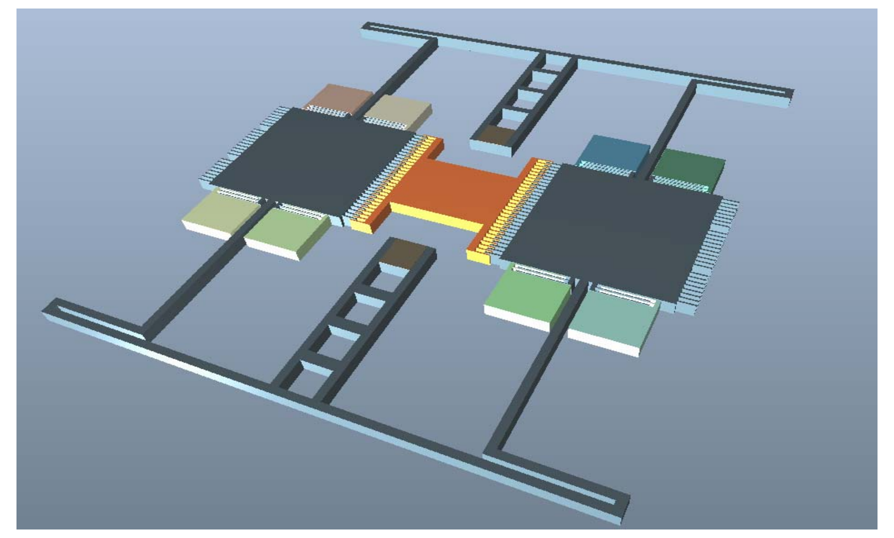
Application Note

Mode Matched Tuning Fork Gyro



Application Note: Mode Matched Tuning Fork Gyro
Version 8.6/PC

Part Number 30-090-101
Jan 2010

© Copyright IntelliSense Software Corporation 2004, 2005, 2006, 2007, 2008, 2009, 2010
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™, SYNPLE, CleanRoom, Blueprint, Fastfield Multiphysics, Hexpresso, TEM, IntelliMask, IntelliMask Pro, MEMS-Synth, 3DBuilder, IntelliFAB, FABViewer, WaveRunner, EDALinker, are trademarks of IntelliSense Software Corporation.
Windows NT, Windows 2000, Windows XP, Vista and Windows 7 are trademarks 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

1. Introduction

In this application note, we will discuss the design of a mode matched tuning fork gyroscope from an initial element-based model through device-level analysis to macromodel extraction and integration with a control circuit. We will analyze the gyro we have developed and discuss the benefits of IntelliSuite when it comes to MEMS-based inertial sensor design. Using IntelliSuite, one can optimize a design without having to go through the costly procedure of prototype development and testing. First, we will do some initial design exploration using a model built from scratch with system-level components (beams, plates, gaps, comb drive actuators, etc.). Taking advantage of the great speed of SYNPLE, IntelliSense's system-level simulator, it is easy to optimize the design of a device using parametric analysis. We will then extract a mask layout from the SYNPLE schematic and use it to automatically create a 3D meshed model to be used for analysis. We will determine the natural frequencies of the gyro and will examine the device response in a static case and a dynamic case. We will then use a unique feature called N-degree-of-freedom System Model Extraction (N-DOF SME) to generate a system model of the device which retains the accuracy of the device-level model. We can incorporate this model into a system-level simulator like SYNPLE to co-simulate the device with the CMOS control circuitry that will be used to govern and export the signals.

1.1. Background

Inertial sensors are common MEMS devices because the benefits greatly outweigh the costs when the MEMS based devices are compared to their earlier, larger, more complex counterparts. Inertial devices were first designed for military applications. They were used for guidance and navigation. Early inertial devices were extremely large, bulky, expensive, and very complex from a manufacturing standpoint. More recently, they have been adopted by commercial industries (air bag sensors, etc.), which has led to a drive to make inertial devices cheaper and smaller. This in turn has resulted in the adoption of MEMS technology in the fabrication of inertial devices because of the bulk manufacturability of the devices as well as the small size and weight associated with all MEMS devices.

Inertial sensors use the inertia of a mass to sense acceleration or rotation. The two major types of inertial devices are gyroscopes and accelerometers. The accelerometer uses an inertial mass to sense acceleration along one of the three axes while the gyroscope uses a mass to sense a rotation around one of the three axes. Some are designed for sensing in multiple axes. Accelerometers are usually single masses supported by a spring structure designed to allow the mass to move within a certain range under certain loading conditions depending on the desired sensitivity of the device. Gyroscopes must be designed with multiple paths of motion in mind. This is because they are constantly vibrated in one direction and when the device is rotated, the combination of the vibration and the rotation cause a Coriolis force which causes displacement along an axis perpendicular to the axis of rotation and the axis of vibration.

1.2. The Gyroscope

In this particular instance, we will examine a single axis gyroscope with two masses (***Figure 1***). In this example, the two masses will be oscillated such that they both move towards the center of the device at the same time. This will cause opposite velocities and keep the forces on the device balanced. The device will use the capacitance between each moving mass and each of the four electrodes (shown in multiple colors in ***Figure 1***) to sense the rotation-induced displacement. Multiple electrodes are being used for noise cancellation and sensitivity.

