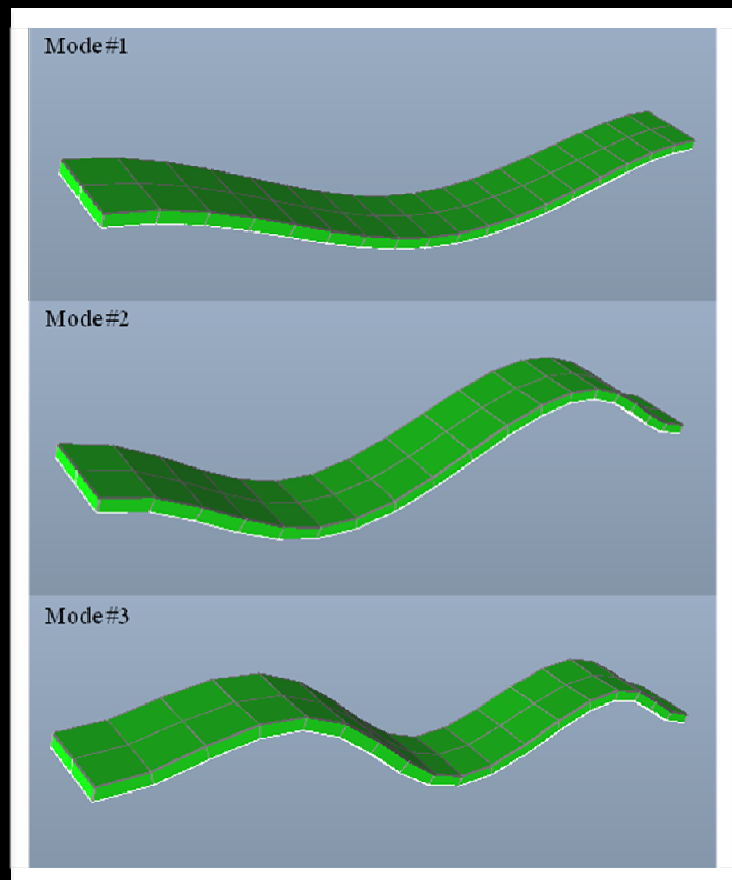


# Application Note

## Sub-Harmonic and Coupled Resonance Simulation



Application Note: Sub-Harmonic and Coupled Resonance Simulation  
Version 8.6/PC

Part Number 30-090-101  
July 2009

© Copyright IntelliSense Software Corporation 2004, 2005, 2006, 2007, 2008, 2009  
All Rights Reserved.

Printed in the United States of America

This manual and the software described within it are the copyright of IntelliSense Software Corporation, with all rights reserved.

#### Restricted Rights Legend

Under the copyright laws, neither this manual nor the software that it describes may be copied, in whole or in part, without the written consent of IntelliSense Software Corporation. Use, duplication or disclosure of the Programs is subject to restrictions stated in your software license agreement with IntelliSense Software Corporation.

Although due effort has been made to present clear and accurate information, IntelliSense Software Corporation disclaims all warranties with respect to the software or manual, including without limitation warranties of merchantability and fitness for a particular purpose, either expressed or implied. The information in this documentation is subject to change without notice.

In no event will IntelliSense Software Corporation be liable for direct, indirect, special, incidental, or consequential damages resulting from use of the software or the documentation.

IntelliSuite™ is a trademark of IntelliSense Software Corporation.  
Windows NT is a trademark of Microsoft Corporation.  
Windows 2000 is a trademark of Microsoft Corporation.

Patent Number 6,116,766: Fabrication Based Computer Aided Design System Using Virtual Fabrication Techniques  
Patent Number 6,157,900: Knowledge Based System and Method for Determining Material Properties from Fabrication and Operating Parameters





# Table of Contents

<b>Introduction</b>	<b>1</b>
Background	1
Sub-harmonic resonance	1
Coupled resonance	2
<b>ThermoElectroMechanical (TEM) Analysis</b>	<b>3</b>
Model generation in 3D Builder	3
Beam creation	3
Meshing the structure	5
Exporting to TEM	7
Setting simulation conditions	9
Applying material properties	9
Applying boundary conditions	10
Load application	12
Frequency Analysis	12
Sub-harmonic resonance analysis	17
Load application	17
Mesh refinement	18
Simulation and results	19
Coupled resonance analysis	25
<b>Summary</b>	<b>27</b>



# 1. Introduction

In this application note, we will discuss the sub-harmonic analysis and coupled-mode vibration simulation features in IntelliSuite, by using a clamped-clamped beam case as an example. First, we will use 3D Builder to create the 3D beam model, mesh the beam and export the model to ThermoElectroMechanical (TEM) Analysis Module for finite element analysis (FEA).

## 1.1. Background

Vibration analysis is one of the most common analyses needed for both large-size and small-size mechanical systems. Mechanical vibrations are commonly classified into two types: free vibration and forced vibration.

Free vibration refers to the dynamics of a system after the application of an initial force or load. For instance, a pendulum that swings around the center of a clock is vibrating freely. Taking the damping factor into account, however, the vibration will decay due to energy dissipation. Another example is pulling a guitar string away from its original position and letting it go. The pulling action is the initial input, which stores kinetic energy in the mechanical system. With no effects from external forcers or the systematic damping, the energy will conserve and the vibration will go on. For free vibrations, frequency analysis can be performed to determine the multiple vibration modes of a mechanical system. The frequencies of the vibration modes are called natural frequencies. Awareness of the frequency differences between multiple vibration modes within a mechanical system helps design mechanical structures. Interferences from undesired vibration modes can lead to unexpected results, and are usually avoided during the design phase.

Forced vibration refers to the dynamics of a system with one or more periodic loads applied on it. Periodic motions that passengers feel in a moving car often arise from the engine vibration or external road interaction acting upon the car. The alternating external action from the engine or road forces the car and passengers to move periodically, and the motion of the car and passengers is forced vibration. Note the external loads do not have to be forces, they can also be motions. Frequency response determines how a mechanical system responds to an external force. A mechanical system is not an ideal brick-wall mechanical filter, and it responds to mechanical loads at undesired frequencies. Frequency response shows how fast the vibration magnitude decays at different frequencies.

## 1.2. Sub-harmonic resonance

In a linear system, the frequency of the output signal is the same as the input signal. When a linear system is subject to a disturbance input that contains multiple frequency components, the output may have a large magnitude at frequencies that are multiples of the fundamental frequency of the disturbance. But large magnitudes at frequencies lower (e.g.,  $\frac{1}{2}$  or  $\frac{1}{3}$ ) than the fundamental frequency of the disturbance will not be excited in a linear system.

In a non-linear system, however, higher frequencies may excite lower frequency modes. For example, when a non-linear system with a natural resonant frequency of  $f$  experiences a disturbance at a frequency that is  $n$  times the natural frequency, ie,  $nf$ , the disturbance input will excite the system at the frequency of  $nf$ , because the free motion generally contains all the higher order frequency harmonics. Since this harmonic at  $nf$  is part of the whole motion, it may act upon other modes so that the other harmonics may show up along with it. Thus, a large magnitude of vibration may occur at a frequency lower than the disturbing frequency. This phenomenon is known as “sub-harmonic resonance” or “frequency demultiplication”.

In IntelliSuite, a mechanical system can be considered a linear or non-linear system, depending on the specific application case. For a non-linear system, its non-linear response can be obtained after non-linear system analysis is performed.

### **1.3. Coupled resonance**

Mechanical systems are often subject to more than one external force at more than one frequency simultaneously. The multiple external forces may be direct loads applied to the system. Or one or more of the external forces may be coupled into the system from its ambient environment by an internal element. In either case, the mechanical system could vibrate at multiple frequencies, and so the coupled resonance response is of interest.

IntelliSuite allows application of multiple loads on a mechanical system and simulation of the coupled resonance response.



## 2. ThermoElectroMechanical (TEM) Analysis

The ThermoElectroMechanical Analysis Module (TEM) is a device level Finite Element Analysis application developed by IntelliSense. This module allows you to incorporate material properties, boundaries, and loads to fully analyze a device in the time and frequency domains. One additional feature that is unique in IntelliSuite's TEM is the ability to create N-DOF System Models from Finite Element Models that can be imported into a system level simulator like SYNPLE. These models retain the accuracy of their Finite Element based models, but can be quickly simulated under multiple loading conditions in the system level solver. Another capability of this feature is that it enables co-simulation of your CMOS control circuitry with your Finite Element based MEMS device. This is a very powerful capability only offered by IntelliSense that allows users to develop their device from layout through processing and device level development all the way to the full integration of the MEMS device with their circuit without having to switch to other CAD tools.

In this section we will discuss the device level simulation of a clamped-clamped beam. We will begin by creating the mechanical model in 3D Builder. Then we will set simulation conditions for the model. Finally, we will run Dynamic simulations to investigate linear, non-linear and coupled resonance responses of the beam.

### 2.1. Model generation in 3D Builder

The clamped-clamped beam on which we will perform analysis is shown as in Figure 1. The mechanical structure consists of only one single layer (green). The beam will be fixed at both ends. The mechanical vibration modes of the beam are of interest, as well as the frequency responses as a non-linear system with loads at different frequencies. In this section, we will:

- i. Construct the mechanical beam model
- ii. Mesh the mechanical structure for finite element analysis
- iii. Export the model to the TEM module



*Figure 1 Clamped-clamped beam*

#### 2.1.1 Beam creation

Open the 3D Builder module, as shown in Figure 2. Maximize the window.

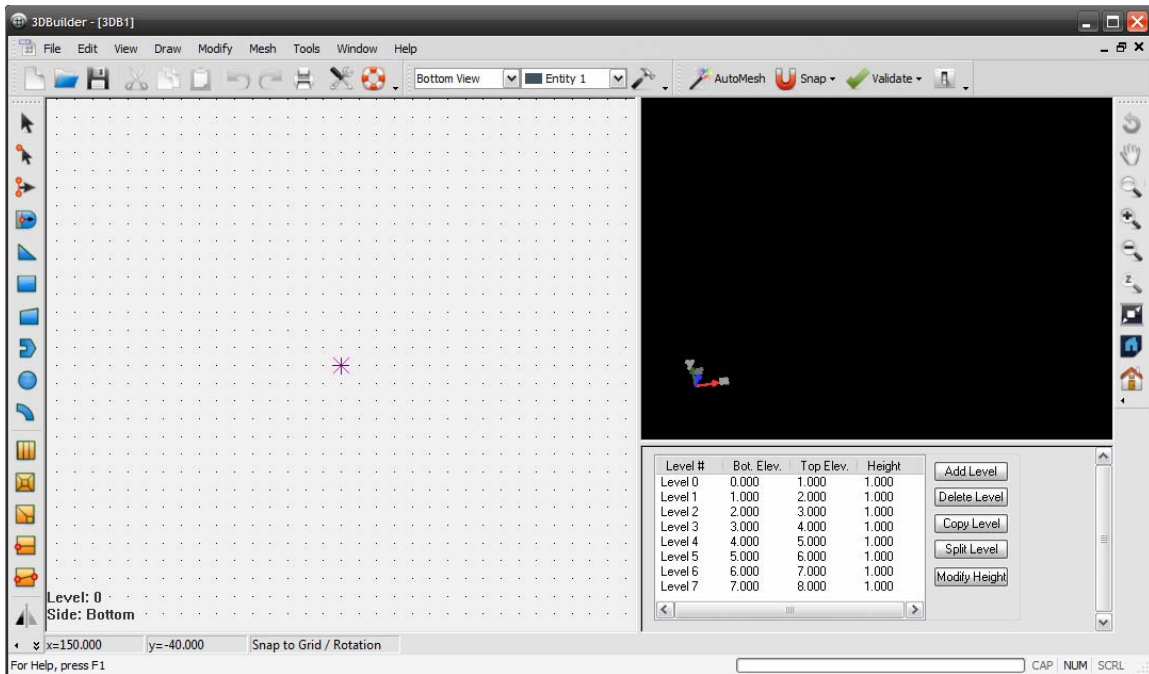


Figure 2 3D Builder window

Default levels have been pre-assigned in the 3D Builder, and each level is automatically assigned a default height of 1  $\mu\text{m}$ . Level 0 for the clamped-clamped beam should be 3  $\mu\text{m}$  thick, so we must adjust the height. Click on “Level 0” in the level window. The level window is the rectangular area located in the lower right-hand side of the 3D Builder window.

**Click** *Modify Height*

Enter a height of 3  $\mu\text{m}$  as shown in Figure 3.

