Custom Solve	ers	~
Solver Excitations Solver / Analysis Type: Data for Response Sp Summation Method: Closely-spaced-mode	Modal Settir Modal Respon ectrum Analysis NRL factor: 1.0	ngs Output Post Prod se Spectrum s 3	Var	iables for Custom Solve Name	ers Value	

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.

Vibrata: Advanced Moo	dal Dynamic A ocessing He	nalysis 5.0.0 Ip		- 🗆 X
Events	,	F		
Name	XY Prefix	Туре	Event Definition File	1
Frame_RSpec_Abs	Abs	Modal Response Spectrum	S:/Vibrata/Frame/Frame_RSpec_Abs.ev	
Frame_RSpec_SRSS	SRSS	Modal Response Spectrum	S:/Vibrata/Frame/Frame_RSpec_SRSS.ev Clear	•
Frame_RSpec_NRL	NRL	Modal Response Spectrum	S:/Vibrata/Frame/Frame_RSpec_NRL.ev	ATA
Frame_RSpec_NRC10	NRC10	Modal Response Spectrum		
2			Open	
<u> </u>			Save	
- Event Details				
EFM Units: Inch (Pr	ound fi 🗸	7		
FEM File: SAVibra	uta\Erame\fram	a modfem		Select EEM
Leavet Detabases Cit/Ghar				Select PEM
Input Database: 5:(vibra	ita\rrame\iran	leo1_modes.op2		Select DB
Solver Excitations	Modal Sett	ings Output Post Pro	ocessing	
Solver / Analysis Type	Modal Respo	nse Spectrum	V	
solice , randijsis typer	RSpec_NRL NRL Modal Response Spectrum Sylvibrata/Frame/Frame_RSpec_NRL.ev Open Save etails its: Inch (Pound f) etails its: Inch (Pound f) :: Sylvibrata/Frame/frame.modfem atabase: Sylvibrata/Frame/frame01_modes.op2 Select DB Analysis Type: Modal Response Spectrum Yariables for Custom Solvers			
Data for Response Sp	ectrum Analys	sis	Variables for Custom Solv	ers
Summation Method:	NR	C 10% 🗸 🗸	Name	Value
Closely-spaced-mode	factor: 1.0			
			<	>
			Į	

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/s².



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.
						• • • • • • • • • • • • • • • • • • • •	

Deemanaa		Summation Method						
Response	ABS	ABS NRL		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. Transient Analysis

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. Isat Model with RCS Thruster Firing.

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.
- 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.

•	odal Dynamic Analysis 5.0.0	×
File Functions Post P	rocessing Help	
Events] .	1
Name X ISat_RCS_Translation T 2	KY Prefix Type Event Definition File New Trans Modal Transient Copy Clear Open Save Save	
Event Details		
FEM Units: Inch (F	Pound f) V	
FEM File: S:\Vibr	rata\ISAT\ISat_Dploy_Sm.modfem	Select FEM
Input Database: S:\Vibr	rata\ISAT\isat_dploy_sm.op2	Select DB
Solver / Analysis Type	Modal Settings Output Post Processing 4	
Data for Transient A	analysis Variables for Custom Solvers	5
Data for Transient A Integration interval:	I.7388e-04 Dynamic uncertainty factor: 1.0 Name	s Value ^
Data for Transient A Integration interval: Event end time:	Induit transient Variables for Custom Solvers 1.7388e-04 Dynamic uncertainty factor: 1.0 4.0 Static uncertainty factor: 1.0	S Value
Data for Transient A Integration interval: Event end time: Initial Conditions:	Indian transient Image: Indian transient Inalysis Image: Variables for Custom Solvers Indian transient Image: Variables for Custom Solvers </td <td>s Value</td>	s Value
Data for Transient A Integration interval: Event end time: Initial Conditions: Prior Event file:	Inalysis A.O Zero J.7388e-04 Dynamic uncertainty factor: J.0 Name Static uncertainty factor: J.0 Name	S Value
Data for Transient A Integration interval: Event end time: Initial Conditions: Prior Event file: Static offset:	Inalysis I.7388e-04 Dynamic uncertainty factor: I.0 Variables for Custom Solvers Name I.7388e-04 Static uncertainty factor: I.0 Variables for Custom Solvers Name IIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIII	S Value ^

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 f(x) 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).