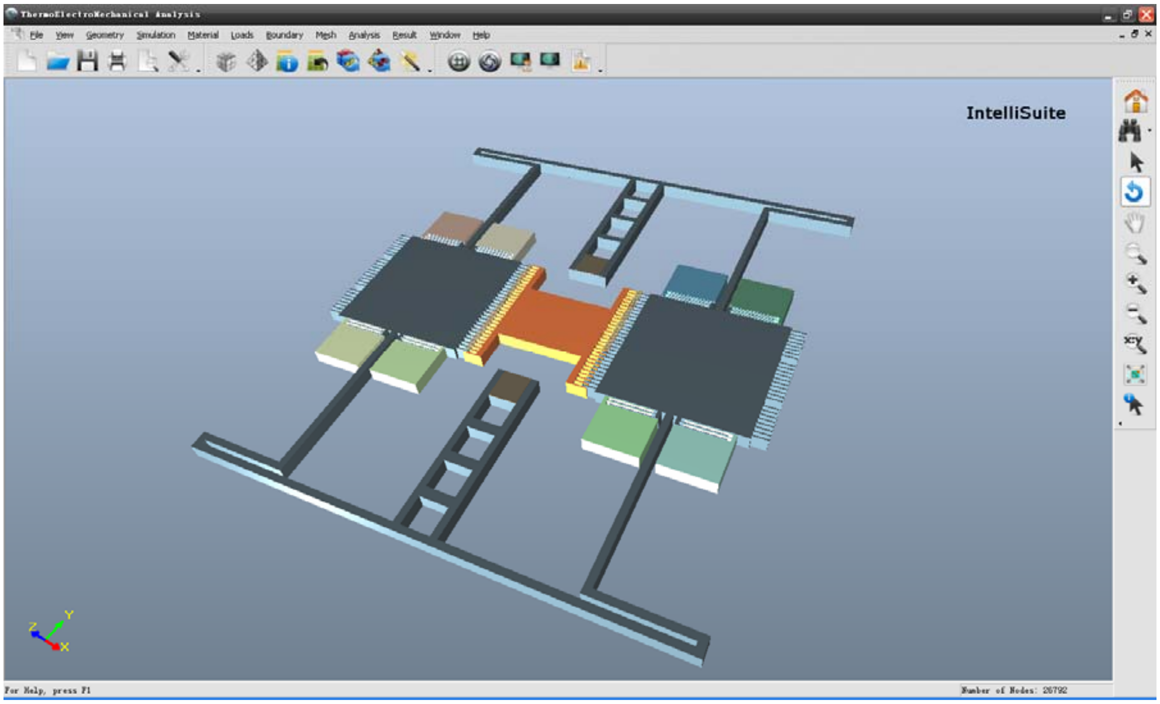


Figure 1 Single-Axis, Two-Moving-Mass Gyro

2. Element-based Design in SYNPLE

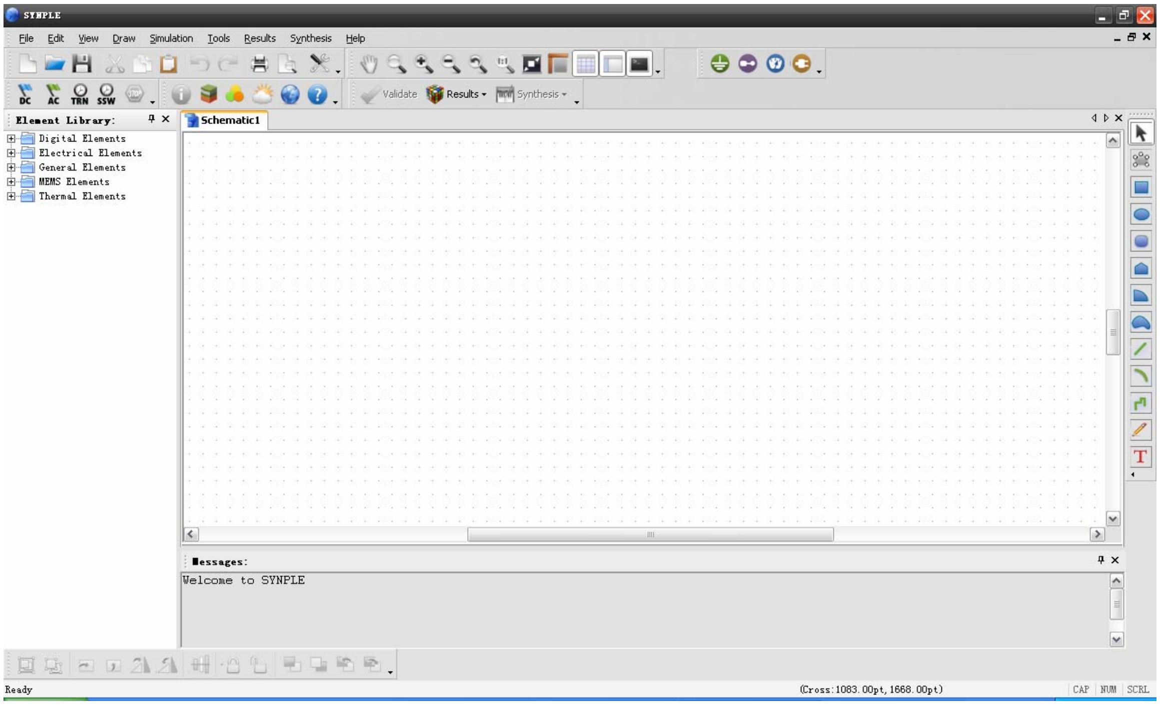
SYNPLE, IntelliSense's system-level simulator, can be used in a variety of ways. On one hand, MEMS elements like beams, plates, and comb drives can be used to build a device from scratch. The other method would be to import a macromodel from the FEA module. In both cases, the device can be wired up to a circuit that will be used to control the device. In this section, we will analyze a model that has been constructed from scratch with beams and plates. In Section 5, we will discuss the analysis of a macromodel.

2.1. Model Construction

Open SYNPLE.

Click *Start…Programs…IntelliSuite…SYNPLE*

The schematic window will open. Users will see the schematic window on the right, the element library on the left, and the message panel on the bottom as shown in Figure 2.