Note: At the bottom of the window, the third box from the left, "Grid/Rotation" is displayed. This means that *Snap to Grid* is active (the default). At the lower left corner of the window, you will see the coordinates of the current location of the pointer. Since *Snap to Grid* is on, this value will be multiples of 10  $\mu\text{m}$ .

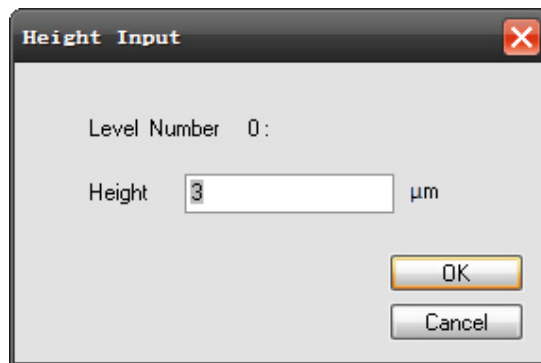



Figure 3 Modify Height

Click once in the 2D area to activate the window.

Click the Add Rectangle toolbar button: 

To create an element, click and drag across the diagonal.

- (1) Move the pointer to (0, 0).
- (2) Click and hold the left mouse button and drag the pointer to the diagonal corner (200, 20).
- (3) Release the button.

Figure 4 shows the level 0 element we just created.

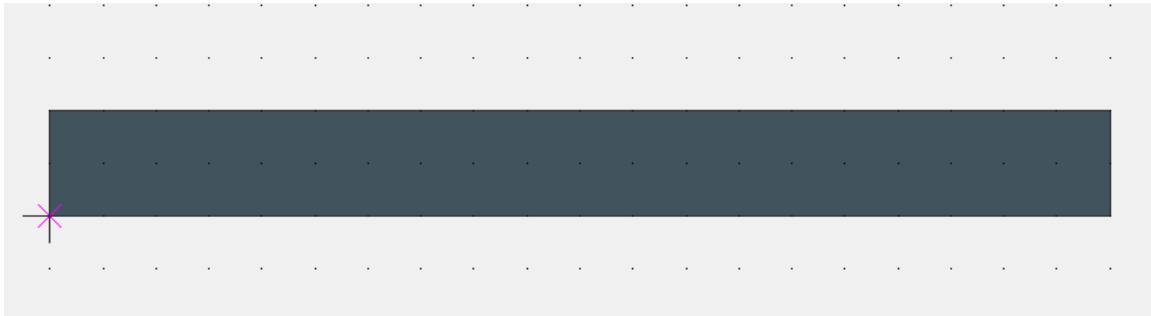


Figure 4 Level 0 element

### **Tips:**

If you click in the 3D viewing area (upper right window), you can rotate, pan, or zoom to see how your construction is progressing. This can also be accomplished with the keyboard arrow keys or via the mouse as follows:

#### **Rotate**

This is the default left mouse button action. To rotate the element in the 3D window, click once on the window and then click and drag (perpendicular to the desired axis of rotation) to rotate the object.

#### **Pan**

To Pan the 3D view, go to the *View...Pan* menu. Click and drag in the 3D view area to move the object.

#### **Zoom**

To zoom in or out the object, go to *View...3D Zoom*. *Zoom In* and *Zoom Out* zoom by 10%. To zoom in or out of a selected window, select *Zoom Window*. Click on the 3D view area. Hold down the mouse button and drag toward the lower left corner until the box just surrounds the object. Release the mouse button. The window will adjust so that you are now viewing what was in the box you just drew. To zoom back out, hold down the **Ctrl** button while holding and dragging toward the upper right with the mouse. The window will adjust proportional to the inside rectangle.

Note: When you are done zooming, we recommend changing back to *View...3D Rotate* or *View...Pan* to prevent accidental zooming.

### **2.1.2 Meshing the structure**

To mesh the structure we just created in 3D Builder for finite element analysis, go to *Mesh...Multiple Slices*, and click on it, as shown in Figure 5.

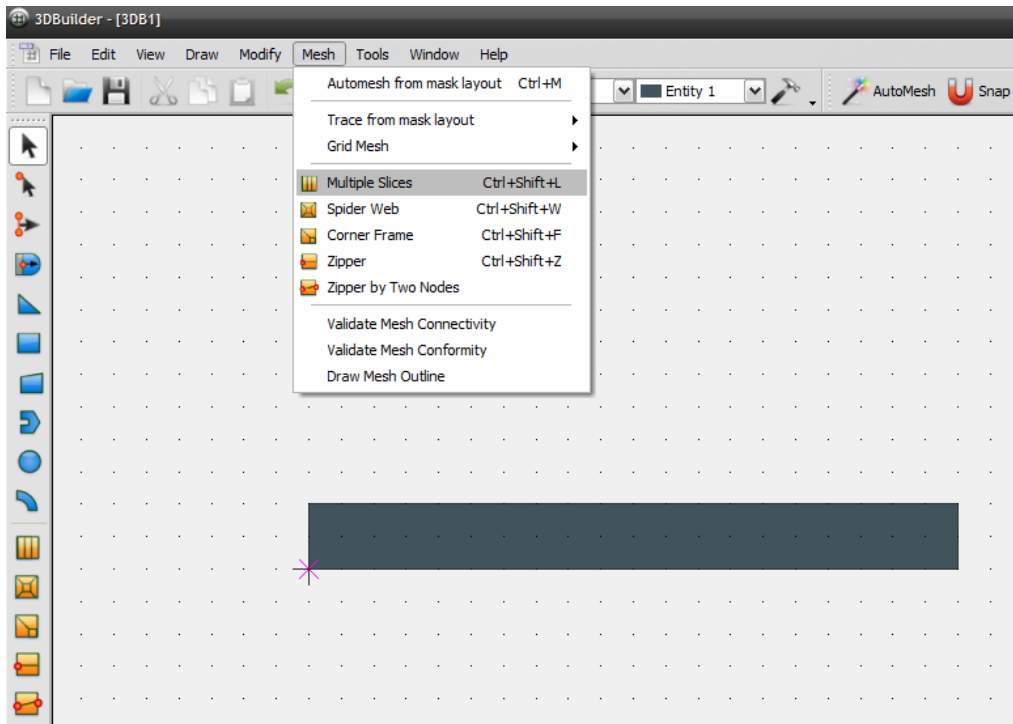


Figure 5 Select mesh generation method

Click on the top or bottom part of the beam entity to select it. A dialog box will pop up for mesh information input, as shown in Figure 6. When you click on the top or bottom part of the entity, one side on the top or bottom of the entity will be highlighted. Enter 18 for this case in the dialog box and click on **OK**.

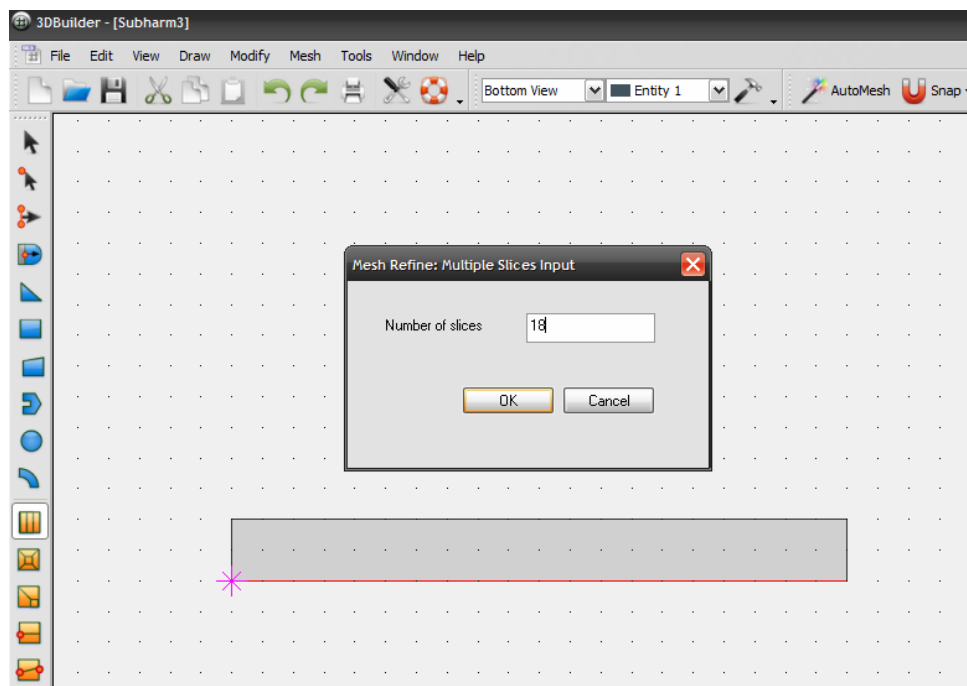


Figure 6 Multiple Slices mesh dialog box

Then go to *Mesh...Multiple Slices* again, and click on it. This time click on the left or right part of the beam entity to select it again. A dialog box will pop up for mesh information input, as shown in Figure 6. When you click on the left or right part of the entity, one side on the left or right part of the entity will be highlighted. This side will be divided for meshing. Enter 2 in the dialog box and click on **OK**.

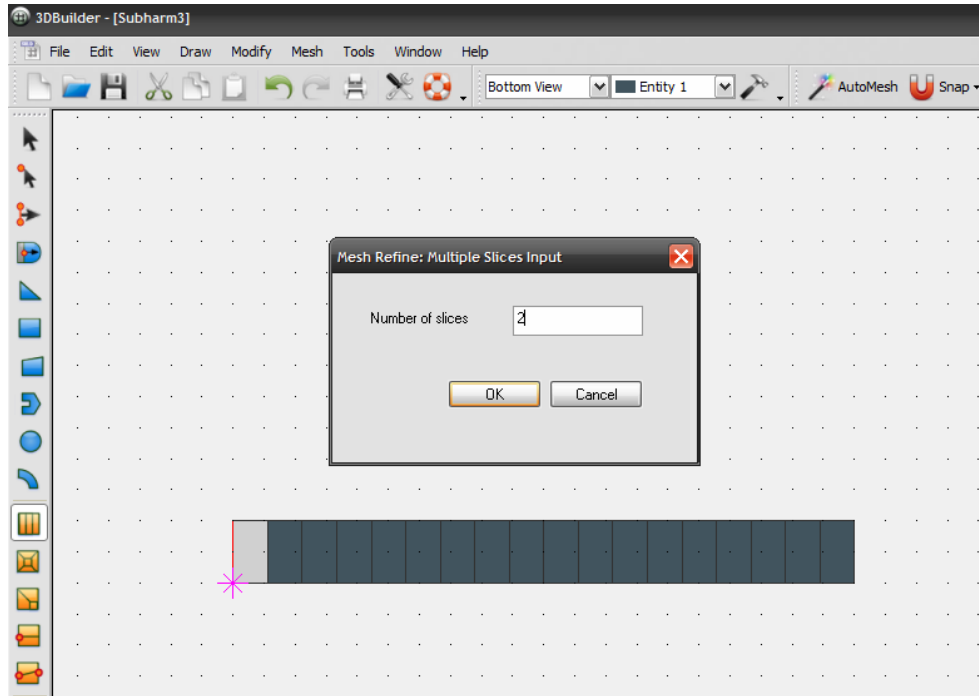


Figure 7 Another Multiple Slices mesh dialog box

Figure 8 shows the structure mesh pattern.