承 Function Manager		- 🗆 ×
Functions	Plot	
Source Files	Manage vraModalTransientExactt V XY	Stacked XYZ
# Name	FunctionType OrdNumDataType NumberEleme +	
1 RCS_UnitFrc_hist 1	Time Response Excitation Force 9 Attributes	
	Edit	
	New Function	- 🗆 X
	Function Type: Transient V	5
	X Spacing Start 0 X	Y
	O Even 0	0
	C.19	0
	Increment 0.1 0.2	1.0
4	0.7	1.0
All	4 # Points 9 0.71	0
	1.19	0
Function Sets	1.2	-1.0
	1.7	-1.0
	-1	
	0 0.5 1 1.5 2	
	Transient Function Attributes	
All N	Name: RCS_UnitFrc_hist	6 Attributes
9 Done Ca	Interpolation Type: LinLin ~	
	Y Avis Type: Force Y 7	
	8	
	Create	

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



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

olve	er Exci	tations	Modal Settings	Output	Post Pro	cessing			
	Туре	Item	Label	CSys	Dir	Scale Factor	Function Name	Fu ^	Point Force
1	Force	Node	3956	basic	3	20.0	RCS_UnitFrc_hist	Silv	Load Set
2	Force	Node	4225	basic	3	20.0	RCS_UnitFrc_hist	StV	Enforced Moti
3	Force	Node	4494	basic	3	20.0	RCS_UnitFrc_hist	S:\	CSD
4	Force	Node	3687	basic	3	20.0	RCS_UnitFrc_hist	StV	Show on FEM
			١	Point Forcing Fu	Force Ex	citations			? ×
		1		Dir X Y	Scale	Type 1	Function Name		f (x)
				Z		20 Force	RCS UnitFrc hist		-7
				Rx			1		
				Ry					È.
				P-					_
				ιν <u>ε</u> <					>
				Node Sele	ction				
					Nod	les	Nod	e Groups	^
					306	57			
					395	00	-		
					422	25			
			\		449	94			
				<	Se	>	<	Select	>
					OK	2	Apply	Close	

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.

Solver	Exci	tations	Modal Settings	Output	Post P	rocessing					
Ite	m		Label	Output	Recovery	Recovery	Contour	Output	Output Location	^	Node XY
				Variables	Point	Location	Intervals	CSys			Elem XY
Contour	Group	ALL_NO	DES_AND_ELEMS	UT, AT	none		Trans_Contours		Femap: Copy of ISat_	F	CSD
											Contour
											× Requests
										1	Solve
											Interactive
										_	Plot XY
										Š	Plot Contour
<									>		× Results

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).

Solver	Exci	tations Modal Settings	Output	Post Pr	rocessing					
lte	m	Label	Output	Recovery	Recovery	Contour	Output	Output Location	^	Node XY
			variables	Point	Location	Intervals	CSys			Elem XY
Contour	r Group	ALL_NODES_AND_ELEMS	UT, AT	none		Trans_Contours		Femap: ISat_RCS_Tra		CSD
Contour	r Group	RCS_PANEL_SHELLS	SVMS	All	Centroid	Trans_PeakStress		Femap: ISat_RCS_Tra]	Contour
									1	X Requests Solve Interactive
									2	Plot XY
									\sim	Plot Contour
<								>		× Results

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. Frame Model Transient Animation.

 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.

Vibrata: Advar File Functions	nced Modal Dynamic Analysis 5.0.0 Post Processing Help	- 🗆 X
Events	· · · · · · · · · · · · · · · · · · ·	
Name Frame_TransHat 2	XY Prefix Type Event Definition File New mmer Modal Transient Copy Clear Open Save Save	
Event Details		
FEM Units:	Inch (Pound f)	C-1 55M
Input Database	S:/Vibrata\Frame\trame\trame.modtem	Select DR
input Database.	Stylerate traine traine or industop2	Select DB
Solver Exc	itations Modal Settings Output Post Processing	3
Solver / Analys	sis Type: Modal Transient 🗸 🗸	
Data for Trar	variables for Custom Solve	rs
Integration in	nterval: 1.3378e-03 Dynamic uncertainty factor: 1.0 Name	Value ^
Event end tim	10 Static uncertainty factor: 1.0	
initial conditi	tons: Quasi-static 5	
Prior Event fil		
Static offset:	✓	
		~
	<	>

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 *interference* 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).

g Functions Scale 0.96723 -0.25390	Type F	Function Na	? me	×		CSD Show on FE Use FastRM
g Functions Scale 0.96723 -0.25390	Type F	Function Na	me	3		Show on FE
Scale 0.96723 -0.25390	Type F	Function Na	me	3		Use FastRM
0.96723 -0.25390						USE FASILIN
-0.25390						
				f (x)		Delete
				2		
				<u> </u>		
e nodal displacem	ent coordinate	system			>	
Selection						
Nodes			Node Groups			
10						
	Entity Selection	- Select Nodes for E	citation		?	×
	5 <u>0 Add</u> C	Remove C	Exclude	Select All	🖲 🔍 🗞 🔌	Pick ^
	ID 10 Y	to	by 1	Previous	Delete	<u>ок</u> 6
				More	Method Ca	ancel
		×		~		
	>	<		>		
4 Select			Select			
			Self-correlated			
	e nodal displacem Selection Nodes 10 4 Select	e nodal displacement coordinate Selection 10 Interview Selection	e nodal displacement coordinate system Selection Nodes 10 Entity Selection - Select Nodes for En Group Group Control of C	e nodal displacement coordinate system Selection Nodes Nodes Node Groups 10 Entity Selection - Select Nodes for Excitation Select 4 Select Select Select Select Self-correlated	e nodal displacement coordinate system Selection Nodes Nodes Node Groups 10 Entity Select Nodes for Excitation Select All Previous More 4 Select Select Self-correlated	e nodal displacement coordinate system Selection Nodes Node Groups 10 Entity Selection - Select Nodes for Excitation Select All @ @ @ @ More Method ^ Ca A Select Select Self-correlated

