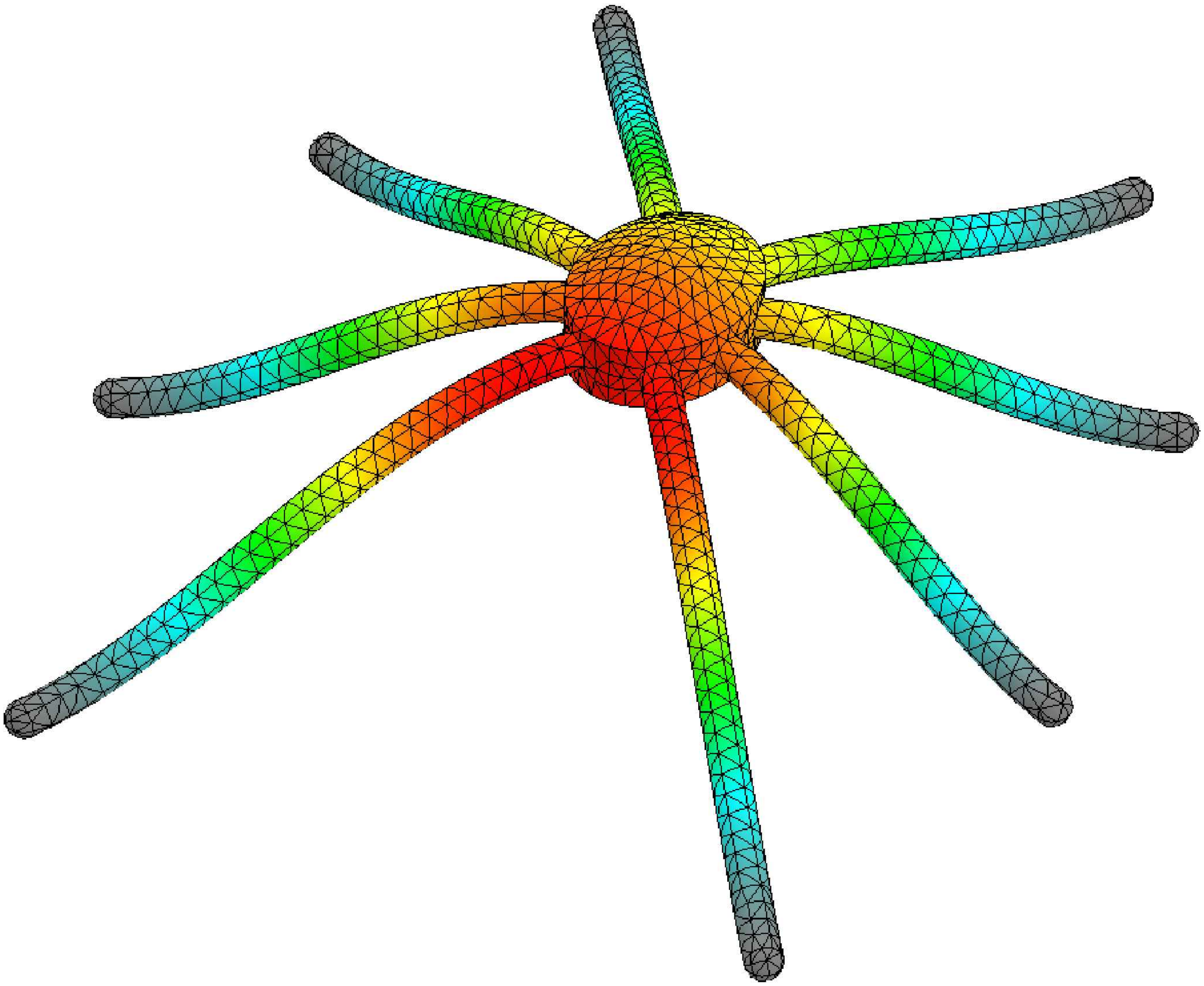
Vibration Analysis

with SolidWorks® Simulation 2014



Paul M. Kurowski

Design Generator Inc.

SOLIDWORKS Solution Partner

SDC PUBLICATIONS

Better Textbooks. Lower Prices.
www.SDCpublications.com

Vibration Analysis with SolidWorks Simulation 2014

Paul M. Kurowski, Ph.D., P.Eng.

SOLIDWORKS Solution Partner

SDC Publications

Design Generator Inc.

SDC Publications
P.O. Box 1334
Mission, KS 66222
913-262-2664
www.SDCpublications.com
Publisher: Stephen Schroff

About the Cover:

The image on the cover shows the first mode of vibration of SPIDER 2014 model.
All models used in this book may be downloaded from www.SDCPublications.com
The SPIDER2014 model can be found in Chapter 00 Cover Page. It has been saved with Modal analysis study ready to run.

ISBN-13: 978-1-58503-910-4
ISBN-10: 1-58503-910-1

Printed and bound in the United States of America.

About the Author

Dr. Paul Kurowski obtained his M.Sc. and Ph.D. in Applied Mechanics from Warsaw Technical University. He completed postdoctoral work at Kyoto University. Dr. Kurowski is an Assistant Professor in the Department of Mechanical and Materials Engineering, at Western University. His teaching includes Finite Element Analysis, Product Design, Kinematics and Dynamics of Machines and Mechanical Vibrations. His interests focus on Computer Aided Engineering methods used as tools of product design.

Dr. Kurowski is also the President of Design Generator Inc., a consulting firm with expertise in Product Development, Design Analysis, and training in Computer Aided Engineering.

Dr. Kurowski has published many technical papers and taught professional development seminars for SAE International, the American Society of Mechanical Engineers (ASME), the Association of Professional Engineers of Ontario (PEO), the Parametric Technology Corporation (PTC), Rand Worldwide, SolidWorks Corporation and others.

Dr. Kurowski is a member of the Association of Professional Engineers of Ontario and SAE International. He can be contacted at www.designgenerator.com

Acknowledgements

I would like to thank the students attending my various courses for their valuable comments and questions. I thank Tom Kurowski for editing and proof reading the text and the exercises. I thank my wife Elżbieta for her encouragement and technical support that made it possible to write this book.

Paul Kurowski

Table of contents

Before you start

Notes on hands-on exercises and functionality of Simulation

This book goes beyond a standard software manual. It takes a unique approach by bridging the theory of mechanical vibrations with examples showing the practical implementation of vibration analysis. This book builds on material covered in "**Engineering Analysis with SolidWorks Simulation 2014**".

We recommend that you study the exercises in the order presented in the book. As you go through the exercises, you will notice that explanations and steps described in detail in earlier exercises are not repeated in later chapters. Each subsequent exercise assumes familiarity with software functions discussed in previous exercises and builds on the skills, experience, and understanding gained from previously presented problems.

Exercises in this book require **SolidWorks Simulation**. The **SolidWorks Simulation Product Matrix** document is available at:

http://www.solidworks.com/sw/docs/solidworks-simulation-matrix.pdf

Information on the **SolidWorks Simulation** Packages is available at:

www.solidworks.com/sw/products/10169_ENU_HTML.htm

All exercises in this book use **SolidWorks** models, which can be downloaded from www.SDCpublications.com. Most of these exercises do not contain any **Simulation** studies; you are expected to create all studies, results plots, and graphs yourself.

Animations of selected results may be found using the YouTube link at www.designgenerator.com

All problems presented here have been solved with **SolidWorks Simulation Premium** running on Windows 7 in a 64 bit operating environment.

Working on exercises you may notice that your results may be slightly different from results presented in this book. This is because numerical results may differ slightly depending on the operating system and software service pack.

We encourage you to explore each exercise beyond its description by investigating other options, other menu choices, and other ways to present results. You will soon discover that the same simple logic applies to all functions in **SolidWorks Simulation**, be it structural, vibration or thermal analysis.

This book is not intended to replace software manuals. The knowledge acquired by the reader will not strictly be software specific. The same concepts, tools and methods apply to any FEA software.

Prerequisites

"**Vibration Analysis with SolidWorks 2014**" is not an introductory text to **SolidWorks Simulation**. Rather, it picks up Vibration Analysis from where it was left in the pre-requisite textbook "**Engineering Analysis with SolidWorks 2014**". If you are new to **SolidWorks Simulation** we recommend reading "**Engineering Analysis with SolidWorks 2014**" to gain essential familiarity with Finite Element Analysis. At the very least go through chapters 6, 18, and 19 of the pre-requisite textbook.

The following prerequisites are recommended:

- An understanding of Vibration Analysis
- An understanding of Structural Analysis
- An understanding of Solid Mechanics
- Familiarity with **SolidWorks**
- Familiarity with **SolidWorks Simulation** to the extend covered in "**Engineering Analysis with SolidWorks 2014**" or equivalent experience
- Familiarity with the Windows Operating System

Selected terminology

The mouse pointer plays a very important role in executing various commands and providing user feedback. The mouse pointer is used to execute commands, select geometry, and invoke pop-up menus. We use Windows terminology when referring to mouse-pointer actions.

Item	Description
Click	Self-explanatory
Double-click	Self-explanatory
Click-inside	Click the left mouse button. Wait a second, and then click the left mouse button inside the pop-up menu or text box. Use this technique to modify the names of folders and icons in **SolidWorks Simulation** Manager.
Drag and drop	Use the mouse to point to an object. Press and hold the left mouse button down. Move the mouse pointer to a new location. Release the left mouse button.
Right-click	Click the right mouse button. A pop-up menu is displayed. Use the left mouse button to select a desired menu command.

All **SolidWorks** file names appear in CAPITAL letters, even though the actual file names may use a combination of capital and small letters. Selected menu items and **SolidWorks Simulation** commands appear in **bold**, **SolidWorks** configurations, **SolidWorks Simulation** folders, icon names and study names appear in *italics* except in captions and comments to illustrations. **SolidWorks** and **Simulation** also appear in bold font. Bold font may also be used to draw reader's attention to particular term.

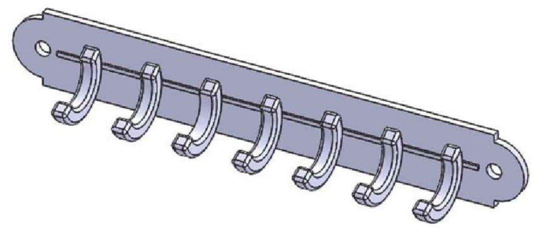
1: Introduction to vibration analysis

Topics covered

- Differences between a mechanism and a structure
- Difference between dynamic analysis and vibration analysis
- Rigid body motion and degrees of freedom
- Kinematic pairs
- Discrete and distributed vibration systems
- Single degree of freedom and multi degree of freedom vibration systems
- Mode of vibration
- Rigid Body Mode
- Modal superposition method
- Direct integration method
- Vibration Analysis with SolidWorks Simulation and SolidWorks Motion
- Functionality of SolidWorks Simulation and SolidWorks Motion
- Terminology issues
- Summary

Differences between a mechanism and a structure

A mechanism is not firmly supported and can move without having to deform; components of a mechanism can move as rigid bodies. On the contrary, any motion of a structure must involve deformation because a structure is, by definition, firmly supported. This motion may take the form of a one-time deformation when a static load is applied, or the structure may be oscillating about the position of equilibrium when a time varying load is present. In short, a mechanism may move without having to deform its components while any motion of a structure must be accompanied by deformation. If an object may move without experiencing deformation then it can be classified as a structure. Examples of mechanism and structure are shown in Figure 1-1.



Mechanism Structure

Figure 1-1: A mechanism and a structure.

A bicycle is a mechanism, it is designed to move. Motion of a bicycle can be studied without considering deformation of its components. A cloth hanger is a structure. It is designed to stand still and can move only about its position of equilibrium. Any motion of a structure is always accompanied by deformation.

Source: GrabCAD, Gorjup Design, Himanshu Singh Chauhan.

Depending on the objective of analysis, an object or its components may be treated either as a mechanism or a structure. A wind turbine rotor is a mechanism as it spins relative to the tower. Depending on the objective of analysis this mechanism may be treated as a rigid body mechanism or an elastic body mechanism. An individual blade may be treated as a rigid body component of a mechanism or as a structure if its vibration characteristics need to be analyzed (Figure 1-2).

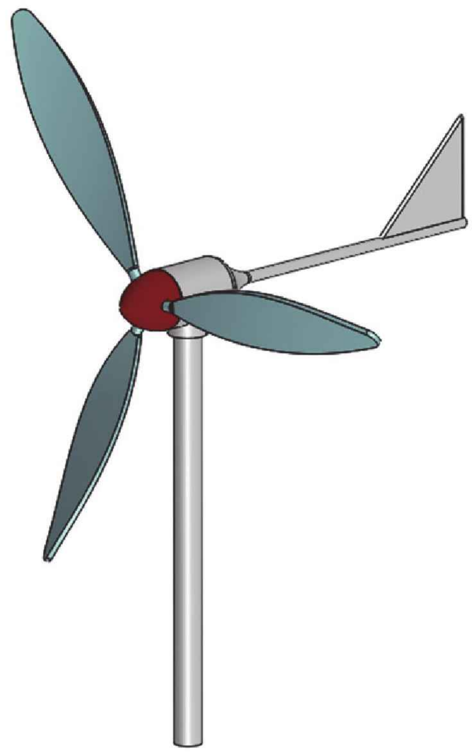


Figure 1-2: A wind turbine may be considered either as a mechanism or as a structure.

The wind turbine tower is a structure; but the rotor can be considered as a mechanism composed of rigid bodies, a mechanism composed of elastic bodies or as a structure.

Source: GrabCAD, Zachary Gilbert.

Difference between Dynamic Analysis and Vibration Analysis

Dynamic Analysis can be performed on a mechanism or a structure. The analyzed object can be an airplane wing, a car, a building subjected to an earthquake, an engine or a door bell making sound. Dynamic Analysis can deal with rigid and/or elastic bodies. **Vibration Analysis** is more specific, it deals only with elastic bodies vibrating about the position of equilibrium.

Rigid body motion and degrees of freedom

If an object moves without deforming, then this object is called a **Rigid Body** and the motion is classified as a **Rigid Body Motion**. The number of independent variables necessary to define the position of a rigid body with respect to a reference coordinate system equals the number of degrees of freedom of that body. The number of degrees of freedom of a rigid body is equal to the number of rigid body motions of that body.

Consider an unsupported rigid body; to define its position with respect to a Cartesian coordinate system we need to provide six pieces of information: linear coordinates x, y, z along any point belonging to this body, and three angles defining angular orientation of the body (Figure 1-3).

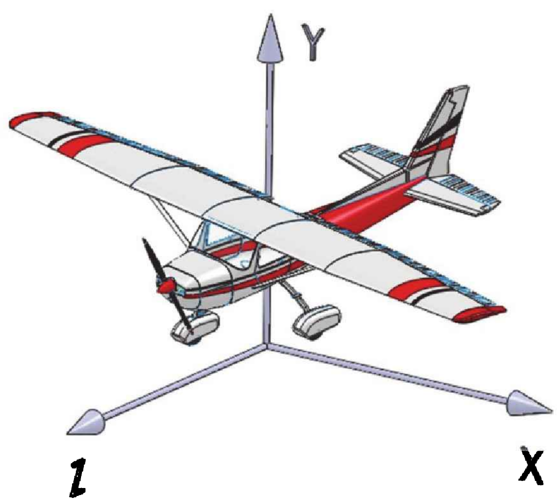


Figure 1-3: A flying plane considered as a rigid body has six degrees of freedom or six rigid body modes.

Source of model: GrabCAD.

In the case of a flying airplane, three linear coordinates are required to define the position of the center of mass and three angular coordinates are required to define the angular position.

Kinematic Pairs

A kinematic pair is comprised of two rigid bodies; one rigid body is fully restrained meaning it has no degrees of freedom. The other rigid body can move and is connected to the fully restrained body it in such a way that the number of degrees of freedom of the moving body is limited.

We will review basic kinematic pairs and their Degrees of Freedom (DOF) using the assembly models listed in Figure 1-4. Notice that all assemblies are not fully defined to allow for relative motion of assembly components.

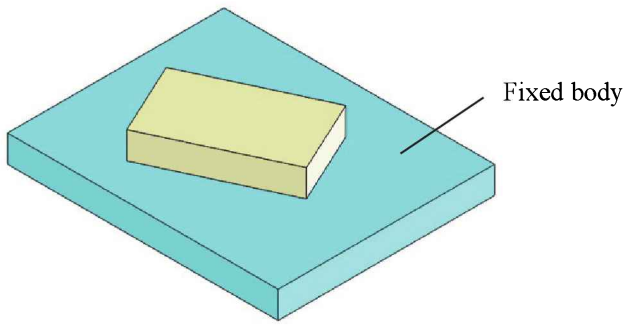
Kinematic pair	Number of translational DOF	Number of rotational DOF	SolidWorks Assembly file name
Planar	2	1	PLANAR
Prismatic	1	0	PRISMATIC
Revolute	0	1	REVOLUTE
Cylindrical	1	1	CYLINDRICAL
Spherical	0	3	SPHERICAL
Screw	1/2	1/2	SCREW

Figure 1-4: A summary of basic kinematic pairs and their number of degrees of freedom.

If opened in SolidWorks, you will find all assemblies consist of two components: the blue part is fixed (fully restrained) and the yellow part can move relative to the blue part.

To review the above models, use assembly commands **Move Component** and **Rotate Component**. No **SolidWorks Simulation** study is required. In all examples featuring kinematic pairs, the fixed bodies are blue, and the moving bodies are yellow. Animation of the above kinematic pairs can be easily done in **SolidWorks Motion**.

A **planar** kinematic pair is shown in Figure 1-5.

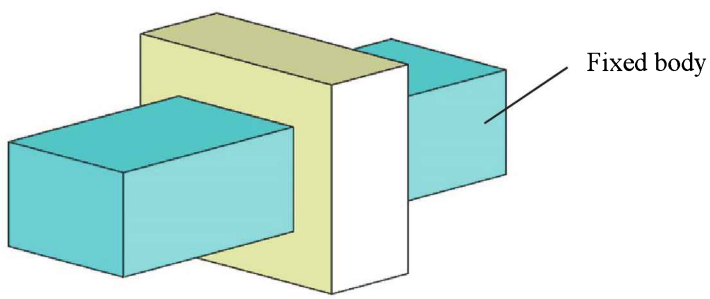


PLANAR.SLDASM

Figure 1-5: A planar kinematic pair removes three degrees of freedom from the moving body.

Two linear coordinates are required to define the position of the center of the sliding block and one angular coordinate is required to define its angular position. Therefore, the pair has two translational degrees of freedom and one rotational degree of freedom.

A **prismatic** kinematic pair is shown in Figure1-6.

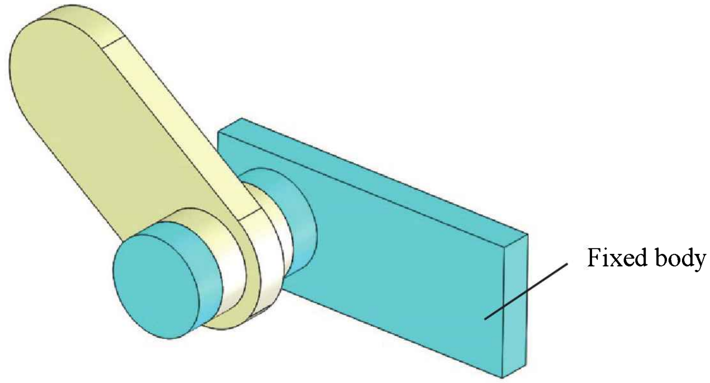


PRISMATIC.SLDASM

Figure 1-6: A prismatic kinematic pair removes five degrees of freedom from the moving body.

One linear coordinate is required to define the position of the sliding block. This pair has one translational degree of freedom.

A **revolute** kinematic pair is shown in Figure 1-7.

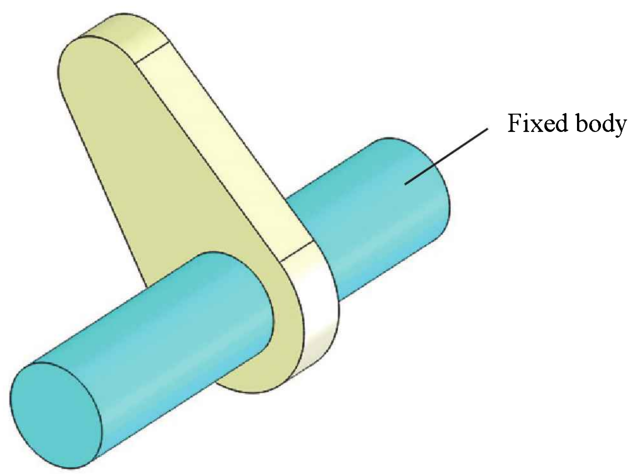


REVOLUTE.SLDASM

Figure 1-7: A revolute kinematic pair removes five degrees of freedom from the moving body.

One angular coordinate is required to define the position of the moving part. The pair has one rotational degree of freedom.

A **cylindrical** kinematic pair is shown in Figure 1-8.

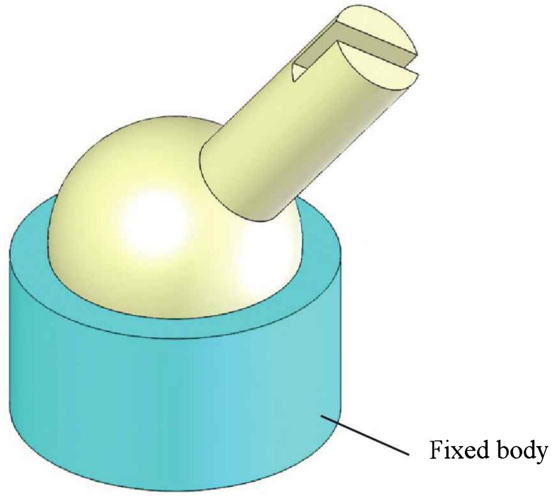


CYLINDRICAL.SLDASM

Figure 1-8: A cylindrical kinematic pair removes four degrees of freedom from the moving body.

One linear coordinate and one angular coordinate are required to define the position of the moving part. The pair has one translational and one rotational degree of freedom.

A **spherical** kinematic pair is shown in Figure1-9.

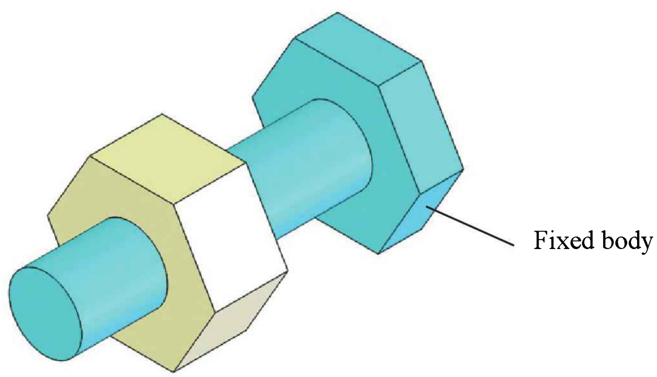


SPHERICAL.SLDASM

Figure 1-9: A spherical kinematic pair removes three degrees of freedom from the moving body.

Three angular coordinates are required to define the position of the moving part. The pair has three rotational degrees of freedom.

A **screw** kinematic pair is shown in Figure1-10.



SCREW.SLDASM

Figure 1-10: A screw kinematic pair removes five degrees of freedom from the moving body.

Either linear or angular coordinates fully define the position of nut. The pair has one degree of freedom that couples translation and rotation. When you review this model, notice that the thread is not modeled explicitly. Instead, its presence is modeled by a mechanical mate.

Discrete and distributed vibration systems

All that is required for vibration is a mass and stiffness. Therefore any real life object can be treated as a vibration system.

Depending on how mass and stiffness are distributed we can classify vibration systems as discrete or distributed systems. A discrete system is one where mass and stiffness are separated meaning that certain portions of the system are responsible for inertial properties while others are responsible for stiffness properties. Figure 1-11 shows two discrete systems: a single degree of freedom system and a two degree of freedom system. Both systems perform linear vibrations; the base and masses are treated as rigid bodies; deformations are limited to the springs which have no mass. Review the assembly model DISCRETE LINEAR in configurations *1DOF* and *2DOF* as shown in Figure 1-11.

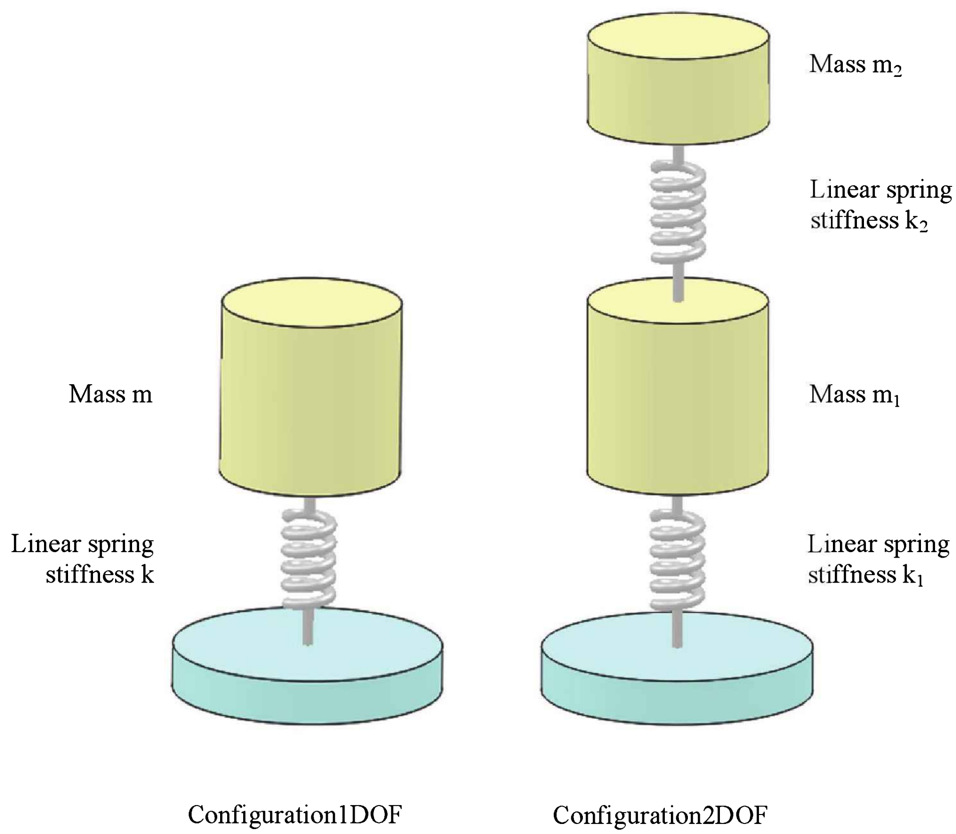


Figure 1-11: A single degree of freedom system (left) and two degree of freedom system (right) have mass and stiffness separated.

The cylindrical masses can move only in the vertical direction, they can't rotate. The base is fixed in both models.

One linear coordinate is required to define the position of the vibrating mass in the 1DOF model; two linear coordinates are required to define the positions of two vibrating masses in the 2DOF model.

A single degree of freedom (SDOF) discrete vibration system is a "workhorse" of the theory of vibration. Vibration responses of a SDOF have simple analytical solutions and many real life problems can be successfully studied by representing the system as a SDOF. We will study a SDOF in the second chapter to introduce essential concepts of vibration analysis with **SolidWorks Simulation** and **SolidWorks Motion**.

A discrete vibrating system can also perform rotary (angular) vibration. An example is the SWING ARM assembly model shown in Figure Figure1-12. The base and arm are treated as rigid bodies; deformation is limited to the massless spring.

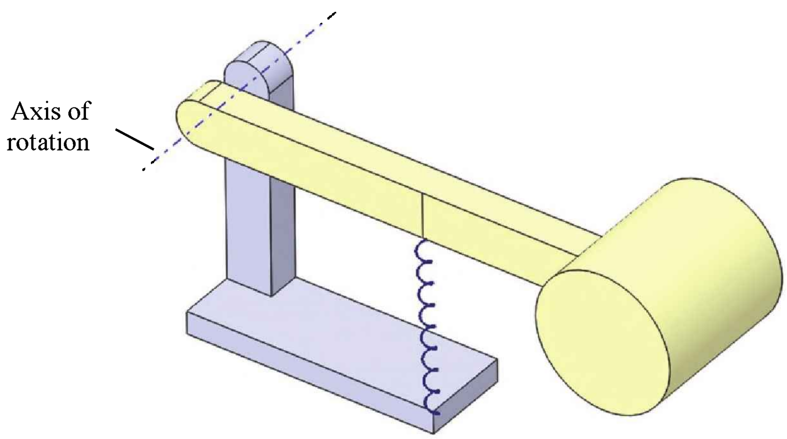


Figure 1-12: A single degree of freedom rotary vibration system has mass and stiffness separated.

One angular coordinate is required to define the position of this system.

Referring to Figure 1-12, notice that the arm performs angular vibration, therefore vibration properties are defined by its mass moment of inertia about the axis of rotation. The spring experiences linear deformation, therefore its vibration properties are defined by linear stiffness. The swing arm could also perform angular vibration if a torsional spring was installed at the hinge.

Discrete systems don't exist in real life and discrete representation is only a modeling simplification. We use discrete system representations because many problems can be described with sufficient accuracy by discrete models, usually with a few degrees of freedom. Degrees of freedom of discrete systems are clearly defined because deformation is limited to springs (linear and angular) present in the model. Analytical solutions of discrete systems are simple and results are often intuitive.

In a distributed vibrating system, mass and stiffness are not separated. All portions of an object are responsible for both mass and stiffness properties. Figure 1-13 shows the model CLIP which represents a distributed vibration system.

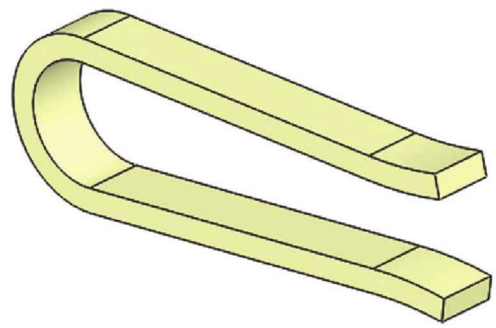


Figure 1-13: In a distributed vibration system, all portions of the object are responsible for both mass and stiffness properties.

Mass and stiffness are not separated.

Mode of vibration

The mode of vibration is the most fundamental property of any vibrating system. Its central role in vibration analysis requires us to discuss it in this introductory chapter. We will refine this definition later.

The mode of vibration can be defined as the preferred way of a structure to vibrate. It is characterized by its frequency of vibration, mass participating in the vibration, and shape of vibration. The number of modes of vibration is equal to the number of degrees of freedom of the vibrating system.

Shapes of vibration in the first (and only) mode of the discrete vibration system DISCRETE LINEAR in the *1DOF* configuration is shown in Figure 1-14. This model is available with **Frequency** studies set up and ready to run. Please run the solutions before proceeding.

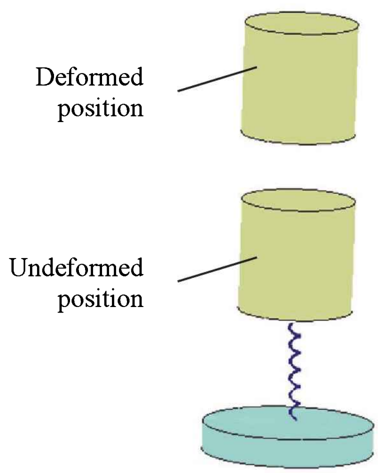


Figure 1-14: The shape of vibration of the 1DOF model. The moving part translates as a rigid body; it does not experience any deformation. The only deforming element is the spring.

The mass moves as a rigid body, its motion is restricted to translation. Motion of the moving part is not restricted by any kinematic pair in this model. Restraints are defined in the SolidWorks Simulation Frequency study.

Notice that the "Deformed shape" shown in Figure 1-14 refers to the deformation of the spring. The cylinder itself is not deforming, it performs linear oscillations as a rigid body.

The SWING ARM model in the *1DOF* configuration also has a **Frequency** study defined; please run the solution before proceeding.

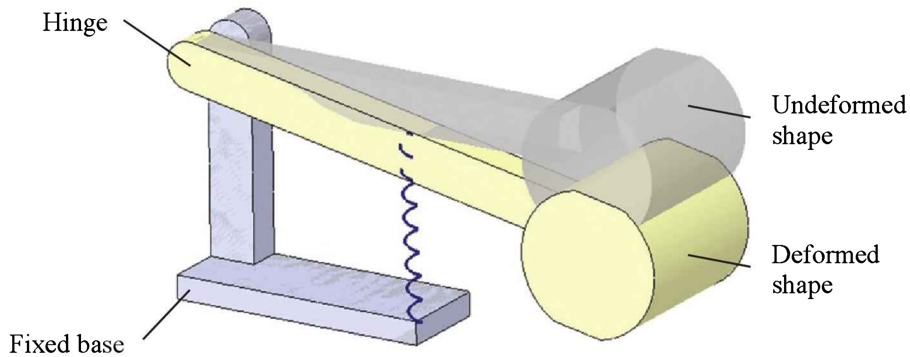


Figure 1-15: Shape of vibration of the SWING ARM model.

The arm rotates as a rigid body; it does not experience any deformation. The only deforming element is the spring. Motion of the moving part is controlled by a hinge.

Review assembly model SWING ARM and notice that the vertical post in the base serves only to locate the hinge position of the swing arm. The hinge itself is not modeled. The hinge is simulated in the **Frequency** study using a **Fixed Hinge** restraint.

A two degree of freedom discrete system has two modes of vibration, each one characterized by its own unique shape as demonstrated by the assembly model DOUBLE PENDULUM, shown in Figure 1-16. Run the **Frequency** study before proceeding.

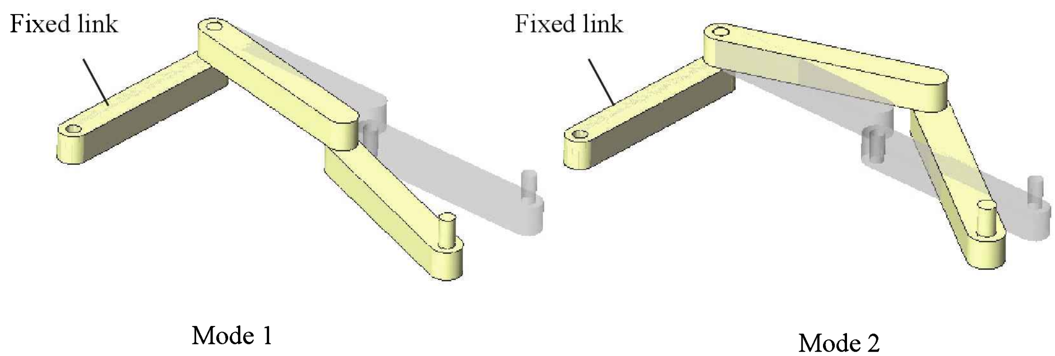
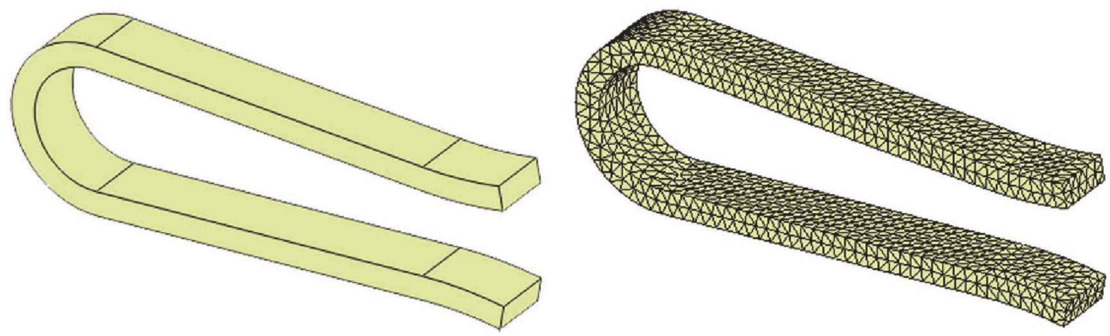


Figure 1-16: Shapes of vibration of the DOUBLE PENDULUM model.

The shape in mode 1 and mode 2 are result plots in the Frequency study. The undeformed shape is superimposed on the deformed model.

Review the **Frequency** study in the DOUBLE PENDULUM model. Notice that **Global Contact** is set to **Allow Penetration** in order to disconnect the touching faces which otherwise would be bonded. The three links are connected by two **Pin Connectors** which have torsional stiffness and therefore act as torsional springs

Distributed systems, where deformation is not limited to discrete springs have an infinite number of degrees of freedom and consequently, an infinite number of modes of vibration. Consider CLIP (Figure 1-13) to be unsupported. CLIP treated as a rigid body has six degrees of freedom: three translations and three rotations; we may say it has six rigid body motions. When treated as an elastic body, CLIP will have an infinite number of degrees of freedom associated with elastic deformation in addition to those six degrees of freedom associated with rigid body motions (RBMs). Consequently is has an infinite number of modes of vibration. When this CLIP is analyzed using FEA, the number of elastic degrees of freedom becomes finite due to discretization, an inherent part of FEA (Figure 1-17).



CAD model Finite element mesh

Figure 1-17: CLIP model before and after discretization.

Meshing was performed using solid elements with a default element size. This produced a mesh with 13275 nodes.

Remembering that solid elements have 3 DOF per node, the number of degrees of freedoms in the finite element mesh is 39825.

The total number of degrees of freedom is found by adding:

	Number of associated degrees of freedom
Rigid body motions	6
Elastic deformation	39825
Total:	39831

The CLIP model after discretization and with Rigid Body Modes removed by restraint has 39825 elastic DOFs. Theoretically, this gives the model 39825 modes of vibration. However, all those higher modes are never excited in the real life. Only the first few modes with frequencies low enough to be excited are important in the vibration response.

When the model is unsupported (as is the CLIP model) **Simulation** detects six **Rigid Body Modes** and assigns them frequencies of 0Hz. The first three modes are rigid body translations and the second three are rigid body rotations (Figure 1-18).

Translation along x

Rotation about x

Translation along y

Rotation about y

Translation along z

Rotation about z

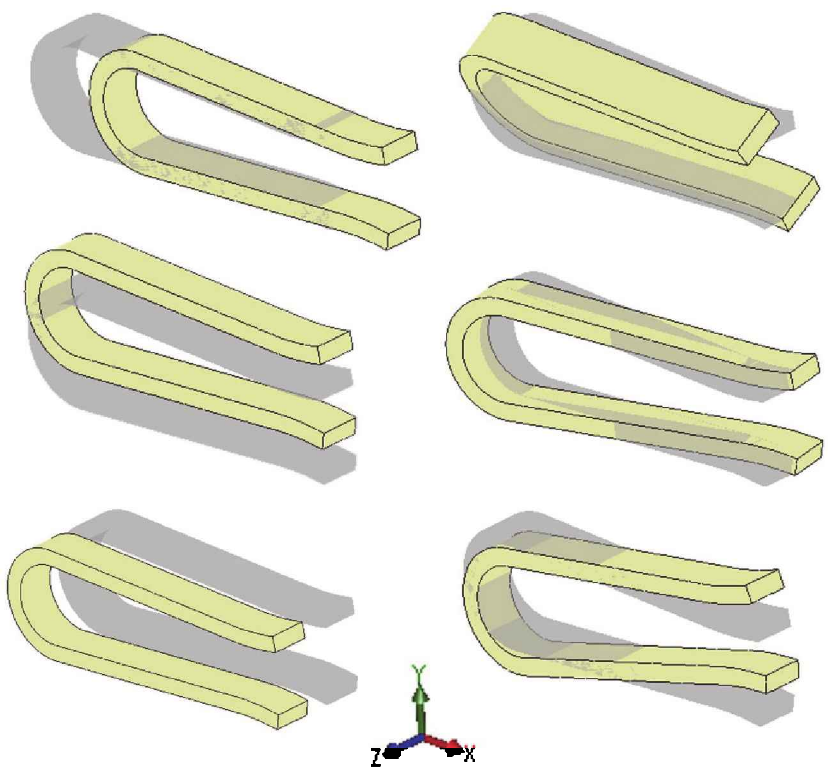


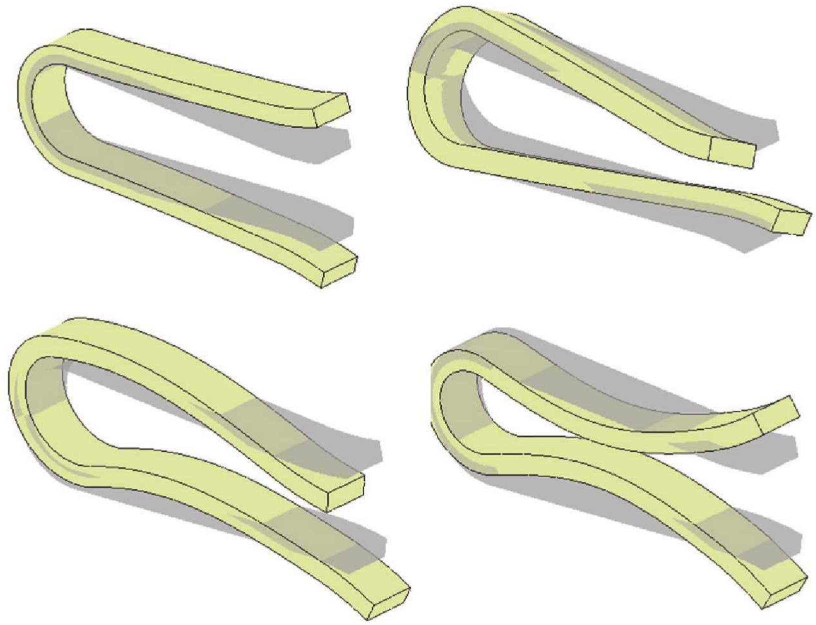
Figure 1-18: Six Rigid Body Modes of the unsupported CLIP model are in this case aligned with the directions of the global coordinate system.

The dark shaded shape shows the original position of the model.

The directions of RBMs happen to be aligned with the global coordinate system in the CLIP model example, but this is not always the case.

Notice that the **Rigid Body Motion** and **Rigid Body Mode** terms may be considered interchangeable. In vibration theory, **Rigid Body Mode** is more frequently used.

Mode 7 and all higher modes are elastic modes of vibration. In most cases we don't count Rigid Body Modes (if they are present at all) and mode numbering starts from the first elastic mode. Figure 1-19 shows the first four elastic modes of vibration which we will simply call the modes of vibration. The frequencies of the first four modes are within the range of 0-2600Hz.



Mode 1
403Hz

Mode 3
1561Hz

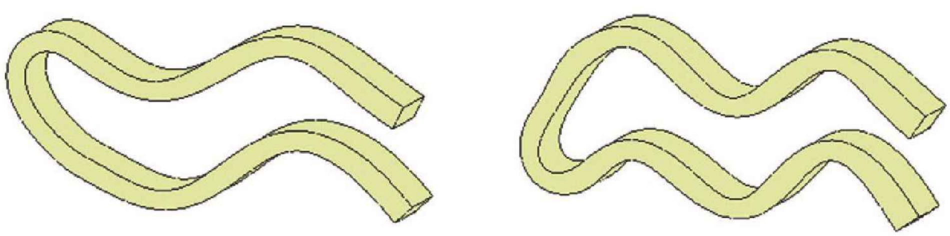
Mode 2
971Hz

Mode 3
2595Hz

Figure 1-19: The first four modes of vibration of the unsupported CLIP model.

The dark shaded shape shows the undeformed model.

How many modes of vibration are of practical importance depends on the expected frequency range of the expected excitation. Most often, higher modes of vibration can't be excited and therefore have no importance in vibration analysis of mechanical systems (Figure 1-20).



Mode 11
9518Hz

Mode 14
17178Hz

Figure 1-20: Higher modes have complex shapes and high frequencies.

The undeformed model is not shown.

Modal Superposition Method

The restrained and meshed with a default element size CLIP model has 39825 degrees of freedom. Solving this vibration problem would require finding a solution to the set of 39825 equations. Considering that vibration problems are time dependent, those equations would have to be solved for all time steps where the number of time steps may be into the thousands. In the view of very high numerical effort required to solve vibration problems directly, we need a way to simplify the formulation of vibration problems. We will describe this process in steps.

Step 1

Let's summarize what we already know about modes of vibration. Each mode is characterized by frequency, shape of deformation, and the mass participating in the vibration. Furthermore, the direction of deformation is associated with certain stiffness. Collecting these facts we may represent an object vibrating in a given mode of vibration by a single degree of freedom oscillator with mass, stiffness and direction that correspond to the properties of the mode under consideration.

Step 2

Now we decide how many modes are important in the vibration response of the system under investigation. For example, if only three modes are important, we can represent the system with three Single Degree of Freedom oscillators (SDOF), each one corresponding to one mode. This way the number of degrees of freedom is reduced to just three, no matter how complex the model is.

Step 3

Instead of studying the original model, we may now study a simple 3DOF system where each mode is represented by a SDOF.

Step 4

We assume that the system is linear and we find its vibration response as a superposition of vibration responses of the three SDOFs.

The principle of the **Modal Superposition Method** is shown schematically in Figure 1-21.

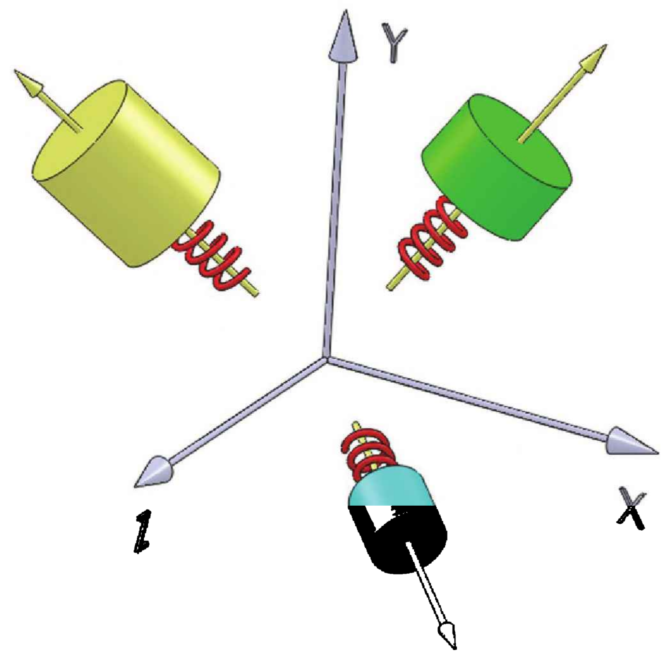


Figure 1-21: A vibration system represented by three single degree of freedom oscillators.

It is assumed that three modes of vibration are sufficient to model the vibration response. Each mode is represented by a SDOF oscillator. Each oscillator is characterized by its mass, stiffness and direction of linear displacement.

The **Modal Superposition Method** is universally used in vibration analysis with FEA. The most important condition that must be satisfied in order to use the **Modal Superposition Method** is that the problem must be **linear**. The second important issue is how many modes should be considered? This is most often decided by considering the frequency range of excitation. A convergence process may also be conducted where the same problem is solved a number of times, each time with more modes calculated to find the sensitivity of the data of interest to the number of modes considered in the **Modal Superposition Method**.

Direct Integration Method

An alternative to the **Modal Superposition Method** is the **Direct Integration Method**. It works with all degrees of freedom of the model. It does not require an assumption of the number of modes to be considered in a vibration response and the problem doesn't have to be linear. The **Direct Integration Method** is available in **SolidWorks Simulation** but its numerical intensity limits practical applications to vibration problems of short time duration.

Vibration Analysis with SolidWorks Motion and SolidWorks Simulation

We have already defined discrete and distributed vibration systems; these definitions are essential to understanding differences between **SolidWorks Simulation** and **SolidWorks Motion** which are add-ins to **SolidWorks**.

Vibration of discrete systems can be analyzed both by **SolidWorks Motion** and **SolidWorks Simulation**. Vibration of distributed systems can be analyzed only with SolidWorks **Simulation.**

SolidWorks Motion is a tool for kinematic and dynamic analyses of rigid bodies. Elasticity can be modeled in **SolidWorks Motion** only as a discrete spring which makes it possible to use it for vibration analysis of discrete systems. Models analyzed by **SolidWorks Motion** are typically mechanisms and have few degrees of freedom associated with rigid body motions of its components. All deformation is limited to springs. Vibration properties of distributed systems cannot be analyzed with **SolidWorks Motion**.

SolidWorks Simulation is a tool of structural analysis of elastic bodies by means of Finite Element Analysis. **SolidWorks Simulation** models everything as elastic bodies with a large number of degrees of freedom and, consequently, with a large number of modes of vibration. It is up to the user to decide how many modes should be used to model a vibration response. Using **SolidWorks Simulation**, a discrete representation of a problem does not offer any advantages besides a more intuitive understanding of the problem and the results. **SolidWorks Simulation** performs discretization and represents every problem as a discrete system with a large but finite number of degrees of freedom.

Figure 1-22 shows the ELLIPTIC TRAMMEL model. This device can be used to trace an ellipse, hence the name. **SolidWorks Motion** treats it as a mechanism. This mechanism has one degree of freedom and consists of two prismatic kinematic pairs and two revolute kinematic pairs. The angular position of the tracer or linear position of either slider fully defines the position of this mechanism. The elliptic trammel may become a vibration system if a torsional spring is added to either hinge. Then, any displacement will be associated with deformation, and the system will satisfy the requirements to be classified as a structure. Notice that the model with the torsional spring added still has one degree of freedom.

If the same model (with or without a spring) is analyzed with **SolidWorks Simulation**, all components are treated as elastic bodies. The number of degrees of freedom will depend on the element size. If the default element size is used, the meshed model will have 14375 nodes and 43125 degrees of freedom.

The model analyzed by **SolidWorks Motion** has one degree of freedom; one piece of information fully defines the position of the mechanism.

The model analyzed by **SolidWorks Simulation** has 43125 degrees of freedom.

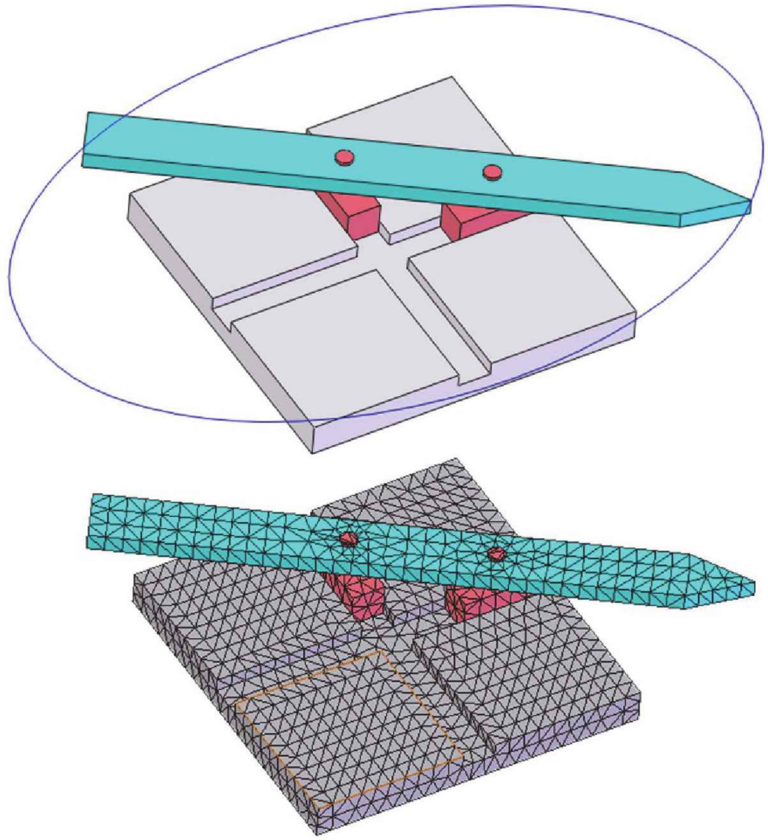


Figure 1-22: Different representations of the *ELLIPTIC TRAMMEL* model by SolidWorks Motion (top) and SolidWorks Simulation (bottom).

The arrowhead traces an ellipse (top). SolidWorks Simulation is not intended for mechanism analysis; hence the ellipse is not shown in the lower image.

SolidWorks Motion works only with assemblies. Assembly components are modeled as solid bodies and are treated as undeformable (rigid bodies).

SolidWorks Simulation works with both parts or assemblies and treats everything as elastic bodies.

Some problems analyzed with **SolidWorks Motion** may give the appearance of analysis of elastic bodies. This applies to all collision problems where elastic properties must be added in contact definitions as is done in the WRECKING BALL model (Figure 1-23).

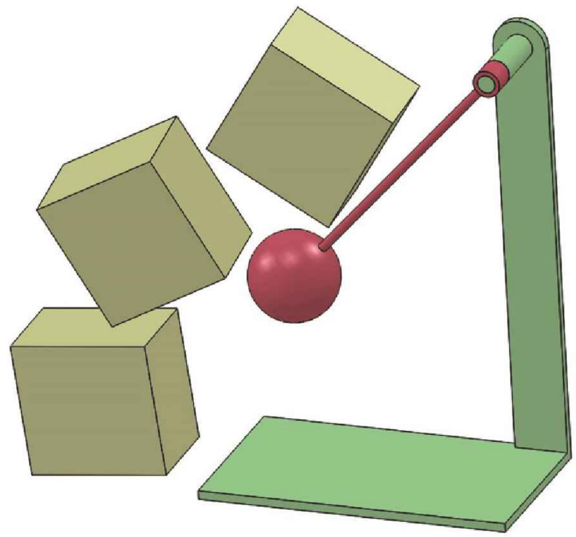


Figure 1-23: WRECKING BALL assembly analyzed with SolidWorks Motion.

A stack of boxes is demolished by a wrecking ball.

As a result of the impact, boxes "fly apart" but deformation is not present anywhere in the model. Elastic properties of contact are assumed in SolidWorks Motion. They must be entered by a user as a part of the model definition or some default number may be used.

Capabilities of **SolidWorks Simulation** and **SolidWorks Motion** with regard to modeling discrete and distributed systems are summarized in Figure 1-24.

	Discrete systems	Distributed systems
SolidWorks Simulation	Yes	Yes
SolidWorks Motion	Yes	No

Figure 1-24: Summary of capabilities of SolidWorks Simulation and SolidWorks Motion with regard to analyzing discrete and distributed systems.

SolidWorks Motion is intended to work with rigid bodies; SolidWorks Simulation is intended to work with elastic bodies.

When a discrete system is analyzed in **SolidWorks Simulation**, it is the user's responsibility to consider only those modes of vibration that are associated with the deformation of the discrete springs and not with the deformation of the assembly components.

SolidWorks Motion and **SolidWorks Simulation**, which are both **SolidWorks** add-ins, serve different purposes and employ different modeling techniques. In **SolidWorks Motion**, assembly mates are translated into kinematic pairs and assembly components become rigid links of a mechanism. Assembly mates that fully define an assembly must be suppressed in order to allow the motion of a mechanism. In **SolidWorks Simulation**, assembly mates are ignored and restraints must be defined independently of the mates in the CAD model. Assembly components with touching faces are bonded unless specific contact conditions are defined.

Functionality of SolidWorks Motion and SolidWorks Simulation

As a **SolidWorks** add-on, **Simulation** and **Motion** are connected to **SolidWorks** CAD as well as between themselves. Connectivity and analysis capabilities are shown in Figure 1-25.

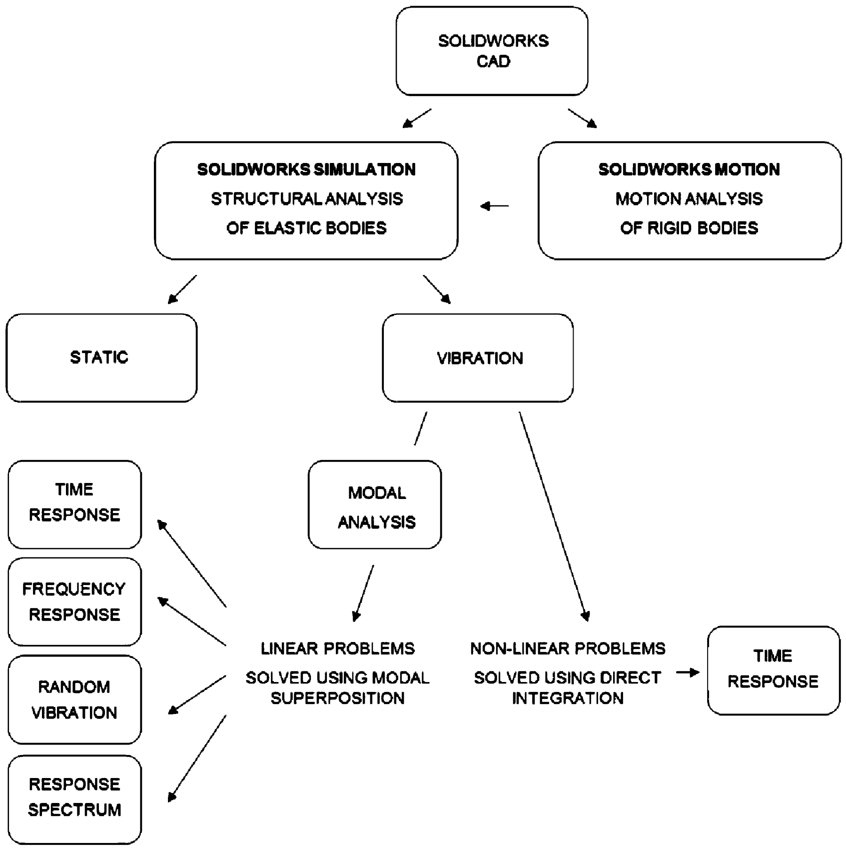


Figure 1-25: Types of analyses and connectivity available in SolidWorks Motion and SolidWorks Simulation.

Results of SolidWorks Motion may be transferred to SolidWorks Simulation.

SolidWorks Motion is not a tool intended for vibration analysis and will not be used in this book. Most vibration problems in this book will be solved with **SolidWorks Simulation** using the **Modal Superposition Method**. These will be problems of **Time Response**, **Frequency Response**, **Random Vibration** and **Response Spectrum**. Nonlinear vibration problems will be solved using the **Direct Integration Method**. We will provide in-depth descriptions of these types of analyses later, but now we have to address some important terminology issues.

Terminology issues

Terms used in **SolidWorks Simulation** differ in many ways from terms you find in a standard textbook on vibration analysis. A summary of important differences along with short descriptions are given in Figure 1-26.

Textbook terminology	**SolidWorks Simulation study**	**Description**
Linear Vibration Analysis	Linear Dynamic	All types of linear vibration analyses based on the Modal Superposition Method
Nonlinear Vibration Analysis	Nonlinear with Dynamic Option	Nonlinear vibration analysis. Excitation is a function of time. Results are given as a function of time
Modal	Frequency	Finds modes of vibration (frequency and shape)
Time Response	Modal Time History (for linear analysis) Nonlinear with Dynamic Option (for nonlinear analysis)	Excitation is a function of time. Results are given as a function of time.
Frequency Response	Harmonic	Excitation is harmonic and is defined as function of frequency. Results are given as a function of frequency.
Random Vibration	Random	Excitation and results are in the form of Power Spectral Density.
Response Spectrum	Response Spectrum Analysis	Base excitation in the form of a Response Spectrum.
-	Drop test	Nonlinear vibration analysis intended for drop test simulation.

Figure 1-26: Summary of capabilities of SolidWorks Simulation and SolidWorks Motion.

Different types of vibration analysis will be described in in the chapters that follow.

We will be alternating between standard terminology and **SolidWorks Simulation** specific terminology.

Definition of linear and nonlinear vibration studies in **Simulation** is shown in Figure 1-27.

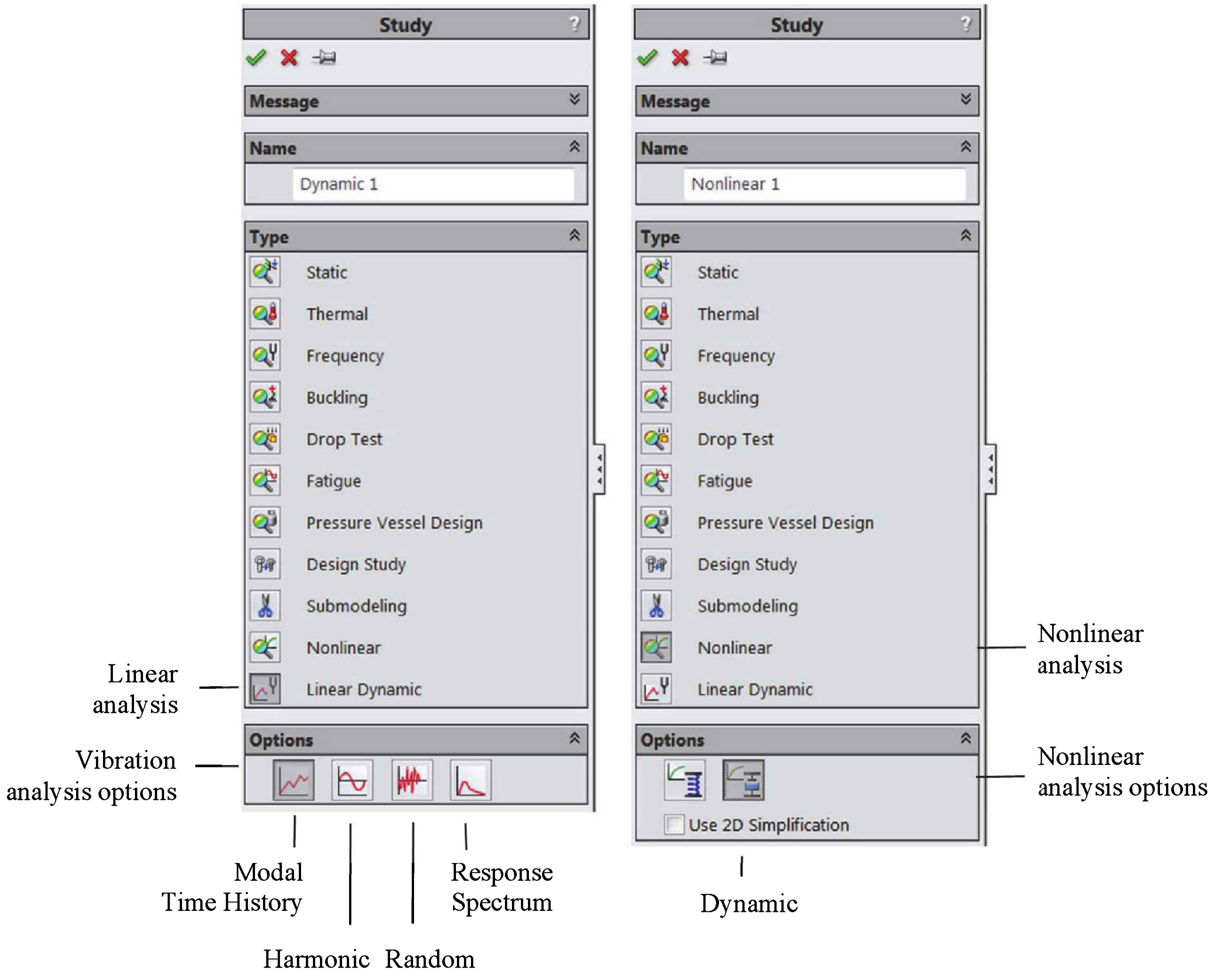


Figure 1-27: Definition of linear and nonlinear vibration analysis studies in Simulation.

Different types of linear vibration studies are defined by selecting analysis option. Nonlinear vibration study is defined as Nonlinear with Dynamic option.

2: Introduction to modal analysis

Topics covered

- Modal analysis
- Properties of a mode of vibration
- Interpreting results of modal analysis
- Normalizing displacement results in modal analysis

Modal analysis

Modal analysis, called a **Frequency** analysis in **Simulation**, finds natural frequencies and shapes of vibration that are associated with natural frequencies. **Modal** analysis assumes that the body vibrates in the absence of any excitation and damping. The vibration textbooks call this free undamped vibration.

Consider the simplest type of vibrating system: a **Single Degree of Freedom** oscillator (SDOF) with mass *m* and stiffness *k* which is vibrating in the absence of damping and excitation (Figure 2-1).

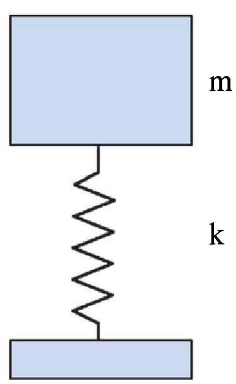


Figure 2-1: SDOF oscillator vibrating in the absence of damping and excitation. The problem is linear; mass m and stiffness k do not change during motion.

Vibration has been caused by an initial condition such as displacement and/or velocity.

The equation of motion of this SDOF can be derived by noticing that inertial forces equal stiffness force at any time (1). The displacement solution has the form of (2) where ω is the circular frequency of oscillations in radians per second (rad/s) and A is the amplitude of displacement. It is easy to prove that the equation of motion of the SDOF is satisfied for any A but only for one value of ω (3).

$$m\ddot{x} + kx = 0 \qquad (1)$$

$$x = A\sin(\omega t) \qquad (2)$$

$$\dot{x} = A\omega\cos(wt)$$

$$\ddot{x} = -A\omega^2\sin(\omega t)$$

$$m(-A\omega^2\sin(\omega t)) + k(A\sin(\omega t)) = 0$$

$$-mA\omega^2 + kA = 0$$

$$\omega = \sqrt{\frac{k}{m}} \qquad (3)$$

Frequency (3) is the natural frequency of the SDOF; this is the only frequency with which the SDOF can oscillate in the absence of excitation.

Review equation (1) and its solution (3) to notice a very important property of the natural frequency found here for an SDOF system but extendable to any vibrating system. When a system is vibrating with its natural frequency, there is a cancellation between inertial and stiffness forces. The resultant stiffness of an undamped system performing free, undamped vibration is therefore equal **zero**. This happens for any amplitude of vibration.

Interpreting results of modal analysis

Modal analysis can only find the natural frequencies and the associated shapes (modes) of the analyzed structure performing free, undamped vibration. Since neither excitation nor damping is modeled in modal analysis, the modal analysis cannot provide any quantitative information on displacements and stresses. Displacement results can only be used to compare relative displacements between different portions of the analyzed structure and only within the same mode of vibration. We'll demonstrate this later when analyzing multi-degree of freedom systems.

In **SolidWorks Simulation** displacement results of modal analysis are normalized to make the generalized mass matrix equal one (4).

$$\left|\varphi^T\right|\left|M\right|\left|\varphi\right| = \left|1\right| \qquad (4)$$

φ eigenvector (shape of vibration)

φ^T transposed eigenvector

M mass matrix

1 unit matrix

The effect of normalization can be demonstrated by running modal analysis on a model with the same geometry and modulus of elasticity but different material densities. This will be done using the BALL VALVE PLATE model. The material in the model is Cast Carbon Steel: restraints are shown in Figure 2-1. The geometry is simplified by omitting fillets.

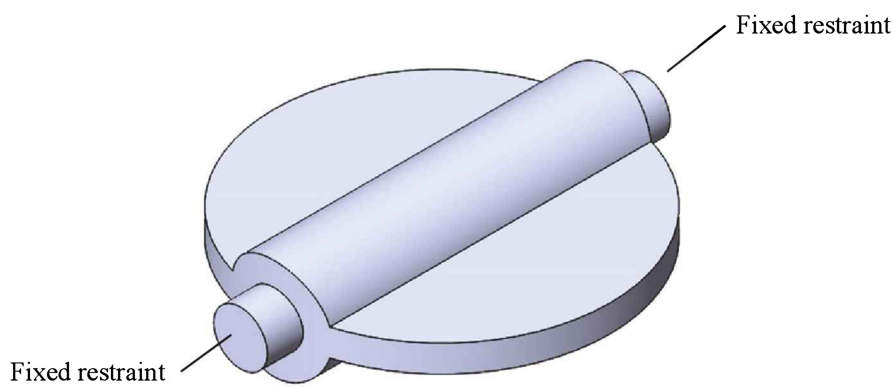


Figure 2-2: BALL VALVE PLATE

Fixed restraints are applied to both end faces of the shaft.

BALL VALVE PLATE features a simplified geometry where fillets have been removed. The sharp re-entrant edges would cause stress singularities if the model were used for stress analysis. If removal of fillets does not change the model stiffness significantly, this simplified geometry may be acceptable for modal analysis.

The finite element mesh used in modal analysis should model stiffness correctly. If the model is not intended for subsequent stress analysis, geometry details that do not affect stiffness globally do not have to be modeled precisely since this will not affect the results of modal analysis.

Create a **Frequency** study titled *01 Cast Carbon Steel* as shown in Figure 2-3.

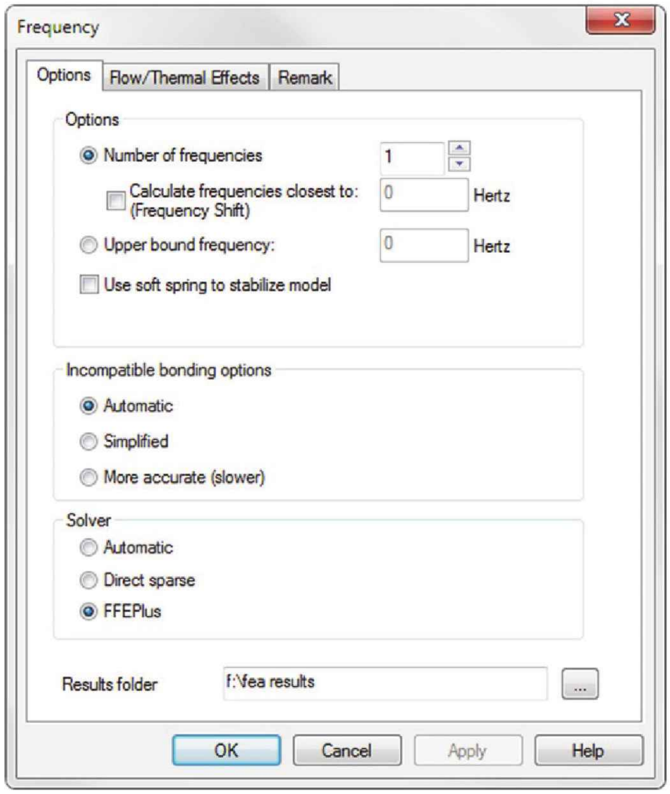


Figure 2-3: Properties of frequency study *01 Cast Carbon Steel.*

Only one mode of vibration will be found. One mode will suffice to demonstrate the effect of modal mass normalization.

Mesh the model using a **Standard Mesh** with second order solid elements of default size and define restraints as shown in Figure 2-2.

Displacement results of mode 1 are shown is Figure 2-4.

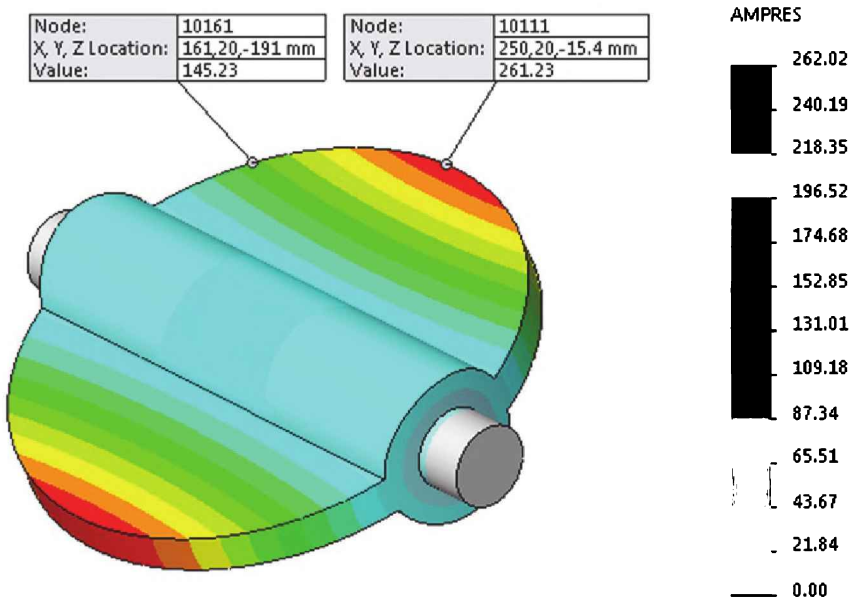


Figure 2-4: Resultant displacement results of mode 1.

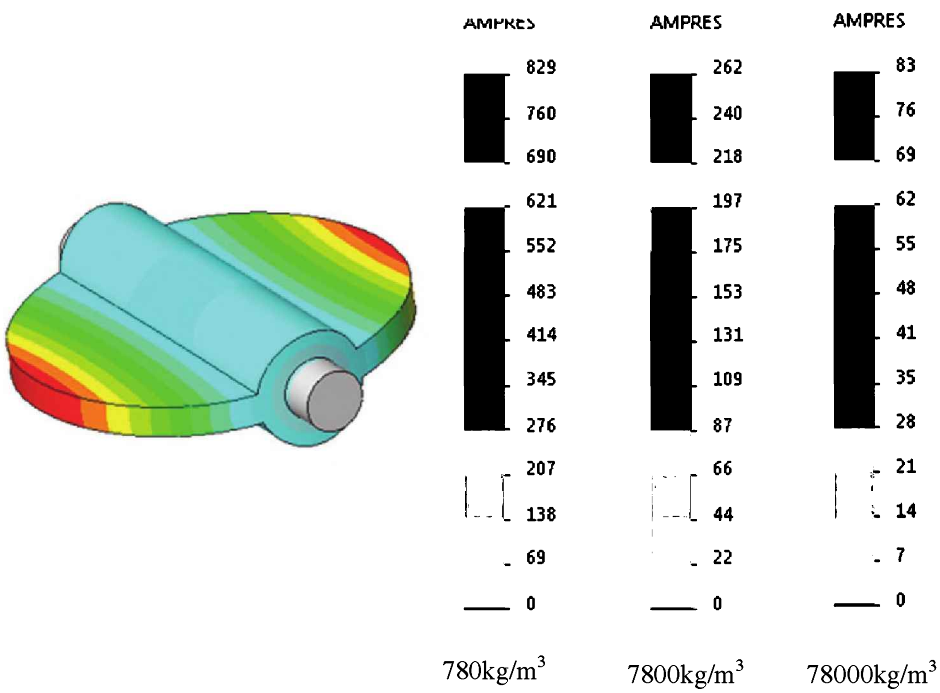
Color fringes along with a color legend are shown in the displacement results of modal analysis, but their numerical values are relatively valid only. For example, the ratio between resultant displacements in the probed locations is 261/145=1.80. Notice that probing disables the deformed shape.

Displacement results of modal analyses do not show any units to avoid a misinterpretation of results.

Numerical values of displacement results are normalized to make the modal mass equal to 1. Therefore, the displacement results will be different for the same model geometry and stiffness but different mass. To demonstrate this, we modify the original density of 7800kg/m^3 and repeat the analysis with two custom materials:

1. Custom material 1: modulus of elasticity equal to that of Cast Carbon steel; mass density 780kg/m^3
2. Custom material 2: modulus of elasticity equal to that of Cast Carbon steel; mass density of 78000kg/m^3

Copy the frequency study *01 Cast Carbon Steel* into two new studies titled *02 10% Cast Carbon Steel* and *03 1000% Cast Carbon Steel*. Assign the corresponding custom materials as explained above. Obtain the results and summarize the displacements as shown in Figure 2-5.



<u>Figure 2-5: Displacement results in mode 1 for different materials.</u>

Color legends are shown in the ascending order or mass density. The modal shape is the same for all material densities.

To gain a better insight on how displacement results are normalized in Modal analysis, we'll summarize the natural frequencies and the maximum displacement magnitudes for the different material densities listed in Figure 2-5. The summary is shown in Figure 2-6.

	Material density kg/m^3	Frequency of the first mode	Normalized maximum displacement
1	780	1353Hz	829
2	7800	428Hz	262
3	78000	135Hz	83

Figure 2-6: Summary of frequency and normalized displacement results for materials of different density.

Material density is modified by using a custom material.

When you review the results in Figure 2-6, remember that while material density changes, the model stiffness remains the same. Even though this exercise has no relevance to real life, it still provides interesting results. As we know, a structure vibrating in a given mode of vibration may be treated as a **Single Degree of Freedom** oscillator (SDOF) with a natural frequency expressed by (2): $\omega = \sqrt{\frac{k}{m}}$

In our case stiffness remains the same; therefore the natural frequency is inversely proportional to the square root of mass. This is why a 100 fold increase in the mass causes the frequency to drop to one tenth of the original frequency; compare frequency results in rows 1 and 3 in Figure 2-6.

Modal analysis normalizes the displacement results in such a way that the ratio of maximum displacements equals the inverse of the square root of the ratio of mass; compare displacement results in rows 1 and 3 in Figure 2-6.

Displacement results of modal analysis are so often misinterpreted that it may be better to show the results of modal analysis without colors, making it a deformation plot.

Deselect Show colors

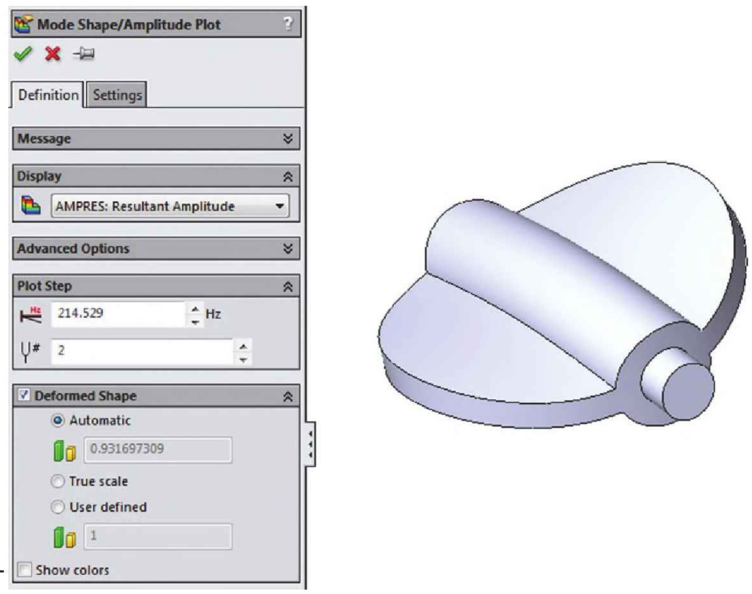


Figure 2-7: Deformation results in mode 1.

Deselecting colors eliminates color fringes and hides the color legend. The deformation plot is less "prone to misinterpretations". This plot shows the shape of the second mode.

3: Modal analysis of distributed systems

Topics covered

- Modal analysis of distributed systems
- Meshing considerations in modal analysis
- Importance of mesh quality in modal analysis
- Importance of modeling supports
- Interpretation of results of modal analysis

The mode of vibration is a combination of frequency and shape; it depends on the stiffness and inertial properties of the analyzed structure. If finding modes of vibration is the only objective, geometry details such as notches or rounds don't have to be included in the analysis because they don't change the stiffness or inertia in a significant way. If the results of a modal analysis are used as pre-requisites to subsequent stress analyses, then the mesh has to comply with all requirements of the stress analyses.

To demonstrate the different requirements of the mesh in a stress and modal analysis, we will conduct both analyses on a simple model. Open part model NOTCH, make sure it is in configuration *02 notch* and observe the small round notch (Figure 3-1).

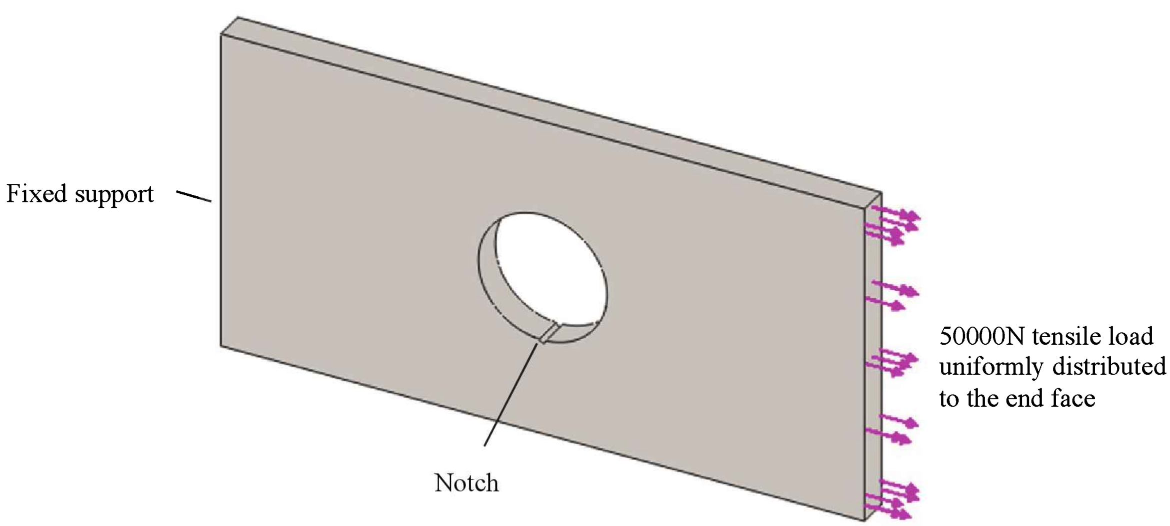


Figure 3-1: A notched hollow plate subjected to a tensile load and fixed support.

Support symbols are not shown.

A stress analysis of the NOTCH model requires careful preparation of the mesh. Create a **Static** study titled *01 stress* and define a mesh control on the notch face to assure correct element size and turn angle (Figure 3-2).

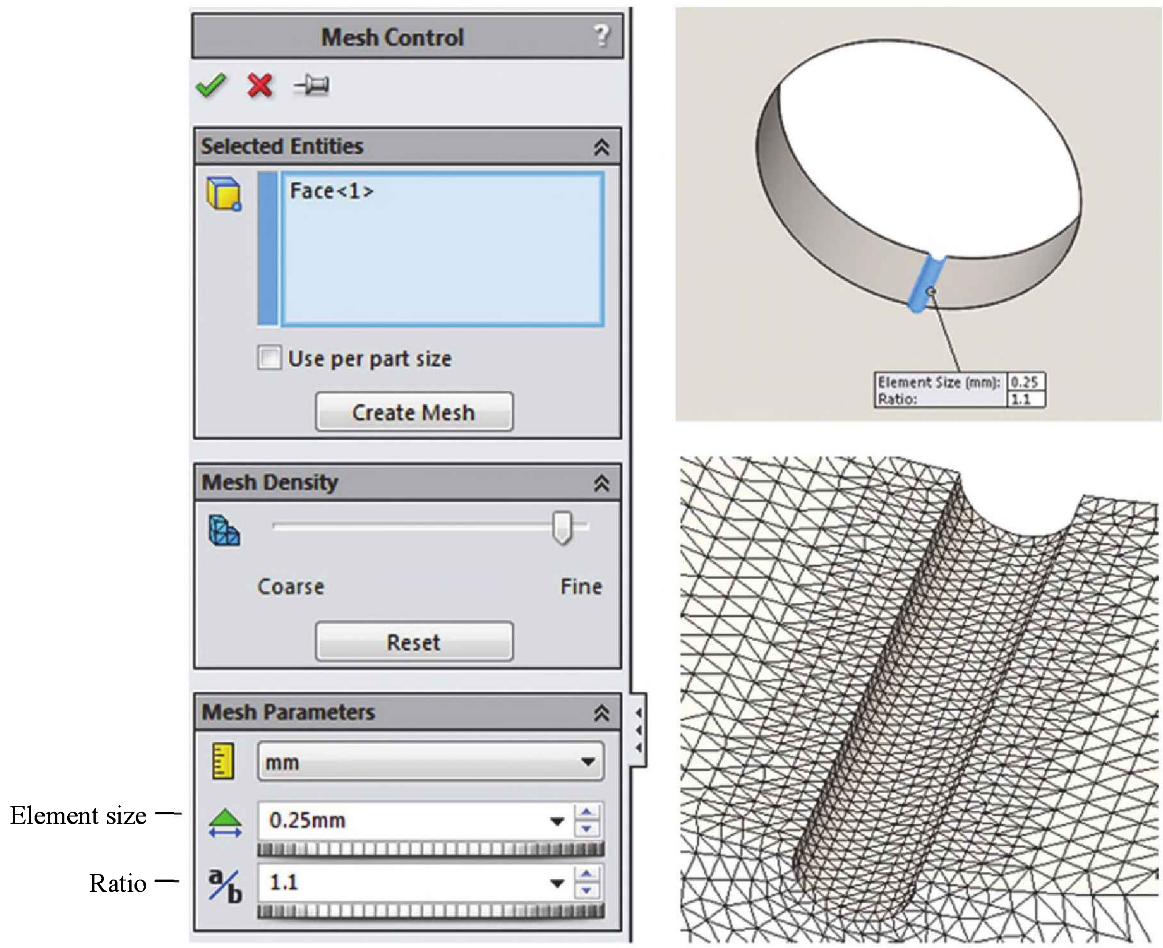
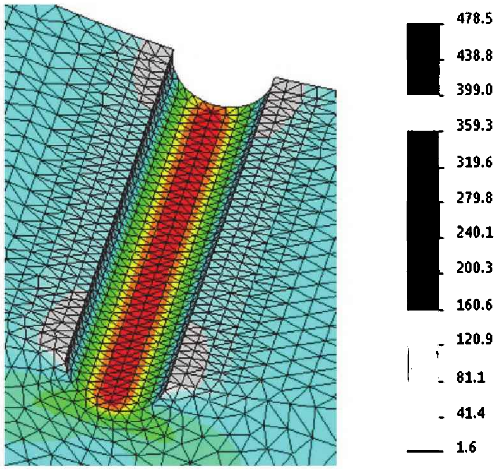
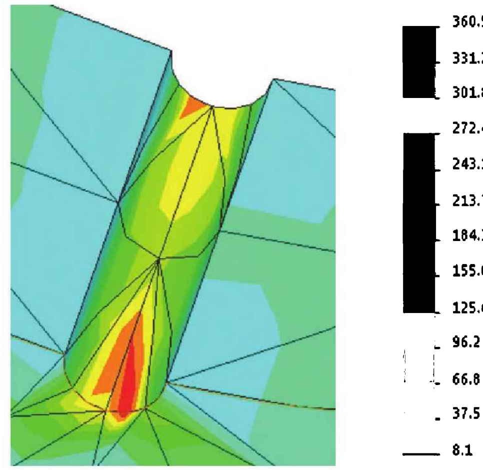


Figure 3-2: Mesh control defined on the face of the notch.

Element size on the controlled entity is 0.25mm; this is controlled independently of the global element size. Ratio defines the relative size of elements in consecutive layers in the transition zone between element size 0.25mm and global element size 5.27mm.

Define loads and restraints as shown ion Figure 3-1 and obtain the solution. Next, copy study *01 stress* into a new study titled *02 stress*. Delete the mesh controls and obtain the solution with the default mesh. Von Mises stress results of both studies are shown in Figure 3-3.





Study *01 stress*
Correct stress results produced by mesh with controls.
Maximum stress 479MPa

Study *02 stress*
Incorrect stress results produced by default mesh.
Maximum stress 361MPa

Figure 3-3: Von Mises stress results produced with the correct mesh (left) and incorrect mesh (right).

Using the default element size and no mesh controls produces highly distorted elements that give incorrect stress results.

Compare the stress plots in Figure 3-3 and notice that the incorrect mesh returns an incorrect stress distribution pattern as well as von Mises stresses that are 25% lower as compared to the correct mesh. This quick review of stress analysis shows the importance of meshing small geometry details when stress results are sought.

We will now study the effects of the mesh on the results of modal analyses. Create two **Frequency** studies titled *03 modal* and *04 modal*. Define the same restraint as shown in Figure 3-1; don't define any load. In study *03 modal* use a mesh with the same controls as in study *01 stress*. In study *04 modal* don't use any mesh controls.

A summary of results of studies *03 modal* and *04 modal* is shown in Figure 3-4. The same figure also shows modal frequencies of a model without a notch. These results may be obtained by analyzing the model in configuration *01 no notch* using a default mesh.

Mode number	Frequency [Hz]		
	Study *03 modal* Model with notch Mesh controls	Study *04 modal* Model with notch No mesh controls	Study *05 modal* Model without notch No mesh controls
1	210.2	210.21	210.25
2	845.06	845.48	845.56
3	1274.4	1274.7	1275.3
4	1730.1	1730.1	1730.2
5	2825.7	2825.9	2826.1

Figure 3-4: Comparison of modal frequencies produced by the model with a notch and mesh controls, with a notch and no mesh controls, and without a notch.