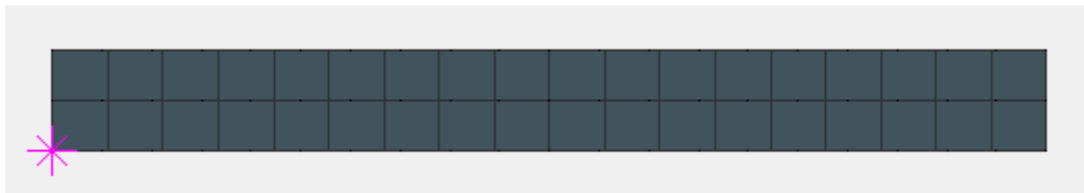


Figure 8 Meshed entity

### 2.1.3 Exporting to TEM

Now, we export the model created within 3D Builder to the TEM module for simulation. In the 2D view window, as shown in Figure 9,

**Click** *File...Export to Analysis Module.*

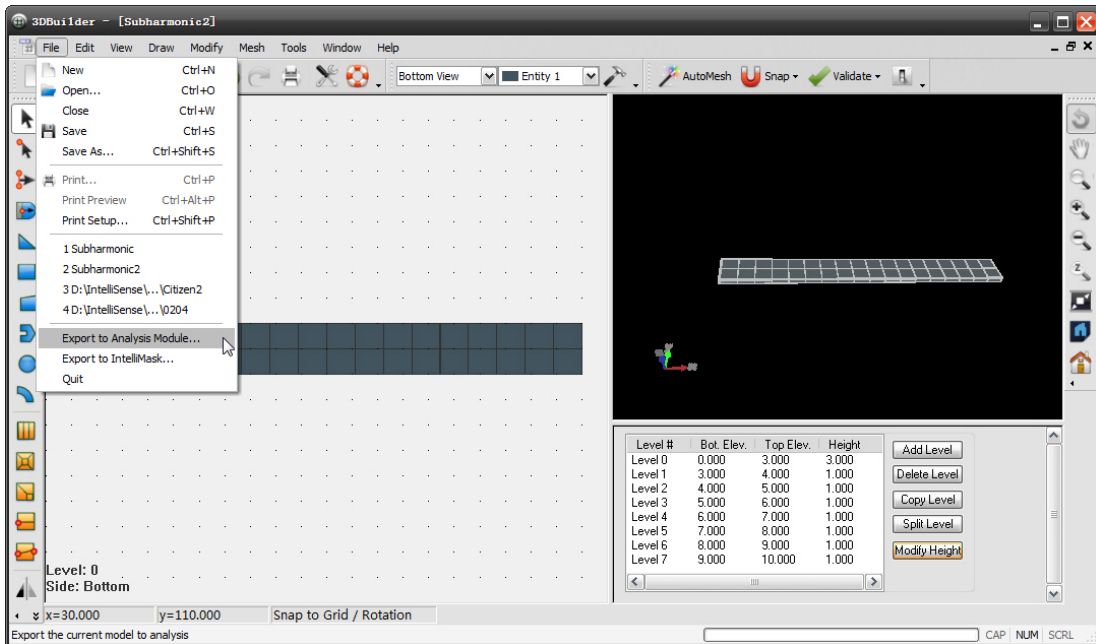


Figure 9 Export to TEM

A *Check Export Elements* dialog box will pop up. Check both boxes: *Validate Mesh Connectivity* and *Validate Mesh Conformity*, as shown in Figure 10, and click on **Check**.

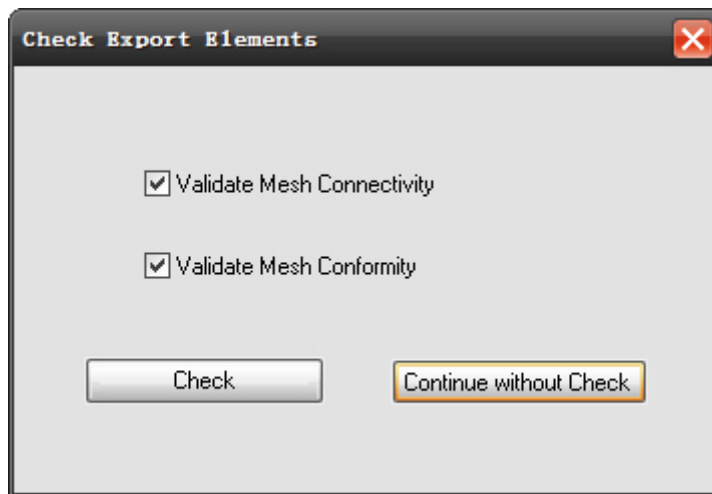


Figure 10 Check Export Elements dialog box

A second *Export to Analysis Module* dialog box will pop up. Check *ThermoElectroMechanical*, and click on **OK**, as shown in Figure 11.

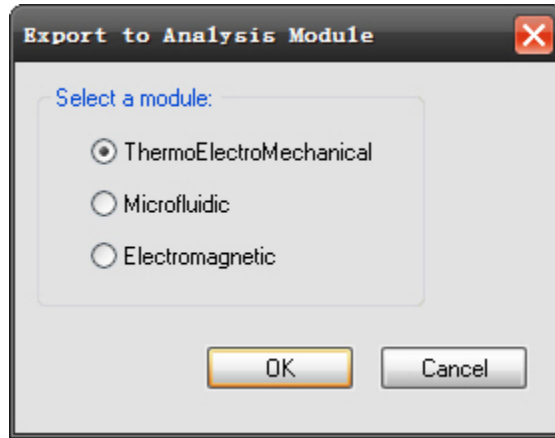


Figure 11 Export to Analysis Module dialog box

After a file saving dialog box pops up requesting a file name, specify a directory and a file name (subharmonic.save for this example) with the file name extension of *.save*. Then the 3D Builder window will close and a new ThermoElectroMechanical Analysis window will open. Figure 12 shows the 3D view of the newly exported mechanical structure.

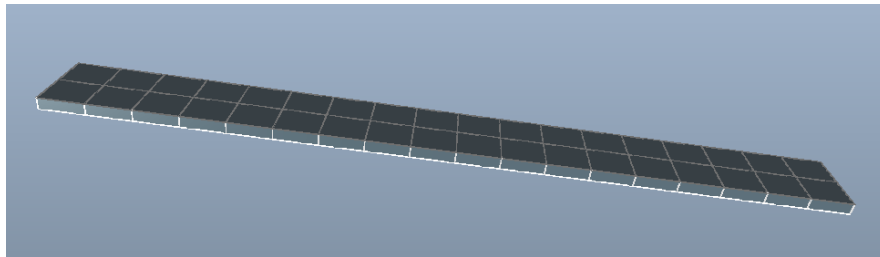


Figure 12 TEM model exported from 3D Builder

## 2.2. Setting simulation conditions

In this section, we will set conditions for simulation by applying proper material properties, boundary conditions and loads.

### 2.2.1 Applying material properties

Once we have the TEM model, we can select proper material properties for the entities. As shown in Figure 13, within the 3D view window,

**Click** *Material... Check/Modify*.

Click on the green entity to select it. It will turn red. A Check/Modify Material Property dialog box will pop up. Click on the *Import Property* button, and select *Si\_Bulk\_General* for this example. Then modify the properties by clicking on the property item and the *Edit Property* button, and specifying the material properties. The modified material properties are shown in Figure 14. Note the stress of the material has been changed to 200 MPa. Click on **OK**.

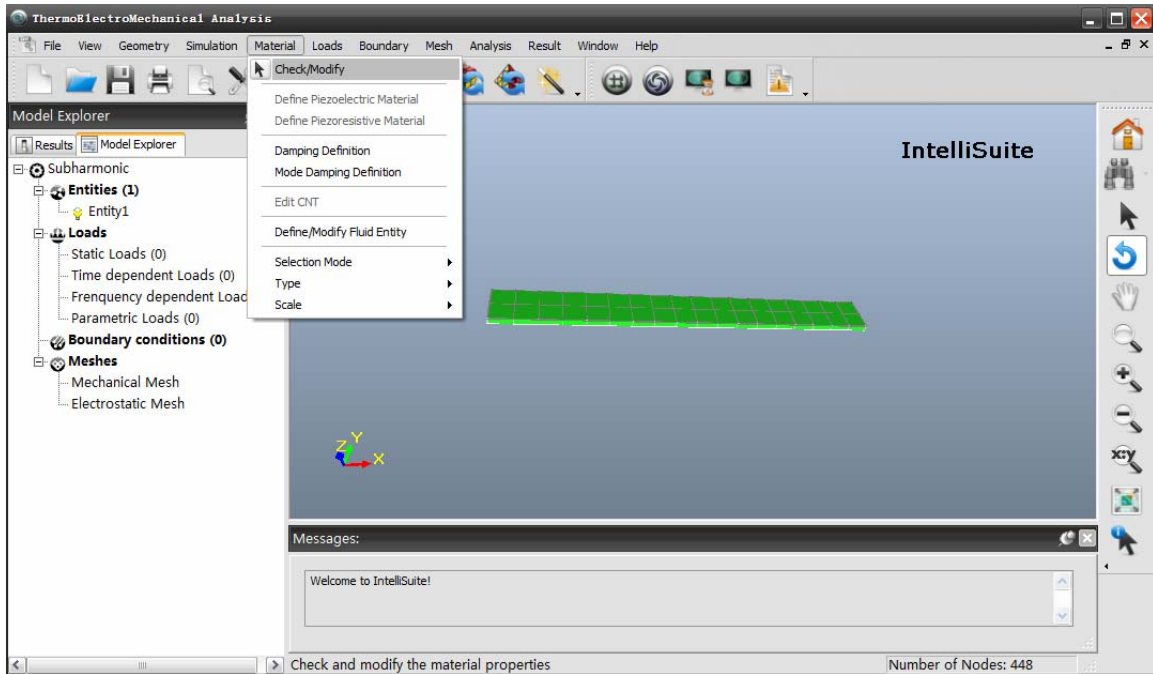


Figure 13 Check/modify material properties

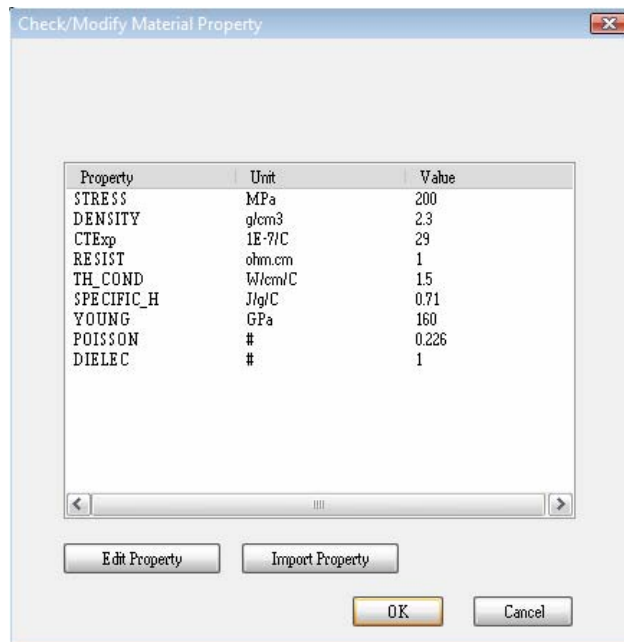


Figure 14 Check/modify material property dialog box

### 2.2.2 Applying boundary conditions

Once we have set the material properties, we need to apply boundary conditions to the model. In this example, we will only use the *Fixed* boundary condition.

**Click** *Boundary...Selection Mode...Pick on Geometry* (Figure 15)

**Click** *Boundary...Fixed*



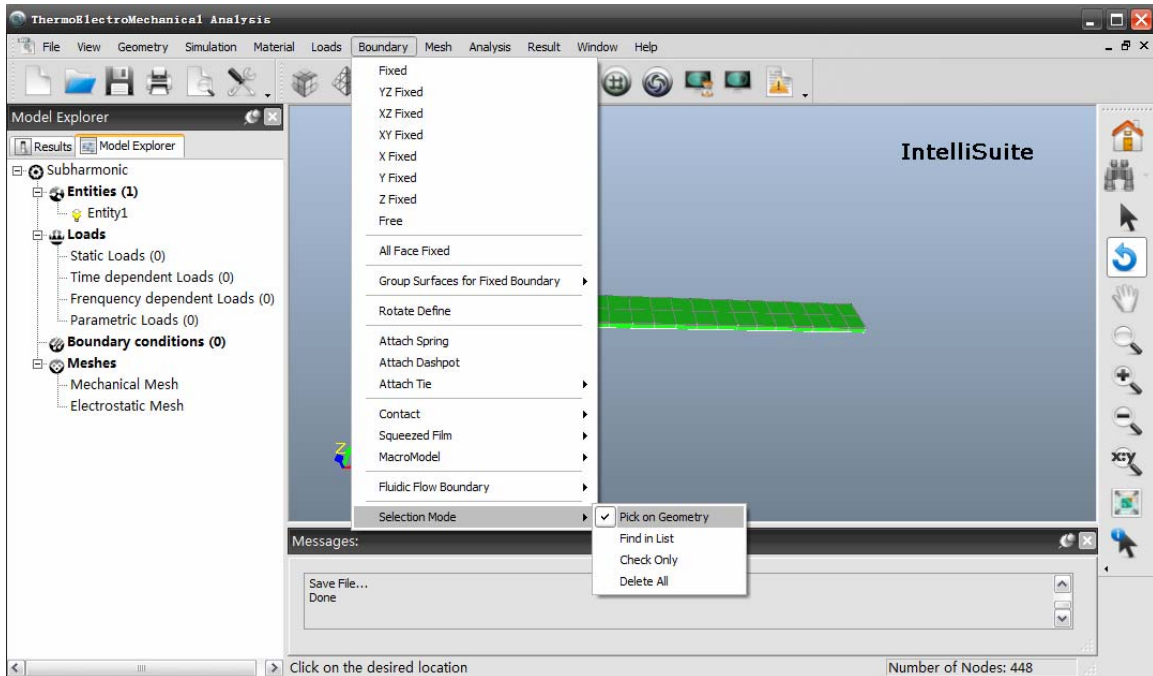


Figure 15 Apply boundary conditions

Select the faces as shown in Figure 16.