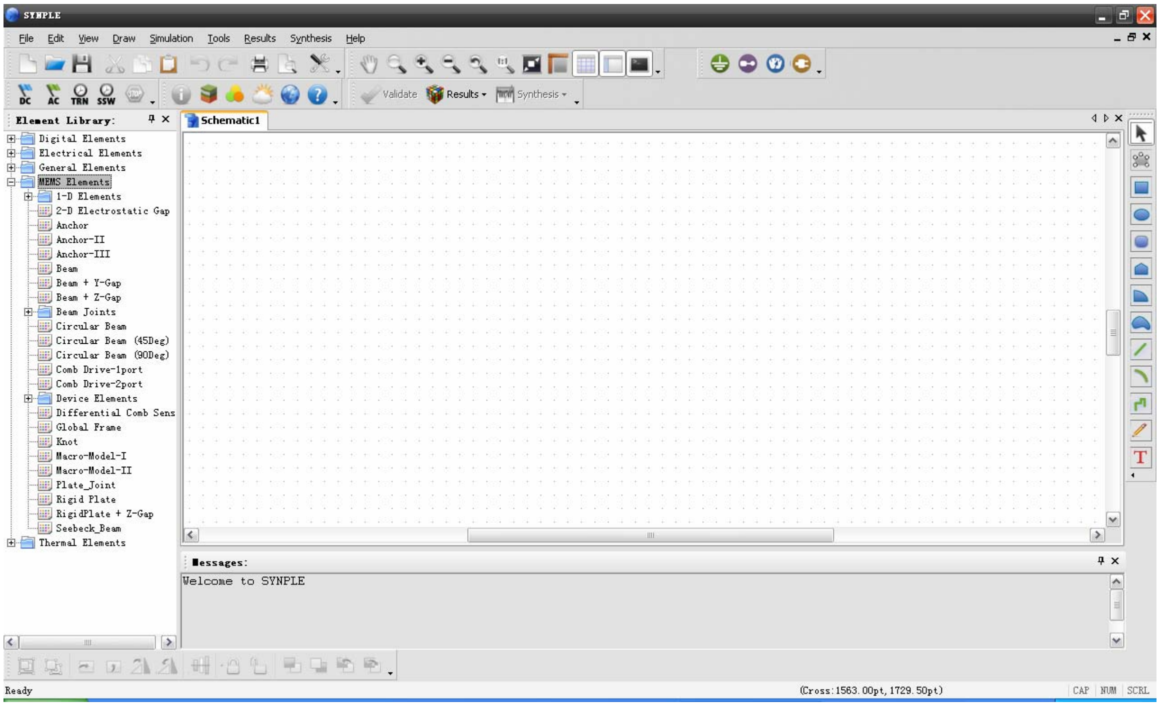
```
Welcome to SYNPLE
```

Figure 2 SYNPLE window

In the element library window on the left,

Click *MEMS Elements*

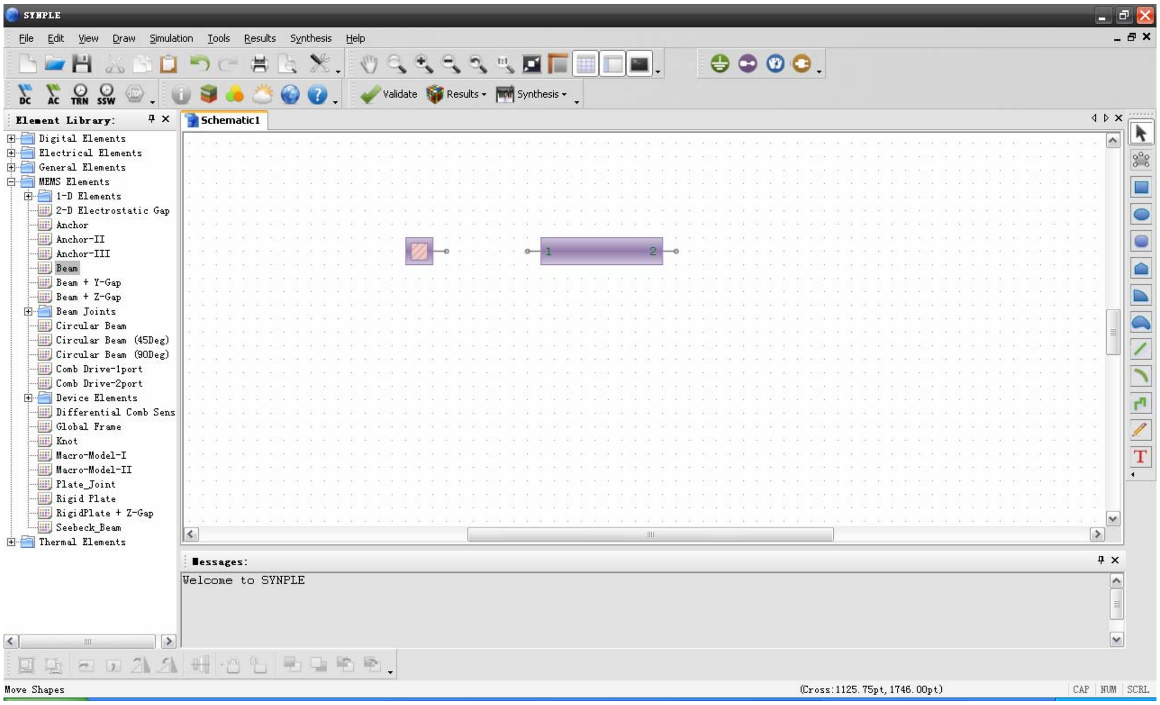
This will open the MEMS element library as shown in Figure 3.



Welcome to SYNPLE

Figure 3 MEMS Element Library

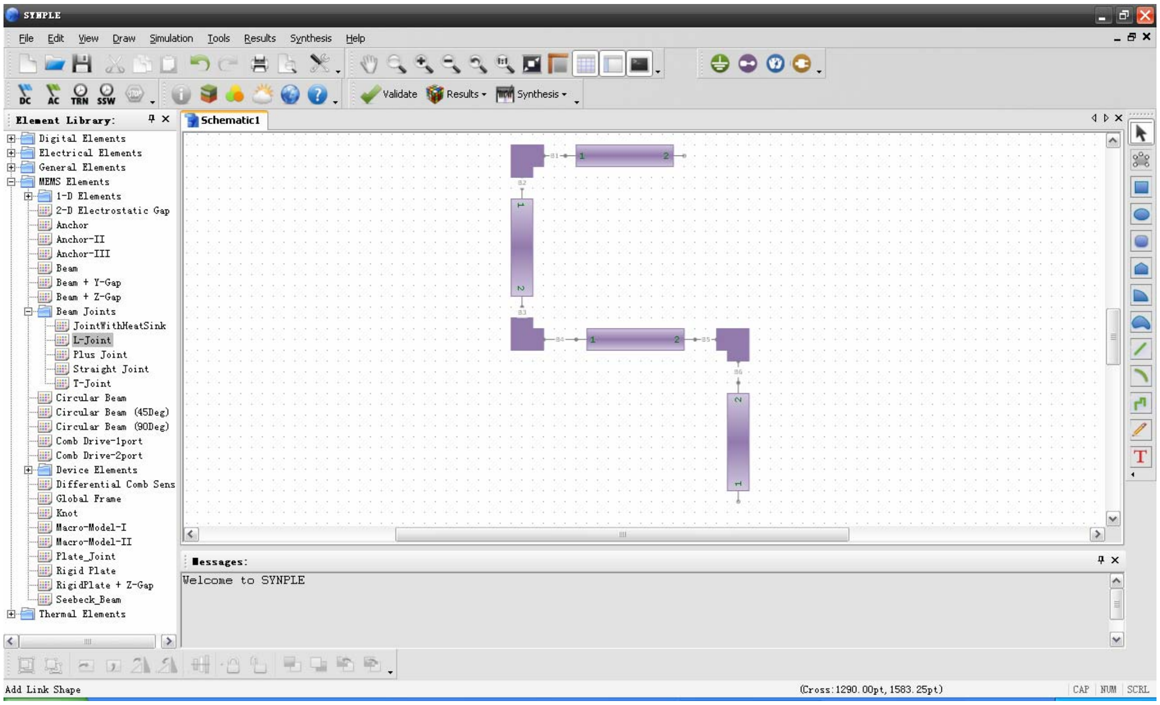
Click and drag an *Anchor* element into the Schematic window. Also, click and drag a *Beam* into the schematic window. Your SYNPLE Window should appear as shown in Figure 4.