Incorrect mesh or removal of the notch has next to no effect on natural frequencies.

The results summarized in Figure 3-4 show that incorrect meshing of small details or removing those details has no effect on the natural frequency. Animate any mode to see that the modal shapes also remain the same.

Stresses may be strongly dependent on small **local** features which must be correctly represented in a mesh intended for stress analysis. Modes of vibration depend on structural stiffness which is a **global** model property. Incorrect meshing or removal of small features does not change stiffness in a significant way. This is the reason that modal analysis is insensitive to incorrect meshing of small features or to the removal of these small features all together.

The above statement applies only if modal analysis is the only objective. If results of modal analysis are used as pre-requisites to subsequent vibration stress analyses, then all rigors of correct meshing apply to the modal analysis.

Stay with the *01 no notch* configuration open to perform a convergence analysis of natural frequencies. Copy study *05 modal* into *06 modal no notch coarse* and mesh the model with a very coarse mesh using 25mm default element size. Then copy *05 modal no notch* into *07 modal no notch fine* and mesh the model with a very fine mesh using 2.5mm default element size. Frequency results in the first mode are shown in Figure 3-5.

Study *06 modal no notch coarse*
Element size 25mm
Frequency 213.2Hz

Study *05 modal no notch*
Element size 5.7mm
Frequency 210.3Hz

Study *07 modal no notch fine*
Element size 2.5mm
Frequency 210.1Hz

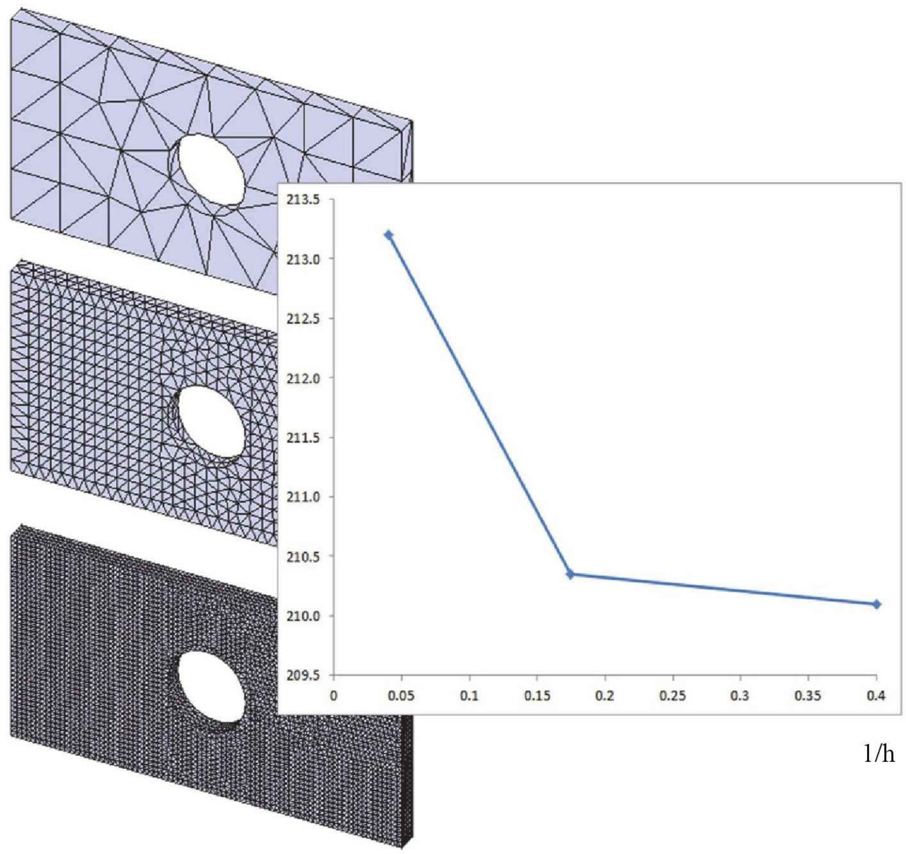


Figure 3-5: Frequency of the first mode as a function of 1/h, where h is the element size.

Frequency drops with element size and converges to the asymptotic value corresponding to a continuous model.

As Figure 3-5 indicates, the natural frequency converges to an asymptotic value. Remember that meshing adds stiffness to the model. The coarse mesh has more "artificial stiffness". With mesh refinement, as the artificial stiffness is gradually removed, the model becomes softer and, consequently, the natural frequency drops. In the limit, frequency tends to the asymptotic value that corresponds to a continuous model.

We have demonstrated this only for the first mode, but the same applies to all modes.

We'll return to the topic of artificial stiffness and its effect on modes of vibration in Chapter 4 discussing similarities and differences between modal and buckling analyses.

4: Modal analysis – the effect of pre-stress

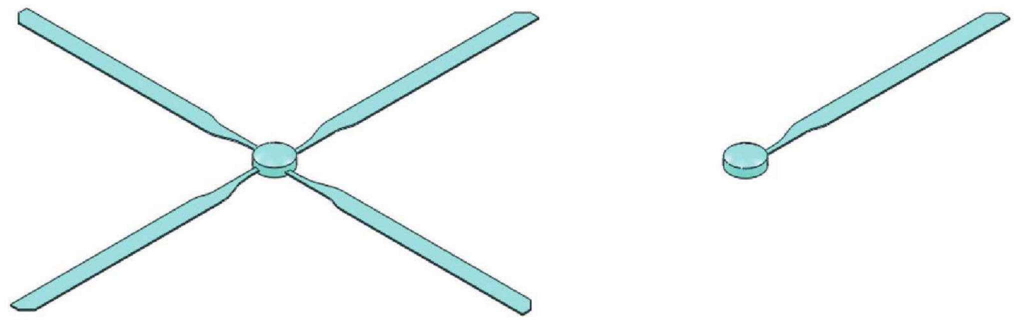
Topics covered

- Modal analysis with pre-stress
- Modal analysis and buckling analysis
- Artificial stiffness

If a structure is subjected to a load that produces significant tensile or compressive stresses, those stresses may significantly change the structure's stiffness and consequently, its modes of vibration. To account for that change, modal analysis needs to be completed on a structure with a modified stiffness caused by the existing state of stress. This type of modal analysis, called pre-load or pre-stress modal analysis is conducted in two steps. First, a static analysis is run to find stresses that develop due to a load; these stresses are used to modify the structure's stiffness. Next, modal analysis is run on the structure with the stiffness modified by the previously found stresses.

Predominantly, tensile stresses will increase the natural frequencies as is illustrated by tuning a guitar string or stress-stiffening of a rotating component like a turbine blade. Rotating machinery typically requires that the effect of pre-stress be considered. Compressive stresses on the other hand will decrease natural frequencies. In this chapter we will illustrate the effect of tensile and compressive stresses on natural frequencies. Then we will demonstrate how a modal analysis with pre-stress relates to a buckling analysis.

Open part model ROTOR which is a simplified representation of a helicopter rotor. We are interested in the effect of centrifugal forces that develop due to rotor rotation on the fundamental natural frequency of the rotor blade. The fundamental frequency is the lowest natural frequency. All four blades are identical and isolated from each other by a support at the hub. Therefore, we may simplify the analysis to only one blade (Figure 4-1).



Rotor with four blades One blade for analysis

Figure 4-1: ROTOR model

Modal analysis can be conducted one blade.

Switch to *02 one blade* configuration and create a **Frequency** study *01 rotating*. Apply **Fixed** restraints as shown in Figure 4-2.

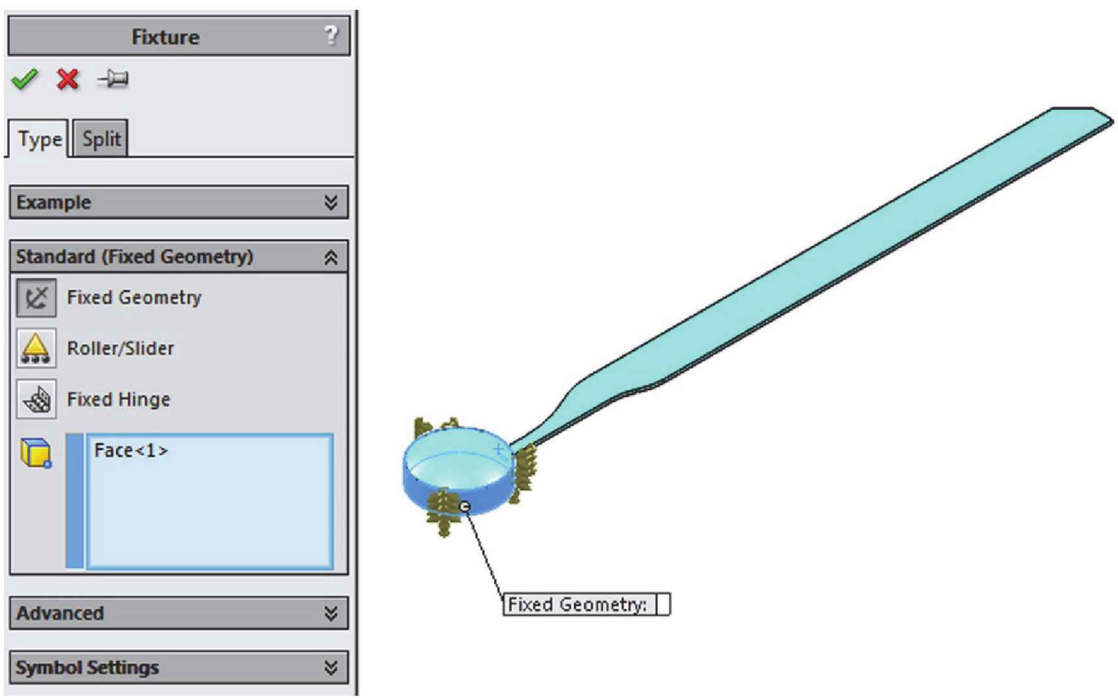


Figure 4-2: Restraints definition

A fixed restraint is defined to the cylindrical surface. You may also apply it to the top and bottom face of the hub; it won't have any significant effect on the vibration of the blade.

Define a centrifugal load of 300RPM by selecting the cylindrical surface where the **Fixed** restraint has been defined. This cylindrical face uniquely defines the axis which is taken as the axis of rotation as shown in Figure 4-3.

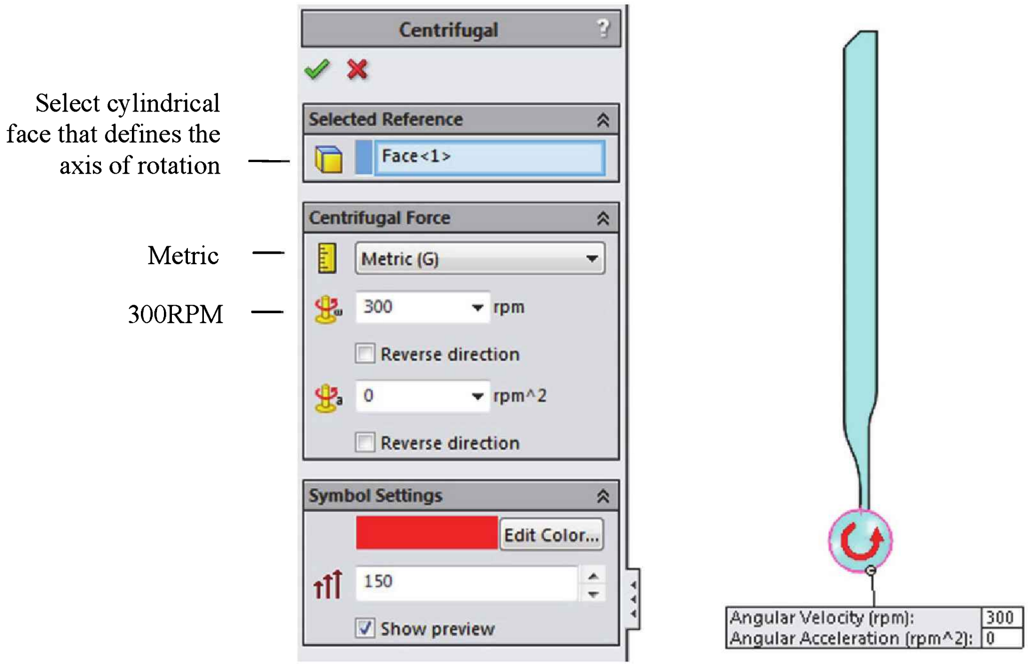


Figure 4-3: Centrifugal load definition

A centrifugal load is defined by an angular rotation about the axis. Use metric units to define it in revolutions per minute (RPM). Do not define any angular acceleration.

In the study properties, specify one mode and use the **Direct sparse** solver (Figure 4-4).

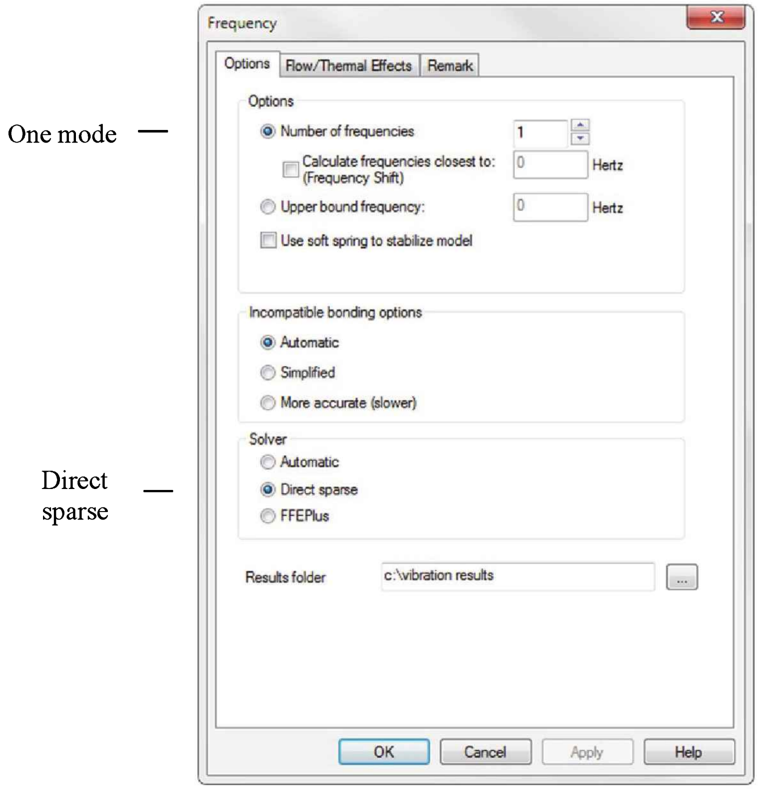
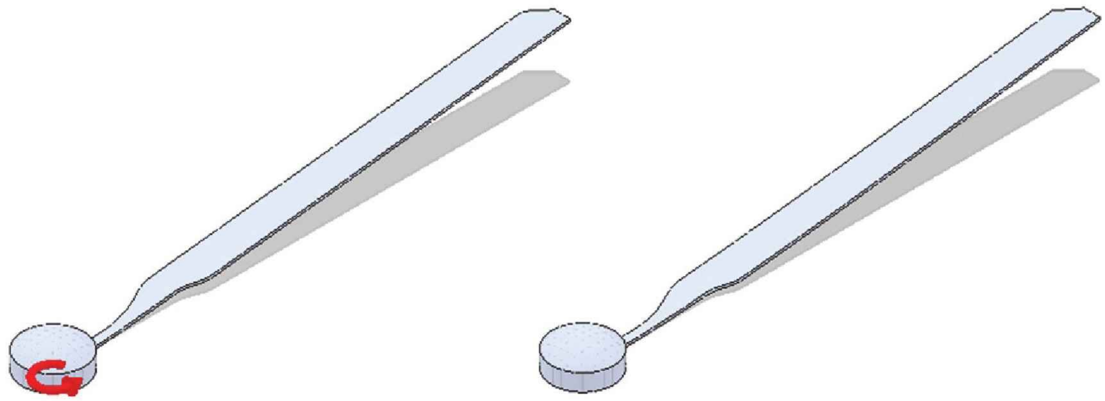


Figure 4-4: Properties of the frequency study

FFEPlus solver is not available for a modal analysis with pre-stress.

Mesh the model with the default mesh size and solve the study. Copy study *01 rotating* into *02 stopped*, suppress the centrifugal load and solve the study without pre-stress.



Fundamental frequency of rotating blade: 5.35Hz

Fundamental frequency of stopped blade: 0.54Hz

Figure 4-5: The first mode of vibration of rotating and stopped blade.

The mode shape is the same for the rotating and stopped blade; the modal frequency is very different. Both plots show the undeformed shape superimposed on the modal plot.

A comparison of results between rotating and stopped blade shows a very strong effect of rotation; the first natural frequency is ten times as high for the rotating blade as it is for the stopped blade. The centrifugal force produces tensile stresses which cause stress stiffening of the blade. The resultant stiffness of the rotating blade is the sum of the elastic stiffness and stress stiffness. Stress stiffness due to tensile stress is positive, and the resultant stiffness of the rotating blade is higher than the stiffness of the stationary blade. Higher stiffness, in turn, produces a higher natural frequency.

When a load is present in a **Frequency** study, this load is used to modify the stiffness. The analysis is then called modal analysis with pre-load. The term modal analysis with pre-stress is also used. The solution progresses in two steps. First, a static analysis is run to find stresses. These stresses are used to modify the model stiffness. Then modal analysis is run using the combined elastic and stress stiffness.

Remember that it is a serious error to confuse pre-load with excitation load!

The effect of tensile stress on the natural frequency may be easily demonstrated by tuning a guitar string, where increasing string tension increases the natural frequency.

The effect of compressive stresses on the natural frequency is the opposite; compressive stresses produce negative stress stiffness which decreases the resultant stiffness and the natural frequency decreases. We will demonstrate this with the COLUMN model subjected to compression by two wedges as shown in Figure 4-6. This illustration schematically shows an apparatus used to demonstrate buckling of a rectangular column.

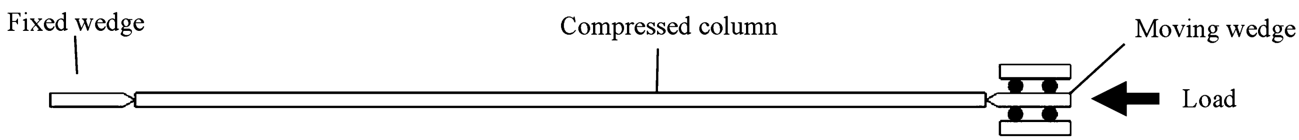


Figure 4-6: A prismatic column compressed by two wedges.

The left wedge is fixed. The right wedge is held in between rollers. It can only translate in the direction of the load.

The beam is "squeezed" between to wedges; one wedge is fixed and provides support, the other one is guided by rollers and can move only in the direction of the load. We want to simulate this test rig to find a relation between the applied load and the first natural frequency of the loaded column in the range of the load magnitude changing from a tensile load to a compressive load up to the point of buckling. Using the test rig as shown in Figure 4-6 we would not be able to apply a tensile load, but using **SolidWorks Simulation**, this is very easy. The analysis does not require an analysis of an assembly; it can be completed on one part.

For your information only, review the COLUMN assembly model, then open the COLUMN part model which will be used for analysis.

To find the magnitude of the buckling load we need to run a buckling analysis. Using the COLUMN part model, create a **Buckling** study titled *00 buckling* and define restraints as shown in Figure 4-7.

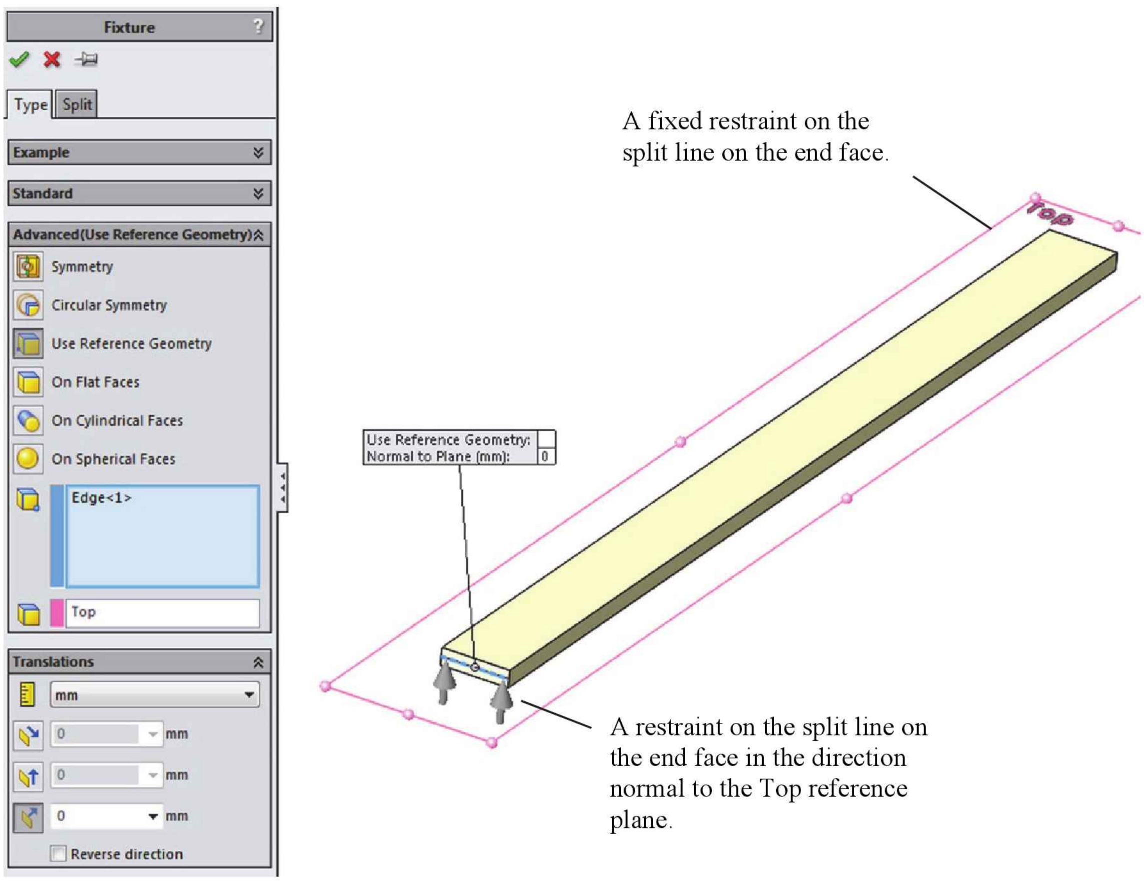


Figure 4-7: Restraints applied to the *COLUMN* model.

The fixture window shows restraint on the side of the moving wedge. Restraint on the side of fixed wedge is a fixed restraint. The fixed restraint window is not shown.

Notice that the **Fixed** restraint, as defined on the split line on the side of the fixed wedge will be transferred to nodes of solid elements which have three degrees of freedom. Therefore, this **Fixed** restraint will effectively produce a hinge support.

Next, define a load as shown in Figure 4-8.

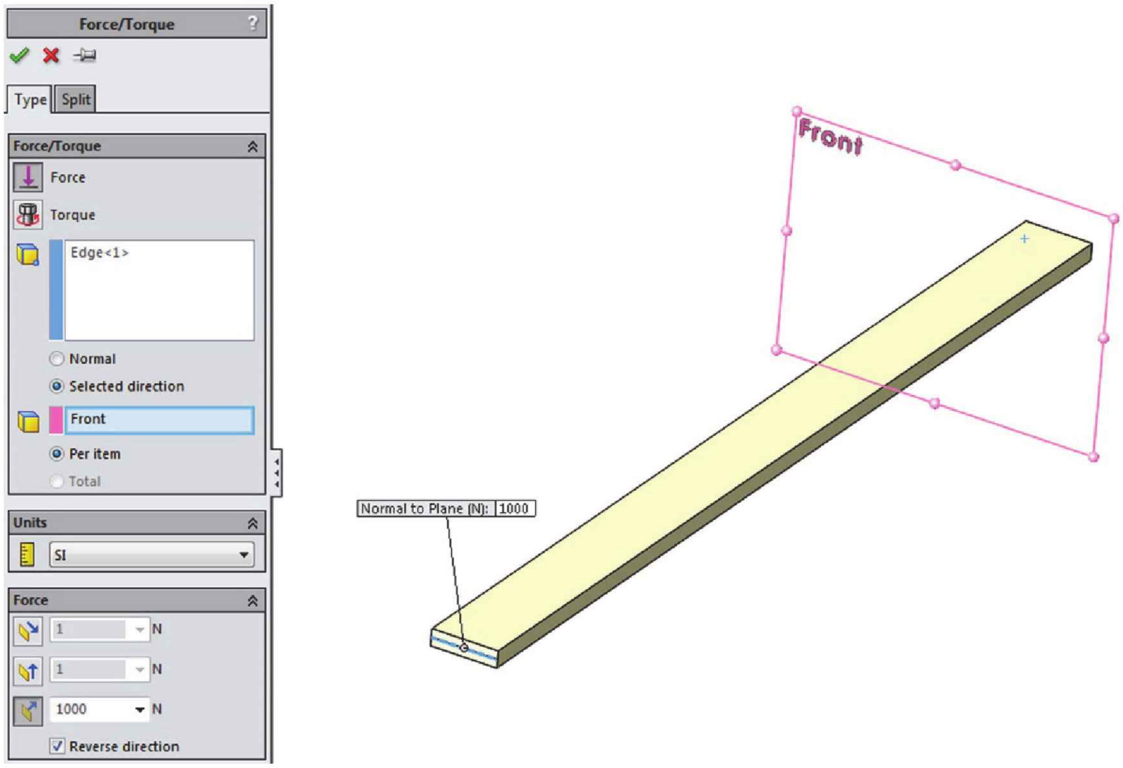


Figure 4-8: Load applied to the *COLUMN* model.

The load is applied to the same split line where the restraint on the side of moving wedge has been applied.

Notice that on the loaded end, the restraint and load are both applied to the same entity (the split line), but in different directions. If the load was applied in the direction of this restraint it would be ineffective.

Mesh the model with the default element size and run the solution of the buckling study. The first buckling mode is shown in Figure 4-9 and the **Buckling Load Factor** (BLF) is 1.5755, meaning that according to the linear buckling model, buckling will happen at a load of 1575.5N.

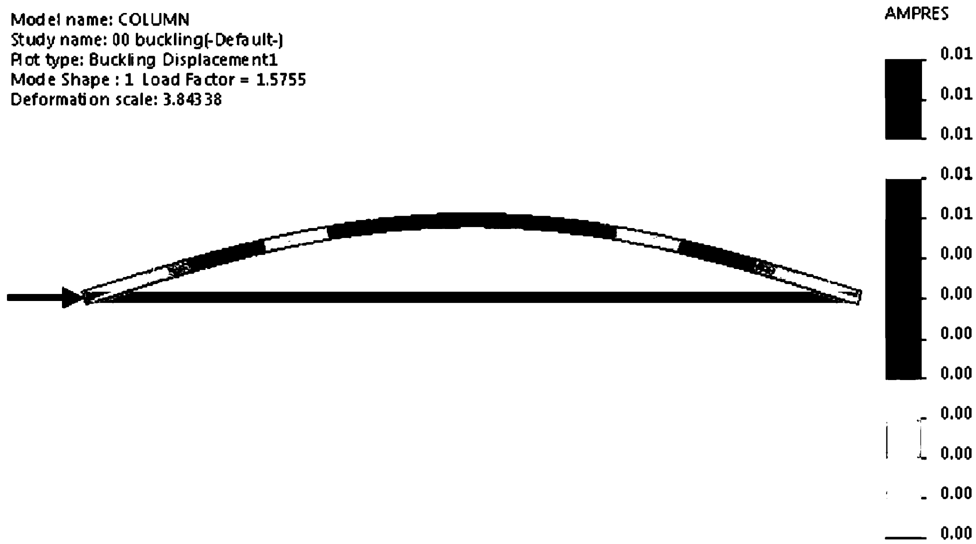


Figure 4-9: Buckled shape shown using displacement plot.

Even though numerical values are shown, their values may be used only to find a displacement ratio rather than absolute displacement. This is in close analogy to modal analysis.

The linear buckling analysis was necessary to establish the range of loads in the modal analysis with pre-stress. We'll now conduct a numerical experiment subjecting the model to different loads ranging from tensile to compressive to study the effect of pre-load on the fundamental natural frequency. The experiment will be conducted in twelve **Simulation** studies while the load is changed from a 1500N tensile load, to a compressive load causing buckling. A tensile load is denoted as positive, compressive as negative.

We should point out that numerical results presented in this experiment may differ slightly depending on the type of solver (we are using FFE Plus), software release, or service pack used.

Create a **Frequency** study titled *01*; you may copy loads, restraints and the mesh from *00 buckling*. Remember to reverse the load direction to tensile and make it 1500N. Obtain the solution and record the first natural frequency. Next, proceed in load steps shown in Figure 4-10.

Study number	Preload N	Frequency Hz
01	1500	177.88
02	1000	162.78
03	500	146.12
04	0	127.31
05	-500	105.19
06	-1000	76.95
07	-1200	62.15
08	-1400	42.49
09	-1500	27.89
10	-1550	16.20
11	-1570	7.52
12	-1575.49	0.31

Figure 4-10: A summary of the 12 studies illustrates the change of the first natural frequency with the applied pre-load. All runs are solved with the default element size of 3.11mm.

Notice that the compressive pre-load in study 12 is just 0.01N less than the bucking load.

Results summarized in the above table are presented as a graph in Figure 4-11. A very steep curve near the buckling load is why the last four steps were conducted with small load increments.

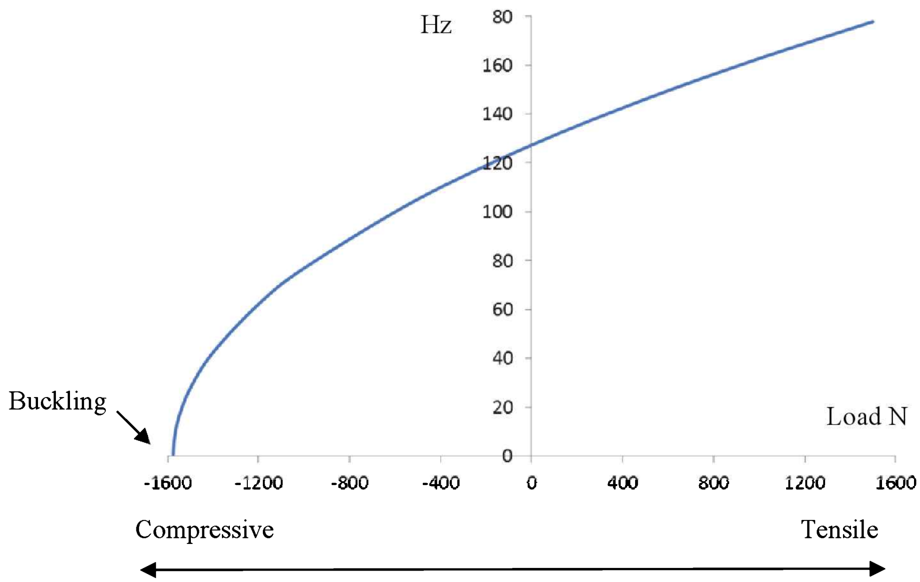


Figure 4-11: Frequency of the first mode of vibration as a function of pre-load.

The frequency reaches zero when the compressive load equals the buckling load.

As Figure 4-11 shows, a pre-load causing the drop of the first natural frequency to zero is equal to the buckling load.

Tensile stresses develop positive stress stiffness that adds up to the elastic stiffness; therefore the resultant stiffness increases, which is reflected by a higher natural frequency as compared to the unloaded column. Compressive stresses develop negative stress stiffness that is subtracted from the elastic stiffness and this decreases the resultant stiffness; consequently the natural frequency decreases. When the pre-load approaches the magnitude of buckling load, the resultant stiffness is very low producing a very low natural frequency. When the pre-load reaches the buckling load, the stiffness drops down to zero and that leads to buckling.

When the magnitude of a compressive load approaches the buckling load, the effective stiffness is very close to zero; this is why it has a near-zero natural frequency. Therefore, we may use the model from study *12*, the one pre-loaded with a 1575.49N compressive load, as a tool to investigate the effect of discretization on the model stiffness. As you know, meshing adds some stiffness to a model; let's call it artificial stiffness. The larger the elements are, the more artificial stiffness is added and vice versa. The artificial stiffness does not change with pre-load; therefore in study *12*, the artificial stiffness is a major contributor to the model stiffness. The natural frequency of 0.31Hz is not a real frequency under that load. Elastic stiffness and stress stiffness have almost canceled themselves out and this 0.31Hz frequency is an artifact produced by the artificial stiffness that is there due to discretization error. The numerical value of that frequency completely depends on the choice of mesh size.

We will now conduct a numerical experiment to demonstrate that the remaining model stiffness in study *12* is a product of discretization, and not a real stiffness.

Keeping in mind that all twelve studies were run with default element size 3.1mm, we will now use 2mm element size. Copy study *12* into *12 fine* and re-mesh it with a 2mm element size. With a smaller element size, the artificial stiffness is reduced and the effective stiffness becomes negative. An attempt to run the solution produces an error message shown in Figure 4-12.

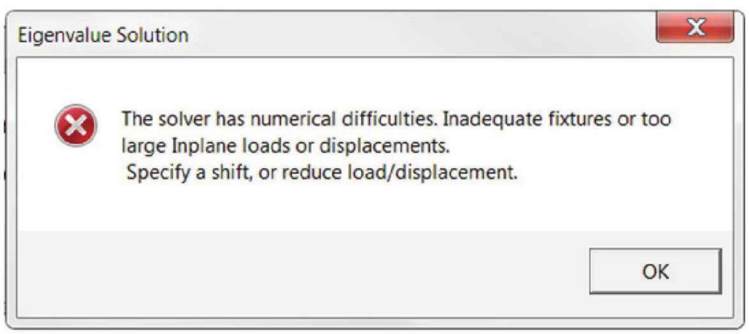


Figure 4-12: Error message caused by a negative effective stiffness.

The explanation offered by the error message is generic and does not apply to our problem.

We now realize that all that was "holding" the model stable in study *12* was the artificial stiffness produced by the process of discretization. The contribution of artificial stiffness to total stiffness is significant only for loads very close to buckling because the real stiffness is then very low.

You may want to analyze the effect of element size on the natural frequency in the model subjected to a compressive load that is very close to the buckling load. You'll find that the natural frequency strongly depends on the element size as shown in Figure 4-13.

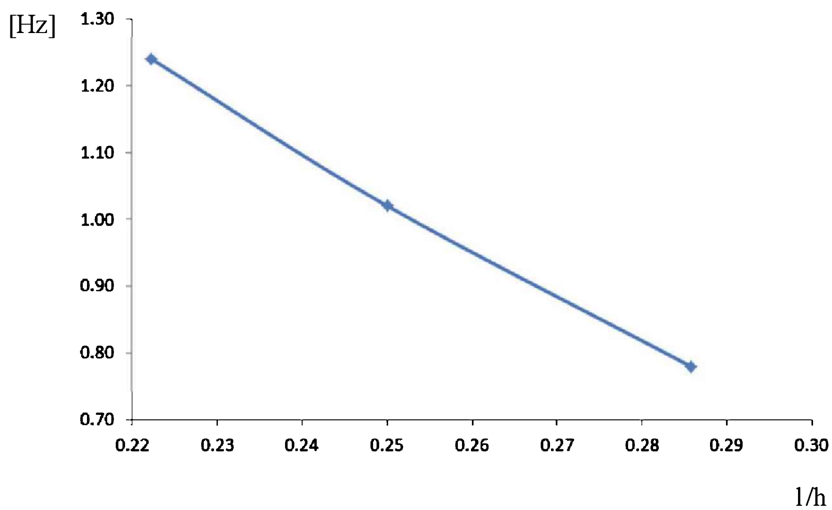


Figure 4-13: Natural frequency at a compressive load of 1575.49N as a function of the inverse of element size, *h*.

A strong dependence of frequency on the element size makes these results useless.

A fundamental rule of using FEA results states that before we use results to make a design decision, we must prove that these results are not significantly dependent on the choice of discretization (element size). The graph in Figure 4-13 proves the opposite: frequency results for load magnitudes close to buckling are strongly dependent on the mesh size.

5: Modal analysis - properties of lower and higher modes

Topics covered

- Modal analysis using shell elements
- Properties of lower and higher modes
- Convergence of frequencies with mesh refinement

Consider a bracket supported by two hinges along both ends. Open the U BRACKET part model and notice that the bracket is modeled as a surface (Figure 5-1).

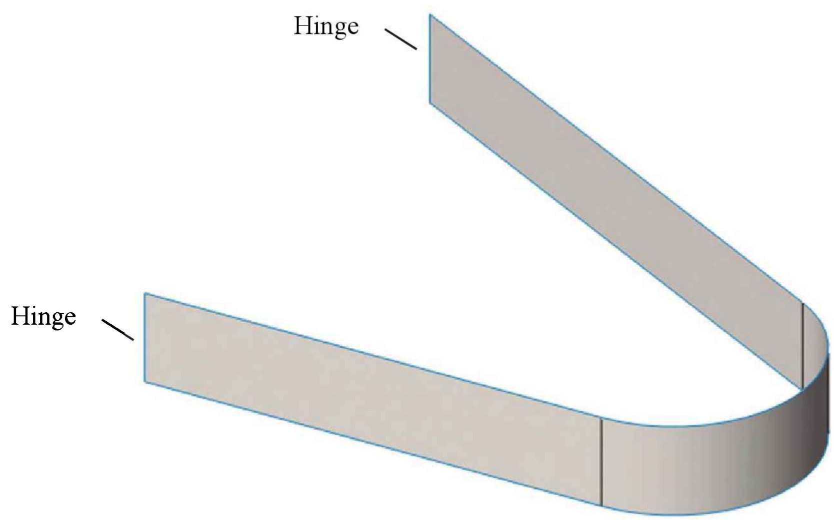


Figure 5-1: U BRACKET model supported by two hinges.

The geometry is represented by a surface. Therefore the thickness information is missing from geometry.

Create a **Frequency** study and define a Shell thickness of 2mm.

Define restraints as shown in Figure 5-2.

Immovable

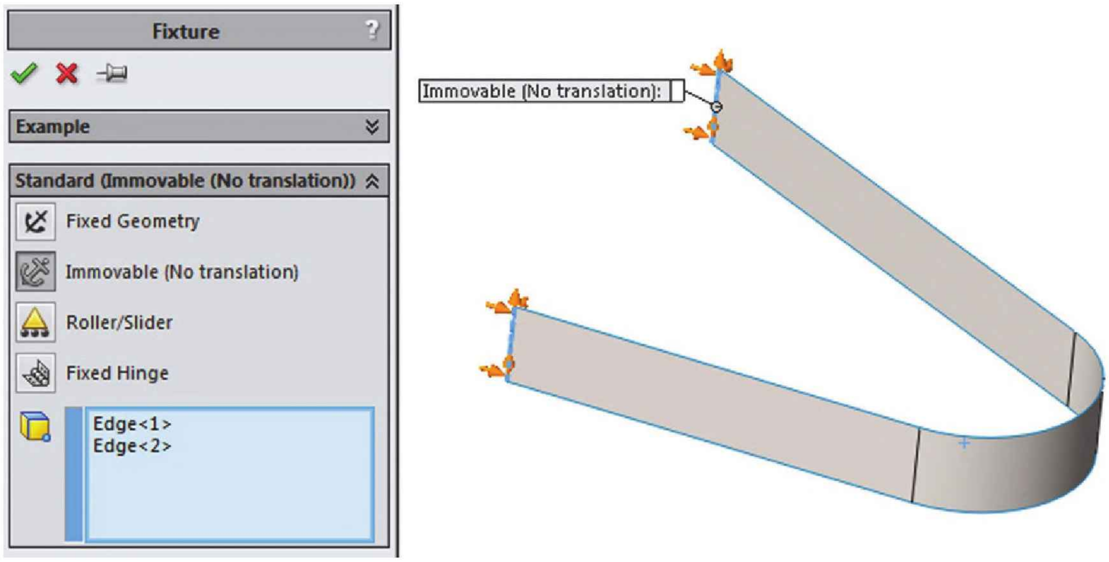


Figure 5-2: U BRACKET restraints

Use Immovable restraints to allow hinging about the two short ends.

Notice that the **Fixture** window differentiates between **Fixed Geometry** and **Immovable (No translation)** restraints. This is because the model geometry is a surface, and **Simulation** recognizes it as a candidate for meshing with shell elements.

Mesh the model using the default settings (Figure 5-3).

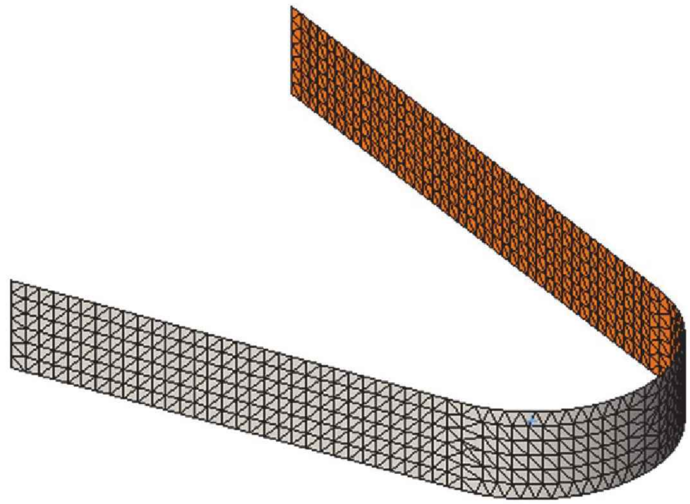


Figure 5-3: U BRACKET meshed with shell elements.

Colors (not visible in this B&W illustration) differentiate between the top and bottom of shell elements. Here, the outside is meshed with tops; the shell's normal vector goes from inside to outside the model.

Shell element orientation is very important in stress analysis. Different stress results are reported on different sides of shell elements. In modal analysis, which does not calculate stress, shell orientation has no impact on results. This is of course correct only if modal analysis is our only objective. If it is followed by vibration analysis where stresses need to be analyzed, then the orientation of shell elements must be carefully controlled.

Obtain a solution for 10 modes of vibration and review the results as shown in Figure 5-4.

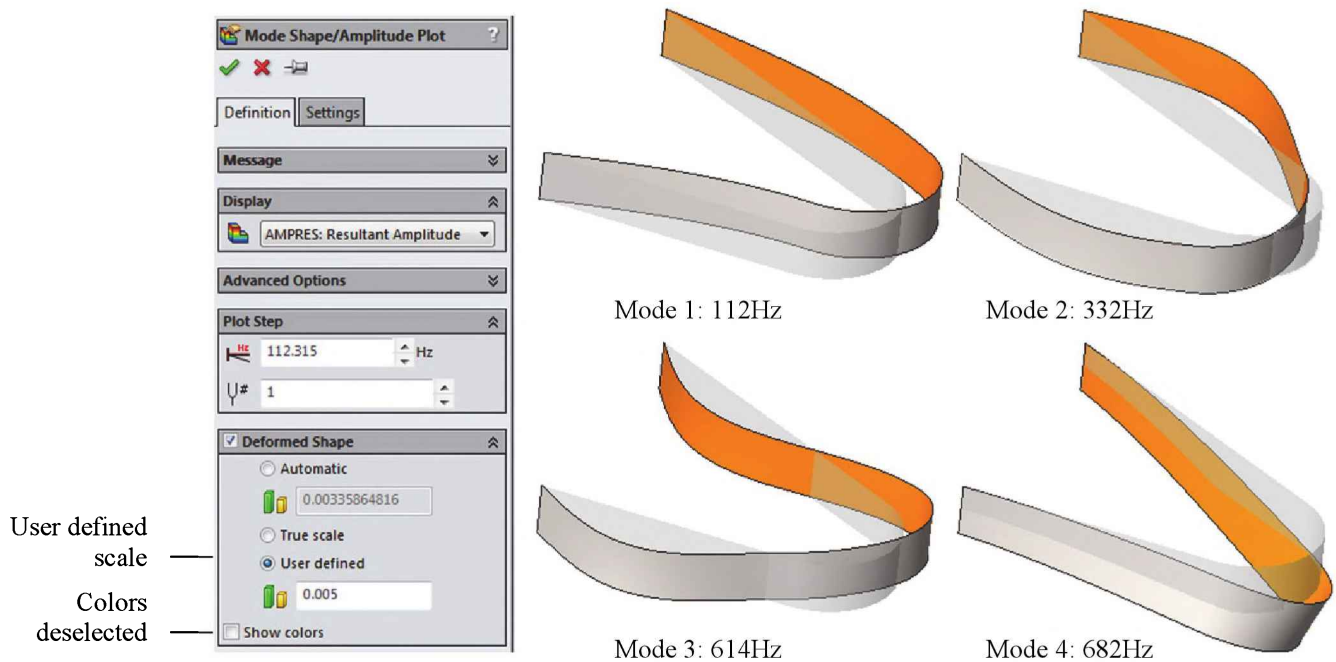


Figure 5-4: The first four modes of U BRACKET.

The undeformed model shape is shown along with the deformed model. To differentiate the undeformed shape from the modal shape, the scale of deformation has been adjusted individually for each mode. The Mode Shape window shows the deformation scale adjusted for Mode 1. Colors are not shown.

A review of shapes of all calculated modes reveals the following qualitative properties of lower and higher modes:

Lower modes "find" the easiest way to move structure; notice that the bracket vibrating in mode 1 hinges about supports, little deformation is present. Higher modes have progressively complex shapes while motion tends to be "more uniformly" distributed over the model.

The above findings indicate the following:

- Lower modes tend to maximize kinetic energy and minimize strain energy of a vibrating structure.
- Higher modes tend to maximize strain energy and minimize kinetic energy.

While the second statement is easy to prove just by observing modal shapes, the first one can't be proven by watching modal shapes because displacement results are normalized and are not comparable between different modes. Vibration analysis following modal analysis is required to demonstrate higher displacement amplitudes of lower modes.

The U BRACKET model is numerically efficient (solution is fast) because of its simplicity and because it uses shell elements. This makes it a convenient tool to demonstrate the effect of mesh refinement on the natural frequency already discussed in the previous chapter.

Obtain frequency results for meshes with element sizes of 10mm, 5mm, and 2mm. The graph in Figure 5-5 demonstrates the convergence of the first frequency.

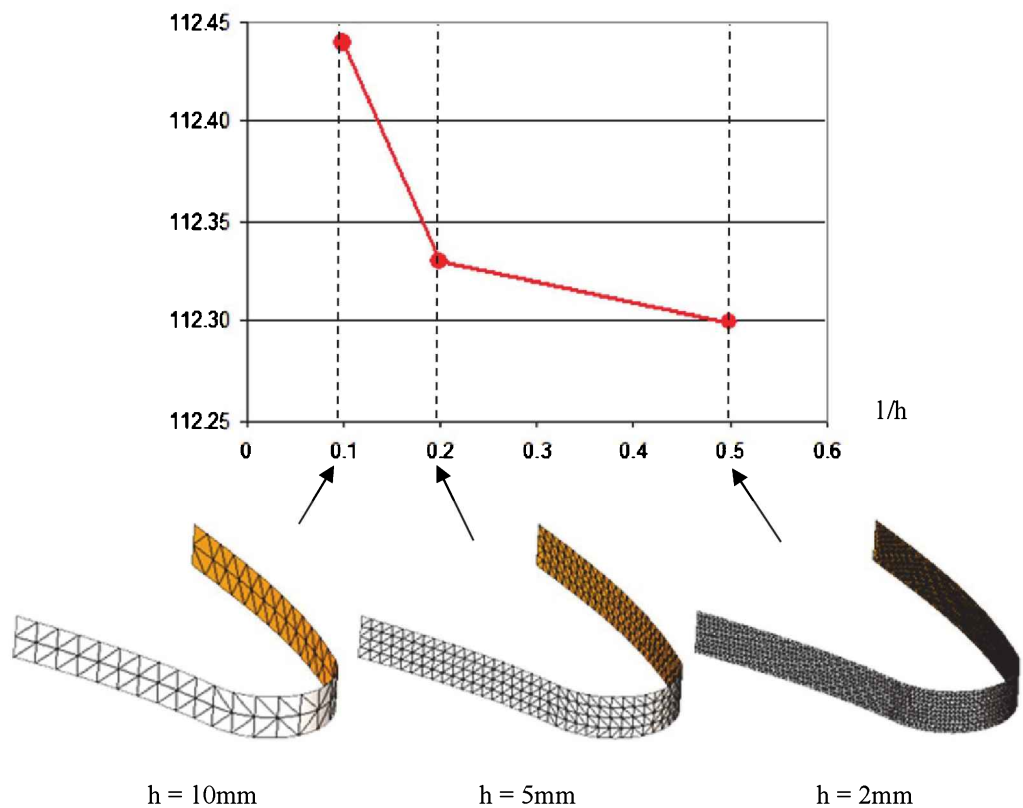


Figure 5-5: Convergence of the frequency of the first mode.

The graph shows the frequency as a function of 1/h where h is the element size.

The effect of mesh density is not strong, but still noticeable.

6: Modal analysis – mass participations, properties of modes

Topics covered

- Modal mass
- Modes of vibration of axisymmetric structures
- Modeling bearing restraints
- Using modal analysis to find "weak spots"

Procedure

We'll analyze the assembly model SHAFT which is a simplified representation of a shaft with two gears, supported by two spherical bearings that allow for some angular displacement. Many small features that are not essential for modal analysis (teeth, chamfers, rounds, undercuts) are not present in this model. They would have to be included if a stress analysis was to be conducted.

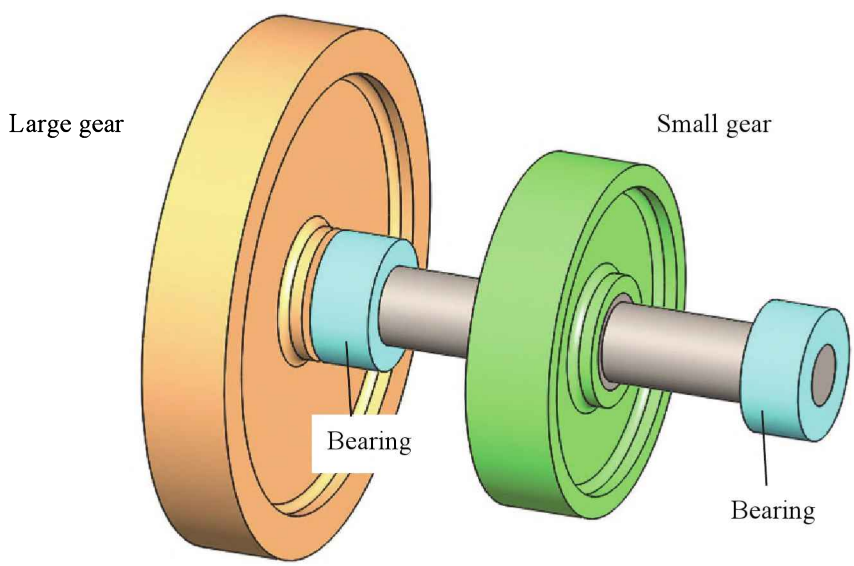


Figure 6-1: SHAFT supported by two spherical bearings.

Many small features important for manufacturing and assembly and function are not included in this model. The model is shown in the 01 bearings configuration.

The objective of this exercise is to find the first few natural frequencies of the shaft supported by spherical bearings, and to compare these results with results of the shaft when unsupported. This comparison will highlight interesting vibration properties of axi-symmetric structures and the effect of restraints on the mass participating in vibration. The major modeling consideration is modeling supports that allow for angular displacement present in spherical bearing supports.

The SHAFT assembly model has three configurations: *01 bearing*, *02 no bearings,* and *03 spherical bearing faces*. Configuration *01 bearings* serves only to illustrate the problem. Notice that applying restraints to cylindrical faces of two bearings would eliminate rotations allowed by the spherical bearings. The difference between a fixed bearing support and a spherical bearing support is easy to explain if we consider modes of vibration of a beam with fixed supports and simple supports.

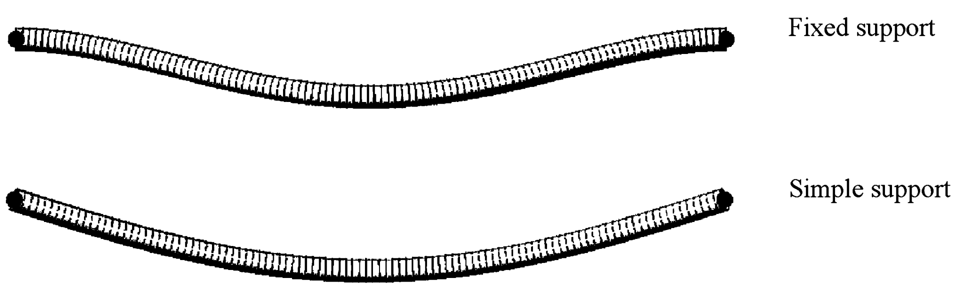


Figure 6-2: Bending mode shape of a beam with fixed supports (top) and simple supports (bottom).

The ends of the beam with simple supports are able to rotate. The model used to create these plots is called BEAM DEMO. It is not directly related to the SHAFT exercise.

The difference between fixed and simple supports is easy to model using beam elements, as shown in the BEAM DEMO model. These two types of supports are differentiated in BEAM DEMO by selecting either **Fixed** or **Immovable** restraints. In the case of the SHAFT model where restraints can't be applied to one point, we'll need to use a **Bearing Support** as shown in Figure 6-3.

Change the model configuration to *02 no bearings*, create a **Frequency** study titled *01 bearing support* and define **Bearing Support** restraints as shown in Figure 6-3.

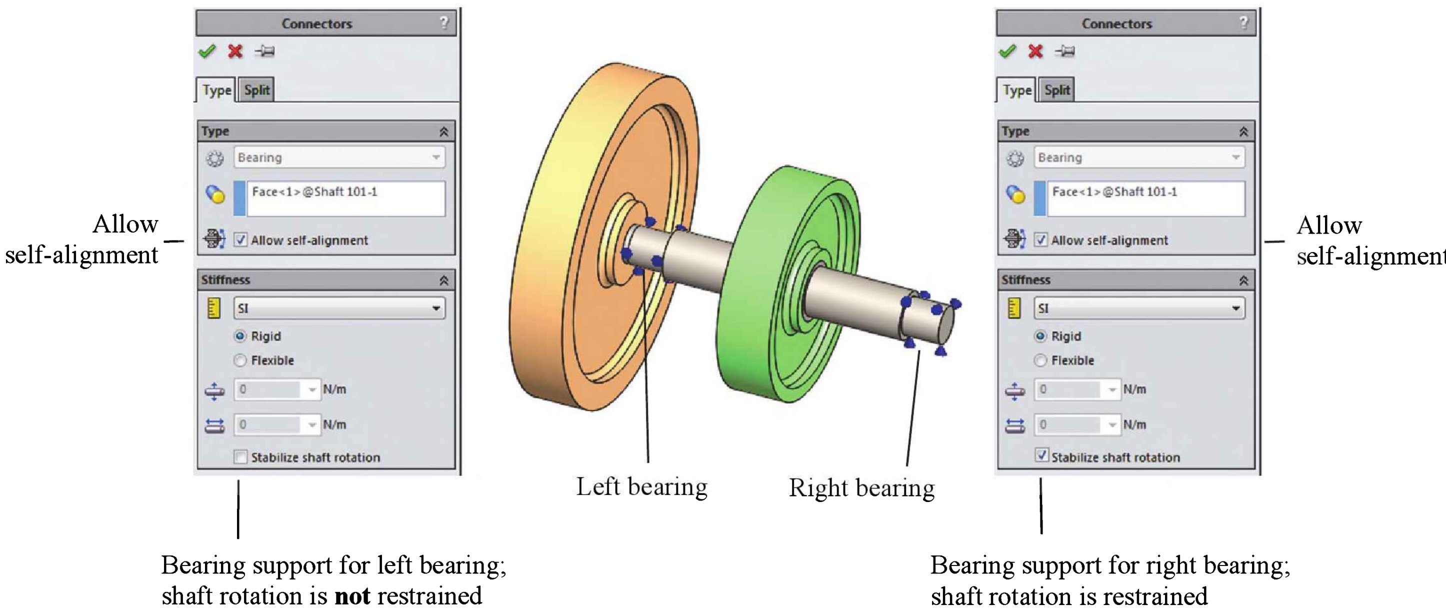


Figure 6-3: Bearing Support

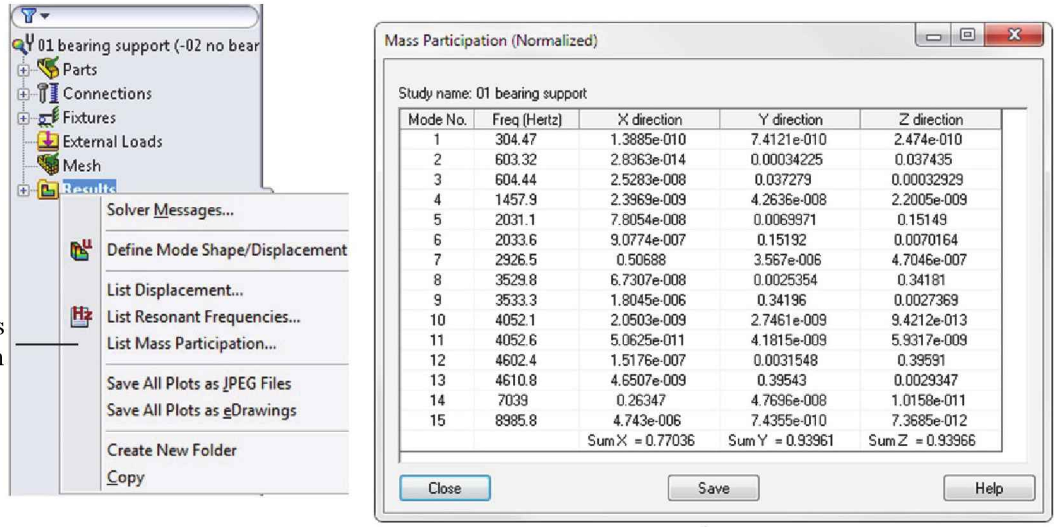
Bearing supports must be defined individually for each side.

If rotation was not restrained on one of the bearings, the model would have one **Rigid Body Mode** – rotation about its axis.

Specify 15 modes in the study properties, mesh the model with default element size.

Obtain solution, right click the results folder and select **List Mass Participation** as shown in Figure 6-4.

List Mass Participation



Mode No.	Freq (Hertz)	X direction	Y direction	Z direction
1	304.47	1.3885e-010	7.4121e-010	2.474e-010
2	603.32	2.8363e-014	0.00034225	0.037435
3	604.44	2.5283e-008	0.037279	0.00032929
4	1457.9	2.3969e-009	4.2636e-008	2.2005e-009
5	2031.1	7.8054e-008	0.0069971	0.15149
6	2033.6	9.0774e-007	0.15192	0.0070164
7	2926.5	0.50688	3.567e-006	4.7046e-007
8	3529.8	6.7307e-008	0.0025354	0.34181
9	3533.3	1.8045e-006	0.34196	0.0027369
10	4052.1	2.0503e-009	2.7461e-009	9.4212e-013
11	4052.6	5.0625e-011	4.1815e-009	5.9317e-009
12	4602.4	1.5176e-007	0.0031548	0.39591
13	4610.8	4.6507e-009	0.39543	0.0029347
14	7039	0.26347	4.7696e-008	1.0158e-011
15	8985.8	4.743e-006	7.4355e-010	7.3685e-012
		Sum X = 0.77036	Sum Y = 0.93961	Sum Z = 0.93966

Save to csv format

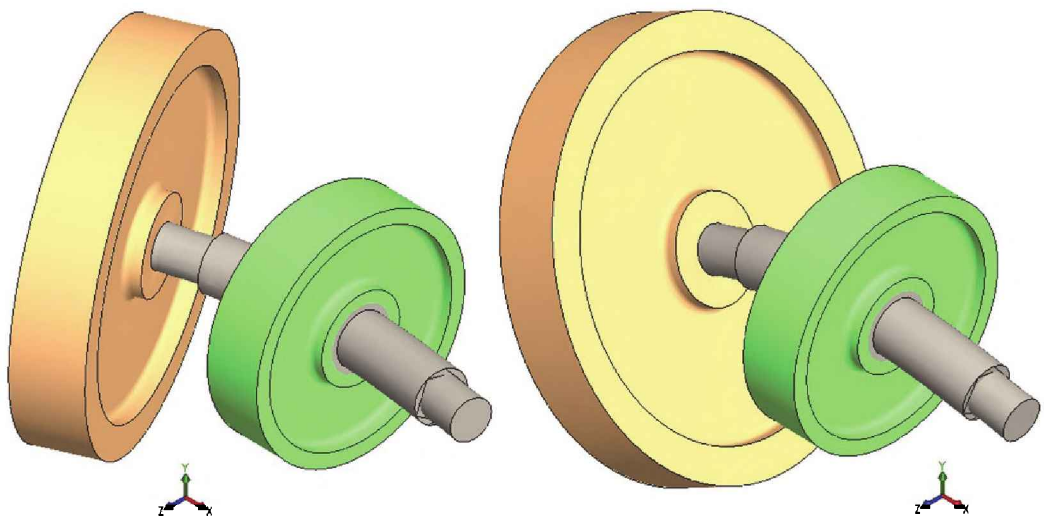
Figure 6-4: List of mass participation in modes 1-15.

The Mass Participation window lists frequencies of all modes along with their normalized mass participation in the X, Y, and Z directions of the global coordinate system.

We'll use results listed in the **Mass participation** window in Figure 6-4 to make important observations about modal frequencies and associated mass participation in the SHAFT model.

Animate the first mode (frequency 304.5Hz) to see that this is a torsional mode. Mode 2 and mode 3 are bending modes rotating about the centers of the bearings. The modal frequencies are very close: 603.3Hz and 604.4 Hz; modal shapes are identical but rotated by 90° about the X axis.

Identical modal shapes and almost identical frequencies are not coincidental. Repetitive modes with the plane of vibration rotated by 90° characterize modal results of axi-symmetric structures. A small difference in the numerical value of frequency is caused by discretization error which makes the stiffness of the finite element model not perfectly axi-symmetric. Review the list of frequencies in Figure 6-4 and identify more repetitive modes: mode 5 and mode 6, mode 8 and mode 9, mode 10 and mode 11. Animate the modal shapes to confirm that the shapes are indeed identical but rotated by 90° (Figure 6-5).



Mode 2: 603.3Hz Mode 3: 604.4Hz

Figure 6-5: Repetitive modes 2 and 3 of the model represent the same physical mode.

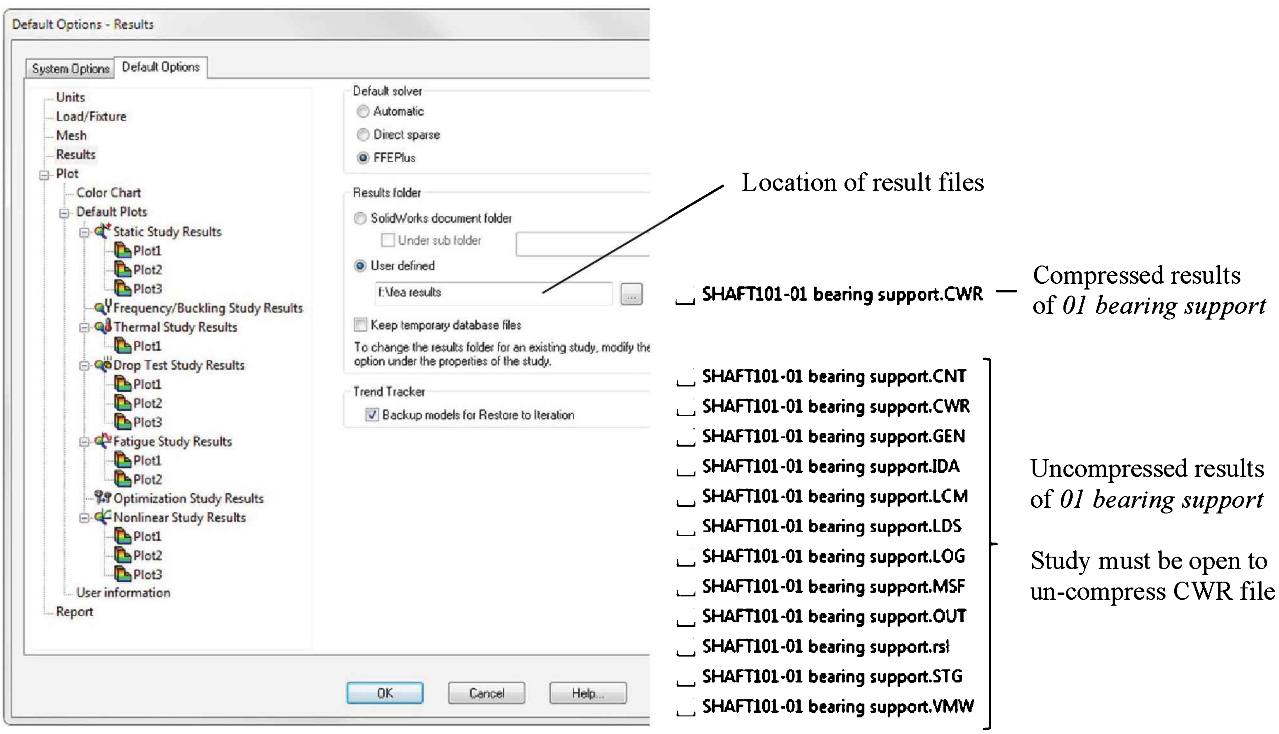
In real life, vibration may take place in any plane crossing the X axis.

Notice that modal shapes are not exactly aligned with XZ and XY planes. The alignment is not a rule for analysis of axi-symmetric problems.

Let's now review the remaining results shown in the **Mass Participation** window in Figure 6-4. Columns titled **X direction**, **Y direction**, **Z direction** list the percentage of total mass of the model that participate in the vibration in the given mode. In the **Modal Superposition Method**, mass participating in a given mode is used to represent the contribution of this mode to the vibration response.

The **Cumulative Mass Participation** in X, Y, and Z at each mode is the sum of the values up to and including that mode. This can be easily calculated in a spreadsheet after saving **Mass Participation** results in a csv file. These results are also readily available in the **OUT** file which is a part of the **CWR** file where **SolidWorks Simulation** stores results from each study. The **OUT** file lists **Participation Factors**, **Modal Masses**, **Mass Participations** and **Cumulative Mass Participations**.

Figure 6-6 explains how to find the OUT files associates with a given study.



Simulation Options window
Default Options tab

Study *01 bearing support*
result files

<u>Figure 6-6: Data base of study *01 bearing support* in model SHAFT101.</u>

The OUT file containing detailed results of the modal analysis is stored in a CWR file once the study is closed.

The file SHAFT101-01 bearing support.OUT is a text file and may be opened with any text editor or a spreadsheet. It has been saved as a text file SHAFT.txt and is located in this chapter folder.

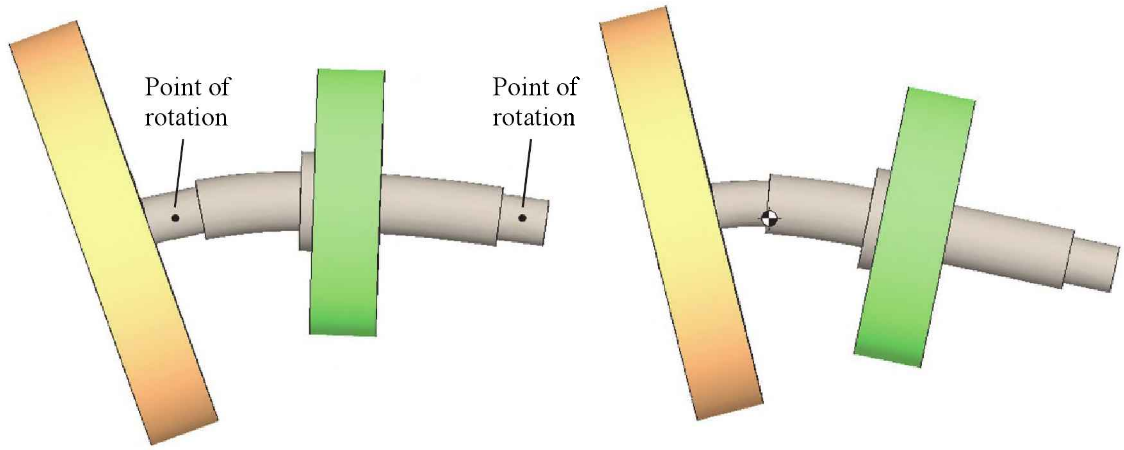
We'll now compare the modes of vibration of the SHAFT supported by two spherical bearings (the *01 bearing support* study we have just completed) with modes of vibration of an unsupported model. Copy study *01 bearing support* into study *02 no support*, delete all restraints and obtain the solution. Export frequency and mass participation results to Excel and prepare a summary of results as shown in Figure 6-7.

Study name	01 bearing support					02 no support			
Mode No.	Hz	X	Y	Z		Hz	X	Y	Z
1	304.47	1.39E-10	7.41E-10	2.47E-10		0	1	0	0
2	603.32	2.84E-14	0.000342	0.037435		0	0	1	0
3	604.44	2.53E-08	0.037279	0.000329		0.000625	0	0	1
4	1457.9	2.40E-09	4.26E-08	2.20E-09		0.000671	7.24E-31	2.70E-21	4.02E-22
5	2031.1	7.81E-08	0.006997	0.15149		0.000923	1.99E-21	2.19E-30	1.09E-20
6	2033.6	9.08E-07	0.15192	0.007016		0.001748	2.95E-22	1.09E-20	0
7	2926.5	0.50688	3.57E-06	4.70E-07		784.58	1.81E-22	1.65E-22	1.04E-21
8	3529.8	6.73E-08	0.002535	0.34181		785.29	5.74E-23	1.10E-21	1.33E-22
9	3533.3	1.80E-06	0.34196	0.002737		791.81	9.82E-26	3.36E-21	5.71E-23
10	4052.1	2.05E-09	2.75E-09	9.42E-13		2982.6	5.91E-21	2.58E-22	2.82E-21
11	4052.6	5.06E-11	4.18E-09	5.93E-09		2986.6	7.71E-22	2.25E-21	3.25E-22
12	4602.4	1.52E-07	0.003155	0.39591		3684.4	9.49E-21	6.16E-23	6.25E-22
13	4610.8	4.65E-09	0.39543	0.002935		4050.8	1.72E-21	3.46E-21	2.03E-22
14	7039	0.26347	4.77E-08	1.02E-11		4051.4	3.83E-22	1.62E-22	3.18E-21
15	8985.8	4.74E-06	7.44E-10	7.37E-12		5709.1	1.40E-21	3.63E-20	5.68E-20
SUM OF MASS PARTICIPATIONS:		0.77	0.94	0.94			1.00	1.00	1.00

Figure 6-7: Summary of results of studies *01bearing support* and *02 no support.*

The border around the first six modes of the unsupported model mark the Rigid Body Modes.

The model in study *02 no supports* has no supports at all. Animate the first six modes to confirm that they are all Rigid Body Modes. The first elastic mode is Mode 7 and its duplicate is Mode 8. Contrary to the results of *01 bearing support* study, this is not a torsional mode but a bending mode. Therefore, Mode 7/8 is comparable to Mode 2/3 in the model with bearing supports.



Spherical bearings support.
Frequency: 603Hz

Vibration takes place about the spherical bearings.

No support.
Frequency: 785Hz

Vibration takes place about the centre of mass.

Figure 6-8: Comparison of the first bending mode of the SHAFT with bearing supports and unsupported .

The scale of deformation is adjusted to show the deformed shapes clearly.

In the absence of supports, vibration always takes place about the center of mass. Removal of the bearing supports affected the model stiffness, and mass participation. The combined effect of both changes resulted in the increase of the frequency of the first bending mode.

Return to Figure 6-7 and notice that mass participation in the model with Rigid Body Modes shows full mass participation in translational modes 1, 2, 3. Therefore, it is not possible to compare mass participations between the modes shown in Figure 6-8.

The SHAFT model can be also used to illustrate that modal analysis provides a qualitative review of "weak spots". While absolute displacements can't be found in modal analysis, the relative displacement results are valid. Keeping this in mind animate the first bending mode of model supported by spherical bearings and observe the bending of the small shaft holding the large gear. This animation reveals a potential design fault: the heavy gear held by a soft cantilever shaft (Figure 6-9).

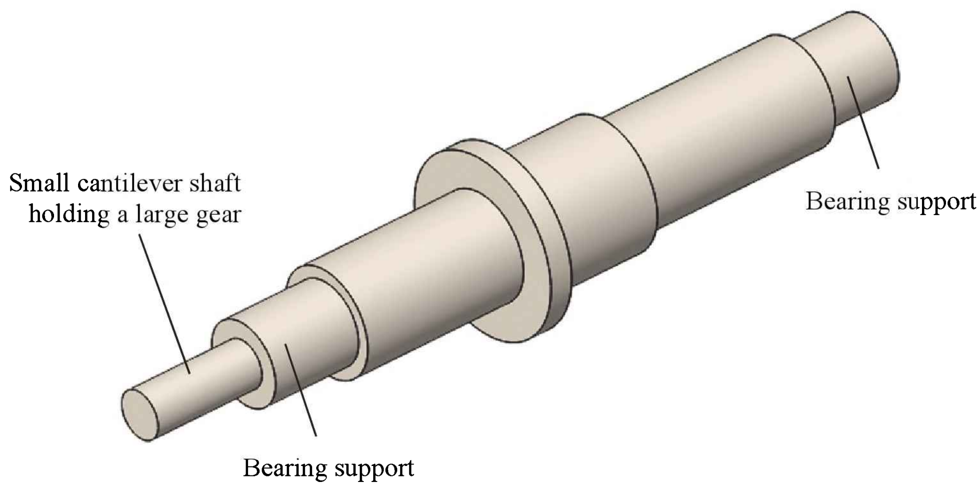


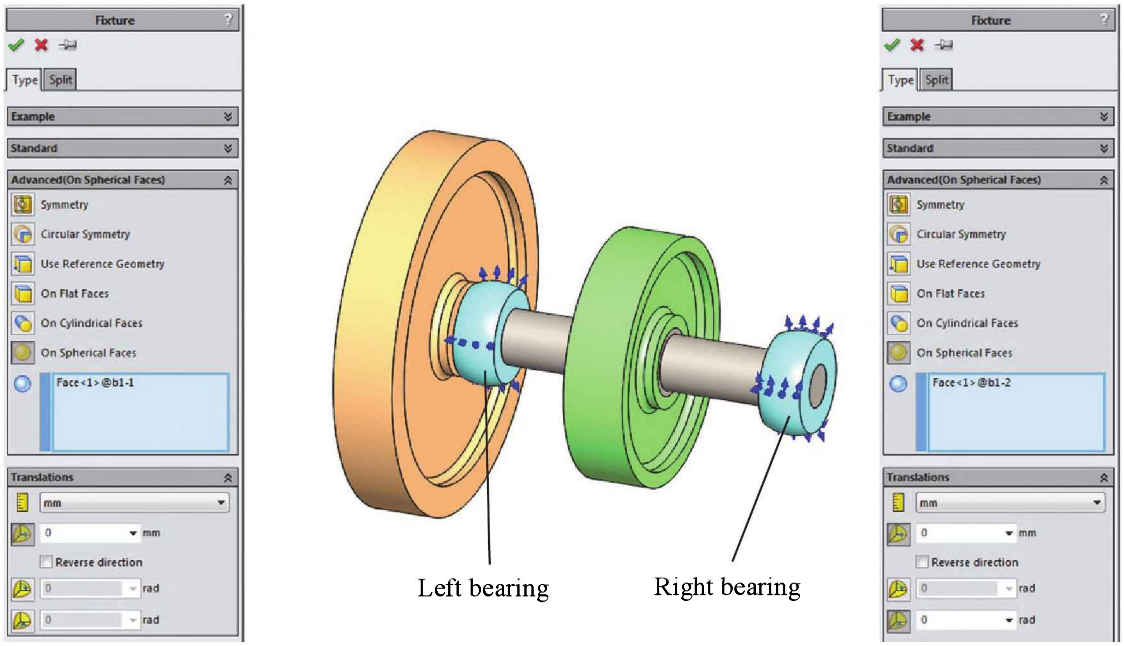
Figure 6-9: Potential design flaw identified by modal analysis: soft cantilever beam supports a heavy gear.

The split face used to define a bearing support near the large gear is not shown.

The flexibility of the large gear sitting on the small cantilever shaft may cause alignment problems, noise and premature wear. Therefore, based on the results of this analysis, the designer may decide to move the bearing to the end of the shaft.

We used a **Bearing Support** to provide the model with the ability to perform rotations about the bearings even though bearings themselves were not included in the model. The same support may be represented if bearings are added to the assembly and their outside face is modeled spherically as in assembly configuration *03 spherical bearing faces.*

Figure 6-10 shows the **On Spherical Faces** restraints defined for both bearings; this is equivalent to the Bearing Support shown in Figure 6-3.



Left bearing:
Suppressed translation in the radial direction.

Right bearing:
Suppressed translations in the radial direction and in the latitudinal direction.

Figure 6-10: Bearing supports modeled using restraints defined in local spherical coordinate systems.

This restraint has to be defined individually for each bearing.

If only radial translation is restrained on both spherical faces, the model will have one RBM (rotation about its axis).To eliminate this RBM restrain latitudinal translation on the right bearing.

Analyze the model with restraints defined as shown in Figure 6-10 and verify that the results are close to those obtained using **Bearing Supports**. The slight difference is due to the fact that the model shown in Figure 6-10 takes into consideration the inertial effects of the bearings and also due to discretization error of the **On Spherical Faces** boundary conditions.

7: Modal analysis – mode separation

Topics covered

- Modal analysis with shell elements
- Modes of vibration of symmetric structures
- Symmetry boundary conditions in modal analysis
- Anti-symmetry boundary condition in modal analysis

Procedure

The CAR model is a simplified representation of a car body; we'll use it to find the first torsional mode of vibration. Frequency of the first torsional mode is an important factor in vehicle handling and ride quality. Modern unibody sedans have the first torsional frequency in the range 25-30Hz. Convertibles have "softer" bodies and, consequently, a much lower torsional frequency of 10-15Hz.

Due to a schematic representation of the car body design, the CAR model will not return realistic numerical values of natural frequencies. Still, it will allow us to review important modeling techniques used in modal analysis.

Review the CAR model in its two configurations *01 full* and *02 half* (Figure 7-1).

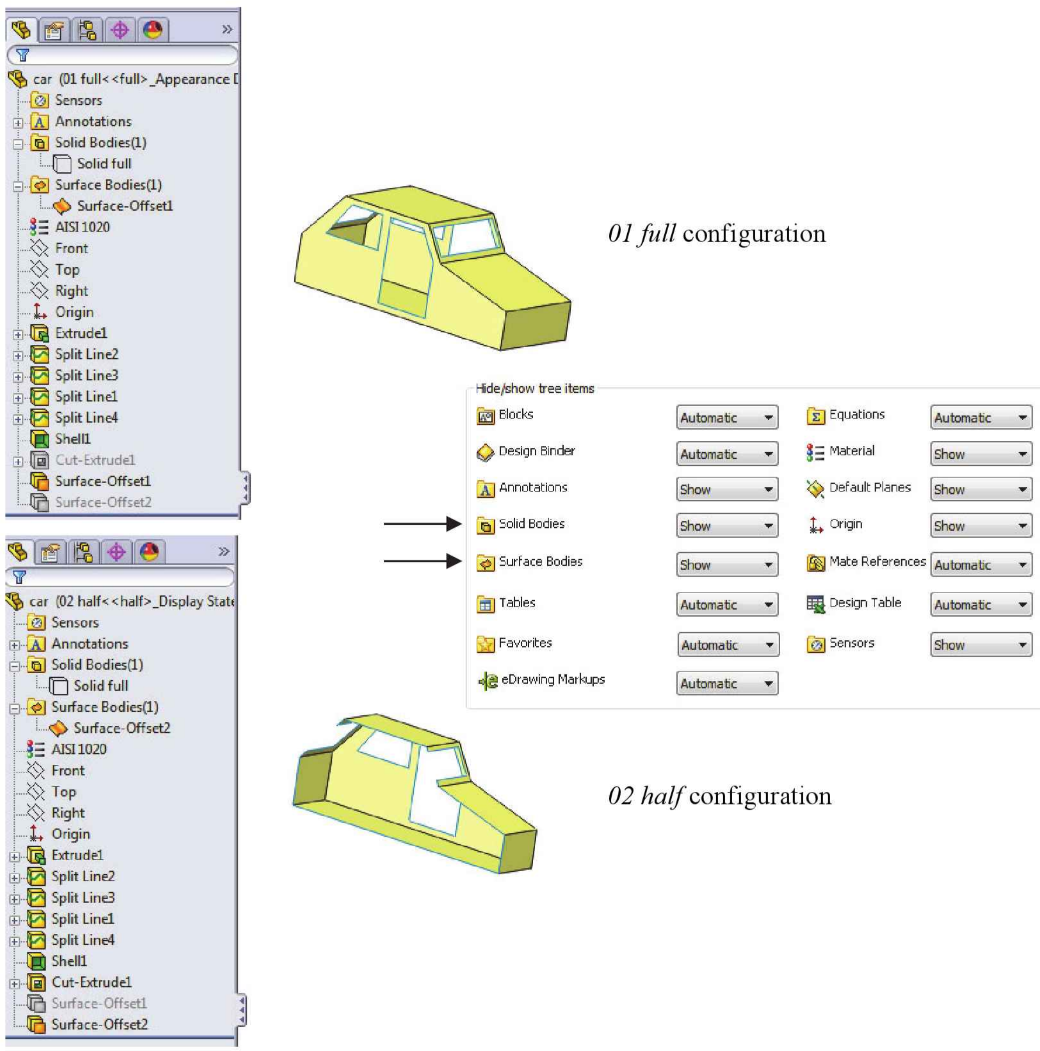


Figure 7-1: The CAR model consists of Solid and Surface bodies. The solid body is hidden.

To show the Solid Bodies and Surface Bodies folder in the SolidWorks Feature Manager Design Tree, right click anywhere in the Feature Manage window and use the pop-up window to show these items.

Activate the *01 full* configuration and create a **Frequency** study called *01 full*. In the study properties, request 12 modes to be found. Exclude the **Solid Body** from the analysis as shown in Figure 7-2.

Right-click SolidBody, select Exclude from Analysis.

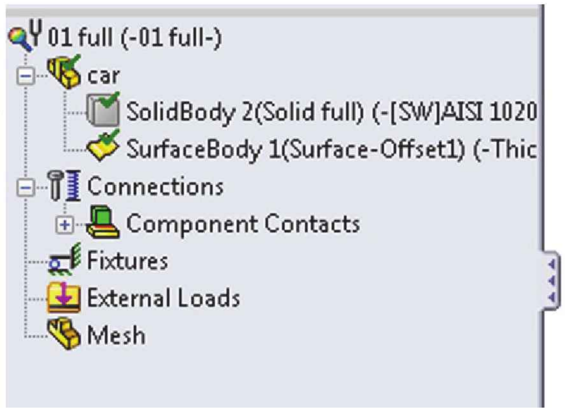
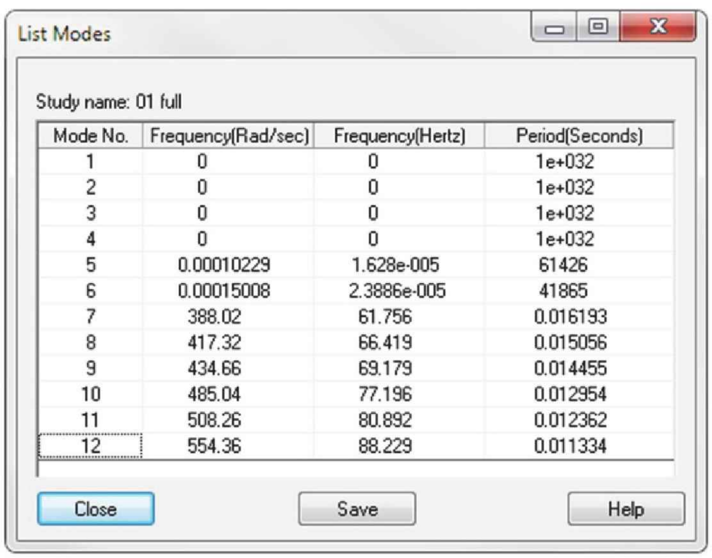


Figure 7-2: Solid Body must be excluded from the analysis to avoid meshing of the Solid Body.

Even though the Solid Bodies have been hidden in the CAD model, they still have to be removed from analysis.

Right-click **Surface Body**, select **Edit Definition** and enter a shell element thickness of 50mm (we do not attempt to model a car body realistically). Obtain the solution using the default element size and review the **List Modes** window (Figure 7-3).


List Modes

Study name: 01 full

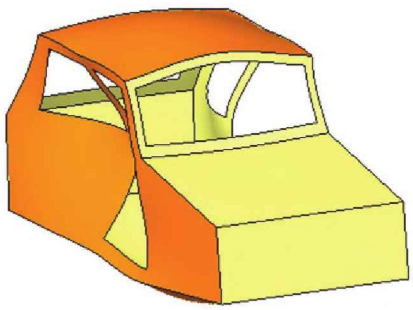
Mode No.	Frequency(Rad/sec)	Frequency(Hertz)	Period(Seconds)
1	0	0	1e+032
2	0	0	1e+032
3	0	0	1e+032
4	0	0	1e+032
5	0.00010229	1.628e-005	61426
6	0.00015008	2.3886e-005	41865
7	388.02	61.756	0.016193
8	417.32	66.419	0.015056
9	434.66	69.179	0.014455
10	485.04	77.196	0.012954
11	508.26	80.892	0.012362
12	554.36	88.229	0.011334

Close Save Help

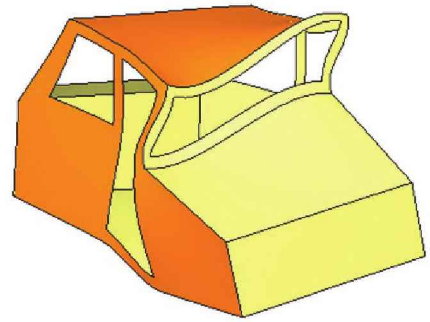
Figure 7-3: In the absence of restraints, the first six modes are Rigid Body Modes with frequencies very close to or equal to 0Hz.

Modes 5 and 6 are not exactly zero due to discretization error.

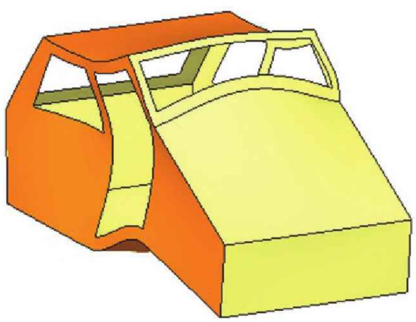
Animate the six elastic modes (modes 7-12) and notice that the second elastic mode of vibration, which is mode number 8, is the torsional mode we are looking for. Also, observe that the deformed shapes are either symmetric or anti-symmetric (Figure 7-4).



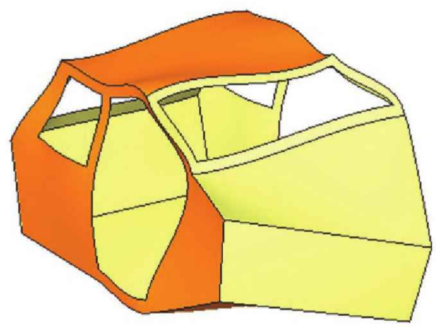
Mode 1: 388Hz; Symmetric



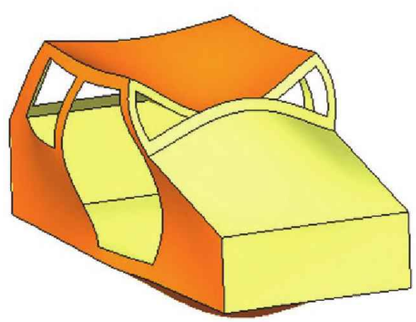
Mode 2: 417Hz; Anti-symmetric



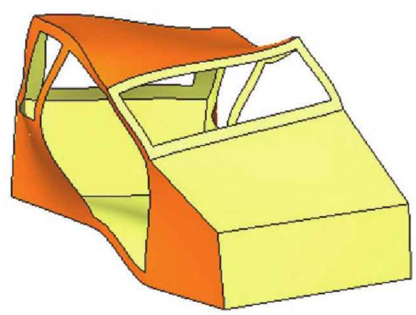
Mode 3: 434Hz; Symmetric



Mode 4: 485Hz; Anti-symmetric



Mode 5: 508Hz; Symmetric



Mode 6: 554Hz; Anti-symmetric

Figure 7-4: Modal shapes and frequencies of the first six elastic modes.