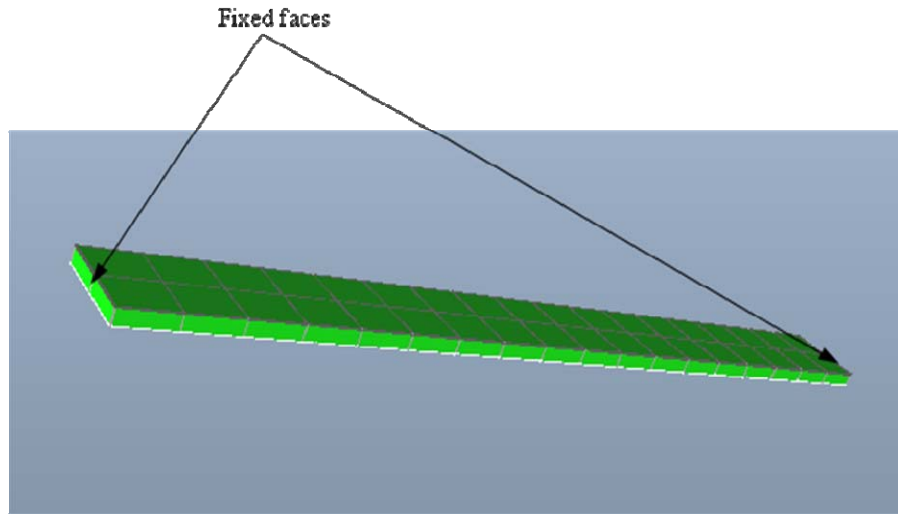


Figure 16 Select fixed faces

When you are finished applying all the fixed boundary conditions, check if the boundary conditions are really there.

**Click** *Boundary...Selection Mode...Check Only*

**Click** *Boundary...Fixed.*

All the fixed faces will be highlighted in bright red as shown in Figure 17 (the opposite face has been selected, but doesn't show up due to the view).

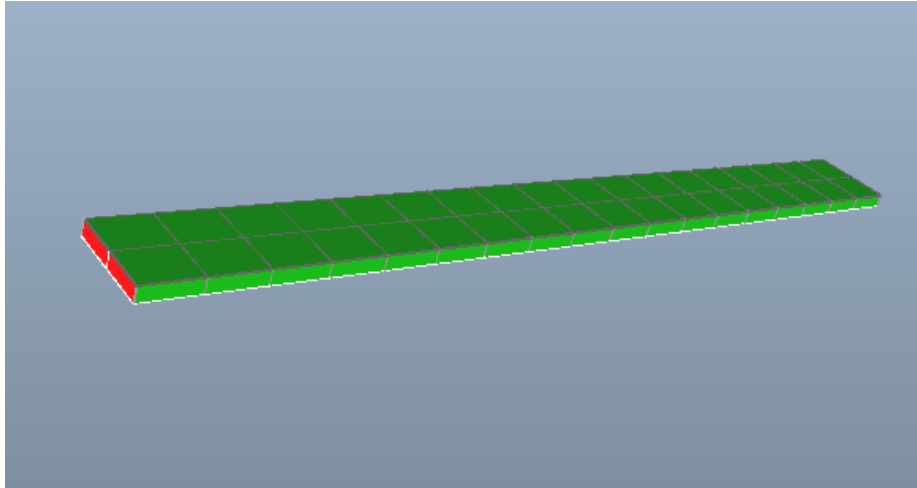


Figure 17 Fixed faces shown in red

### 2.2.3 Load application

The last step of setting simulation conditions is load application. IntelliSuite allows you to apply a large number of loads to the device. These range from forces, pressures, Coriolis forces, temperatures and other stimuli. The loads can be constant loads, time varying loads or frequency varying loads. Because the first analysis we will perform is Frequency Analysis, no load application is needed actually to find the frequencies of different free vibration modes. So we will apply loads during the sub-harmonic and coupled resonance analyses.

## 2.3. Frequency Analysis

In this example, we will find the natural frequency of the first 3 vibration modes of the structure. Once we have the TEM model, we should specify the simulation settings before running simulation. Within the TEM 3D view window,

**Click** *Simulation...Simulation Setting* (Figure 18).

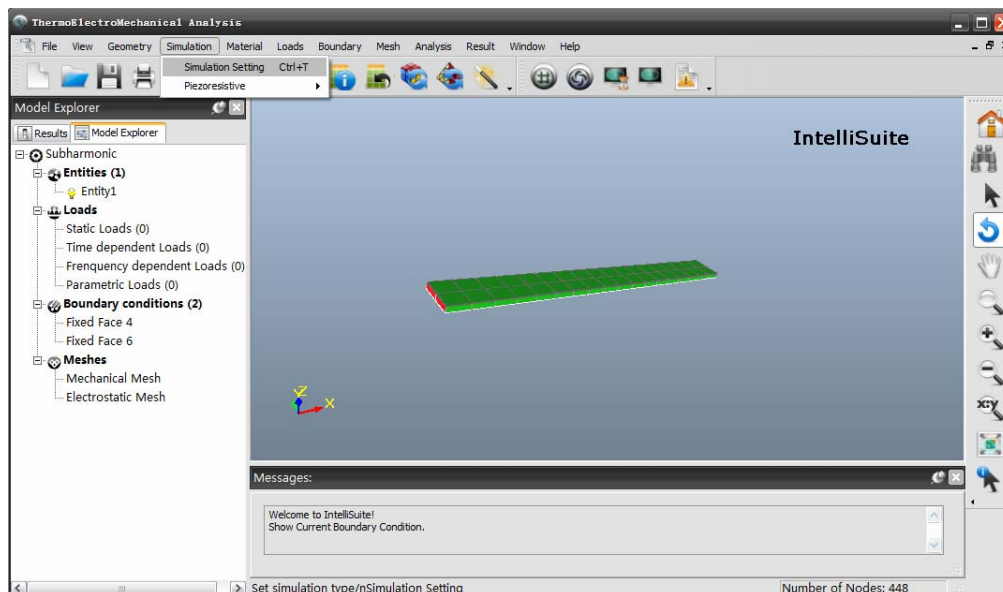


Figure 18 Access to simulation settings

A dialog box will pop up, requesting simulation settings. In this window you will find all the calculation types and the corresponding options. Specify the simulation settings as follows. The simulation we will run now is a natural frequency analysis. So check *Frequency* for the *Calculation Type*, and *Static Stress* for *Analysis Type*. Enter 3 for *Mode Number* to find the first 3 modes and 0 for *Frequency of Interest* using *Small* scale displacement theory. Choose *Undeformed* as the start shape, and no piezo material. The simulation setting dialog box should look like Figure 19. Click **Apply** and **OK**.

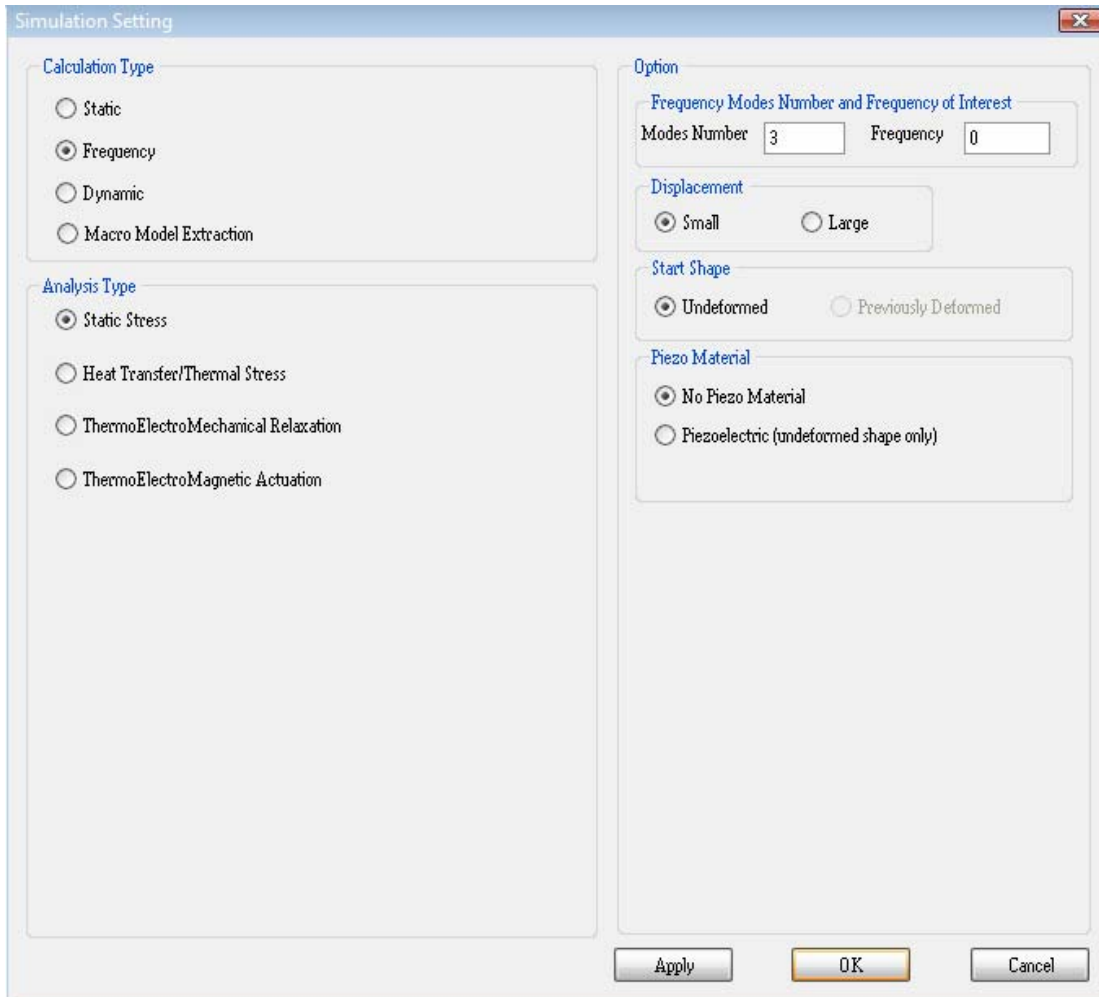


Figure 19 Simulation setting window

Because we have already set the correct material properties and boundary conditions, to perform a frequency analysis,

**Click** *Analysis...Start Frequency Analysis* (Figure 20).

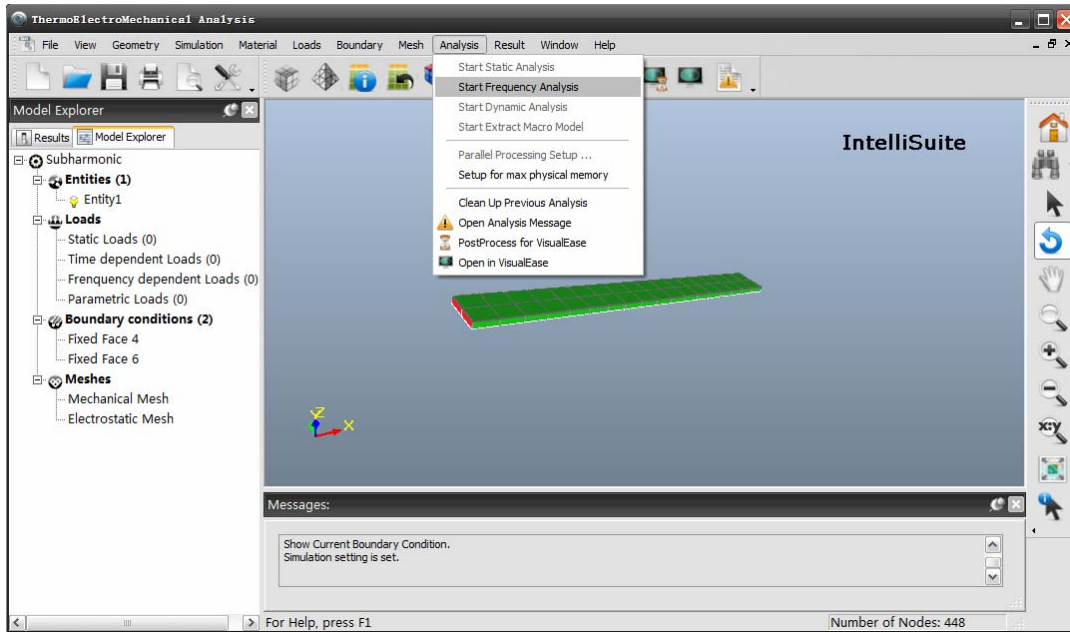


Figure 20 Start Frequency Analysis

The frequency analysis will take a few minutes. While the simulation is running, a window will pop up on your computer screen, showing the progress of the simulation. The window will close as soon as the simulation is finished. After the simulation, the results available for you to check are the natural frequencies and animations of each vibration mode. These can be found on the *Result* menu (Figure 21).