Welcome to SYNPLE

Figure 4 Dragging Elements into Schematic Window

Now that you know how to add elements to the schematic window, we can start to build up the gyro device. Delete the elements you just created, and drag four *Beam* elements and three *L-Joint* elements into the drawing window (the *L-Joint* element can be found in the *Beam Joints* sub-menu). Connect these elements as shown in Figure 5. To connect a beam to a joint, drag one element so that its pin lines up with the pin of the other element. A red square will highlight the connection when the pins are lined up. You can also hover the mouse over a pin and a pencil icon will show up. Click and drag to draw a wire from one pin to another.



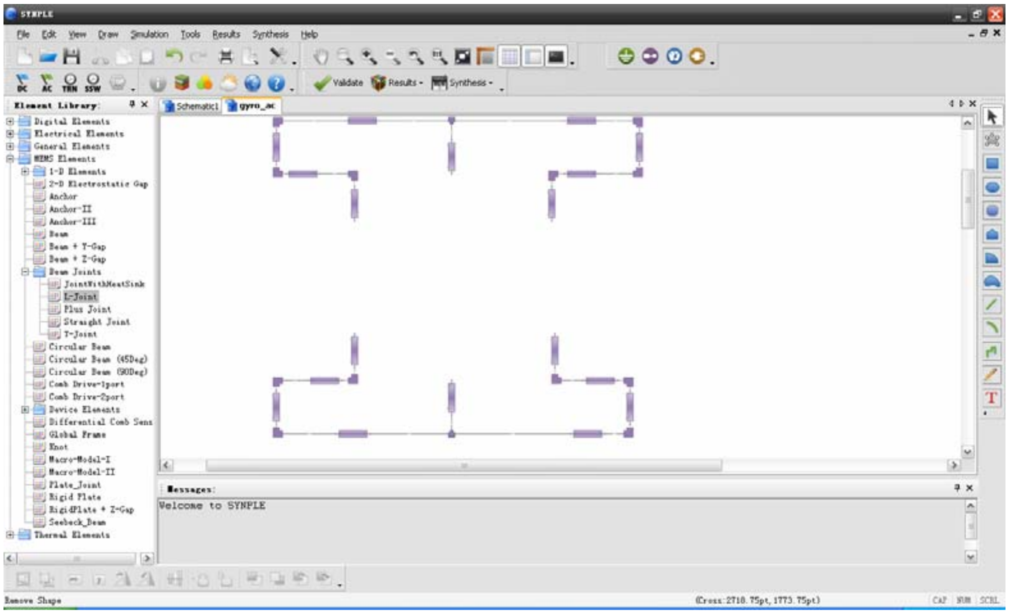
Welcome to SYNPLE

Figure 5 Connecting Beams

To rotate elements, you can use the *Rotate* buttons in the toolbar at the bottom of the window.

Figure 6 Format Toolbar

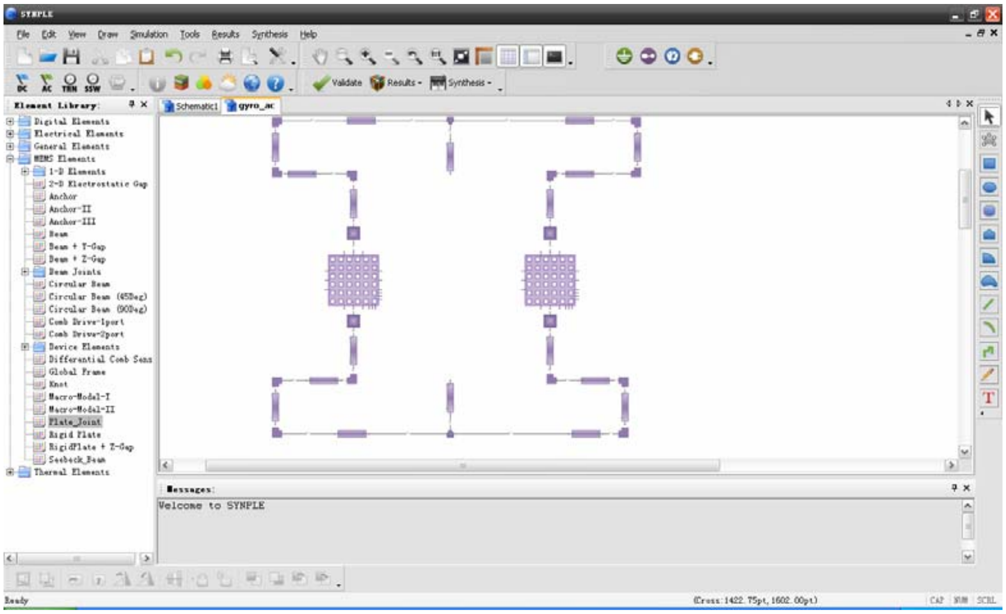
Continue connecting beam and joint elements in the schematic window until the structure looks like that shown in Figure 7. You can continue dragging elements into the window from the element library, or you can use the Copy/Paste commands.