It is a coincidence that the symmetric and anti-symmetric modes are alternating.

The property of a modal shape to be either symmetric or anti-symmetric applies to all structures with a plane of symmetry. You may want to return to the SHAFT model in chapter 6 to confirm this.

Having found this important property of vibration of symmetric structures, we may ask if analysis can be performed on one half of the model. If so, and what boundary conditions should be defined? We will investigate this using the CAR model in the *02 half* configuration. We will first run a modal analysis with symmetry boundary conditions, then a modal analysis with anti-symmetry boundary conditions.

Switch to configuration *02 half*, create a **Frequency** study titled *02 sym*, exclude **Solid Body** from the analysis and define the same shell element thickness as before (50mm). Define **Symmetry Boundary Conditions** as shown in Figure 7-5.

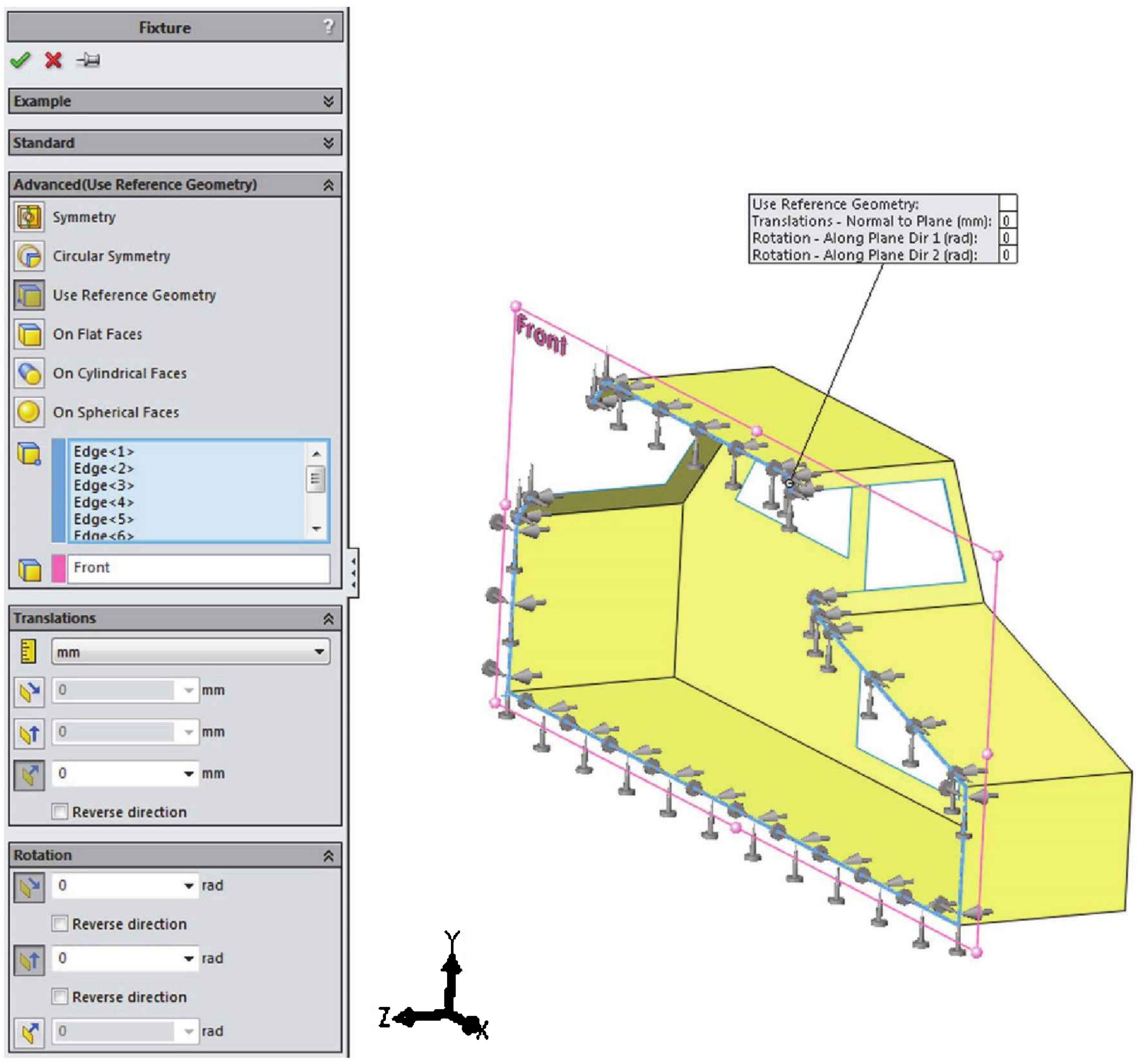


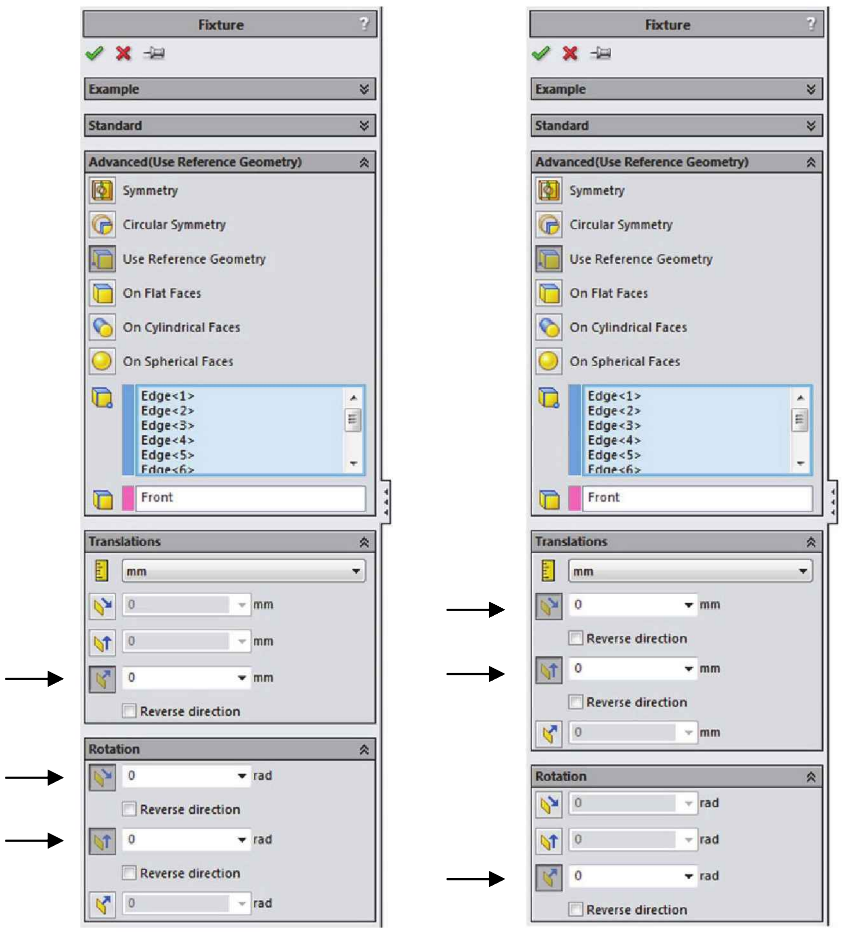
Figure 7-5: Symmetry boundary conditions applied to the edges of the shell element model.

Symmetry boundary conditions leave the model with 3 Rigid Body Motions.

Symmetry boundary conditions must be defined in terms of six degrees of freedom: three translations and three rotations. This is because the model uses shell elements which have six degrees of freedom per node.

Symmetry boundary conditions do not restrain the model fully; it can translate in X and Y directions and can rotate about Z direction. Therefore, the first three modes will be **Rigid Body Modes**. To find the first six elastic modes, request 9 modes in the properties of study *02 sym*.

Obtain the solution of the *02 sym* study and copy it into a new study titled *03 anti-sym*. The *03 anti-sym* study uses anti-symmetry boundary conditions which are easy to define by editing the symmetry boundary conditions. The editing is just a “flip”: degrees of freedom that were free in symmetry boundary conditions are restrained in anti-symmetry boundary conditions and vice-versa. Symmetry and anti-symmetry boundary conditions applied to edges in the plane of symmetry are shown in Figure 7-6.



Symmetry BC Anti-Symmetry BC

<u>Figure 7-6: Symmetry and anti-symmetry boundary conditions defined using the Front plane as reference geometry. Arrows indicate restrained directions.</u>

The symmetry boundary condition window is a repetition from Figure 7-5.

The anti-symmetry boundary conditions do not fully restrain the model either. A summary of symmetry and anti-symmetry boundary conditions is shown in Figure 7-7.

Rigid Body Displacement	Symmetry BC	Anti-symmetry BC
X translation	Free	Fixed
Y translation	Free	Fixed
Z translation	Fixed	Free
X rotation	Fixed	Free
Y rotation	Fixed	Free
Z rotation	Free	Fixed

Figure 7-7: Summary of Rigid Body Motions in a model with symmetry and anti-symmetry boundary conditions.

Notice that what is fixed in symmetry boundary conditions is free in anti-symmetry boundary conditions and vice versa.

Obtain the solution of *03 anti-sym* and proceed to review of results of both studies. Animate the first three modes in study *02 sym* and in study *03 anti-sym* and confirm that they are **Rigid Body Modes** corresponding to rigid body displacements identified in Figure 7-7.

Animate the elastic modes found in study *02 sym*, these are Modes 4-9 and notice that all shapes are symmetric. Next, animate Modes 4-9 found in study *03 anti-sym* and notice that all shapes are anti-symmetric.

Symmetry boundary conditions eliminate anti-symmetric modes and anti-symmetry boundary conditions eliminate symmetric modes. Performing modal analysis on one half of the model first with symmetry boundary conditions then with anti-symmetry boundary conditions is called the **Modal Separation** technique.

One half of a symmetric model can be used to extract all modes of vibration by combining results of a modal analysis with symmetry boundary conditions with the results of modal analysis of the same model with anti-symmetry boundary conditions as it is schematically shown in Figure 7-8.

Study name	01 full		02 sym		03 anti-sym	
			All modes are symmetric		All modes are anti-symmetric	
	Mode No.	Hz	Mode No.	Hz	Mode No.	Hz
	1	0	1	0	1	0
	2	0	2	0	2	0
	3	0	3	0	3	0
	4	0	4	62	4	65
	5	0	5	68	5	76
	6	0	6	81	6	87
sym	7	62	7	88	7	91
anti-sym	8	66	8	96	8	145
sym	9	69	9	117	9	164
anti-sym	10	77				
sym	11	81				
anti-sym	12	88				

Figure 7-8: Summary of results of studies *01 full*, *02 sym*, and *03 anti-sym.*

Borders around the first six modes of the unsupported full model (study 01 full) and around the first three modes of the partially supported models (studies 02 sym and 03 anti-sym) indicate Rigid Body Modes.

The solutions of study *01 full* may be obtained by merging solutions of study *02 sym* obtained with symmetry boundary conditions and *03 anti-sym* obtained with anti-symmetry boundary conditions.

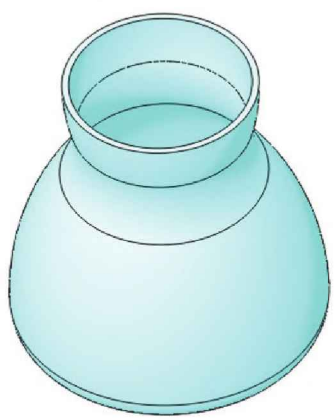
The **Modal Separation** technique may be also used to directly find the first torsional mode, which is the first elastic mode in the model with anti-symmetry boundary conditions.

8: Modal analysis – axi-symmetric structures

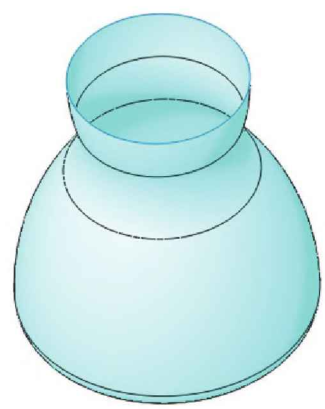
Topics covered

- Modes of vibration of axi-symmetric structures
- Repetitive modes
- Solid and shell element modeling

The analysis of the SHAFT model in Chapter 6 and the CAR model in Chapter 7 revealed important properties of modal shapes of symmetric models and axially symmetric models. In this chapter we continue with the analysis of an axi-symmetric model. The model VASE comes in two configurations *01 solid* and *02 shell*. The wall thickness in *01 solid* is 2mm. The surface in *02 shell* has been offset 1mm into the material. Either configuration may be used for analysis of modes of vibration of this axi-symmetric model. We will use *02 shell* which is suitable for meshing with shell elements. If you use *01 solid*, make sure to use a sufficiently small element size to avoid excessive element distortion. Your results will be slightly different because of the different modeling approach and different discretization error.



Configuration *01 solid*
Suitable for meshing with solid elements



Configuration *02 shell*
Suitable for meshing with shell elements

Figure 8-1: VASE model in two configurations.

Either solid or shell elements could be used for modal analysis.

Make sure the model is in the *02 shell* configuration and create a **Frequency** study titled *01 modal*. Verify that the material, **Ceramic Porcelain**, has been transferred from the CAD model. Exclude the **SolidBody** from the analysis and define the shell thickness as 2mm (Figure 8-2).

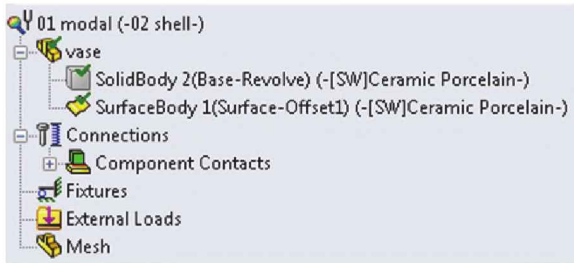


Figure 8-2: SolidBody removed from analysis, thickness defined for SurfaceBody.

SolidBody must be removed or else the automesher will mesh it with solid elements.

Define a **Fixed** restraint to the bottom of the VASE as shown in Figure 8-3.

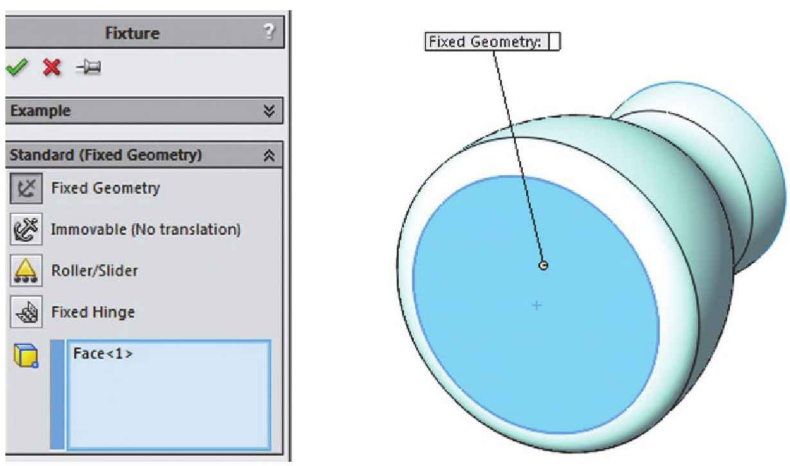
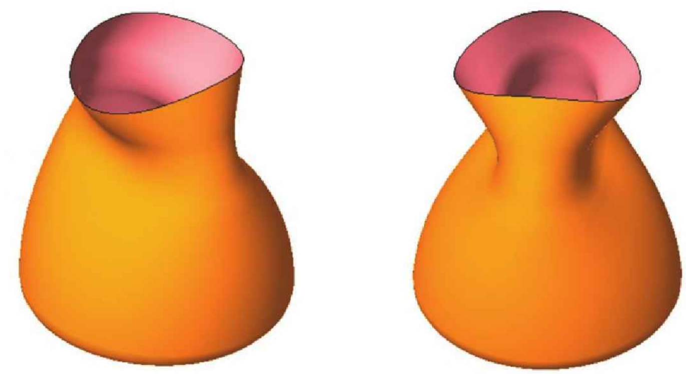


Figure 8-3: Fixed restraint applied to the bottom face.

This restraint fully restrains the model but also eliminates any vibration of the bottom face. Restraint symbols are not shown.

Mesh the VASE model with the default element size and obtain the solution for 12 modes. Vibration shapes and frequencies of the first two modes are shown in Figure 8-4.



Mode 1 3390.1Hz Mode 2 3391.7Hz

Figure 8-4: The first two modes of vibration.

The shape of both modes is identical but rotated about the axis of symmetry.

Repetitive modes characterize modal analysis results of axi-symmetric structures. The only reason these two modes do not have the same frequency is because of numerical error. The shape is also identical but rotated by about the axis of symmetry. Theoretically, shapes of repetitive modes should be rotated by 90º. In the case of the VASE, the angle of rotation is lower because of numerical error. More repetitive modes are listed in Figure 8-5.

Mode number	Frequency [Hz]	
1	3390.1	Repetitive
2	3391.7	
3	4416.4	Repetitive
4	4416.4	
5	7885.8	Repetitive
6	7911.9	
7	8789.5	Axial mode
8	10855	Repetitive
9	10857	
10	12953	Repetitive
11	12955	

Figure 8-5: The first twelve frequencies of VASE. Repetitive modes are indicated.

Numerical error makes frequencies of each pair of repetitive modes slightly different.

Notice that the only non-repetitive mode is mode 7, which vibrates axially.

9: Modal analysis – locating structurally weak spots

Topics covered

- Modal analysis with beam elements
- Modes of vibration of symmetric structures
- Using results of modal analysis to identify potential design problems

Procedure

Open the BAJA FRAME part (Figure 9-1). This model is an early iteration of a tubular frame designed by students of Western Engineering for the SAE BAJA competition. At an early stage of the design process, loads and restraints are largely unknown. Since modal analysis may be run without any restraints, an insight offered by modal analysis into structural properties of this design is particularly valuable.

Review the BAJA FRAME model and notice that all trims of Structural Members have been suppressed. Nodes are constructed in such a way that the center lines of the tubes all meet in one point. This was done to facilitate meshing with beam elements; manufacturing considerations will later require modifications to this design.

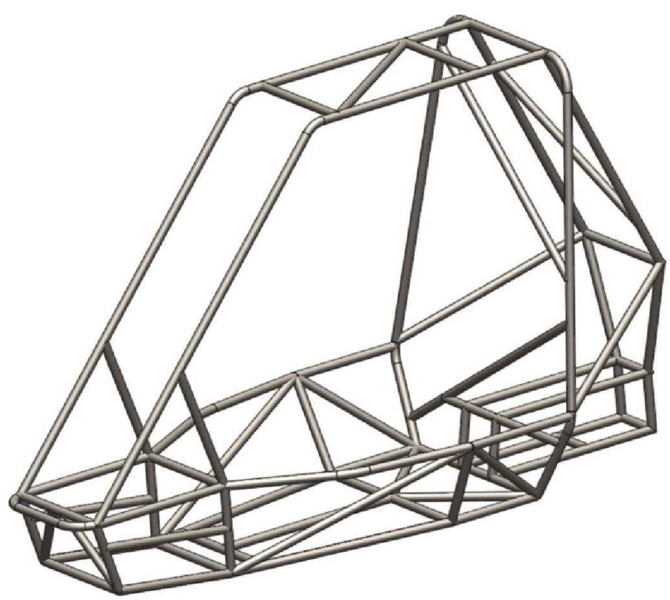
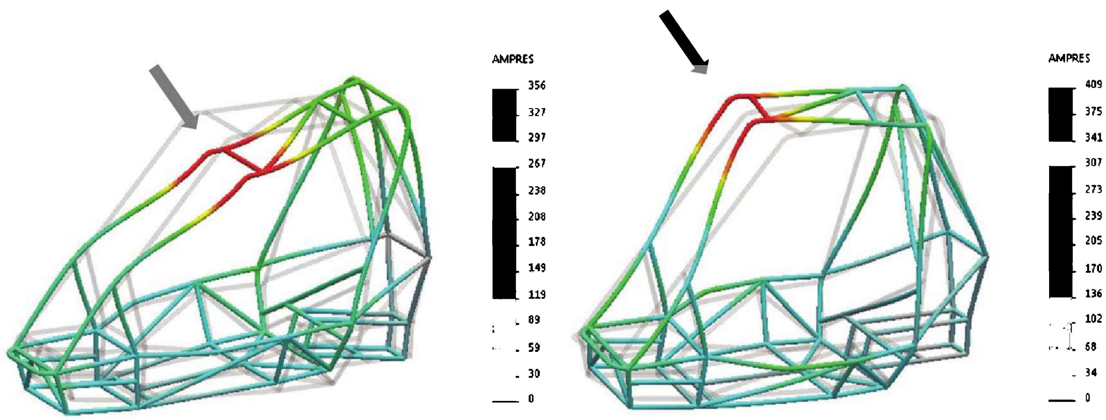


Figure 9-1: An early design iteration of Western Engineering's SAE BAJA frame.

The SolidWorks model uses structural members suitable for meshing with beam elements.

We will use modal analysis to review how stiffness is distributed in the model and to locate potential "weak spots" of this space frame. This can be done without defining any restraints. Create a **Frequency** study titled *Study 1* and specify 12 modes in the study properties, remember that the first six modes will be RBMs. The frame is symmetric, except for two diagonal members in the back; the modes will be very close to symmetric and anti-symmetric.

Obtain the solution and animate the first two elastic modes. Considering that in the absence of restraints, the model has six Rigid Body Modes, the first two elastic modes will be mode 7 and mode 8. Figure 8-2 shows these two modes using displacement plots. The use of colors (invisible in this black and white illustration) makes it easy to locate areas with the largest displacements. While absolute displacements results are meaningless in modal analysis, relative displacements are valid; just remember to compare displacements only within the same mode. Comparing displacement results between different modes is a severe error on the interpretation of modal analysis results.



Mode 1
Bending 36Hz

Mode 2
Torsion 47Hz

<u>Figure 9-2: The first two elastic modes of vibration. Arrows locate areas with large displacements. In both modes this is the same location.</u>

The undeformed shape is superimposed on displacement results. Numerical displacement results are shown; they may be used for finding relative displacements within the same mode.

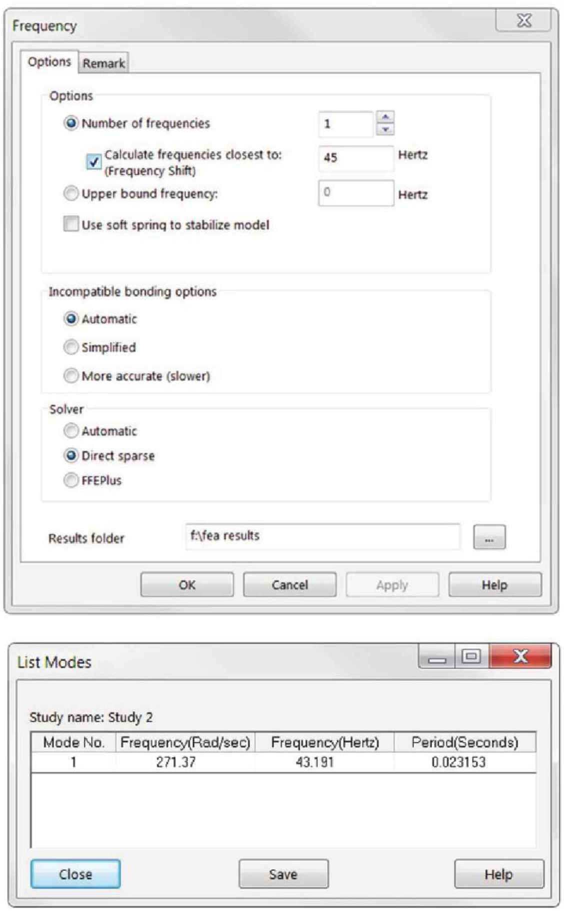
The first mode is a bending mode; the second mode is a torsional mode. Review all results to see which "weak spots" are highlighted by higher modes.

Numerical values of modal frequencies are of little use except to find out which portions of the frame are "soft". Notice that the BAJA FRAME will be stiffened by installing the engine, transmission, suspension, steering and driver's seat. At the same time these components will also add mass. Therefore, at this point the modal analysis provides only qualitative results.

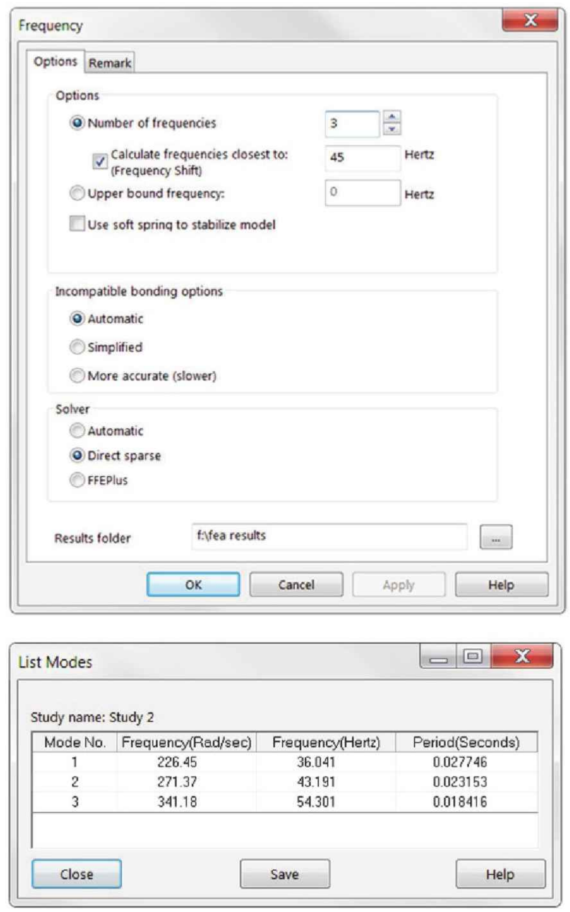
Based on these qualitative results, Western Engineering SAE BAJA team decided to reinforce locations indicated in Figure 8-2. The redesign was important for the driver's safety because these locations are critically important in the case of a roll-over.

We'll use the BAJA FRAME model to introduce **Frequency Shift**. This solver option allows for calculating frequencies clustered around the specified frequency value. It may be used to eliminate calculation of 0Hz frequencies in models where Rigid Body Modes are present.

Copy the completed study into two more studies: *Study 2* and *Study 3* and try out the **Frequency Shift** option using the settings shown in Figure 9-3.



Study 2
Frequency Shift 45Hz
1 mode requested



Study 3
Frequency Shift 45Hz
3 modes requested

<u>Figure 9-3: Solver Options with Frequency Shift selected (top) and corresponding list of calculated modes (bottom).</u>

Modes with frequencies clustered around the specified Frequency Shift value are calculated.

A **Frequency Shift** of 45Hz with just one mode requested finds the closet mode (43.2Hz) which is the torsional mode. A **Frequency Shift** with three modes requested finds modes 36Hz, 43.2Hz, and 54.3Hz. In both cases all rigid body modes are eliminated from the solution.

The **Frequency Shift** solver option is available only for the **Direct Sparse** solver. Notice that frequency results are slightly different between the first study run without **Frequency Shift** and studies *Study 1* and *Study 2* executed with this option. This is because *Study 1* found 12 modes of vibration and results have accumulated higher numerical error as compared to *Study 2* which has found one mode and *Study 3* which has found three modes.

10: Modal analysis – a diagnostic tool

Topics covered

- Modal analysis used to detect problems with restraints
- Modal analysis used to detect connectivity problems
- Rigid Body Motions of assemblies

The ability of modal analysis to detect Rigid Body Motions can be used to diagnose modeling problems such as insufficient restraints or disconnected parts. If static analysis fails with an "insufficient restraints" or similar error message, you may still run a modal analysis on the same model and identify the direction(s) of movement corresponding to the 0Hz mode(s). This will identify the direction where restraints are missing.

Procedure

The objective of the analysis of the PLIERS assembly is to find stresses in the arms of the pliers squeezing a plate as shown in Figure 10-1.

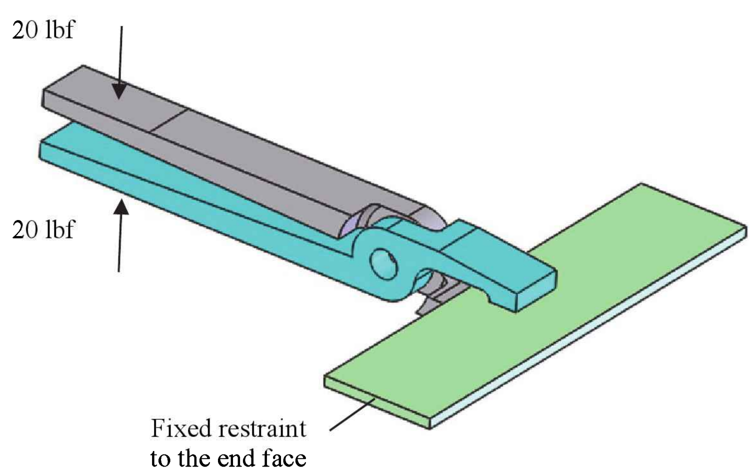


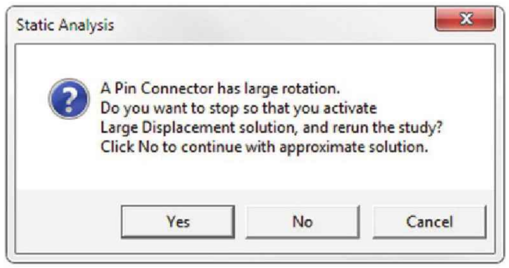
Figure 10-1: A plate being held by pliers is modeled in the PLIERS assembly.

Two arms of the pliers are connected with a pin connector. The arms are loaded with a normal force of 20lbf applied to the split faces as shown.

Open the assembly model PLIERS; make sure it is in its *01 error* configuration. The assembly model has a static study *01stress analysis* defined. The two arms are connected with a pin connector and loaded with 20lbf each; the plate has a fixed support applied to one side (Figure 10-1).

An attempt to solve the study produces an error message for which the exact wording depends on the type of solver used, but the meaning is the same: the model is not sufficiently restrained.

FFEPlus
solver message



Direct Sparse
solver message

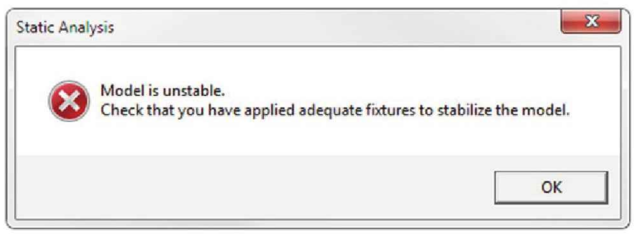


Figure 10-2: Error message issued by solvers due to Rigid Body Motions detected in the model.

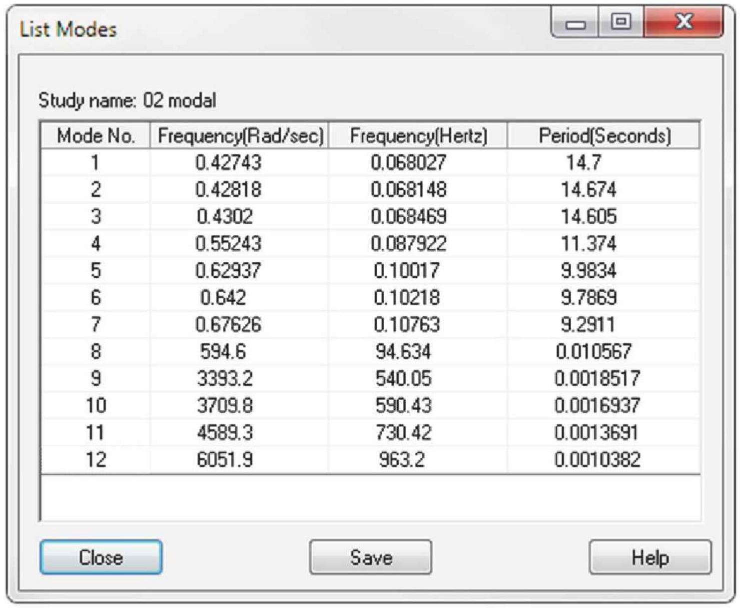
Both windows have the same message: there is a problem with restraint in the model.

The lack of "adequate fixtures" reported by both solvers is related to the presence of **Rigid Body Motions** (RBM) in the model; we will use modal analysis to find them.

Create a **Frequency** study titled *02 modal;* copy the **Fixtures** and **Connectors** from the failed study *01 stress analysis*.

At this point we know that there are unintentional RBMs in the model but we don't know how many are there; define the number of frequencies in study *02 modal* as 12.

Obtain the solution and open the **List Modes** window to identify the number RBMs in the model (Figure 9-3).

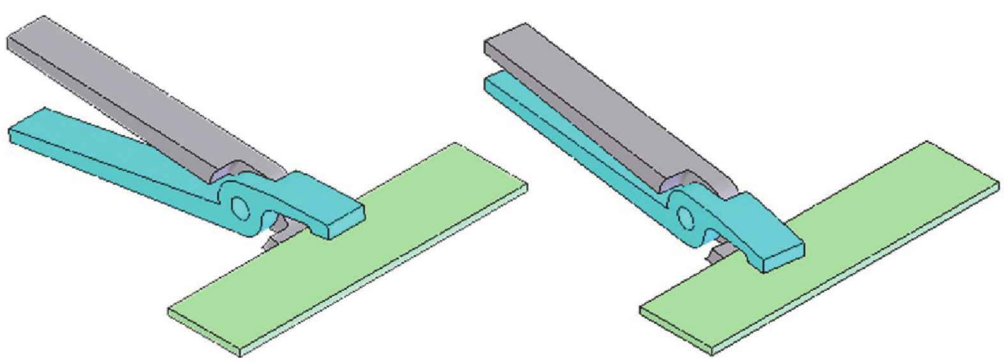


Mode No.	Frequency(Rad/sec)	Frequency(Hertz)	Period(Seconds)
1	0.42743	0.068027	14.7
2	0.42818	0.068148	14.674
3	0.4302	0.068469	14.605
4	0.55243	0.087922	11.374
5	0.62937	0.10017	9.9834
6	0.642	0.10218	9.7869
7	0.67626	0.10763	9.2911
8	594.6	94.634	0.010567
9	3393.2	540.05	0.0018517
10	3709.8	590.43	0.0016937
11	4589.3	730.42	0.0013691
12	6051.9	963.2	0.0010382

<u>Figure 10-3: Modes 1 through 7 have near zero frequency.</u>

The number of RBMs in the model is seven; the first seven modes do not have a zero frequency because of the combined effect of discretization and solution error.

Before we address the surprising number of seven RBMs, animate these seven modes; the second and the fifth modes are shown in Figure 10-4.



Mode 2
Two arms move relative to each other

Mode 5
Two arms move in union

<u>Figure 10-4: Deformed plots of mode 2 and mode 5.</u>

An animation demonstrates that the arms move as rigid bodies either relatively to each other or in union.

An animation of all Rigid Body Modes demonstrates that the arms are disconnected from the flat plate. Review the model to find small gaps between the jaws and the plate. This is the cause of "insufficient restraints" in the model and, consequently the failed solution.

The first mode where the deformation is present is mode 8 shown in Figure 10-5.

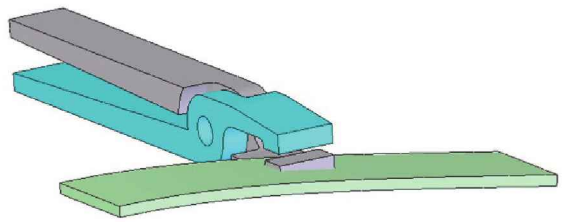


Figure 10-5: Deformed plot of mode 8.

The flat plate vibrates as a cantilever beam and the pliers don't move.

We have demonstrated that modal analysis can be used to identify problems with missing restraints. Now we need to explain why there are seven and not six Rigid Body Modes in the model.

The maximum number of RBMs a rigid body may have is 6, this happens when it has no restraints at all. In this model there are two unsupported bodies: the two arms. These two bodies would have the total of 12 RBMs if they were not connected by the pin. The pin connector eliminates 5 Degrees of Freedom (DOF) allowing only one DOF which is the rotation about the axis of the pin connector.

By allowing only 1 DOF, the pin connector removes 5 DOFs from the connected arms. Therefore, the resultant number of DOFs of the connected two arms is 12-5=7. These 7 DOFs produce 7 RBMs; remember that we are talking about DOFs of rigid bodies.

You may want to switch to the *02 correct* assembly configuration. This configuration uses a thicker flat plate which touches both arms. The touching faces are bonded by the automesher and the FEA model behaves as one solid body avoiding the problem of missing connectivity.

The problems with the PLIERS assembly have been identified using modal analysis which worked as a tool to detect RBMs. This in turn led us to the problem of disconnected faces. An alternative approach would be to use a **Static** study with the option **Use soft springs to stabilize model** selected.

Open the part titled FLAT, which is a component of the PLIERS assembly. We'll use it to demonstrate an application of modal analysis to identify a problem with restraints. The model comes with two studies defined: *01 static* and *02 modal*, both ready to run. An attempt to run *01 static* study produces error message shown in Figure 10-6.

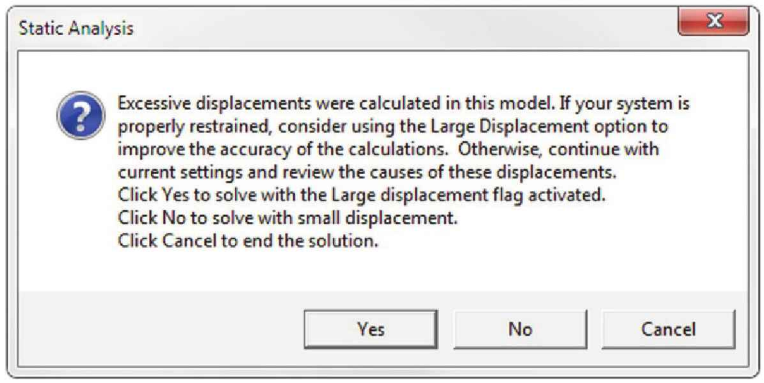
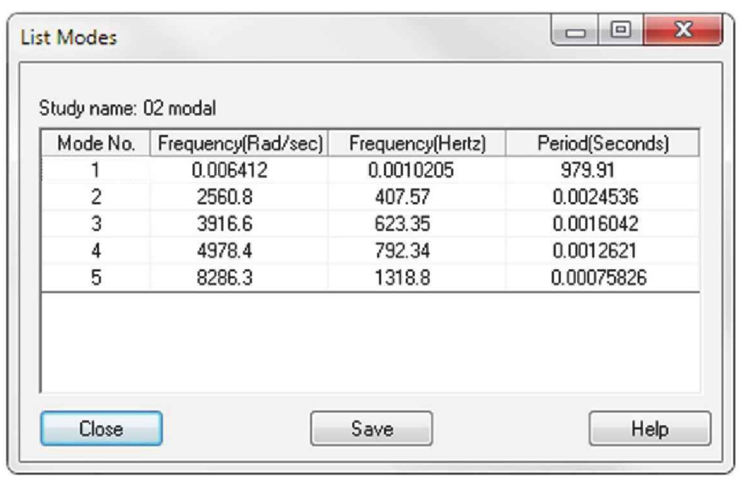


Figure 10-6: Solver error message in study *01 static.*

Error message identifies a problem with excessive displacements.

Selecting **Yes** fails the solution, selecting **No** produces clearly incorrect results with ridiculously large displacements.

The solution of study *02 modal* completes without an error message but the **List Modes** window shows the first mode with a frequency of 0Hz (Figure 10-7).



Mode No.	Frequency(Rad/sec)	Frequency(Hertz)	Period(Seconds)
1	0.006412	0.0010205	979.91
2	2560.8	407.57	0.0024536
3	3916.6	623.35	0.0016042
4	4978.4	792.34	0.0012621
5	8286.3	1318.8	0.00075826

Figure 10-7: Results of *02 modal* study.

Mode 1 is a Rigid Body Mode.

Animation of mode 1 reveals the problem: the restraint is not fixed even though it is called **Fixed**. The model has one Rigid Body Mode (RBM) which is rotation about the edge where the **Fixed** restraint has been defined (Figure 10-8).

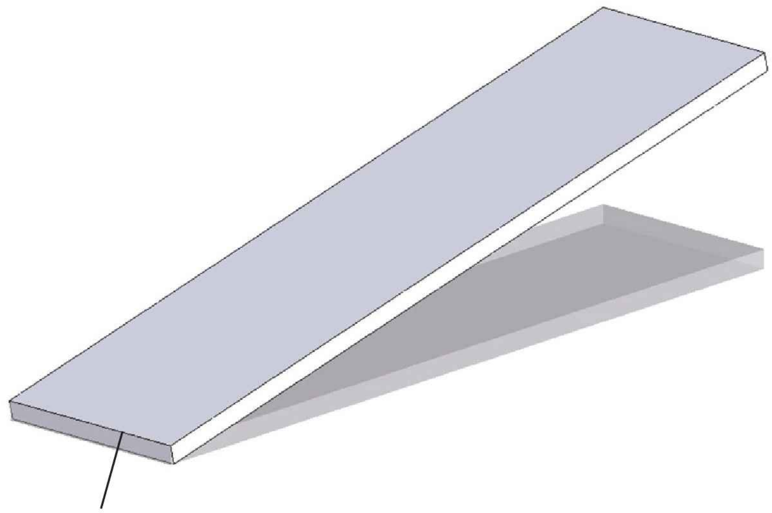


Figure 10-8: Shape of mode 1.

Even though we use the terms "Undeformed model" and "Deformed model", there is no deformation at all present in the model. The model rotates about the line where the Fixed restraints have been defined.

The reason why an RBM is present in model where the **Fixed** restraint has been applied is that solid elements have three degrees of freedom per node and these are all translations. Consequently nodes of solid elements can't generate a moment reaction necessary to support the model and model rotates about the line of support.

These are two simple examples of using modal analysis as a diagnostic tool. Both are based on the ability of modal analysis to identify rigid body modes which must not be present in a structural analysis.

11: Harmonic excitation of discrete systems

Topics covered

- Steady state harmonic excitation
- Frequency sweep
- Displacement base excitation
- Velocity base excitation
- Acceleration base excitation
- Resonance
- Modal damping

In the previous chapters, we studied different applications of modal analysis which investigated structures' propensity to vibrate but were not vibration analyses per se. In chapter 1 we have introduced the **Modal Superposition Method** which is a ubiquitous tool in Vibration Analysis. We will be using the **Modal Superposition Method** in this chapter and throughout the rest of this book; therefore it deserves this short review.

Modal Superposition Method

The **Modal Superposition Method** (MSM) represents a vibration response of a structure by using the superposition of responses that characterize Single Degree Of Freedom (SDOF) systems. The natural frequencies and directions of vibration of these SDOF systems correspond to the natural frequencies of the analyzed structure. The number of SDOFs contributing to a dynamic response is equal to the number of modes calculated by a pre-requisite modal (frequency) analysis.

How many modes should be calculated to represent a vibration response using the MSM? The first few modes are the most important, but the exact number of required modes is not known prior to analysis. Ideally, one should use a convergence process to demonstrate that increasing the number of modes past a certain number no longer significantly affects results.

Vibration analysis based on the MSM may be categorized into two major types: **Time Response** analysis and **Frequency Response** analysis. **Time Response** analysis is called **Modal Time History** in **SolidWorks Simulation**; a **Frequency Response** analysis is called **Harmonic**. We'll be alternating between these terms just like we alternate between the terms **Modal** analysis and **Frequency** analysis.

Time Response (Modal Time History) analysis

In a **Time Response** analysis, the applied load is an explicit function of time. Damping is taken into consideration and the vibration equation appears in its full form:

$$[M]\ddot{d} + [C]\dot{d} + [K]d = F(t)$$

Where:

[M]	mass matrix
[C]	damping matrix
[K]	stiffness matrix
F(t)	vector of nodal loads (this vector is a function of time)
d	unknown vector of nodal displacements

A **Time Response** analysis requires the definition of a damping coefficient which is most often expressed as a percentage of critical damping. Readers are referred to (9) listed in Chapter 21 for numerical values of damping coefficients.

A **Time Response** analysis is used to model events of a short duration. A typical example would be an analysis of a structure's vibrations due to an impact load or acceleration applied to the base (called base excitation). Results of the **Time Response** analysis capture both the transient response and steady state vibration response.

Frequency Response (Harmonic) analysis

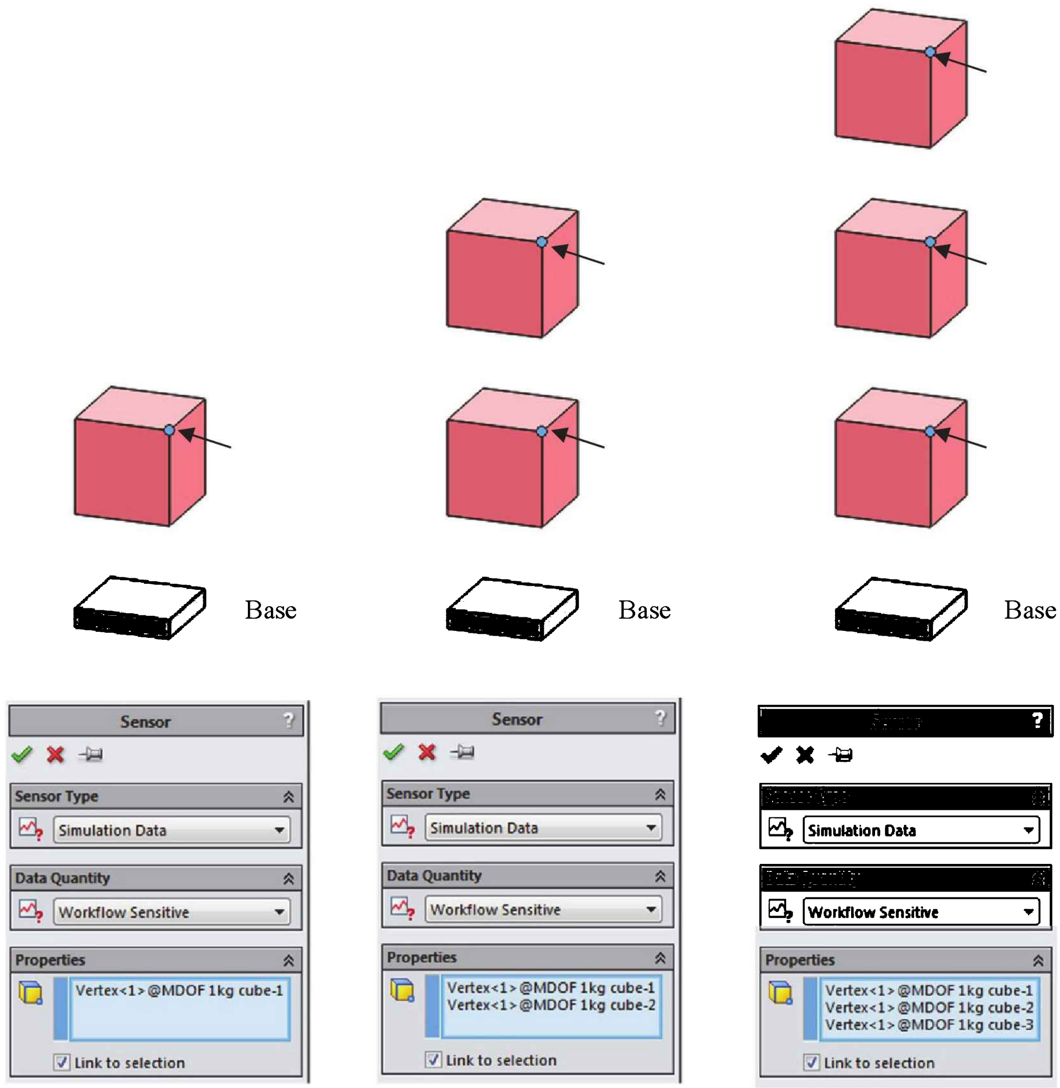
A **Harmonic** analysis assumes that the load is a function of frequency rather than being directly dependent on time as is the case of a **Time Response** analysis.

$$[M]\ddot{d} + [C]\dot{d} + [K]d = F(Asin(\omega t) + Bcos(\omega t))$$

A **Frequency Response** analysis models a structure's response to a forced excitation or base excitation (excitation applied to its supports) that is a harmonic function of time. It is assumed that the excitation frequency changes very slowly, hence the alternative name **Steady State Harmonic Response** is often used for this type of analysis. A **Frequency Response** analysis also uses the modal superposition method and requires that damping be defined, usually as a percentage of critical damping.

A typical application of a **Frequency Response** analysis is a simulation of a shaker table test discussed in Chapter 12.

In this chapter we introduce both **Time Response** and **Frequency Response** analyses using a simple discrete vibrating system. Open assembly model MDOF and review the three configurations *1DOF, 2DOF, and 3DOF* as shown in Figure 11-1. The same figure shows sensor locations in each configuration.



Configuration *1DOF* Configuration *2DOF* Configuration *3DOF*

<u>Figure 11-1: Assembly model MDOF in three configurations; all cubes are identical, arrows indicate sensor positions in each configuration.</u>

Verify that the mass of each cube is 1kg.

Cubes are connected by linear springs. The stiffness of each spring is 10000N/m. Each cube may move only in the vertical direction (Figure 11-2).

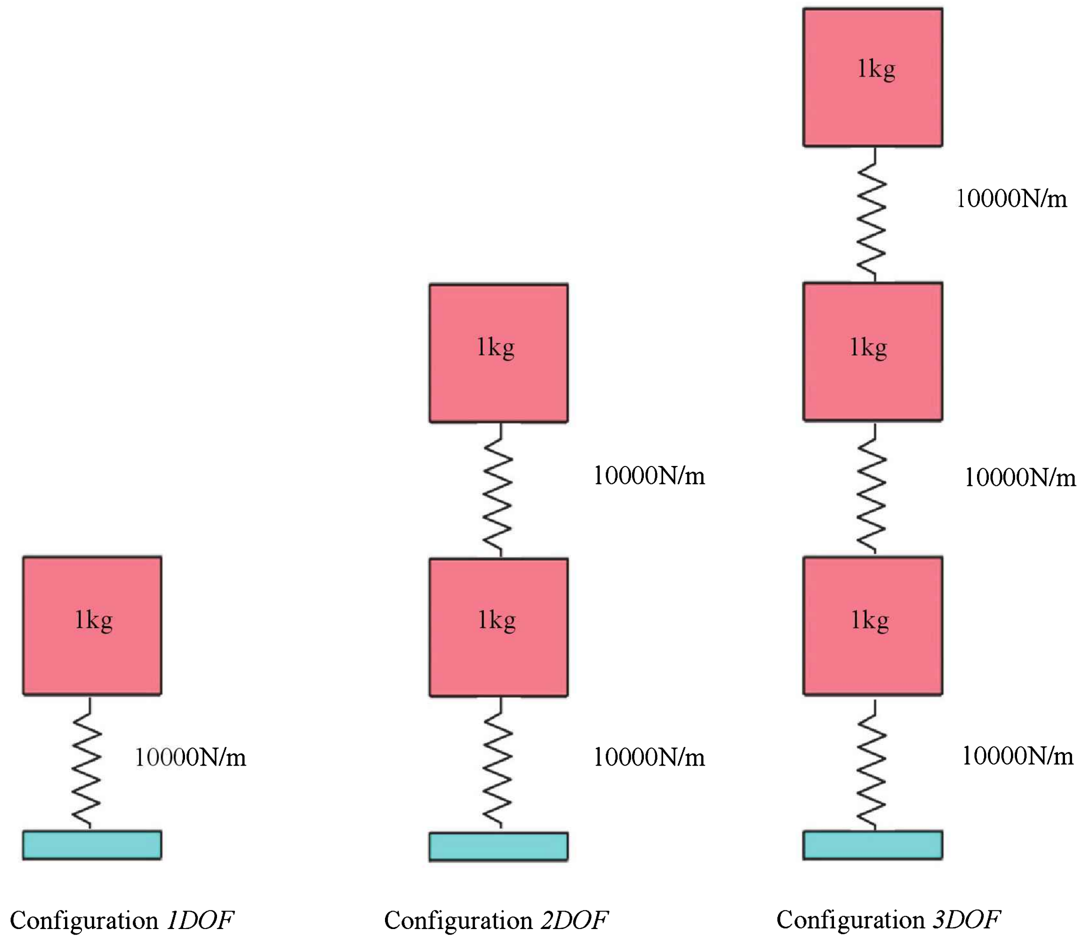


Figure 11-2: Assembly components are connected with linear springs.

All springs have the same stiffness of 10000N/m.

We'll use the *MDOF* assembly model to run as many as ten studies:

Study name	**Study type**	**Configuration**	**Description**
01 1DOF modal	Frequency	*1DOF*	Modal analysis
02 2DOF modal	Frequency	*2DOF*	Modal analysis
03 3DOF modal	Frequency	*3DOF*	Modal analysis
04 1DOF disp fr	Linear Dynamic Harmonic	*1DOF*	Frequency Response analysis with displacement base excitation
05 1DOF vel fr	Linear Dynamic Harmonic	*1DOF*	Frequency Response analysis with velocity base excitation
06 1DOF acc fr	Linear Dynamic Harmonic	*1DOF*	Frequency Response analysis with acceleration base excitation
07 2DOF disp fr	Linear Dynamic Harmonic	*2DOF*	Frequency Response analysis with displacement base excitation
08 3DOF disp fr	Linear Dynamic Harmonic	*3DOF*	Frequency Response analysis with displacement base excitation
09 1DOF shock tr	Linear Dynamic Modal Time History	*1DOF*	Time Response analysis with shock load
10 1DOF initial cond tr	Linear Dynamic Modal Time History	*1DOF*	Time Response analysis with initial condition

We'll now discuss restraints, spring connectors and meshing which are common to all configurations and all studies.

All cubes have **Roller/Slider** restraints defined to all side faces (Figure 11-3).

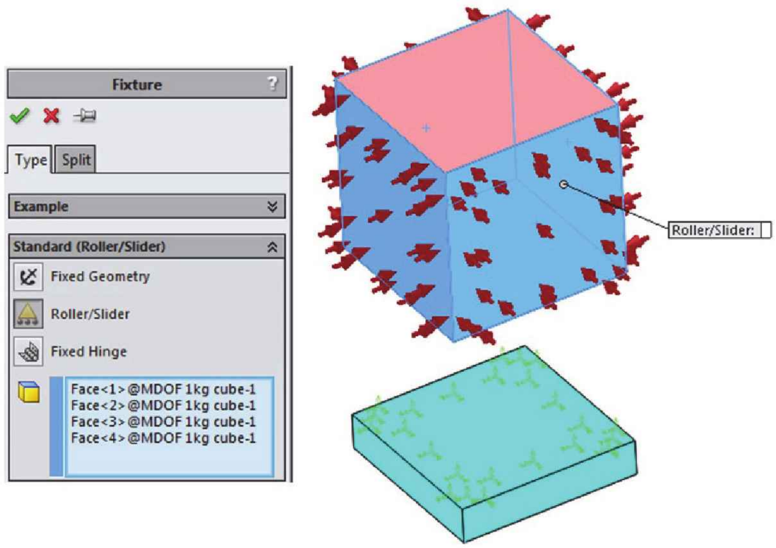


Figure 11-3: Roller/Slider restraints applied to four sides of each cube shown here for configuration 1DOF.

Roller/Slider restraints may be applied only to two perpendicular faces; this will be sufficient to limit the motion to the vertical direction only.

The base part has **Fixed** restraints applied, they may be applied to one face, selected faces or all faces.

All springs are modeled as spring connectors (Figure 11-4).

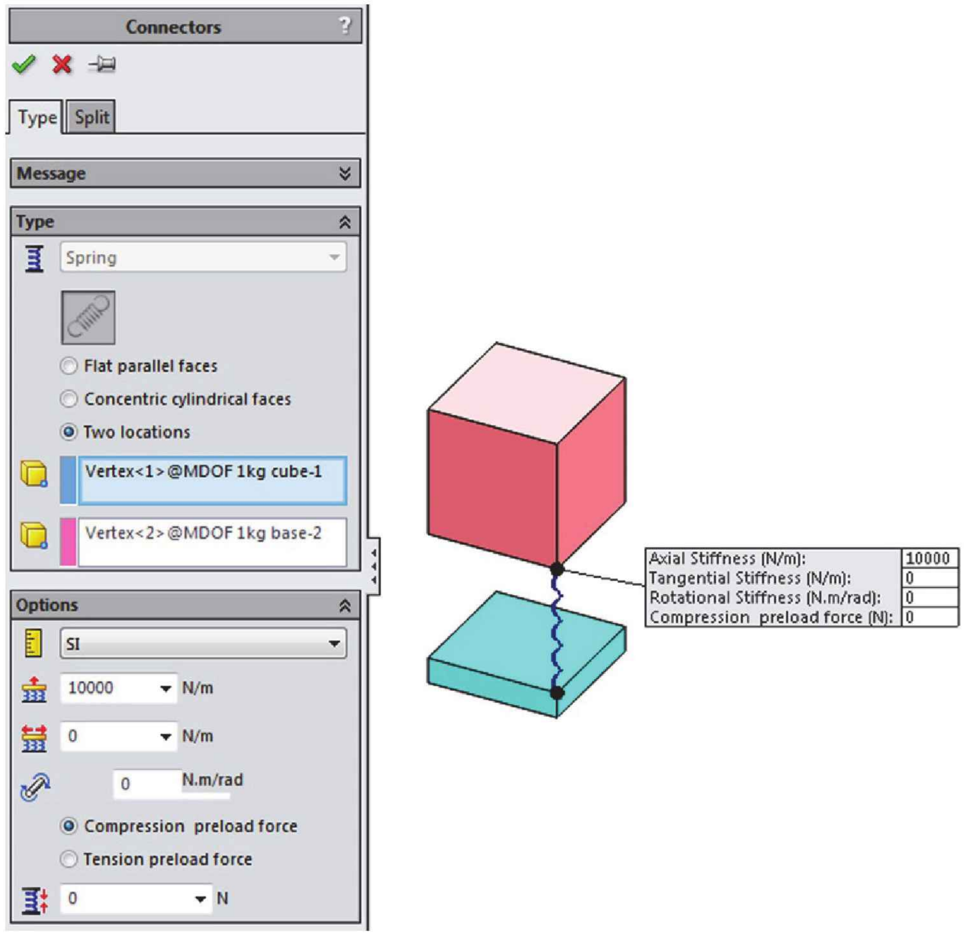


Figure 11-4: Spring connector shown here for configuration 1DOF.

10000N/m is the total stiffness.

We are modeling discrete systems. In these systems stiffness properties are fully defined by the springs and inertial properties are defined by the cubes. As we'll soon demonstrate that the cubes do not deform in these modes which are of interest to us. Therefore, the cubes and base may be meshed with a coarse mesh; use element size 20mm in all configurations.

Run **Frequency** studies on configurations *01 1DOF modal, 02 2DOF modal*, and *03 3DOF modal* to find the modes of vibration. You will find that the number of modes that are associated with the deformation of the springs corresponds to the number of degrees of freedom in each configuration. Remember, when we say "the number of degrees of freedom" we are referring to all the solid elements (cubes and base) as rigid bodies.

Results of modal analyses are summarized in Figure 11-5.

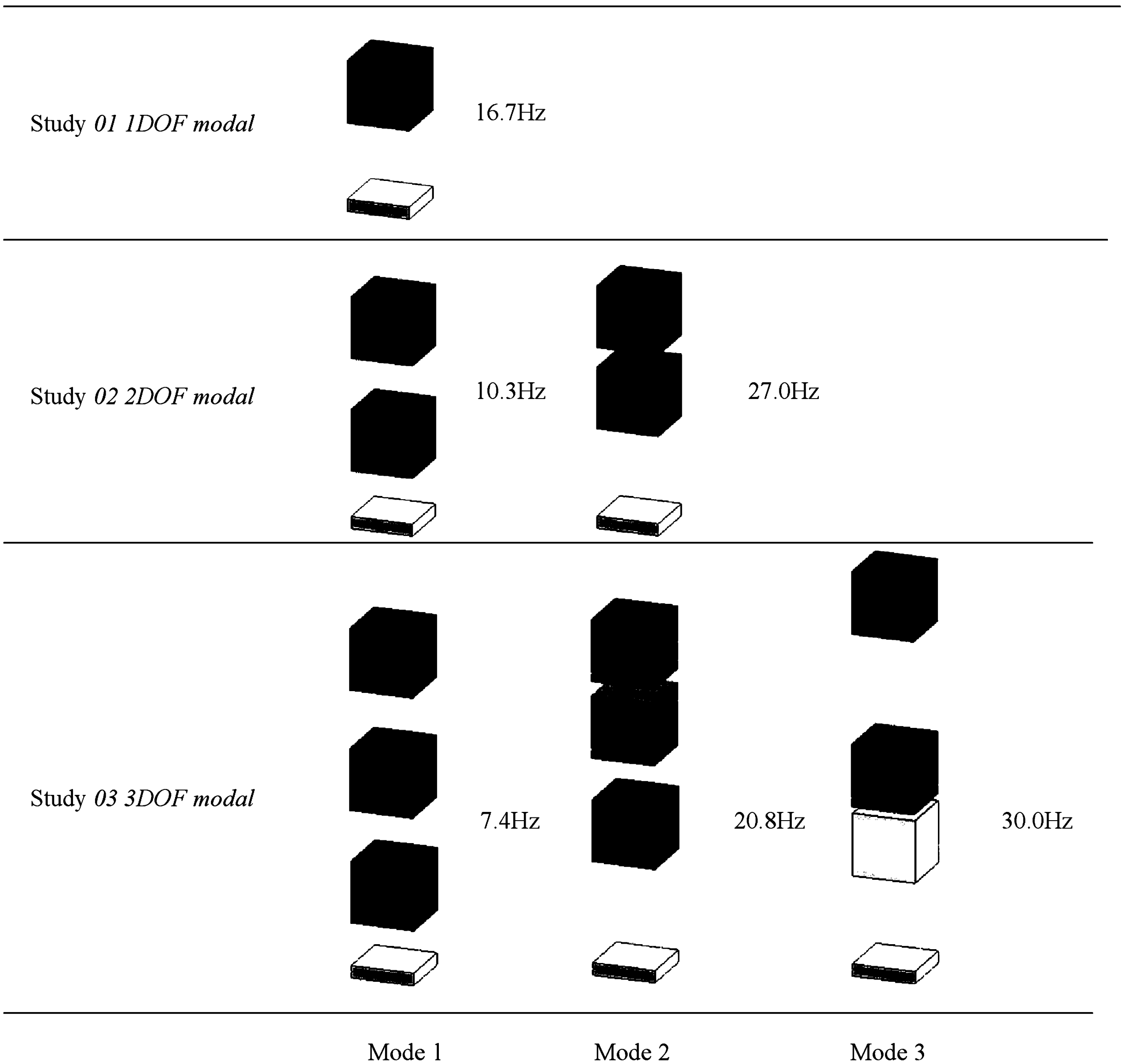


Figure 11-5: Summary of the results of modal analysis in all assembly configurations.

Configurations are arranged in rows, modes are arranged in columns.

You may find more modes to see that they are associated with deformation of the cubes and do not correspond to the discrete system we analyze here.

Configuration 1DOF has an analytical solution $\omega = \sqrt{k/m}$ = 100rad/s which is very close to the numerical solution f =16.7Hz ($\omega = 2\Pi f$).

From the results of the modal analysis presented in Figure 11-5, we learned that the range of frequencies, 0 – 30Hz, includes all natural frequencies in the system in all assembly configurations.

In the **Frequency Response** analysis that we are about to perform we subject the base to oscillations with frequencies from zero (no motion) to frequencies much higher than the highest frequency. A frequency three time as high as the natural frequency is generally considered as a "much higher" frequency; therefore the range of base excitation frequencies will be 0-90Hz. In the **Harmonic** analysis with base excitation, the base is subjected to harmonic motion as shown in Figure 11-6.

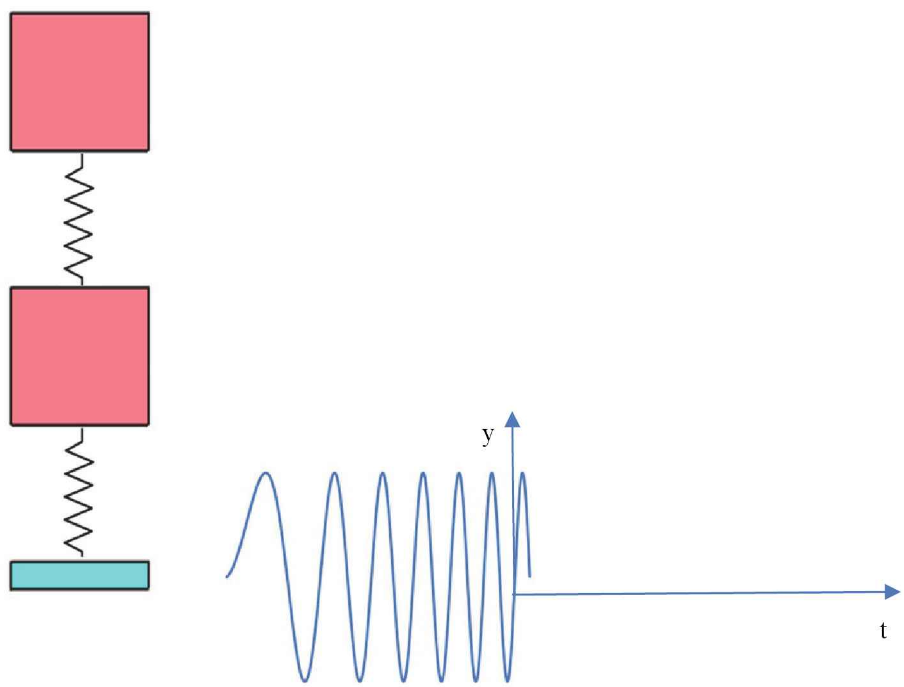


Figure 11-6: Harmonic base excitation applied to the base shown here for configuration *2DOF.*

The base performs harmonic motion with constant amplitude and increasing frequency.

.

Three different types of base excitation will be applied:

- excitation with constant displacement amplitude
- excitation with constant velocity amplitude
- excitation with constant acceleration amplitude

Start in the *1 DOF* assembly configuration and create a **Harmonic** study titled *04 1DOF disp fr* as shown in Figure 11-7. This is the first study in the series of three **Harmonic** studies: *04 1DOF disp fr, 05 1DOF vel fr,* and *06 1DOF acc fr.*

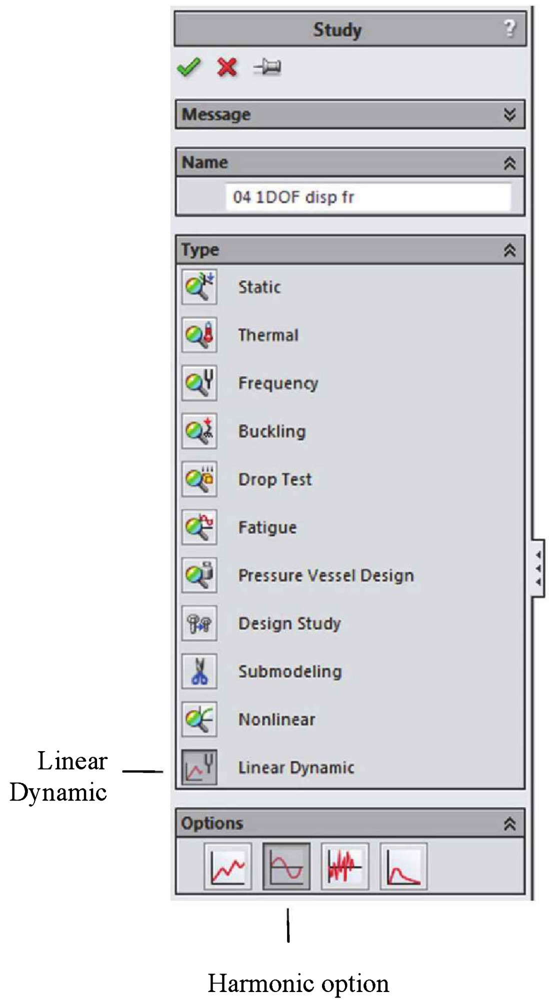
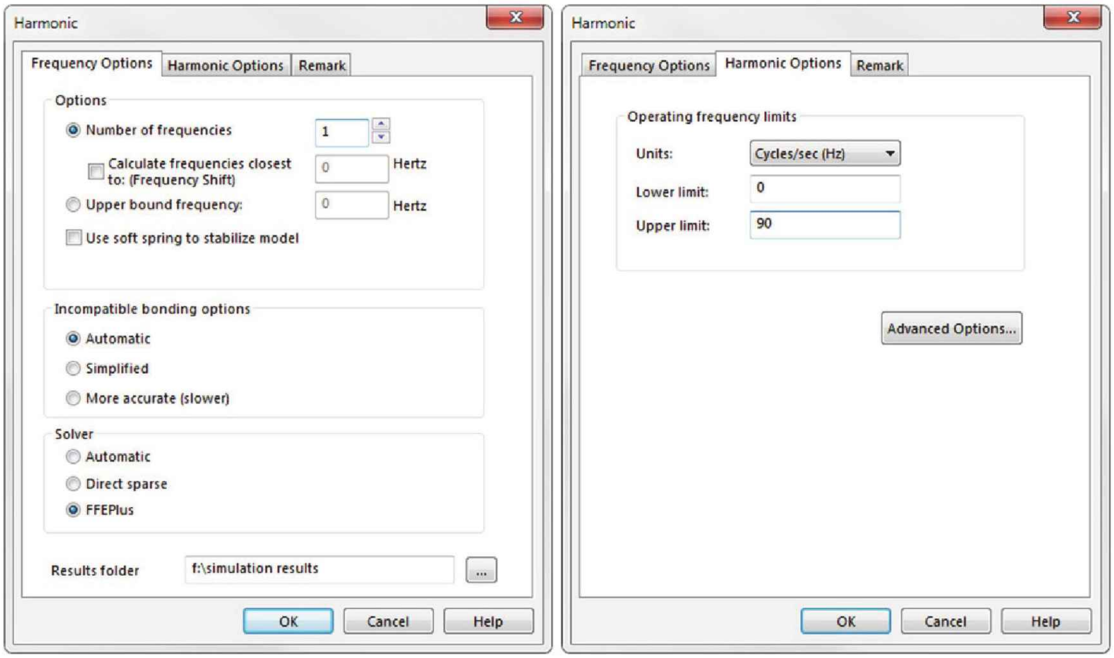


Figure 11-7: Definition of a Harmonic study.

A harmonic study is one of four options in a Linear Dynamic study.

Define study properties as shown in Figure 11-8.



Frequency Options Harmonic Options

Figure 11-8: Definition of the Harmonic study.

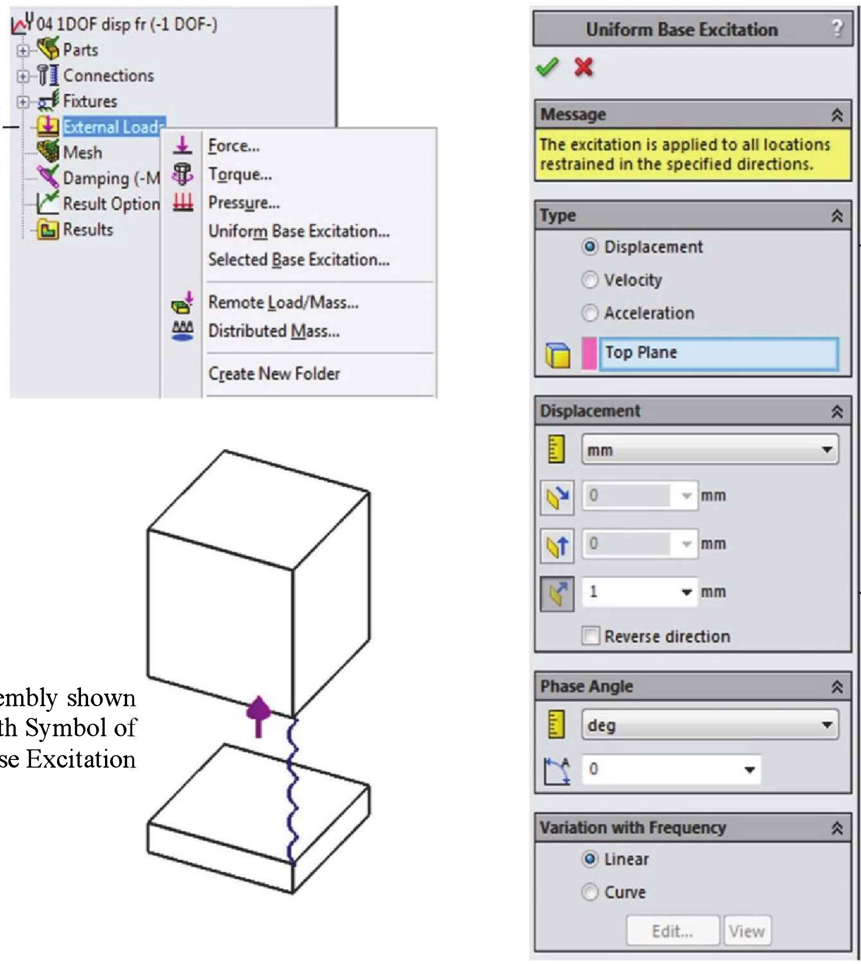
The definition consists of specifying Frequency Options and Harmonic Options.

Like all **Linear Dynamic** studies, the **Harmonic** study is based on the **Modal Superposition Method**. The **Number of frequencies** specified in **Frequency Options** gives the number of modes to be used to find the vibration response. In this exercise we analyze a discrete vibration system with one degree of freedom which had one mode of vibration. For this reason the **Number of frequencies** is 1. The **Operating frequency limits** defined in **Harmonic Options** specifies the range of frequencies to which the model will be subjected. The vibration response will be found as function of the excitation frequency slowly changing from 0 to 90Hz.

Define a **Fixed** restraint to base and **Roller/Slider** restraints the same as in the previously completed **Frequency** analyses.

Define **Base Excitation** as shown in Figure 11-9.

(1)
Right-click External Loads folder, Select Uniform Base Excitation



Assembly shown with Symbol of Uniform Base Excitation

(2)
Select Displacement

(3)
Enter 1mm as the amplitude of displacement
In the direction perpendicular to Top plane.

(4)
Leave this as Linear but notice the terminology error.
"Linear" means here that the amplitude is constant; it doesn't change with frequency

<u>Figure 11-9: Definition of Uniform Base Excitation.</u>

The wireframe display is used to clearly show the symbol of Uniform Base Excitation. The symbols of the Roller/Slider restraints applied to all four sides of the cube and symbols for the Fixed restraints applied to all faces of the base are not shown in this illustration.

Uniform Base Excitation means that the base excitation is applied through all restraints active in the specified direction. In our case it means that it is applied to the base but not to the cube, because the cube does not have any restraints is the direction of excitation. **Variation with Frequency** is defined as **Linear** but this should say **Constant**; the amplitude doesn't change with the excitation frequency.

Apply **Modal Damping** as shown in Figure 11-10.

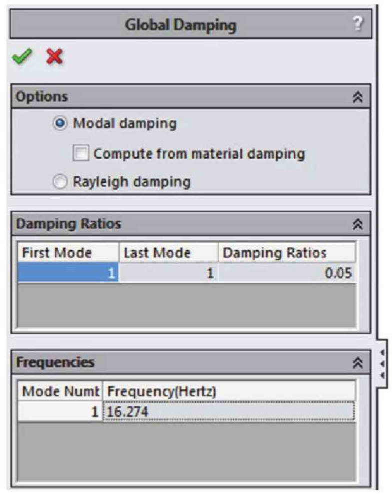


Figure 11-10: A Modal Damping of 5% is defined for the only mode present in this discrete vibrating system.

Modal damping may be defined directly or may be computed from material damping. Here, we use the first method.

A **Modal damping** of 5% means that damping in the system equals 5% of critical damping. Oscillatory motion is not possible for critical damping or higher. This definition of damping doesn't specify which model component is responsible for dissipating energy. This is why the window in Figure 11-10 is called **Global Damping**.

Define **Results Options** as shown in Figure 11-11:

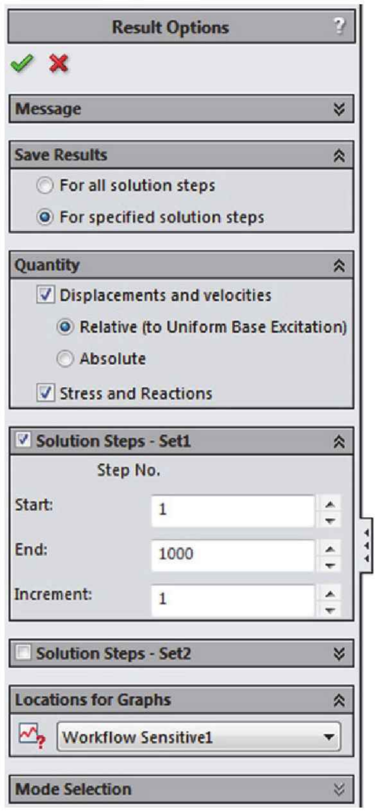


Figure 11-11: Result Options definition.

This window defines which steps will be used for creating Response Graphs. Here, all steps will be used. The number referring to the last step may be defined as higher than the actual number of preformed steps. Here, the end step is defined as step 1000 – the highest number allowed by the system.

Obtain the solution and define a **Response Graph** as shown in Figure 11-12.

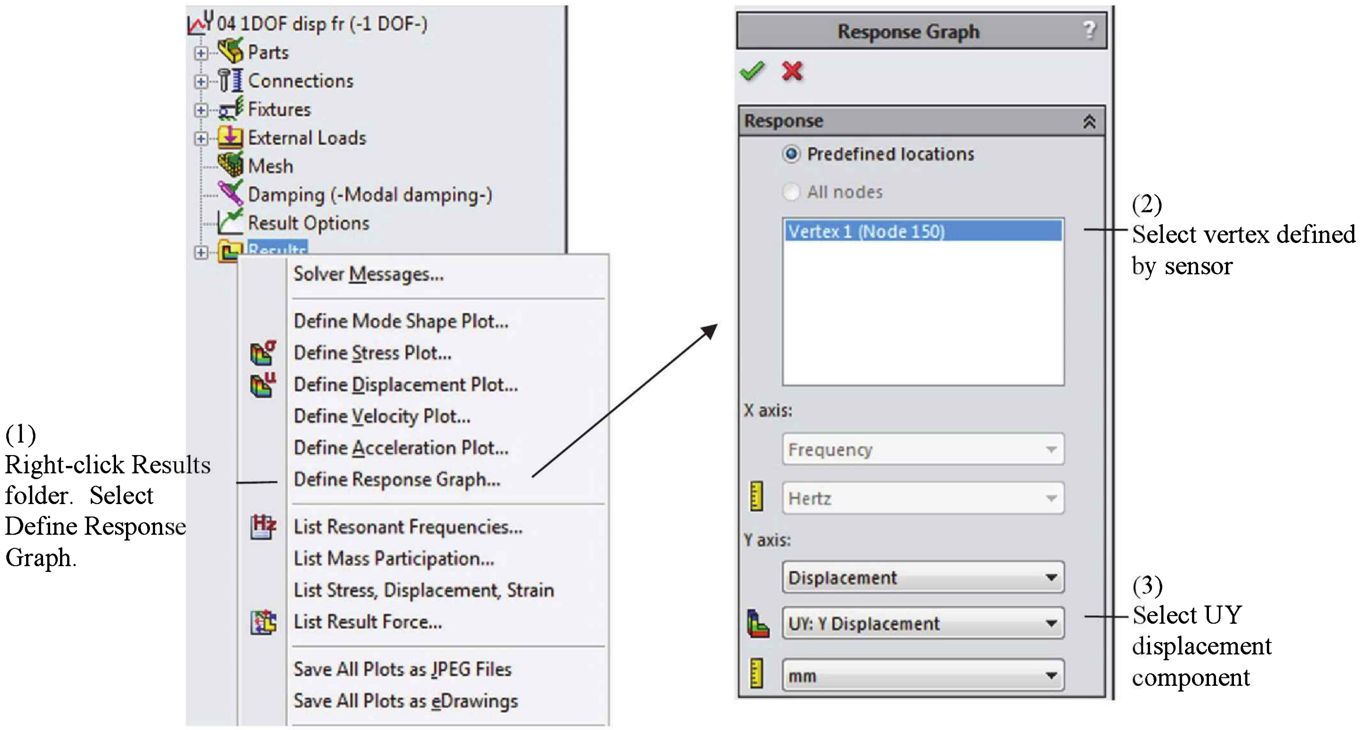
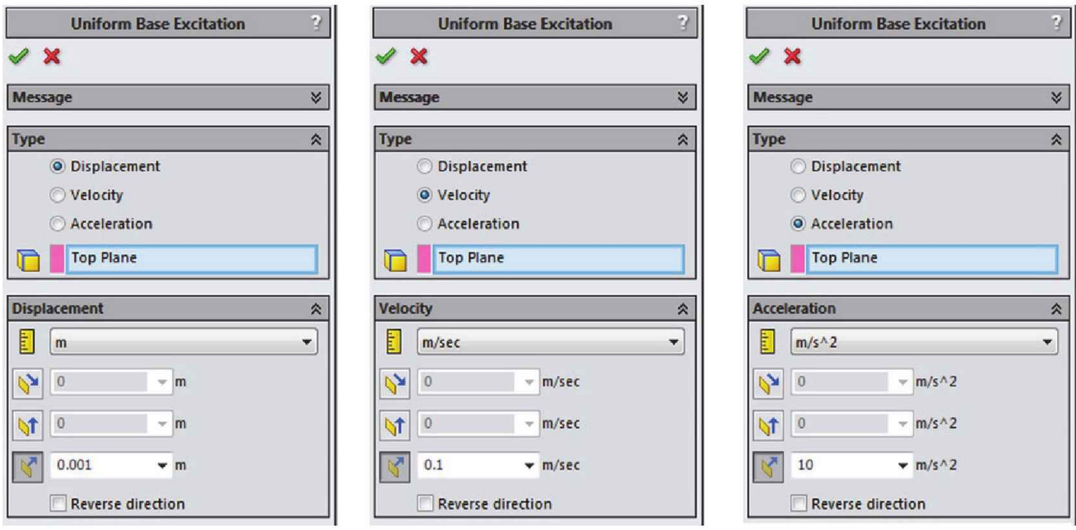


Figure 11-12: Result Options definition.

Displacement component UY is in the direction of base excitation.

Before analyzing results of study *04 1DOF disp fr*, we'll define and run two more **Harmonic** studies, then we'll analyze all the results together. Copy study *04 1DOF disp fr* into *05 1DOF vel fr*, then copy it into *06 1DOF acc fr*. Edit the two new studies as shown in Figure 11-13; the only difference between these three studies is in the base excitation definition.



Study *04 1DOF disp fr*
Displacement excitation
amplitude 0.001m

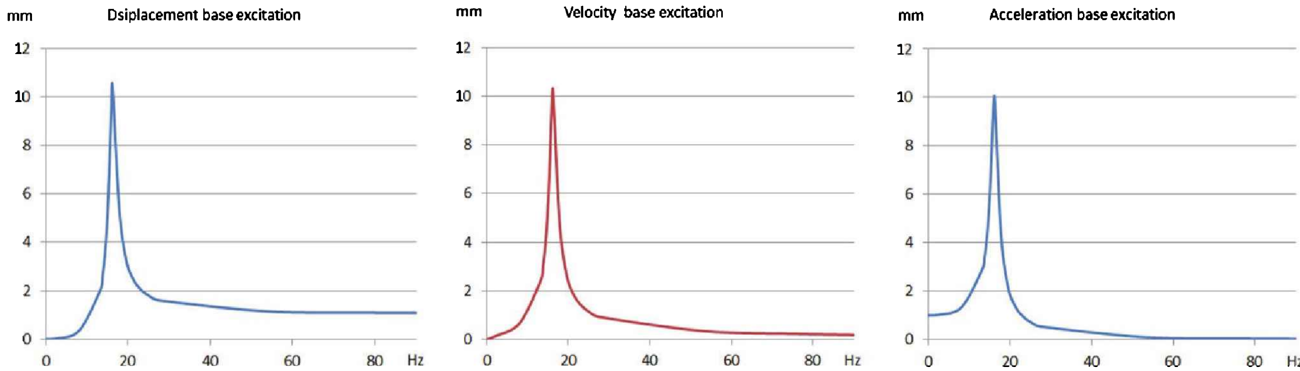
Study *05 1DOF vel fr*
Velocity excitation
amplitude 0.1m/s

Study *06 1DOF acc fr*
Acceleration excitation
amplitude 10m/s^2

Figure 11-13: Base excitation definition in three Harmonic studies with base excitation.

The left window is a repetition of Figure 11-9.

The results are summarized in Figure 11-14.



Study *04 1DOF disp fr* Study *05 1DOF vel fr* Study *06 1 DOF acc fr*

Figure 11-14: UY displacement amplitude as a function of the excitation frequency.

These graphs have been formatted in Excel.

UY displacement amplitude response graphs presented in Figure 11-14 all show one peak that corresponds to16Hz (100rad/s) which is the natural frequency of the system in the *1DOF* configuration. For low damping, as is the case in this exercise, the amplitude of displacement reaches a maximum when the excitation frequency equals the natural frequency, this is called resonance.

Let's explain why in this exercise the displacement amplitude at resonance is the same for all three types of base excitation. Displacement, velocity, and acceleration in a harmonic motion are:

$$x = A sin(\omega t)$$

$$\dot{x} = A\omega cos(\omega t)$$

$$\ddot{x} = -A\omega^2 sin(\omega t)$$

Displacement amplitude is proportional to the base excitation, be it displacement, velocity, or acceleration excitation. Considering that $\omega = 100rad/s$ is the frequency of excitation at resonance, the numerical value of acceleration amplitude is 100 times higher than the velocity amplitude and 10000 higher than displacement amplitude. Our definition of acceleration base excitation (10m/s^2), velocity base excitation (0.1m/s) and displacement base excitation (0.001m) follow the same ratio. Therefore, the displacement amplitude at resonance is the same for all three types of base excitation.

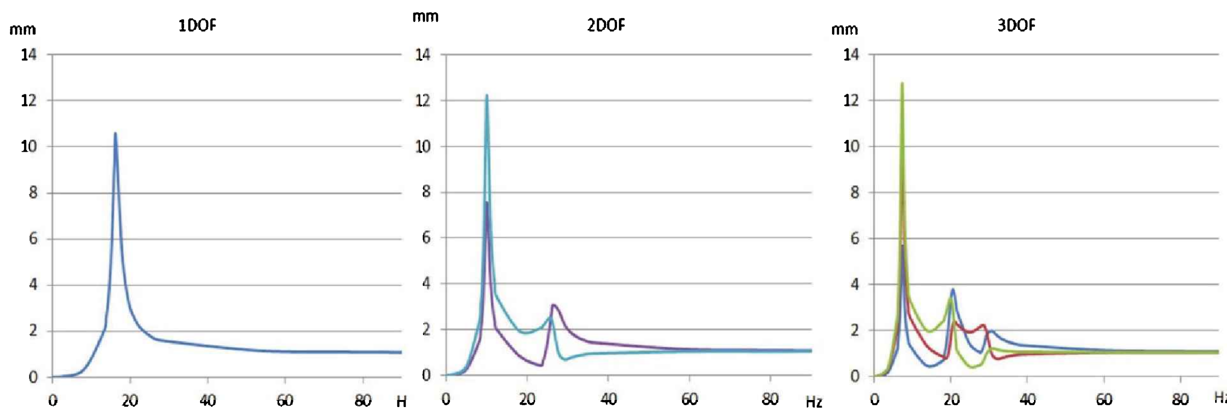
Examination of the response graphs in Figure 11-14 reveals an interesting property of displacement base excitation: the amplitude of vibration remains constant for frequencies much higher than the resonance.

We'll now conduct the analysis of the assembly in *2DOF* and *3DOF* configurations, limiting it to displacement base excitation with amplitude of 1mm and a frequency sweep of 0-90Hz. In the properties of study *07 2DOFdisp fr* define 2 modes and in the properties of study *08 3DOF disp fr* define 3 modes in **Frequency Options**.