Click *Result...Natural Frequency*

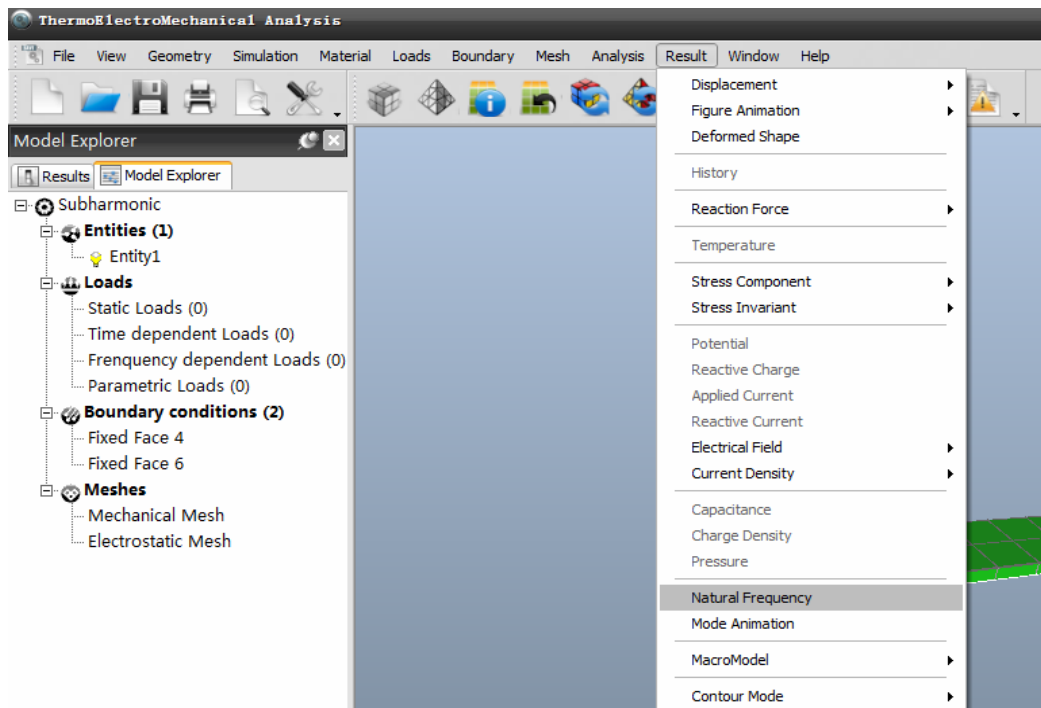


Figure 21 Access to natural frequency results

You will see the list of natural frequencies as shown in Figure 22.

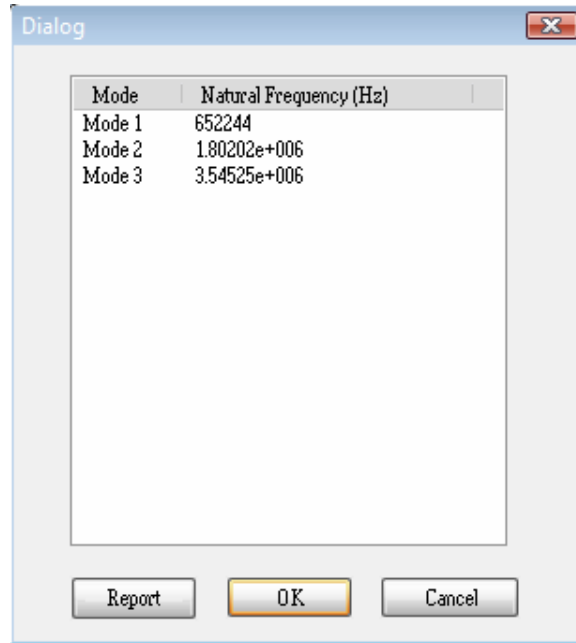


Figure 22 List of natural frequencies

If you want to see the animation of any of the vibration modes,

**Click** *Result...Mode Animation*

This will bring up a dialog box (Figure 23) that asks you the mode number of your interest and an appropriate vibration magnitude scale factor. The default vibration magnitude is 1  $\mu\text{m}$  maximum. The scale factor allows you to view the vibration animation more clearly by magnifying the vibration magnitude.

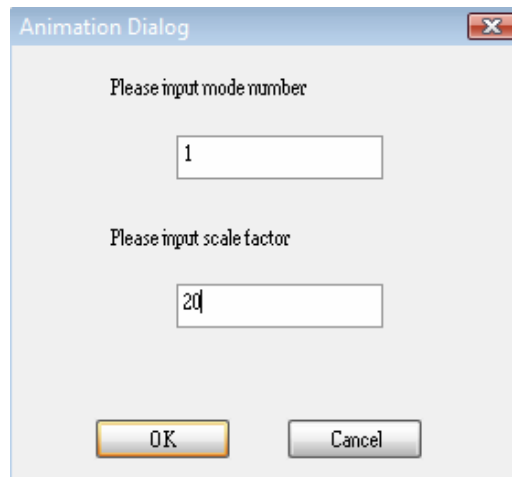


Figure 23 Animation selection dialog box

By entering 1, 2 and 3 as the input mode number respectively and 20 as the input scale factor, you will be able to view the animations of the first 3 vibration modes by a magnitude magnification factor of 20. Figure 24 shows the shots taken during the beam vibration mode animations. The mode animations give us hints where to apply the loads for sub-harmonic and coupled resonance simulations.

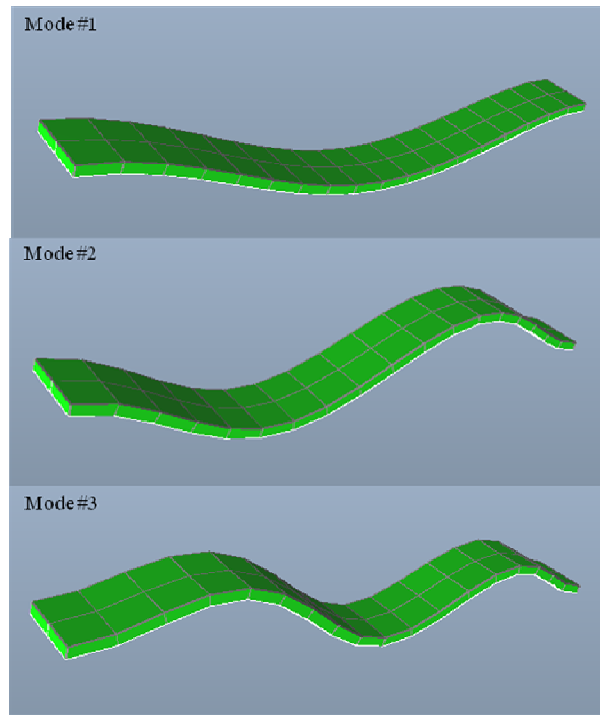


Figure 24 Beam vibration modes

Now that we understand the vibration modes of the clamped-clamped beam, we will move on to dynamic simulations to analyze the beam responses of sub-harmonic and coupled resonances. To exit the TEM module,

**Click** *File...Clean Up & Exit* (Figure 2).

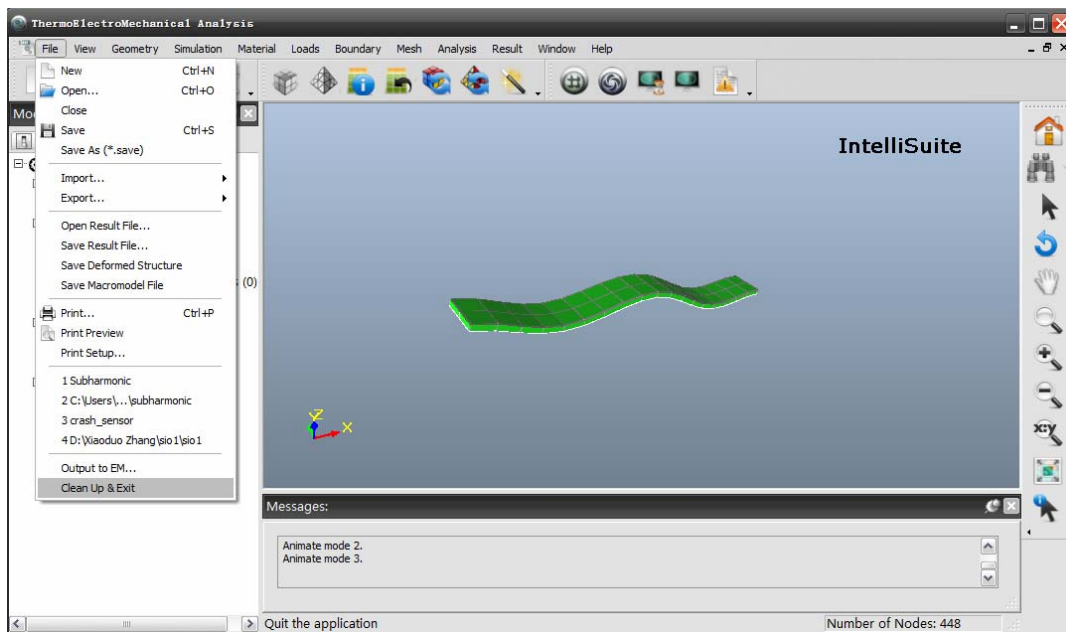


Figure 25 Clean up and exit

When asked if you want to save the file, click on **Yes**.

## 2.4. Sub-harmonic resonance analysis

We didn't specify the damping factors when we set up the simulation for the Frequency Analysis as above, because the damping factors don't affect the vibration frequencies or modes. If the clamped-clamped beam is considered a dynamic system, however, the damping factors within or associated with the beam will decay the vibrations, and so must be specified. The damping factors of a structure are also specified as material properties in the TEM module. So, to set the damping factors for a structure,

**Click** *Material...Damping Definition.*

A dialog box of Check/Modify Material Property will pop up. Click on *Edit Property*, and enter 6512 for the Mass\_damping and 1.535e-8 for the Stiffness\_damping, as shown in Figure 26. These values are obtained through experiments or from the user's experiences.

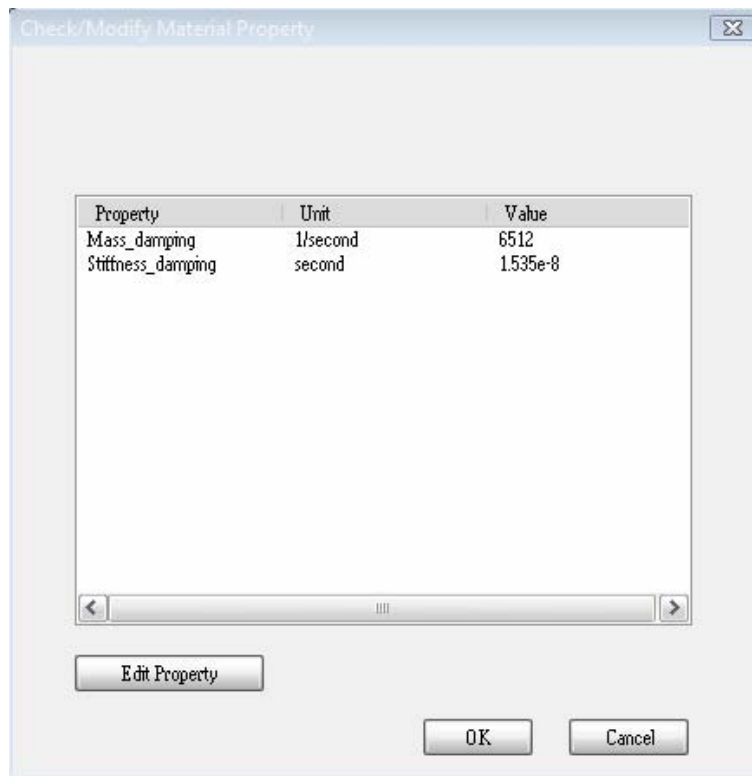


Figure 26 Damping definition

### 2.4.1 Load application

Once the damping factors are defined, we need to set force loads that will be applied at the center node of the clamped-clamped beam in this example.