Welcome to SYNPLE

Figure 7 Beam Elements

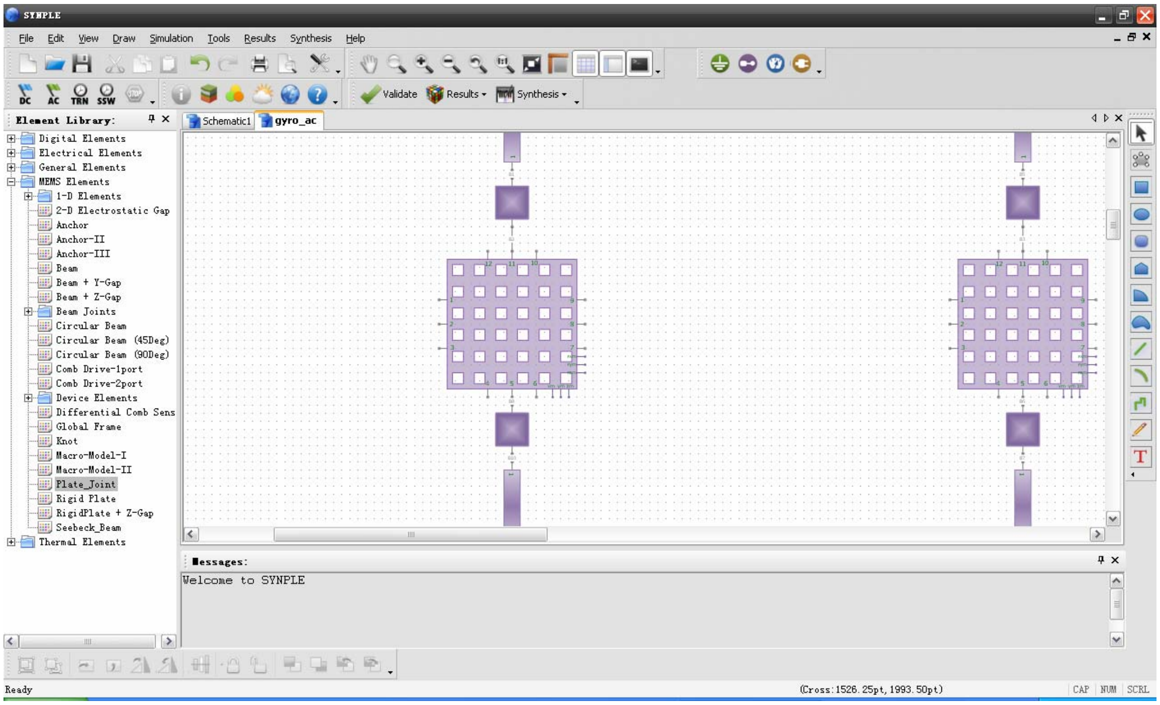
Drag two *Rigid Plate* elements and four *Plate Joint* elements into the schematic window.



Welcome to SYNPLE

Figure 8 Adding Rigid Plate Elements

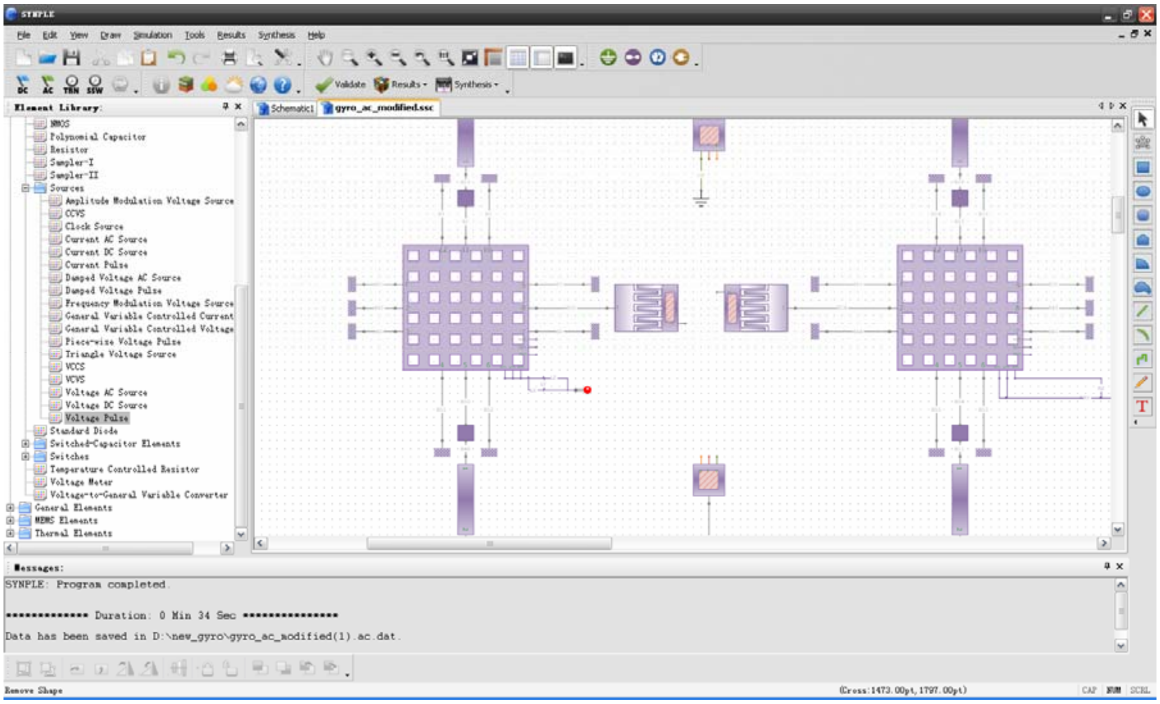
Connect the *Rigid Plate* elements to the *Beam* elements with the *Plate Joints.* Note that it is always necessary to use the joint elements when connecting beams and plates.



```
Welcome to SYNPLE
```

Figure 9 Connecting Beams and Plates

Next, we will add the comb drives. Drag two *Comb Drive-1port* elements into the schematic and connect them to the *Rigid Plate* elements as shown in Figure 10. Then, connect the nine other force pins on each plate to *Knot* elements as shown below. The *Knot* elements show that no forces or moments will be applied at these points.



```
SYNPLE: Program completed.

************* Duration: 0 Min 34 Sec ***************

Data has been saved in D:\new_gyro\gyro_ac_modified(1).ac.dat.
```

Figure 10 Comb Drive Elements

Drag two *Anchor_II* elements into the schematic window and connect them to the middle beams as shown below. It is possible to connect voltage, temperature, and power loads to this particular anchor. We need to define an electrical reference for the schematic, so select *Electrical Ground* from the *Draw* menu at the top of the screen, click in the schematic window to insert the element, and connect it to the voltage pin of the top anchor.

Drag a *Global Frame* element into the schematic window. This element allows you to transfer global orientation, angular velocities, and accelerations from the global (chip) frame of reference to the local frame of reference. (More information on this or any element can be found in the Elements Documentation in the *Help* menu.) You will also need to connect a *Constant* element to each of the pins of the *Global Frame* (note that one *Constant* element can be connected to multiple pins). The *Constant* element can be found in the *Sources* sub-menu under the *General Elements* library. In this case, the value of all the constants connected to the *Global Frame* will be left at zero.

Your schematic should now look like the one below.