Define a **Base Excitation** as shown in Figure 11-9. Define **Modal Damping** for all modes present in each study. Spring connectors, restraints and the mesh remain the same.

Study *07 2DOFdisp fr* has two sensors; study *08 3DOFdisp fr* has three sensors; define **Results Options** for all sensors present in each study. Obtain solutions and define UY **Response Graphs**.

The frequency responses of UY displacement amplitude for all three studies are summarized in Figure 11-15.



Study *04 1DOF disp fr*

Study *07 2DOF disp fr*

Study *08 3DOF disp fr*

Figure 11-15: UY displacement amplitude as a function of the excitation frequency for the three assembly configurations.

The graph on the left has been already shown in Figure 11-14.

The number of peaks in these three frequency sweeps illustrated in Figure 11-15 equals the number of modes of vibration present in each study. The first mode dominates the displacement amplitude response in all cases. Please review the spreadsheet MDOF.xlsx for more details impossible to show in these black and white illustrations.

We now move to a **Time Response** analysis using the same *MDOF* assembly in configuration *1DOF*. Make sure the assembly is in *1DOF* configurations and create a study titled *09 1DOF shock tr* as shown in Figure 11-16.

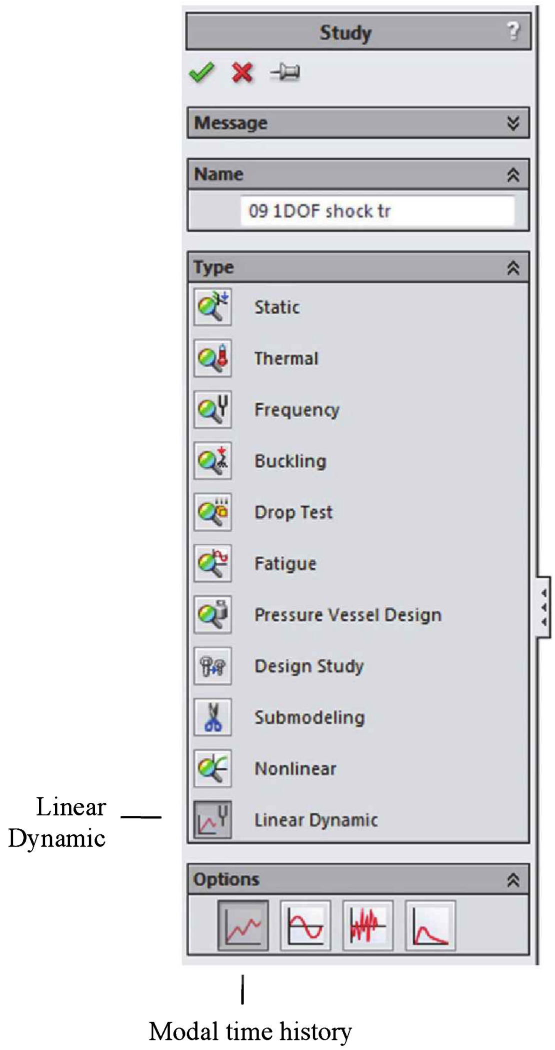
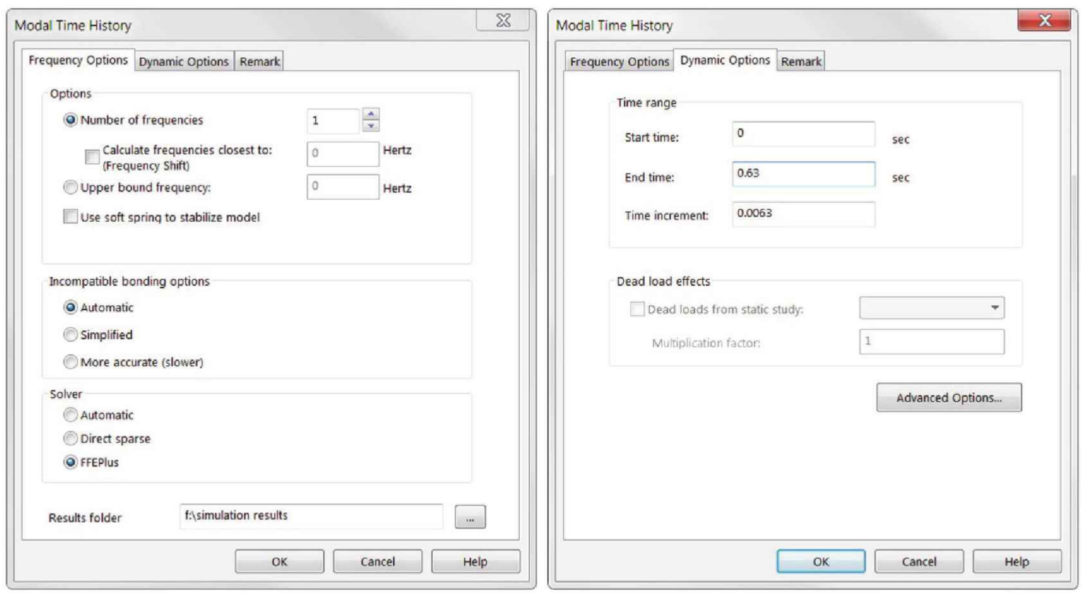


Figure 11-16: Definition of a Time response study.

A vibration response in a **Time Response** study is a function of time. In the study properties we must specify the time duration of the analysis and the time step. The duration of analysis will be 10 times the period of the mode of vibration with 10 time steps per oscillation. Remembering that the natural frequency is 100rad/s, the period of vibration is:

$$T = \frac{2\,\Pi}{\omega} = 0.063s$$

The duration of analysis is 0.63s and the time step 0.0063s. Define the properties of the **Modal Time History** study as shown in Figure 11-17.



Frequency Options   Harmonic Options

Figure 11-17: Properties of the Time response study.

Given a time duration of 0.63s and a time step of 0.0063, the analysis will complete in 100 steps.

Use the same restraints, **Results Options**, and mesh size as in study *04 1DOF disp fr*.

Damping in a **Modal Time History** analysis may be defined in two different ways: as **Modal Damping**, the same as in a **Harmonic** study, or as linear damping in a **Spring-Damper** connector.

We'll work with the same 5% modal damping as before but we'll express it as linear damping in the definition of a **Spring-Damper** connector. The critical damping in a Single Degree of Freedom oscillator is:

$$c_{cr} = 2\sqrt{km} = 200\frac{Ns}{m} \qquad k = 10000\frac{N}{m} \quad m = 1kg$$

$$c = 0.05c_{cr} = 10\frac{Ns}{m}$$

Therefore, to define damping as 5% of critical damping we can either enter 10Ns/m in the **Spring-Damper Connector** window, or as 0.05 in the **Global Damping** window (Figure 11-18).

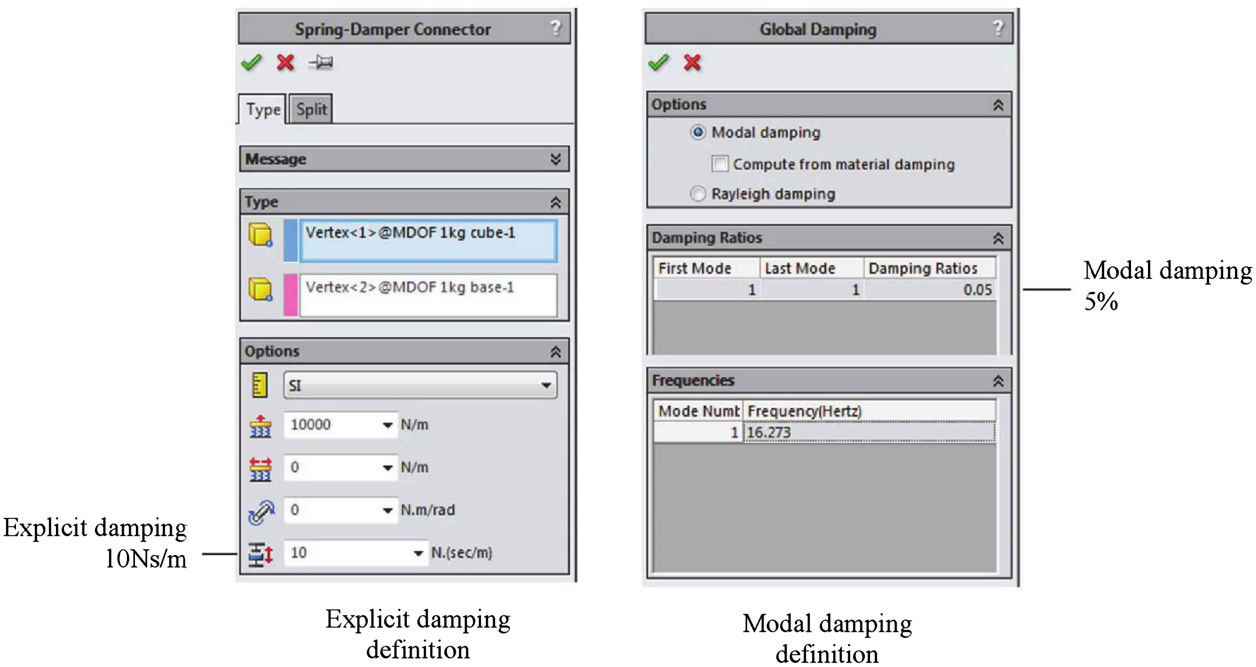


Figure 11-18: Damping can be defined explicitly (left) or as a fraction of critical damping (right). The entries in both windows define the same damping.

Define damping explicitly as a linear damping. If you define damping using both methods at the same time, the resultant damping will be 10% modal.

A **Time Response** analysis requires a load defined as a function of time. Here, we want to model the system response to a shock load, which is a load of short duration. In our example the load duration is 0.01s, the maximum load magnitude is 1000N, and the shape of the load time history curve is half a sine curve. (Figure 11-19).

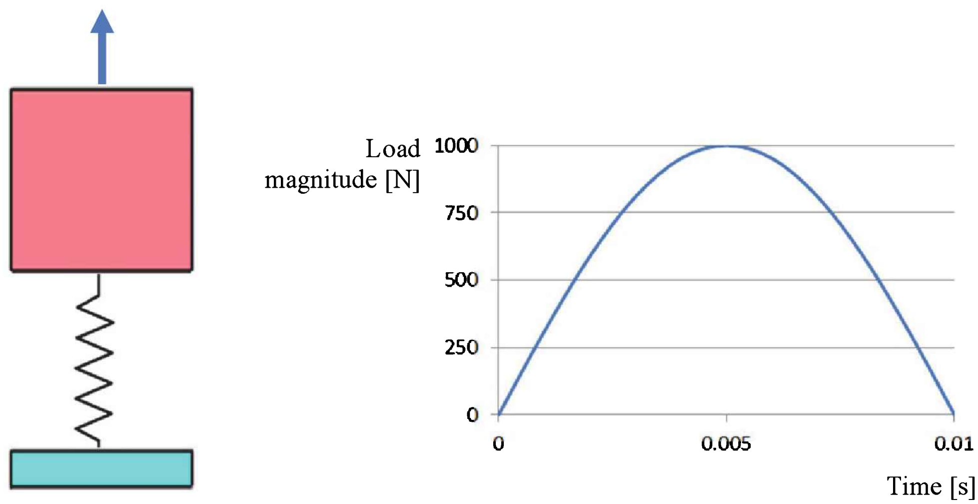
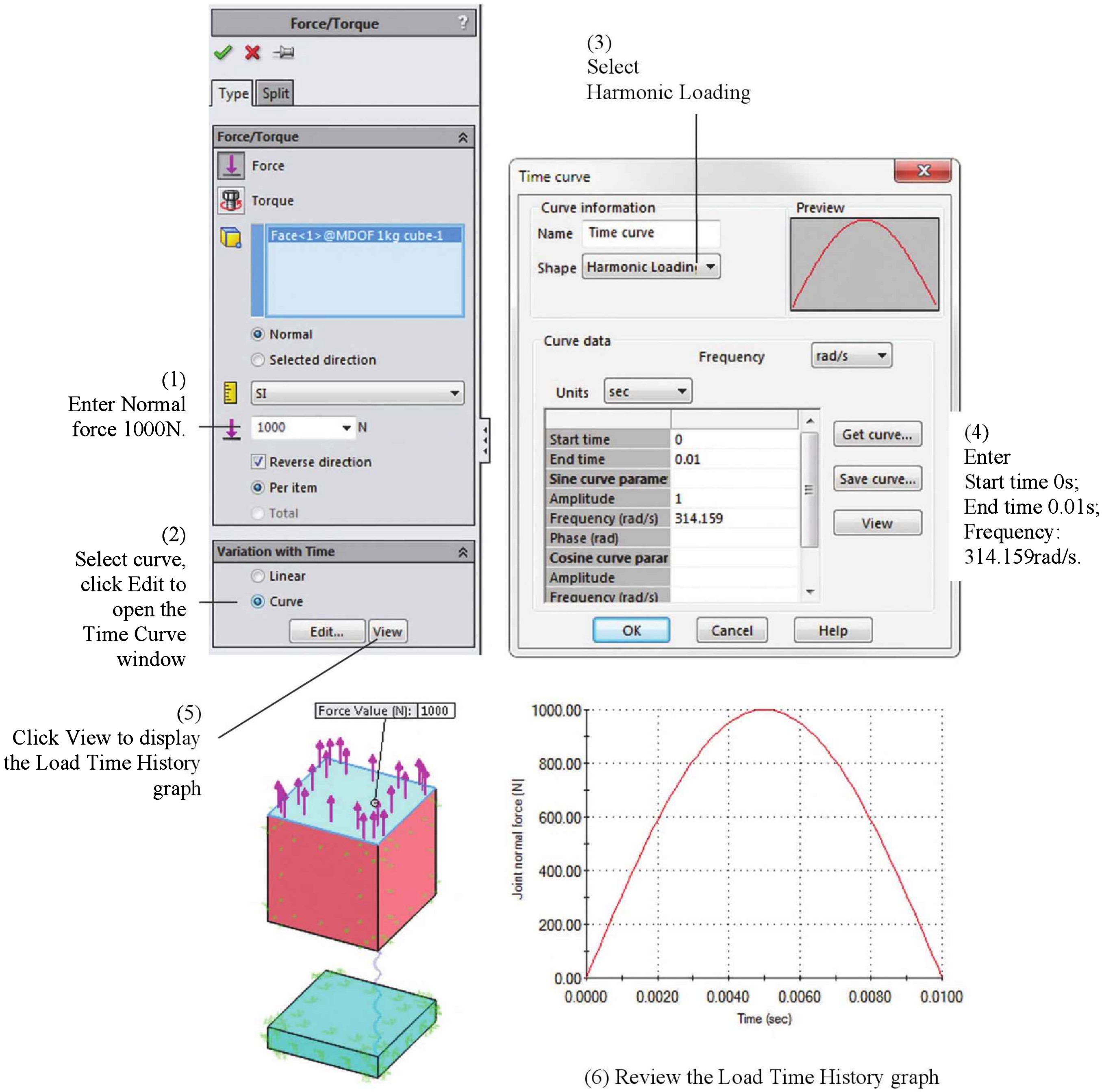


Figure 11-19: Shock load applied in the direction of motion of the 1DOF model.

Load is applied to the top face; it disappears after 0.01s.

To define this load, follow the steps in Figure 11-20.



<u>Figure 11-20: Defining load as a function of time. The force takes 0.005s to reach the maximum of 1000N, then another 0.005s to drop back to zero.</u>

Select Variation with Time as Curve (2), and click Edit to open the Time curve definition window. Define the shape as harmonic loading (3) and enter values as shown (4). Click View (5) to examine the Load time history curve (6).

Notice that neither the entry in the **Force** window (here 1000N) or the values defining the **Time curve** define the load time history on their own. The corresponding values are multiplied to calculate the force magnitude as a function of time.

The duration of the load is 0.01s (Figure 11-19), while the duration of analysis is 0.63s (Figure 11-17) meaning that the load disappears after 0.01s and the system enters into free vibration. Run *09 1DOF tr shock* study and observe that it completes in 100 steps.

Right-click the *Results* folder and follow the steps illustrated in Figure 11-21 to create a graph showing displacement in the sensor location as a function of time.

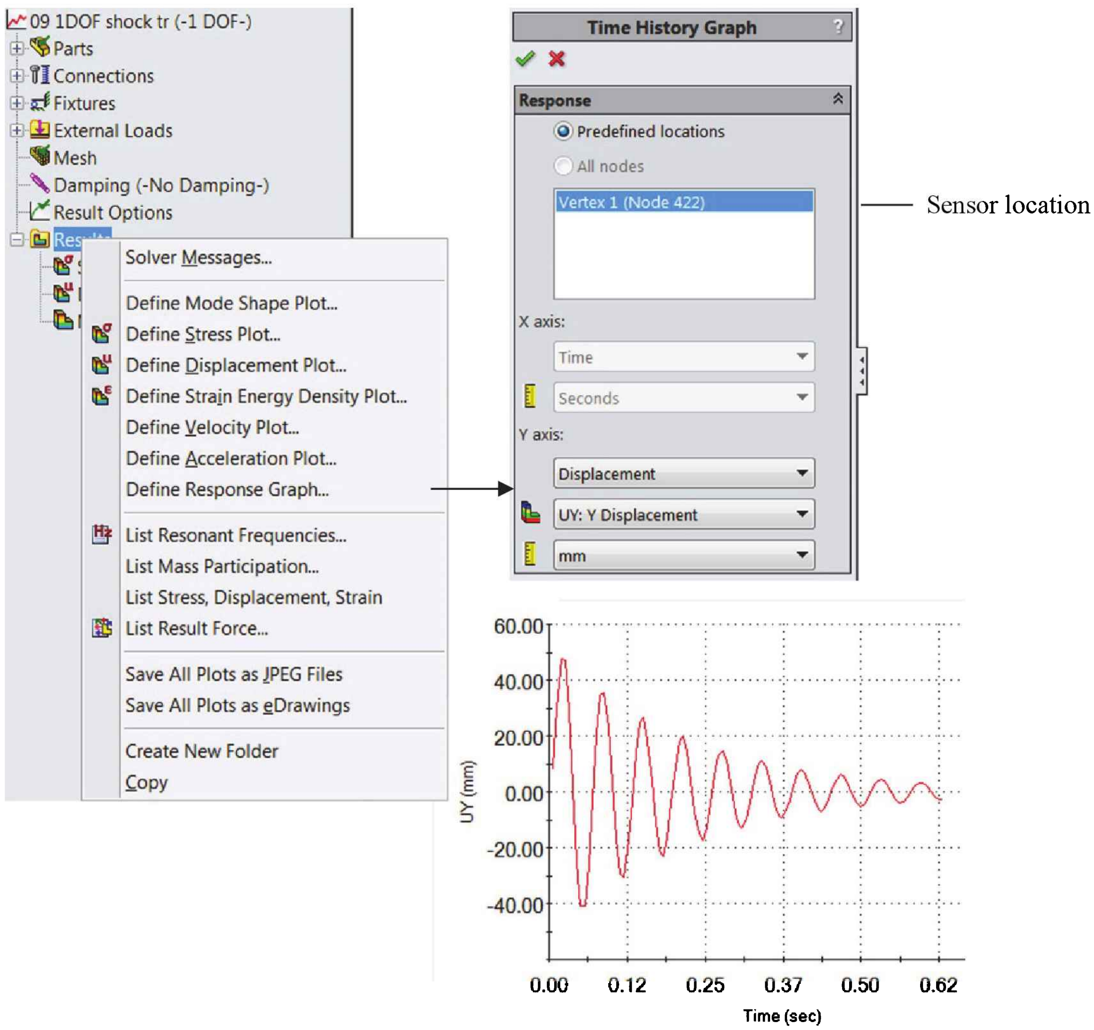


Figure 11-21: Displacement of the 1kg mass for the first 0.63s after load application.

Notice that after 0.01s, the load becomes zero and the mass performs free damped oscillations.

There is one more study left to run. Copy the last study *09 1DOF shock tr* into *10 1DOF initial cond tr*. These two studies are identical except for the method used to excite vibration. In study *09 1DOF shock tr* we used a load of very short duration and called it a Shock Load. This time dependent load acting over 0.01s generates an impulse load of 6.31Ns (Figure 11-22).

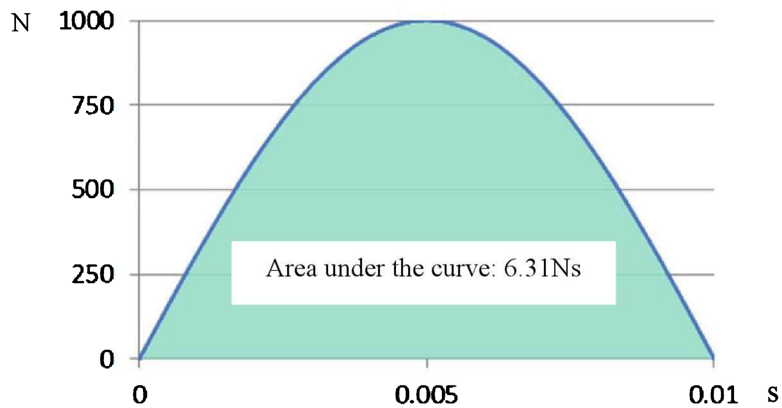


Figure 11-22: Impulse of the time dependent load is equal to the area under the curve.

See spreadsheet MDOF IMPULSE.xlsx.

Considering that:

$$mv_0 = Ft \qquad where\ \ m = 1kg \qquad Ft = 6.31Ns$$

we find the initial velocity of the 1kg mass that gives an equivalent excitation as the above time dependent load:

$$v_0 = \frac{Ft}{m} = 6.31\frac{m}{s}$$

In study *10 1DOF initial cond tr* we set the system in motion by using an initial condition rather than a time dependent load.

Notice that after 0.01s, the load becomes zero and the mass performs free damped oscillations.

Delete the load that was copied from study *09 1DOF shock tr* and define an initial condition as shown in Figure 11-23.

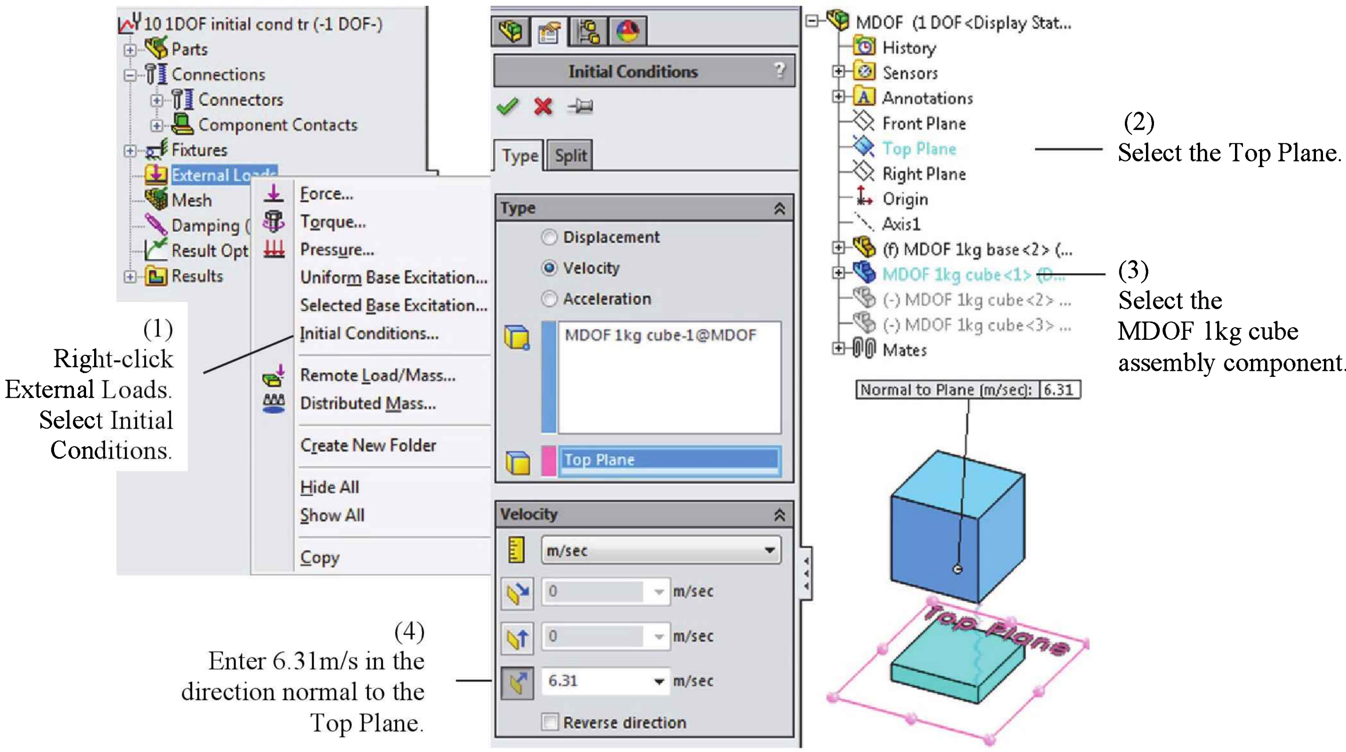
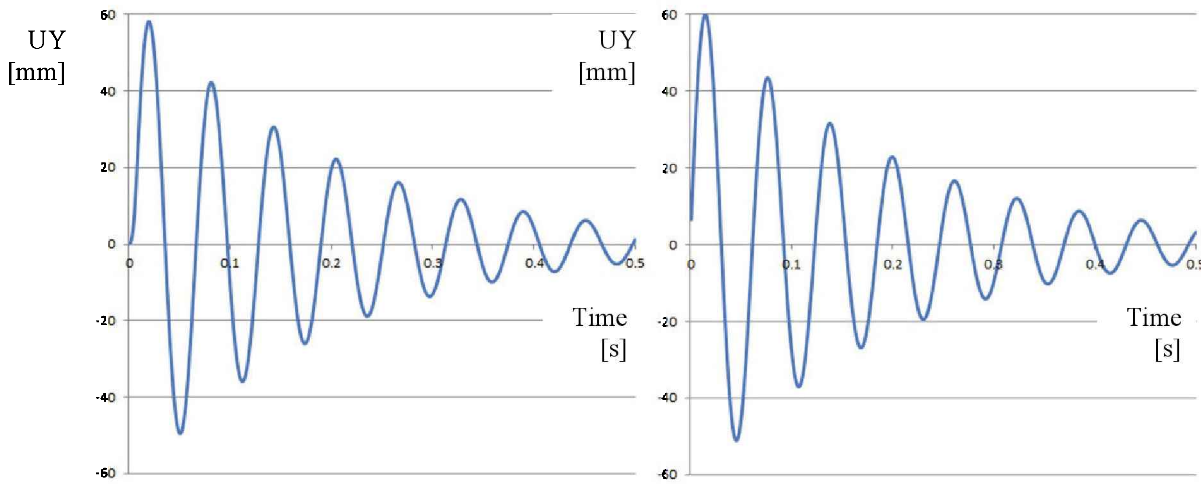


Figure 11-23: Definition of the initial condition.

The initial velocity definition applies to the entire 1kg cube.

Obtain the solution and review the UY displacement response graph and compare it with the same UY displacement response graph from the previous study. Both graphs are shown side by side in Figure 11-24.



Shock load

Initial velocity

Figure 11-24: UY displacement response to the shock load and to the initial velocity.

Both graphs are identical except for the first 0.01s when the shock load is active.

Results shown in Figure 11-24 show that in a single degree of freedom system the effect of a shock load (load of a very short duration) may be modeled by an equivalent initial velocity.

You may want to extend this exercise even further and investigate the effect of different damping on the vibration response. This may be easily done using, for example, study *06 1DOF acc fr.* This study was run with a modal damping 5%. Re-run the same study with damping 2% and 10% and summarize the UY displacement responses in one graph as shown in Figure 11-25.

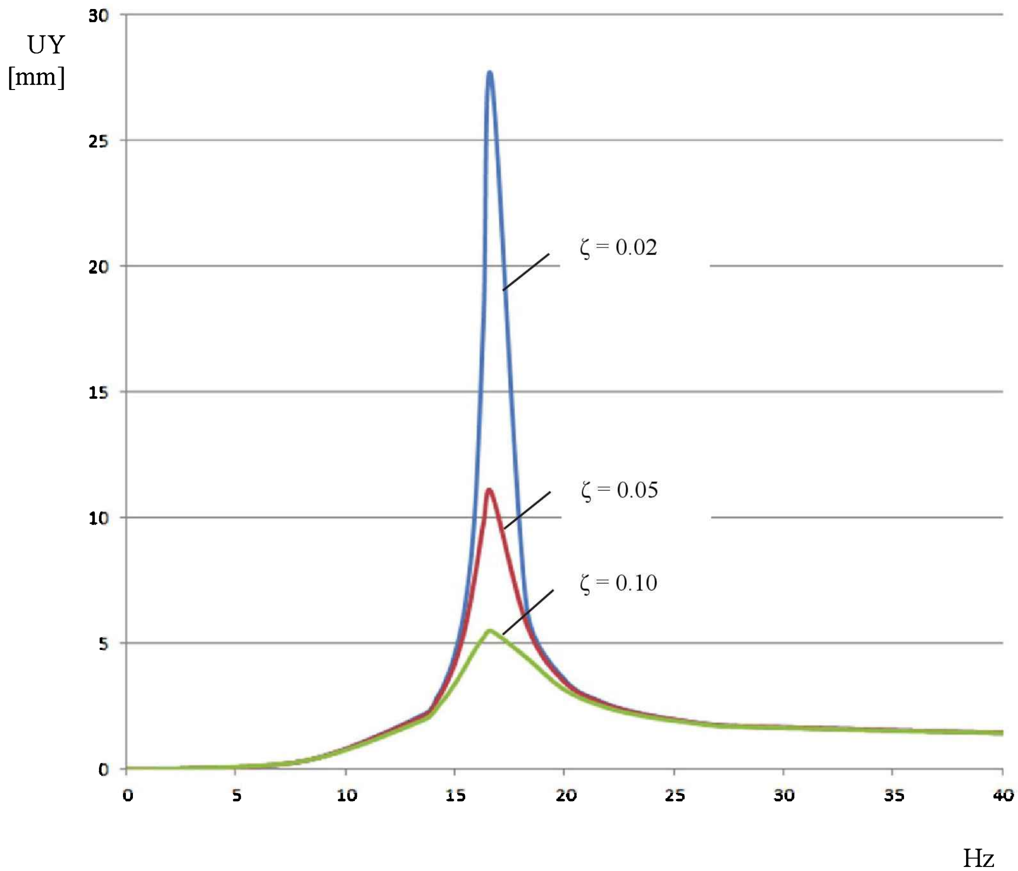


Figure 11-25: The amplitude of vibration as a function of excitation frequency for different modal damping values.

Damping strongly affects the amplitude for excitation frequencies close to the resonant frequency. It has no effect for excitation frequencies much lower or much higher than the resonant frequency.

Notice that the amplitude of vibration is always measured from the neutral position, not between negative and positive peaks.

As a side topic, not directly related to any of the analyses performed in this chapter, we'll point out the following. Since **Modal Time History** requires results of a **Frequency** analysis, a **Frequency** analysis is always run prior to a **Modal Time History**. Within **Modal Time History**, you may select to run only the **Frequency** analysis (Figure 11-26).

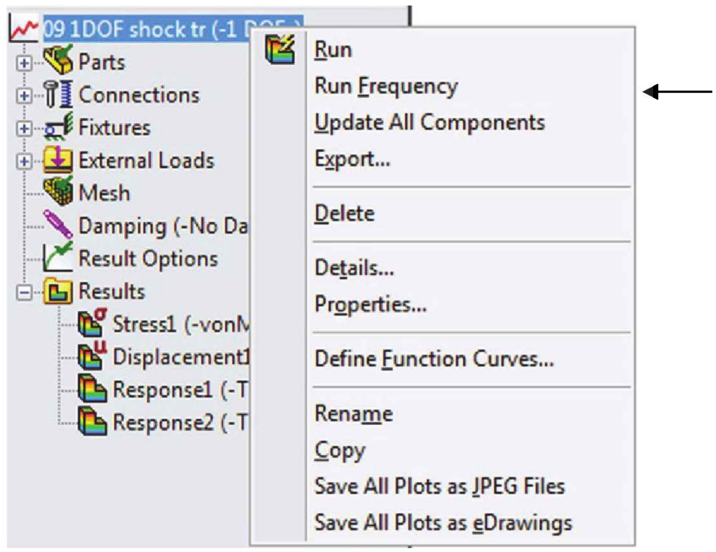


Figure 11-26: The pop-up menu invoked by right-clicking the Time Response study folder.

The Modal Time History study gives an option to run just the Frequency study without subsequent dynamic analyses.

Since a **Frequency** analysis is a part of **Modal Time History** and **Harmonic** studies, results of both these studies include the same results that are available in a **Frequency** study. To verify this, go back to the results of any **Modal Time History** or **Harmonic** studies and define a mode shape plot. Notice that you will see only as many frequencies as specified in the properties of the corresponding studies.

12: Harmonic base excitation of distributed systems

Topics covered

- Steady state harmonic excitation
- Frequency sweep
- Displacement base excitation
- Velocity base excitation
- Acceleration base excitation
- Resonance
- Modal damping

This chapter partially repeats the topic of harmonic base excitation introduced in Chapter 11 for a discrete system and expands it to include vibration induced stress analysis of a distributed system.

Steady state harmonic excitation

The harmonic excitation may take the form of harmonic force excitation or a base excitation where harmonic displacement, velocity, or acceleration is applied to the structure's restraints. Harmonic excitation is very common in engineering design problems. Rotating equipment may be subjected to harmonic excitation due to unbalanced loads, and many other types of excitations can be approximated successfully by harmonic excitation. In this chapter we'll model the vibration response of a structure subjected to harmonic base excitation where displacement of restraints is a sine function of time:

$$x = Asin(\omega t)$$

where:

A - displacement amplitude

ω - angular frequency.

Harmonic base excitation may be realized experimentally by placing the analyzed structure on a shaker table which oscillates with displacement amplitude A and frequency ω (Figure 12-1).

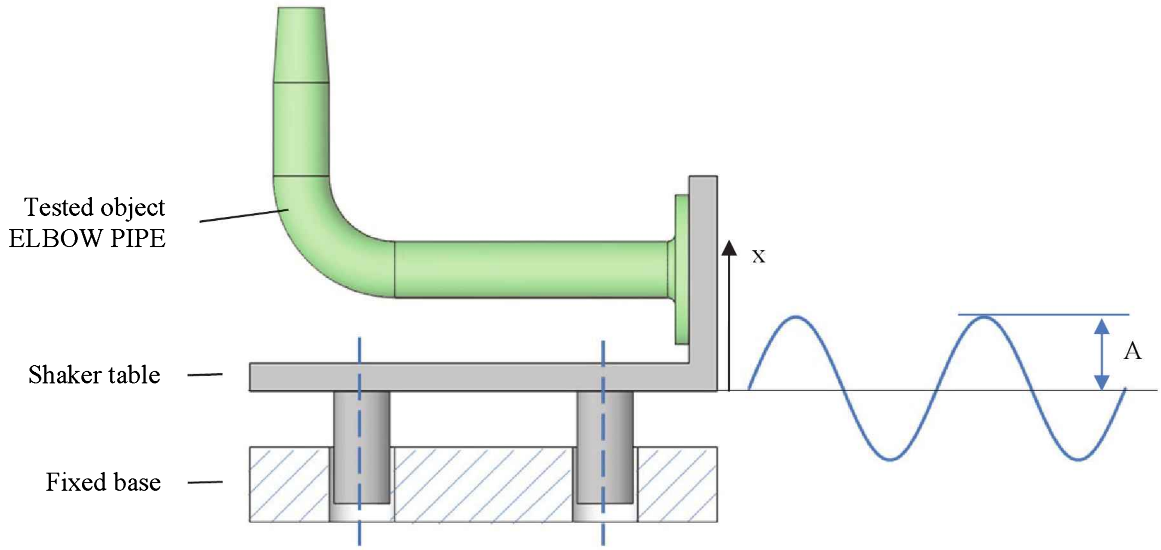


Figure 12-1: Setup of a shaker table.

The shaker table oscillates up and down with displacement amplitude A and frequency ω. The frequency changes slowly from 0 to 400Hz.

The model shown in Figure 12-1 is the SHAKER assembly. It is for show only and won't be analyzed. A component of this assembly, ELBOW PIPE, will be analyzed.

A shaker test is often conducted in such a way that the displacement amplitude remains constant while the frequency of excitation changes within a certain range. This is done to investigate the response of a tested object to excitation with different frequencies. The test is called a **Frequency Sweep** and will be simulated in this chapter by subjecting part model ELBOW PIPE to base excitation in the frequency range of 0 - 400Hz. This is the range of frequencies to which the part is subjected to in service. A very important assumption has to be made: the rate of change of the excitation frequency is assumed to be very low. Therefore, the vibration response is a steady state response; transient responses are not modeled. As we already know from Chapter 11, in **SolidWorks Simulation** this analysis is called a **Linear Dynamic** study with a **Harmonic** option. Vibration analysis textbooks call this a **Steady State Harmonic Response**.

In preparation of the **Steady State Harmonic Response** analysis, we have to run two preparatory analyses.

First we'll run a static study to identify the location of stress concentration and to design a mesh that will correctly model it. Using information from this static study we'll define the locations of stress sensors.

Next, we'll run a modal analysis to see how many modes of vibration are in the 0 - 400Hz frequency range. This way we'll know how many modes have to be considered in the vibration response analysis based on the **Modal Superposition Method**.

Open part model ELBOW PIPE and create a **Static** study *Gravity*. Apply a gravity load of 9.81m/s^2 as shown in Figure 12-2.

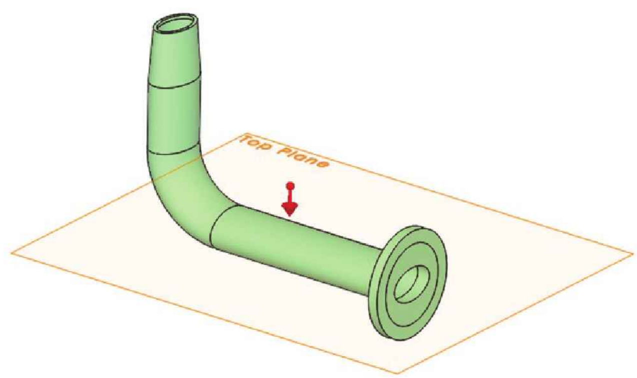


Figure 12-2: Load and restraints in the static study.

Gravity load 9.81m/s^2 *is normal to the Top reference plane.*

This load is not supposed to model real life loading conditions during the frequency sweep, but being a volume load is qualitatively close to the loading that the model will experience during the **Frequency Sweep**. We need it only to locate stress concentrations and to design the mesh. Absolute magnitudes of stress do not matter.

Apply restraints as shown in Figure 12-3; the same restraints will be used in all studies in this chapter.

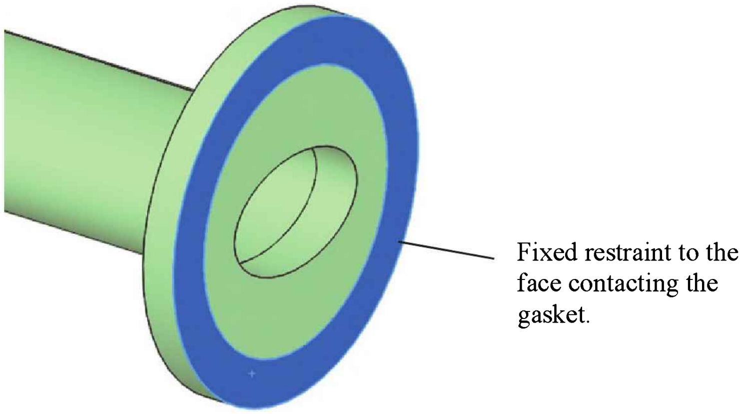


Figure 12-3: Restraints to the ELBOW PIPE model.

Apply a fixed restraint to the face created by the split line. This restraint approximates the attachment to another pipe with the same sized flange.

Define a mesh control on the fillet near the flange as shown in Figure 12-4.

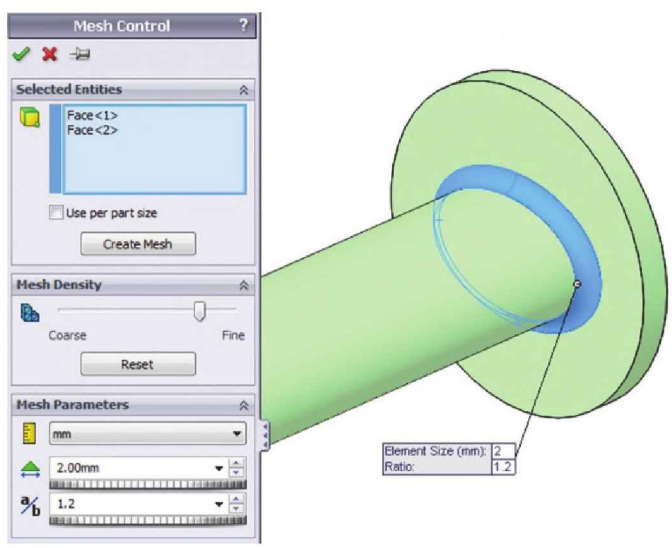


Figure 12-4: Mesh control to the fillet; notice that the fillet face is divided into two faces by a split line.

Use a 2mm element size on the controlled entity and the ratio of transition, a/b, equal to 1.2.

Having applied the mesh control, mesh the model with a global element size of 8mm and obtain the static solution. Review **P1 stress** results and confirm the location of the stress concentrations as shown in Figure 12-5. Remember that von Mises stress is not applicable to brittle material such as **Gray Cast Iron**, which has been assigned to the model.

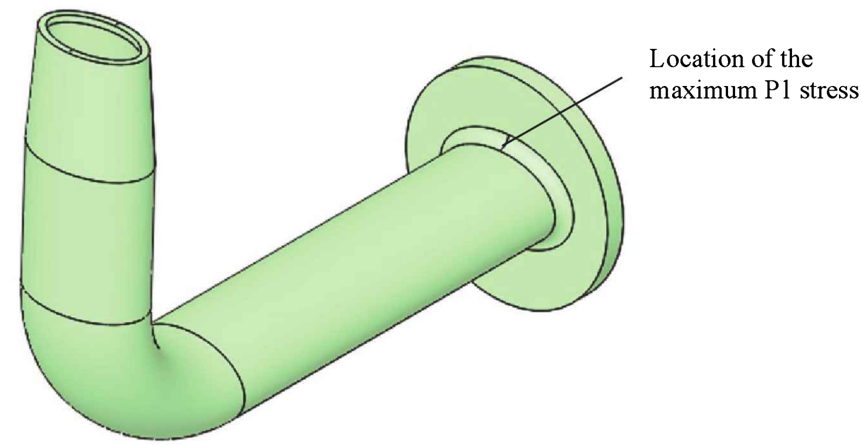


Figure 12-5: Location of the highest P1 stress.

This location coincides with the split lines placed there to mark the location of a stress sensor.

Based on the results of a qualitative stress analysis, we define the first sensor in the location shown in Figure 12-5; it will be used to create **Stress Response Graphs**. Define the second sensor as shown in Figure 12-6; this one will be used to create **Displacement Response Graphs**.

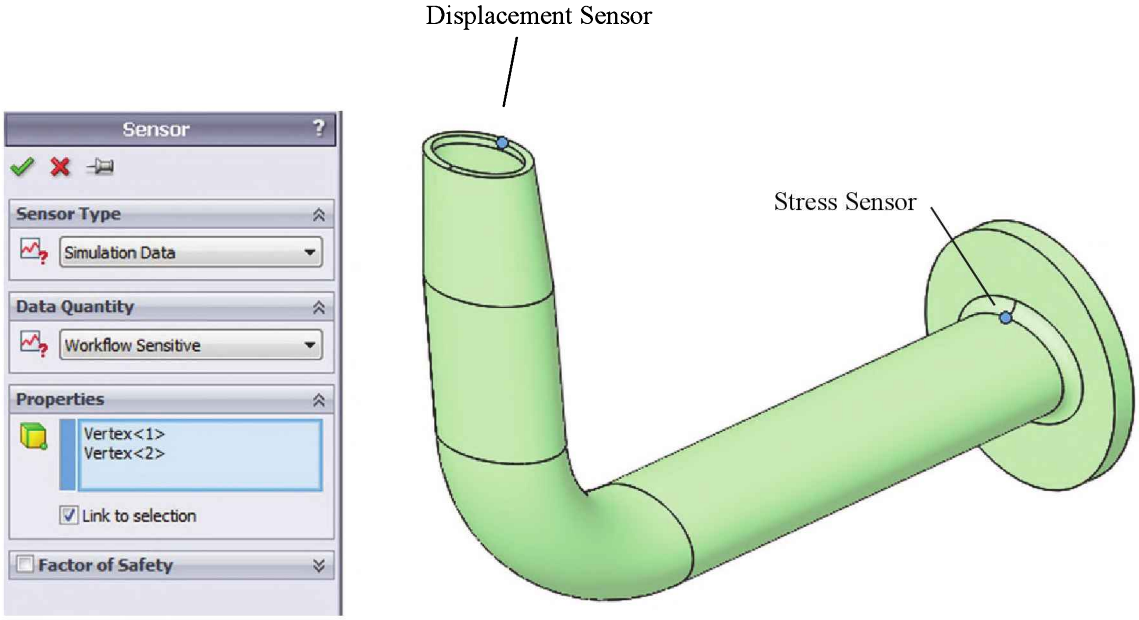


Figure 12-6: Sensor locations; two locations are defined in one Sensor window.

Both sensors use split lines in the model for defining their locations. These sensors are already defined in the SolidWorks model.

Review once more the results of the *Gravity* study and notice that the highest P1 stress is not exactly in the **Stress Sensor** location and that a somewhat irregular shape of stress fringes (use Discrete Fringes) indicates that a more refined mesh is required. We'll use this sensor location for simplicity of its definition. We'll use a relatively coarse mesh to speed up the otherwise long solution time of the **Harmonic** studies. Once done, you are encouraged to repeat the analysis with a **Stress Sensor** located on the curved face using a more refined mesh.

Create a **Frequency** study titled *modal* with properties shown in Figure 12-7.

Upper bound frequency 400Hz

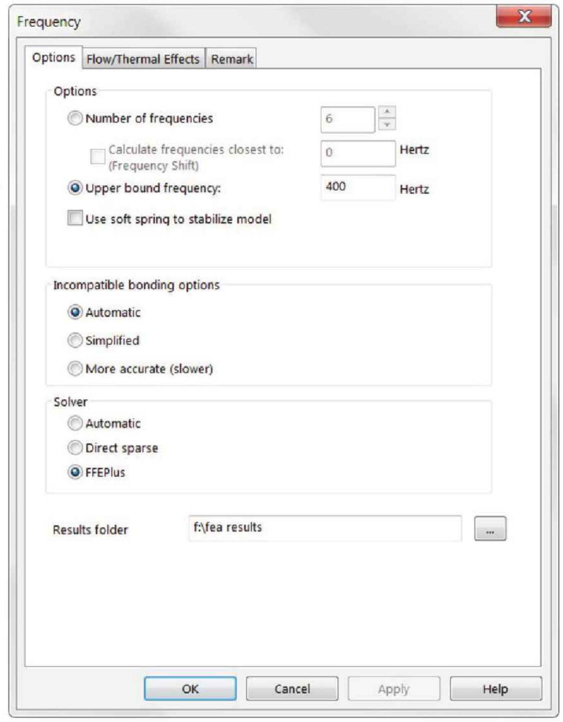
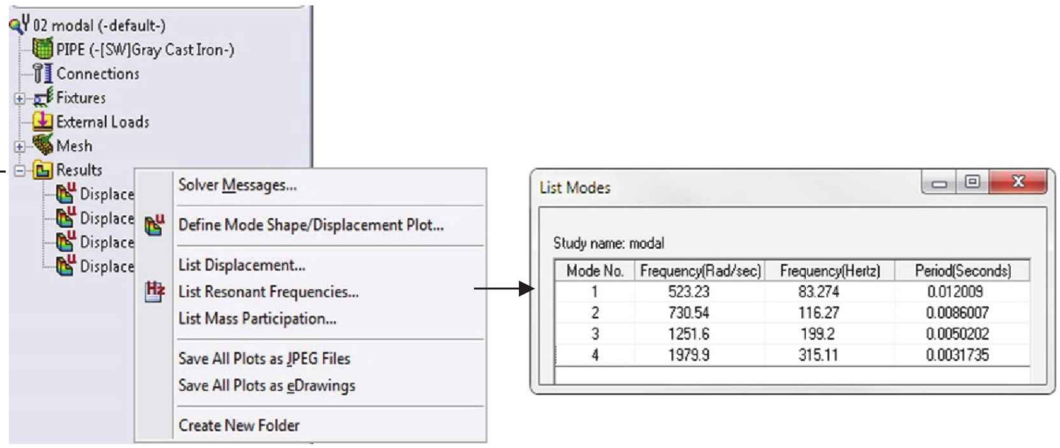


Figure 12-7: Properties of the frequency study; upper bound frequency of 400Hz selected.

Specifying an Upper bound frequency of 400Hz means that all modes in the frequency range of 0-400Hz will be found.

Obtain the modal solution and list the modes in the 0-400Hz range as shown in Figure 12-8. Remember that you may also run a modal analysis within the **Harmonic** study as explained in Chapter 11.

Right-click Results folder. Select List Resonant Frequencies.



<u>Figure 12-8: List Modes window shows four modes within the range of 0-400Hz.</u>

Modal frequencies are shown using three units: rad/s, Hz, and s. The relation between frequency ω (rad/s) and f (Hz) is: ω =2Πf.

Review modal shapes shown in Figure 12-9:

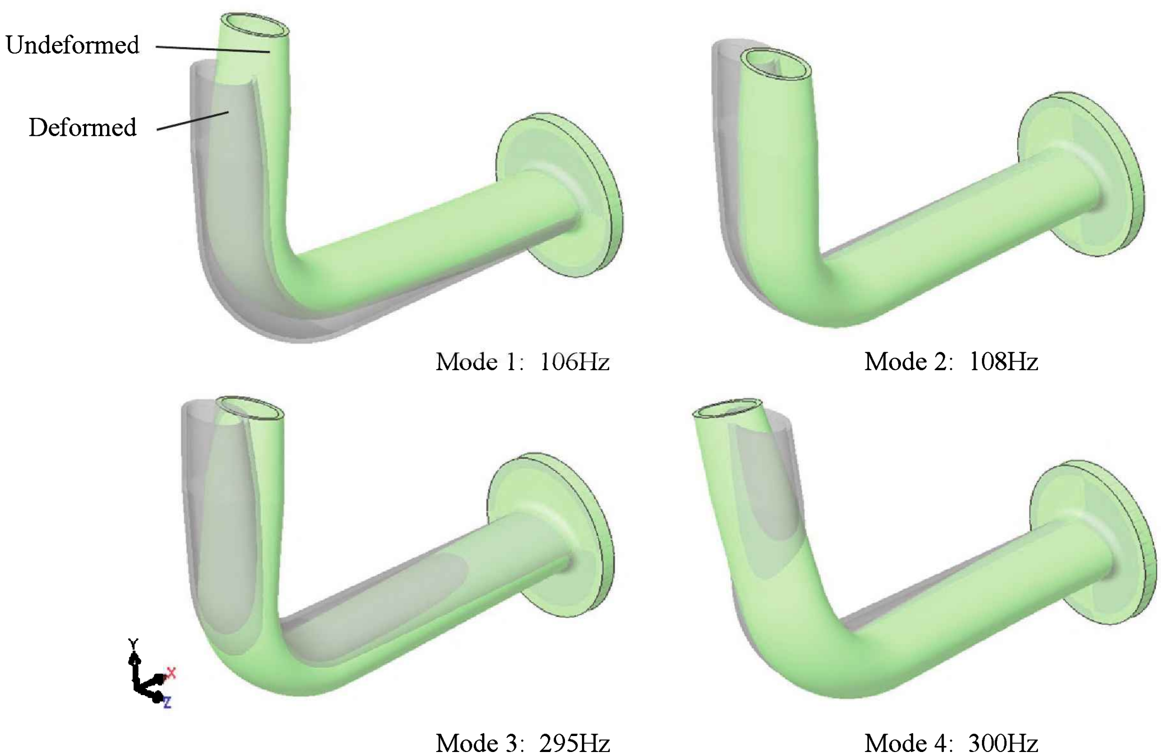


Figure 12-9: Modal shapes shown with the undeformed model superimposed on the deformed model.

The undeformed model is shown in darker color. All modes are shown in the same view.

As shown in Figure 12-9, vibration in modes 1 and 3 are in the XY plane; vibration in modes 2 and 4 are in YZ plane. The base excitation is applied in the Y direction; therefore only mode 1 and mode 3 will be excited in the **Frequency Sweep** simulation. In real life the direction of excitation would never be perfectly aligned with one plane and all modes would be excited to some extent.

In preparation for the **Frequency Sweep** analysis, recall the relation between displacement, velocity, and acceleration base excitation (discussed in Chapter 11). As shown in Figure 12-1, the shaker table is excited with a harmonic displacement of amplitude A and frequency ω. Assuming that displacement at $t = 0$ is $x = 0$, displacement x at any given point of time t is:

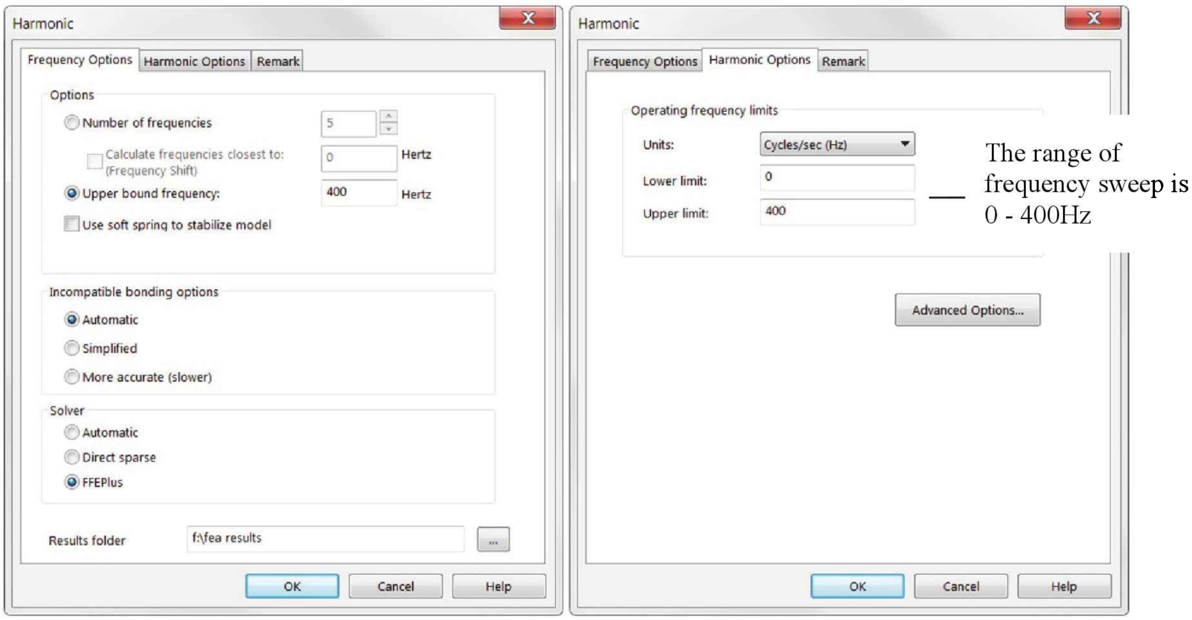
$$x = Asin(\omega t)$$

Velocity is the first derivative of displacement with respect to time and acceleration is the second derivative of displacement with respect to time. Therefore, at any given point of time t velocity v and acceleration a are:

$$v = \mathrm{A}\omega \cos(\omega \mathrm{t})$$

$$a = -A\omega^2 \, sin(\omega t)$$

To demonstrate the equivalence of displacement, velocity, and acceleration excitations, we'll conduct three **Harmonic** studies: *01 harmonic disp*, *02 harmonic vel*, *03 harmonic accel*. These are all **Linear Dynamic** studies with the **Harmonic** option; properties of these studies are all the same as shown in Figure 12-10.



Frequency options Harmonic Options

<u>Figure 12-10: Frequency Options and Harmonic Options of the three studies simulating a Frequency Sweep.</u>

Frequency Options define how many modes will be used in the Modal Superposition method. Harmonic Options define the range of frequencies in the simulated Frequency Sweep.

Frequency Options are identical to the settings of the already completed modal analysis.

Damping in the **Harmonic** study must be defined as a modal damping. See Figure 12-11 for the damping definition.

Figure 12-11: Modal damping definition for the three studies simulating a Frequency Sweep.

Modal damping is 5% for all studies. The Last Mode is listed here as 15 but only four modes are present in the analysis.

Defining the modal damping the same for all modes is conservative, meaning that displacement and stress results for higher modes may be overestimated. In reality damping in higher modes is usually higher because of more deformation present in those modes.

In all three studies define restraints the same as in the previously completed static and modal analyses; the mesh is also the same. What differentiates the studies is the type of Base Excitation.

Figure 12-12 shows the **Base Excitation** definition in study *01 harmonic displ*. **Uniform Base Excitation** means that the same excitation is applied to all restraints which are active in the direction of excitation.

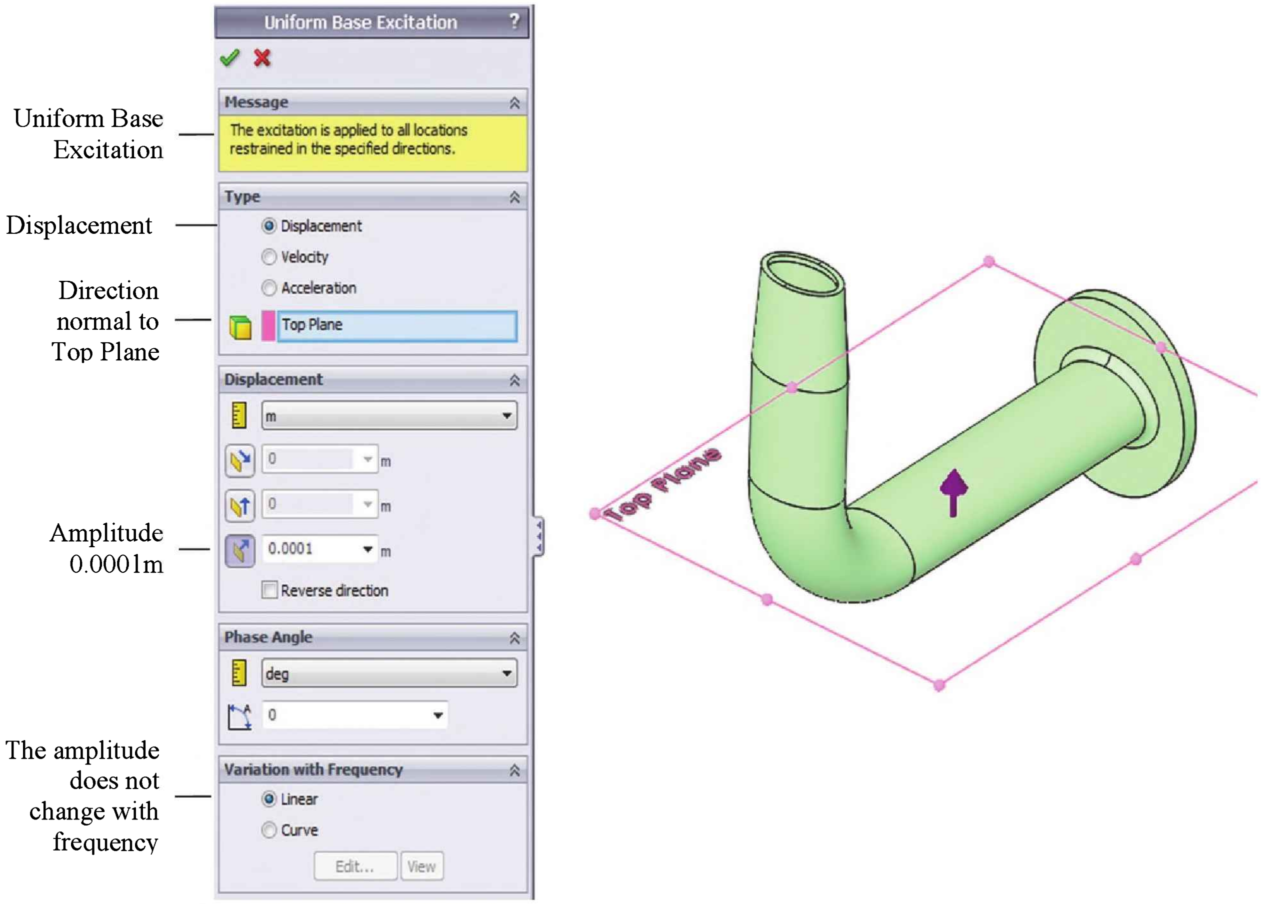
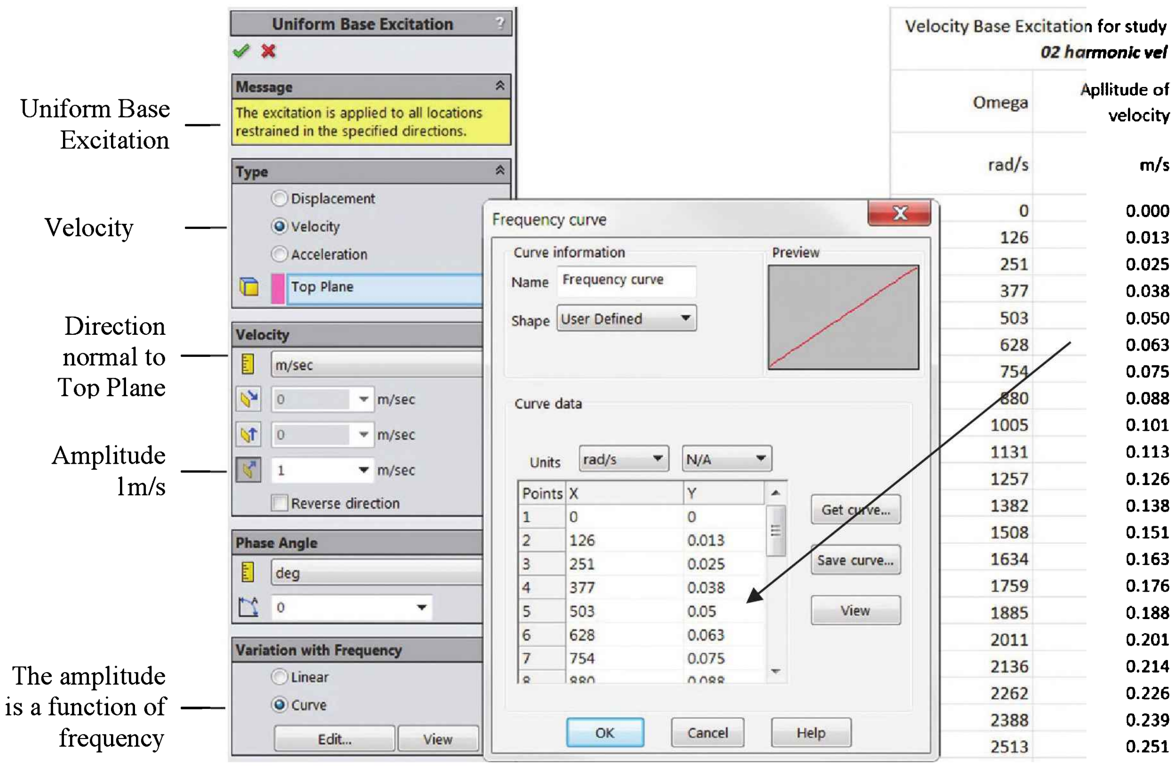


Figure 12-12: Base excitation in study *01 harmonic displ.*

The amplitude of displacement remains constant throughout the entire range of frequencies. The amplitude of displacement is entered in meters. This will make it easier to compare displacement excitation with velocity excitation and acceleration excitation.

Notice an imprecise terminology of **Variation with Frequency**. The term **Linear** actually means constant amplitude throughout the range of frequencies.

Copy study *01 harmonic displ* into *02 harmonic vel* and change the **Base Excitation** definition from displacement excitation to velocity excitation as shown in Figure 12-13.



Velocity Base Excitation for study *02 harmonic vel*	
Omega	Apllitude of velocity
rad/s	m/s
0	0.000
126	0.013
251	0.025
377	0.038
503	0.050
628	0.063
754	0.075
880	0.088
1005	0.101
1131	0.113
1257	0.126
1382	0.138
1508	0.151
1634	0.163
1759	0.176
1885	0.188
2011	0.201
2136	0.214
2262	0.226
2388	0.239
2513	0.251

Figure 12-13: Base excitation in study *02 harmonic vel.*

The amplitude of velocity is a linear function of frequency as the Preview window shows.

Click **Edit** in the **Uniform Base Excitation** window to open the **Frequency curve** window. Delete all default rows and paste the two columns from spreadsheet ELBOW PIPE.xlsx. The amplitude of velocity is 1m/s because the multiplier $A\omega = 0.0001\omega$ is already included in the amplitude of velocity column; review the ELBOW PIPE.xlsx to confirm this.

Notice that the **Frequency curve** is defined up to 2639 rad/s (420Hz) to exceed the range of the **Frequency Sweep** requested in the study properties (Figure 12-10).

Figure 12-14 shows the **Uniform Base Excitation** definition in study *03 harmonic accel*.

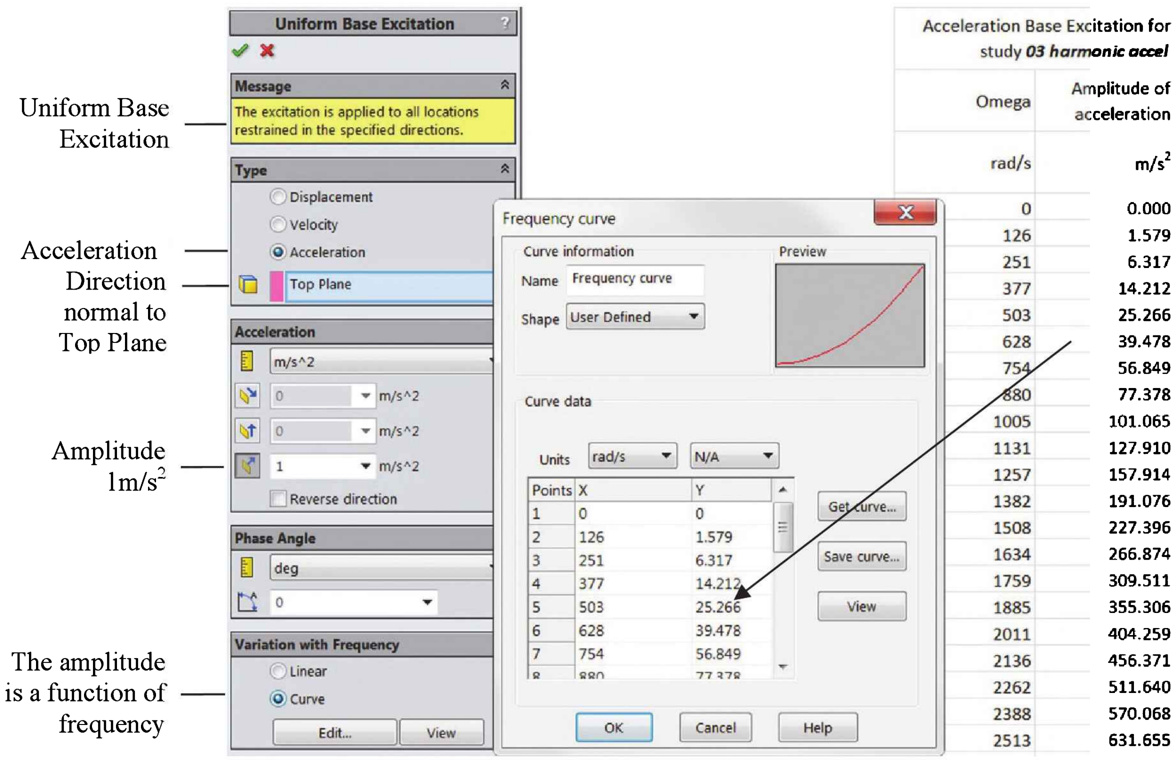


Figure 12-14: Base excitation in study *03 harmonic accel.*

The amplitude of velocity is a parabolic function of frequency as the Preview window shows.

Solve all studies and create a **Response Graphs** of UY displacement amplitude recorded by the **Displacement Sensor** and P1 stress amplitude recorded by the **Stress Sensor**; both sensors are shown in Figure 12-6. An example of the definition of the **Response Graph** for P1 stress amplitude is shown in Figure 12-15.

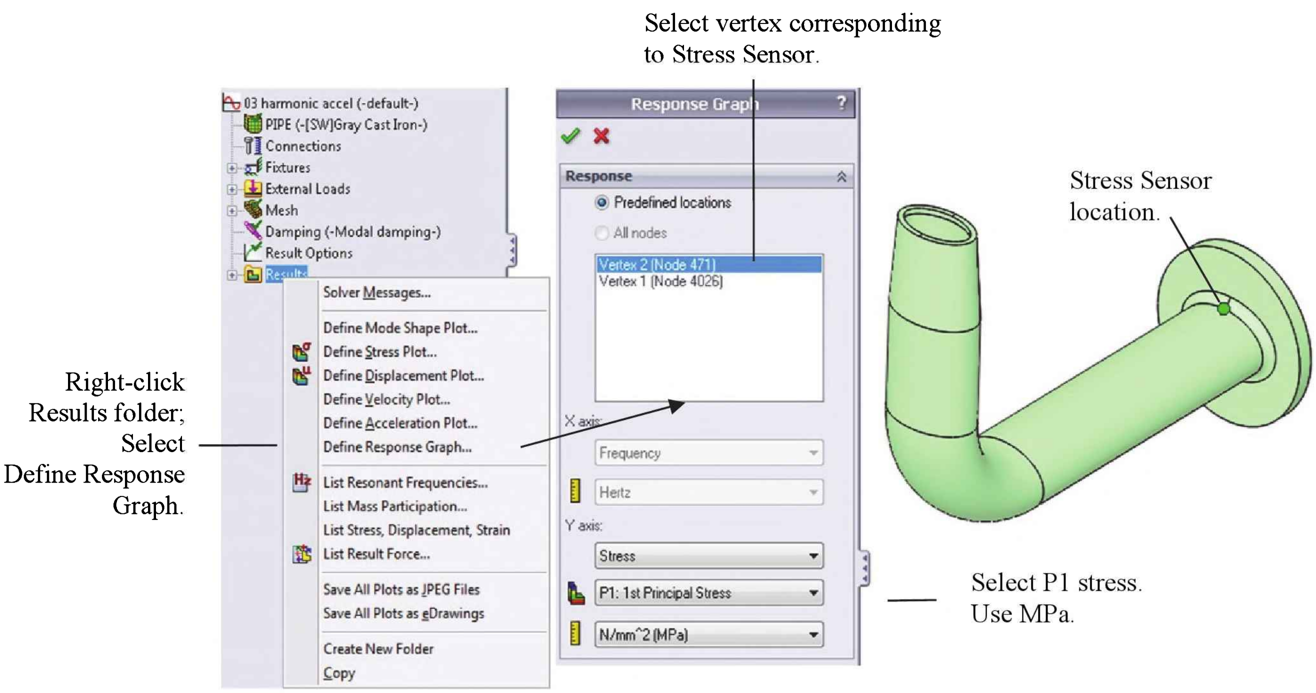


Figure 12-15: Definition of the Response Graph shown here for P1 stress.

Select the vertex where the required sensor is located.

All three UY displacement **Response Graphs** and all three P1 stress **Response Graphs** from the three studies are identical because the excitation with constant displacement amplitude, linear velocity amplitude, and parabolic acceleration amplitude all produce the same effect.

Automatically created graphs may be saved as a csv file for processing in a spreadsheet or in a graphics program (Figure 12-16).

Click File
Select Save As
csv

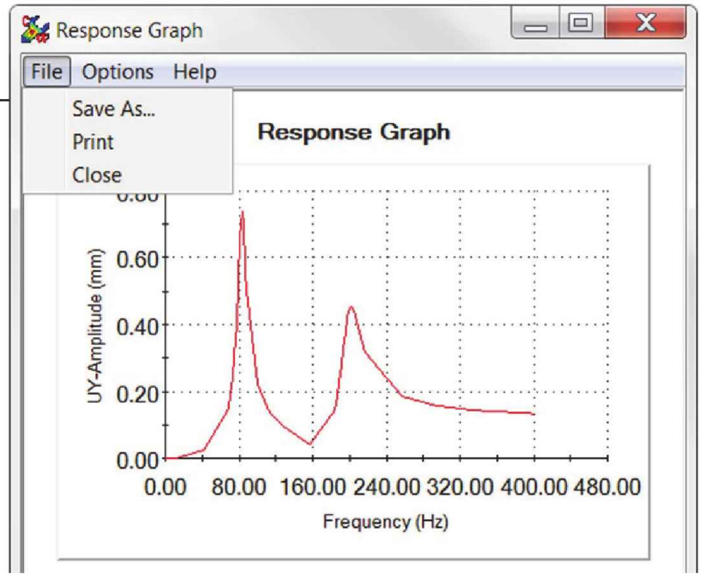


Figure 12-16: Saving the Response Graph for processing in another program.

Select csv format to export data to Excel. Selection of the data format is not shown in this illustration.

The graph in Figure 12-17 shows UY displacement magnitude as a function of frequency; it has been formatted in Excel.

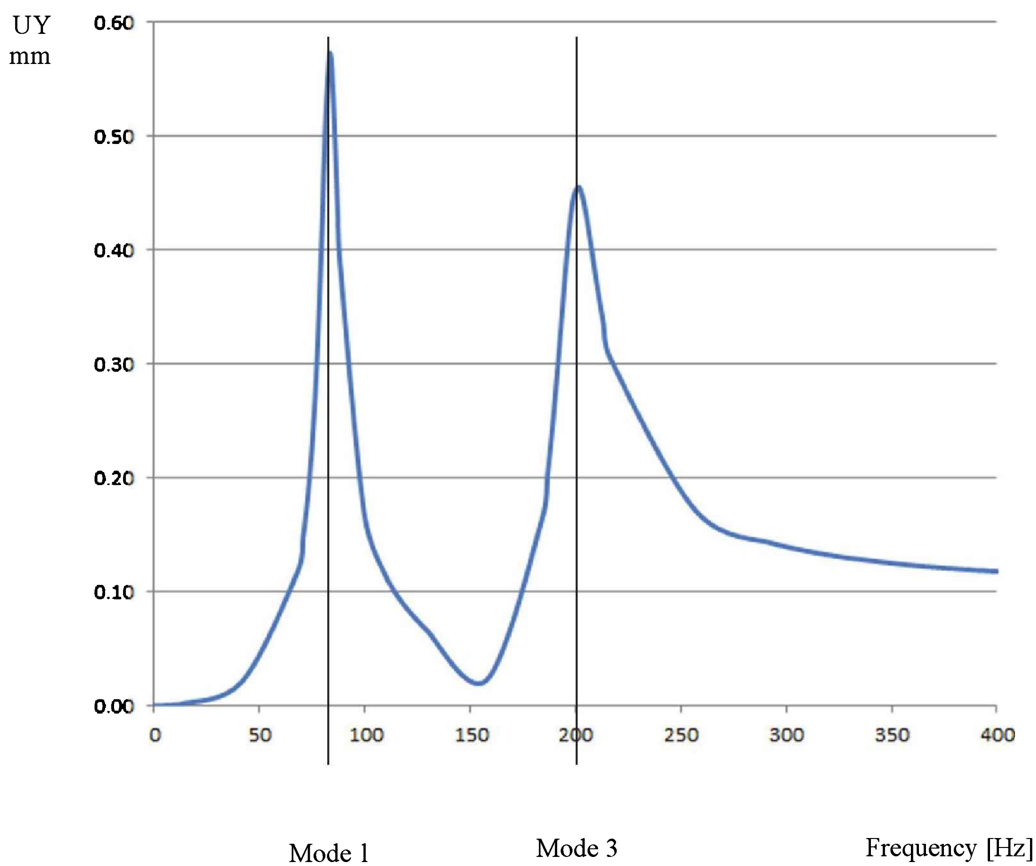


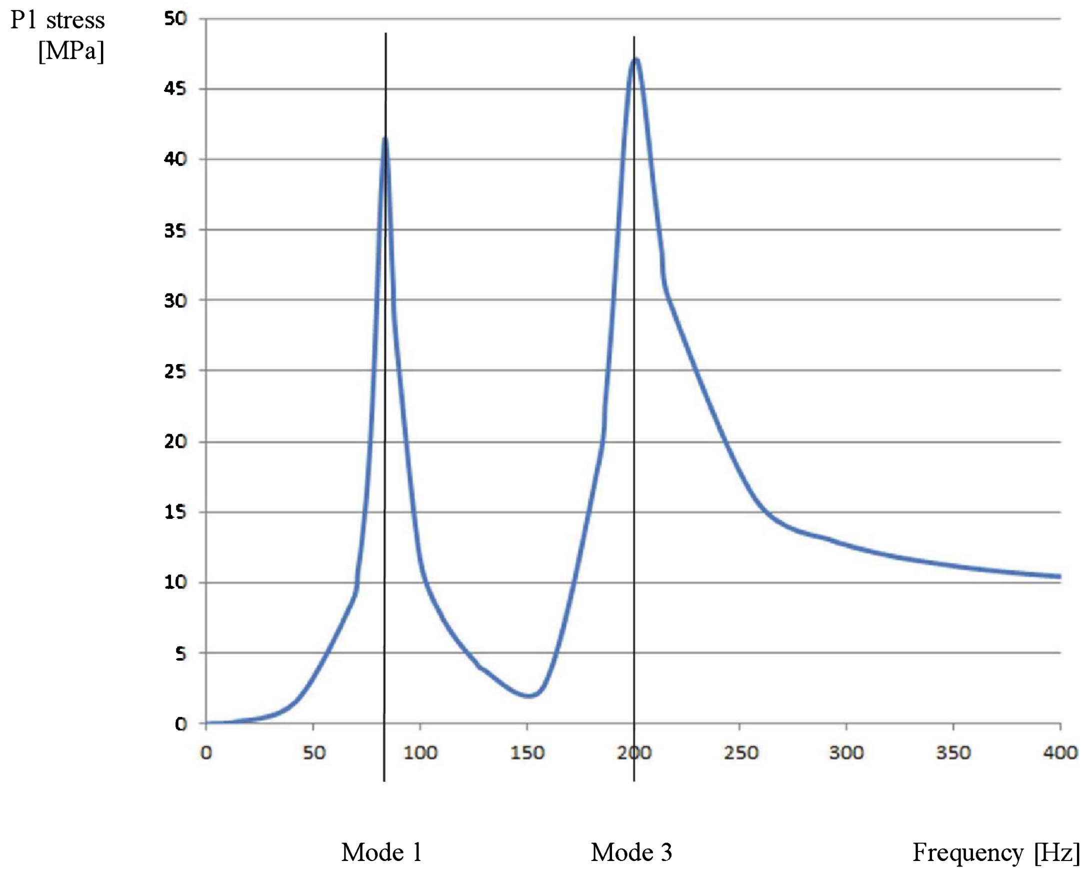
Figure 12-17: Displacement amplitude as a function of excitation frequency. The excitation frequency changes within the range of the frequency sweep (0-400Hz).

The excitation frequency equal to the first modal frequency, 84Hz, produces an oscillation within the range of -0.58mm to +0.58mm.

The excitation is in the XY plane. Therefore, modes 2 and 4, which have shapes moving out of the XY plane are not excited even though their frequencies are within the range of the **Frequency Sweep**.

Review the Response Graph of the UZ displacement component and notice that UZ displacement is practically zero. The only reason some displacement is shown is due to discretization error of the excitation. Discretization error makes stiffness not exactly symmetric about the XY plane.

The graph in Figure 12-18 shows P1 stress as a function of frequency. It has also been formatted in Excel.



<u>Figure 12-18: P1 stress amplitude in the indicated location as a function of excitation frequency. The excitation frequency changes within the range of the frequency sweep (0-400Hz).</u>

The excitation frequency equal to the third modal frequency, 199Hz, produces a P1 stress of 47MPa.

Excitation with frequencies equal to or very close to the modal frequency and along the corresponding direction of the modal shape is called **Resonance**. Inertial forces cancel with stiffness forces and the vibration response is controlled purely by damping. Repeat the analysis using any of the threes studies with lower modal damping to see a very significant increase of displacement and stress amplitudes.

Compare Figures 12-17 and 12-18 and notice that while the displacement response is higher for the first resonant frequency, the stress response is higher for the second resonant frequency.

13: Omega square harmonic force excitation

Topics covered

- Unbalanced rotating machinery
- Resonance
- Modal damping
- Omega square excitation
- Steady state response

Procedure

A centrifuge is shown schematically in Figure 13-1.

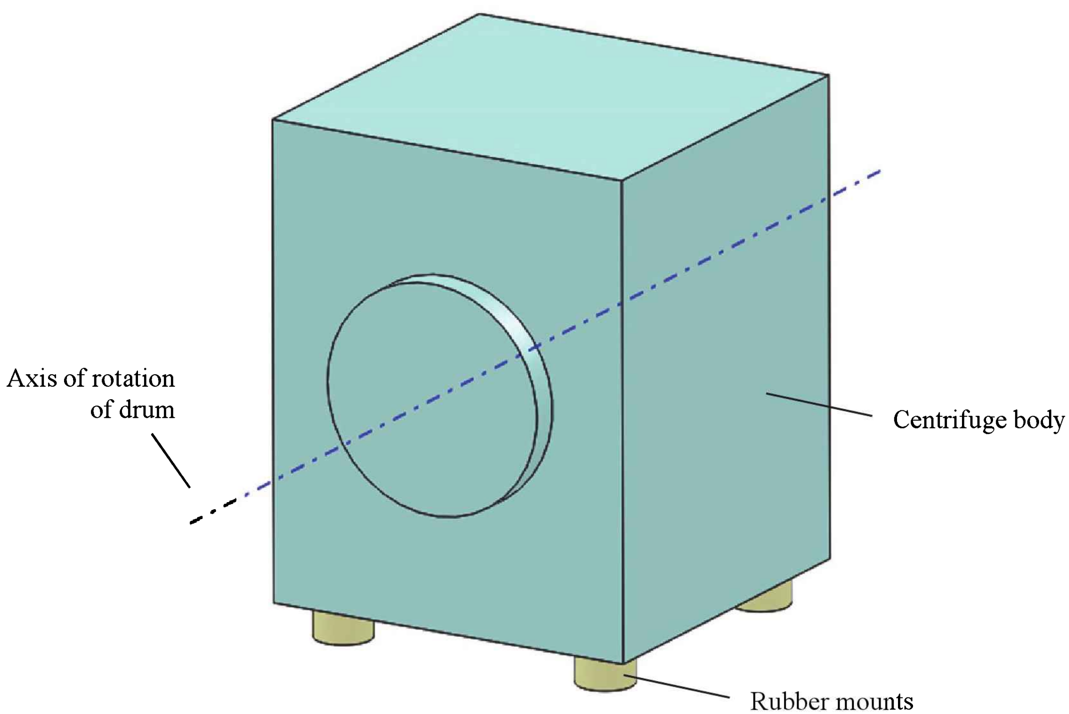


Figure 13-1: The centrifuge sits on four rubber mounts. The drum rotates about the horizontal axis. The motion of the centrifuge is restricted to the vertical direction.

The drum is merged with the body in this schematic representation.

The drum has a dynamic imbalance of 0.5kg at a radius of 100mm. Therefore, the centrifugal imbalance force F is a function of angular velocity, ω:

$$F = me\omega^2$$

$$m = 5kg \; - \; imbalanced\ mass$$

$$e = 0.1m \; - \; eccentricity$$

$$\omega \; - \; angular\ velocity\ rad/s$$

The operating speed of the centrifuge is 2000RPM which corresponds to a frequency of 33.3Hz. Our objective is to find the amplitude of vibration of the centrifuge body as a function of the angular velocity when the machine slowly reaches its operating speed. This is a Steady State Harmonic Response problem requiring a **Harmonic** study.

Create a **Harmonic** study titled *centrifuge* and define restraints as shown in Figure 13-2.

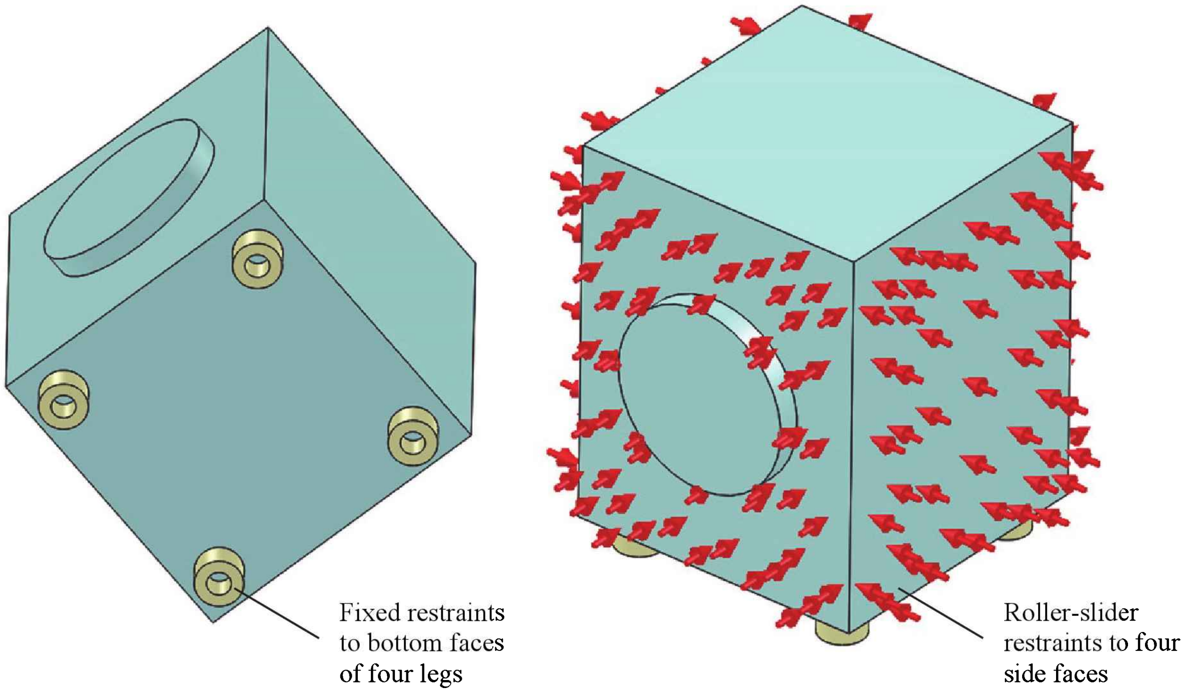
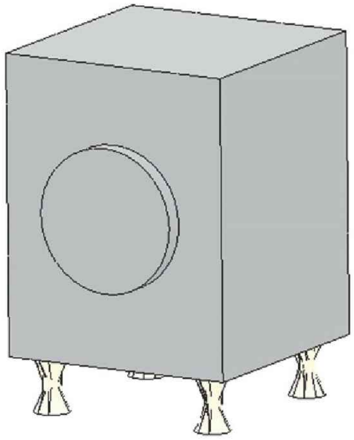


Figure 13-2: Restraints applied to the assembly model allow for vertical movement of the body accompanied by deformation of the rubber mounts.

Symbols of Fixed restraints defined on the bottom faces of the four rubber mounts are not shown.

Mesh the model using a Standard Mesh with a default element size. Before running the study, we'll run a modal analysis to investigate frequencies in the range of 0-2000RPM. Run a modal analysis from inside the **Harmonic** study and review the frequencies of the first five modes. To speed up the solution time you may define 5 modes instead of the default 15 modes in the **Frequency Options** of the **Harmonic** study.

Results of the modal analysis are shown in Figure 13-3.



List Modes

Study name: centrifuge

Mode No.	Frequency(Rad/sec)	Frequency(Hertz)	Period(Seconds)
1	58.007	9.2321	0.10832
2	3708	590.15	0.0016945
3	3709.6	590.4	0.0016938
4	3714.1	591.12	0.0016917
5	3717.6	591.68	0.0016901

Modal shape 1 at a frequency of 9.2Hz. Deformation is limited to the rubber legs. The centrifuge performs oscillations in vertical direction.