**Click** *Loads...Amplitude vs. Time...Periodic*

**Click** *Loads...Pressure...Node*

Click to select the center node of the beam (Figure 27).

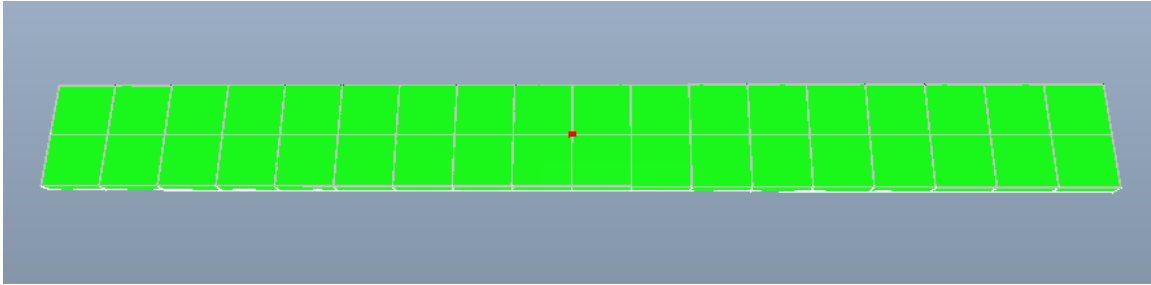


Figure 27 Clamped-clamped beam with the force at the center

Input the values as shown in Figure 28 into the *Amplitude Definition* dialog box. The force amplitude is 1  $\mu\text{N}$ , and the frequency is half the frequency of the first vibration mode.

**Amplitude Definition** ✕

Node 169 is Selected

Define Applied Force Curve

$\ddot{A} = A_0 + A_1 \cos W(t-t_0) + B_1 \sin W(t-t_0)$  for  $t \geq t_0$   
 $\ddot{A} = A_0$  for  $t < t_0$

	X	Y	Z	
t0	0	0	0	sec
W	0	0	2046000	rad/sec
A0	0	0	0	1e-6N
A1	0	0	0	1e-6N
B1	0	0	1	1e-6N

Figure 28 Amplitude definition of a periodic force load at  $\omega/2$

#### 2.4.2 Mesh refinement

Mesh refinement is used in finite element analysis to improve computation efficiency, accuracy and convergence probability. It is often performed for irregular geometric shapes and at high load density area. For regular geometric shapes, large mesh patterns are used for simulation to save computation time. For irregular geometric shapes or local areas, fine mesh patterns help the computation converge because those areas are often where stresses are relatively very high within a body.

In this example, the shape of the clamped-clamped beam is a rectangle. So no mesh refinement is done for this simple shape for Dynamic Analysis.

#### 2.4.3 Simulation and results



## Linear analysis

Set up the simulation as we did for the Frequency Analysis previously. Figure 29 shows the dialog box of the simulation settings for the linear analysis. The simulation we will run is a *Dynamic Analysis*. The *Stress/Displacement (Direct Integration)* method is used. We are only interested in the *Last State* of the results. Since we are performing linear analysis for this simulation, select *Small* for the displacement. Choose *Undeformed* as the start shape, and *No Piezo Material*. Check *No* for dynamic steady state option, and specify a 10  $\mu$ s for the fixed transient simulation time and an increment number of 500. We don't do *Contact Analysis* here. Click **Apply** and **OK**.

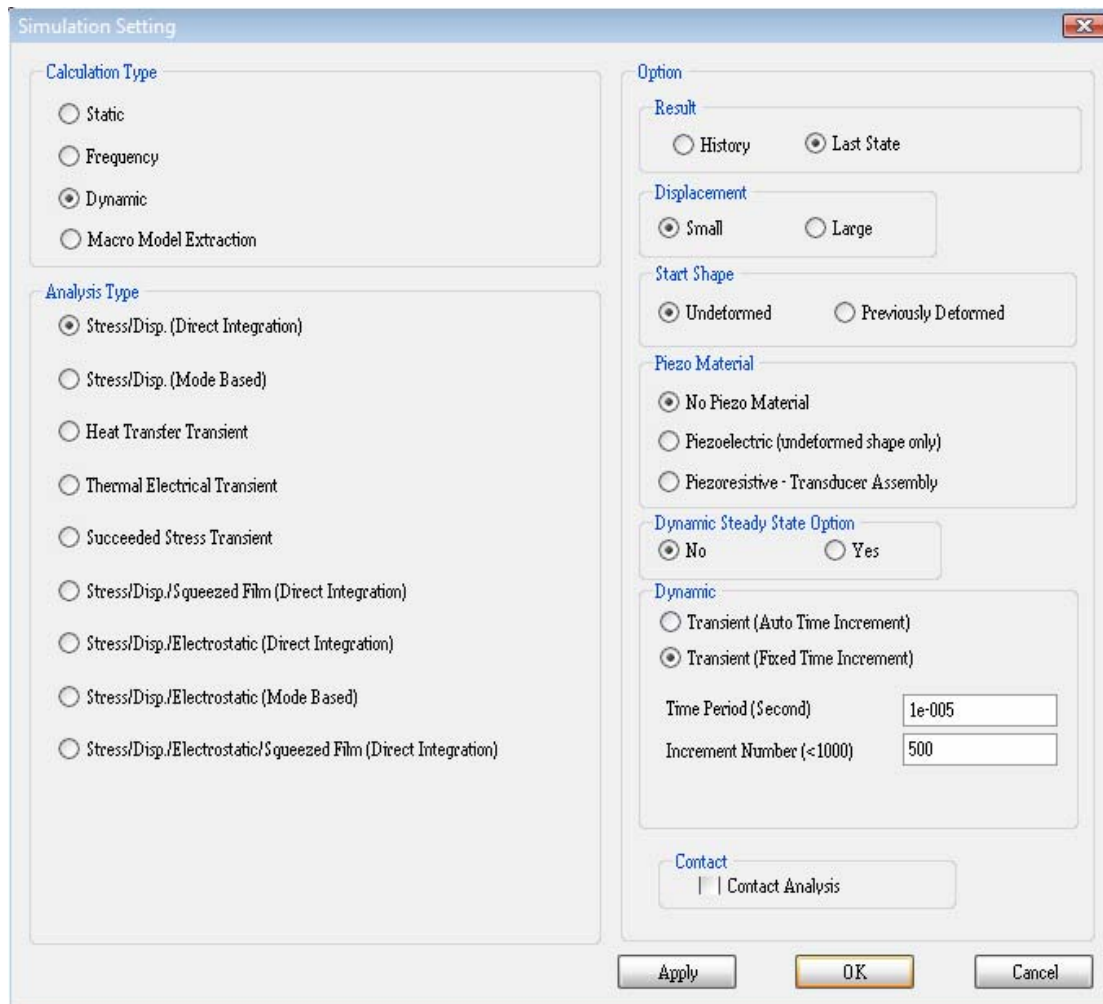


Figure 29 Dynamic Simulation settings

To run time-domain dynamic analysis simulation,

**Click** *Analysis...Start Dynamic Analysis*

When the simulation is finished,

**Click** *Result...Node Curve...Magnitude* (Figure 30)

**Click** *Result...Displacement...Z*

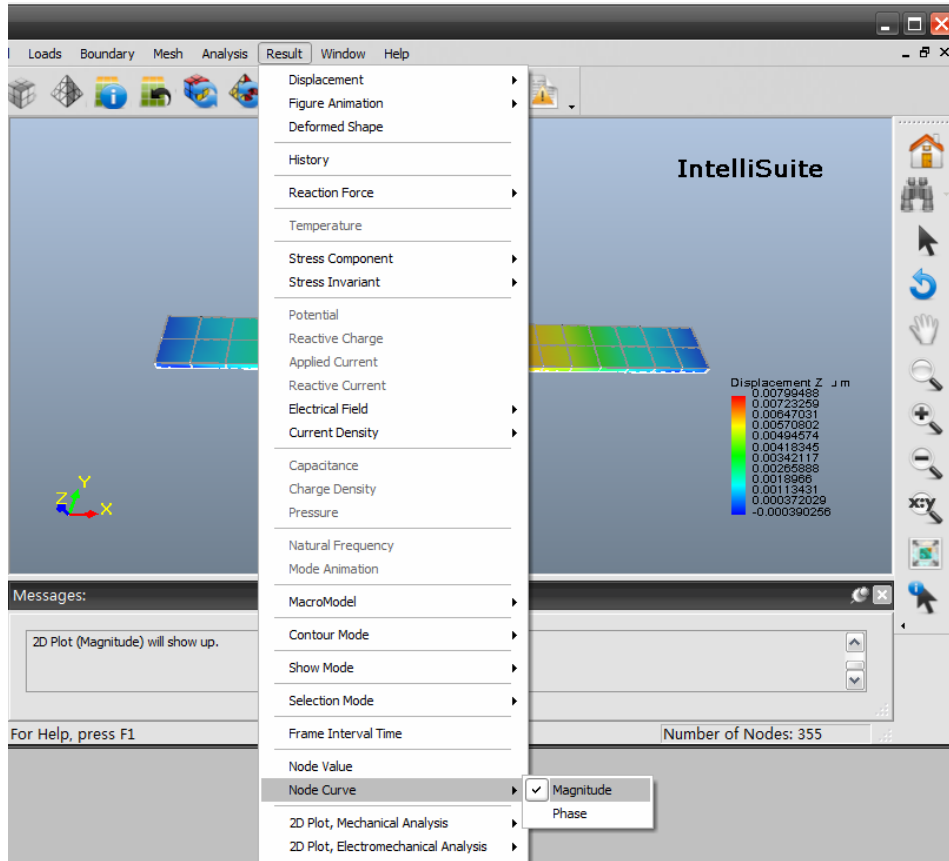


Figure 30 Access to Z displacement curve of the beam

Then click on the node where we applied the load, i.e., the center node of the beam. The beam Z displacement versus time curve will be drawn and a window will pop up to show it, as in Figure 31, for the linear analysis case of a force load at the frequency of  $\omega/2$ , where  $\omega$  is the frequency of the first vibration mode of the beam.

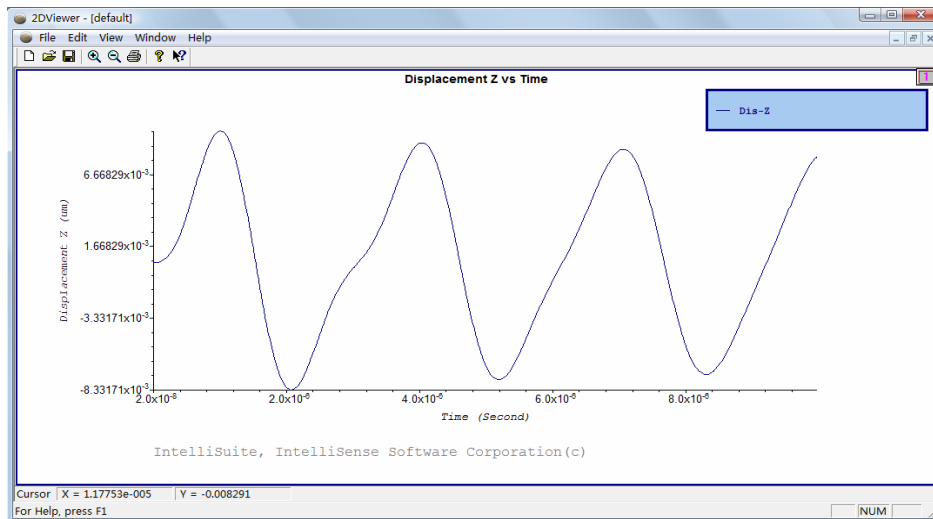


Figure 31 Linear analysis displacement results with a load at  $\omega/2$

Now change the load frequency to  $\omega/3$ , as shown in Figure 32, and run the simulation again with the same setup.