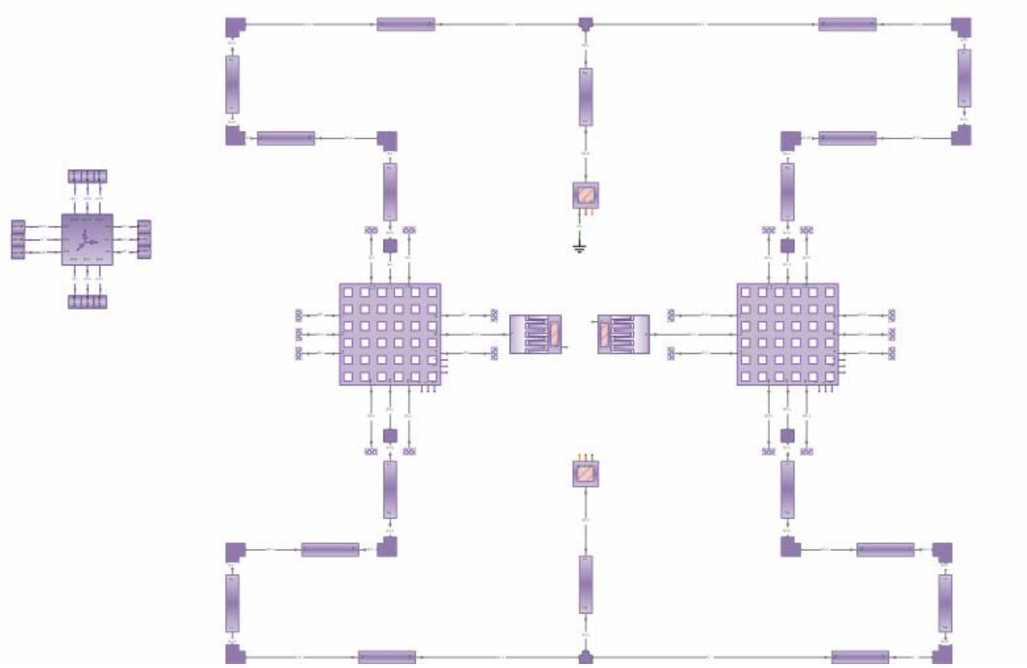


Figure 11 Structural Elements of the Gyro

Insert a *Voltage AC Source* element, located in the *Sources* sub-menu in the *Electrical Elements* library. Connect the positive pin to the voltage pin on each of the comb drive elements. Add another *Electrical Ground* (from the *Draw* menu) and connect it to the negative pin of the voltage source.

Finally, select *Draw…Output Probe,* and add an output probe near the bottom right-hand corner of each of the plate elements. Connect the output probe to the “xm”, ”ym”, and ”zm” pins. These pins represent the displacement of the center of the plate, and connecting them to the output probe means that we will be monitoring the displacement as an output signal.

Your schematic should now appear as shown below.

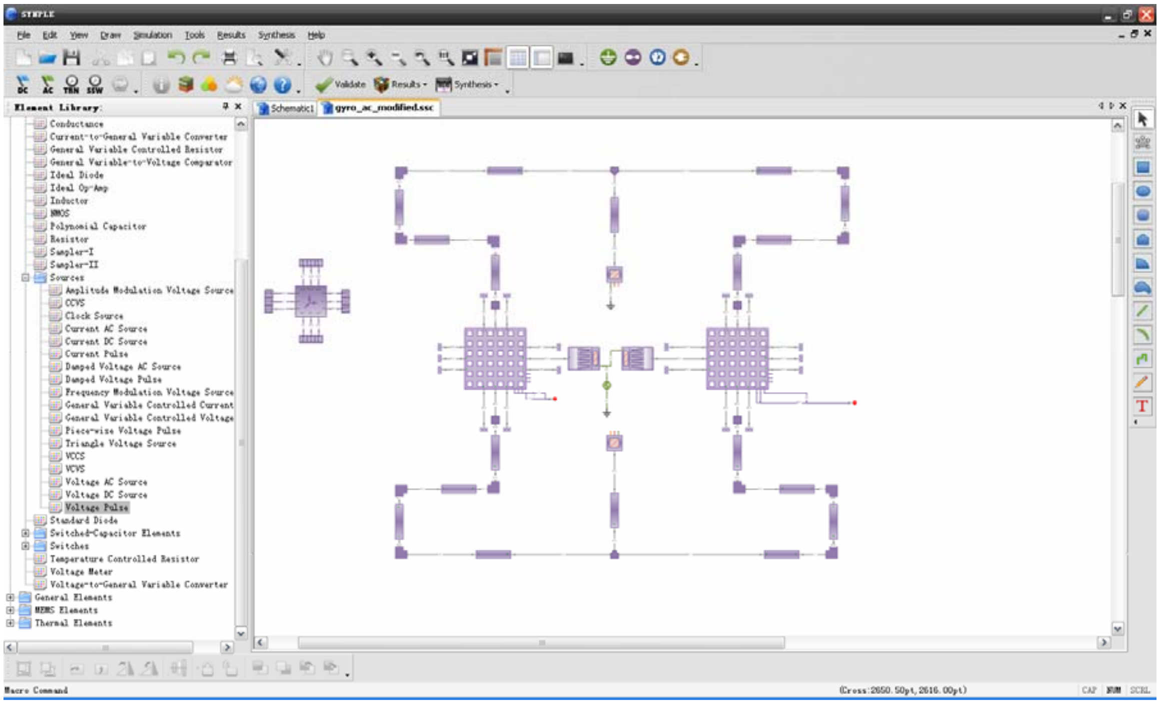
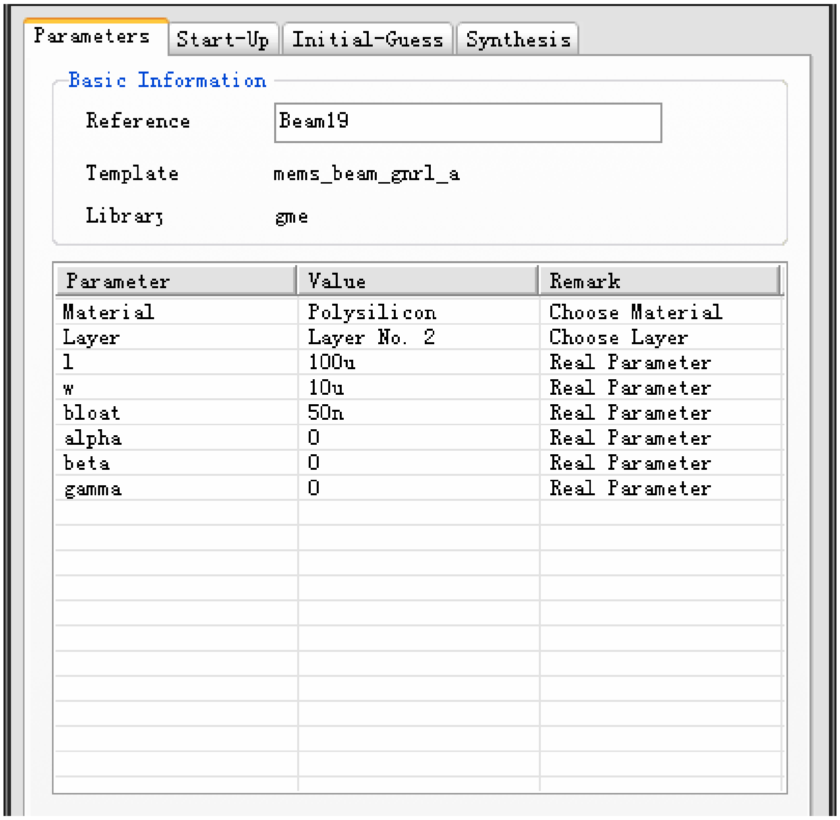