Only mode 1 falls in the range of the operating frequencies: 0 – 33Hz.

Figure 13-3: Results of the modal analysis.

Animate modes 2-5 to see that they correspond to local deformations of the rubber legs.

We may now proceed with a **Harmonic** study. Using the results of the modal analysis, we know to define only one mode in **Frequency Options**; in **Harmonic Options**, define the operating frequency limits of 0 -33Hz or 0 - 207rad/s (Figure 13-4).

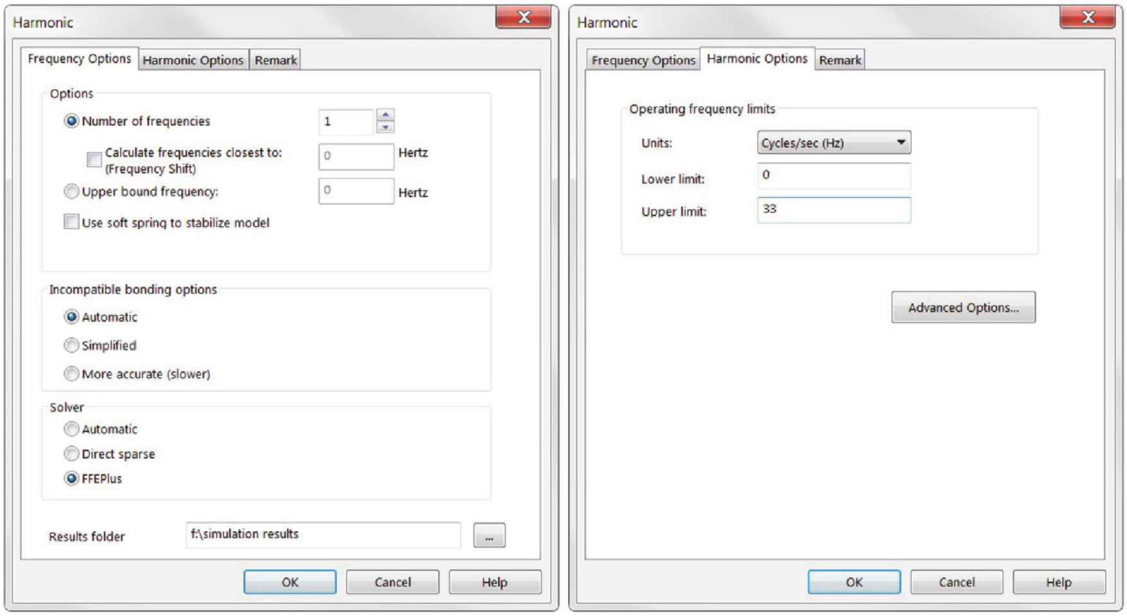


Figure 13-4: Options of the Harmonic study.

The operating frequency limi6ts correspond to a range of 0-2000RPM.

Define Modal Damping as 5%, which corresponds to the damping in the rubber material (Ref. 8, Chapter 21). The damping definition is shown in Figure 13-5.

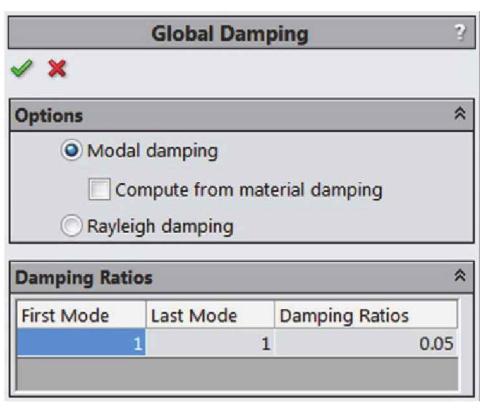
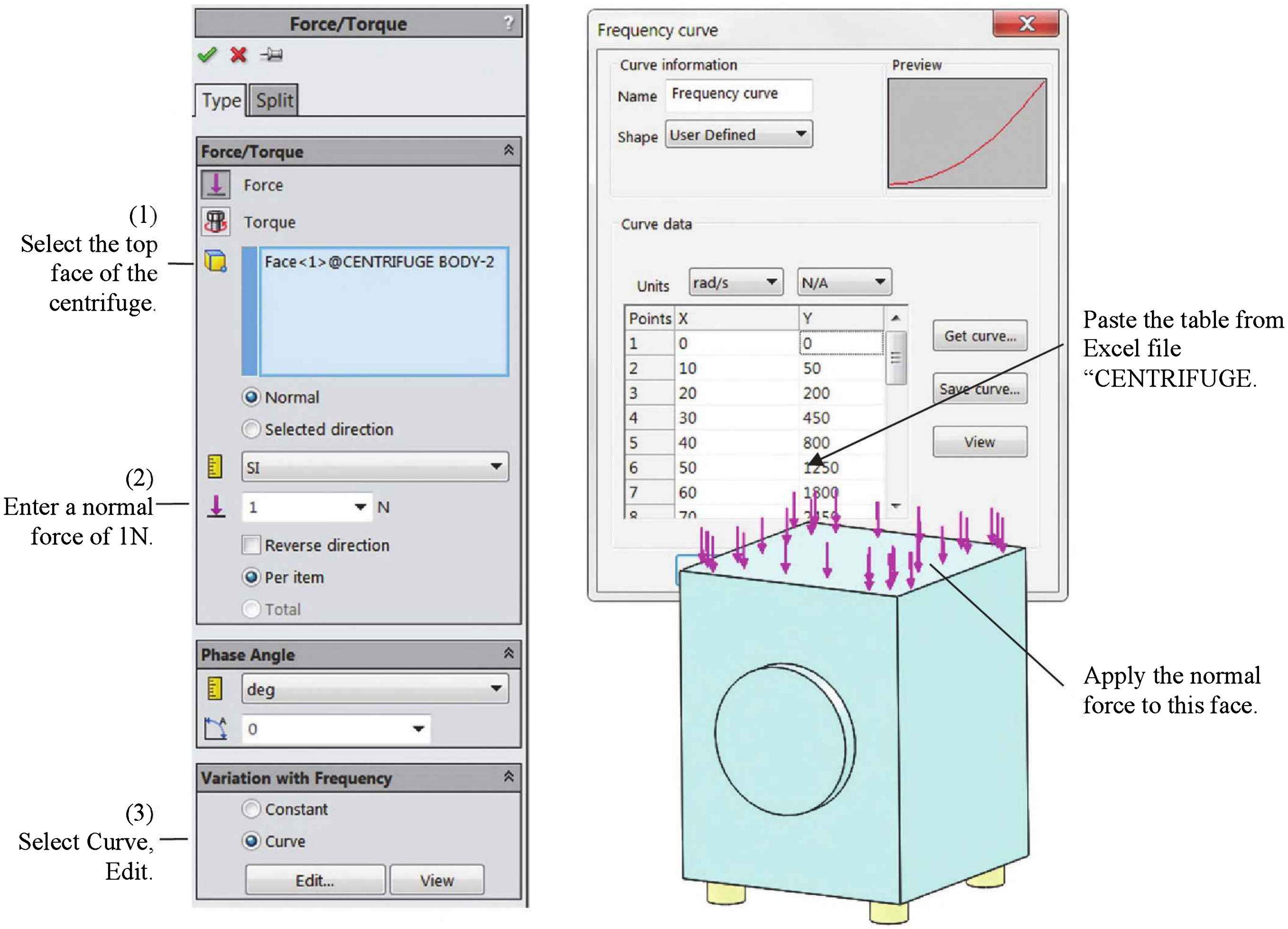


Figure 13-5: Definition of Modal Damping.

Damping only needs to be defined for one mode.

The load has to be defined as a square function of angular velocity; $= me\omega^2$. Using a schematic representation of the centrifuge as in the assembly model *CENTRIFUGE*, we may represent the imbalanced force in a simplified manner by defining a load to the top face. Open the spreadsheet CENTRIFUGE, copy the highlighted table and follow the steps shown in Figure 13-6 to define the load.



<u>Figure 13-6: Definition of the imbalanced load.</u>

1N is a multiplier to the Y column of the table copied from the spreadsheet.

Go to the CAD model to review the **Sensor** defined in one of the top corners of the centrifuge body. Define the **Results Options** shown in Figure 13-7.

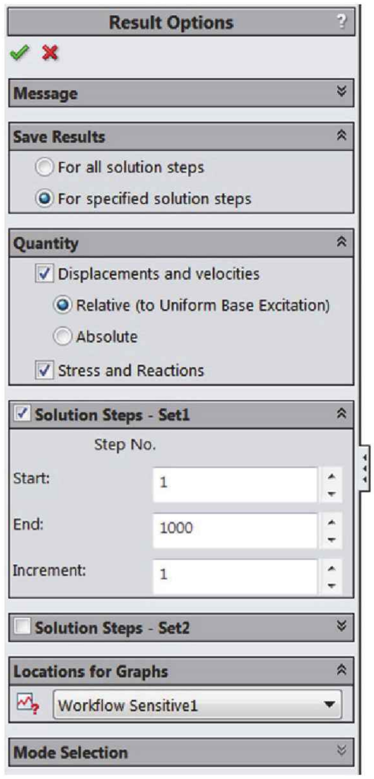


Figure 13-7: Definition of Results Options.

This is done in preparation to graphing the magnitude of displacement as a function of the angular velocity. Even though the end step is defined here as 1000, the analysis will complete in only 37 steps.

Solve the **Harmonic** study and define a response graph showing the amplitude of displacements as a function of the angular velocity (Figure 13-8).

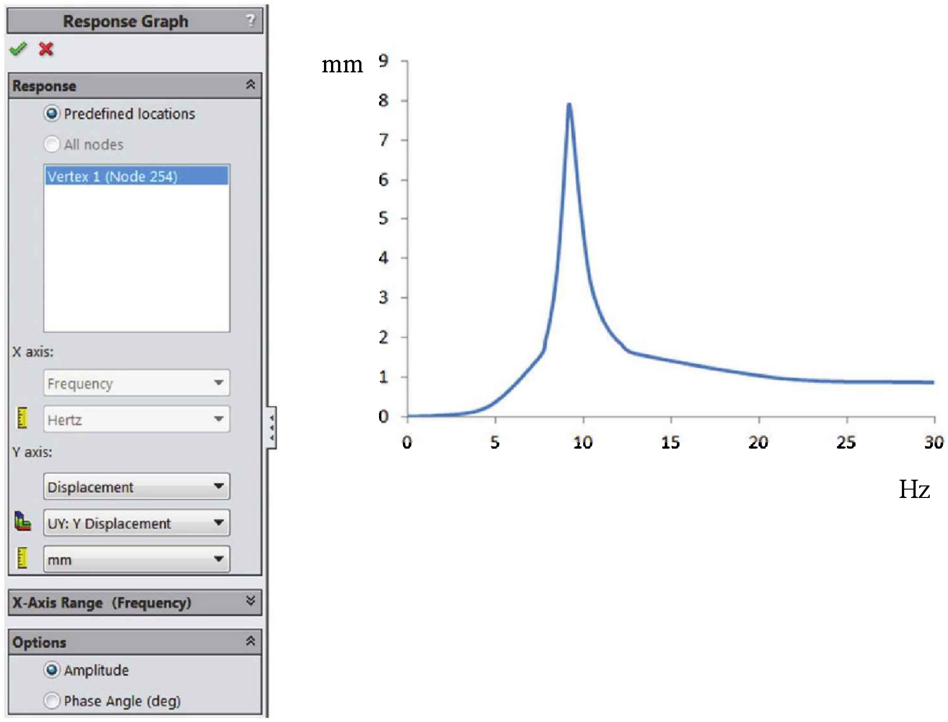


Figure 13-8: Amplitude to vertical displacement as a function of the excitation frequency.

The graph has been formatted in Excel.

The one peak in Figure 13-8 corresponds to the only natural frequency, 9.2Hz, present in the analyzed range of excitations. The maximum displacement amplitude is ~8mm and it happens when the excitation frequency equals the natural frequency 9.2Hz or 552RPM.

Notice that the amplitude reaches a constant value for excitation frequencies much higher than the resonant frequency; just like in the case of base excitation with constant displacement amplitude.

The displacement amplitude for omega square excitation for the excitation frequency much higher than the excitation frequency has analytical solution:

$$d = \frac{me}{M} = 0.77mm$$

$$m = 5kg \;-\; imbalanced\ mass$$

$$e = 0.1m \;-\; eccentricity$$

$$M = 645kg - mass\ of\ centrifuge\ body$$

You’ll have to extend the range of the excitation frequency to 4000RPM to see displacement amplitude converging to the above value.

14: Time response analysis, resonance, beating

Topics covered

- Time Response analysis
- Base excitation
- Resonance
- Modal damping
- Beating phenomenon
- Transient response
- Steady state response
- Mass participation

A support bracket is formed from a steel sheet in the SUPPORT model, which lends itself well to a surface model shown in Figure 14-1. The wide end is fixed over the area defined by the split face.

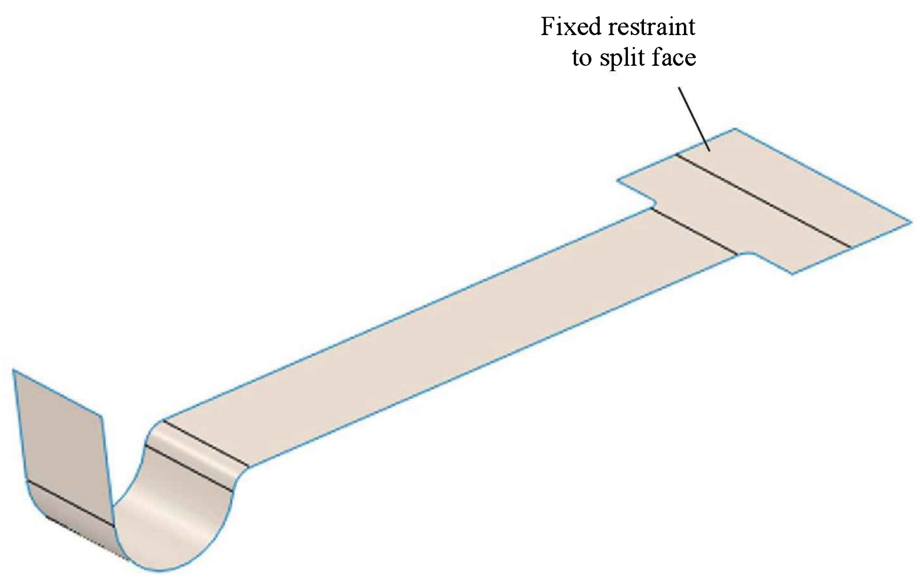


Figure 14-1: The SUPPORT model consists entirely of surfaces.

Restraint symbols are not shown.

The SUPPORT bracket is subjected to harmonic base excitation with displacement amplitude of 0.05mm (Figure 14-2).

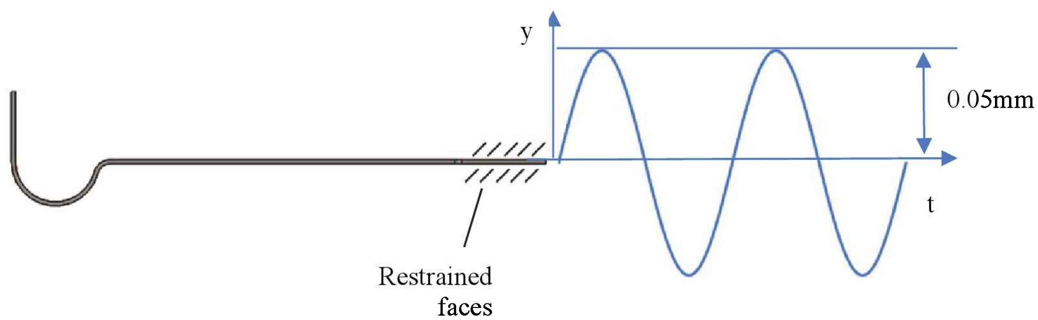


Figure 14-2: Harmonic base excitation applied to the SUPPORT model.

The restrained face performs oscillations with an amplitude of 0.05mm

We want to investigate the bracket response to the harmonic base excitation shown in Figure 14-2 when the frequency of excitation is close to the natural frequency of the SUPPORT. This exercise will serve to study the differences between **Modal Time History** (**Time Response)** and **Harmonic** (**Frequency Response**) analyses. Both these analyses are based on the **Modal Superposition Method** and require the pre-requisite results of a **Frequency** (**Modal**) analysis.

Create a **Frequency** study titled *Modal* and define **Fixed** restraint as shown in Figure 14-1. Define the shell element thickness as 3mm and mesh the model with the default element size.

Run the *Modal* study to find the first four modes shown in Figure 14-3.

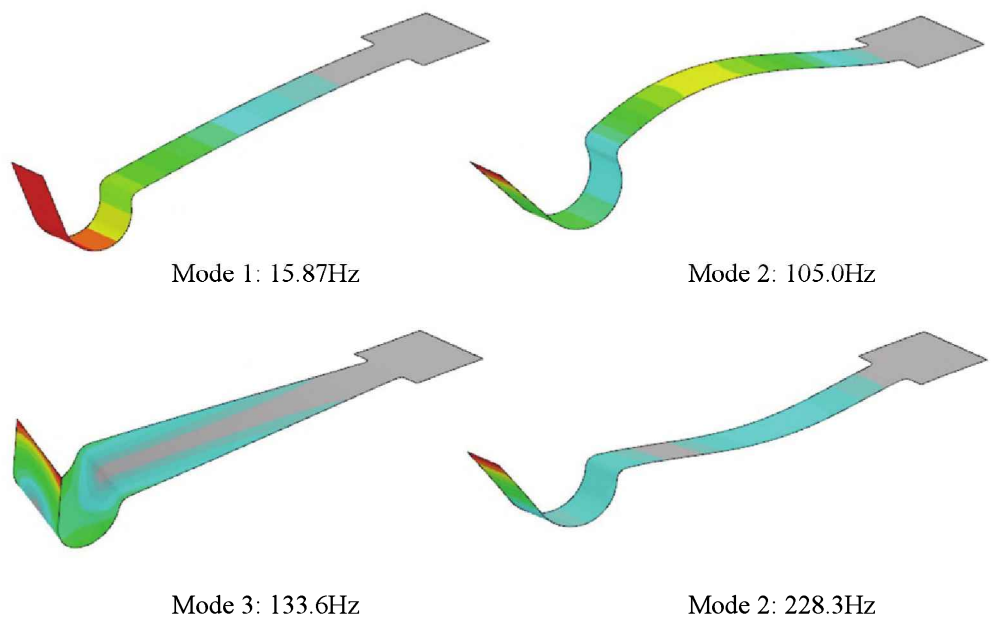
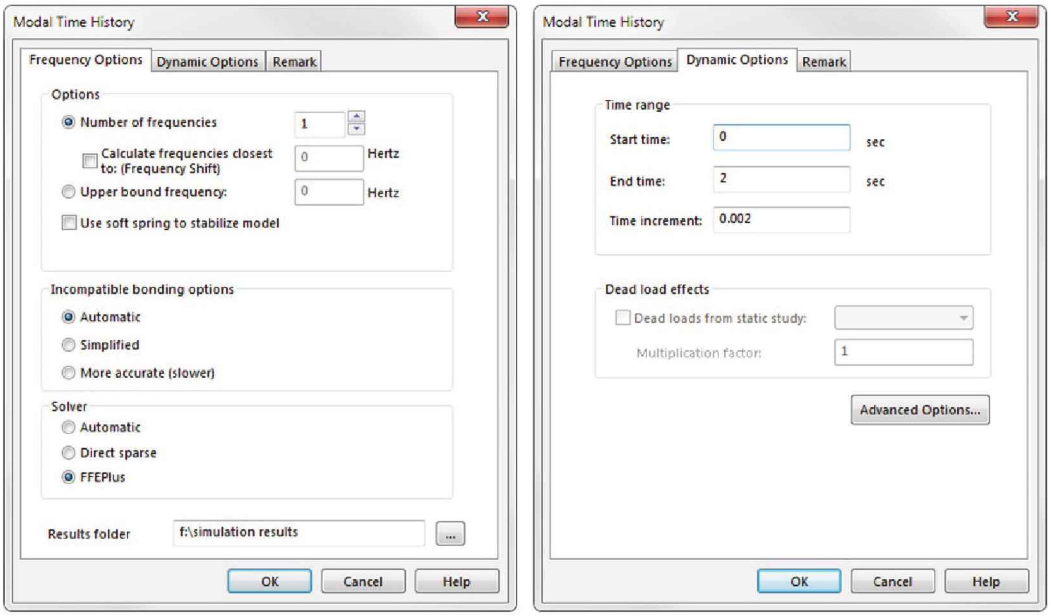


Figure 14-3: The first four modes of the SUPPORT model.

Review the above modes with the undeformed model superimposed on the deformed plots. The undeformed shape is not shown here to avoid overcrowding of this illustration.

A review of the modal shapes demonstrates that the first modal shape is coincident with the direction of the base excitation shown in Figure 14-2. Therefore, a base excitation with a frequency close to 15.87Hz will excite this mode.

Create a **Modal Time History** study titled *01 time response 0%*; there will be no damping present in this study. Define study properties as shown in Figure 14-4.



Frequency Options Dynamic Options

Figure 14-4: Properties of Modal Time History study *01 time response 0%.*

Only one mode is specified in the Frequency Options.

We'll study the model's response to base excitation with frequencies close to the first natural frequency. The first mode will dominate the vibration response, therefore only this first mode is specified in **Frequency Options**. In the **Dynamic Options** specify the duration of the analysis as 2s with a time step of 0.002s which consequently defines 1000 solution steps (Figure 14-4).

The assumption that the vibration response may be represented by the response of the first mode will only work well to study the response when the excitation frequency is close to the natural frequency. It wouldn't be valid for a more general study.

In the **Modal Time History** study, the base excitation must be specified as an explicit function of time. Since the base excitation is a harmonic function of time we may use a pre-defined **Harmonic Loading** to define the **Time Curve**. Details of the base excitation definition are explained in Figure 14-5.

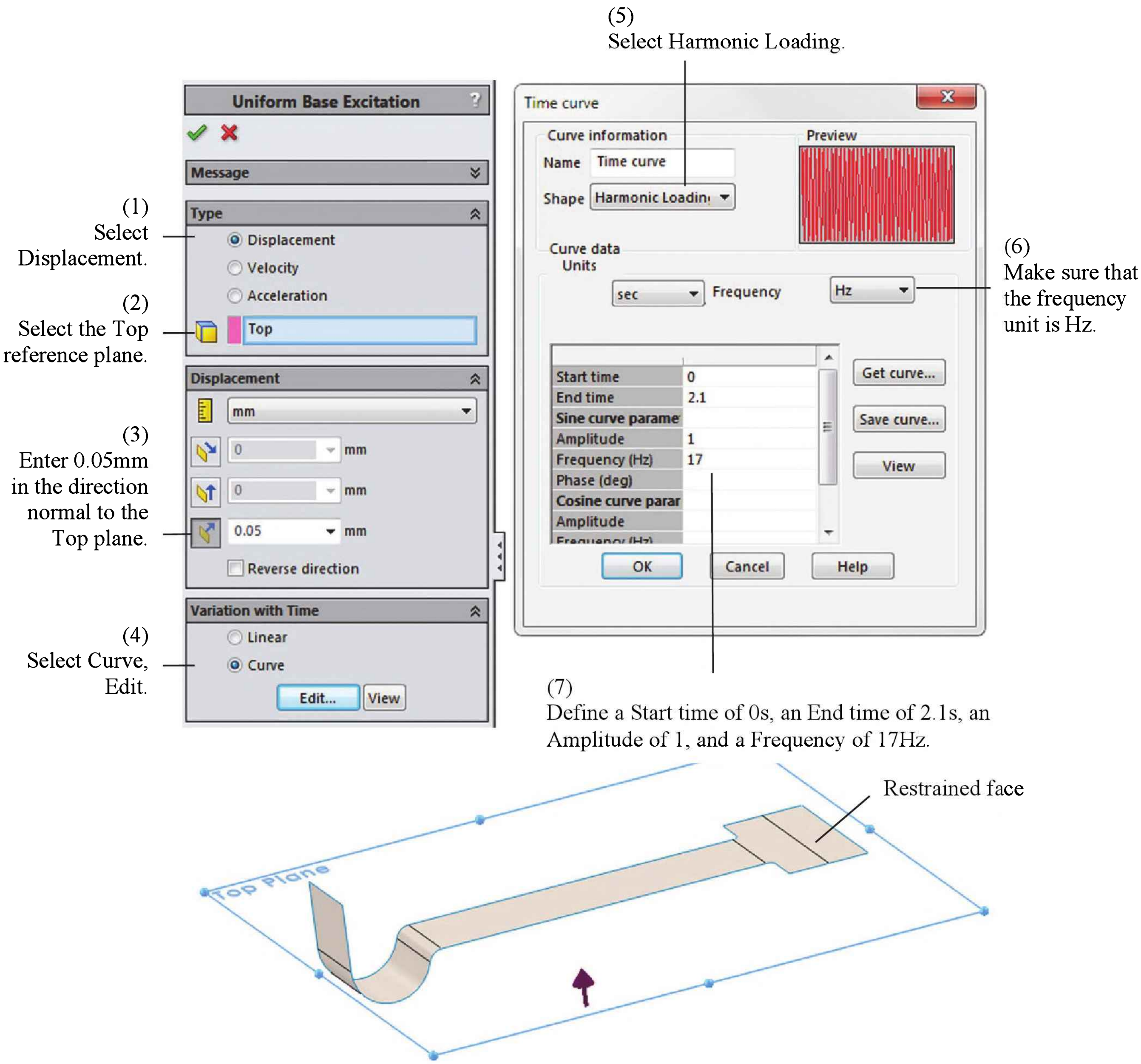


Figure 14-5: Base excitation definition.

Amplitude is defined as 1 in the Time Curve window. This is multiplied by the displacement defined in the Uniform Base Excitation window.

The End time is slightly longer than the duration of that defined in the study properties.

The excitation is applied to all locations restrained in the direction normal to the **Top** reference plane. Therefore, even though the **Uniform Base Excitation** symbol is located in the geometric center of the model, the excitation is applied only to the restrained face.

Review the **Sensor** definition as shown in Figure 14-6.

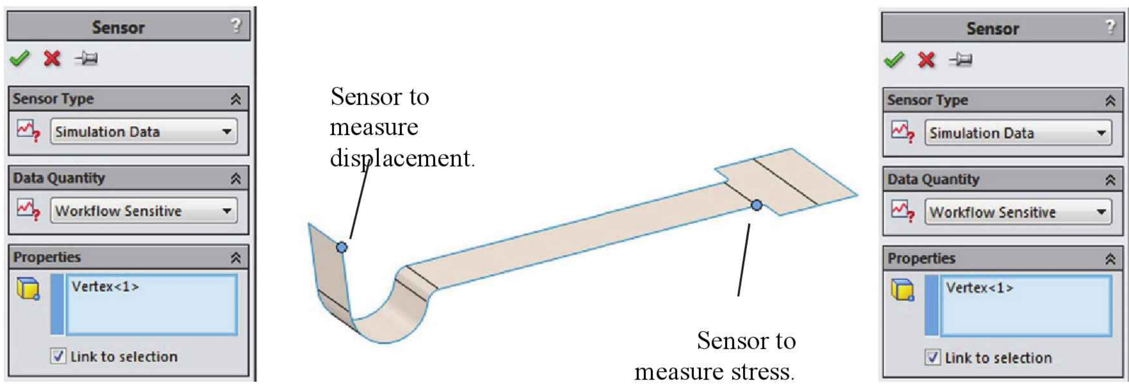
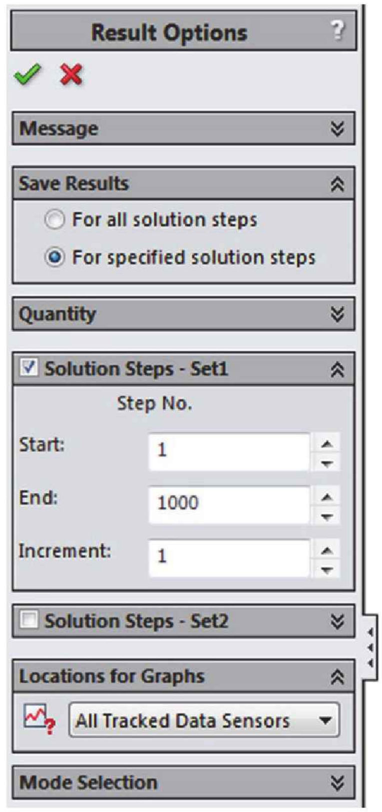


Figure 14-6: The two sensor definitions.

This is for review only; the SUPPORT model comes with this sensor already defined.

Figure 14-6 shows two sensors; the sensor on the left called “Displ” will be used to construct displacement response graphs. The sensor on the right called “Stress” is located close to where the maximum stress will be found and will be used to construct stress response graphs.

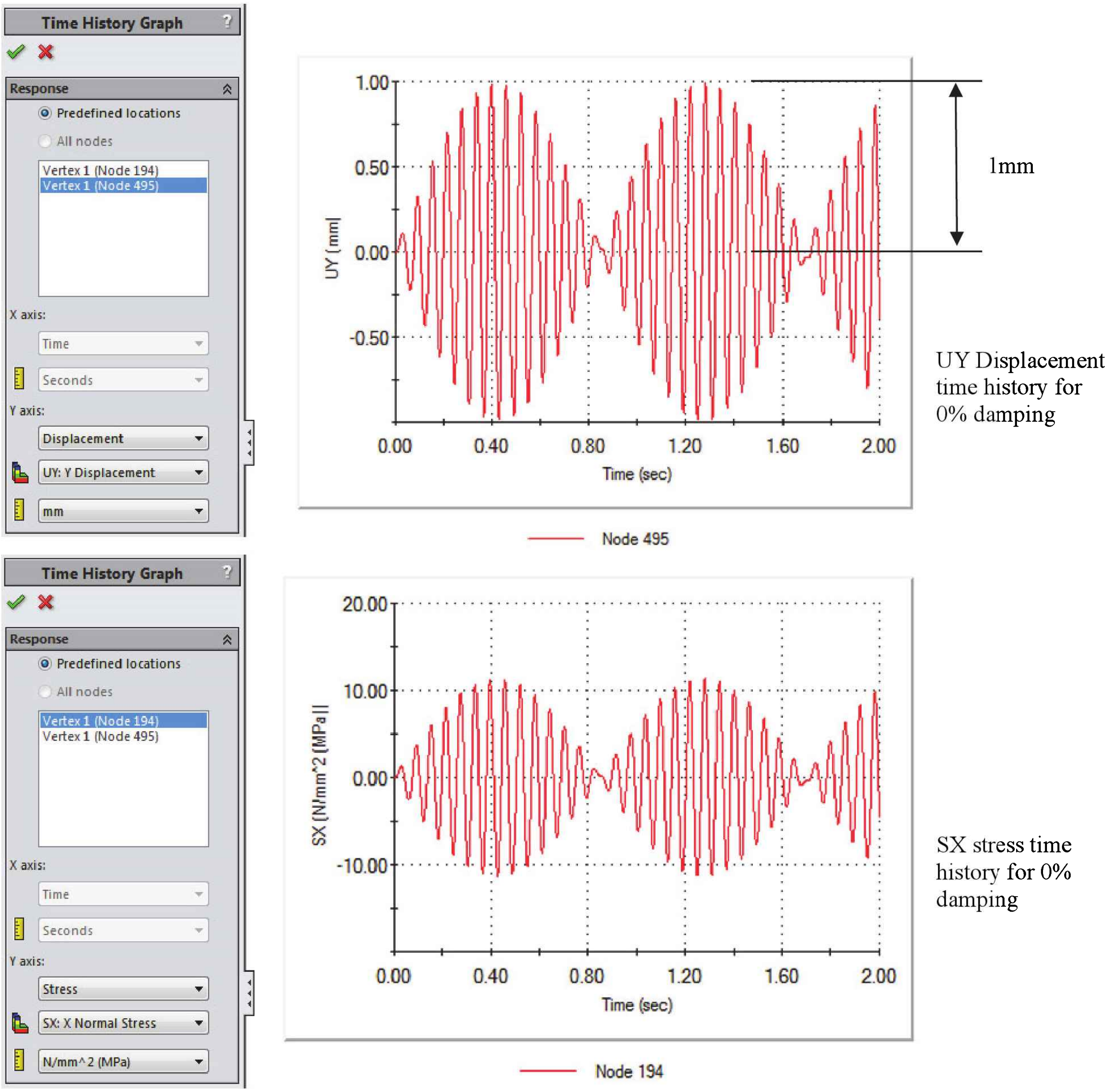
Define **Result Options** as shown in Figure 14-7.



<u>Figure 14-7: Result Options definition; accept all defaults in the Result Option window.</u>

"All Tracked Data Sensors" selection means both sensors.

Run the solution and define the response graphs as shown in figure 14-8.



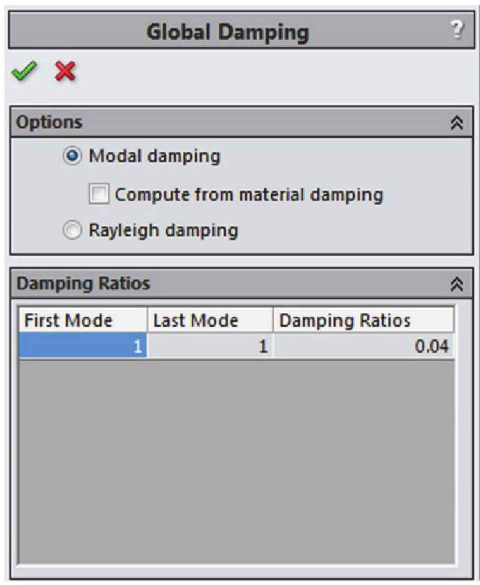
<u>Figure 14-8: Time History of UY displacement and SX stress in the locations of the sensors with 0% damping in the model.</u>

The total time of 2s corresponds to the time duration defined in the study Properties.

The SX stress sensor registers stress on the top of the shell element.

The graphs in Figure 14-8 illustrate the phenomenon called **Beating**. These changes happen for as long as the base excitation is active. The vibration response never reaches a steady state where the amplitude of vibration would stabilized. Remember that no damping has been defined in this study.

To study the effect of damping, copy study *01 time response 0%* into study *02 time response 4%*. The difference between these two studies will be the damping. In study *02 time response 4%* define **Modal Damping** as 4% (Figure 14-9).

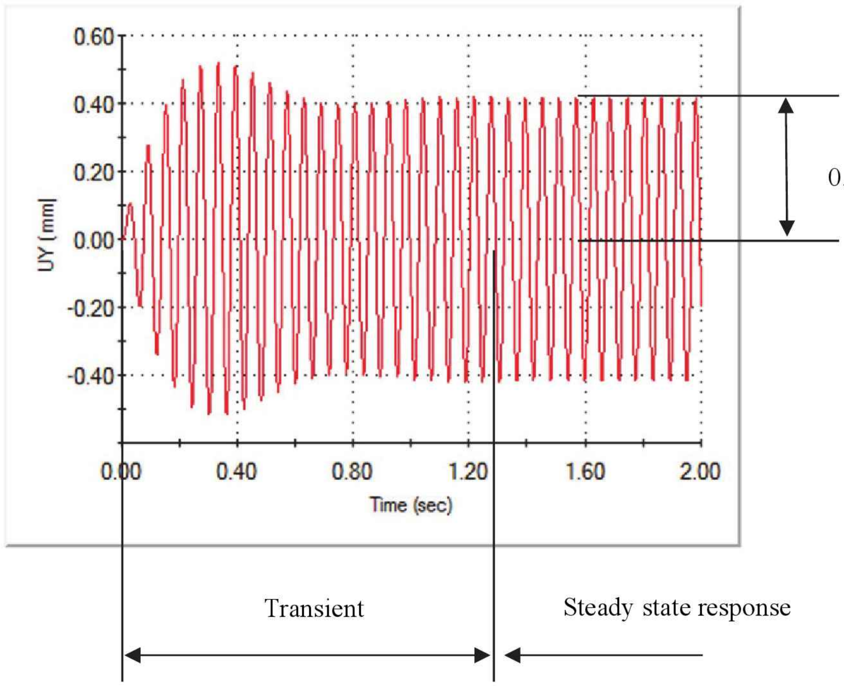


Damping 4%

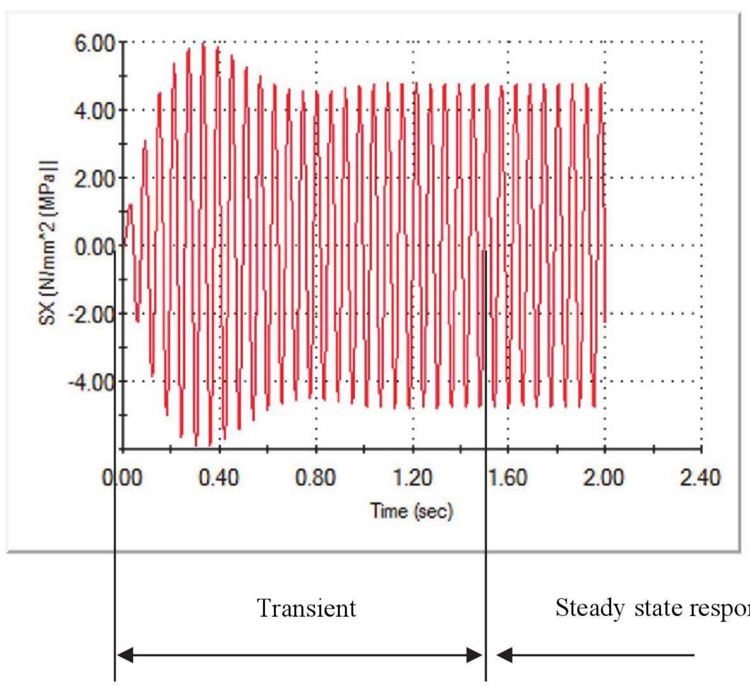
Figure 14-9: Modal Damping definition.

Study is based on one mode; therefore damping is also defined only for one mode.

Obtain the solution of study *02 time response 4%* and review the displacement and time histories the same way as was shown in Figure 14-10.



UY Displacement time history for 4% damping



SX stress time history for 4% damping

<u>Figure 14-10: Time History of UY displacement and SX stress in the locations of sensors with 4% damping.</u>

The transient response disappears after ~1.5s and the displacement amplitude reaches a steady state of 0.41mm.

Here is a summary of both cases: without damping, and with 4% modal damping.

Undamped response

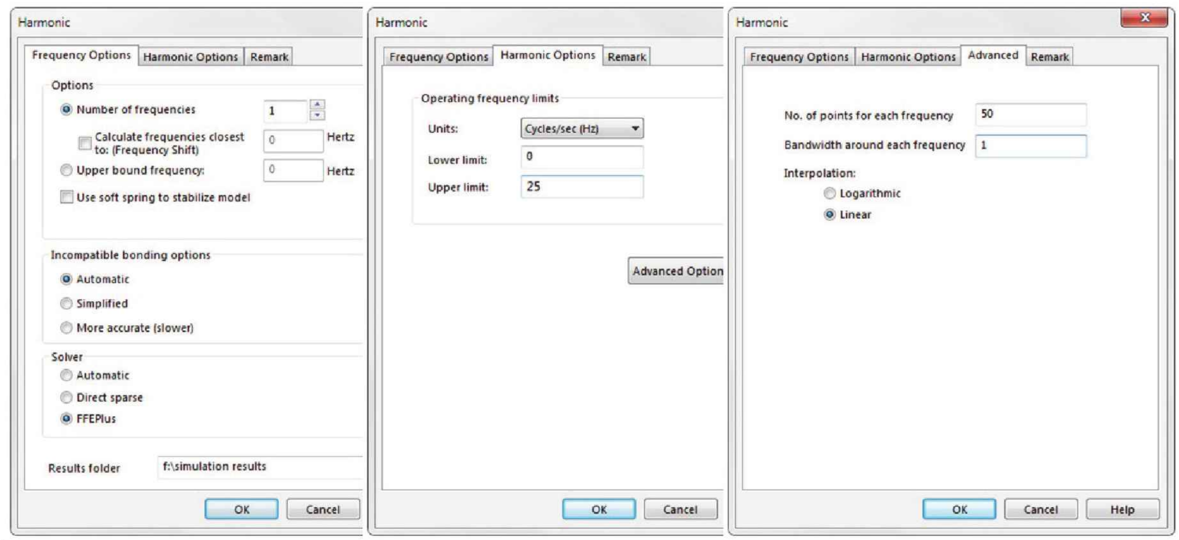
The maximum UY displacement amplitude without damping is ~1mm. This maximum displacement repeats periodically for as long as the base excitation continues. The maximum displacement amplitude amplification is 1/0.05=20. It is so large because the excitation frequency is very close to the natural frequency; the system is very close to resonance. The time distance between the maximum amplitudes is called the **Beating Period**. The **Beating Period**, T, depends on the natural frequency of the system, f_n, and the frequency of excitation, f, which in our case gives the **Beating Period** equal to 0.88s:

$$T = \frac{1}{|fn - f|} = \frac{1}{|15.87 - 17|} = 0.88s$$

Damped response

The maximum UY displacement amplitude with 4% modal damping happens at t=0.3s when the amplitude reaches ~0.52mm. The maximum displacement amplitude amplification is ~10 and periodic amplifications of displacement amplitude diminish with time. The **Beating Period** remains the same as in the case of undamped vibration but it continues for about 1.5 seconds. Then the amplitude of vibration reaches **Steady State** when the amplitude stabilizes at 0.41mm.

We'll now perform a **Frequency Response** study on the same model. To produce results that relate closely to the **Time Response** studies, we will again use only one mode, and a range of excitation frequencies of 0-25Hz to only cover the first natural frequency of 15.87Hz. Create a **Harmonic** study titled *03 frequency response 4%* and define the study properties as shown in Figure 14-11.



Frequency Options Harmonic Options Advanced Options

Figure 14-11: Harmonic study options.

Click Advanced Options in Harmonic Options to open the Advanced tab.

In the **Frequency Options**, specify only one mode. This way the study will only be based on the first mode (15.87Hz). In the **Harmonic Options**, specify the range of excitation frequencies as 0-25Hz. This captures both the natural frequency (15.84Hz) and the excitation frequency (17Hz) used in the previous **Modal Time History** studies. In the **Advanced Options**, specify the number of points to be 50, a bandwidth of 1, and linear interpolation. This way the frequency range will be covered in 101 equal sized frequency steps with no bias around the natural frequency.

The number of frequency steps performed is 101: 50 steps below the natural frequency, and 50 steps above the natural frequency. The step count includes the step at time t=0s, therefore the total number of steps is 101.

Define **Base Excitation** and **Result Options** the same as in the **Time Response** studies. Define a **Modal Damping** of 4%. Run the study and construct the **Response Graph** showing the UY displacement amplitude as a function of the excitation frequency (Figure 14-12).

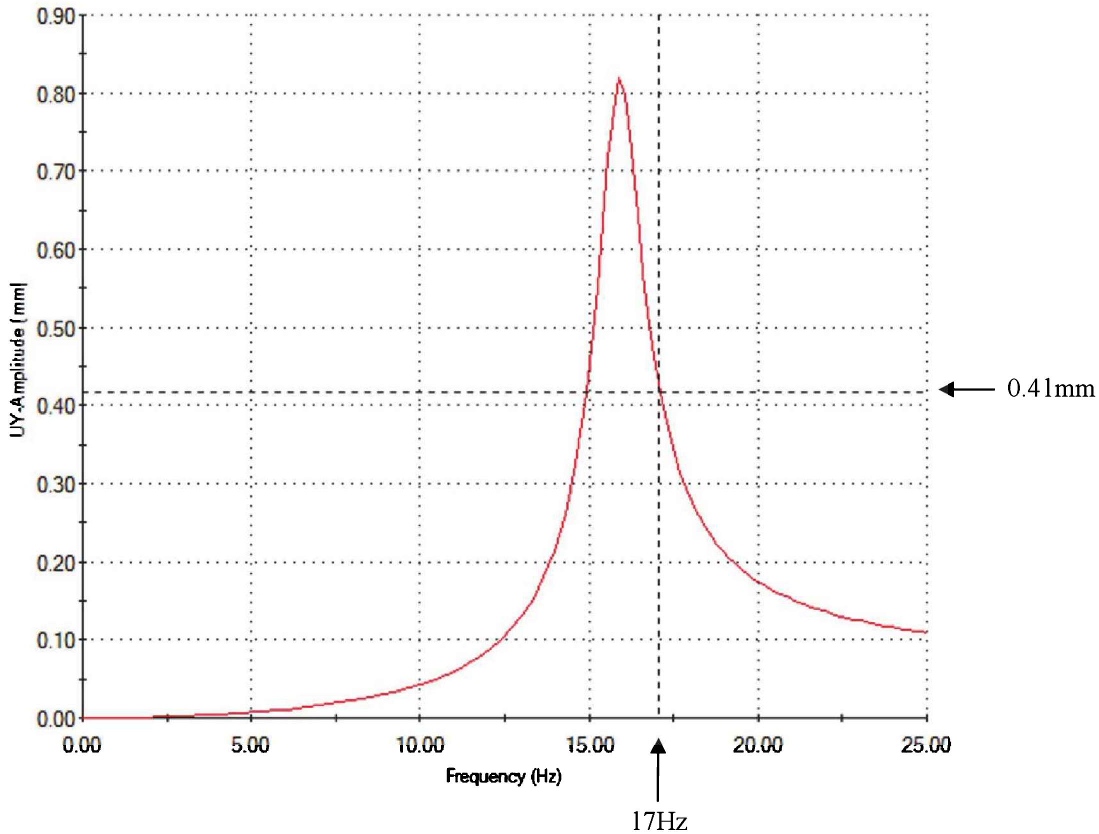


Figure 14-12: Displacement amplitude as a function of the excitation frequency.

The dotted lines cross at the point that corresponds to the steady state response in Figure 14-10.

Figure 14-12 shows the steady state displacement amplitude for all frequencies within the range of 0-25Hz, including the frequency of 17Hz, which was the excitation frequency in the **Time Response** studies. Review the graph and notice that the steady state displacement amplitude for the excitation frequency at 17Hz is 0.41mm. This is the same amplitude as the steady state amplitude found in the *02 frequency response 4%* study.

The graph in Figure 14-12 was constructed using steady state results of a large number (here one hundred and one) of time response analyses, clearly a very time consuming approach. The **Frequency Response** analyses provided these results within one study.

To expand on the topics presented in this chapter you may want to copy the study *02 time response 4%* into *04 time response 4% resonance*. In this new study change the frequency of harmonic excitation from 17Hz to 15.87Hz to make it equal to the first natural frequency of the model with two decimals of accuracy. The displacement time history in the location of the sensor is shown in Figure 14-13.

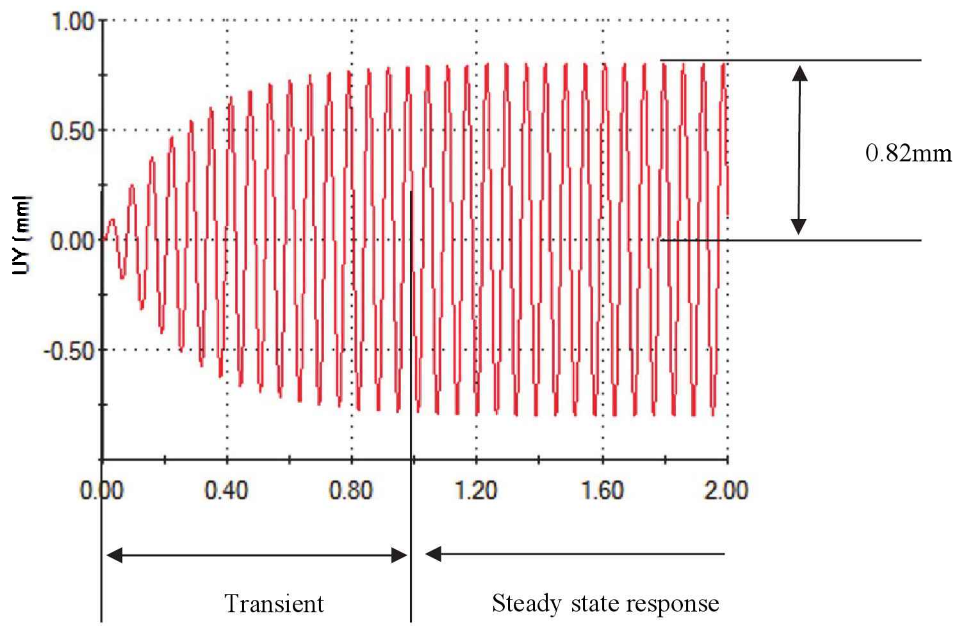


Figure 14-13: Displacement time history for harmonic excitation with a frequency of 15.87Hz which is equal to the natural frequency.

After ~ 1s the system reaches steady state, where the amplitude stabilizes at 0.82mm.

The steady state amplitude of vibration in resonance is 0.82mm. The **Frequency Response** study *03 frequency response 4%* gives the same results. Go back to the graph in Figure 14-12 and notice that the displacement amplitude at resonance is ~0.82mm.

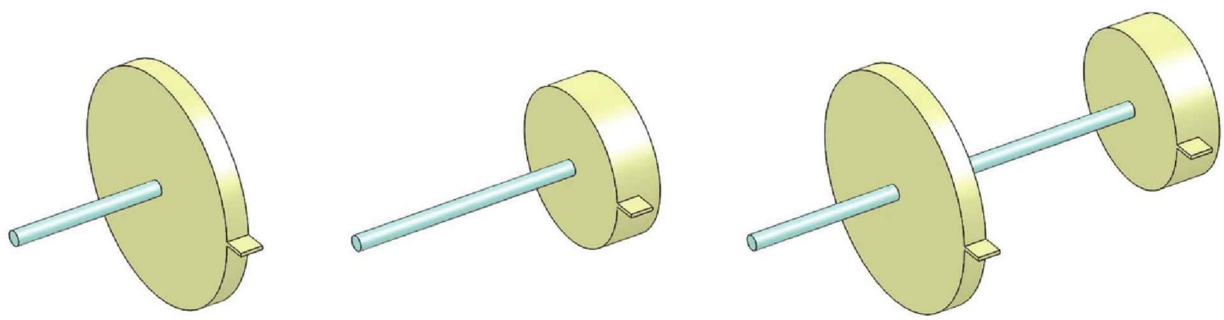
15: Vibration absorption

Topics covered

- Torsional vibration
- Resonance
- Modal damping
- Vibration absorption
- Frequency Response

Procedure

Open assembly model VIB ABSORBER and examine the three assembly configurations shown in Figure 15-1.



01 one disk *02 absorber* *03 full model*

Figure 15-1: Three configurations of the VIB ABSORBER assembly.

Little tabs on both disks help place sensors and aid in visualizing the torsional modes of vibration.

The assembly comes with sensors defined as shown in Figure 15-2.

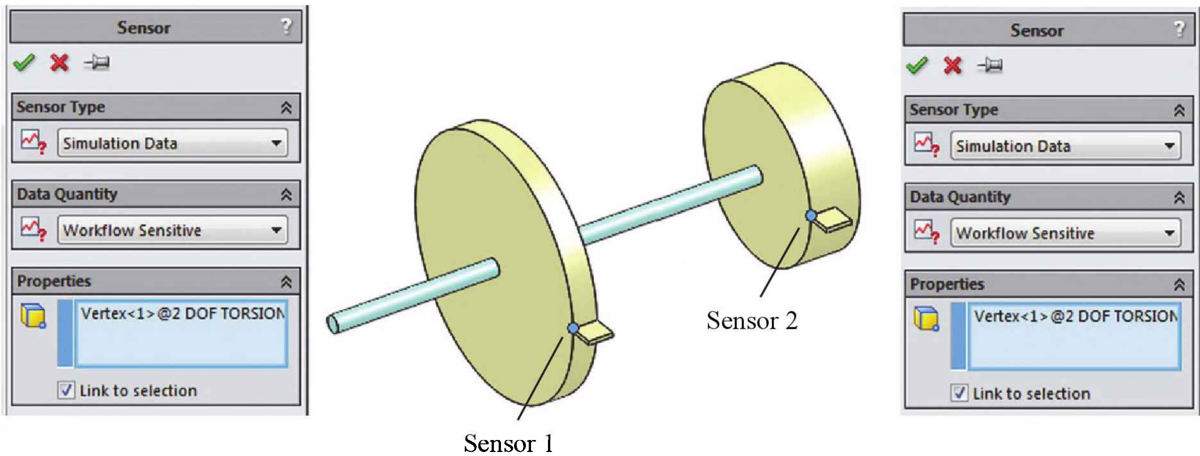


Figure 15-2: Sensor locations in the VIB ABSORBER model.

Sensors are located at the base of each tab.

We start an analysis in the *01 one disk* configuration to model the torsional vibration of one disk subjected to a harmonic torque load. The torque amplitude is 10Nm and does not change with the frequency of oscillations which is 0-80Hz. While 0-80Hz is the range of excitation frequencies, we are particularly interested in the steady state displacement amplitude at a frequency equal to 28Hz. For this purpose we use a **Harmonic** study.

Create a **Harmonic** study titled *01 one disk*. The disk is restrained as shown in Figure 15-3. The cylindrical faces of the shaft and disk are restrained in the radial direction using an **On Cylindrical Faces** restraint. The end face of the shaft has a **Fixed** restraint applied.

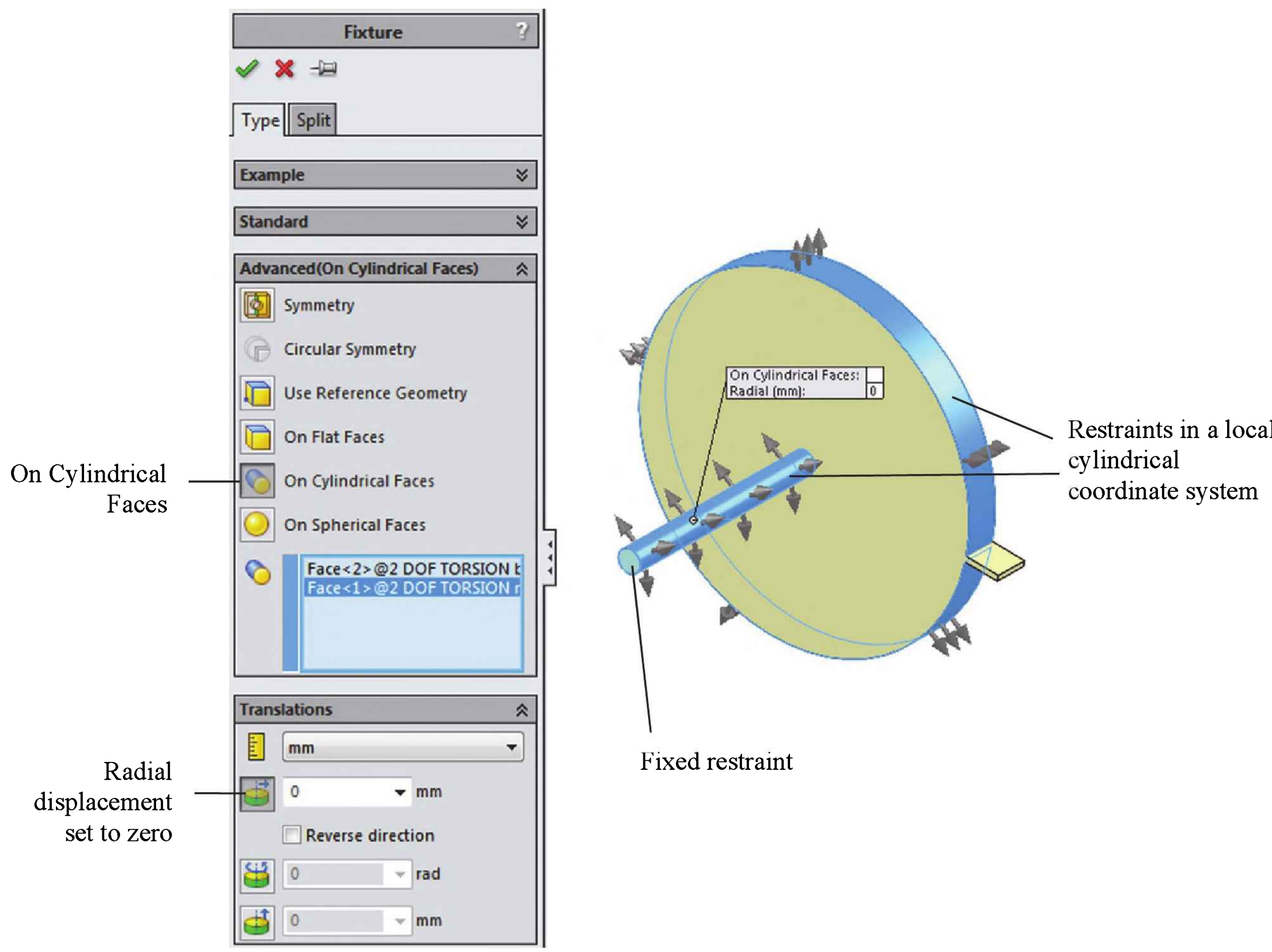
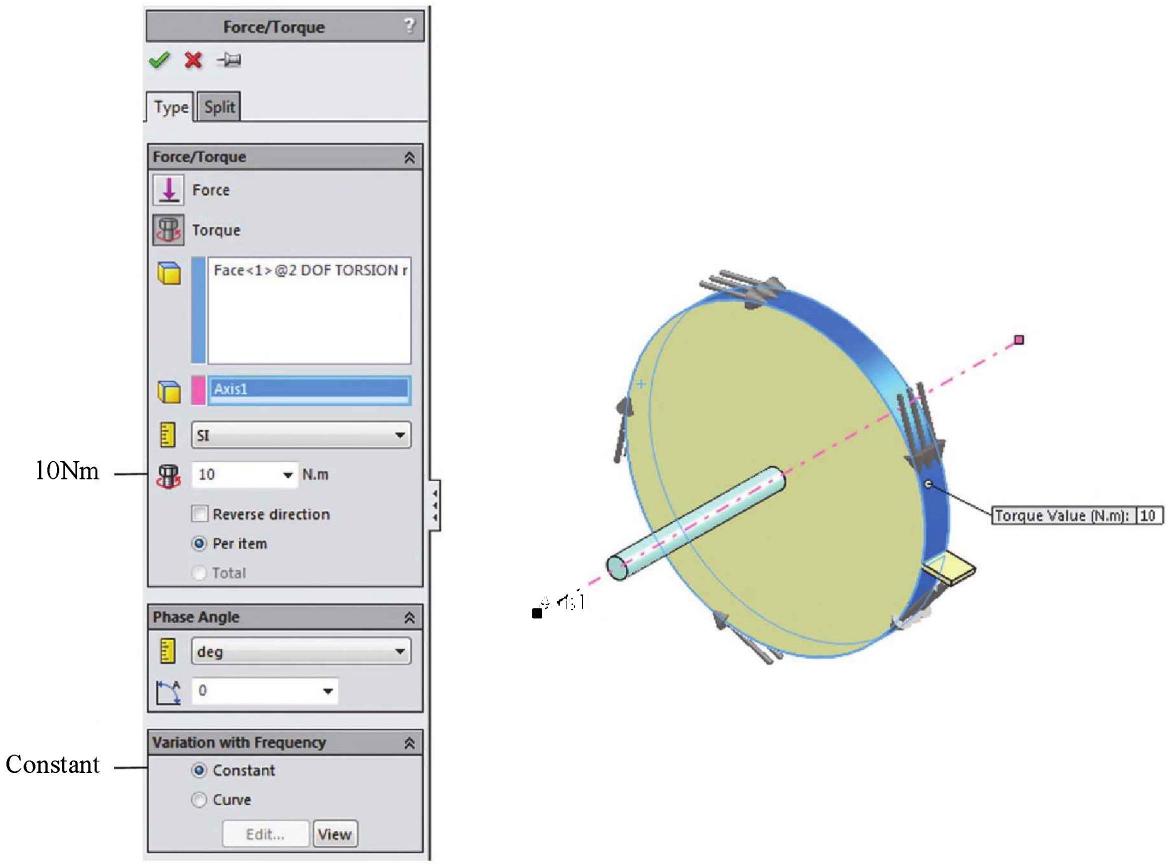


Figure 15-3: Both cylindrical surfaces are restrained in the radial direction.

The Fixed restraint definition window is not shown.

As a result of the restraints shown in Figure 15-3, the disk has only one mode of vibration in the range 0-80Hz and this is a torsional mode with a frequency of 28.1Hz. You may want to confirm this by running a modal analysis.

The disk is subjected to an oscillating torque of 10Nm as shown in Figure 15-4.

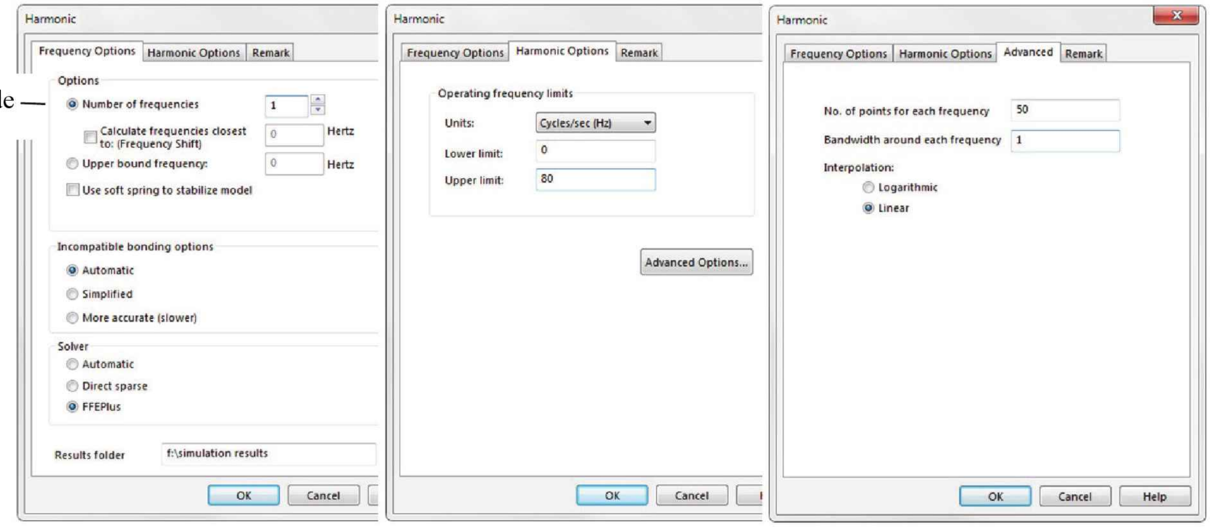


<u>Figure 15-4: Harmonic torque load applied to the outer face of disk.</u>

Selecting Constant for Variation with Frequency means that the torque amplitude does not change with the excitation frequency.

Define study properties as shown in Figure 15-5.

One mode



Frequency Options Harmonic Options Advanced Options

Figure 15-5: Properties of Harmonic study *01 one disk.*

Notice the modified Advanced Options.

In this exercise we are interested in the vibration response between natural frequencies. This is why **Advanced Options** have to be modified.

Considering that only one mode is specified, 50 points defined in **Advanced Options** means that 50 frequency steps will be performed before reaching the natural frequency, another 50 after that and one for the natural frequency. Linear interpolations means that frequency steps won't be biased towards natural frequencies but will be evenly distributed over the frequency range 0-80Hz.

Define modal damping as 1% (Figure 15-6).

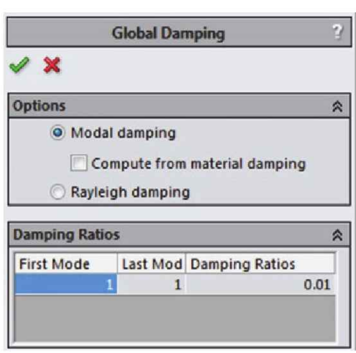


Figure 15-6: Modal damping defined as 1%.

Only one mode is shown in the Global Damping window because only one mode has been specified in the Frequency Options.

Define the **Result Options** as shown in Figure 15-7.

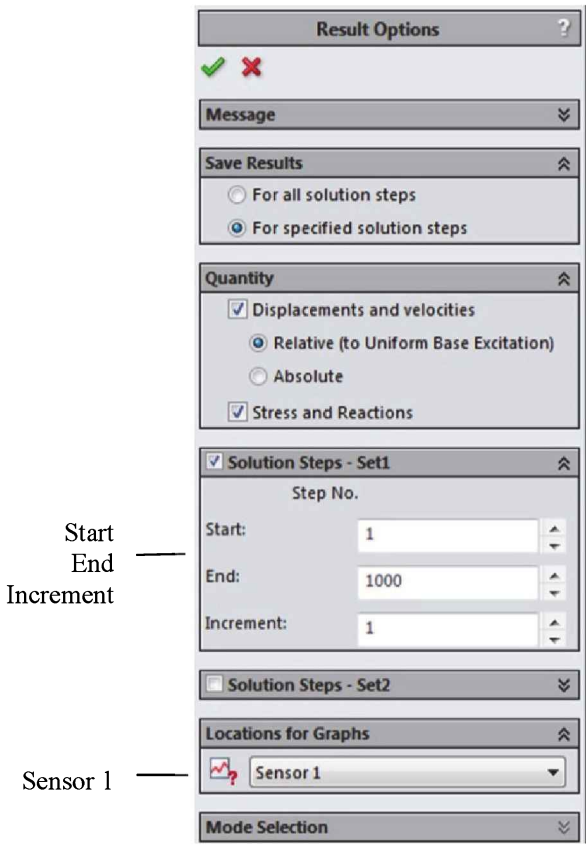


Figure 15-7: Definition of Result Options.

Only one sensor is available in the 01 one disk study which is associated with the 01 one disk assembly configuration.

A start step of 1 and an end step of 1000 with increments of 1 means that that graphs may contain no more than 1000 points. It is not the number of actual time steps.

Mesh the model with the default element size and obtain the solution. Define a **Response Graph** as shown in Figure 15-8:

Vertex 1 is in the Sensor 1 location

UY displacement component

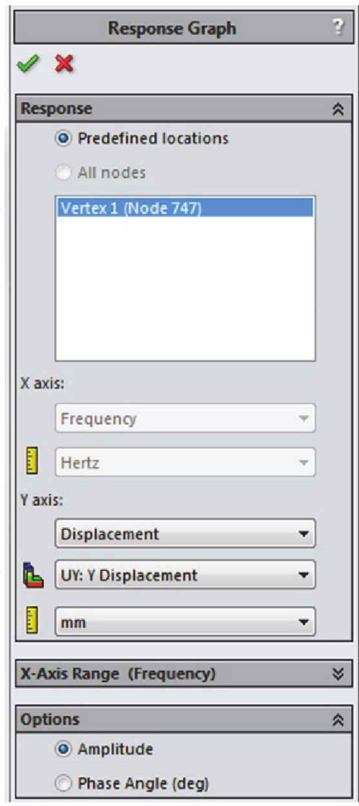
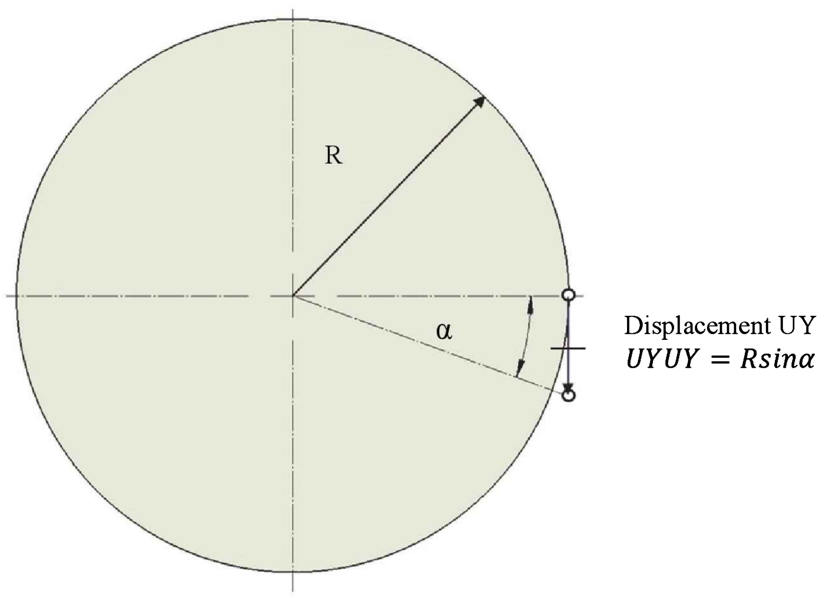


Figure 15-8: Definition of the displacement amplitude response graph.

We use a linear displacement on the circumference of the disk as a measure of angular rotation.

The UY displacement component selected in the **Response Graph** in Figure 15-8 is a linear displacement in the circumferential direction. If desired, it can be translated into angular displacement as shown in Figure 15-9.



$$UY = Rsin\alpha$$

Figure 15-9: UY displacement can be translated into angular displacement.

The above equation is valid only for small displacements.

The UY displacement amplitude frequency response graph is shown in Figure 15-10.

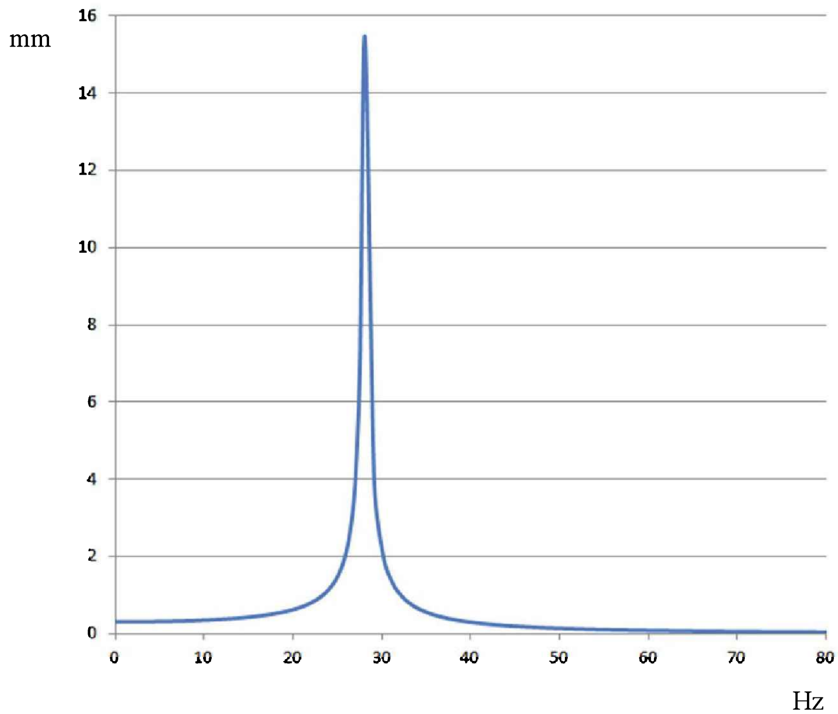


Figure 15-10: UY displacement amplitude frequency response. The peak corresponds to the natural frequency of 28.1Hz.

This graph has been formatted in Excel.

Damping in the system is low (1%), therefore the maximum displacement amplitude happens when the excitation frequency equals the natural frequency. In other words, the system experiences resonance at 28.1Hz.

Now, let's imagine a situation where the disk is subjected to harmonic oscillations with a frequency of 28.1Hz which is the resonant frequency. We find the displacement amplitude too large and want to reduce it. We'll do that by attaching a **vibration absorber**, the natural frequency of which will be equal to the frequency of excitation. Notice that this is the frequency of the absorber before it is attached to the disk.

See the absorber before attaching it to the disk. Change to configuration *02 absorber*. Create a **Frequency** study; apply restraints to the shaft and to the disk the same way as shown in Figure 15-3 and find the first mode of vibration to confirm that it is a torsional mode with a frequency of 28.1Hz (Figure 15-11).

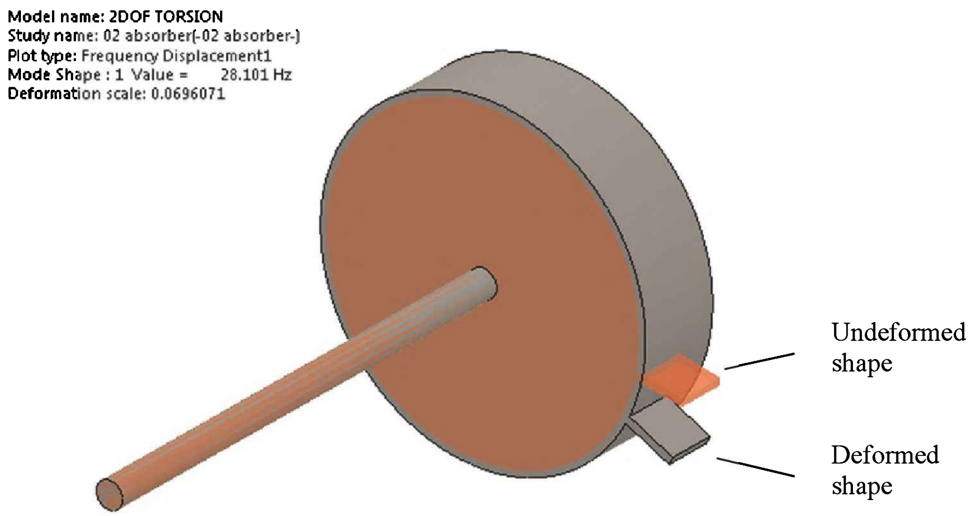


Figure 15-11: The first mode of vibration of the absorber: a torsional mode with a frequency of 28.1Hz.

The circular shape makes visualization of the torsional mode difficult. Tabs help the visualization.

The apparent growth of the disk diameter visible during modal animation is caused by the fact that displacement trajectories are straight lines tangent to the circumference of disk. This is an inherent property of linear analysis.

Now we are ready for the main part of this exercise where we analyze the effect of the vibration absorber attached to the disk. Switch to assembly configuration *03 full model* and create a **Harmonic** study titled *03 full model*. In this configuration, the assembly consists of the disk and the absorber.

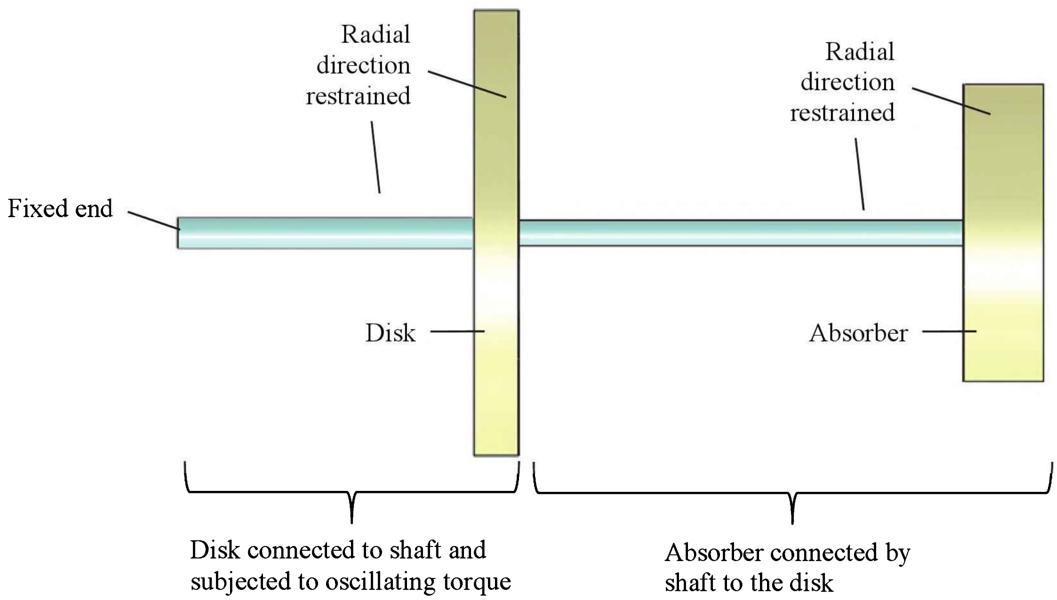


Figure 15-12: The assembly model with the vibration absorber attached to the disk.

All cylindrical faces are restrained in the radial direction.

Define a **Harmonic** study with properties shown in Figure 15-13; two modes are now specified.

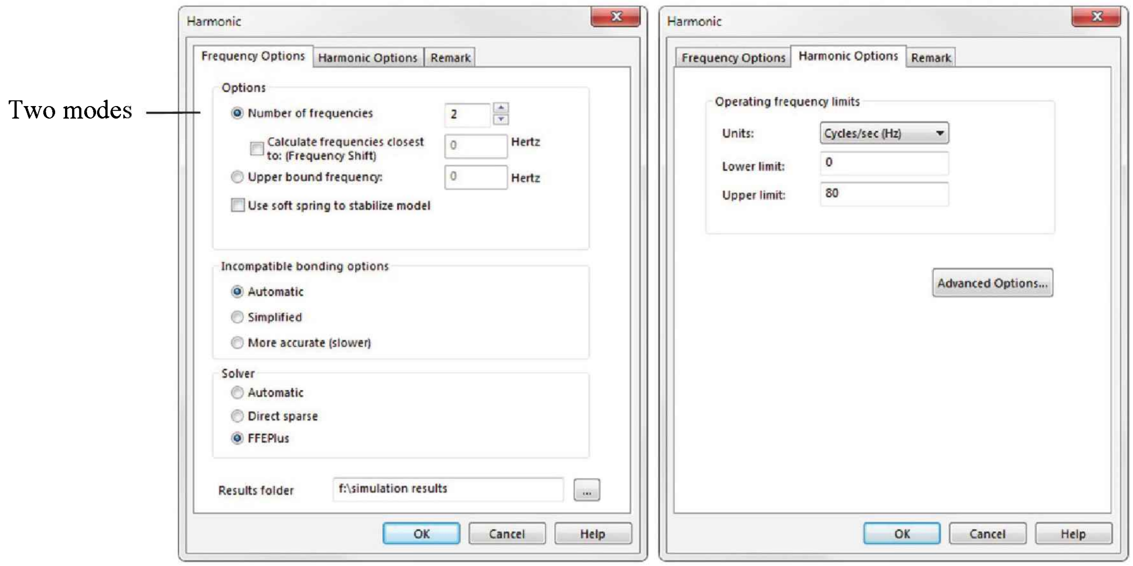


Figure 15-13: Properties of the Harmonic study for the model in the 03 full model configuration.

Advanced Options are not shown; make them the same as in Figure 15-5.

Apply restraints as shown in Figure 15-12. Define a harmonic torque excitation the same as in Figure 15-4 and **Results Options** as shown in Figure 15-14.

All Tracked Data Sensors

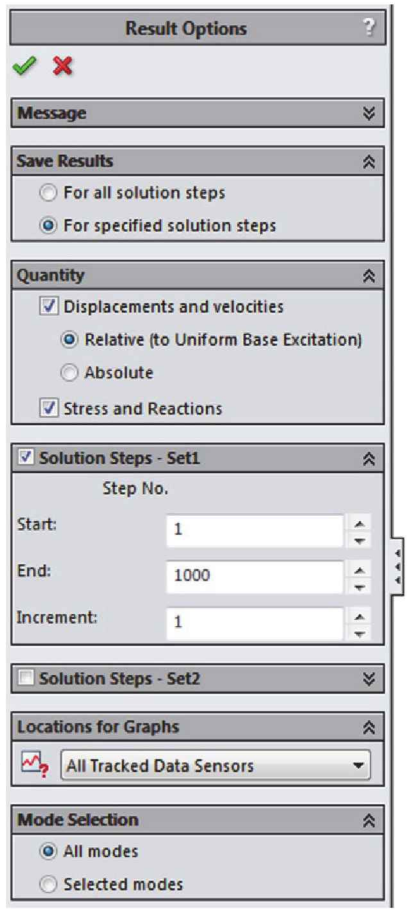
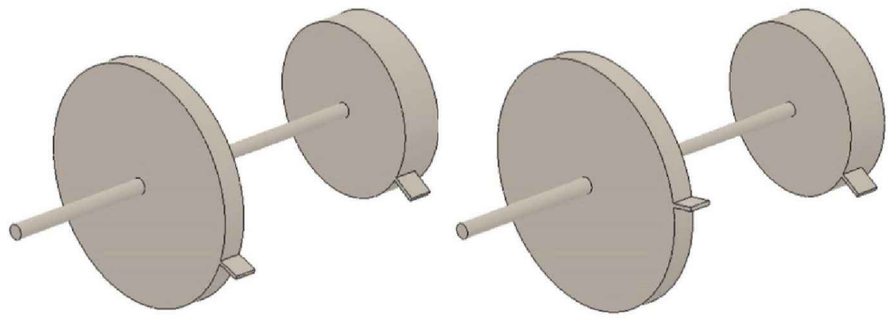


Figure 15-14: Results Option for the *03 full model* configuration.

"All Tracked Data Sensors" means Sensor1 and Sensor2.

The system has two natural frequencies in the range of 0-80Hz; you may confirm this by running a separate **Frequency** analysis or a modal analysis from inside the **Harmonic** study. Modal frequencies and shapes are shown in Figure 15-15.



Mode 1: 21.0Hz   Mode 2: 37.5Hz

Figure 15-15: Two modes of vibration in the 0-80Hz range are torsional modes.

Animate the modal shape to see that in mode 1 both disks oscillate in the same direction and in mode 2 they oscillate in the opposite directions.

Run the solution and notice that it proceeds in 151 steps because the **Advanced Options** in the study properties specify 50 points for each frequency (Figure 15-16).

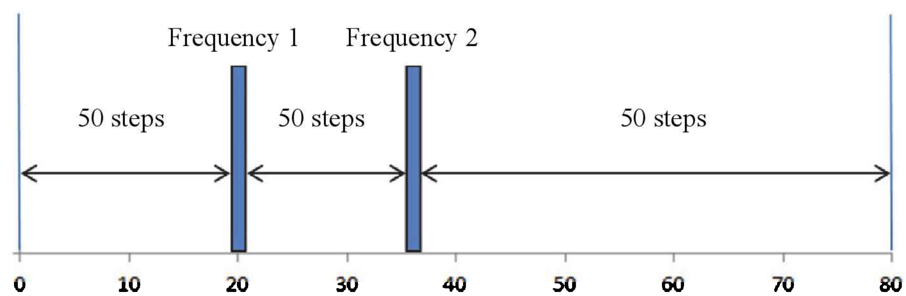


Figure 15-16: The range of frequencies is divided in three parts: 50 steps each.

The step count includes the step at time t=0, therefore the total count is 151.

Once the solution completes, construct the **Response Graph** showing UY displacement amplitude as a function of the excitation frequency.

Response graphs formatted in Excel are shown in Figure 15-17.

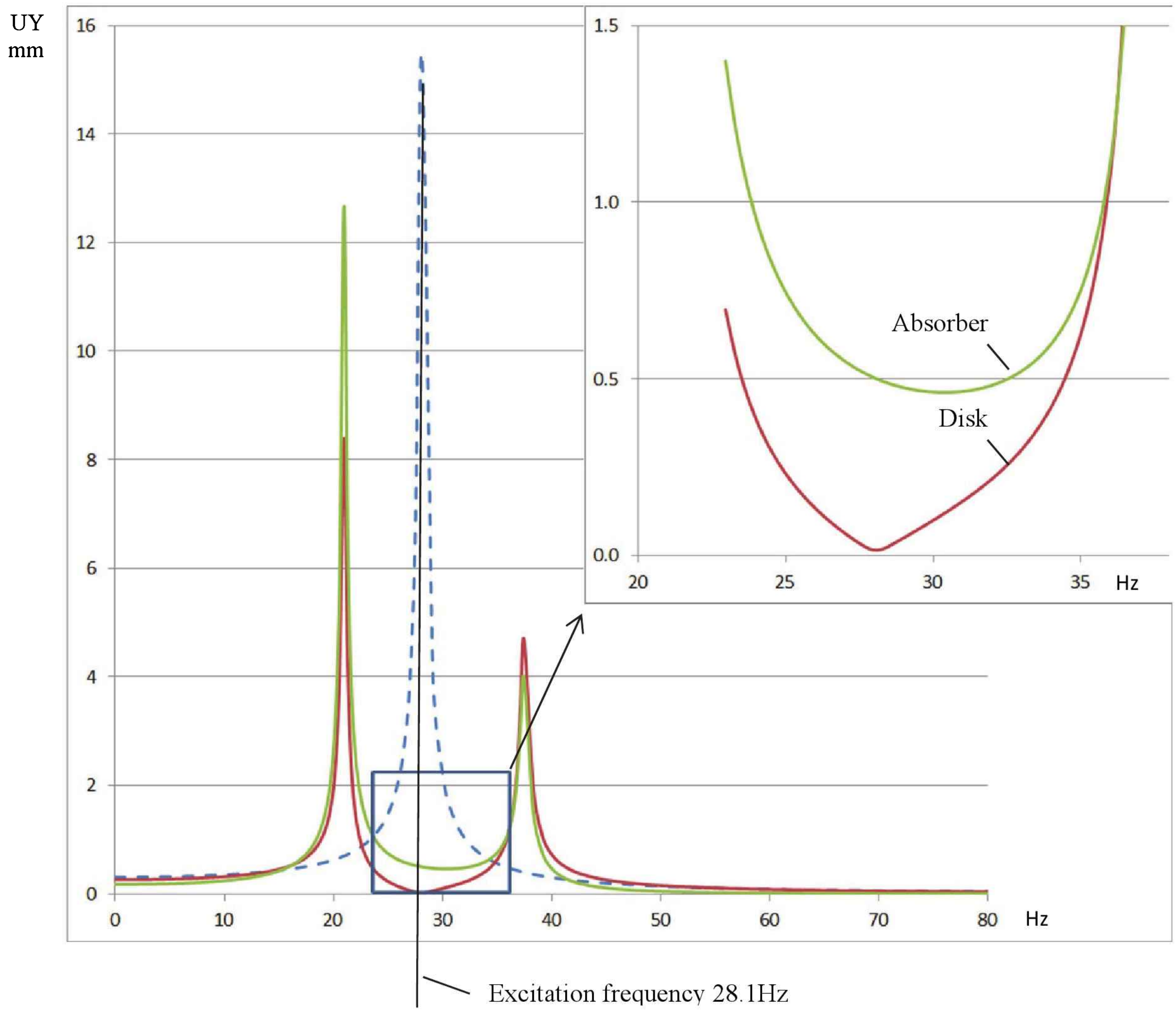


Figure 15-17: A summary of UY amplitude frequency response from the single disk study (dotted line) in *study 01 one disk*, and the disk with the absorber in study *03 full model.*

The main graph summarizes results from studies *01 one disk* (dotted line) and *03 full model* (solid lines). The dotted line repeats the results shown in Figure 15-10. The insert shows results from study *03 full model* in the narrow range of frequencies 24-36Hz.

It can be seen that attaching the absorber de-tunes the system from the resonant frequency of 28.1Hz. The system now has two resonant frequencies in the range 0-80Hz: 21.0Hz and 37.5Hz as shown by the two peaks in Figure 15-17 and modal shapes in Figure 15-14.

The results of the **Harmonic** study provide an overview of the system response in the specified range of frequencies. We are particularly interested in the frequency of 28.1Hz which is the frequency of the torque excitation and the frequency of the absorber before attaching it to the main disk. As Figure 15-17 illustrates it, when the absorber is attached to the main disk and the main disk is excited with a 28.1Hz frequency, the vibration of the main disk stops at this frequency of excitation; see the insert in Figure 15-17 showing a very low amplitude of vibration. We say that the vibration has been "absorbed" by the added vibrating system, and hence the name **Vibration Absorber**.

Controlling vibrations by means of adding an absorber is practical only in cases where the frequency of excitation is constant and adding components is possible. Assembly model 2DOF TORSION presents an idealized system performing angular vibration.

We may treat it as a discrete system because:

1. The shafts are responsible for stiffness properties and the disks (the main disk and absorber) are responsible for inertial effects.
2. The system is subjected to restraints in such a way as to have two separate torsional modes of vibration within the range of the investigated frequencies.

The model does not address important problem such as stress, fatigue, etc., but works well to illustrate the principle of vibration absorbing which applies to both linear and angular vibration systems.

16: Random Vibration

Topics covered

- Random vibration
- Power Spectral Density
- RMS results
- PSD results
- Modal excitation

Random vibration

Random vibrations are non-periodic. Knowing the history of random vibration, we can predict the probability of occurrence of acceleration, velocity and displacement magnitudes, but cannot predict the precise magnitude at a specific instance in time.