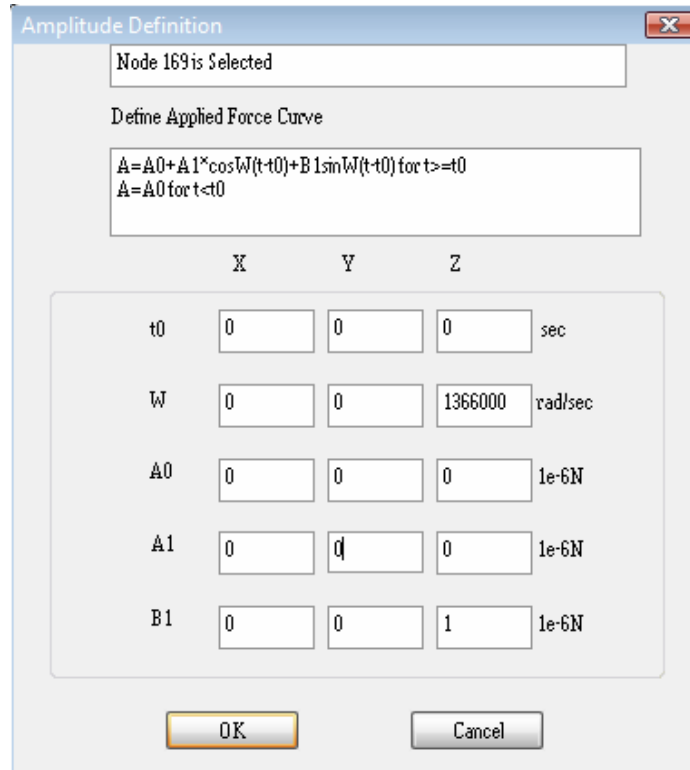


Figure 32 Amplitude definition of periodic force load at  $\omega/3$

Figure 33 shows the simulation results of Z displacement of the beam.

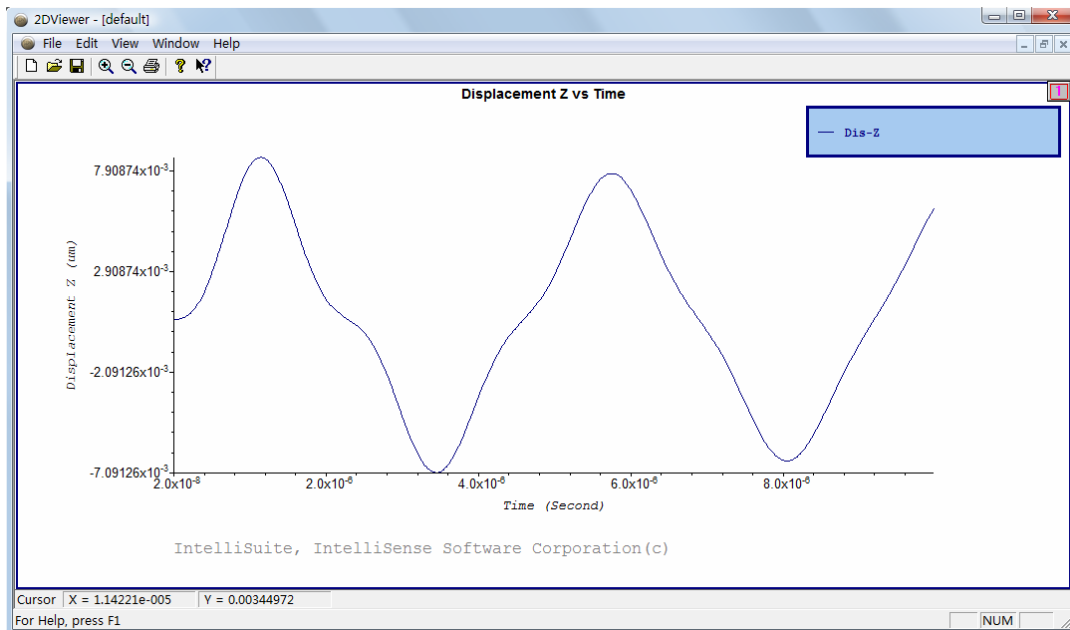


Figure 33 Linear analysis displacement results with a load at  $\omega/3$

### Non-linear analysis

All systems in reality are non-linear systems. A dynamic system (e.g., mechanical or electrical) is often considered a linear system for analysis when the input signal is so small that the consideration will not

significantly affect the output signal. Also, a system is often linearized at a small local area of interest where the linearization will not result in an intolerant error. Such linearization techniques simplify the systems under consideration to save computation time.

When the input signal is large or local linearization will lead to inaccurate analysis results, however, non-linear system analysis needs to be carried out. In this section, non-linear analyses will be performed for a large force load at frequencies of  $\omega/2$  and  $\omega/3$ .

Define the force load frequency and amplitude again as we did for the linear analyses. Figure 34 shows the Amplitude Definition dialog box. Enter 1000  $\mu\text{N}$  for the force amplitude. This force is 1000 times as large as the force for the linear analysis. Treating the beam as a linear system would lead to inaccurate simulation results due to the large force.

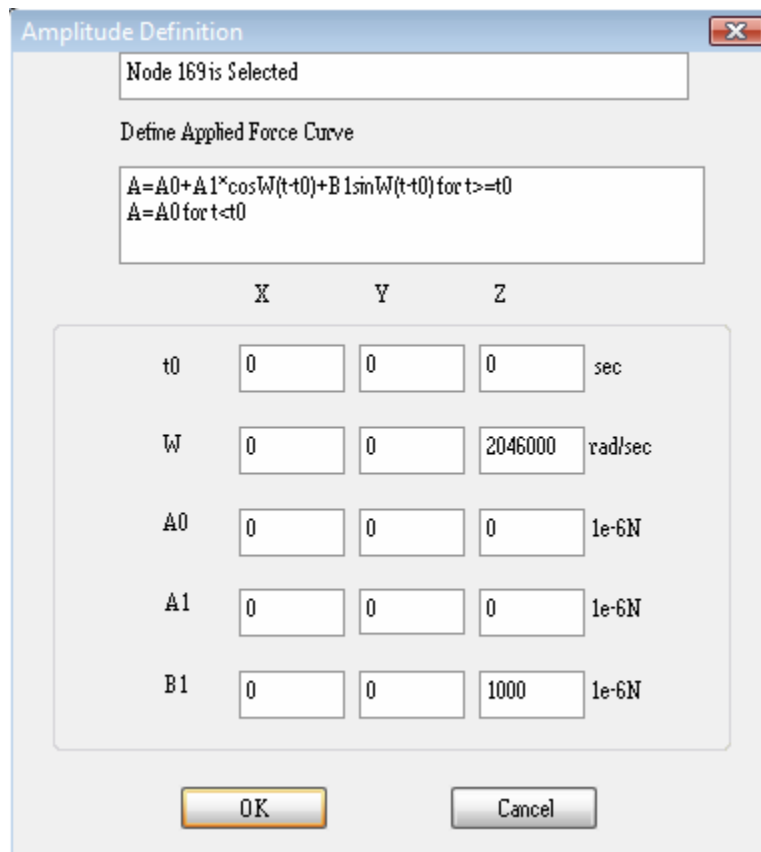


Figure 34 Amplitude definition of periodic force load at  $\omega/2$  for non-linear analysis

Therefore, we choose *Large* for Displacement in the Simulation Setting dialogue box as shown in Figure 35. That is, a large signal analysis will be performed, and the beam will be considered a non-linear system under such a large force during simulation.

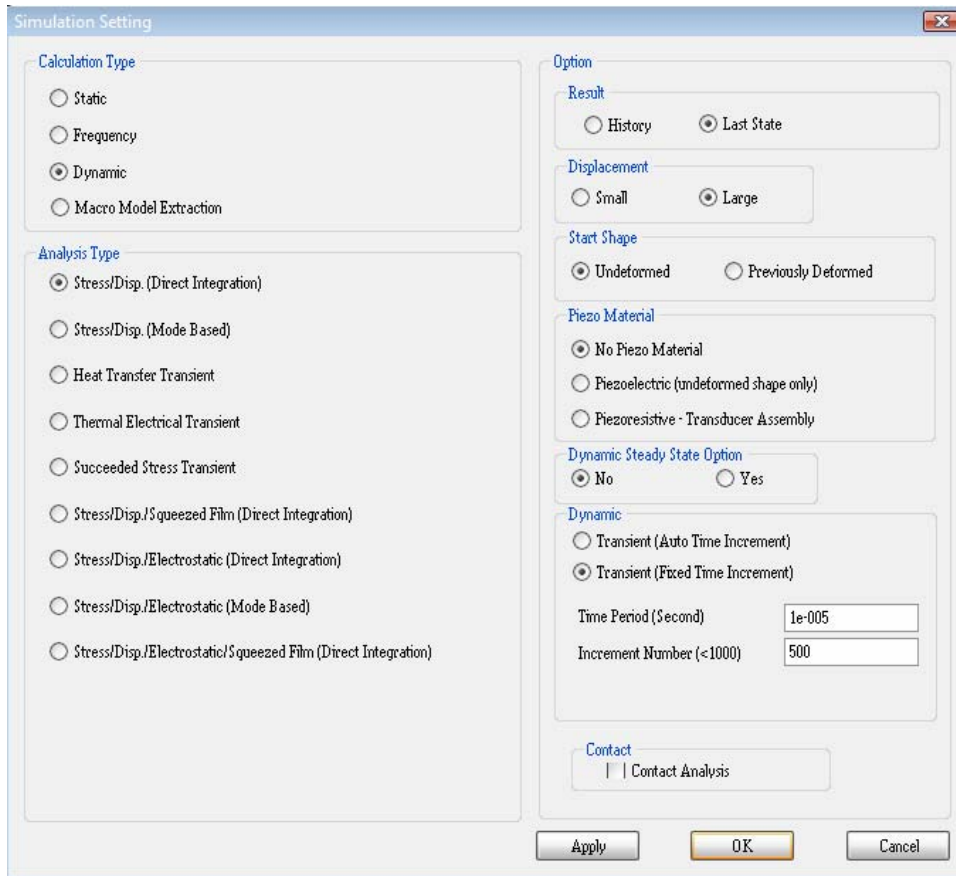


Figure 35 Dynamic Simulation settings for non-linear analysis

Run the time-domain dynamic analysis simulation again and view the results (Figure 36).

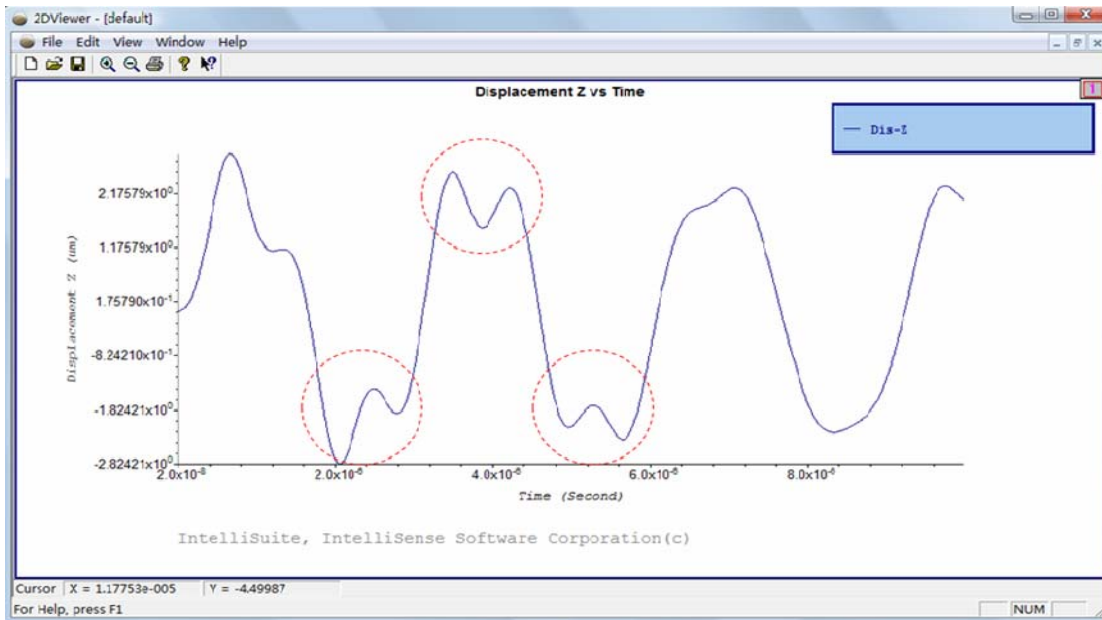


Figure 36 Non-linear analysis displacement results with a load at  $\omega/2$

Then change the load frequency to  $\omega/3$ , as shown in Figure 37, and run the simulation again with the same setup.