Figure 12 Gyro Model

You can double click on any element to bring up the *Device Properties* window where you can set up the properties for that element. For example, double-clicking on any *Beam* element will allow you to change the material properties, layer information, and dimensions of the beam.



Basic Information

Reference: Beam19

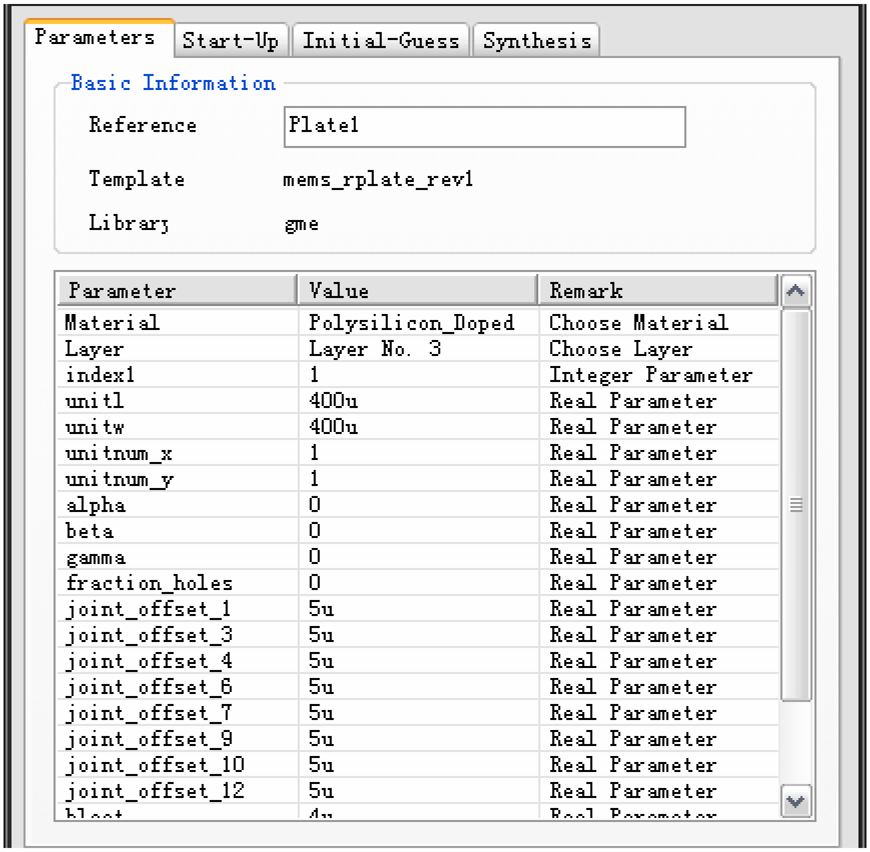
Template: mems_beam_gnrl_a

Library: gme

Parameter	Value	Remark
Material	Polysilicon	Choose Material
Layer	Layer No. 2	Choose Layer
l	100u	Real Parameter
w	10u	Real Parameter
bloat	50n	Real Parameter
alpha	0	Real Parameter
beta	0	Real Parameter
gamma	0	Real Parameter

Figure 13 Beam Parameters

For the *Rigid Plate* element, you can also add holes and define where each of the pins is located.



Reference Plate1

Template mems_rplate_rev1

Library gme

Parameter	Value	Remark
Material	Polysilicon_Doped	Choose Material
Layer	Layer No. 3	Choose Layer
index1	1	Integer Parameter
unitl	400u	Real Parameter
unitw	400u	Real Parameter
unitnum_x	1	Real Parameter
unitnum_y	1	Real Parameter
alpha	0	Real Parameter
beta	0	Real Parameter
gamma	0	Real Parameter
fraction_holes	0	Real Parameter
joint_offset_1	5u	Real Parameter
joint_offset_3	5u	Real Parameter
joint_offset_4	5u	Real Parameter
joint_offset_6	5u	Real Parameter
joint_offset_7	5u	Real Parameter
joint_offset_9	5u	Real Parameter
joint_offset_10	5u	Real Parameter
joint_offset_12	5u	Real Parameter

Figure 14 Rigid Plate Parameters

For a detailed description of each parameter associated with each element, you can refer to the Elements Documentation in the *Help* menu. From this point forward, we will use a saved schematic that includes all of the correct element parameter settings. For further information and any practice on creating SYNPLE schematics from scratch, please refer to the “Getting Started with SYNPLE” Guide in the *Help* menu.

Click *File…Open…*

Open the following file: `<Installation Directory>/IntelliSuite/Training/ /Application_Notes/Gyro/SYNPLE_Models/Element_Models/gyro_ac.ssc`.

This file should appear as shown in Figure 15.