Random vibration is composed of a continuous spectrum of frequencies. The huge amount of time history data makes it impractical or impossible to solve random vibration problems using time response analysis.

For most structural vibrations, the excitation, such as a base acceleration, alternates about zero. Consequently, mean values characterizing the excitation, as well as responses to that excitation such as displacement or stress, are equal to zero and can't be used to characterize random vibration responses. For this reason, results of a random vibration analysis are given in the form of Root Mean Square (RMS) values.

To explain the concept of an RMS value, refer to the graph in Figure 16-1 which shows the acceleration time history (acceleration as a function of time) of random vibration expressed in units of gravitational acceleration [G].

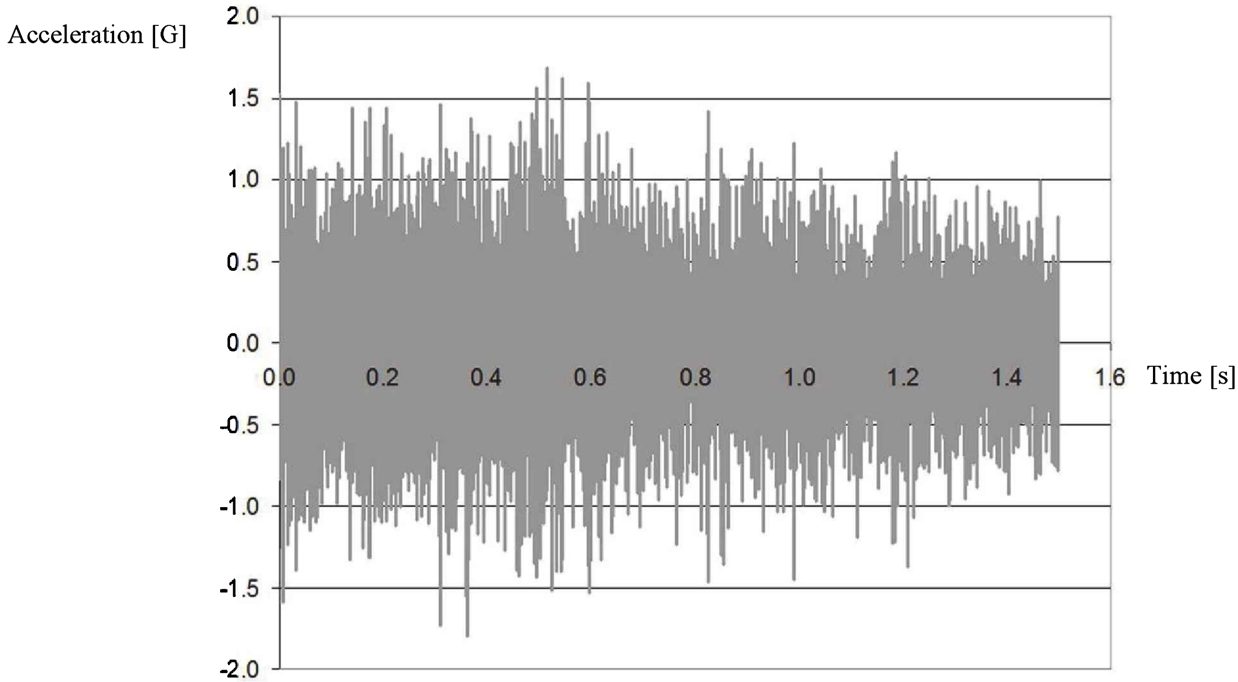


Figure 16-1: An example of acceleration time history data collected during 1.5s.

Considering the sampling rate of 5000 samples per second, this time history curve contains 7500 data samples.

The acceleration time history shown in Figure 16-1 has a zero mean value. However, if we multiply the function by itself, we obtain a function with a positive value. This squared function will be well suited to characterize the acceleration time history because it's mean value will not be zero. This mean value of square acceleration time history is the mean square value and has units of [G^2].

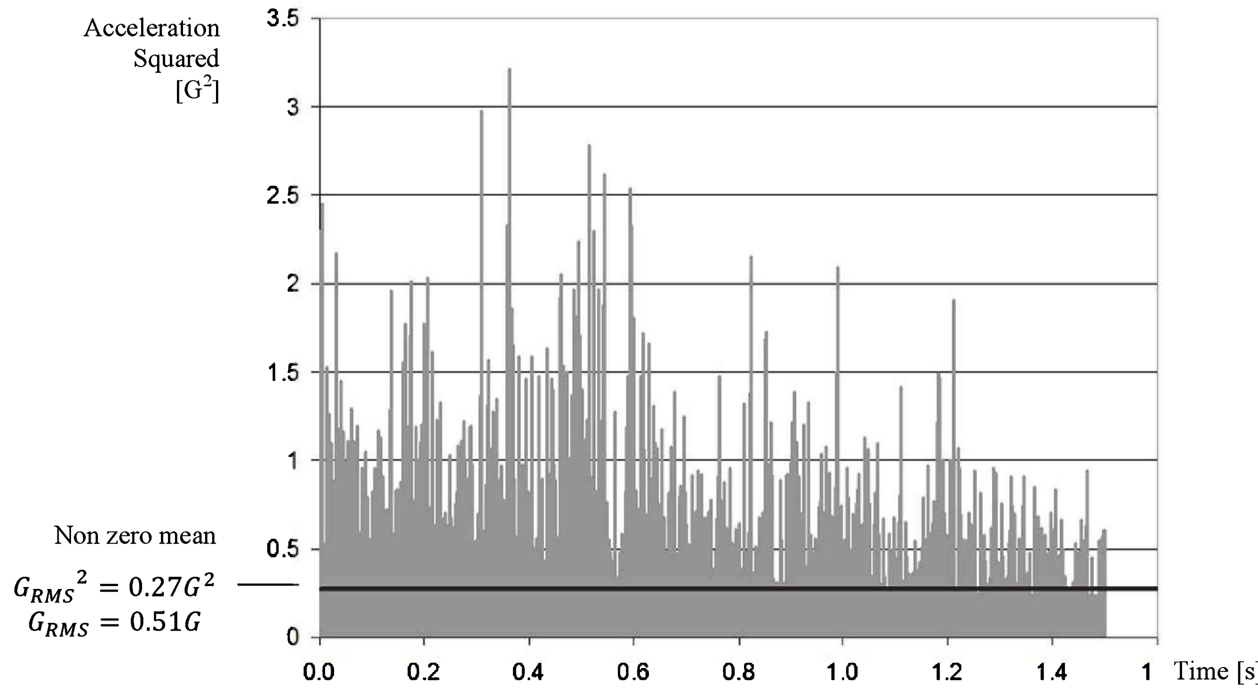


Figure 16-2: Squaring the acceleration time history function shown in Figure 16-1 produces a function with a non-zero mean.

As shown in Figure 16-2, calculating the mean square value gives $G_{RMS}{}^2$ = 0.27G. The square root of the mean square value gives G_{RMS} = 0.51G.

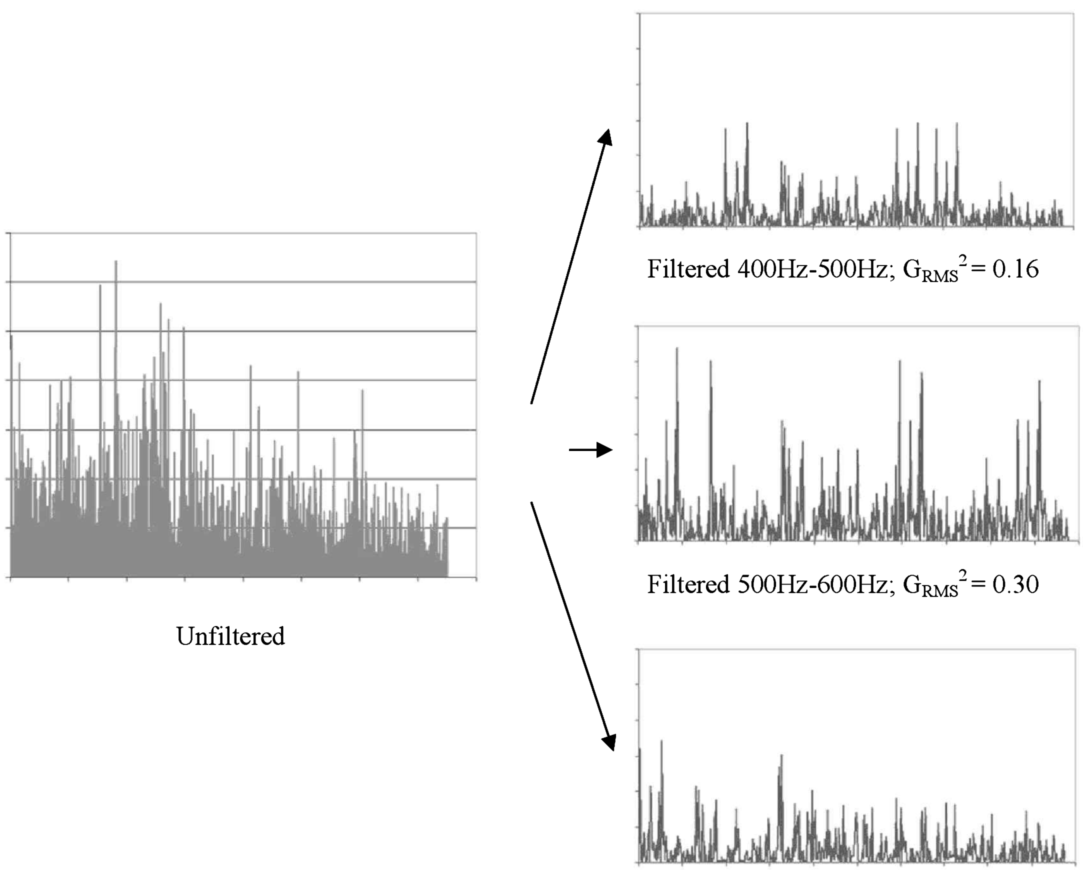
The square root of the mean value is the root-mean-square (RMS) acceleration and has units of [G]. The same applies to RMS displacement, velocity, stress etc.

In random vibration, the magnitudes of acceleration, velocity, displacement etc. all follow a normal distribution. The RMS value corresponds to one standard deviation σ characterizing the normal distribution. To explain this, we refer again to Figure 16-2. The acceleration, as characterized by the given acceleration time history, has a 68% probability of remaining between -0.51G and +0.51G. Consequently, it has a 32% probability of being less than -0.51G or more than 0.51G.

Acceleration Power Spectral Density

Let's assume that the acceleration time history in Figure 16-1 is a stationary random process where probability numbers characterizing this process do not change with time. In this case, the acceleration time history can be used to calculate the Acceleration Power Spectral Density (PSD) curve (the variation of any property with respect to frequency is called "spectrum").

The overall G_{RMS}^2 of random vibrations shown in Figure 16-2 is $0.27G^2$. However, random vibration is composed of a large number of frequencies. Let us say we wish to investigate G_{RMS}^2 individually for a number of frequencies in the range from 0 to 2000Hz. Therefore, we divide the 0-2000Hz range into 20 sections (bins), each 100Hz wide and calculate G_{RMS}^2 characterizing each section by filtering out all frequencies falling outside of the section (Figure 16-3).



Filtered 400Hz-500Hz; $G_{RMS}^2 = 0.16$

Filtered 500Hz-600Hz; $G_{RMS}^2 = 0.30$

Unfiltered

Filtered 600Hz-700Hz; $G_{RMS}^2 = 0.19$

Figure 16-3: G_{RMS}^2 calculated individually for specified frequency ranges.

The graph on the left shows the squared acceleration time history from Figure 16-2. Only three frequency ranges (sections) are illustrated here for brevity.

Having found G_{RMS}^2 values obtained for each frequency range, we can now calculate individual "densities" of G_{RMS}^2 in each section by dividing G_{RMS}^2 in each section by the width of the section. Results obtained for all sections may be plotted as a function of the frequency in the center of each section. This function is called the **Acceleration Power Spectral Density** (Figure 16-4).

Band pass filter	Band center	G_{RMS}^2	Bandwidth	Acceleration PSD
	Hz	$(m/s^2)^2$	Hz	$(m/s^2)^2/Hz$
400Hz - 500Hz	450	0.16	100	0.0016
500Hz - 600Hz	550	0.30	100	0.0030
600Hz - 700Hz	650	0.19	100	0.0019

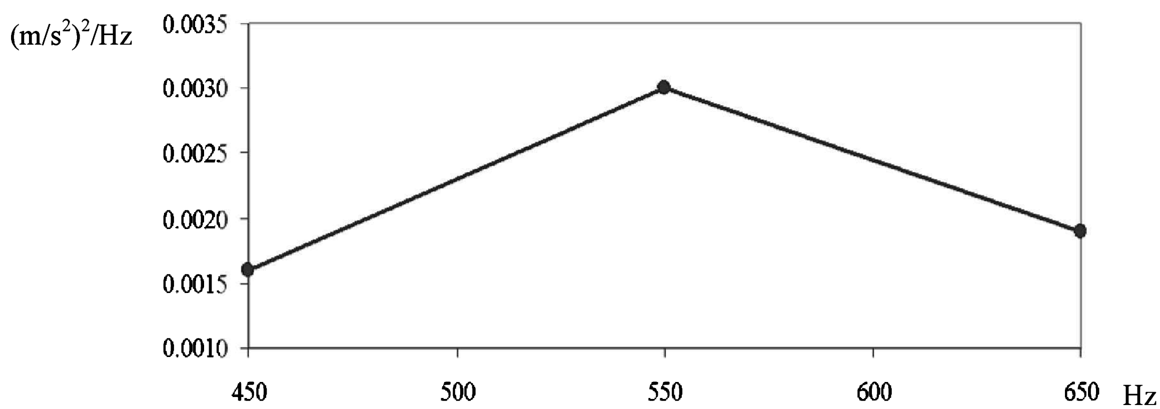


Figure 16-4: Constructing the Acceleration Power Spectral Density function (PSD).

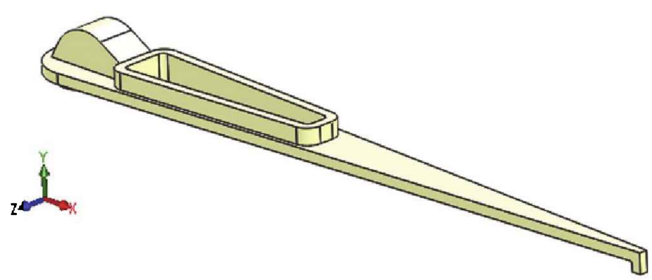
Three points of the Acceleration PSD curve have been calculated by dividing G_{RMS}^2 in each section (each frequency range) by the width of the section.

The Acceleration Power Spectral Density (PSD) allows for a compression of data and is commonly used to characterize a random process. In particular, mechanical vibrations are commonly described by the Acceleration Power Spectral Density, which is easily generated by testing equipment. Design specifications and test results of devices subjected to random vibration are typically given in the form Acceleration PSD.

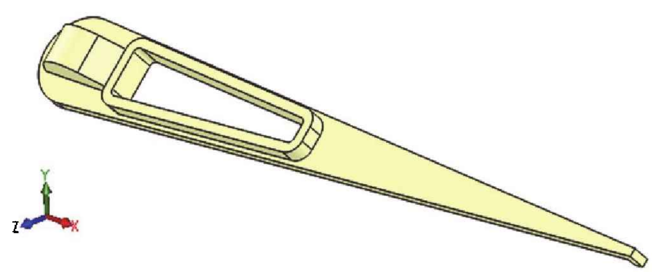
Analysis of random vibration of a hard drive head

Having completed this short introduction of random vibration, we now begin a random vibration analysis of a hard drive head. Open the assembly titled HD HEAD and examine the two configurations: *01aligned* and *02 misaligned.* The assembly contains only one part: HD HEAD.

What is the reason for this complication and why can't we analyze the part instead of the assembly? In a **Random** study, the base excitation may only be applied along only one of the global X, Y or Z directions, which are aligned with the global coordinate system; no other directions are allowed. To apply a base excitation in any other direction, we must place the analyzed part in the desired position respective to the global coordinate system. The assembly enables positioning of the part with respect to the global coordinate system. In configuration *01 aligned* the HD HEAD is aligned with the global coordinate system. In configurations *02 misaligned*, it is rotated 45° about the X axis.



Configuration *01 aligned*

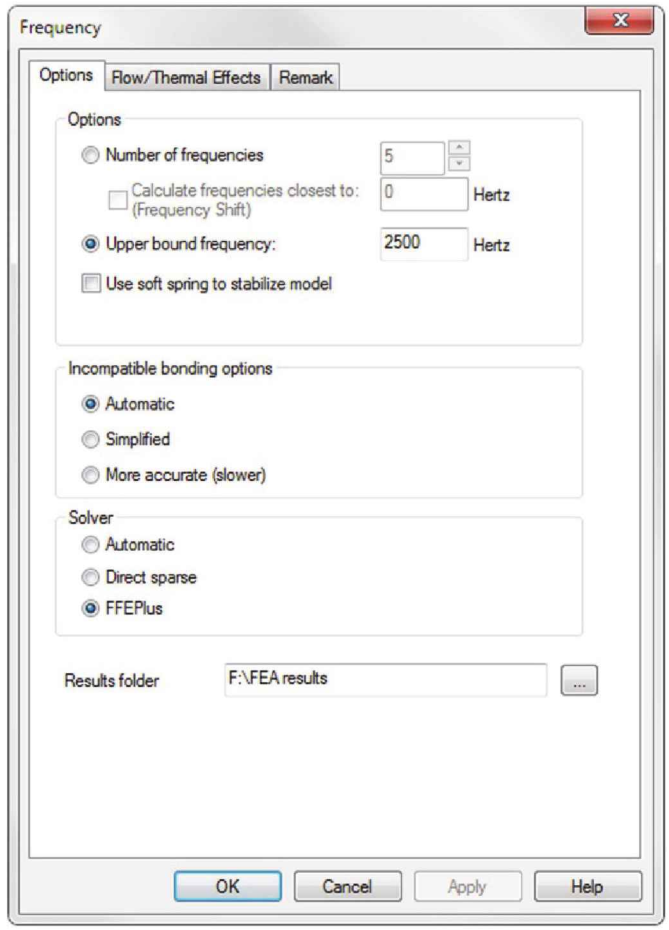


Configuration *02 misaligned*

<u>Figure 16-5: Two assembly configurations are different only in the part position.</u>

In configuration 01 aligned, all part reference planes are aligned with the corresponding assembly reference planes. In configuration 02 misaligned, the part is rotated 45° about the global X axis.

We'll start in configuration *01 aligned*. Create a **Frequency** study with properties shown in Figure 16-6. Call the study *Modal*.



Upper bound frequency: 2500Hz

Figure 16-6: Properties of the Modal study.

All frequencies in the range of 0-2500Hz will be calculated.

Apply restraints as shown in Figure 16-7.

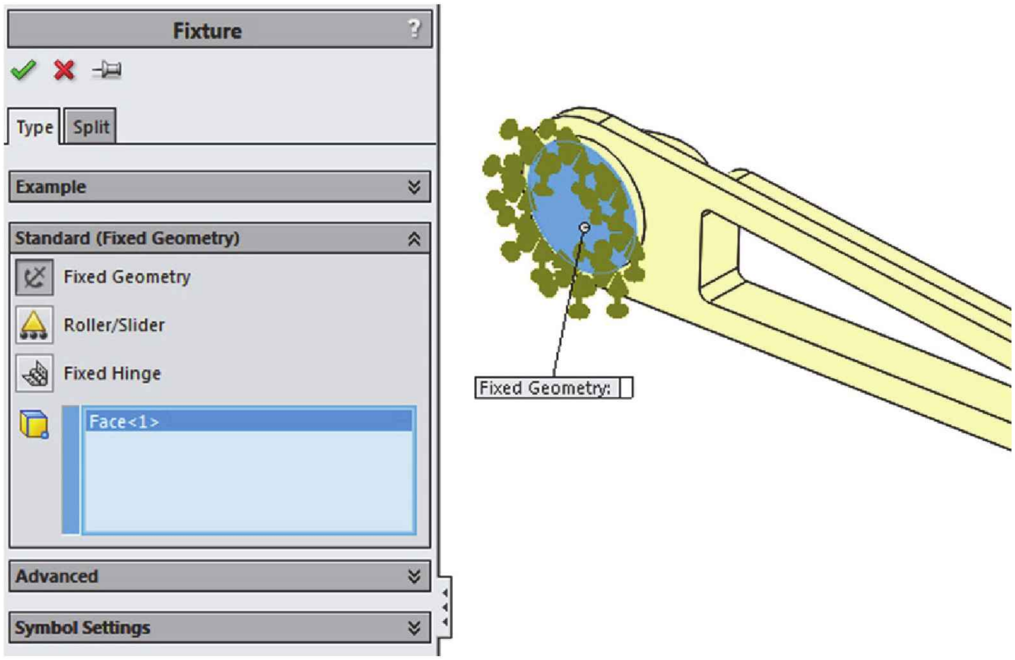


Figure 16-7: Restraints applied to the hard drive head model.

Apply a **Mesh Control** as shown in Figure 16-8 and mesh the model with the default element size.

Element size on the controlled faces: 0.2mm

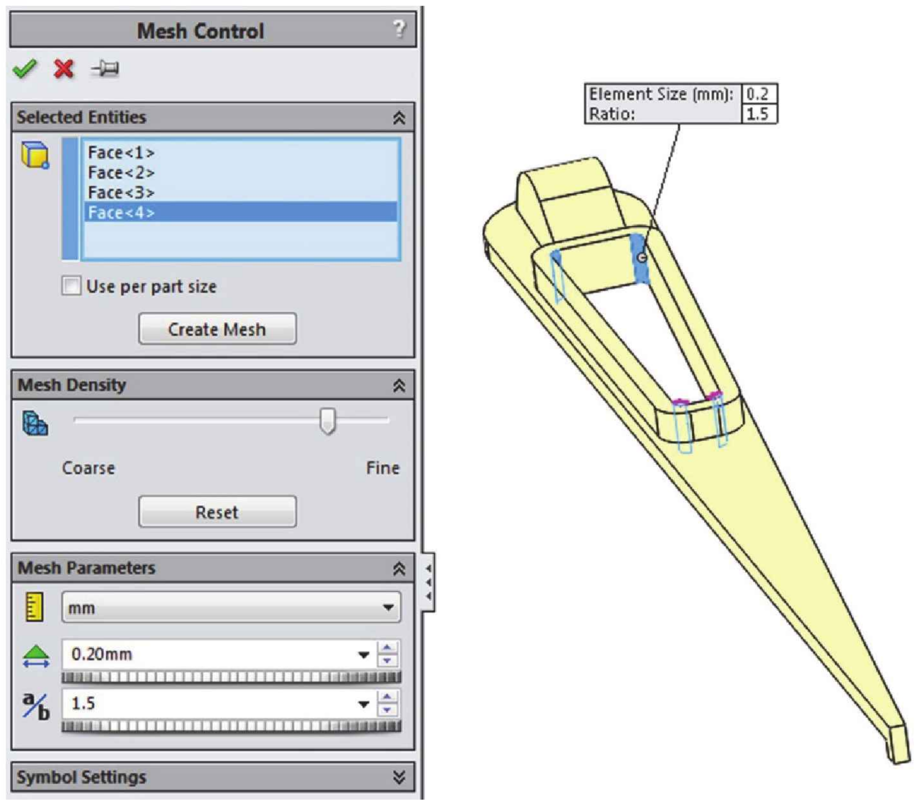
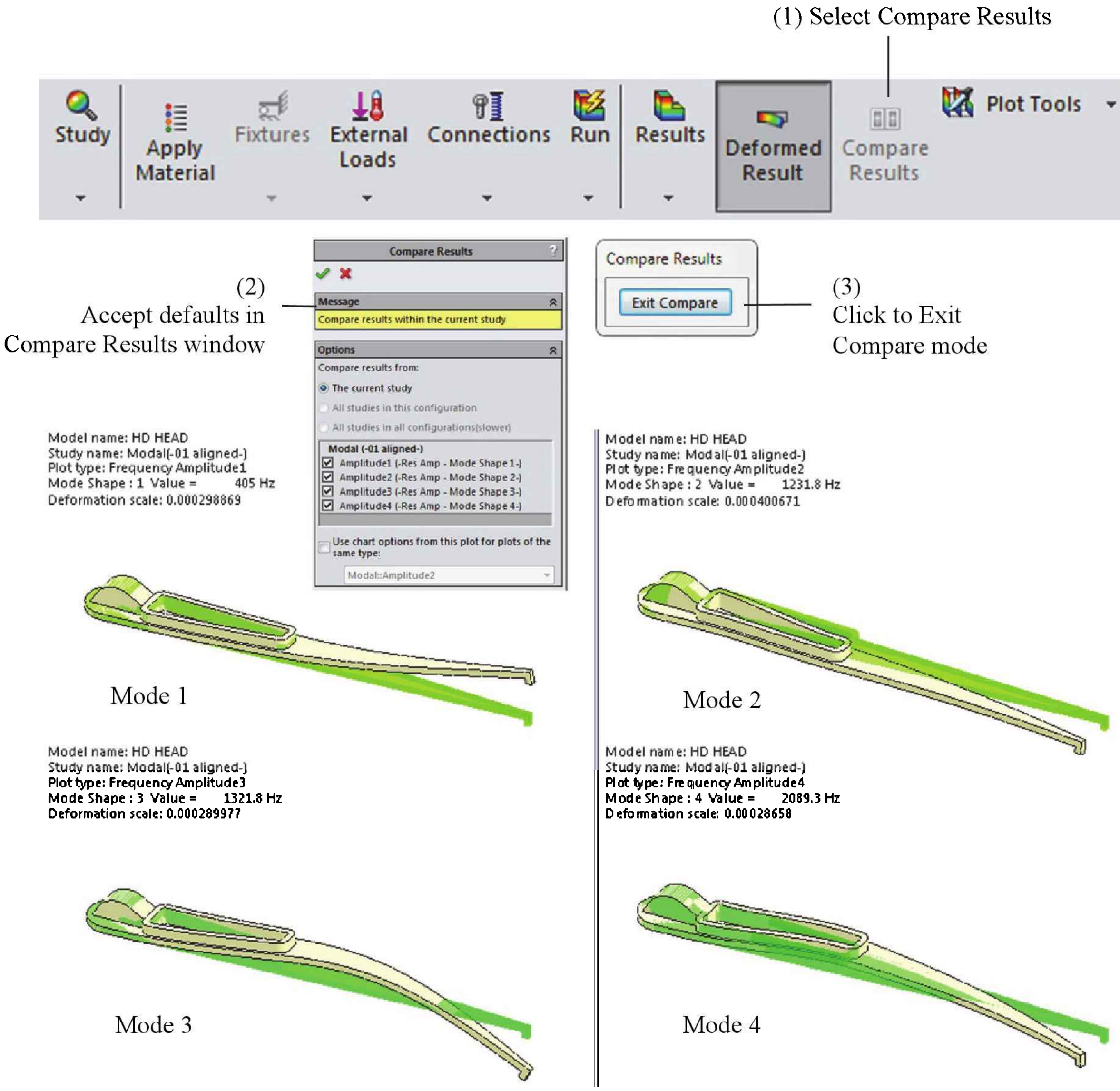


Figure 16-8: Mesh controls are applied to four round fillets.

Use the default global element size on the remainder of the model.

If displacement results were our only objective, the default mesh would be acceptable for both **Frequency** and **Random** studies. However, since in the next study we intend to analyze displacements and stresses, mesh controls are required to ensure correct element shape and size in the area of stress concentrations. Prior to this exercise, analyses were run to find where mesh controls should be specified.

Solve the *Modal* study and review the results shown in Figure 16-9. Notice that there are four modes of vibration within the specified range of frequencies.



<u>Figure 16-9: Modes of vibration within the range of 0 – 2500Hz.</u>

Vibration in mode 1 and 3 take place in the XY plane, vibrations in mode 2 and 4 take place in the XZ plane. The undeformed model is overlaid on the modal shape plots. All four plots can be easily shown together using the Compare Results function in the Command Manager. To use Compare Results follow step shown above.

Proceeding to the analysis of the **Random** study, we could create a new **Dynamic** study independent from the completed **Frequency** study, this time with the **Random** option selected. However, a **Frequency** analysis would then have to be repeated within a **Dynamic Random** study. To avoid this repetition, we can copy the results of the **Frequency** study into a **Dynamic Random** study as shown in Figure 16-10.

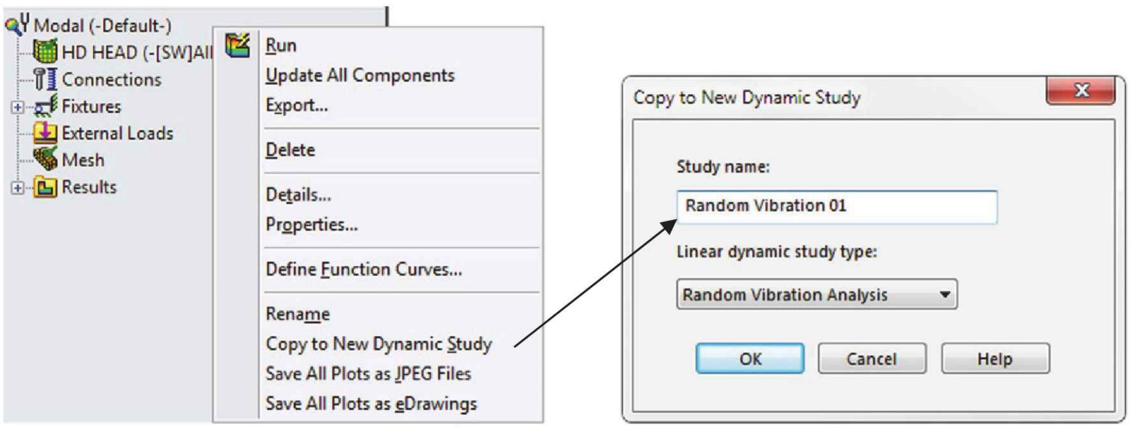
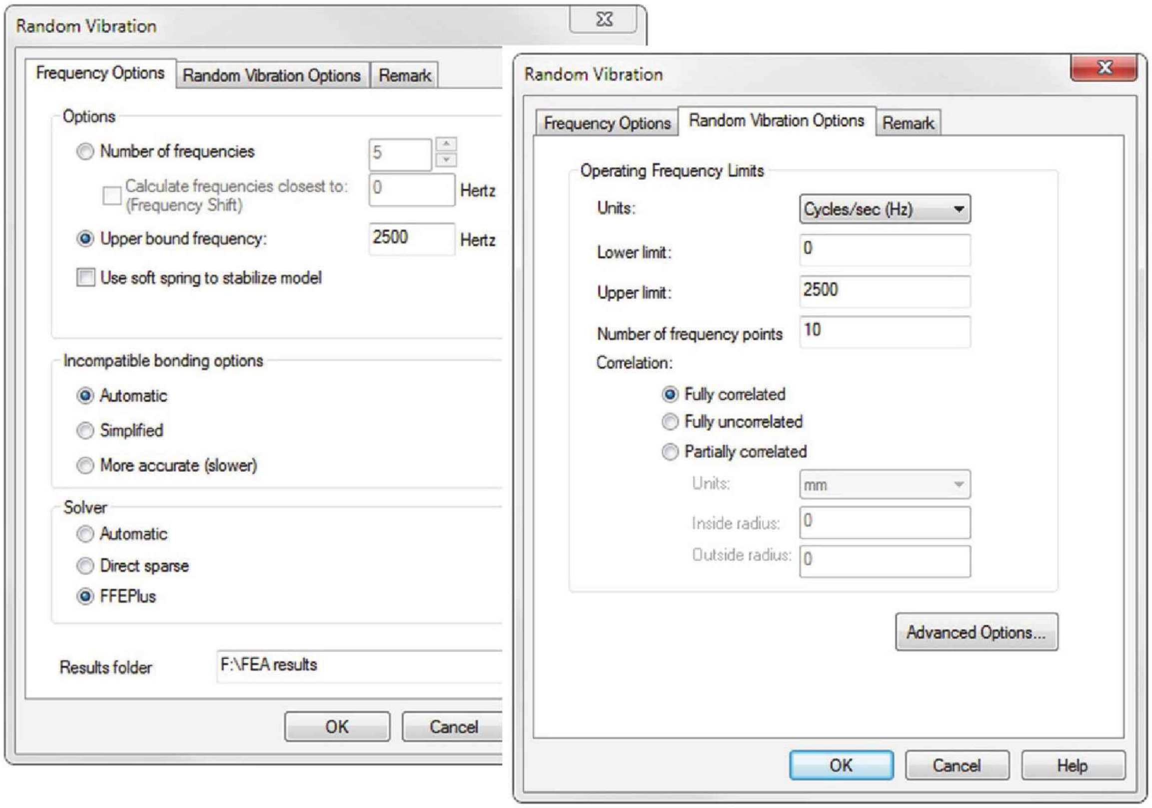


Figure 16-10: Results of a Frequency study can be copied to a new Dynamic study.

Right-click the Modal study folder and select Copy to New Dynamic Study. Select Random Vibration as the type of dynamic study. Name the study Random Vibration 01.

Copying the **Frequency** study into a **Dynamic Random** study also copies the mesh and restraints information.

The required properties of the **Random Vibration** study are shown in Figure 16-11.



Frequency Options　　　　Random Vibration Options

Figure 16-11: Properties of the Random Vibration study.

The Frequency Options specify all modes in the range of 0-2500Hz to be considered in the analysis of random vibrations. The Frequency Options have been copied from Modal study; you don't have to define them.

In the Random Vibration options, specify the Upper limit as 2500Hz to investigate responses to Random Vibration in the frequency range from 0 to 2500Hz.

Define **Global Damping** as shown in Figure 16-12.

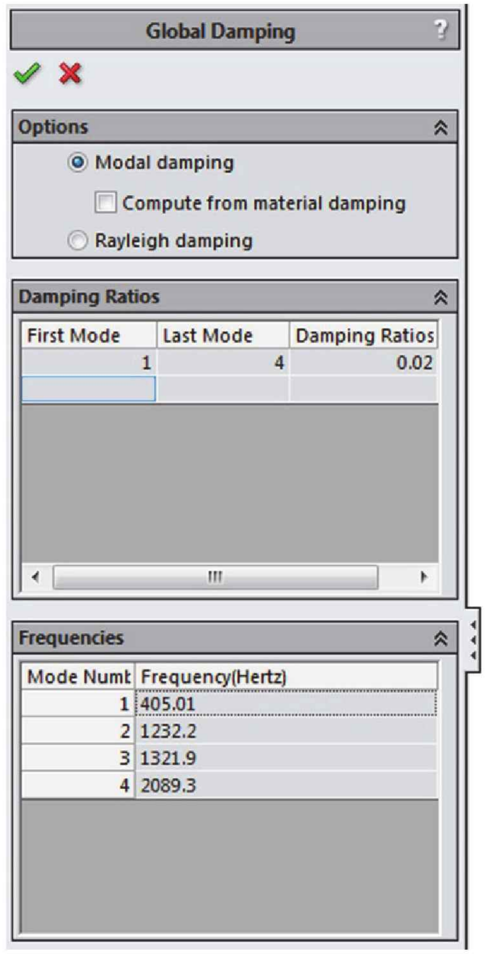
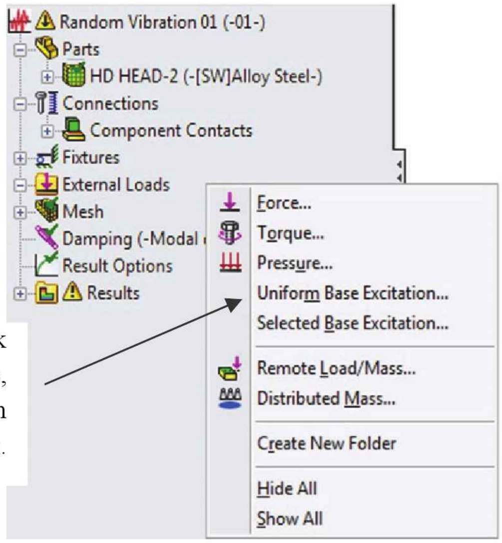


Figure 16-12: Global damping definition.

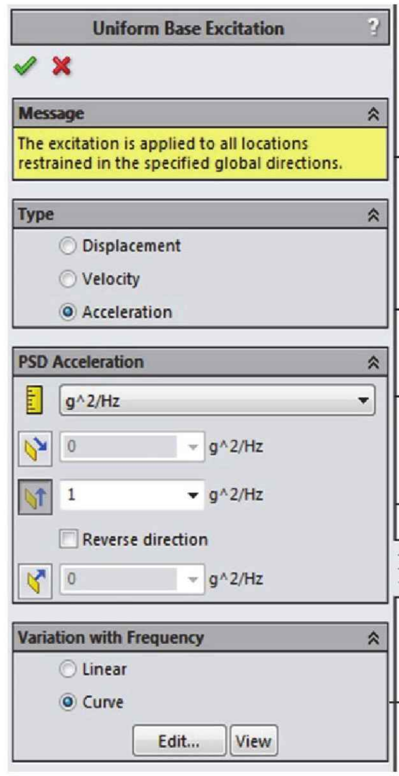
Global damping is defined as 2% of critical damping. The number of modes is 4 and corresponds to the number of modes in the frequency range as defined in Figure 16-11.

Assigning the same damping ratio to all modes represents a simplified and conservative approach. In most cases damping for higher modes will be higher than for lower modes.

The load on the hard drive head comes from random excitation of the base in the global Y direction. Follow Figure 16-13 to define the Acceleration PSD. The Acceleration PSD has been obtained from testing.



(1) Right-click External Loads, select Uniform Base Excitation.



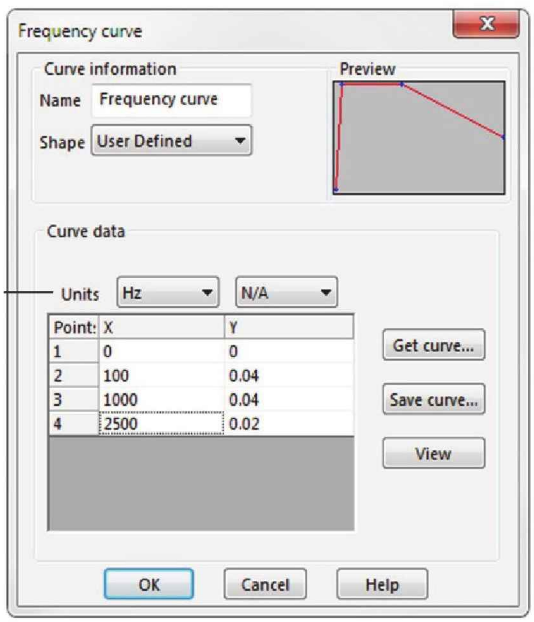
(2)
Read this message.
Make selections indicated below

Acceleration PSD

Unit G^2/Hz

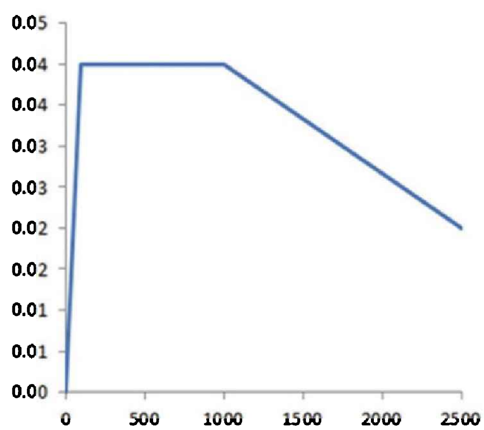
Along global Y

Select curve
Click Edit



(3)
Select Hz.

Enter the coordinates of the four points defining the PSD curve.



(4) Click View in the Uniform Base Excitation window to review the PSD curve. This graphs has been formatted in Excel

<u>Figure 16-13: Uniform Base Excitation defined as the Acceleration PSD in the global Y direction acting on all restraints present in the model. In our model only one restraint is present.</u>

Follow the above steps to create the Acceleration PSD curve.

Define two **Sensors** as shown in Figure 16-14.

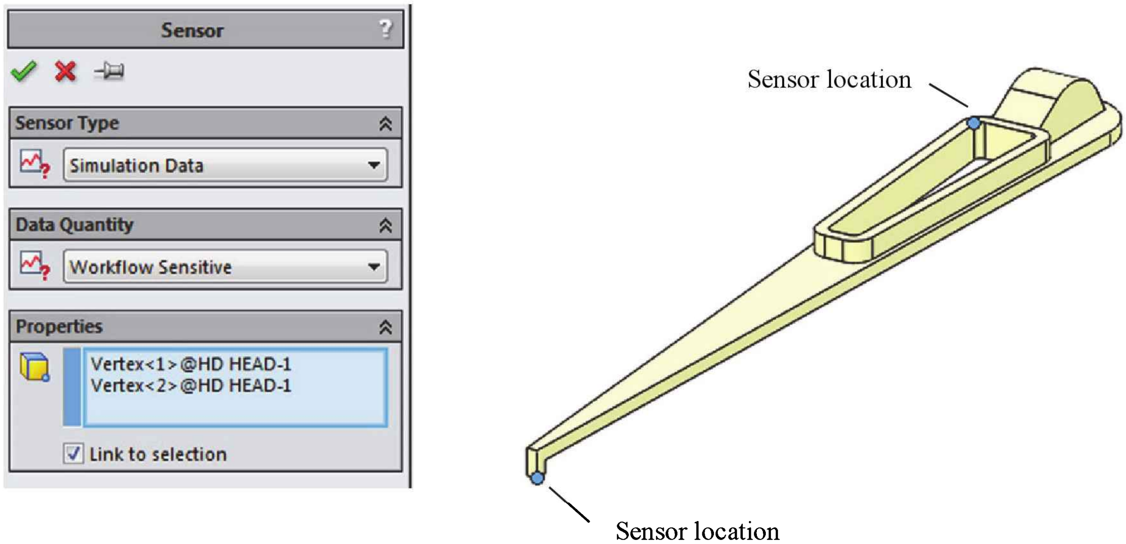


Figure 16-14: Sensor locations

Two vertices are selected: at the tip of the head and at the end of the fillet.

The above is for information only; the HD HEAD model comes with sensors already defined.

Define **Result Options** as shown in Figure 16-15.

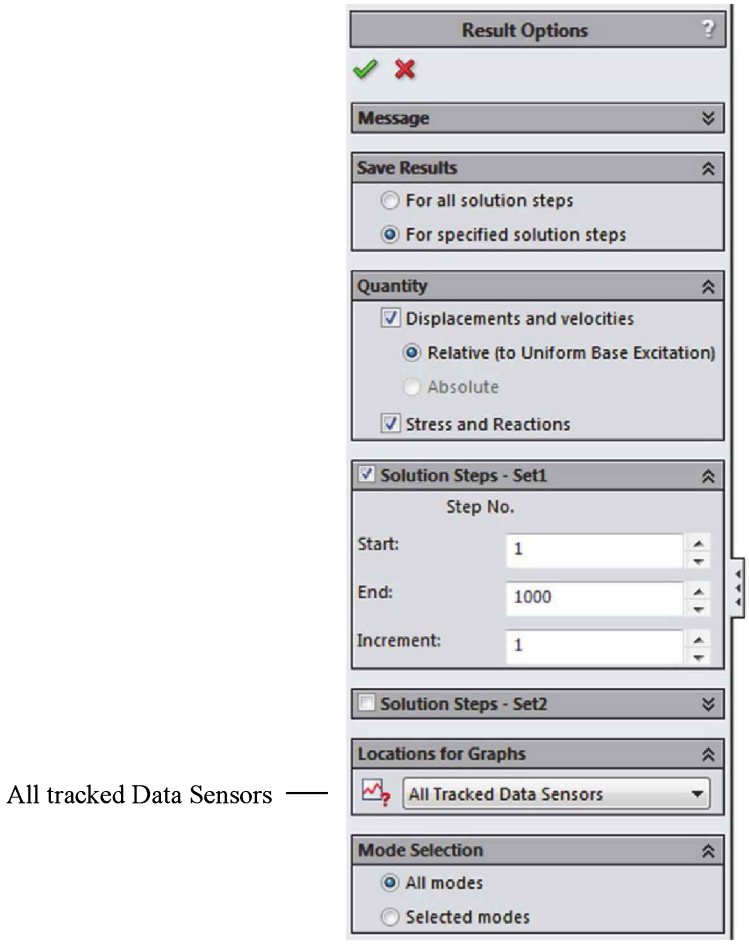


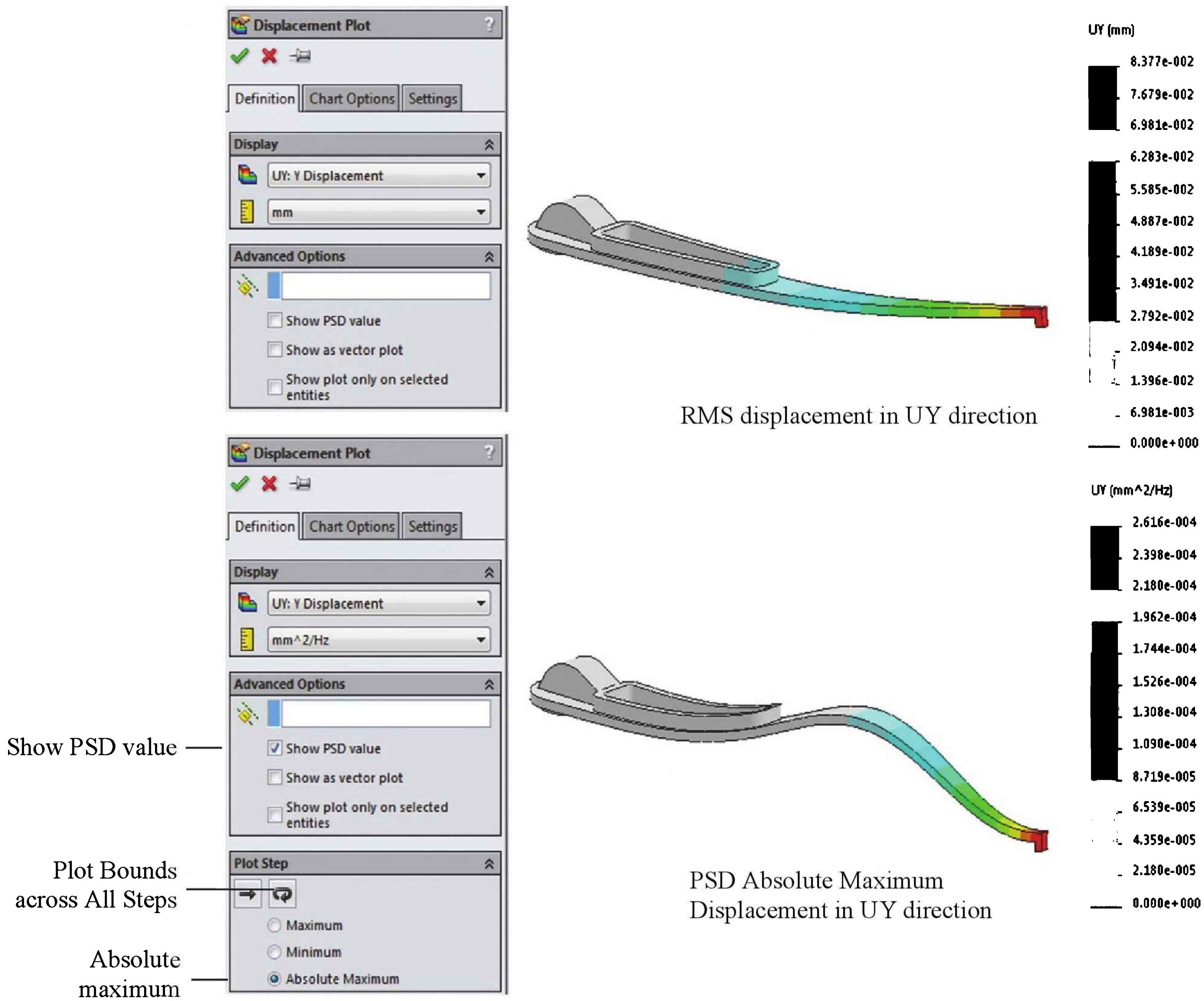
Figure 16-15: Result Options

All tracked Data Sensors includes the two locations selected in the Sensor definition window in Figure 16-14.

Obtain the solution and analyze the RMS displacement results and the PSD displacement results.

In order to analyze the displacement results, new plots need to be created. Plots copied from the *Modal* study are not valid since modal analysis does not give displacement results.

Make a UY displacement plot of RMS and PSD displacements as shown in Figure 16-16.



<u>Figure 16-16: RMS displacement results and PSD Absolute Maximum UY displacement component.</u>

The maximum RMS displacement is 0.084mm.The maximum PSD displacement is 0.00026mm^2/Hz.

PSD displacements are displayed in units of mm^2/Hz for the **Absolute Maximum** found in the frequency range. The **Absolute Maximum** of PSD displacement corresponds to the first mode frequency 405Hz. See Figure 16-17 for explanations of the PSD results.

It is important to understand the meaning of results in a **Random Vibration** analysis. The displacement results in Figure 16-14 (top), are the RMS displacements. The maximum RMS displacement is 0.084mm meaning that the magnitude of displacement has a 68% probability of remaining under 0.084mm. The probability of the maximum displacement magnitude exceeding 0.084mm is 100% - 68% = 32%.

Remembering that the probability of a given displacement is defined by a normal distribution for which $\sigma = 0.084$mm, we can calculate the probability of displacement magnitude exceeding any defined value (Figure 19-17).

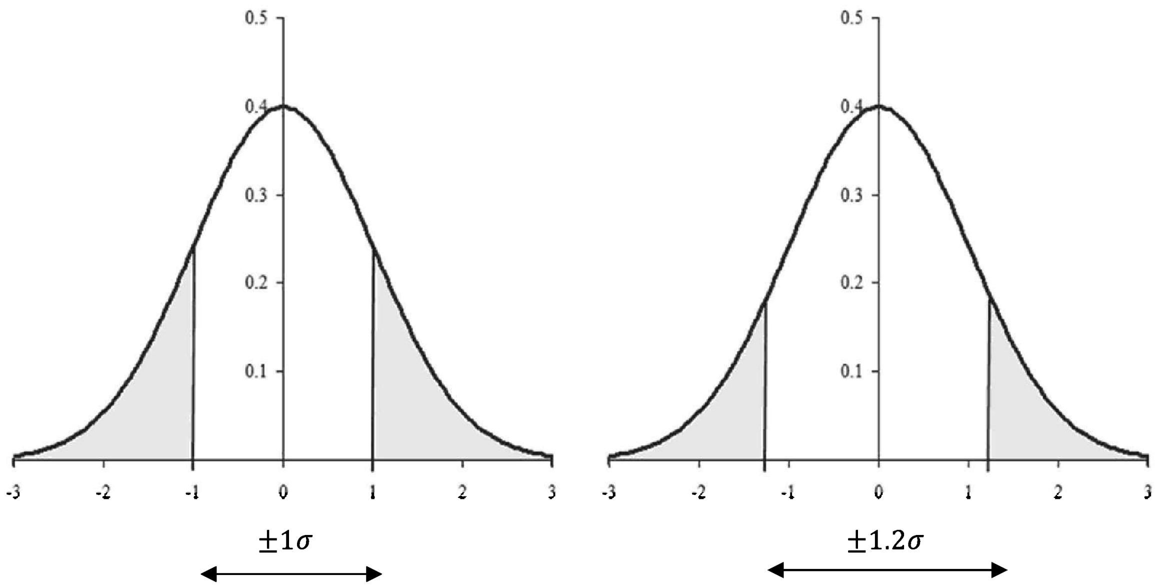


Figure 16-17: The total area under the normalized Gauss curve is 1. The probability of displacement magnitude exceeding $\pm\sigma$ (left) and $\pm 1.2\sigma$ (right) is equal to the corresponding shaded areas.

*The probability of the displacement magnitude exceeding 1*RMS displacement (here 0.084mm), is given by the area outside* $\pm 1\sigma$ *which is 32% (left).*

*The probability of displacement magnitude exceeding 1.2*RMS displacement (0.10mm) is given by the area outside* $\pm 1.2\sigma$ *which is 23% (right).*

The same applies to all results of a Random Vibration analysis. The RMS von Mises stress result is shown in Figure 16-18.

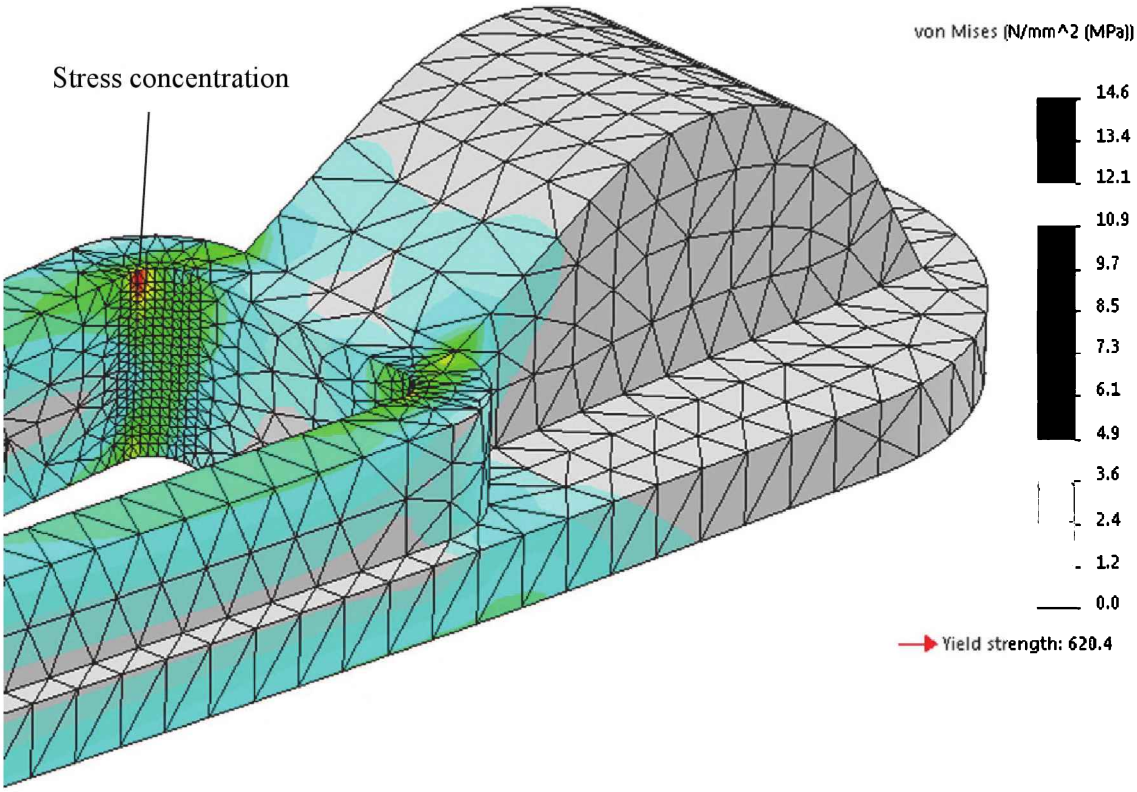


Figure 16-18: RMS P1 stress results

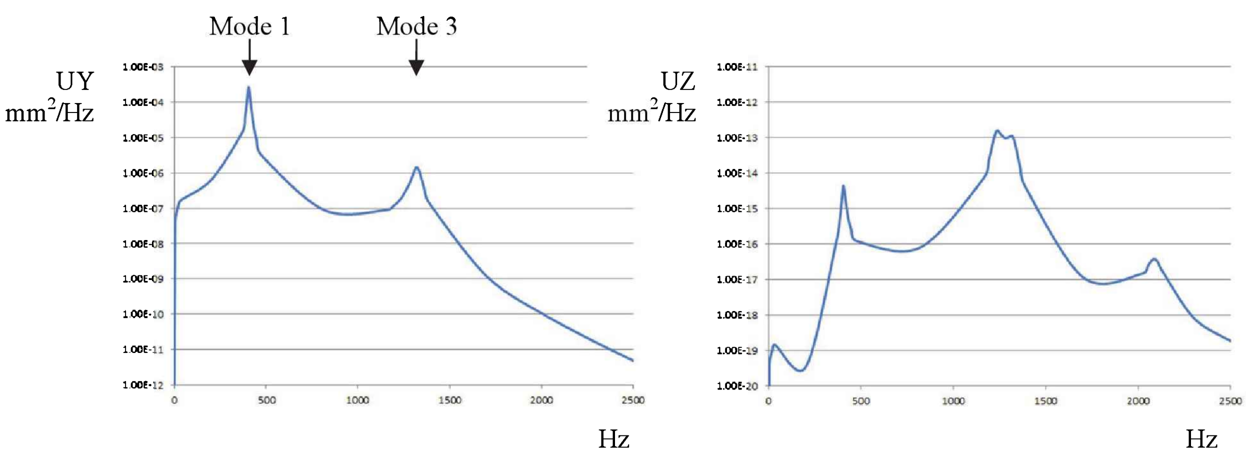
The maximum von Mises stress has a 68% probability of remaining below 14.6MPa.

Notice that stress singularities present in the model geometry (not in the above detail) do not show as stress concentrations because of a large element size used for meshing.

Refer to Figure 16-18 and compare the size of elements to the size of the stress concentration. Even with mesh controls applied, the mesh is at best marginal to model stress concentrations. Repeat the analysis with a more aggressive mesh control.

The results of the **Dynamic Random** analysis presented as RMS values provide one result for the entire frequency range of excitation. Results of Random Vibration analysis such as displacements or stresses may also be presented as PSD values, which are different for each excitation frequency. Examine the PSD options of displacements and stress result plots and notice that displacement results are given in mm^2/Hz, and stress results are given in MPa^2/Hz. These units are a consequence of the base excitation being defined as acceleration squared per frequency range, in our case $G_{RMS}{}^2/Hz$.

The most informative way to review PSD results is to graph them over the frequency range. Create two graphs showing PSD displacement of the tip of the head defined by the sensor shown in Figure 16-14. These two graphs are shown in Figure 16-19.

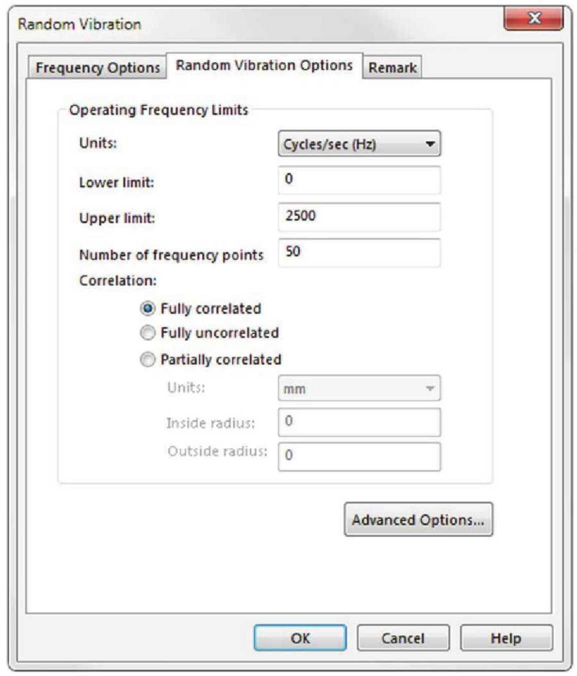


<u>Figure 16-19: PSD UY and UZ displacement components of the tip as a function of excitation frequency in configuration *01 aligned.*</u>

Notice a very different scale of Y axis in UY and UZ response graphs. These graphs have been formatted in Excel.

Modal shapes of mode 1 and mode 3 are aligned with the direction of base excitation. Indeed, upon examination of the UY response graph in Figure 16-19 we see that mode 1 and mode 3 are excited, even though the effect of mode 3 is visible only because a logarithmic scale is used. The UZ response graph does not show any modes excited. The peaks visible on the UZ graph are results of discretization error.

The area under the PSD UY response graph equals the RMS^2 displacement of the vertex where the sensor has been defined. This can be proven by integrating the PSD UY displacement function. If you wish to perform the numerical integration, then for better accuracy run the study *Random Vibration 01* again with settings shown in Figure 16-20.



50 frequency points

Figure 16-20: Modified settings of *Random Vibration 01* study.

Specify 50 frequency points.

Four frequencies are found in the 0-2500Hz range. Therefore with 50 frequency points, the solution will proceed in 251 frequency steps.

Figure 16-21 shows the UY PSD response graph based on 251 frequency steps; it uses a linear scale on the Y axis. See spreadsheet HD HEAD.xlsx for numerical integration results.

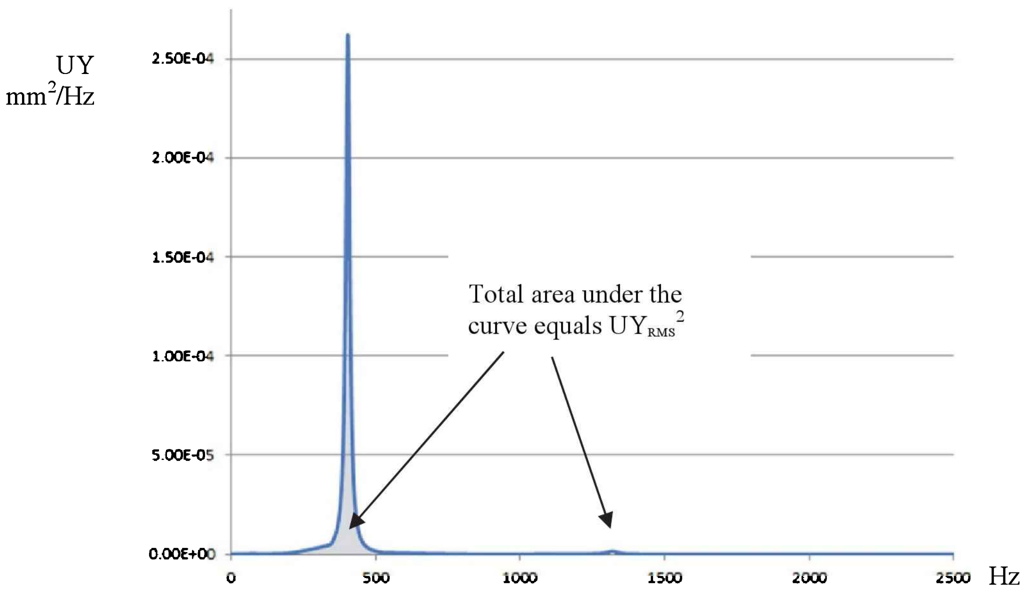


Figure 16-21: Correspondence between RMS and PSD UY displacement.

See the spreadsheet HD HEAD.xlsx for details.

Now, change to configuration *02 not aligned*, create a **Random** study titled *Random Vibration 02* and repeat the analysis. Everything except the model position will be identical to study *Random Vibration 01*. Obtain the solution and construct PSD response graphs for UY and UZ displacement components of the tip. These graphs are shown in Figure 16-22 and are directly comparable to the graphs presented in Figure 16-19.

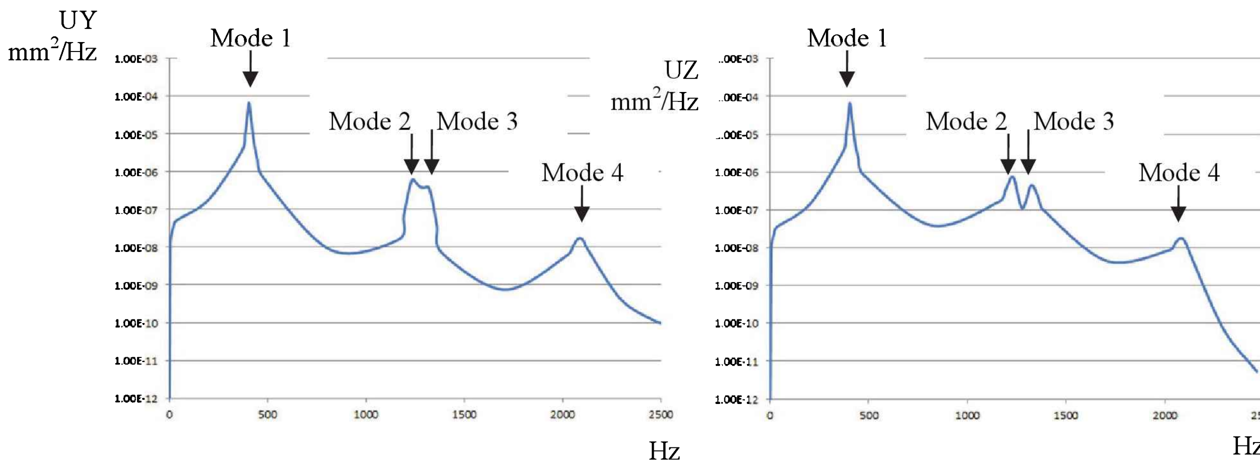


Figure 16-22: PSD UY and UZ displacement components of the tip as a function of excitation frequency in configuration *02 misaligned.*

Four peaks in each graph correspond to the four modes exited by the base excitation.

Upon examination of the graphs in Figure 16-22, we find that in configuration *02 misaligned*, all four modes are excited; this is easiest to show using a logarithmic scale. Mode 1 still dominates the vibration response.

17: Response Spectrum analysis

Topics covered

- Non stationary random base excitation
- Seismic response analysis
- Seismic records
- Response spectrum method
- Generating response spectra
- Methods of modal combinations

Non stationary excitation

Many random events are not stationary meaning that parameters such as mean or variance do not remain the same but change with time. Important examples of a non-stationary process are earthquake and pyrotechnic shock. Figure 17-1 shows an acceleration time history of an earthquake recorded in California zone 4. The event duration is 31s; considering a sampling rate of 0.005s the curve consists of 6145 data points.

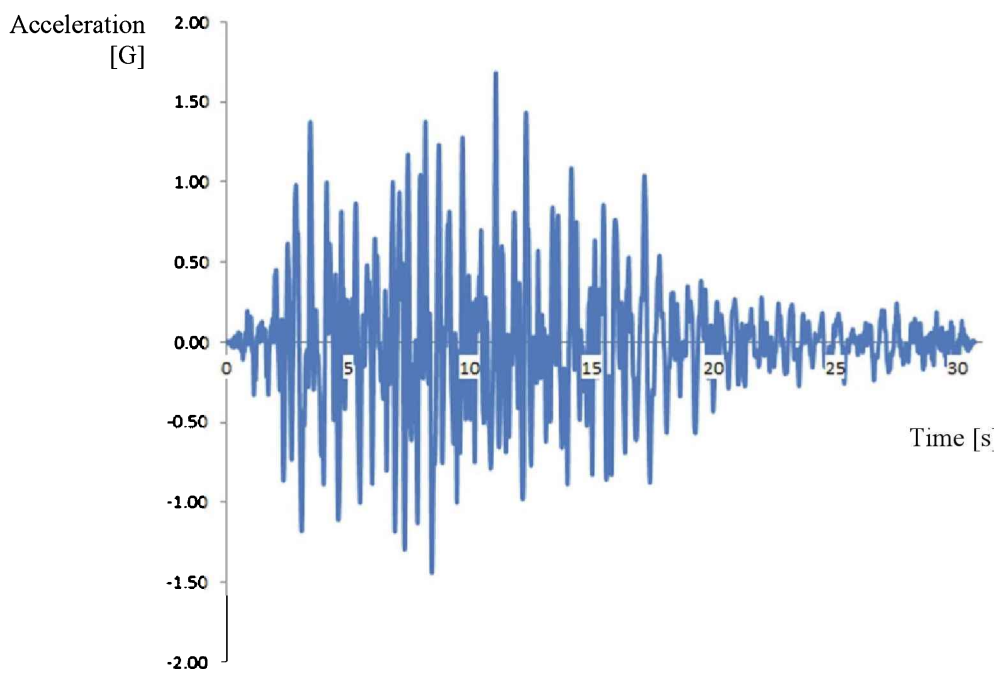


Figure 17-1: An example of acceleration time history of an earthquake.

Source: vibrationdata.com.

The earthquake is a non-stationary process and the methods of Random Vibration analysis cannot be used. At the same time, given the amount of data points, a **Time Response** analysis is impractical. For these reasons, a special analysis method called the **Response Spectrum Method** has been developed to analyze long duration non-stationary processes like the earthquakes or pyrotechnic shock events.

We will explain the concept of the **Response Spectrum Method** in the following steps:

1. A system vibrating in resonance can be described as a single degree of freedom harmonic oscillator system characterized by mass, stiffness and damping.

2. A response of a system with more than one resonant frequency can be represented as a combination of responses of harmonic oscillators, with each harmonic oscillator corresponding to a particular resonant frequency. Notice that this is the basis of the modal superposition method.

3. If the excitation frequency is equal to one of structure's resonance frequencies, then the system response to that excitation is controlled only by system damping. Mass does not matter, stiffness does not matter; they have no impact on the system's response. The **only** thing controlling the system response is its damping!

Imagine that two vastly different systems, a harmonic oscillator and a bridge, have two important things in common: the resonant frequency and damping. Now imagine that both the harmonic oscillator and the bridge are excited by the same excitation that happens to have the same frequency as the resonance frequency of both the harmonic oscillator and the bridge. The vibration response (e.g. the maximum displacement) will be the same for the harmonic oscillator and the bridge!

4. Let's assume that the vibration response can be adequately modeled with the modal superposition method based, for example, on five modes. Using the observations made in point **3.**, we may simplify the analysis very significantly. Rather than studying the response of the actual structure, we can study the response of five harmonic oscillators with natural frequencies corresponding to those five modes and associated damping being the same as the modes and damping of the structure we wish to analyze. In particular, if we study the structure's response to seismic excitation, then rather than testing the actual structure, we can just subject those five oscillators to the earthquake acceleration time history and record the maximum displacement, velocity, and acceleration of each oscillator. Next, we plot the maximum displacement, velocity, and acceleration of each oscillator as a function of the oscillator's frequency. This way a **Response Spectrum** curve is built. The maximum displacement recorded as a function of frequency is the **Displacement Response Spectrum**, the maximum velocity recorded as a function of frequency is the **Velocity Response Spectrum** and the maximum acceleration recorded as a function of frequency is the **Acceleration Response Spectrum**. The above reasoning may be extended to any number of harmonic oscillators.

5. The examined structure does not have to have the same resonant frequencies as our set of harmonic oscillators described in point **4**. If the resonant frequencies of the analyzed structure fall "in-between" frequencies of the oscillators used to construct the **Response Spectrum** curve, the structure response can be interpolated. So if resonant frequencies of the analyzed structures are known and we also know the **Acceleration Response Spectrum** curve that has just been constructed by examining the response of harmonic oscillators, then we can find out by interpolation, the maximum acceleration of the structure corresponding to each mode. Double integration will give the maximum displacement.

6. The information of interactions between modes has been lost in the above process. Therefore, it must be re-built using one of the methods of combining the maximum vibration response of each mode. The commonly used methods are: the **Square Root of Sum of Squares** (SRSS) and the **Absolute Sum**. These methods will be described later.

We will expand on point **4**. Figure 17-2 shows a series of Single Degree of Freedom (SDOF) oscillators all attached to a common base. Their natural frequencies are different, changing from the lowest 3Hz to the highest 30Hz. All SDOF oscillators have the same damping ratio. This base is subjected to excitation with the acceleration time history shown in Figure 17-1.

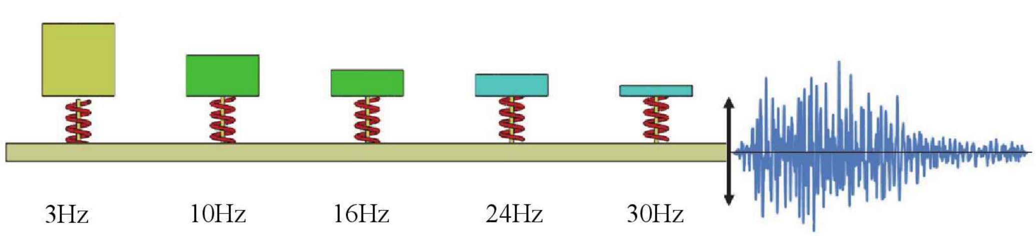


Figure 17-2: An illustrative explanation of the Response Spectrum concept: a set of SDOF oscillators is subjected to base excitation with acceleration time history recorded from an earthquake.

The range of frequencies 3-30Hz is divided into four parts: 3-10Hz, 10-16Hz, 16-24Hz, and 24-30Hz.

A **Time Response** analysis of the set of SDOFs in Figure 17-2 is conducted and the maximum absolute displacement, velocity, and acceleration are found for each oscillator. The **Time Response** analysis is easy because of simplicity of this system. The corresponding **Response Spectrum** curves will have five points. An example of an **Acceleration Response Spectrum** curve is shown in Figure 17-3.

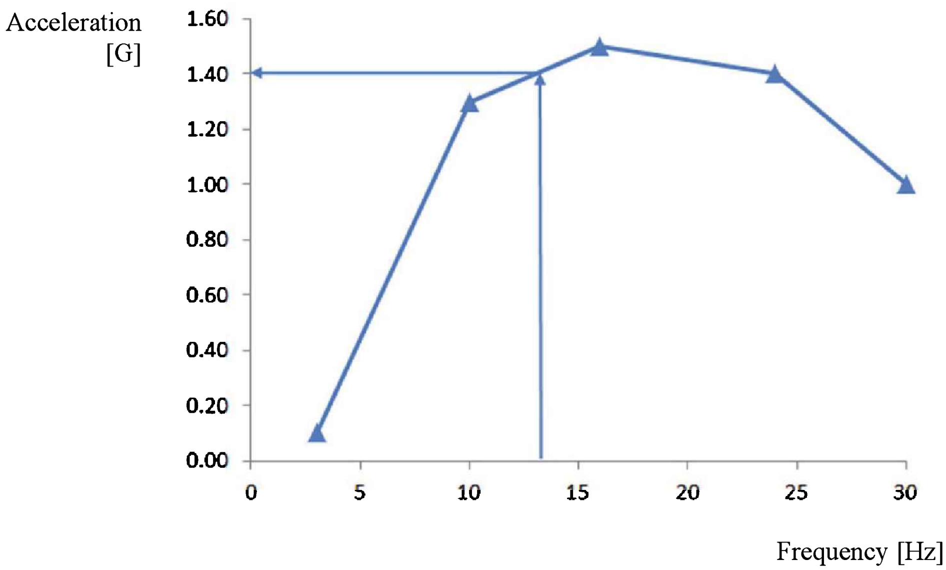


Figure 17-3: An Acceleration Response Spectrum curve generated from the results of a Time Response analysis of a series of SDOFs with frequencies 3Hz, 10Hz, 16Hz, 24Hz, and 30Hz.

The curve serves as an illustrative example only; it does not correspond to any seismic shock.

Response for frequencies falling "in-between" frequencies of the oscillators shown in Figure 17-2 may be interpolated. Figure 17-3 shows a linear interpolation.

Assume for a moment that natural frequencies of a structure are dominated by a single mode and the direction of excitation is aligned with that mode. In this case the response (e.g. maximum acceleration) can be read from a graph. Finite Element Analysis programs merely automate interpolation and combine effects of multiple modes and different excitation directions.

The direction of excitation with regard to the direction of modal shape must be taken into consideration when a **Response Spectrum** curve is constructed. This is illustratively shown in Figure 17-4.

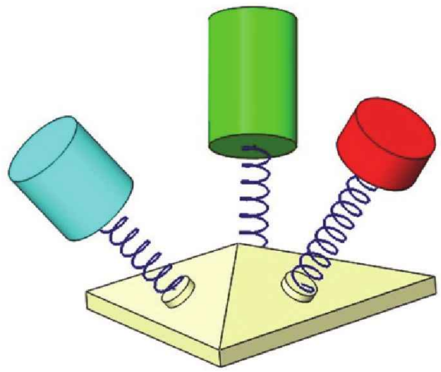


Figure 17-4: Illustrative explanation of the Response Spectrum concept taking into consideration that the direction of excitation and the directions of modal shapes may not be aligned.

Excitation is applied perpendicularly to the base; oscillators are located "at an angle" to the direction of excitation.

Most often the **Response Spectra** curves are produced by subjecting the series of SDOFs to a synthesized acceleration time history that does not correspond to any particular earthquake but represents an earthquake that may happen in a given geographical region. The synthesized **Response Spectra** curves can be found in seismic codes and are used as inputs to seismic analyses. An example of some **Response Spectra** for equipment installed in a building during a seismic event is shown in Figure 17-5.

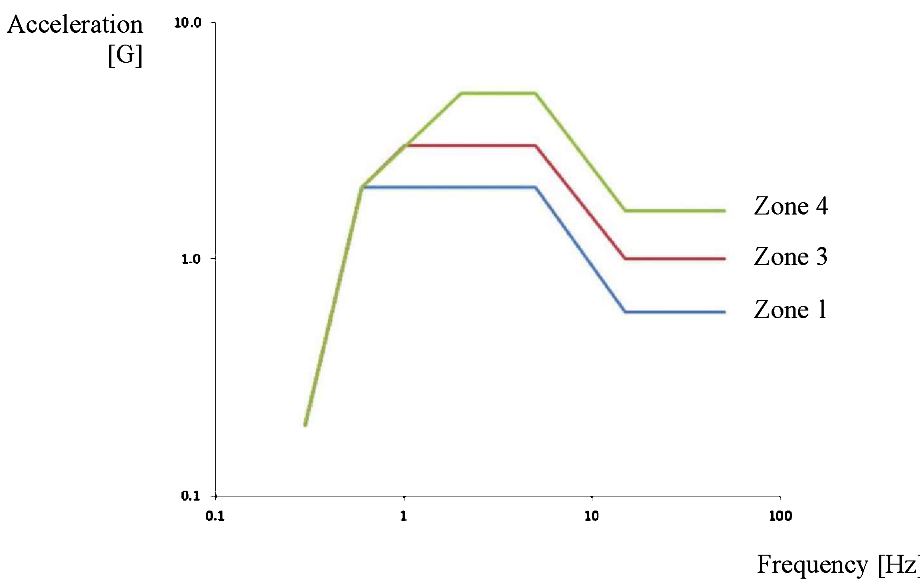


Figure 17-5: Response Spectra for different seismic zones.

For more information see: en.wikipedia.org/wiki/Network_Equipment-Building-System.

The **Response Spectra** shown in Figure 17-5 are not meant to be used as an input to seismic analysis of an entire building. These are accelerations in the horizontal direction of an upper floor of a building during an earthquake.

Even though the **Response Spectrum** method is ubiquitous in earthquake engineering, it has important limitations. Finite Element programs like **SolidWorks Simulation** calculate the response of each mode independently and then must somehow combine them together. The maximum responses in different modes do not have to coincide in time; therefore these responses cannot be combined directly. **SolidWorks Simulation** offers a choice of four approximate methods of combining individual modal responses:

Square Root Sum of Squares (SRSS)

Absolute Sum (ABS)

Complete Quadratic Combination (CQC)

Naval Research Laboratory (NRL)

The **SRSS** estimates the peak response by the square root of the sum of the maximum responses squared. The **ABS** method assumes that the maximum modal responses occur at the same time for all modes. This is the most conservative method of modal combination. The **CQC** is based on random vibration concepts, and **NRL** takes the absolute value of the largest response and adds it to the **SRSS** response of other modes.

How many modes should be considered when modeling a structure's response to a seismic shock? Most often seismic analysis is conducted to demonstrate compliance with certain code requirements. That specific code will specify the minimum mass participation factor in the model used for seismic response. Hence, we need to use as many modes as necessary to ensure the required mass participation.

We'll demonstrate the use of **SolidWorks Simulation** for the analysis of a seismic response using the assembly model titled FRAME, shown in Figure 17-6. Base excitation in the form of a **Response Spectrum** will be similar to the Zone 4 excitation in Figure 17-4. Our objective is to analyze displacements and accelerations experienced by the FRAME installed on an upper floor of a building experiencing **Zone 4** seismic shock. Sharp re-entrant edges in the model geometry are acceptable because stress analysis is not an objective of this exercise.

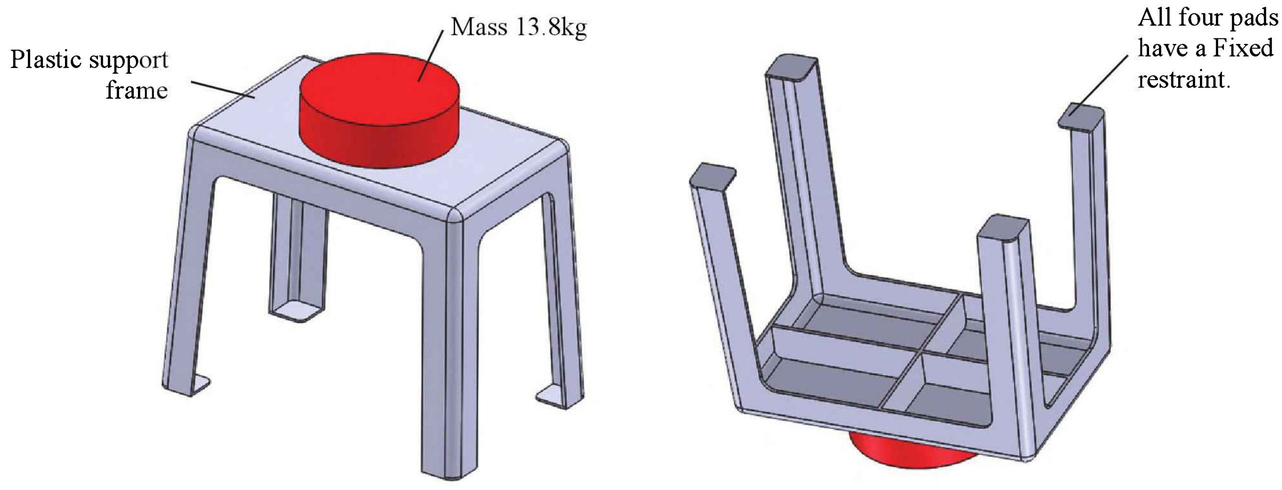


Figure 17-6: The FRAME model used for the analysis of a seismic shock.

Notice the sharp re-entrant edges in the model.

A **Response Spectrum** study does not allow for use of reference geometry in the definition of the direction of excitations. They can be only applied in the global X, Y, or Z directions. Therefore, the assembly model is located "at an angle" and excitation will be applied in the X direction (Figure 17-7).

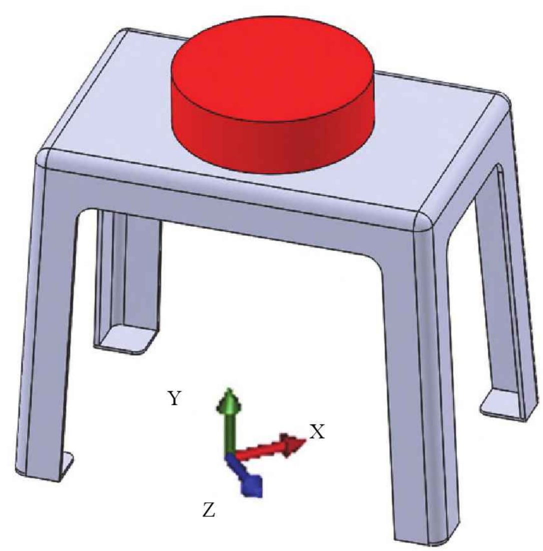
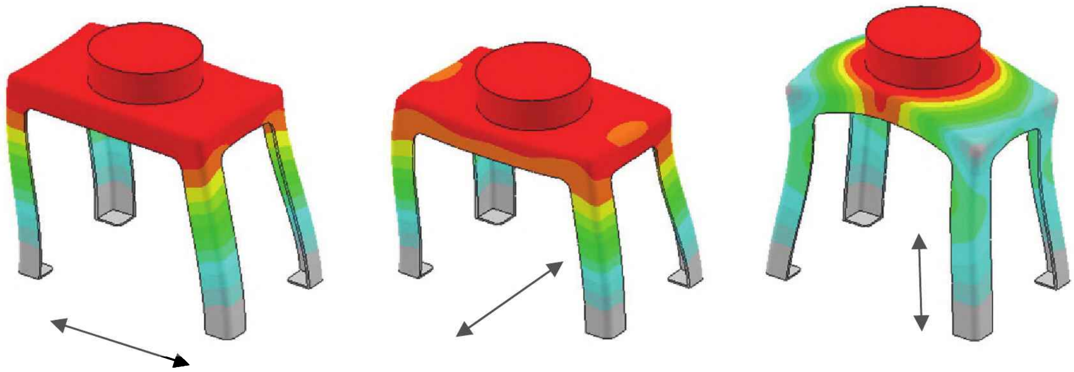


Figure 17-7: Location of the FRAME relative to the global coordinate system of the assembly.

Excitation will be applied in the global X direction.

Prior to the **Response Spectrum** analysis, run a modal analysis to find modes within the range of frequencies 0-50Hz. Mesh the assembly with a **Curvature base mesh** with the default element size and define **Fixed** restrains to the four pads as shown on Figure 17-6. The three modes within the range of 0-50Hz are shown in Figure 17-8.



Mode 1: 19Hz Mode 2: 20Hz Mode 3: 48Hz

Figure 17-8: The three modes of vibration of the FRAME within the range of 0-50Hz.

Arrows indicate the direction of vibration.

The direction of base excitation is along the global X direction, therefore mode 1 and mode 2 will participate in the vibration response. Mode 3 is orthogonal to the direction of excitation and will not be excited by the applied base excitation. The **Response Spectrum** method uses the **Modal Superposition Method** to find the vibration response; therefore, the vibration response will be a superposition of the responses of mode 1 and mode 2.

Create a **Response Spectrum** study as shown in Figure 17-9, call it *Seismic shock*.

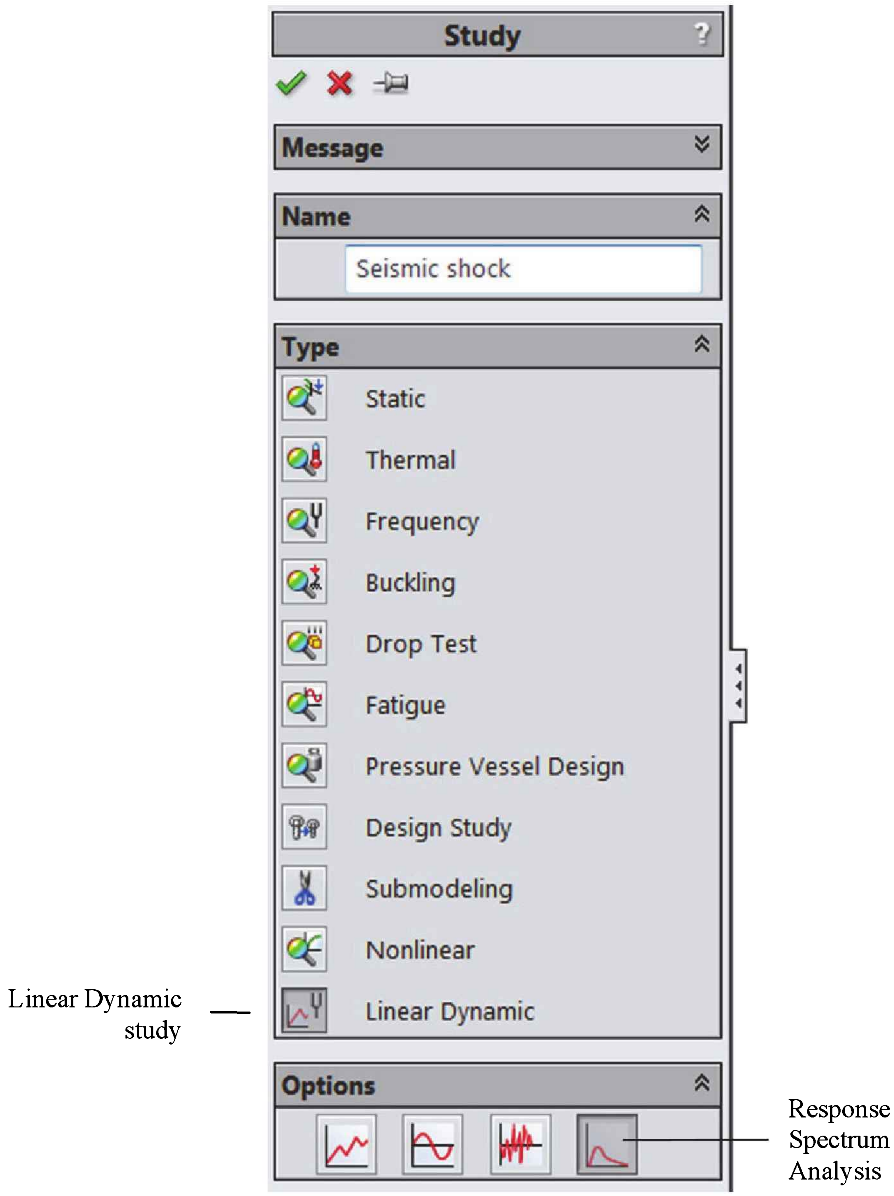


Figure 17-9: Definition of a Linear Dynamic study with the Response Spectrum Analysis option.