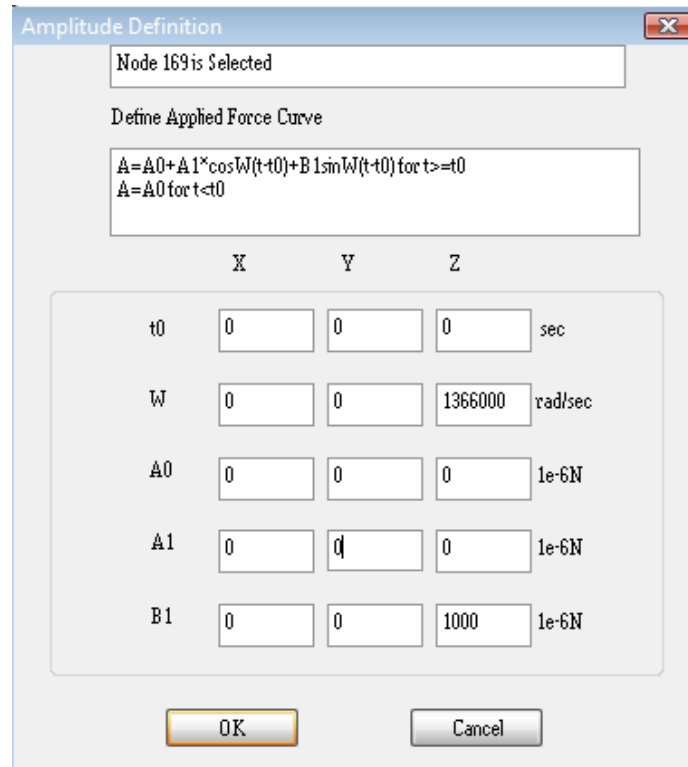


Figure 37 Amplitude definition of periodic force load at  $\omega/3$  for non-linear analysis

Figure 38 shows the simulation results of Z displacement of the beam.

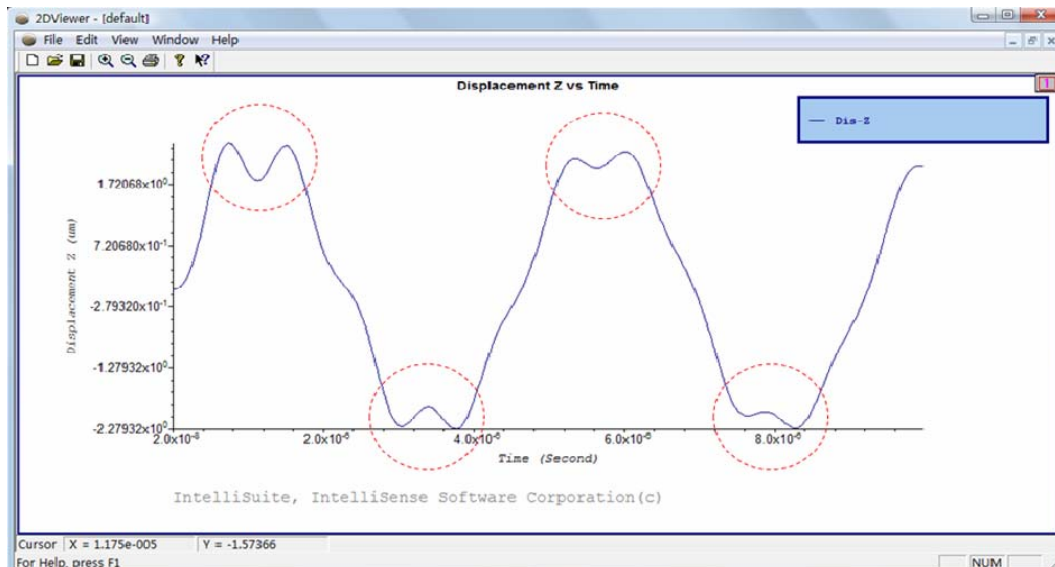


Figure 38 Non-linear analysis displacement results with a load at  $\omega/3$

A comparison is made between the linear and non-linear analysis results (Figure 31 vs Figure 36, and Figure 33 vs Figure 38) as follows. The fundamental vibration frequencies of the linear and non-linear systems are both around  $\omega/2$  and  $\omega/3$  on the Z displacement curves, respectively, as expected. However, the

Z displacement curves from the linear analyses do not show obvious harmonic waves, while the non-linear analysis results contain small but obvious harmonic waves as shown in the circled areas in Figure 36 and Figure 38.

## 2.5. Coupled resonance analysis

In this section, coupled resonance analysis of the clamped-clamped beam will be carried out by applying 2 force loads at 2 different frequencies simultaneously, one at the frequency of the first vibration mode (denoted  $\omega_1$  in this section), and the other at the frequency of the third vibration mode (denoted  $\omega_3$ ). Figure 39 shows the nodes on which to place the force loads.

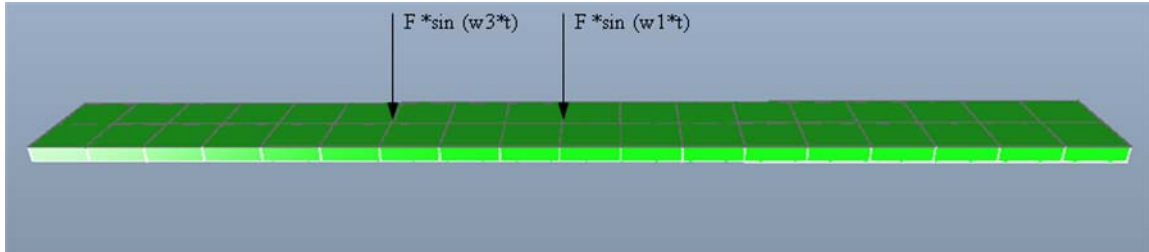


Figure 39 Meshed structure with 2 force loads at different frequencies

Figure 40 and 41 define the force amplitudes at the frequencies of  $\omega_1$  and  $\omega_3$ . Large signal non-linear analyses need to be performed. Check if the Simulation Settings are correct as we did before.

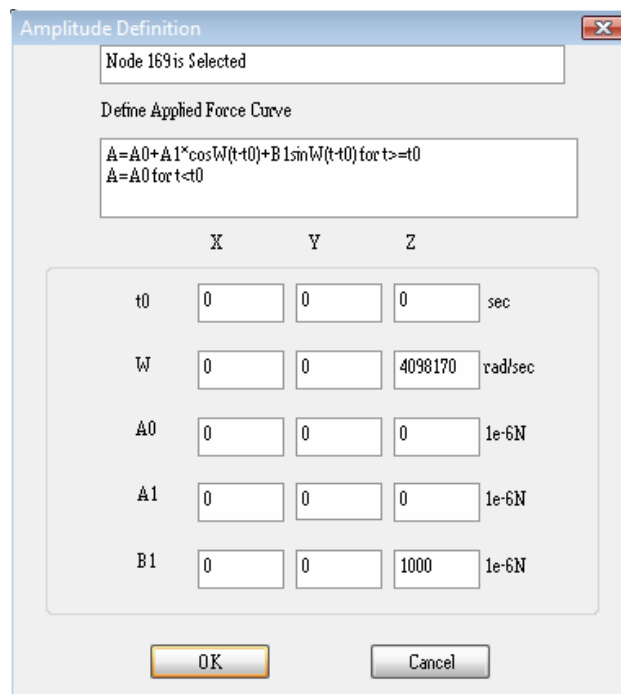


Figure 40 Amplitude definition of force at  $\omega_1$

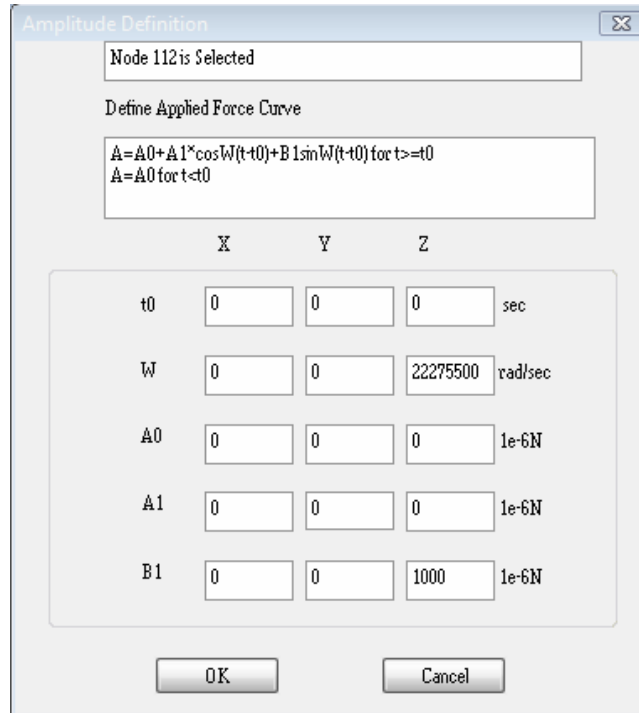


Figure 41 Amplitude definition of force at  $\omega_3$

Run the time-domain dynamic analysis simulation again and view the results (Figure 42). As can be seen, the output Z displacement curve contains obvious large waves with a frequency of  $\omega_1$  and small waves at frequencies hard to determine at first glance. These small waves are actually the superposition results of the system responses of higher order harmonics (*not only* at  $\omega_3$ ).

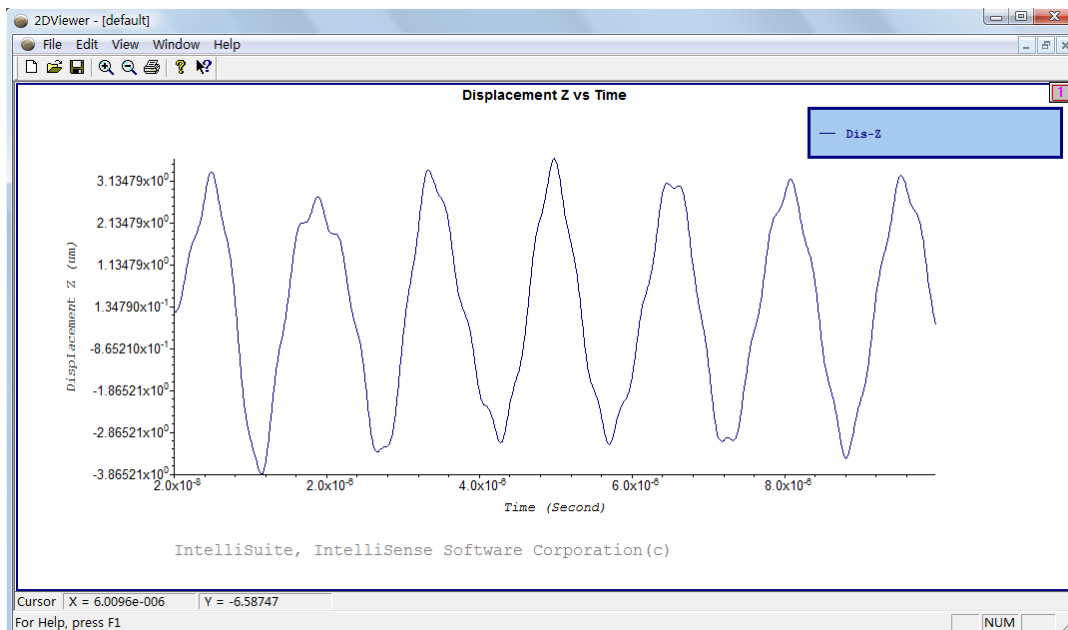


Figure 42 Non-linear analysis displacement results with load at  $\omega_1$  and  $\omega_3$



### 3. Summary

Sub-harmonics originate from non-linear systems when the output signal contains components at frequencies lower than the input signal. For such non-linear systems, sub-harmonic resonance responses are of system designers' interest. But linear analysis will not give designers accurate results due to oversimplification. Non-linear analysis needs to be carried out to find the sub-harmonic frequencies and the system responses.

Coupled resonance responses of systems to large input signals are also non-linear. When any of the coupled loads or the overall load is large, the entire system should be considered non-linear. For such cases, only accurate non-linear analysis results can provide designers trustworthy information about the real system.

IntelliSuite provides users the capability of performing accurate non-linear analysis. As examples, sub-harmonic resonance and coupled resonance responses are simulated for a clamped-clamped beam. Linear analysis is done for a small input force load, and non-linear analysis is performed for a large input force. A comparison of the results shows that the non-linear simulation results reveal the harmonics contained within the output Z displacement curves while linear analysis results do not contain such information. The coupled resonance responses to large force loads show that the output Z displacement curves contain the frequency components not only at the input frequencies, but also at their higher order harmonic frequencies.







# IntelliSense

Total MEMS Solutions

[www.intellisense.com](http://www.intellisense.com)