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 f(x) 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).

Vibrata Documentation: Example Problems

	🕴 Point	Force Excitat	ions		? ×]			
	Forcing F	unctions							
	Dir	Scale	Туре	Function Name					
	X	0.96723			E Carl	1			
	Y	-0.25390				•			
	Z				-2				V
Function M	anager							- U	×
Functions					_	Plot			
Source File	es		Manage	vraModalTransientExactl \	/	XY	Stacked	i XYZ	
#	Name	Fun	ctionType	OrdNumDataType Nun +	New	2			
1 HammerFr	rc_Unit_Tri_	Pulse Time I	Response	Excitation Force 4	Attributes	Math			
	[Now Fun	ction	_	Edit	_		Save	
			cuon		_			Curo	_
		Function Ty	pe: Pu	lse 🗸	3				
		X Spacing		Start	0.0	X	γ		
		⊖ Even		End	4	0	0		
		Unever	n	Incromont	4.0	0.1	0		
			-	Increment	1.0	0.3	0		
				# Points	4	1			
<									
All	N	1				≤ 1			
		0.5							
Function Se	ts								
		0 0.0	0.1	0.15 0.2 0.25	0.3				
		Pulse Fun	ction Attri	butes					
		Name:		HammerFrc_Unit_Tri_Pu	Ilse		Attributes		
		Interpolati	on Type:	LinLin ~		4	ı\		
		Y Axis Ty	pe:	Force 🗸	5				
		Pulse Sh	ape:	Triangle	6	7			
All							<u> </u>	Help	
Done		Start	0.0 V	Vidth 0.2 Amplitu	de 1.0 E	nd .3	Apply		
		Create	9 Cancel						

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

承 New Funct	ion		<u> </u>	×	
Choose a dest	nation file:				
H:\Vibrata\F	rame\frame_e>	citations_har	mmer.fcn	×	
ОК	Cancel				

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



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

 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.

Solver Excitations Modal Settings Output	Post Processin	g				
Item Label Output	Recovery	Recovery Location	Contour Intervals	Output CSys	Output Locat ^	Node XY
Variables	Point					Elem XY
					1	CSD
Contour Output Requests				×		Contour
Output Variables						X Requests
Select nodal variables						- Andresis
UT, Translations 2	e Only)					Solve
> UR, Rotations		-	-			Interactive
V, Nodal Velocities (Flexible Only A Nodal Accelerations (Elexible	y) Only)					
> TU, Total Nodal Displacements (Base + Flexible)					
TV, Total Nodal Velocities (Base TA, Total Nodal Accelerations (B	+ Flexible)					
RFM, Reaction Forces and Mom	ents					
> S, Stress in Shell/Solid Elements				×		
						Plot XY
Output Groups					Ý	Plot Contour
< Conto	ur Groups				>	X Results
ALL_NODE	S_AND_ELEMS			~		
<)			
Select	Select all	nodes 3				
Shell/Beam Stress Recovery Point: none	e Comor			<u> </u>		
Plies for Stress/Strain Results: V All Ra	inge:					
	gei					
Output Intervals						
Contour Interval Set: Trans_Contour	S	`	< 📒 🖻 🗡			
Set Times for Phy	ysical Response I	Recovery				
Set times selection intent:	O Set specif	ic output tim	es:			
First output time: 0	_	Times		^		
and any two of				-		
Last output time: 3.0				-		
Output interval: 0.025	·			-		
Total outputs:				-		
4	L					
				- V		
	<		>			
Select quantities to store: 5						
Responses at each time Max	Min Deak	DMC				
	Will D Peak	KIVIS				

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.




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. <u>fcn–Vibrata Function Class.</u>
- 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. <u>Compute Output.</u>
- 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. <u>Response Spectrum Analysis</u>

- 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 <u>http://www.ata-</u><u>e.com/software/rmsserver/</u>. 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 <u>software@ata-e.com</u>. 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.





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.

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.

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

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 <u>vibrata@ata-e.com</u> 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. <u>REFERENCES</u>

- [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