Define a **Uniform Base Excitation** as shown in Figure 17-10.

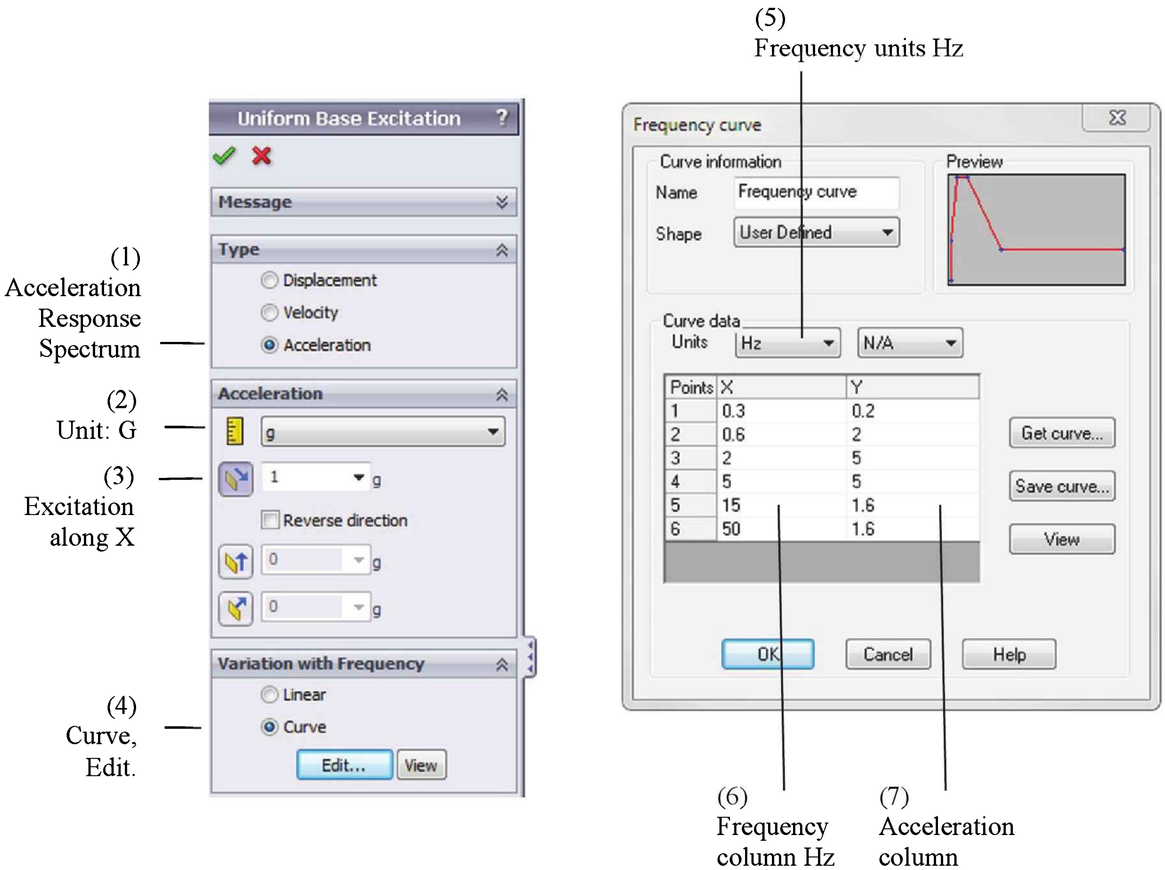


Figure 17-10: Definition of the base excitation.

Numbers in the acceleration column are multipliers to the gravitational acceleration entered in step 3. You may copy the Response Spectrum table from the spreadsheet titled FRAME.XLSX and paste it into the Frequency curve window.

Define restraints as shown in Figure 17-6 and mesh using a **Curvature based mesh** with default settings. The mesh is shown in Figure 45-11.

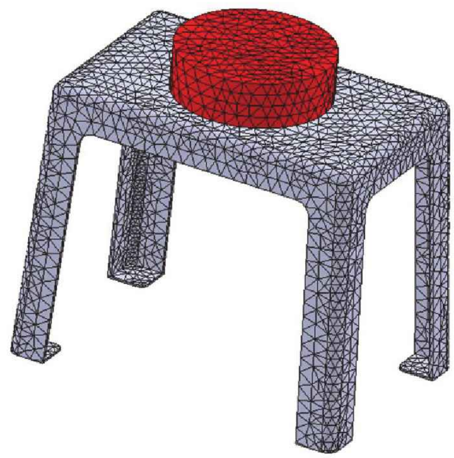


Figure 17-11: The FRAME assembly after meshing.

A curvature based mesh is recommended here because of curvilinear geometry.

Define the study properties as shown in Figure 17-12.

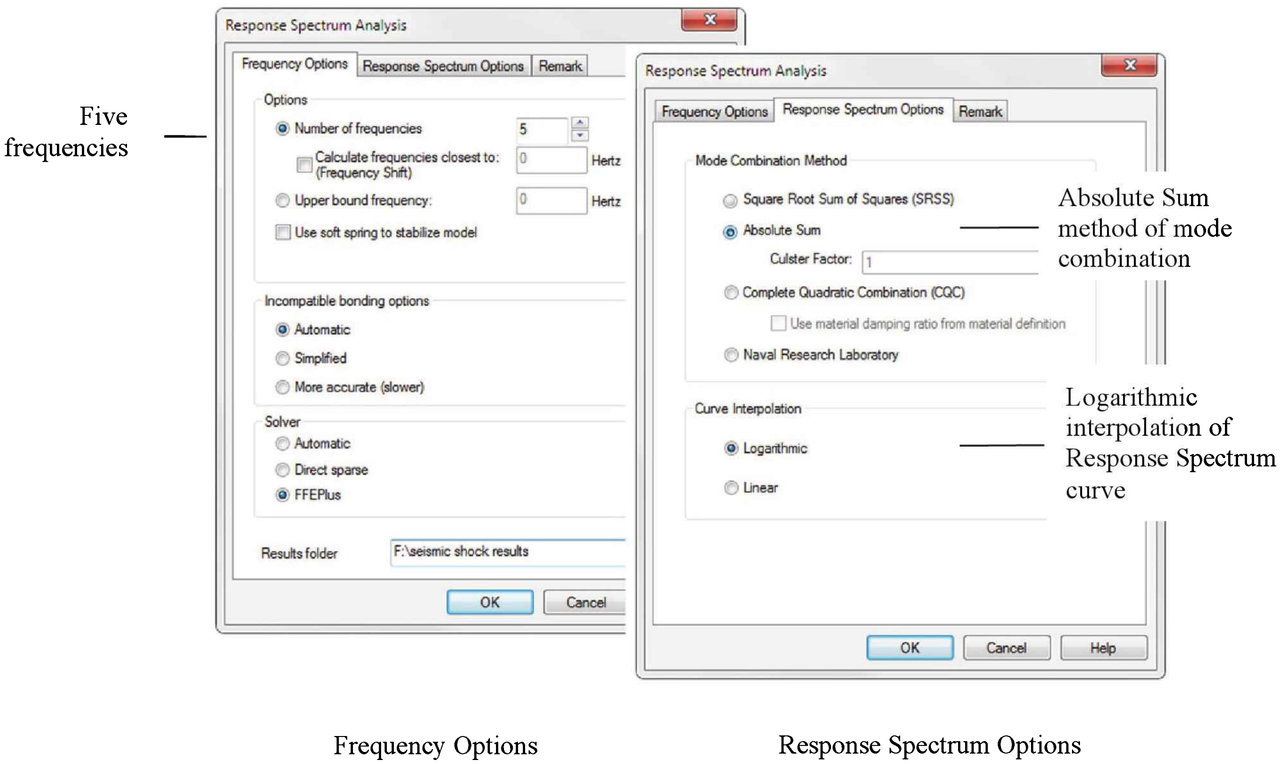


Figure 17-12: Response Spectrum Study properties.

Since five frequencies are specified, the first five modes may possibly contribute to the seismic response.

In preparation for a **Response Spectrum** analysis, we conducted a modal analysis. The results of the modal analysis indicated that only the first two modes will participate in the seismic response. Yet we specify five modes in **Frequency Options**. This is to prove that indeed only the first two modes are important in the seismic response; it will be clearly seen from the modal mass participation results.

Obtain the solution and create a plot of acceleration as shown in Figure 17-13.

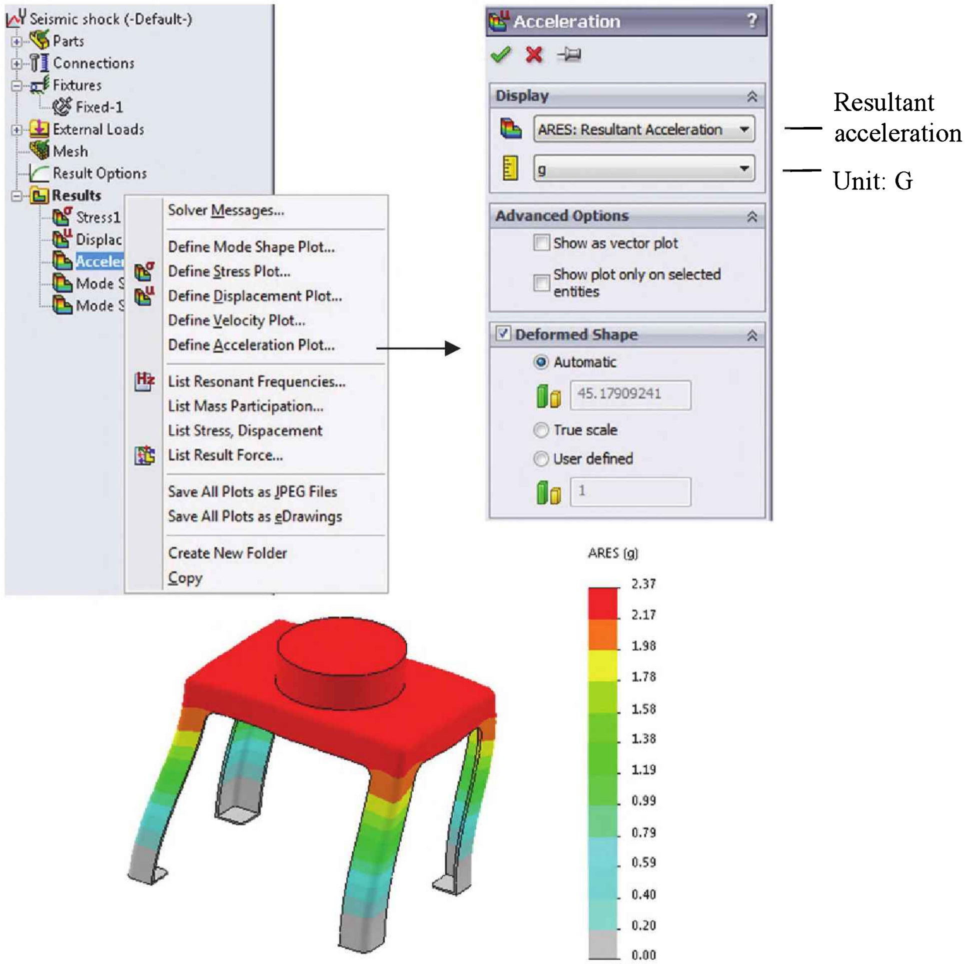


Figure 17-13: The maximum resultant acceleration during the Zone 4 seismic event.

The maximum resultant acceleration relative to the base is 2.37G.

Even though the maximum acceleration plot (as well as other plots) may be animated, the animation has no physical meaning in a **Response Spectrum** study. Remember that the results show the maximum values of acceleration during the entire duration of excitation. It is unknown at which point in time the maximum response happens. Also remember that the maximum response is the result of an arbitrary combination of individual modal responses, here performed using the **ABS** method. The same applies to all results of a **Response Spectrum** analysis.

Open the **Mass Participation** window as shown in Figure 17-14.

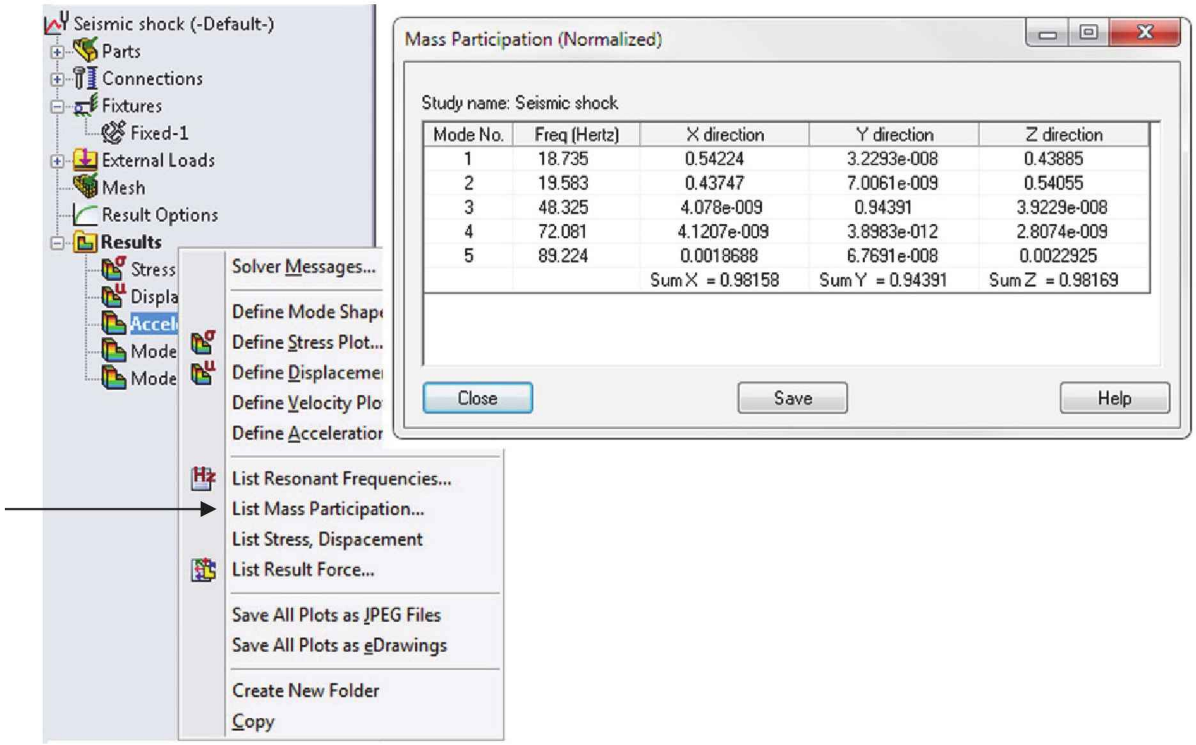
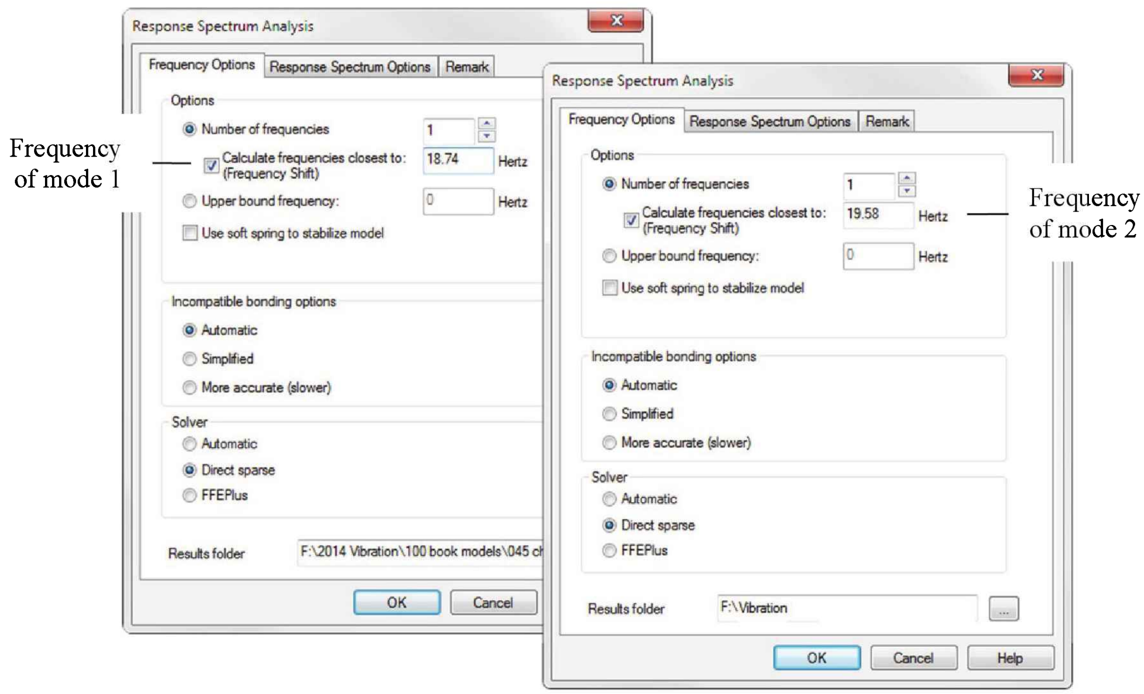


Figure 17-14: Mass participation of the first five modes.

Modal mass participation is normalized to add up to 1 in each of the global X, Y, and Z directions.

A review of the **Mass Participation** indicates that only the first and second modes participate in response to the seismic shock. The third mode with its frequency of 48.3Hz is still within the range of excitation frequencies 0-50Hz, but its direction is orthogonal to the direction of excitation.

To gain a better understanding of how individual modes contribute to the seismic response we will create two more **Response Spectrum** studies: *Seismic shock mode 1* and *Seismic shock mode 2* with **Frequency Options** shown in Figure 17-15. Each study will model a seismic response using only one mode.



Frequency Options in study *Seismic shock mode 1*

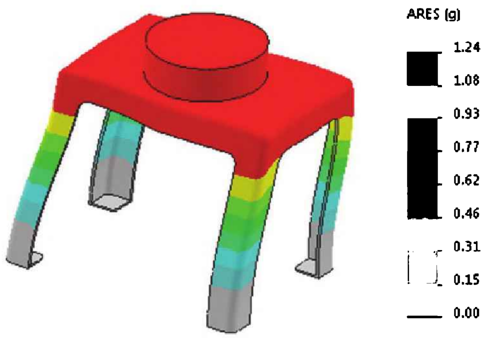
Frequency Options in study *Seismic shock mode 2*

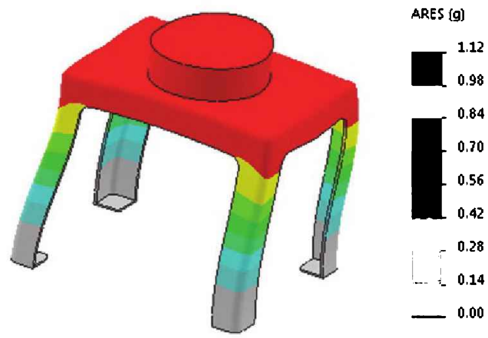
<u>Figure 17-15: Frequency Options in the two studies used to analyze the individual mode contributions to the seismic response.</u>

Each study is based on only one mode.

Notice that the **Frequency Options** of study *Seismic shock mode 1* could also be defined by specifying the number of frequencies equal to one. This is because 18.74Hz is the frequency of the first mode.

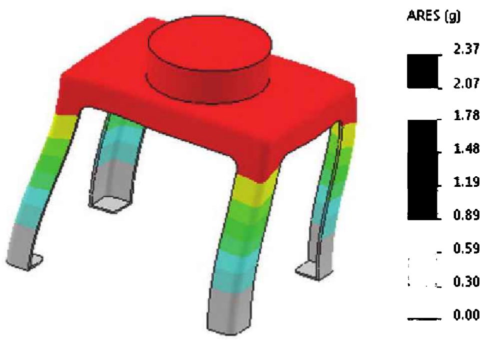
Results of the three studies: *Seismic shock*, *Seismic shock mode 1* and *Seismic Shock mode 2* are shown in Figure 17-16:





Study *Seismic shock mode 1*

Study *Seismic shock mode 2*



Study *Seismic shock*

Figure 17-16: Acceleration results of studies based on mode 1 and mode 2 (top), and acceleration results based on the five modes out of which only the first two contribute to the response (bottom).

The maximum acceleration based on mode 1 is 1.24G, the maximum acceleration based on mode 2 is 1.12G. The acceleration based on both modes is 2.37G.

The maximum acceleration found in study *Seismic shock* is the **Absolute Sum** of the maximum acceleration in each mode. The difference in the second decimal place is caused by numerical error.

Presently, the **Absolute Sum** is the only method of modal combination that can be used in a **Response Spectrum** study. Due to a programming error in Simulation 2014 other methods of modal combination give the same results as the **Absolute Sum**.

18: Nonlinear vibration

Topics covered

- Differences between linear and nonlinear structural analysis
- Types of nonlinearities
- Bending stiffness
- Membrane stiffness
- Modal damping
- Rayleigh damping
- Linear Time response analysis
- Nonlinear Time response analysis
- Modal Superposition Method
- Direct Integration Method

Difference between linear and nonlinear structural analysis

A structural analysis problem, be it static or time dependent, is always concerned with stiffness of the analyzed structure. If stiffness remains constant during the process of load application then the problem is linear. If stiffness changes then the problem is nonlinear. Nonlinear analyses are classified into different types based on the origin of the nonlinear behavior. For example, in nonlinear geometry analysis stiffness changes because of changes to a structure's geometry, in nonlinear material analysis stiffness changes because of changes in material properties etc. There are many other types of nonlinear analyses. For more information refer to "Engineering Analysis with SolidWorks Simulation".

The nonlinearity in the problems analyzed in this chapter is geometric; it is caused by changes in the structure's shape and requires **Large displacement** analysis. However, the term **Large displacement** is incorrect because it implies that displacements must be large in order for the nonlinear effects to show up even though in many cases small displacements are sufficient to change the stiffness significantly.

Bending and membrane stiffness

Consider a beam represented by the model PLANK, supported on both sides by hinges and subjected to 1MPa pressure as shown in Figure 18-1. The beam material is Alloy Steel, with a thickness of 25.4mm (1").

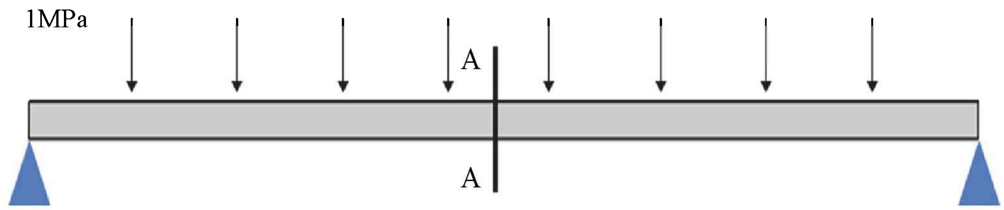


Figure 18-1: A flat beam simply supported on both ends.

The two hinge supports can't move in the horizontal direction.

The bending stress distribution across the beam thickness in the cross section A-A changes as the beam deformation progresses (Figure 18-2).

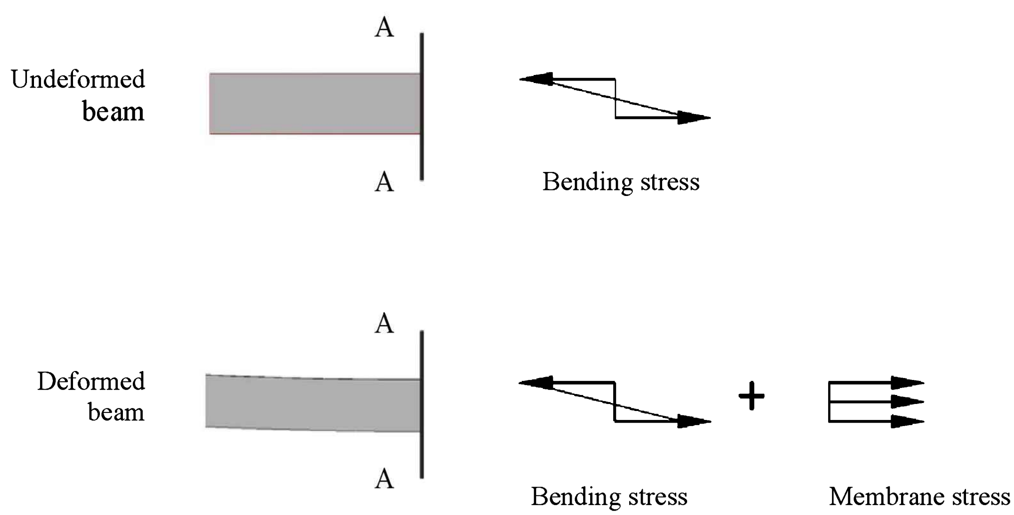


Figure 18-2: Bending stress distribution in cross section A-A in the undeformed beam (top) and in the deformed beam (bottom).

The flat beam responds to load only with bending stress. The curved beam responds to load with bending stress and with tensile stress called membrane stress. Notice that membrane stiffness wouldn't come into existence if one the supports (Figure 18-1) could slide in the horizontal direction.

When the load is first applied and the beam is still flat, the beam can respond only with bending stresses. We may say it only has bending stiffness. As deformation progresses and the beam becomes curved, tensile stresses develop and the beam acquires a new type of stiffness: membrane stiffness. The name "membrane" comes from the analysis of very thin walls called membranes where bending stiffness is negligible and all stiffness is generated by membrane stress. In our case, the stiffness of the curved beam is a superposition of bending and membrane stiffness.

The PLANK model has two configurations: *01 solid* and *02 surface*. We'll use *02 surface* which allows for meshing with shell elements.

A static analysis accounting for nonlinear effects requires either a **Static** study with the **Large displacements** option, or a **Nonlinear** study with the **Large displacements** option. We'll use **Static** studies.

Setup and run two **Static** studies: a linear analysis titled *01 static LIN* and a nonlinear static analysis titled *02 static NL*. To create the nonlinear analysis *02 static NL* you may copy *01 static LIN* into *02 static NL* and select **Large displacements** in the study properties. Remember to define restraints as **Immovable** (not **Fixed**) to model hinge supports on both sides. Contrary to solid elements, this is because shell elements differentiate between these two types of restraints.

Figure 18-3 shows the comparison of displacement results obtained with a **Static** study with and without the **Large displacement** option.

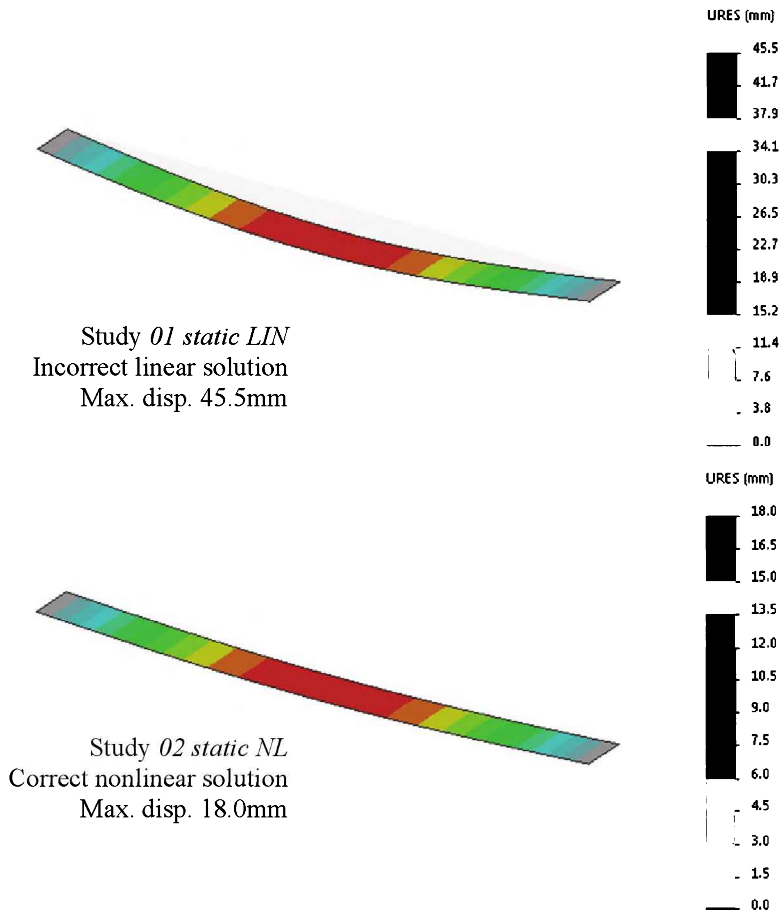


Figure 18-3: Comparison of displacement results of the PLANK treated as linear (top) and nonlinear (bottom) problem.

The scale of deformation (1:1) is the same in both plots.

As can be seen in Figure 18-3, the linear solution ignores the stiffening effect of membrane stresses. Consequently, it severely underestimates the stiffness. Having reviewed displacement results, compare von Mises stress results and notice that the linear analysis produces stresses above yield and the nonlinear analysis produces stresses below yield (Figure 18-4).

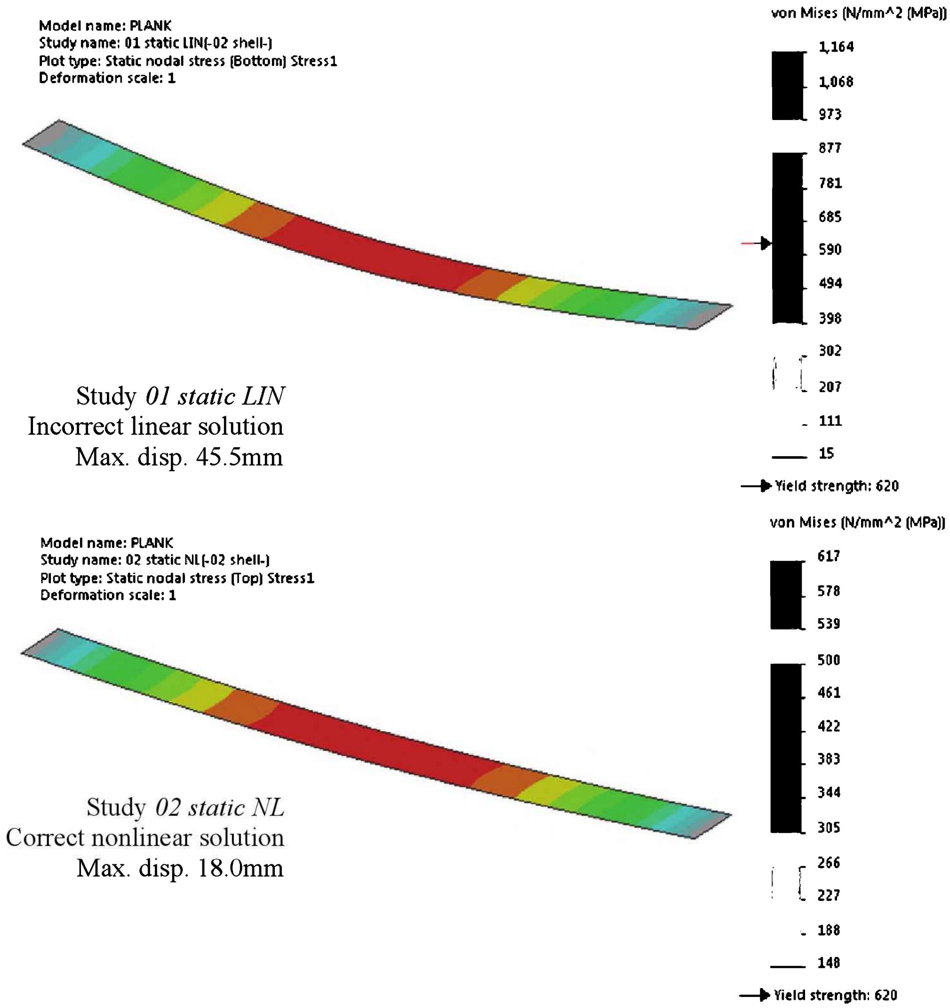


Figure 18-4: Comparison of stress results of the PLANK treated as linear (top) and nonlinear (bottom).

The scale of deformation (1:1) is the same in both plots. Stress results on the Top of shell elements are shown. The top of elements is the side opposite to where pressure is applied. Review the results on the bottom side. Notice that the linear analysis produces the same von Mises stress on both sides while the nonlinear analysis produces a much higher stress on the top side. This is the effect of membrane stress present in nonlinear analysis.

After this short review explaining the nonlinear nature of the PLANK problem, we move on to the analysis of vibration which will include nonlinear vibration response. This type of analysis is computationally demanding, this why we use the PLANK model in the *02 surface* configuration suitable for meshing with shell elements. The use of shell elements reduces the solution time significantly. To reduce the solution time even more, use shell elements with a size of 20mm. Large elements are acceptable because the stress distribution across the shell element thickness is built into the element definition and is not dependent on the element size. Furthermore, the overall model stiffness is weakly dependent on the element size. You may want to confirm this by running the same problem with several meshes using different element sizes.

Set up a **Frequency** study titled *03 modal* to find the first two modes of vibration. Again, remember to define **Immovable** (not **Fixed**) restraints. Results of the modal analysis are shown in Figure 18-5.

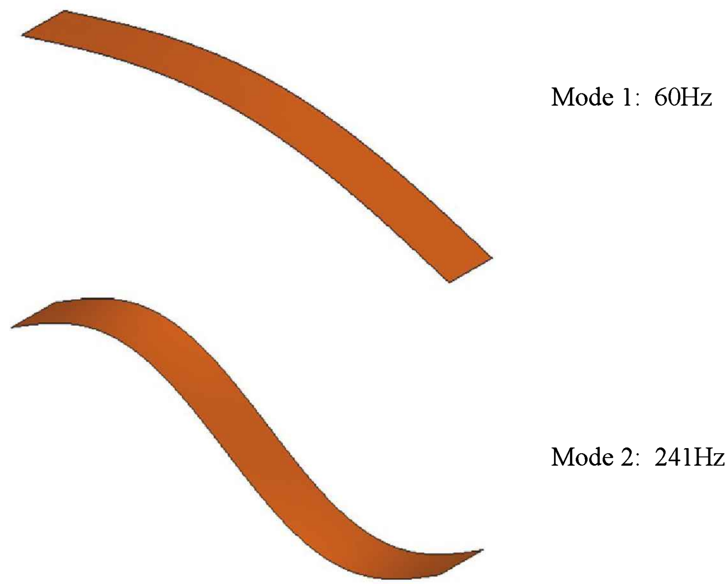


Figure 18-5: Results of modal analysis.

The first two modes are shown. The rotation on both ends is enabled by hinge supports on both sides.

Notice that modal analysis works with the initial stiffness; it does not take into consideration the membrane stiffening effect.

Having found the modes of vibration, we proceed with a linear **Time Response** analysis available in a **Linear Dynamic** study with the **Modal Time History** option. We already know that the problem is nonlinear. The results of the linear **Time Response** will serve only as a comparison to the results of the succeeding nonlinear **Time Response** analysis.

In the **Modal Time History** study, the PLANK model is being subjected to an impulse pressure shown in Figure 18-6.

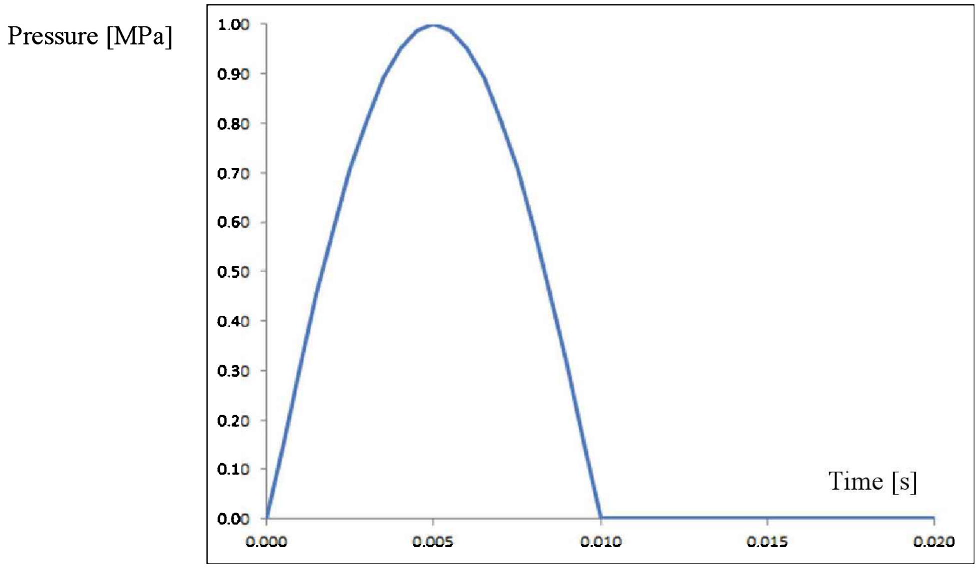
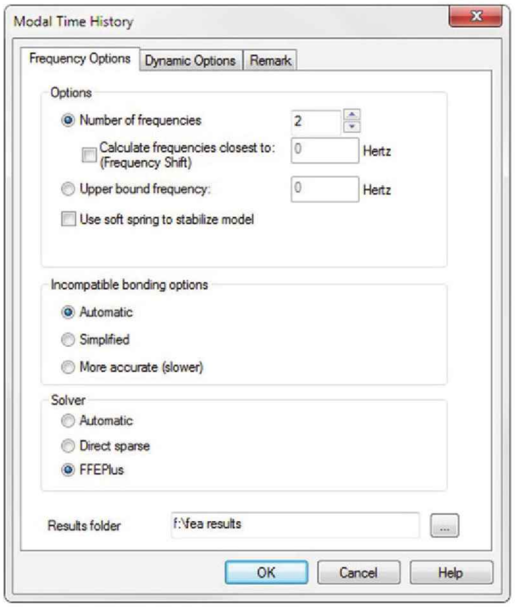
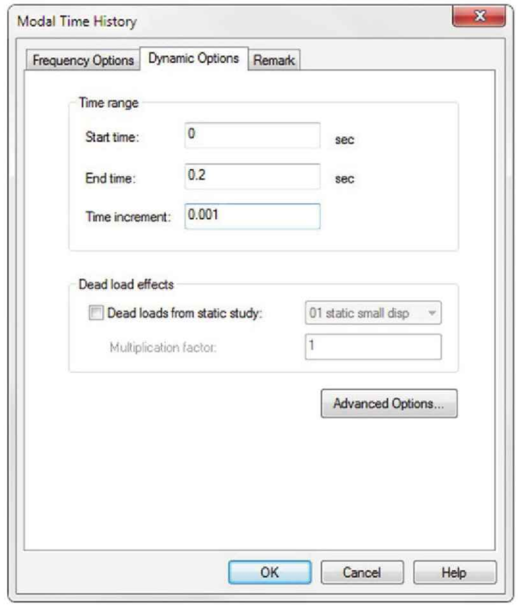


Figure 18-6: Time history of pressure acting on the top face of the PLANK.

During the first 0.01s, pressure rises from zero to 1MPa and drops back to zero. It remains zero until the end of the analysis. The frequency of the sine wave is 50Hz.

Define a **Modal Time History** study titled *04 dynamic LIN* with properties shown in Figure 18-7.





Frequency Options Dynamic Options

Figure 18-7: Properties of the Modal Time History study.

The study is based on two modes; its duration is 0.2s with a time step of 0.001s.

Define modal damping individaully for each mode (Figure 18-8).

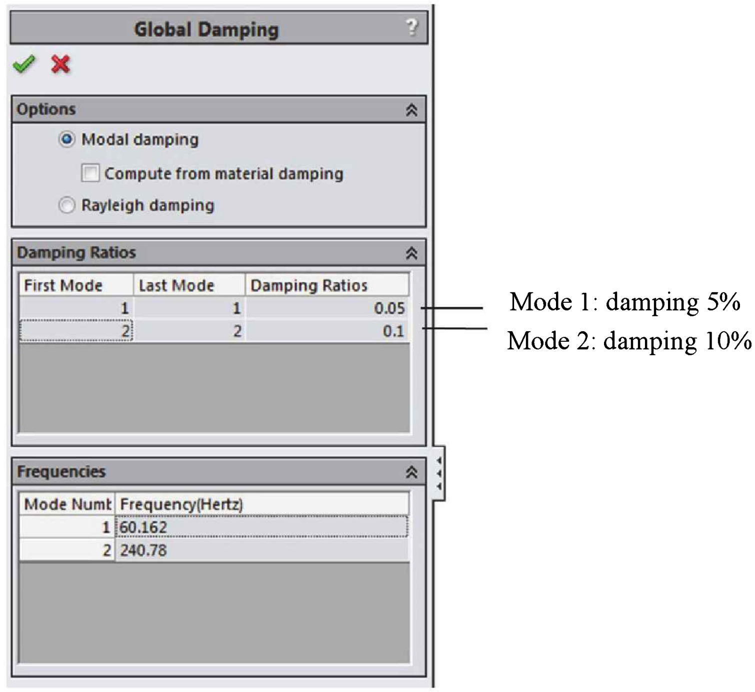


Figure 18-8: Modal damping definition.

Modal damping is defined individually for each mode. We assume these values are known from testing.

Following the definition of the study properties (Figure 18-7), the **Modal Time History** study is based on the superposition of the responses of two modes, each one with different modal damping (Figure 18-8).

Apply a 1MPa pressure to the top face (Figure 18-1); the same as in a static analysis and make it time dependent as shown in Figure 18-9.

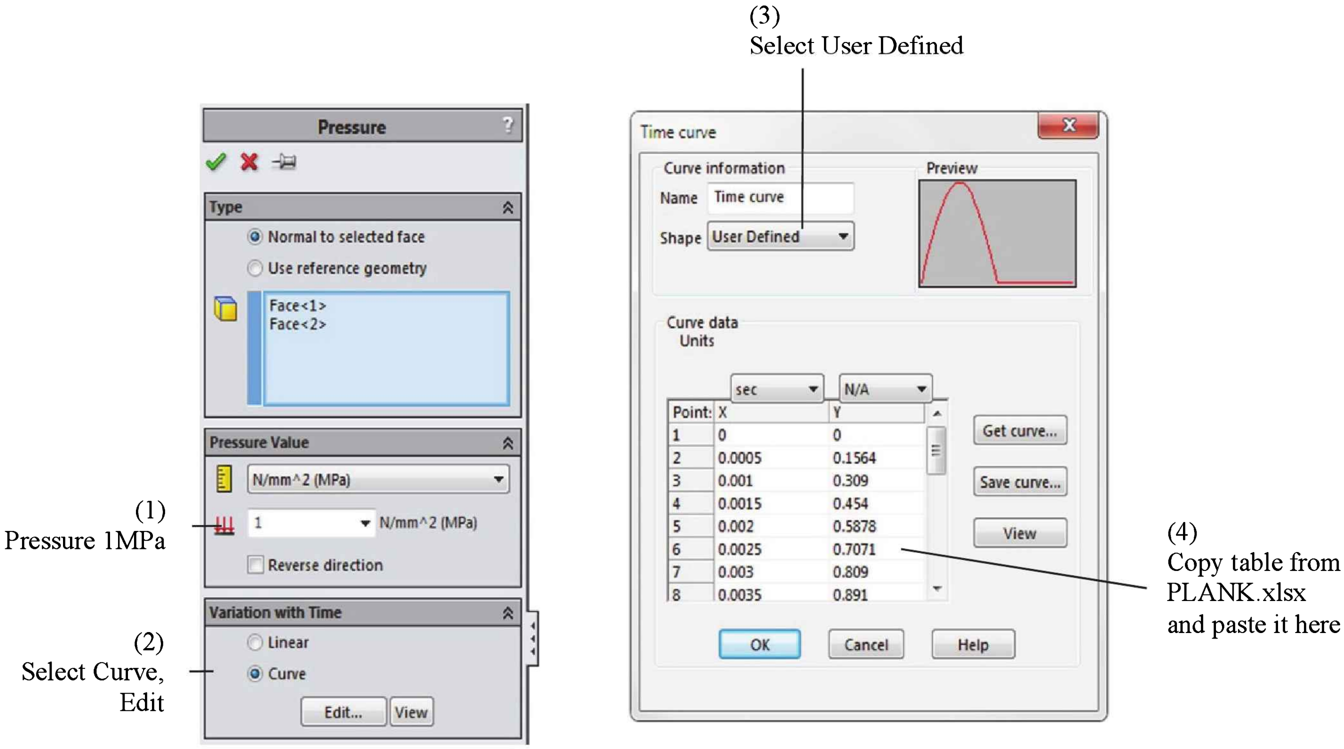


Figure 18-9: Definition of time dependent load.

Copy the table from the spreadsheet titled "PLANK.xlsx" into the Time Curve definition window. The table lists the time dependent multiplier to the specified pressure magnitude 1MPa.

Notice that the time duration in the table copied from PLANK.xlsx in only 0.02s, the same as shown in Figure 18-6. The analysis duration specified in the study properties shown in Figure 18-6 is 0.2s. The program will extrapolate the time curve as zero from 0.02s to 0.2s.

Define **Immovable** restraints identical to the previously completed studies. Define **Results Options** as shown in Figure 18-10.

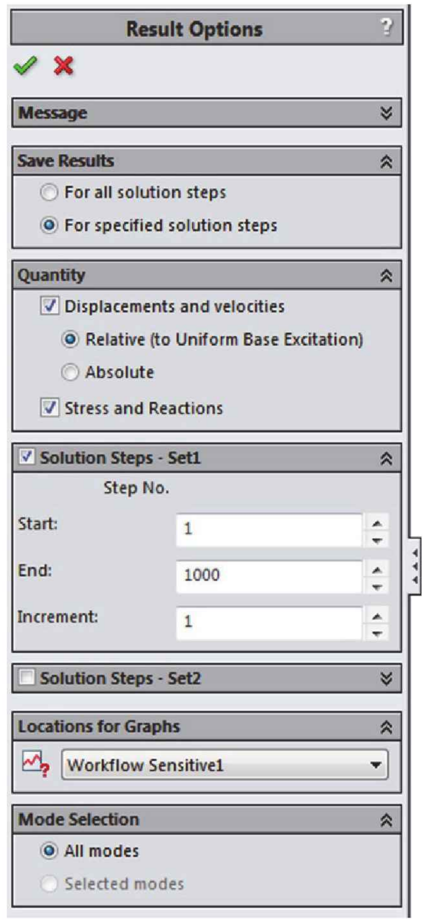


Figure 18-10: Definition of Result Options.

Select the Workflow Sensitive sensor which has been defined in the SolidWorks model.

The study is now ready to run. Obtain the solution and notice that it solves very quickly because it is a linear analysis, and as such, it is based on the **Modal Superposition Method**.

To review results, construct a graph of the time history of displacement in the mid-span, where the **Workflow Sensitive** sensor has been defined (Figure 18-11).

Workflow Sensitive sensor

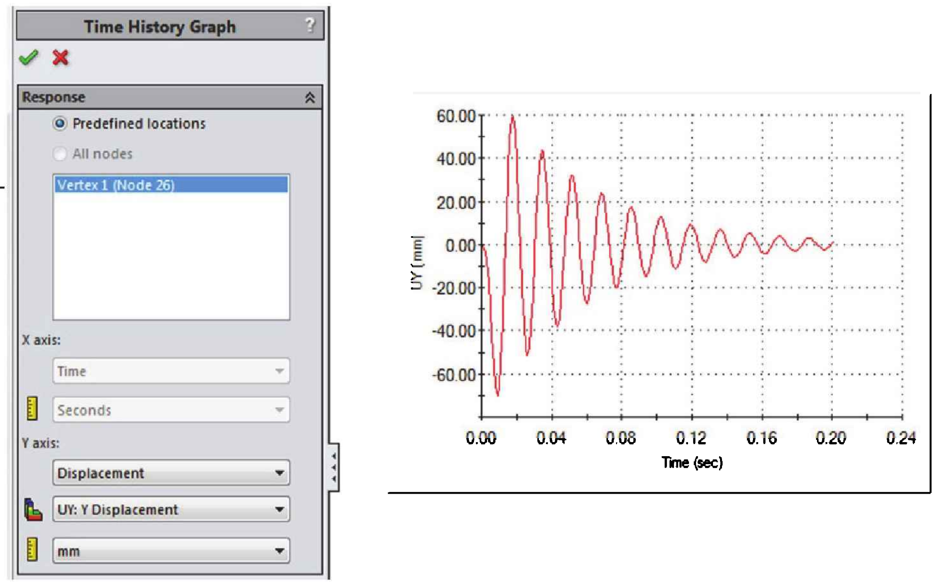


Figure 18-11: Displacement time history in the middle of the beam span.

Select the Workflow Sensitive sensor which has been defined in the SolidWorks model.

Select the UY displacement component.

Notice that when the impulse load disappears after 0.02s the beam performs free damped vibration behaving as a single degree of freedom oscillator. Even though the **Modal Time History** study is based on two modes, the first mode dominates the vibration response.

Previously completed static analysis of this model indicated the importance of nonlinear effects. The nonlinear effects are also present in the vibration response. As we'll soon demonstrate, the above linear solution is incorrect.

We will now perform analysis of the nonlinear vibration response starting with a comparison of the number of Degrees of Freedom (DOF) in the linear and nonlinear vibration problems. The linear vibration problem uses the **Modal Superposition Method**. Two modes have been specified in the study properties; therefore the problem is represented by two DOFs. The nonlinear problem can't use the **Modal Superposition Method** and must solve all equations of motion in terms of all degrees of freedom present in the model using the **Direct Integration** method.

Look up the **Solver Message** in any of the completed studies to see that the number of DOFs in the model is 6408 (Figure 18-12).

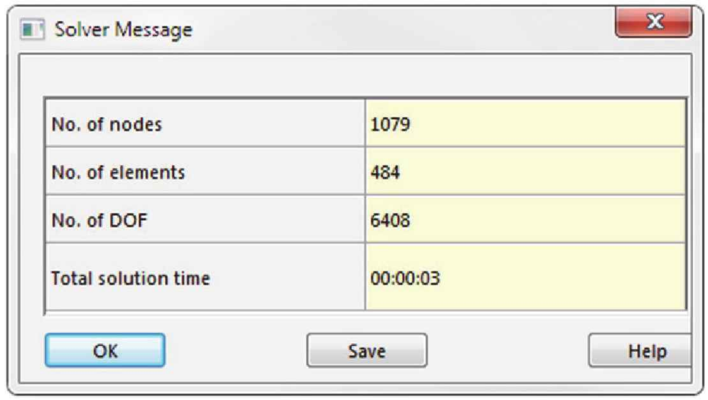


Figure 18-12: The number of DOFs in the PLANK model.

The number of DOFs in the model is 6408. It is not equal to the number of nodes multiplied by six because some DOFs are removed by the restraints.

The comparison between 2 DOFs in the linear problem and the 6408 DOFs in the nonlinear problem gives us an idea how numerically intensive nonlinear vibration problems are. This is why the PLANK model is so simple – otherwise it wouldn't solve reasonably fast.

Another consequence of the inability to use the **Modal Superposition Method** is that we can't use modal damping. Damping in nonlinear problems must be defined as **Rayleigh damping**. **Rayleigh damping** makes an arbitrary assumption that the damping matrix is a linear combination of the mass and stiffness matrices. This assumption is a mathematical convenience for the purpose of simplification since there is no physical justification for this. **Rayleigh Damping** is specified by two damping constants: α and β which are used as multipliers of the mass matrix M and the stiffness matrix K when calculating damping matrix C:

$$[C] = \alpha[\mathrm{M}] + \beta[\mathrm{K}] \qquad \text{(Eq. 18-1)}$$

$$\frac{\alpha}{2\omega} + \frac{\beta\omega}{2} = \zeta \qquad \text{(Eq. 18-2)}$$

where ω is the frequency and ζ is the damping ratio.

Alpha damping α is a viscous damping component also known as mass damping. It characterizes damping of lower frequencies.

Beta damping β is a hysteresis damping component also known as solid or stiffness damping. It characterizes damping of higher frequencies.

Rayleigh damping components α and β can be found if two modes and their modal damping are known. In our case:

	Frequency Hz	Frequency rad/s	Modal damping %
Mode 1	60.2	298	5
Mode 2	240.8	1192	10

Figure 18-13: Modal frequencies and their damping ratios in the PLANK model.

We need to know the above values to find Rayleigh damping.

Using the data in the above table, α and β can be found by solving two equations:

$$\frac{\alpha}{2\omega_1} + \frac{\beta\omega_1}{2} = \zeta_1 \qquad \frac{\alpha}{2\omega_2} + \frac{\beta\omega_2}{2} = \zeta_2 \qquad \text{(Eq. 18-3)}$$

You may also use the spreadsheet “PLANK PAYLEIGH.xlsx” to find α = 20.17 and β = 0.00012.

Knowing α and β, equation (18-2) may be plotted to visualize how damping coefficients change with frequency.

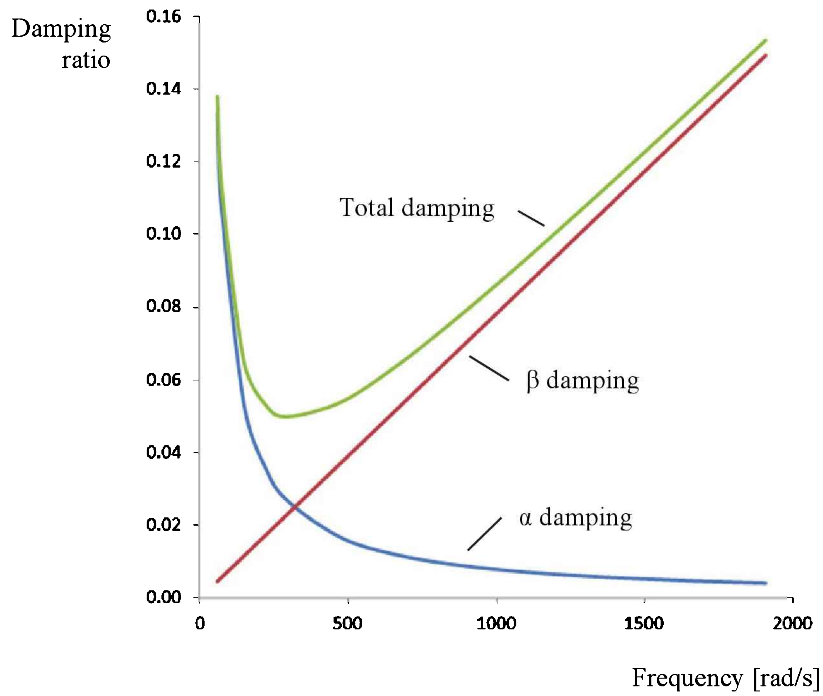


Figure 18-14: Damping ratio and its components as a function of frequency.

α damping drops with frequency, β damping is a linear function of frequency.

The graph in Figure 18-14 serves only to show the importance of **Alfa Damping** and **Beta Damping** for different frequencies. The total damping is not used in solving nonlinear vibration problems, but its components, α and β, used to formulate damping matrix C, are shown in equation (18-1). Create a **Nonlinear** study titled *05 dynamic NL* with the **Dynamic** option (Figure 18-15):

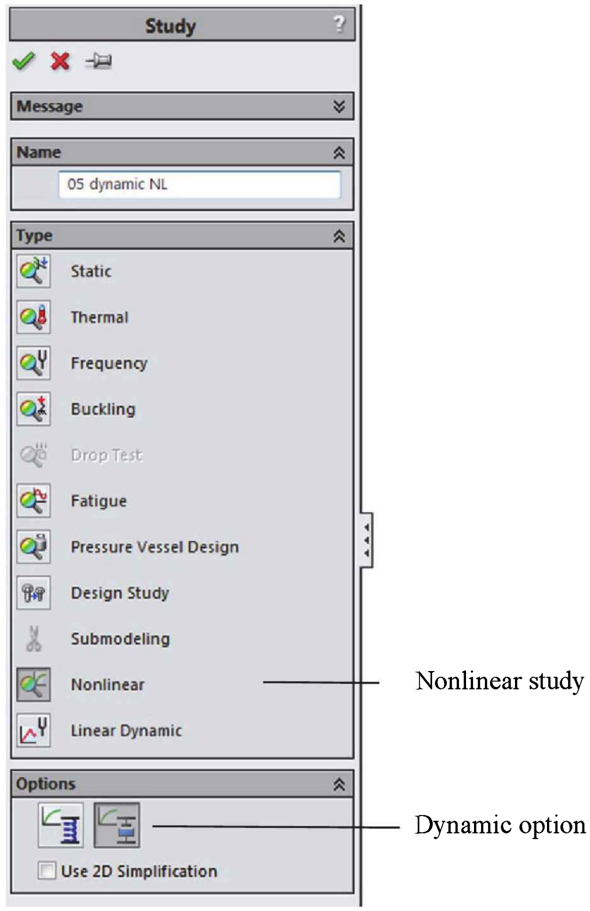


Figure 18-15: Definition of a Nonlinear study with the Dynamic option.

Define damping as shown in Figure 18-16:

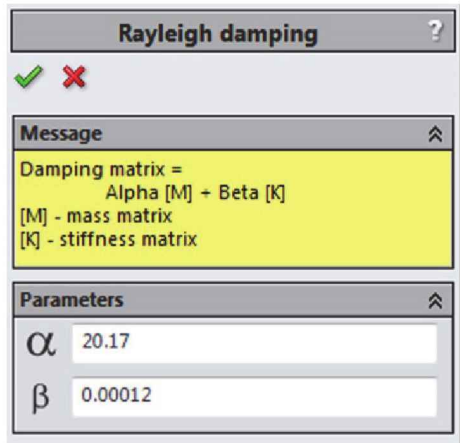


Figure 18-16: Definition of damping in a nonlinear vibration problem.

Enter the previously calculated constants α and β.

Apply the pressure load as a function of time the same way as defined in the linear study. Apply the restraints and mesh size the same as well.

We are now ready to define the properties of the nonlinear vibration study as shown in Figure 18-17:

End time 0.2s

Auto stepping with initial time increment 0.002s

Large displacement

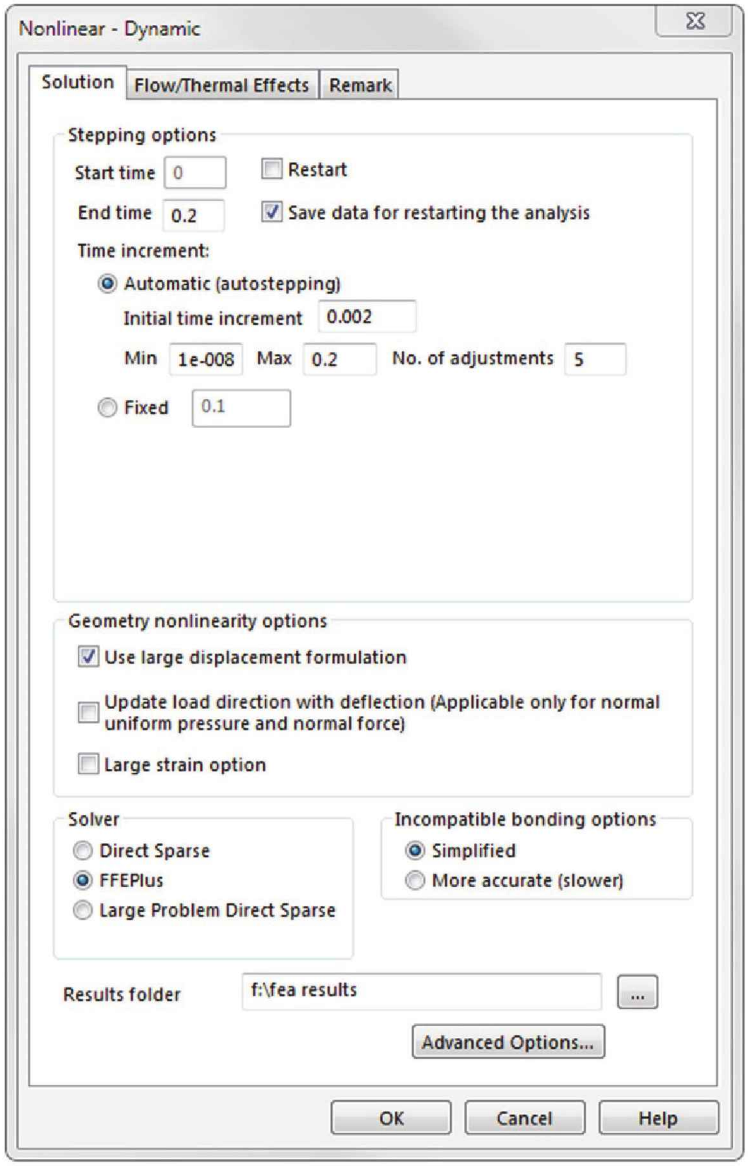


Figure 18-17: Properties of the nonlinear vibration study.

Specify the analysis duration as 0.2s, accept the default auto stepping.

Large Displacements must be specified or else the problem would not be nonlinear and membrane stiffening would not be accounted for.

Run the solution and acknowledge the warning message produced by the nonlinear solver (Figure 18-18).

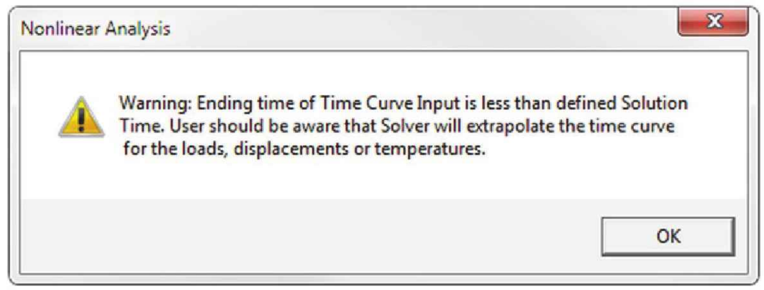


Figure 18-18: A warning message of the nonlinear solver issued when the duration of load is shorter than the duration of analysis.

The load duration is 0.02s, analysis duration is 0.2s. The end value of load is zero, therefore the extrapolated load will be zero for the remaining 0.18s.

Displacement responses for the linear and nonlinear solution are summarized in Figure 18-19.

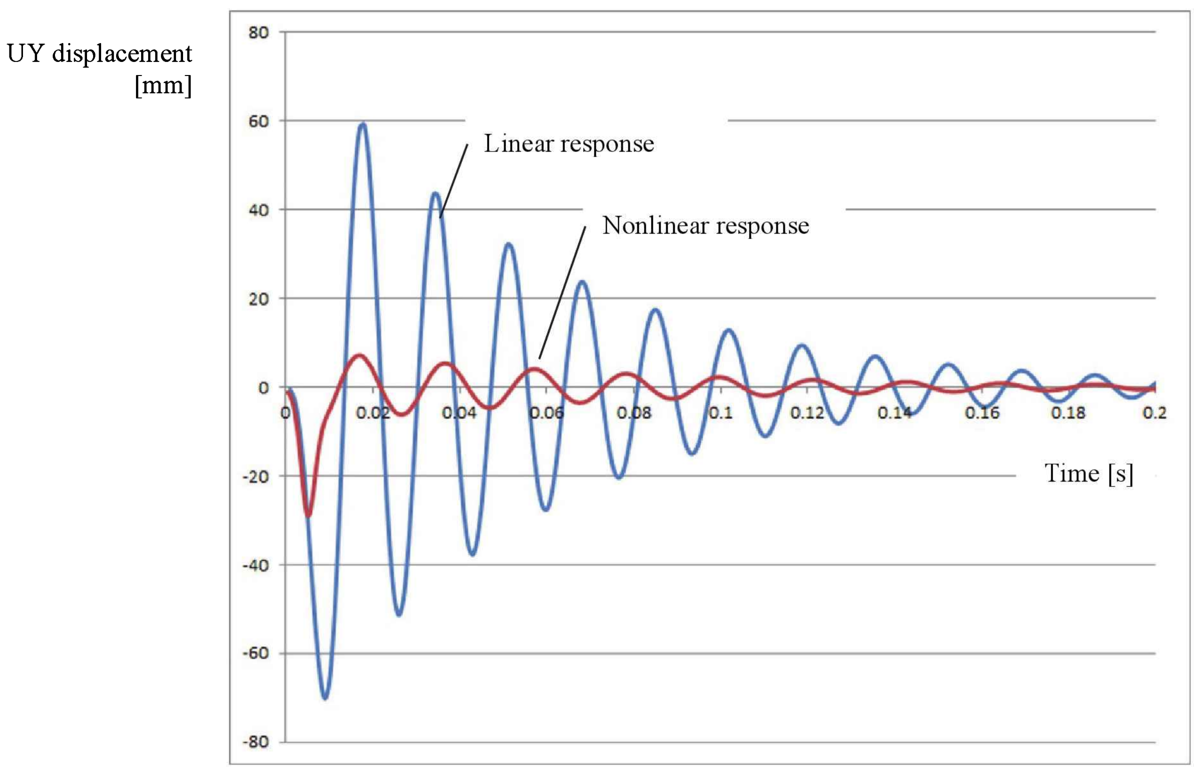


Figure 18-19: Comparison of linear and nonlinear UY displacement results in the mid-span of the PLANK.

The linear response doesn't account for membrane stiffening and produces displacement results with ~300% error.

The comparison between SX stress in the linear and nonlinear solutions in the sensor location is shown in Figure 18-20.

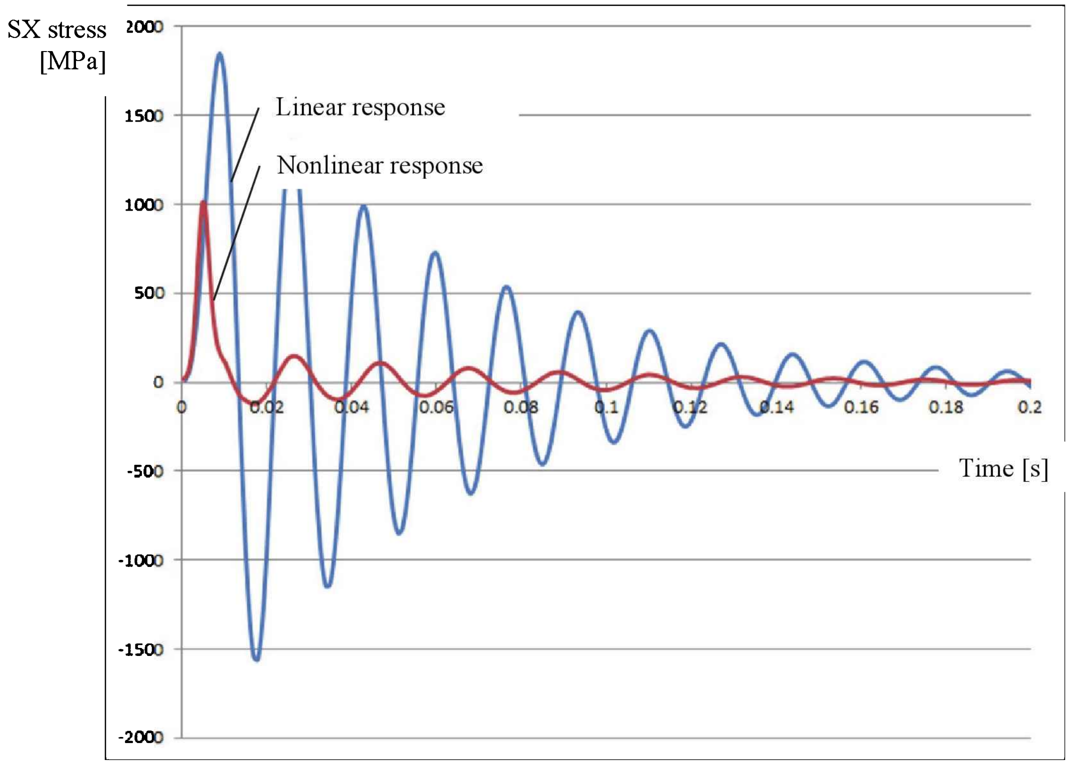


Figure 18-20: Comparison between linear and nonlinear SX stress results in the mid-span of the PLANK.

In the nonlinear response, the magnitude of tensile stress (positive) is larger than the magnitude of compressive stress (negative). This is due to the fact that the sensor location is on the top of the shell elements and does not change location from top to bottom with the reversal of beam deformation.

The frequency of the load time history (50Hz) is close to the first natural frequency of the model, this is the reason for high displacement and stress in the first cycle. Notice that SX stress exceeds the yield strength of the PLANK material (620MPa) in the first vibration cycle indicating that PLANK will experience yield. Whether or not this will result in a structural failure would require an analysis that includes a nonlinear material model.

Different magnitudes of tensile and compressive stresses in the nonlinear SX response proves that the nonlinear solution models membrane (tensile) stresses that develop in the PLANK. Remember that all results presented above are reported at the sensor location (Figure 18-21).

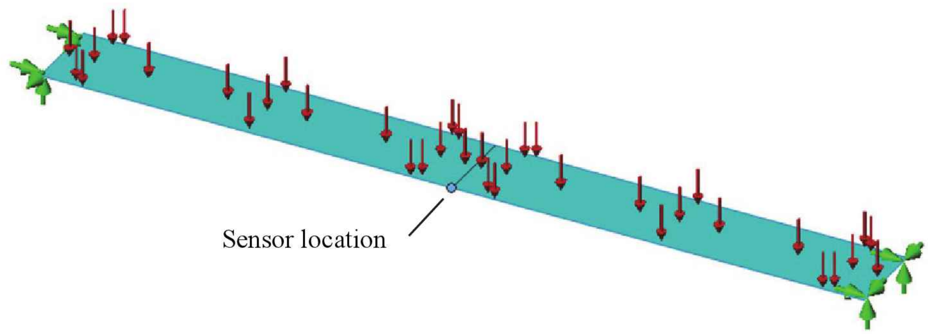


Figure 18-21: Sensor definition and its importance in the interpretation of results.

The sensor is defined at the end of a split line in the mid-span on the PLANK.

Review the shell element orientation (bottoms are orange) to confirm that the top of the PLANK (the side where pressure is applied) has been meshed with the bottoms of shell elements (Figure 18-22).

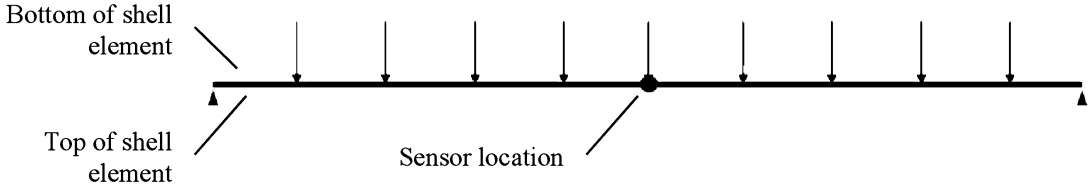


Figure 18-22: The sensor defined for the shells refers to the top of the shell element. The shell element model has no physical thickness.

The physical location of the sensor does not differentiate between the top and bottom of the model.

The sensor reads stress at the top of elements (which is the bottom of PLANK) and its location does not change with the reversal of beam deformation. This explains the different magnitudes of tensile and compressive stress in the nonlinear SX time response observed in Figure 18-20.

19: Vibration benchmarks

A benchmark is a standard point of reference against which things may be compared or assessed. In vibration analysis these benchmarks are known analytical solutions. Solutions of commercial FEA programs maybe compared to those known solutions to assess a program's performance.

A large number of vibration benchmarks are available. In this chapter we solve several NAFEMS benchmarks to see how well our results match those benchmark targets.

Since you are already well familiar with all types of vibration analyses with **SolidWorks Simulation**, we'll take shortcuts discussing the setup of these benchmark problems. In some cases the same model will be used for more than one benchmark test and studies will be created in different model configurations.

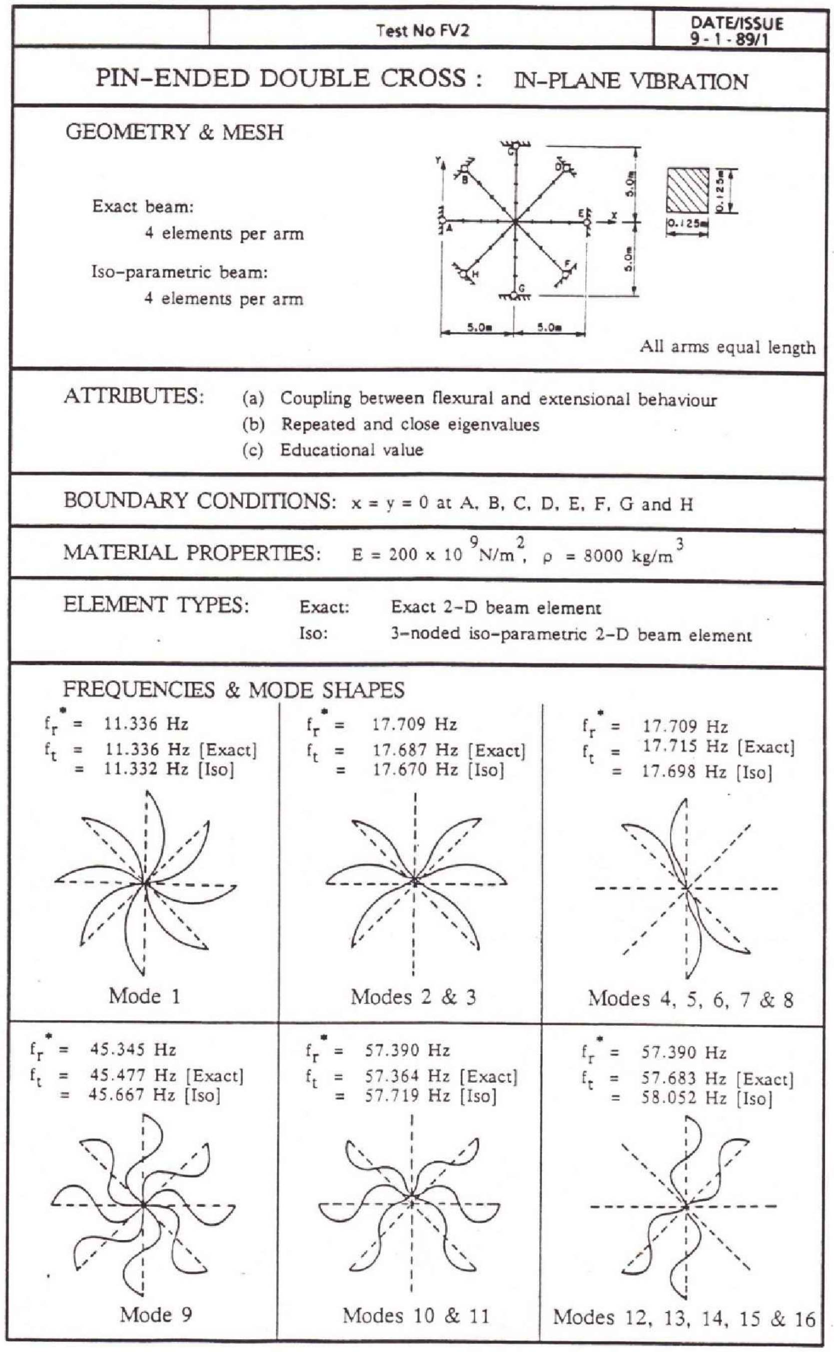
Models, studies and configurations used in this chapter are summarized in table in Figure 19-1.

Benchmark test	**SW model**	**Simulation study**	**SW model configuration**
NAFEMS test FV2	NAFEMS TEST FV2	Frequency *01 solids*	*01 solids*
		Frequency *02 shells*	*02 shells*
		Frequency *03 beams*	*03 beams*
NAFEMS test 5	NAFEMS TEST 5	Harmonic limited to Frequency *TEST 5 solids*	*01 solids*
		Harmonic limited to Frequency *TEST 5 shells*	*02 shells*
		Harmonic limited to Frequency *TEST 5 beams*	*03 beams*
NAFEMS test 5H	NAFEMS TEST 5	Harmonic *TEST 5 solids*	*01 solids*
		Harmonic *TEST 5 shells*	*02 shells*
		Harmonic *TEST 5 beams*	*03 beams*
NAFEMS test 5T	NAFEMS TEST 5	Modal Time History *TEST 5T*	*03 beams*
NAFEMS test 13	NAFEMS TEST 13	Frequency *TEST 13*	*Default*
NAFEMS test 13R	NAFEMS TEST 13	Random *TEST 13R*	*Default*

Figure 19-1: Summary of NAFEMS Benchmark test, corresponding models and their configurations.

NAFEMS test FV2

Pin-ended double cross: in-plane vibration



Test No FV2 — DATE/ISSUE 9-1-89/1

PIN-ENDED DOUBLE CROSS : IN-PLANE VIBRATION

GEOMETRY & MESH

Exact beam:
4 elements per arm

Iso-parametric beam:
4 elements per arm

ATTRIBUTES: (a) Coupling between flexural and extensional behaviour
(b) Repeated and close eigenvalues
(c) Educational value

BOUNDARY CONDITIONS: x = y = 0 at A, B, C, D, E, F, G and H

MATERIAL PROPERTIES: $E = 200 \times 10^9\ N/m^2$, $\rho = 8000\ kg/m^3$

ELEMENT TYPES: Exact: Exact 2-D beam element
Iso: 3-noded iso-parametric 2-D beam element

FREQUENCIES & MODE SHAPES

$f_r^* = 11.336$ Hz $f_t = 11.336$ Hz [Exact] $= 11.332$ Hz [Iso] Mode 1	$f_r^* = 17.709$ Hz $f_t = 17.687$ Hz [Exact] $= 17.670$ Hz [Iso] Modes 2 & 3	$f_r^* = 17.709$ Hz $f_t = 17.715$ Hz [Exact] $= 17.698$ Hz [Iso] Modes 4, 5, 6, 7 & 8
$f_r^* = 45.345$ Hz $f_t = 45.477$ Hz [Exact] $= 45.667$ Hz [Iso] Mode 9	$f_r^* = 57.390$ Hz $f_t = 57.364$ Hz [Exact] $= 57.719$ Hz [Iso] Modes 10 & 11	$f_r^* = 57.390$ Hz $f_t = 57.683$ Hz [Exact] $= 58.052$ Hz [Iso] Modes 12, 13, 14, 15 & 16

Figure 19-2: NAFEMS benchmark test FV2.

Material properties require custom material definition.

Open part model NAFEMS TEST FV2 and review its geometry: eight beams 5m long form a double cross, the beam cross section is a square with side length 0.125m. Material property: E=200000MPa, G=76923MPa, ν=0.3, ρ = 8000kg/m^3. All beams are pin supported at the ends (Figure 19-2).

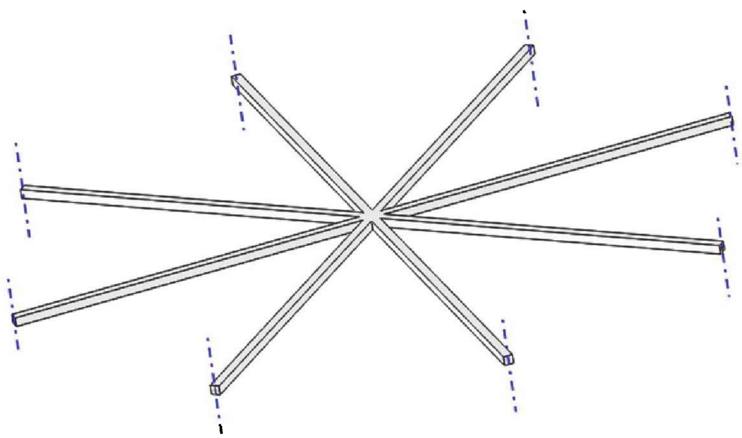


Figure 19-3: Geometry of the double cross model in *NAFEMS FV2* shown in the *01solids* configuration.

Pin support allows for rotation of each end about its own axis. The axes are shown as dotted lines.

Notice that the model geometry lends itself to different representations: volume (solid) geometry for meshing with solid elements, surface geometry for meshing with shell elements and curve (wireframe) geometry for meshing with beam elements. You'll find these three representations in configurations *01 solids*, *02 shells*, and *03 beams*. All configurations have the same custom material with properties specified in the NAFEMS test.

We'll test the model to see how well the results match the benchmark target frequency of 11.336Hz in the first mode. In each configuration create a **Frequency** study with the same name as the configuration title.

A **Frequency** analysis in **SolidWorks Simulation** doesn't have a 2D representation option. Therefore, we'll have to apply proper restraints to make sure that the first mode of vibration will be in-plane and corresponds to what is required in this benchmark test.

Change to *01 solids* configuration, create a **Frequency** study and define the restraints for the solid element model as shown in Figure 19-4.

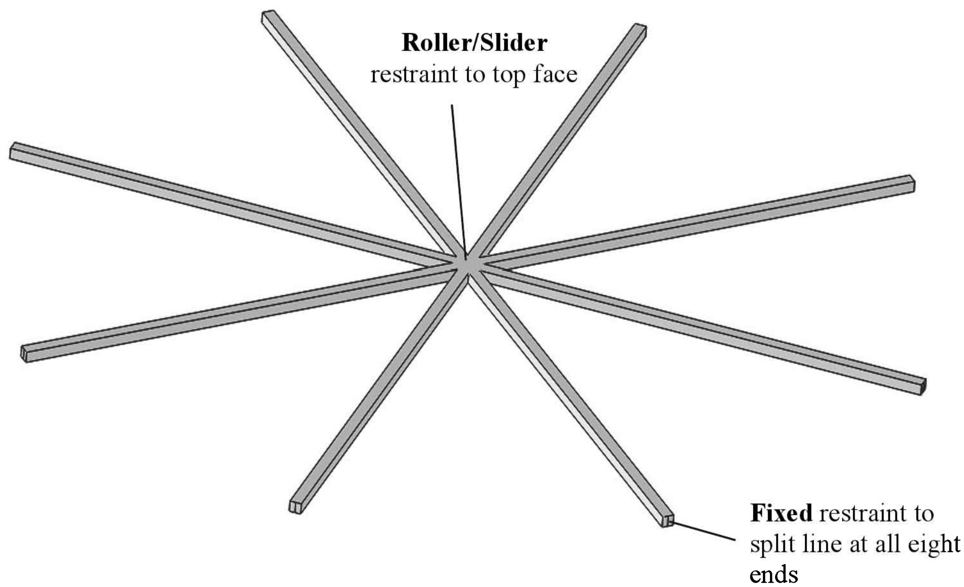


Figure 19-4: Restraints to the model in the *01 solids* study.

Restraints symbols are not shown. Roller/slide support may be applied to either top or to bottom face.

Remember that nodes of solid elements have three degrees of freedom and can't generate moment reactions. Therefore **Fixed** restraints to a straight line produce hinges. The **Roller/Slider** restraint eliminates out of plane modes to produce in-plane vibration as required by the test.

Use the default element size to mesh this model and run the solution.

Change to the *02 shells* configurations, create a **Frequency** study and define restraints for the shell element model as shown in Figure 19-5. To apply pin supports, use **Immovable** restraints. To apply restraints enforcing 2D vibration, use **Use Reference Geometry** restraints.

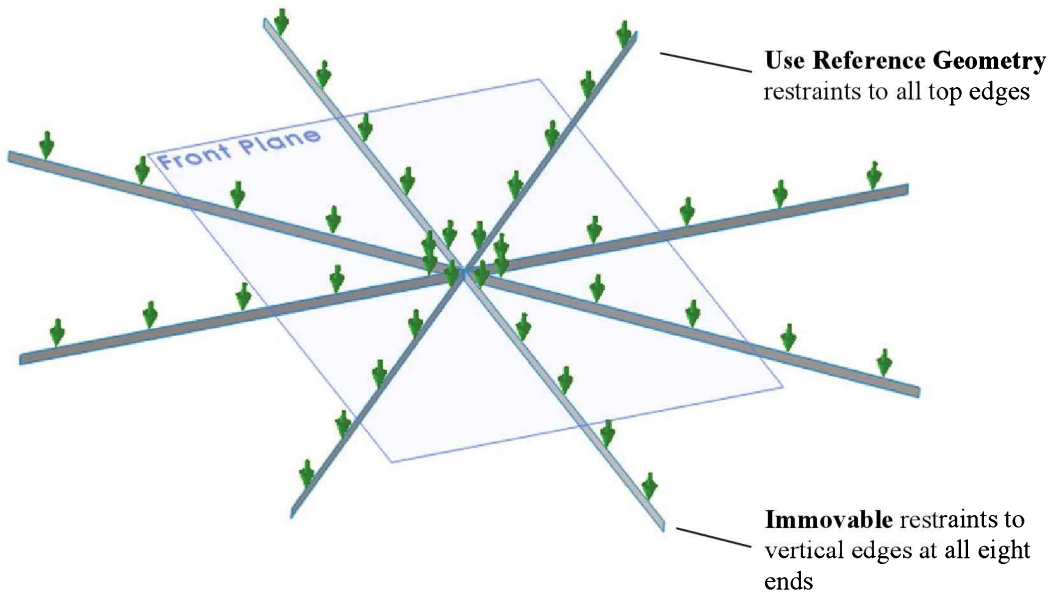


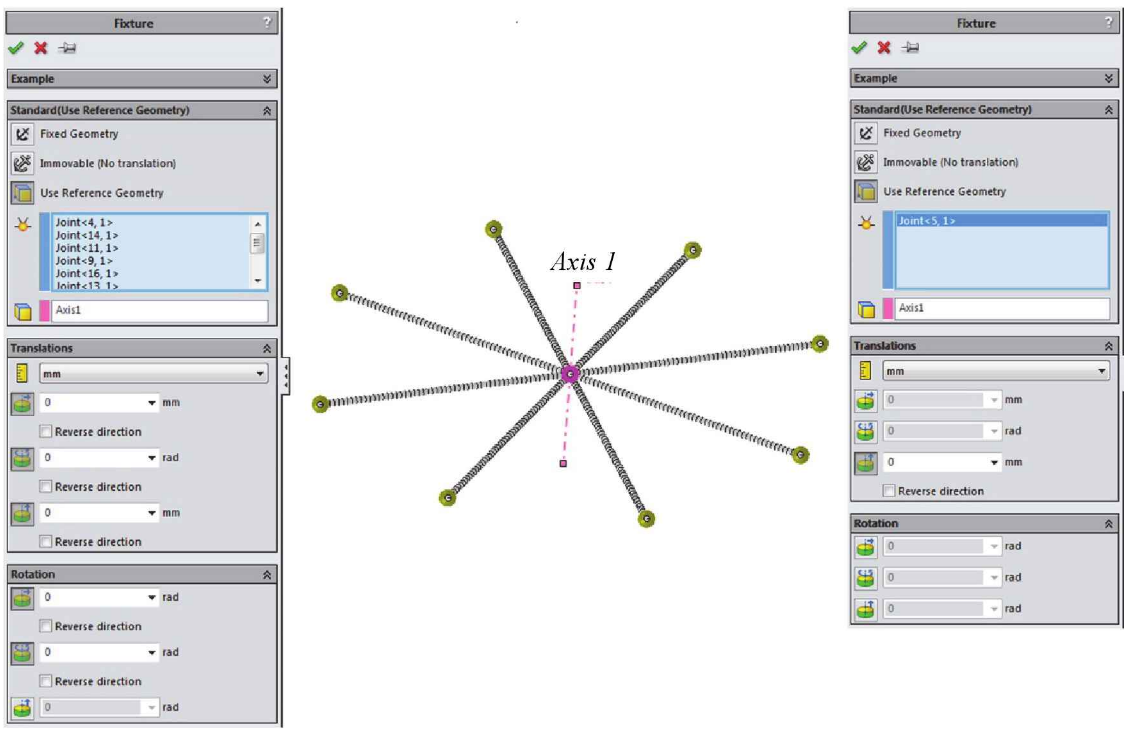
Figure 19-5: Restraints to the model in the *02 shells* study.

The Use Reference Geometry support may be applied to either the top or bottom face. Immovable restraint symbols are not shown.

The **Immovable** restraints do not affect rotational degrees of freedom of the shell elements and allow for rotation. **Use Reference Geometry** restraints use the **Front Plane** for reference and eliminate translations in the direction normal to this plane. This is necessary to eliminate out of plane modes of vibration. These restraints may be applied to either top or bottom edges.

Define the shell element thickness as 125mm, use the default element size to mesh this model and run the solution.

Change to the *03 beams* configuration. The model geometry is visually identical to the model in the *03 solids* configuration but the eight beams are not merged into one solid body. Instead, the model has eight solid bodies. Create a **Frequency** study and change all solid bodies to beams (right-click solid body and select **Treat as Beam**); calculate joints using **Edit Joints;** define restraints for beam element model are shown in Figure 19-6.



Restraints to the peripheral joints

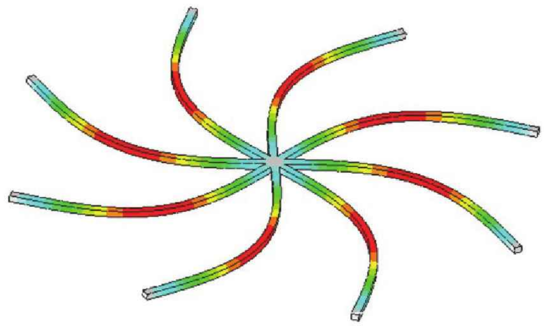
Restraint to the central joint

<u>Figure 19-6: Restraints to peripheral joints and to the central joint in the *03 beams* study.</u>

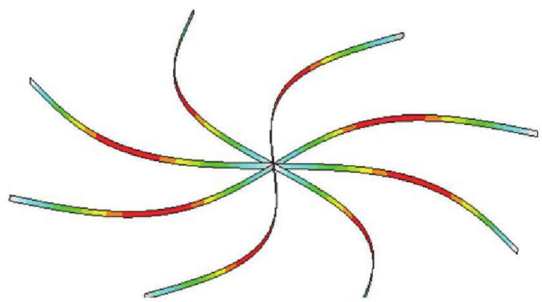
Restraints are defined in the cylindrical coordinate system associated with Axis1. Restraint symbols are not shown.

In the peripheral joints shown in Figure 19-6 (left), only rotation in the direction parallel to *Axis1* is allowed; this defines pin supports and eliminates out of plane motions. The central joint has axial translation restrained shown in Figure 19-6 (right) to eliminate lower out of plane modes. Higher out of plane modes are still possible but for us they are irrelevant. All we want to find is the first mode of vibration to see how well the results of **SolidWorks Simulation** match the benchmark target of 11.336Hz.

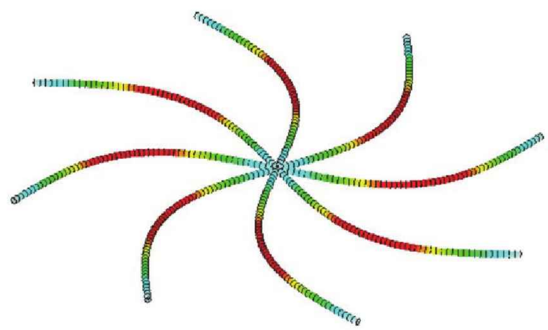
Results of all three studies are summarized in Figure 19-7.



Solid element model:
11.466Hz



Shell element model:
11.333Hz



Beam element model:
11.321Hz

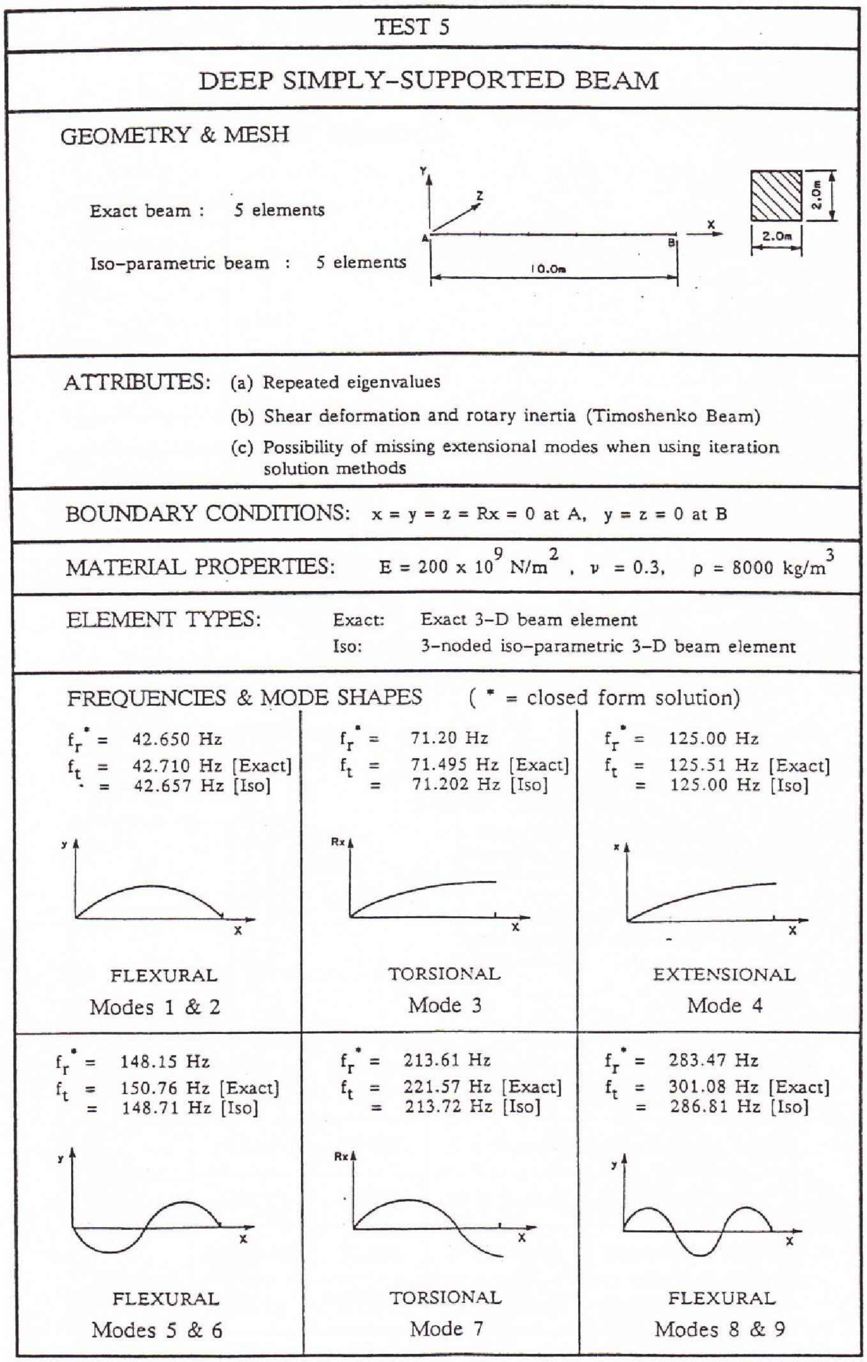
Figure 19-7: First mode of vibration of solid, shell, and beam models.

All shapes correspond to the same first mode of vibration which is an in-plane mode.

All results closely match the exact solution of 11.336Hz.

NAFEMS test 5

Deep Simply-Supported Beam



<u>Figure 19-8: NAFEMS TEST 5; Deep Simply-Supported beam.</u>

Custom material properties are the same as in test FV2.

Open part the model NAFEMS TEST 5 and review its geometry: the beams is10m long, the beam cross section is square with a side length of 2m; the material property is: E = 200000MPa, ρ = 8000kg/m^3. The beam has simple supports on both ends (Figure 19-9).

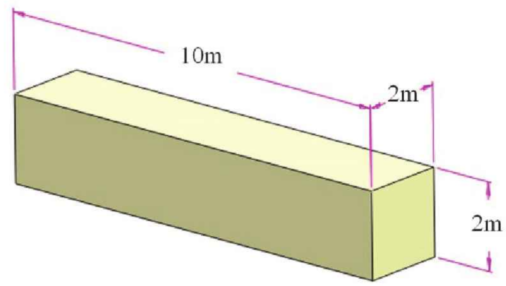


Figure 19-9: Deep Simply-Supported Beam, NAFEMS TEST 5.

Supports are shown schematically in the lower illustration.

We'll test the model to see how well the results match the benchmark target frequency of 42.65Hz in the first mode. The model geometry lends itself to different representations: volume (solid) geometry for meshing with solid elements, surface geometry for meshing with shell elements, and curve (wireframe) geometry for meshing with beam elements. You'll find these different representations in configurations *01 solids*, *02 shells*, and *03 beams*. In each configuration create a **Harmonic** study with names corresponding to names of the configurations: *TEST 5 solids, TEST 5 shells,* and *TEST 5 beams*. Why are we creating **Harmonic** studies rather than **Frequency** studies? Modal analysis can be conducted in both **Frequency** and **Harmonic** studies. Once we complete NAFEMS TEST 5 we'll continue with NAFEMS TEST 5H which does require a **Harmonic** analysis and we'll use the same studies for both tests.

Review the sensors defined in each configuration named *Solids*, *Shells*, and *Beams*. Notice that split lines are used to locate the sensors.

Change to the *01 solids* configuration, create a **Harmonic** study titled *TEST 5 solids* and define restraints for the solid element model as shown in Figure 19-10.

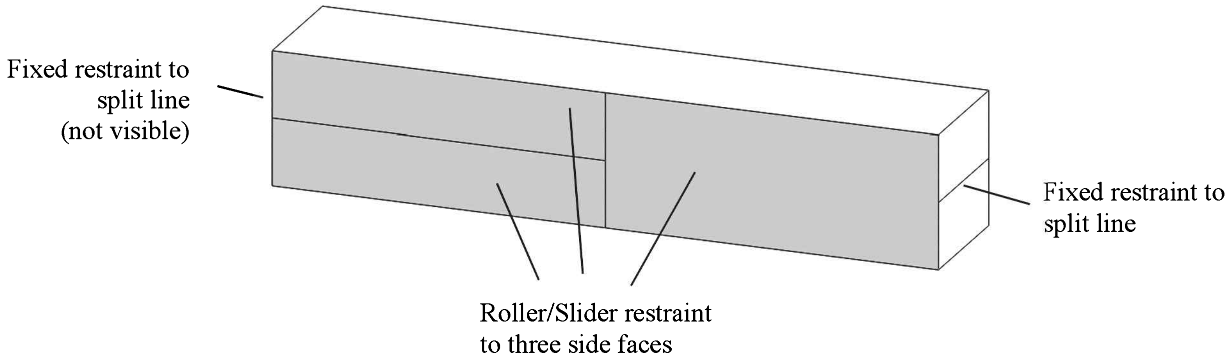


Figure 19-10: Restraints of the model in the *TEST 5 solids* study.

The Fixed restraint applied to a straight edge of a solid model defines a hinge restraint. The Roller/Slider restraints eliminates out of plane modes. Restraints symbols are not shown.

Notice that modal analysis is based on the assumption of linearity and that displacements are considered small. Therefore, modal analysis can't distinguish between a sliding and fixed support shown in Figure 19-11. Refer to "Engineering Analysis with SolidWorks Simulation" Chapter 14 for more information.

Figure 19-11: Simply supported beam with one fixed and one sliding support (top) and with both supports fixed (bottom). Modal analysis does not distinguish between these two configurations.

The top configuration is statically determinable; the bottom configuration is statically undeterminable.

Change to the *02 shells* configuration, create a **Harmonic** study titled *TEST 5 shells* and define restraints for shell element model as shown in Figure 19-12.

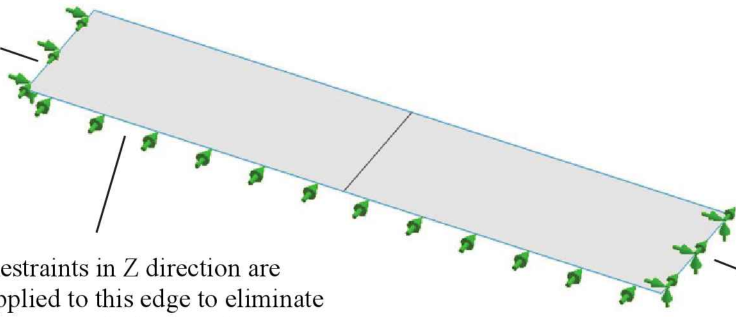


Figure 19-12: Restraints in the of model in the *TEST 5 shells* study.

The shell element model does differentiate between Immovable and Fixed restraints. Use Immovable to define a simple support.

In the **Shell Definition**, enter a shell element thickness of 2m and select the **Thick** shell formulation (Figure 19-13).

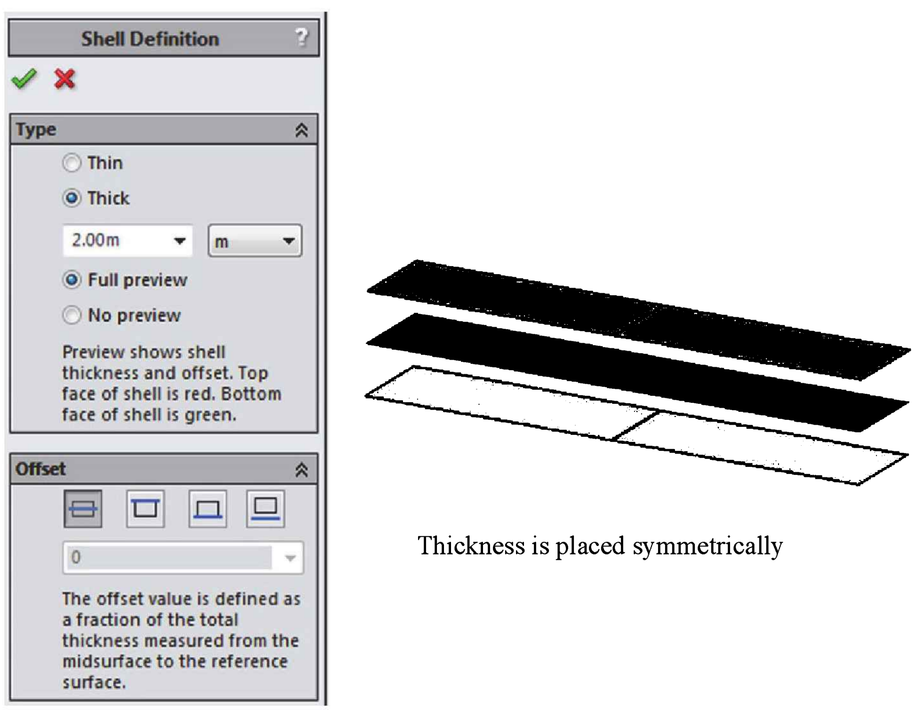


Figure 19-13: The Shell Definition uses the Thick shell formulation.

Select Middle surface for symmetric placement of thickness.

Change to the *03 beams* configuration and create a **Harmonic** study titled *TEST 5 beams*. Instruct **Simulation** to treat both solid bodies as beams; ignore the warning that the beams are too short. This will have no effect on the results produced by the beam element model. Define restraints for the beam element model as shown in Figure 19-14.

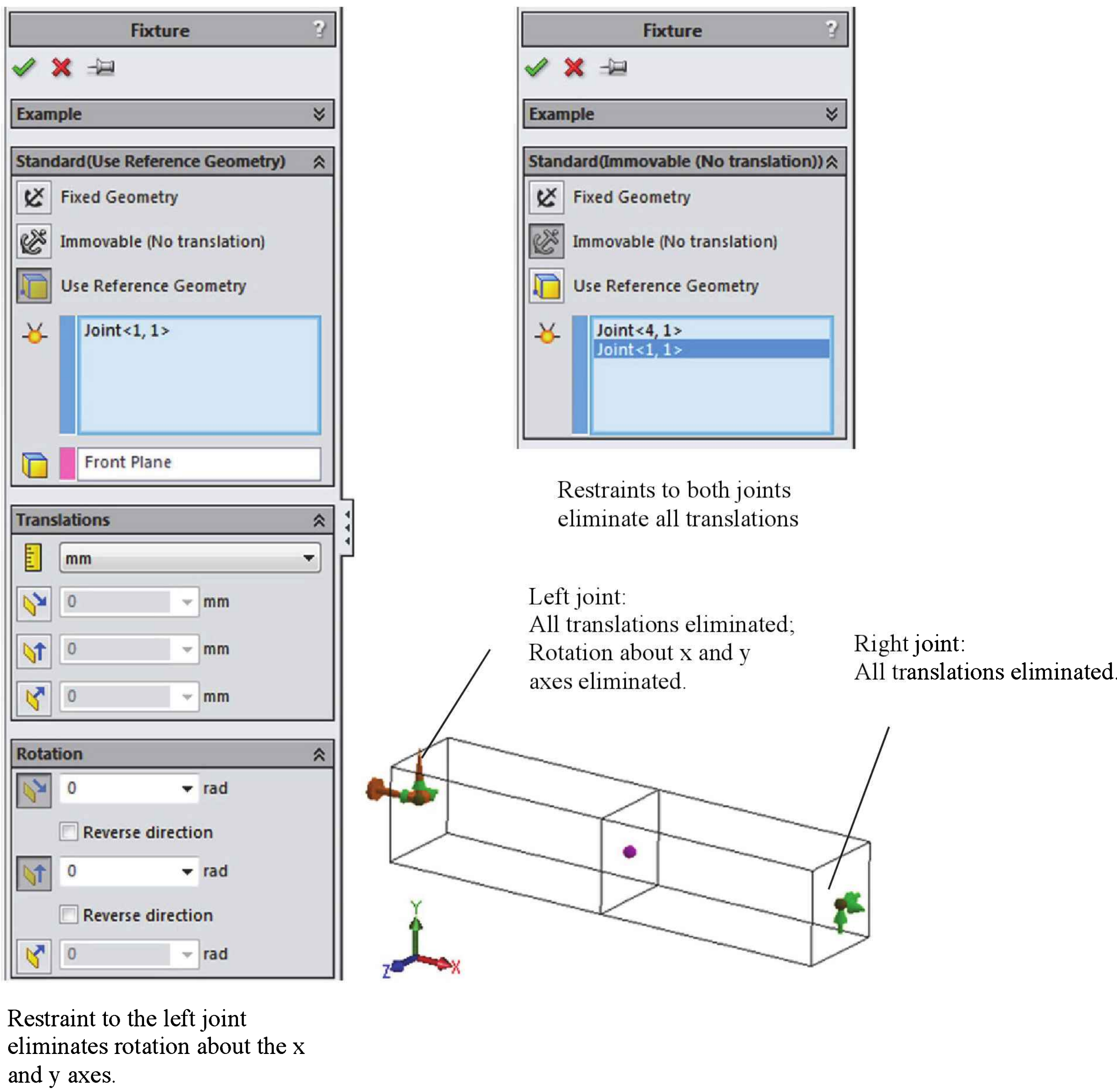
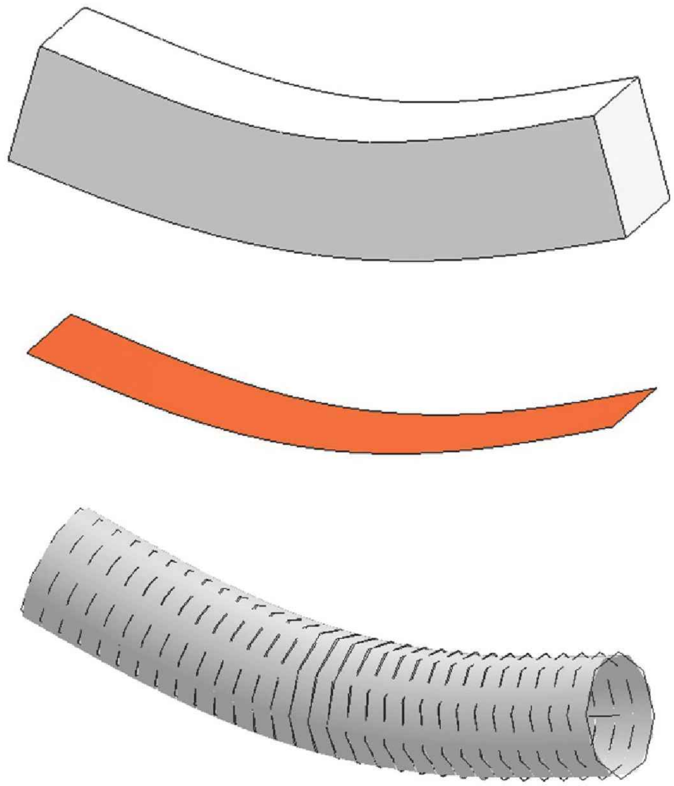


Figure 19-14: Restraints of the model in the *TEST 5 beams* study.

Immovable restraints to both sides leave the model with one Rigid Body Motion. This is why we need to restrain the model from rotation.

We have defined all three studies to the point where modal analysis can be run. Use the default mesh in all studies. Use **Run Frequency** to obtain modal solution of the above three studies.

Results are summarized in Figure 19-15.



Solid element model:
42.116Hz

Shell element model:
43.193Hz

Beam element model:
42.053Hz

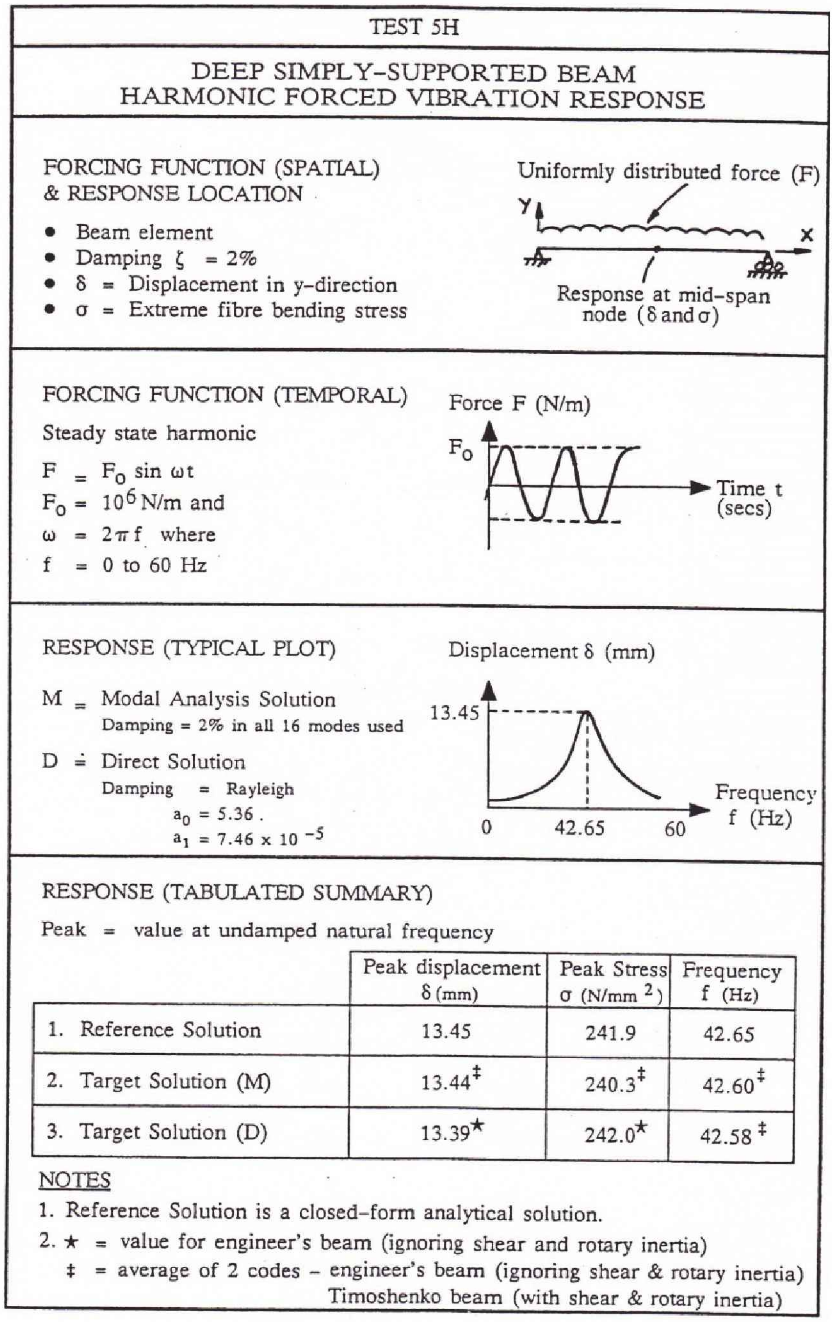
Figure 19-15: First mode of vibration of the solid, shell, and beam models.

All shapes correspond to the same first mode of vibration which is an in-plane mode. Colors are not available in modal plots in a Harmonic study.

All results closely match the exact solution of 42.710Hz.

NAFEMS TEST 5H

Deep Simply-Supported Beam; Harmonic Forced Vibration Response.



TEST 5H

DEEP SIMPLY-SUPPORTED BEAM
HARMONIC FORCED VIBRATION RESPONSE

FORCING FUNCTION (SPATIAL)
& RESPONSE LOCATION

- Beam element
- Damping ζ = 2%
- δ = Displacement in y-direction
- σ = Extreme fibre bending stress

FORCING FUNCTION (TEMPORAL)

Steady state harmonic

$F = F_0 \sin \omega t$
$F_0 = 10^6$ N/m and
$\omega = 2\pi f$ where
f = 0 to 60 Hz

RESPONSE (TYPICAL PLOT)

M = Modal Analysis Solution
Damping = 2% in all 16 modes used

D = Direct Solution
Damping = Rayleigh
$a_0 = 5.36$.
$a_1 = 7.46 \times 10^{-5}$

RESPONSE (TABULATED SUMMARY)

Peak = value at undamped natural frequency

	Peak displacement δ (mm)	Peak Stress σ (N/mm ^{2})	Frequency f (Hz)
1. Reference Solution	13.45	241.9	42.65
2. Target Solution (M)	13.44‡	240.3‡	42.60‡
3. Target Solution (D)	13.39★	242.0★	42.58 ‡

NOTES
1. Reference Solution is a closed-form analytical solution.
2. ★ = value for engineer's beam (ignoring shear and rotary inertia)
‡ = average of 2 codes – engineer's beam (ignoring shear & rotary inertia)
Timoshenko beam (with shear & rotary inertia)

Figure 19-16: NAFEMS TEST 5H

Geometry and material properties are the same as in NAFEMS TEST 5.

This test uses the same **SolidWorks** model as NAFEMS TEST 5. The model is subjected to harmonic excitation by a uniformly distributed force:

$F = 10^6 * sin\omega t$ where $\omega = 2\Pi f$ $0 < f < 60Hz$

Modal Damping is 2% in all 16 modes to be used for analysis. **Rayleigh Damping may** be used alternatively to **Modal Damping**, the constants are $\alpha = 5.36$, $\beta = 7.46*10^{-5}$.

The test target is the peak displacement in the mid-span of 13.45mm, and the peak stress is 241.9MPa for an excitation frequency of 42.65Hz.

We'll expand the already defined studies for *TEST 5 solids, TEST 5 shells and TEST 5 beams* to obtain frequency response results.

Review the **Sensors** in each model configuration and define a load as shown in Figure 19-17. Define a **Modal Damping** of 2% for all modes.

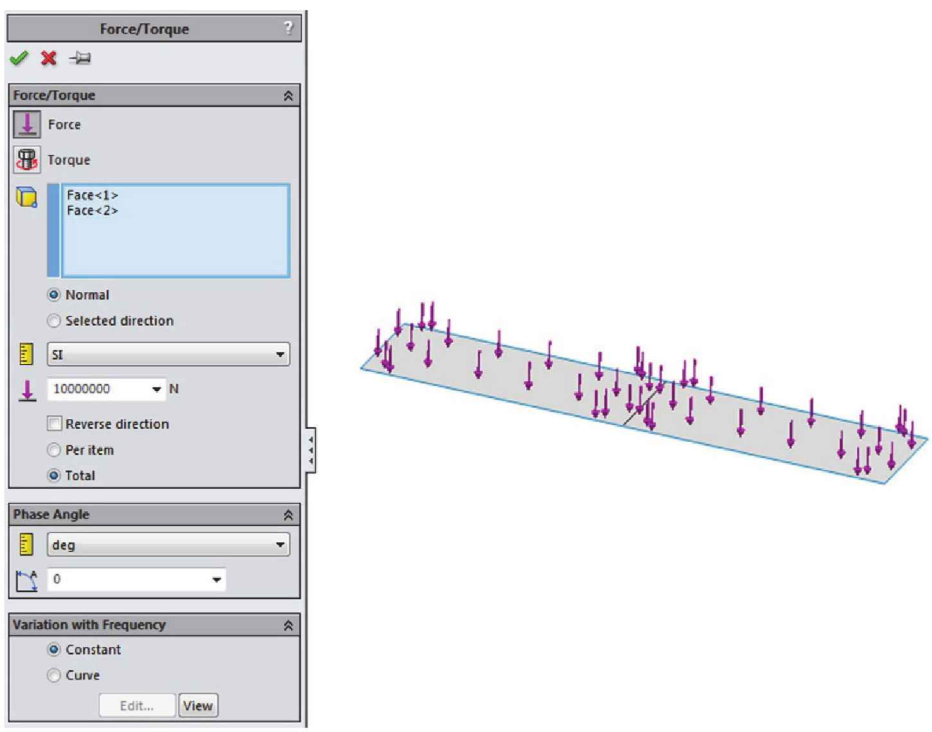
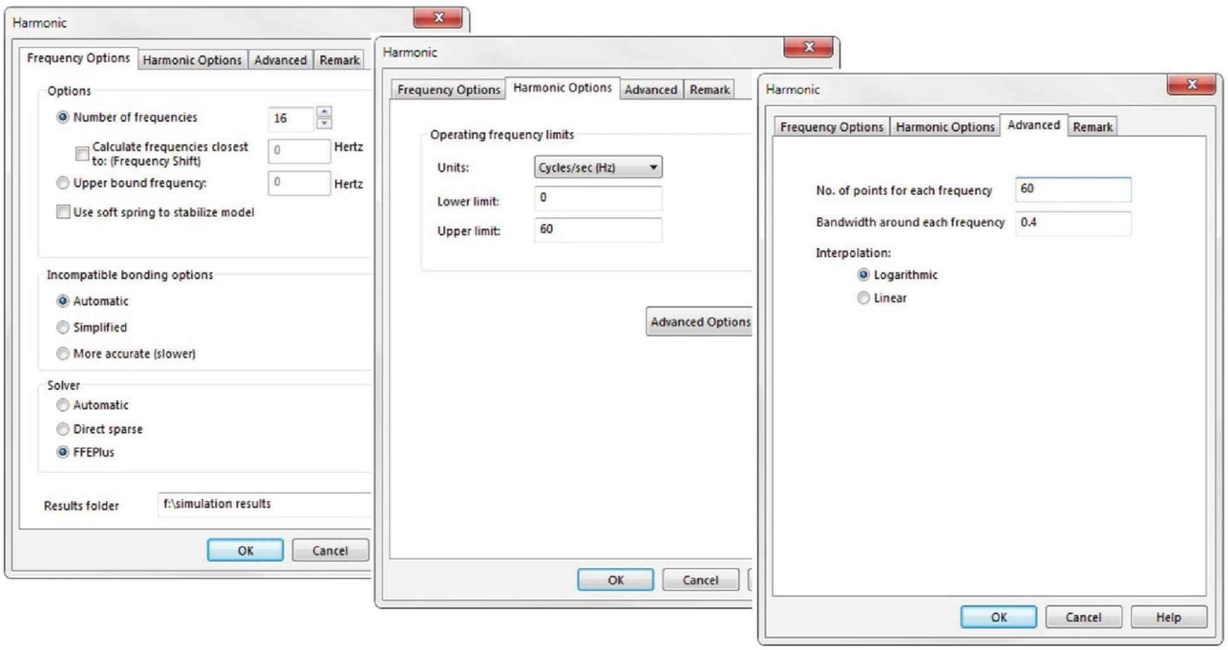


Figure 19-17: Load to be used in the Frequency Response study.

The load is shown on the surface model. Apply the same to the solid and beam models.

Define study properties as shown in Figure 19-18.



Frequency Options:
16 modes

Harmonic Options:
The range of excitation
frequencies 0-60Hz

Advanced Options:
60 points for each frequency

<u>Figure 19-18: Properties of Harmonic studies *TEST 5 solids*, *TEST 5 shells*, and *TEST 5 beams*.</u>

16 modes are specified in the Frequency Options to satisfy the NAFEMS TEST 5H requirements.

Obtain the solutions and notice that there is only one modal frequency present in the specified range of frequencies 0-60Hz (Figure 19-19).

	Frequency [Hz]		
Mode No.	Solids	Shells	Beams
1	42.3	43.2	42.1
2	143.6	152.8	77.5
3	156.6	177.1	143.0
4	266.9	251.3	155.3
5	342.4	294.6	215.4
6	390.8	350.0	232.3
7	435.7	446.9	249.9
8	476.8	488.0	270.1
9	505.9	515.9	386.5
10	524.2	601.2	407.6
11	545.5	607.1	415.3
12	598.3	646.4	437.2
13	605.5	664.8	499.1
14	617.7	672.8	539.6
15	634.4	699.0	549.1
16	634.9	754.4	549.4

Figure 19-19: Modal frequency results of the three studies: *TEST 5 solids*, *TEST 5 shells*, and *TEST 5 beams*.

In all studies there is only one mode in the range of 0-60Hz.

Observe that all solutions complete in 121 steps. This is because we specified 60 points for each frequency in the **Advanced Options** of the study properties. Considering that there is only one frequency in the range of 0-60Hz, there are 60 points on each side of the natural frequency plus one point for t = 0.

Review the sensor definitions for all configurations and construct response graphs of the UY displacement component for all studies. All graphs report very close to the results of the NAFEMS TEST 5H benchmark results: 13.45mm at a frequency of excitation of 42.65Hz. A sample graph with results of the shell element model is shown in Figure 19-20.

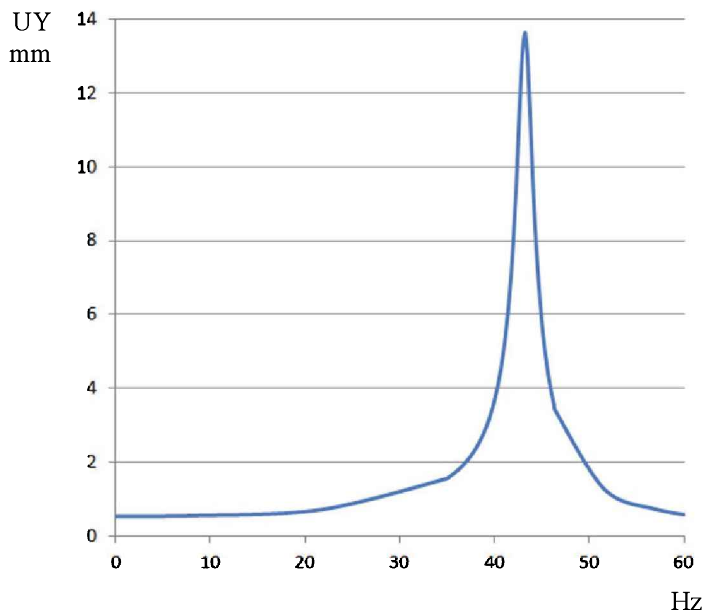


Figure 19-20: UY displacement in the mid-span of the beam as a function of the excitation frequency.

In all studies there is only one mode in the range of 0-60Hz.

Another response graph could be constructed to see how close the stress results are to the NAFEMS TEST 5H stress target of 241.9MPa. Rather than constructing a response graph we review the stress plot for frequency step 61.

NAFEMS TEST 5H does not specify what stress component should be compared to the target value. Since the state of stress is predominantly bending, we'll look at the maximum bending stress component, which is the SX stress component. Notice that the SX stress reported is the same on the bottom and top of the shell elements. This is a **Frequency Response**, not a **Time Response** analysis, and in a **Frequency Response,** the maximum responses for a given excitation frequency are reported.

Follow Figure 19-21 to construct a plot of SX stress at frequency step 61.

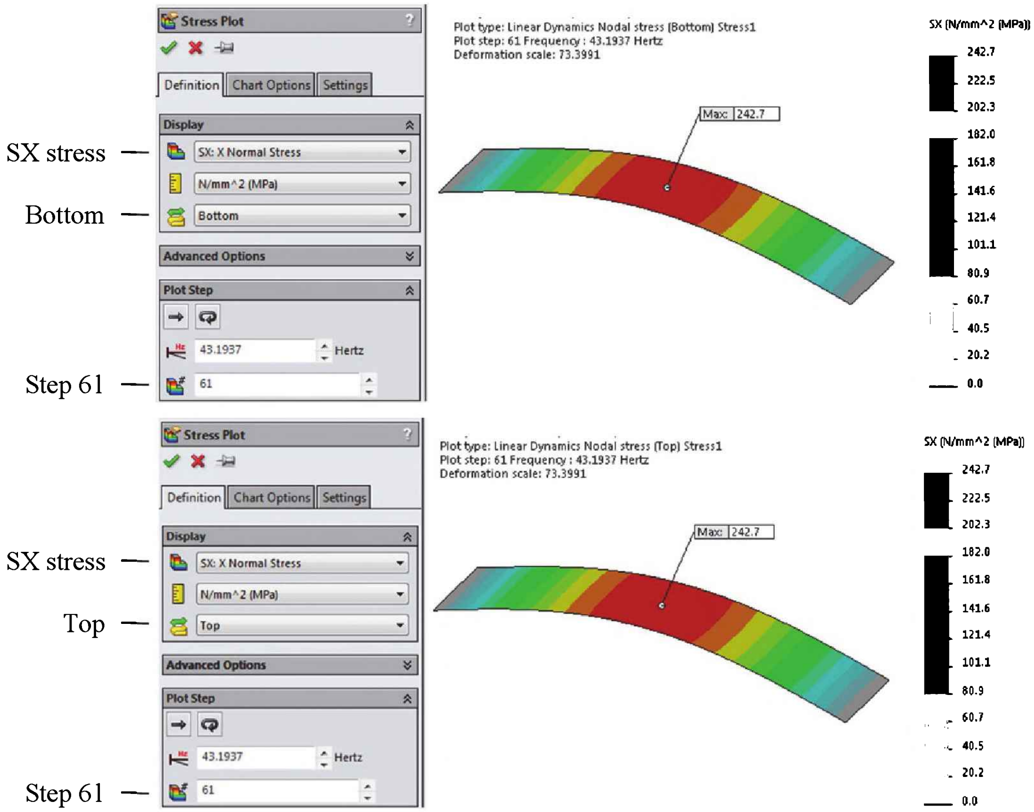
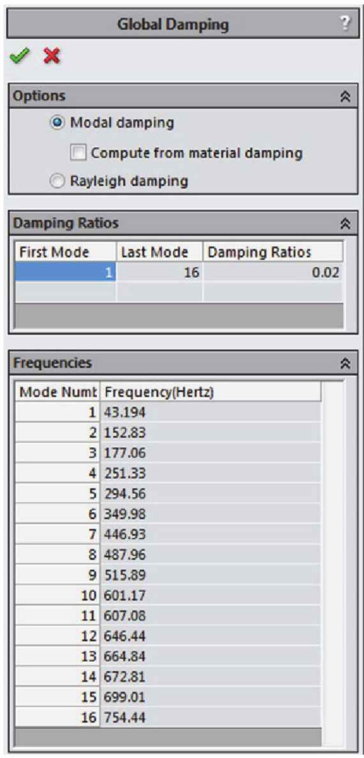
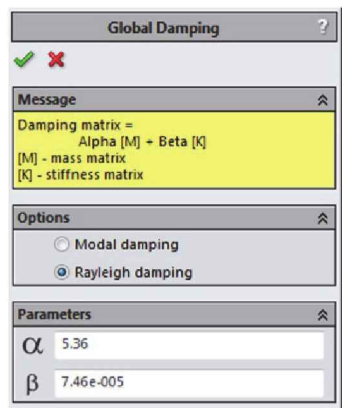


Figure 19-21: The SX stress component on the bottom and top of the shell elements at frequency step 61.

Step 61 corresponds to the natural frequency of 43.19Hz. Both plots show the same SX stress results.

NAFEMS TEST 5H specifies an alternative method of defining damping: **Rayleigh Damping** rather than **Modal Damping**. Repeat any of the completed studies with damping defined as **Rayleigh Damping** as shown in Figure 19-22. Don't forget to delete the **Modal Damping** that has been defined.





α =5.36
β =7.46*10^{-5}

Modal damping   Rayleigh damping

<u>Figure 19-22: Damping defined as Modal Damping and as Rayleigh damping.</u>

Rayleigh damping coefficients are α =5.36, β =7.46*10^{-5} *as specified in the NAFEMS TEST 5H benchmark test.*

Obtain the solution with **Rayleigh Damping** to see that results are identical to those using **Modal Damping**.

NAFEMS TEST 5T

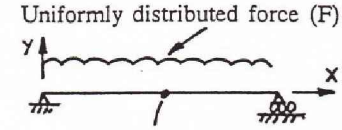
Deep Simply-Supported Beam; Transient Forced Vibration Response

TEST 5T

DEEP SIMPLY-SUPPORTED BEAM
TRANSIENT FORCED VIBRATION RESPONSE

FORCING FUNCTION (SPATIAL) & RESPONSE LOCATION

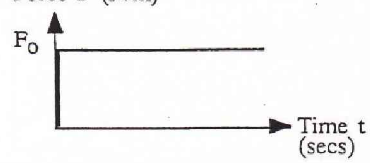
- Beam element
- Damping ζ = 2%
- δ = Displacement in y-direction
- σ = Extreme fibre bending stress



FORCING FUNCTION (TEMPORAL)

Suddenly applied step load

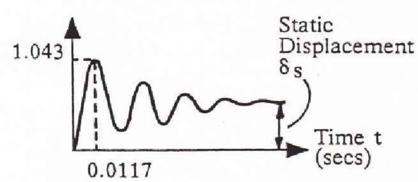
$F_0 = 10^6$ N/m



RESPONSE (TYPICAL PLOT)

M = Modal Analysis Solution
Damping = 2% in all 16 modes used
Time step = 0.0001 secs

D = Direct Solution
Damping = Rayleigh
a_0 = 5.36
a_1 = 7.46 x 10^{-5}
Time step = 0.0001 secs



RESPONSE (TABULATED SUMMARY)

Peak = value at undamped natural frequency

	Peak displacement		Peak Stress	Static displacement
	δ_p (mm)	t_p (sec)	σ (N/mm ^{2})	δ_s (mm)
1. Reference Solution	1.043	0.0117	18.76	0.538
2. Target Solution (M)	1.040‡	0.0116‡	18.71★	0.536‡
3. Target Solution (D)	1.057‡	0.0117‡	18.77‡	0.536‡

1. Reference Solution is a closed-form analytical solution.
2. ★ = value for engineer's beam (ignoring shear and rotary inertia)
   ‡ = average of 2 codes – engineer's beam (ignoring shear & rotary inertia)
   Timoshenko beam (with shear & rotary inertia)

Figure 19-23: NAFEMS TEST 5T

The model and material properties are the same as in NAFEMS TEST 5.

We'll stay with the same model NAFEMS TEST 5; benchmark test NAFEMS TEST 5T will be conducted with a **Modal Time History** study titled *TEST 5T* in only one configuration: *03 beams*. The model is subjected to a suddenly applied load $F_0 = 10^6 N/m$ as shown in Figure 19-24.

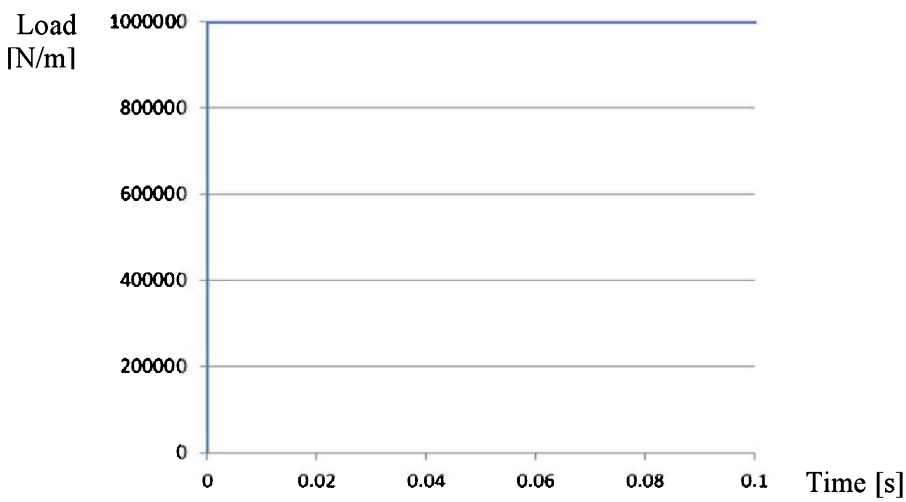


Figure 19-24: The load time history in NAFEMS TEST 5T.

Within the first 0.0001s the load increases to 100000N/m, then remains at this value for the rest of the analysis.

The **Time Step** to be used is 0.0001s; the **Modal Damping** is 2% in all 16 modes to be used for analysis. **Rayleigh Damping**, to be used alternatively to **Modal Damping**, is α =5.36, β =7.46*10^{-5}.

The test target is the displacement UY = 1.043mm in the mid-span of beam at time t = 0.0117s and static displacement ρ = 0.538mm (Figure 19-25).

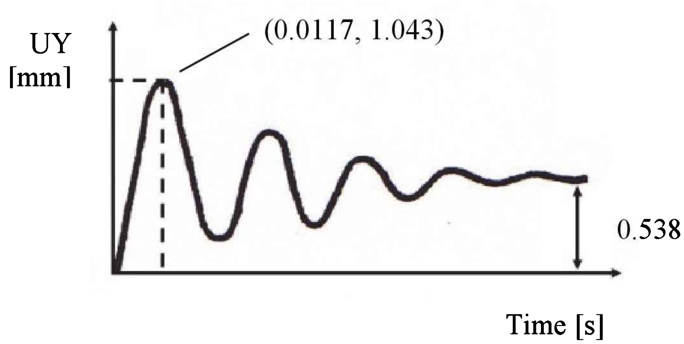
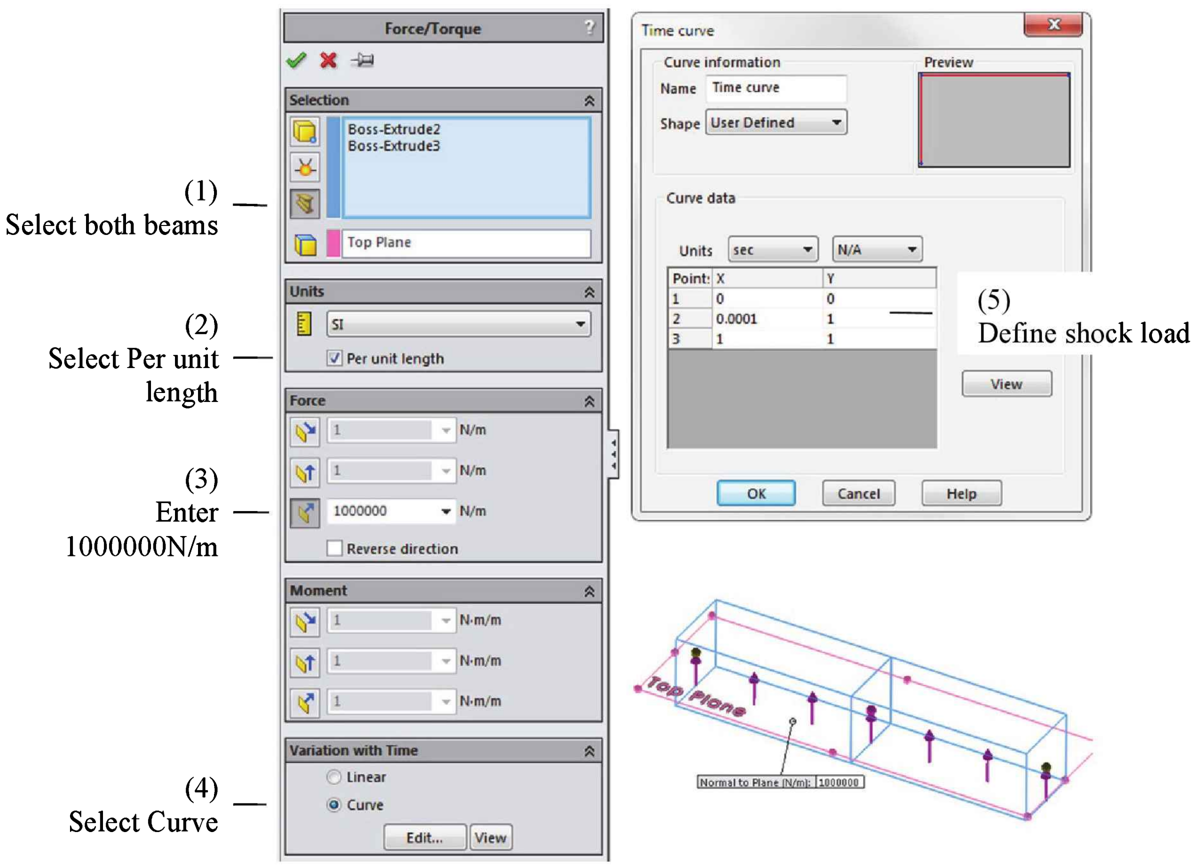


Figure 19-25: Targets in NAFEMS TEST 5T

Positive displacement is measured in the direction of the load.

The model NAFEMS TEST 5 already has three studies from the previous benchmark test NAFEMS TEST 5H. These studies are: *TEST 5 solids, TEST 5 shells,* and *TEST 5 beams*. Move to the *03 beams* configuration and create a **Modal Time History** study titled *TEST 5T.* Instruct **Simulation** to treat the two solid bodies as beams and define restraints identical to those in study *TEST 5 shells.* Go through the **Edit Joints** dialog to recalculate the joints, and apply the load as shown in Figure 19-26.



<u>Figure 19-26: Load definition in the *Test 5T* study</u>

The load is uniformly distributed over two beams. Considering the length of 10m, the total load is 10^7*N.*

Define the **Modal Time History** study properties as shown in Figure 19-27.

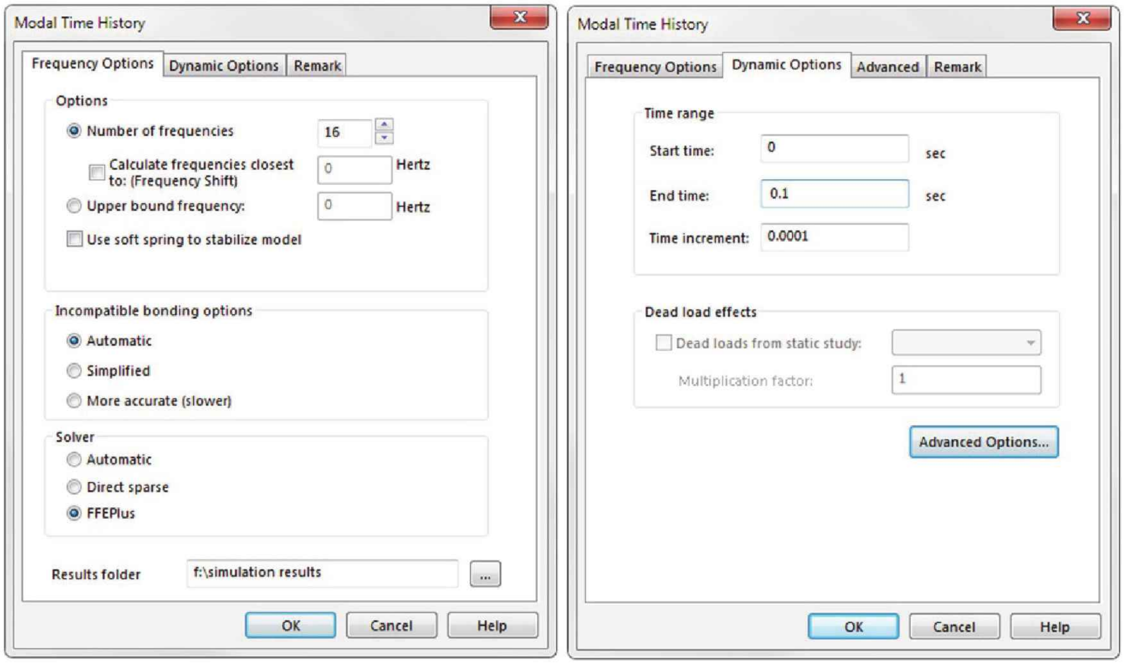


Figure 19-27: Properties of the Modal Time History (Time Response) study *TEST 5T.*

16 modes will be considered in the Modal Time History conducted in 1000 steps.

With a time duration of 0.1s and a time step of 0.0001s, the analysis will complete in 1000 steps. This is the maximum number of steps allowed in a **Response Graph**.

Define a **Modal Damping** of 2% and run the solution. Construct the UY displacement **Response Graph** as shown in Figure 19-28.

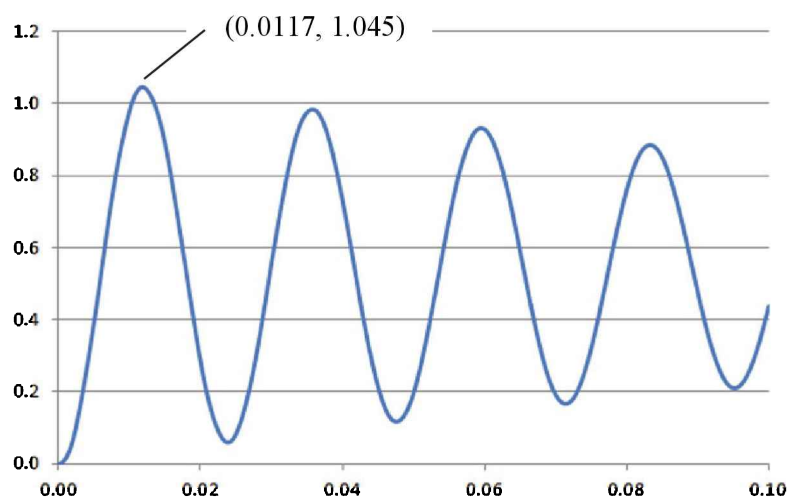


Figure 19-28: Time history of UY displacement in the mid-span of the beam during the first 0.1s.

The UY displacement at t = 0.0117s is 1.045mm, which closely matches the result of the NAFEMS TEST 5T.

Review the UY displacement time history and notice that the beam performs free damped vibration. After 0.0001s the shock load becomes a static load that causes static displacement once vibration stops. To see the static vibration we need to run the study for much longer than 0.1s.

Modify the study properties to make the total time 1s with a time step of 0.001s. This will again produce 1000 steps. The UY displacement time history during 1s is shown in Figure 19-29.

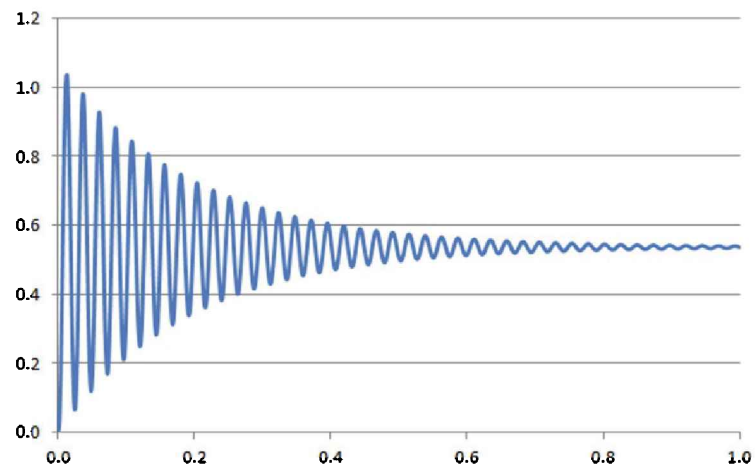


Figure 19-29: Time history of UY displacement in the mid-span of the beam during 1s.

Once the shock load ends, the beam performs free, damped vibration.

NAFEMS TEST 13

Simply-Supported Thin Square Plate

TEST 13

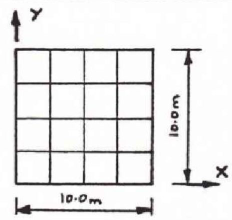
SIMPLY-SUPPORTED THIN SQUARE PLATE

GEOMETRY & MESH

$t = 0.05$ m

H.O.E. : 4 x 4 (as shown)

L.O.E. : 8 x 8



ATTRIBUTES: (a) Well established
(b) Repeated eigenvalues

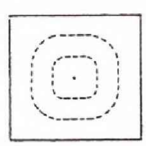
BOUNDARY CONDITIONS: $x = y = Rz$ at all nodes, $z = 0$ along all 4 edges
$Rx = 0$ along edges $x = 0$ & $x = 10$m, $Ry = 0$ along edges $y = 0$ & $y = 10$m

MATERIAL PROPERTIES: $E = 200 \times 10^9$ N/m^2, $\nu = 0.3$, $\rho = 8000$ kg/m^3

ELEMENT TYPES: H.O.E.: 8-noded semi-loof thin shell element
L.O.E.: 4-noded iso-parametric shell element

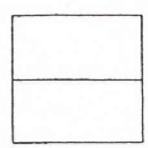
FREQUENCIES & MODE SHAPES (* = closed form solution)

$f_r^* = 2.377$ Hz
$f_t = 2.378$ Hz [HOE]
$= 2.418$ Hz [LOE]



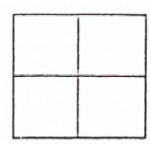
Mode 1

$f_r^* = 5.942$ Hz
$f_t = 5.907$ Hz [HOE]
$= 6.332$ Hz [LOE]



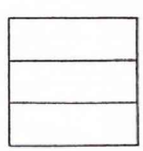
Modes 2 & 3

$f_r^* = 9.507$ Hz
$f_t = 9.588$ Hz [HOE]
$= 10.192$ Hz [LOE]



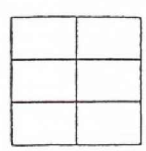
Mode 4

$f_r^* = 11.884$ Hz
$f_t = 11.699$ Hz [HOE]
$= 13.991$ Hz [LOE]



Modes 5 & 6

$f_r^* = 15.449$ Hz
$f_t = 15.416$ Hz [HOE]
$= 17.748$ Hz [LOE]



Modes 7 & 8

Figure 19-30: NAFEMS TEST 13

Carefully review the boundary conditions.

Open part model NAFEMS TEST 13 and notice a small triangular split face in the center. It is there to locate a sensor in the center of the plate. Materials properties require a custom material – the same as the one used in the previous tests.

Create a **Frequency** study titled *TEST 13*; specify eight modes in the study properties.

Define restraints to all four edges in the direction normal to the plate (Figure 19-31).

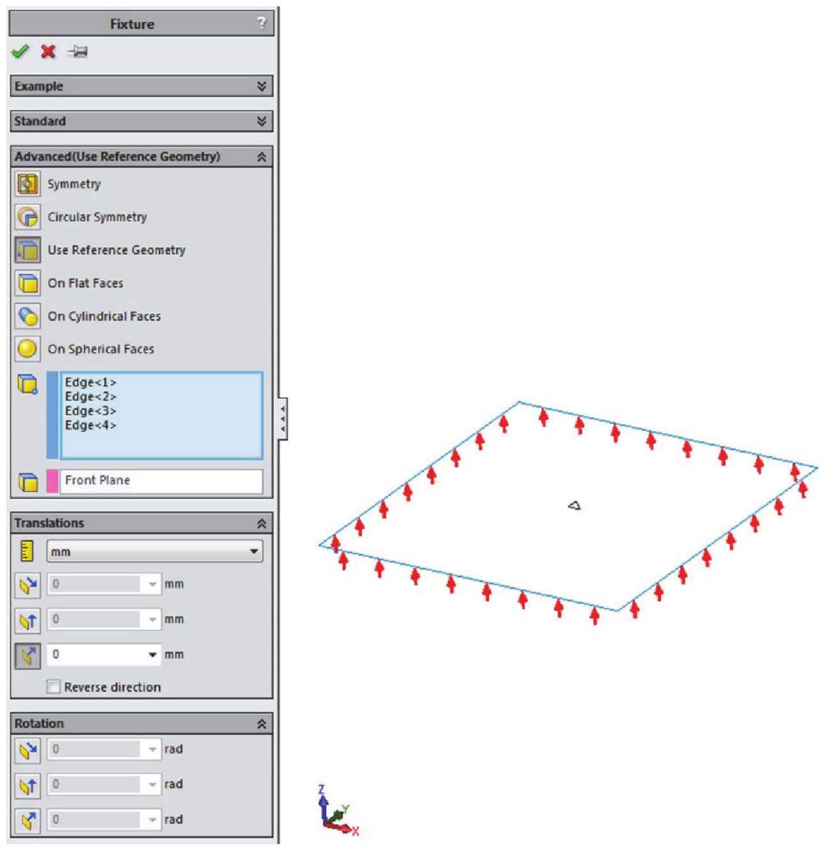


Figure 19-31: Restraints to all four edges.

In this example, the Front Plane is used to define the direction of the restraints.

Define restraints to the two edges parallel to the X axis as shown in Figure 19-32.

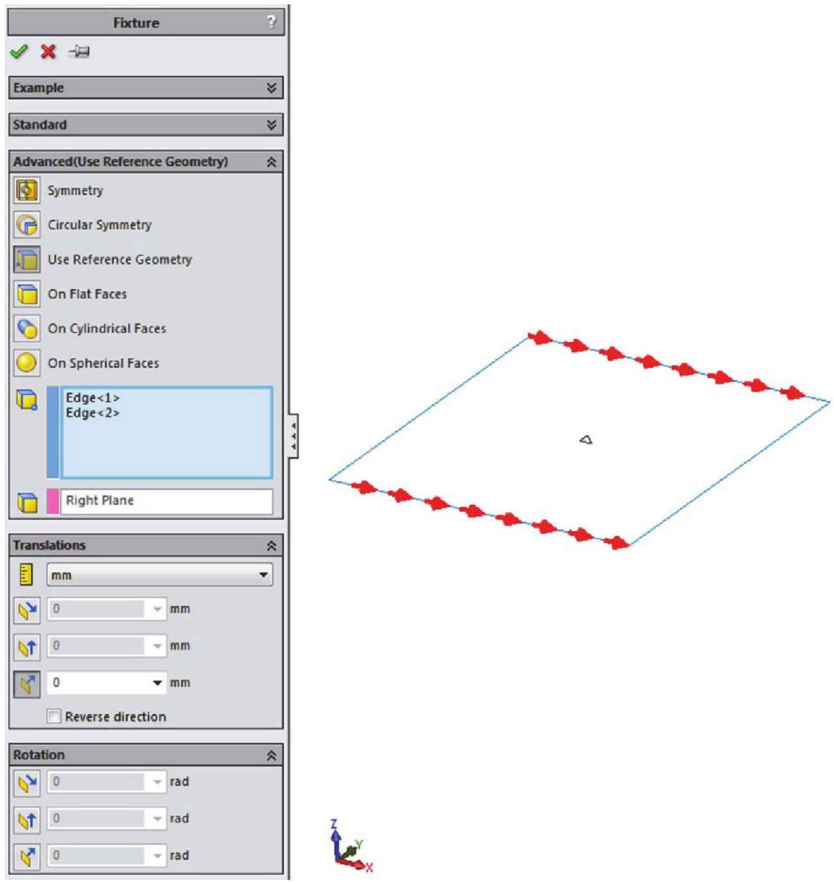


Figure 19-32: Restraints to the two edges parallel to the X axis.

The Right Plane is used here as reference geometry.

Define restraints to the two edges parallel to the Y axis as shown in Figure 19-33.

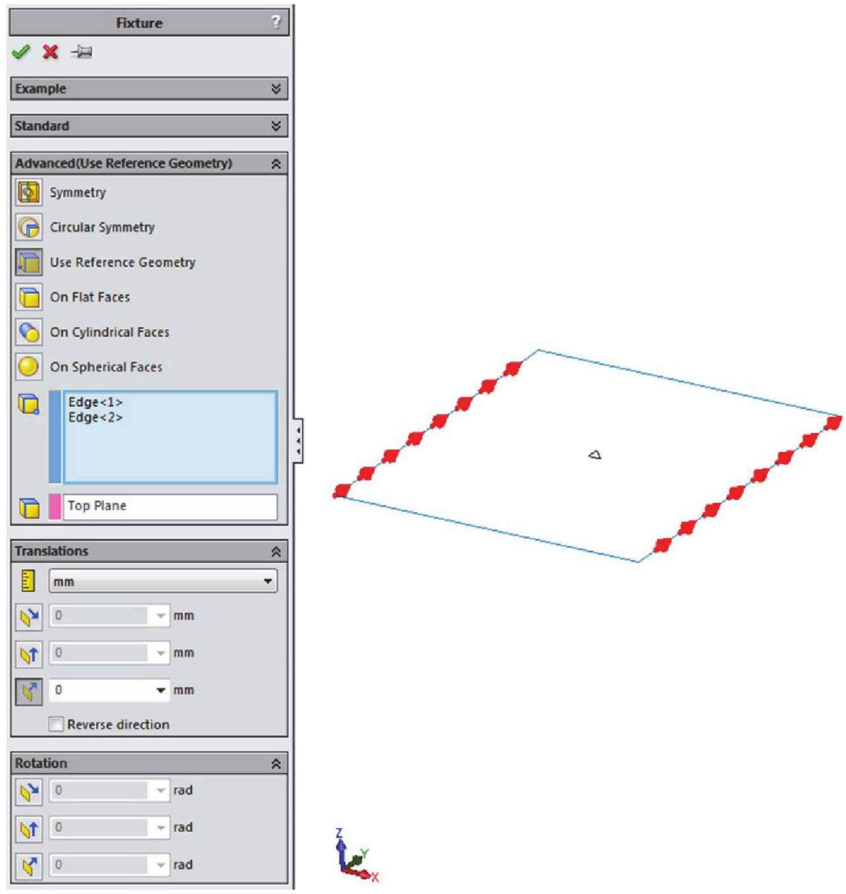


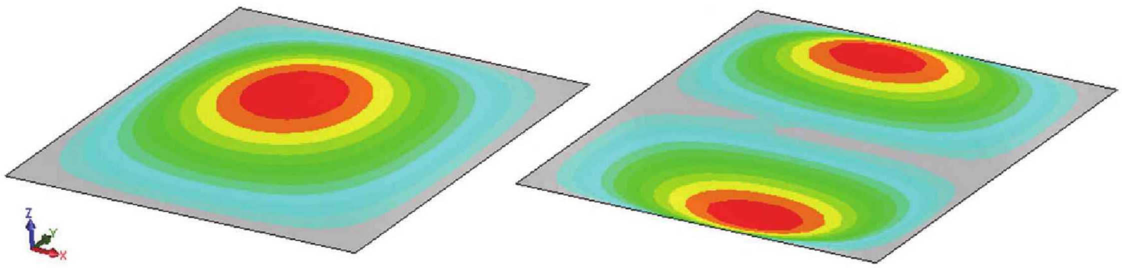
Figure 19-33: Restraints to the two edges parallel to the Y axis.

The Right Plane is used here as reference geometry.

Restraints shown in Figures 19-31, 19-32, and 19-33, fully restrain the model. There is no need to apply x = y = Rz = 0 at all nodes as specified in the test.

Define the shell thickness as 50mm and mesh the model with the default element size.

The first two modes of vibration are shown in Figure 19-34.



Mode 1: 2.376Hz   Mode 2: 5.924Hz

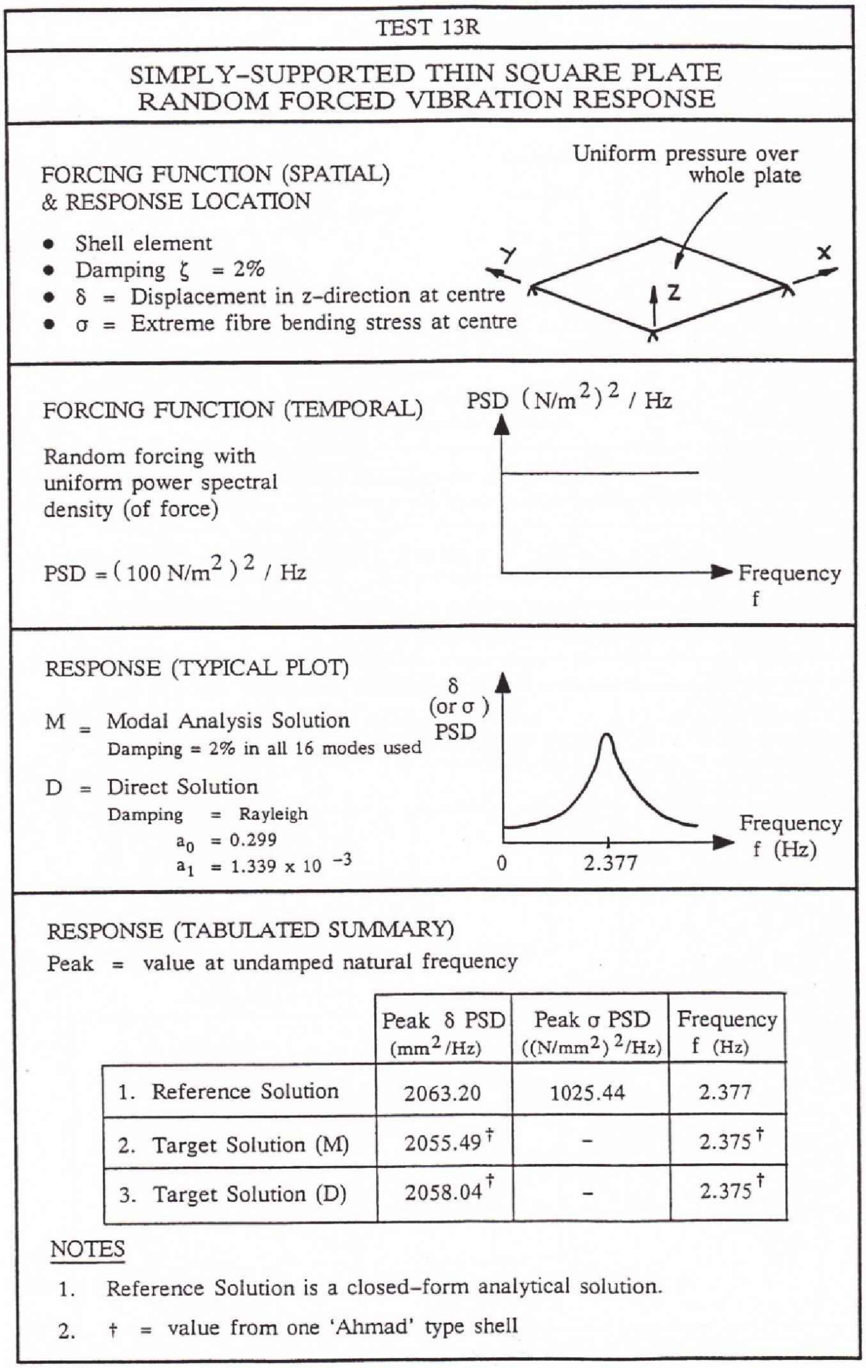
Figure 19-34: The first two modes of vibration and their frequencies.

Mode 1 is symmetric, mode 2 is anti-symmetric.

The results of the modal analysis closely match the results of NAFEMS TEST 13.

NAFEMS TEST 13R

Simply-Supported Thin Square Plate, Random Forced Vibration Response. This study uses the NAFEMS TEST 13 model.



TEST 13R

SIMPLY-SUPPORTED THIN SQUARE PLATE
RANDOM FORCED VIBRATION RESPONSE

FORCING FUNCTION (SPATIAL) & RESPONSE LOCATION

- Shell element
- Damping ζ = 2%
- δ = Displacement in z-direction at centre
- σ = Extreme fibre bending stress at centre

FORCING FUNCTION (TEMPORAL)

Random forcing with uniform power spectral density (of force)

PSD = $(100\ N/m^2)^2$ / Hz

RESPONSE (TYPICAL PLOT)

M = Modal Analysis Solution
Damping = 2% in all 16 modes used

D = Direct Solution
Damping = Rayleigh
a_0 = 0.299
a_1 = 1.339×10^{-3}

RESPONSE (TABULATED SUMMARY)

Peak = value at undamped natural frequency

	Peak δ PSD (mm^2/Hz)	Peak σ PSD ($(N/mm^2)^2$/Hz)	Frequency f (Hz)
1. Reference Solution	2063.20	1025.44	2.377
2. Target Solution (M)	2055.49†	-	2.375†
3. Target Solution (D)	2058.04†	-	2.375†

NOTES

1. Reference Solution is a closed-form analytical solution.
2. † = value from one 'Ahmad' type shell

Figure 19-35: NAFEMS TEST 13R

The model geometry and restraints are identical to study TEST 13

Create a **Random** study titled *TEST 13R* and copy all restraints from the *TEST 13* study.

The load is a random force with uniform Power Spectral Density (white noise).

The force is a pressure; $p = (100N/m^2)^2/Hz$. Define the load as shown in Figure 19-36. Remember that the pressure must be entered in units of $(N/m^2)^2/Hz$. Therefore, its numerical value is; $p = (100N/m^2)^2/Hz = 100^2\ (N/m^2)^2/Hz = 10000\ (N/m^2)^2/Hz$. The use of incorrect units is a common source of error in random vibration analyses.

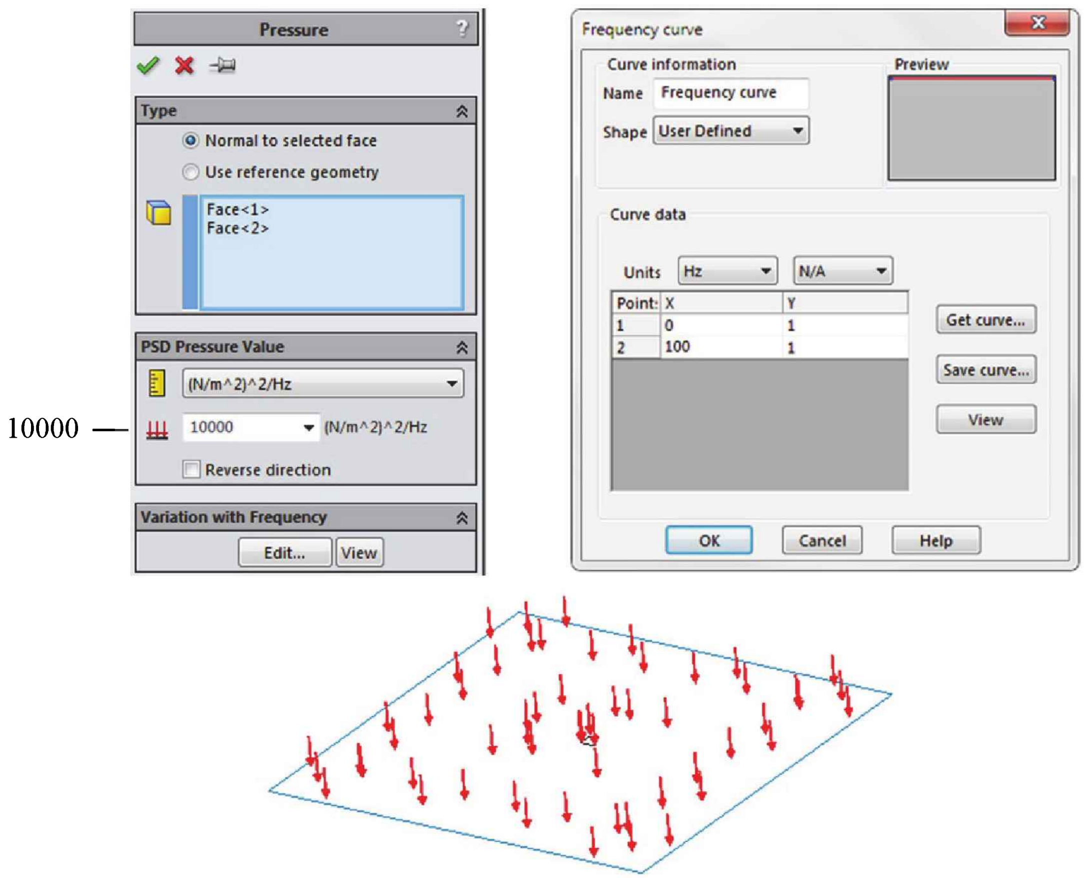


Figure 19-36: NAFEMS TEST 13R

The frequency curve window defines a uniform PSD curve in the frequency range of 0-100Hz.

Define a **Modal Damping** of 2%.

Define a **Random** study with properties as shown in Figure 19-37.

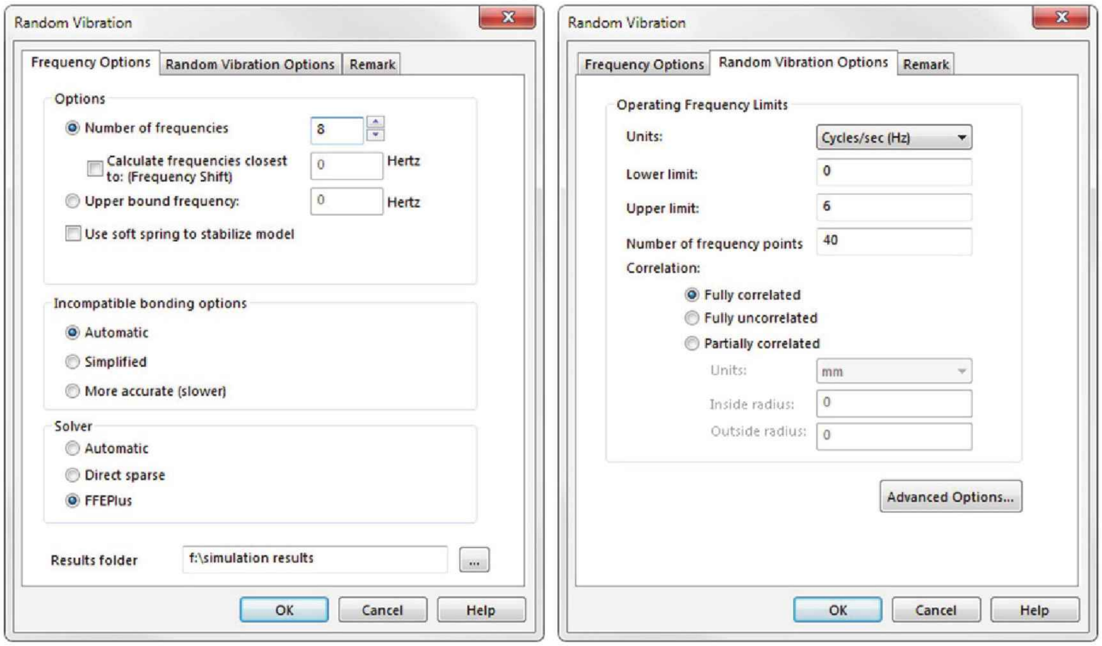


Figure 19-37: Properties of study *TEST 13R*

40 frequency points are specified in the Random Vibration Options to have smooth graphs.

Define the **Result Options** using the Sensor in the center of plate and obtain the solution.

Construct a **Response Graph** of the PSD displacement (Figure 19-38).

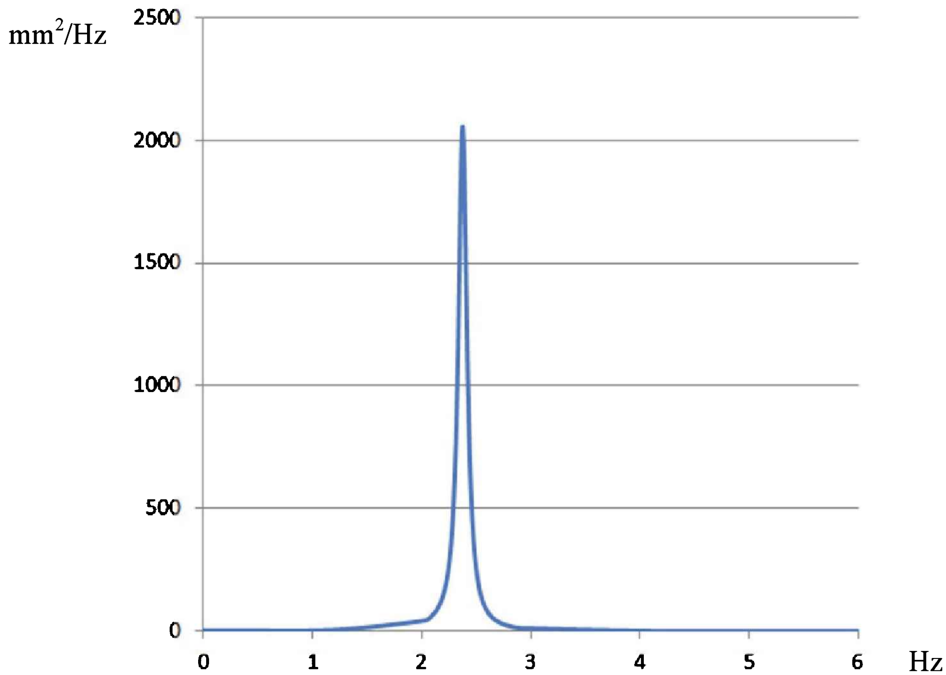


Figure 19-38: The PSD displacement response graph.

The first mode dominates the vibration response in the frequency range of 0-6Hz.

Review the graph and notice that the maximum PSD displacement closely matches the NAFEMS TEST 13R result of 2063mm^2/Hz.

20: Glossary of terms

Critical Damping

System damping above which oscillations are not possible

Degrees of Freedom

Independent variables describing position (linear or angular) of a rigid body or of a node in a finite element mesh.

Discrete Vibrating System

System components responsible for inertial, damping and elastic properties are separated. It takes form of an assembly of rigid bodies connected by springs and dampers.

Distributed Vibrating System

System components responsible for inertial, damping and elastic properties are not separated. It takes form of an assembly of elastic bodies.

Dynamic Analysis

Analysis of motion of rigid bodies and flexible bodies

Flexible body

Body that deforms under applied loads. Flexible body must have elastic properties defined.

Frequency Analysis

SolidWorks Simulation term used to denote Modal Analysis.

Frequency Response Analysis

Analysis of vibration caused by loads defined as function of frequency rather than of time.

Impulse load

Load of a short duration defined as a function of time.

Harmonic Analysis

SolidWorks Simulation term used to denote Frequency Response Analysis

Kinematic Pair

Connection between two bodies that imposes constraints on relative movements of the bodies.

Linear Damping

Damping in linear motion

Mechanism

System of rigid and/or elastic bodies connected by kinematic pairs, working together in a machine.

Modal Analysis

Finding modes of vibration of a vibrating system.

Modal Damping

% of critical damping present in a Single Degree of Freedom System

Modal Superposition Method

Method that represents a vibrating system as a collection of Single Degree of Freedom Oscillators (SDOF); each SDOF represents the system vibrating in a given mode of vibration. The vibration response of the system is found as a superposition of vibration responses of those individual SDOFs.

Mode of Vibration

Certain combination of frequency and shape of vibration for which cancellation between inertial and elastic effects takes place.

Modal Time History

SolidWorks Simulation term used to denote Time Response Analysis

Random Vibration Analysis

Analysis of vibration due to stationary, random excitation given in the form of the Power Spectral Density.

Rayleigh Damping

Damping used in nonlinear vibration analysis; damping matrix is expressed as a linear combination of mass and stiffness matrices.

Response Spectrum Analysis

Type of vibration analysis used to analyze vibration caused by non-stationary excitation such as a seismic shock.

Rigid Body Mode

Mode of vibration of a structure without support or with a partial support. Program assigns 0Hz frequency to modes corresponding to rigid body displacements.

Rigid Body Motion

Translation and/or rotation of a body without deformation.

Single Degree of Freedom Oscillator

A mass (rigid body) connected to a base with a spring (and sometimes damper). It performs linear or angular oscillations and its motions is fully described by one unknown.

Structure

An elastic body capable of supporting loads. Structure is fully supported and it can't move without deforming. Movement of structure takes form of oscillations about the position of equilibrium.

Time Response Analysis

Analysis of vibration caused by loads defined as function time.

Vibration Analysis

Analysis of motion of a deformable (elastic) body or system about an equilibrium point.

Viscous Damping

Damping force is proportional to velocity.

21: References

This textbook builds on an understanding of structural analysis with the Finite Element Method and assumes familiarity with structural and introductory modal analysis to the extent covered in the introductory textbook: Kurowski P. "Engineering Analysis with SolidWorks Simulation 2014", SDC Publications 2014

For a review of the topics discussed in this book, readers are referred to the following literature listed here in the order of relevance:

1. Kurowski P. "Engineering Analysis with SolidWorks Simulation 2014", SDC Publications 2014

   www.SDCpublications.com/Textbooks/Engineering-Analysis-SolidWorks-Simulation-2014/ISBN/978-1-58503-858-9/

2. Inman D., "Engineering Vibration" Prentice Hall 2007
3. The Standard NAFEMS Benchmarks, NAFEMS 1990
4. Selected Benchmarks for Forced Vibration, NAFEMS 1989
5. How to Do Seismic Analysis Using Finite Elements NAFEMS 2007
6. How to undertake Finite Element Based Vibration Analysis NAFEMS 2002
7. Vibrationdata (www.vibrationdata.com)
8. Logan D. "A First Course in the Finite Element Method", Brooks/Cole 2007
9. Adams V., Askenazi A. "Building Better Products with Finite Element Analysis", OnWord Press, 1998.
10. Kim N.H., Sankar B.V. "Introduction to Finite Element Analysis and Design", John Wiley & Sons, Inc., 2009.

The full list of NAFEMS publications can be found at www.nafems.org. Another internet site with a number of FEA related publications is presented by Design Generator Inc. Publications related to FEA fundamentals, training, and implementation can be found at:
www.designgenerator.com/publications.htm

22: List of exercises

Chapter	Part	Assembly	Spreadsheets Animations Text files
Cover	SPIDER2014*		SPIDER.avi
1	CLIP*	PLANAR PRISMATIC REVOLUTE CYLINDRICAL SPHERICAL SCREW DISCRETE LINEAR* SWING ARM* DOUBLE PENDULUM ELLIPTIC TRAMMEL WRECKING BALL**	
2	BALL VALVE PLATE		
3	NOTCH		
4	ROTOR COLUMN	COLUMN***	
5	U BRACKET		
6	BEAM DEMO*	SHAFT	SHAFT.txt
7	CAR		
8	VASE		
9	BAJA FRAME		
10	FLAT	PLIERS	
11		MDOF	MDOF.xlsx MDOF IMPULSE.xlsx
12	ELBOW PIPE	SHAKER TABLE***	ELBOW PIPE.xlsx
13		CENTRIFUGE	CENTRIFUGE.xlsx
14	SUPPORT		
15		VIB ABSORBER	
16		HD HEAD	HD HEAD.xlsx
17		FRAME	FRAME.xlsx

18	PLANK		PLANK.xlsx PLANK RAYLEIGH.xlsx
19	NAFEMS TEST FV2 NAFEMS TEST 5 NAFEMS TEST 13		

* Includes Simulation study ready to run
** Includes Motion study ready to animate
*** For illustration only

Vibration Analysis with SolidWorks® Simulation 2014

- Introduces you to both vibration analysis and its implementation in SolidWorks Simulation 2014
- Covers all types of vibration analysis available in SolidWorks Simulation
- Uses hands on exercises that build on one another throughout the book
- Designed for users already familiar with Finite Element Analysis using SolidWorks Simulation

Description

Vibration Analysis with SolidWorks Simulation 2014 goes beyond the standard software manual. It concurrently introduces the reader to vibration analysis and its implementation in SolidWorks Simulation using hands-on exercises. A number of projects are presented to illustrate vibration analysis and related topics. Each chapter is designed to build on the skills and understanding gained from previous exercises.

Vibration Analysis with SolidWorks Simulation 2014 is designed for users who are already familiar with the basics of Finite Element Analysis (FEA) using SolidWorks Simulation or who have completed the book Engineering Analysis with SolidWorks Simulation 2014. Vibration Analysis with SolidWorks Simulation 2014 builds on these topics in the area of vibration analysis. Some understanding of structural analysis and solid mechanics is recommended.

Topics Covered

- Differences between rigid and elastic bodies
- Discrete and distributed vibration systems
- Modal analysis and its applications
- Modal Superposition Method
- Modal Time History (Time Response) analysis
- Harmonic (Frequency Response) analysis
- Random Vibration analysis
- Response Spectrum analysis
- Nonlinear Vibration analysis
- Modeling techniques in vibration analysis

Table of Contents

SDC PUBLICATIONS

Better Textbooks. Lower Prices.
www.SDCpublications.com

Suggested Retail Prices
Retail Bookstores/Internet Vendors: $75
College & Universities: $48

ISBN: 978-1-58503-910-4

9 781585 039104