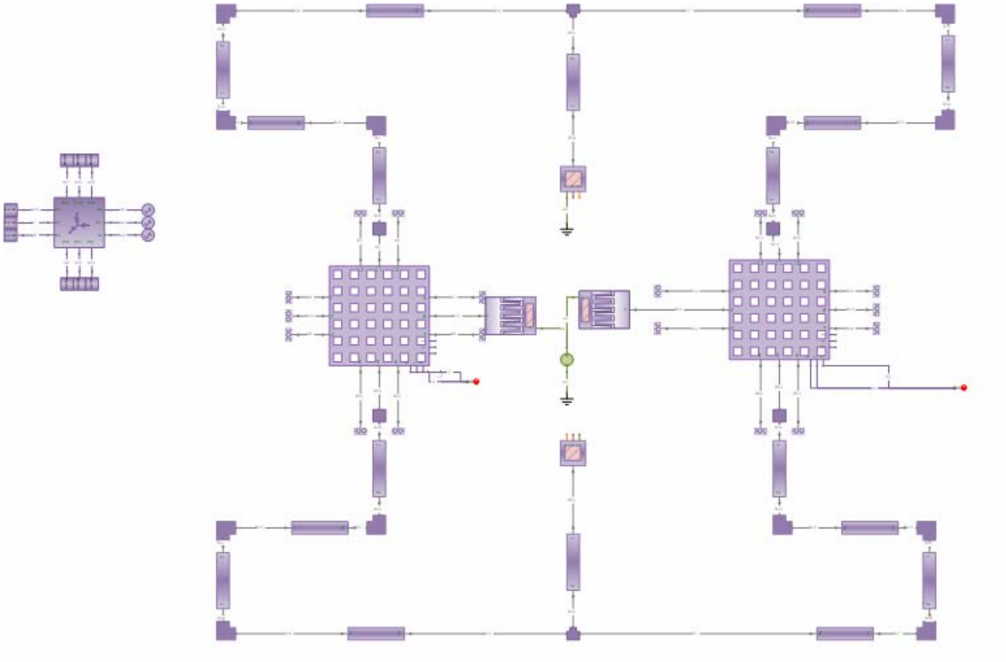
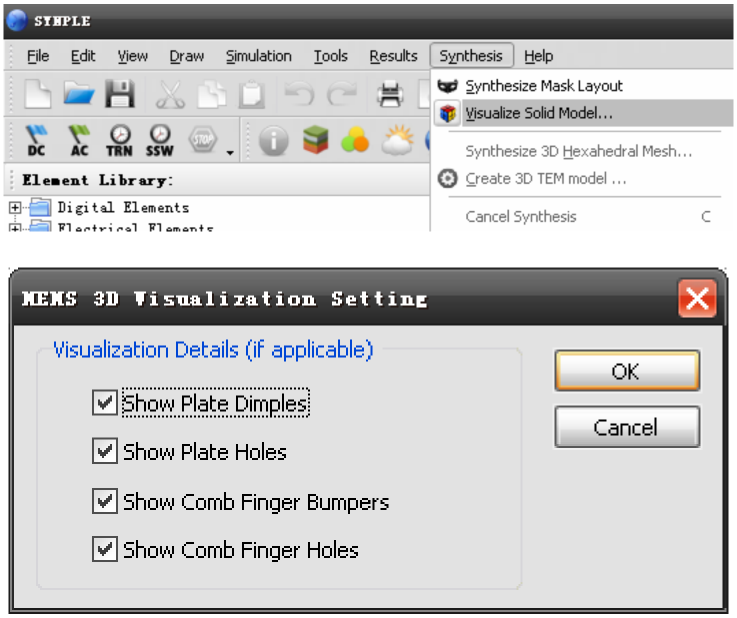


Figure 15 Completed Gyro Model

You can double click any element to view the parameter settings. To get a better idea of what this model really looks like, we can quickly view a solid model representation of the schematic.

Click *Synthesis…Visualize Solid Model…*

Click *OK*



Visualization Details (if applicable)

☑ Show Plate Dimples

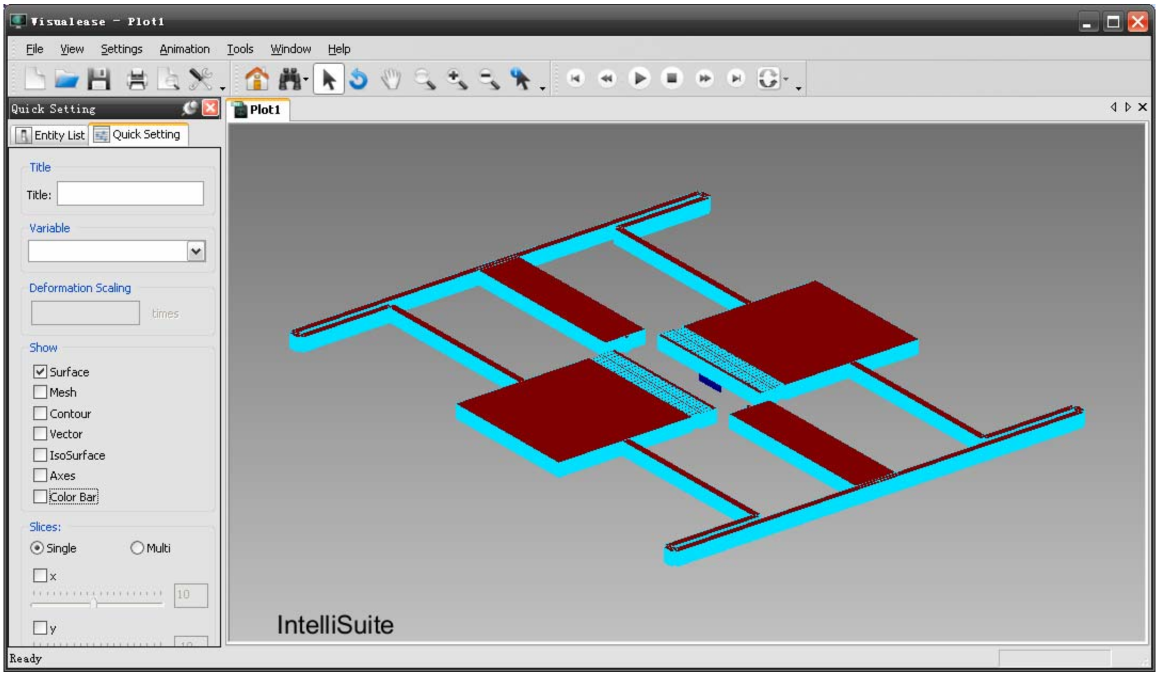
☑ Show Plate Holes

☑ Show Comb Finger Bumpers

☑ Show Comb Finger Holes

Figure 16 Visualize Solid Model

A 3D model will appear as shown in Figure 17.



Title

Title:

Variable

Deformation Scaling

times

Show

☑ Surface
☐ Mesh
☐ Contour
☐ Vector
☐ IsoSurface
☐ Axes
☐ Color Bar

Slices:

● Single ○ Multi

☐ x 10

☐ y

Figure 17 3D Model

When you are finished examining the model, close VisualEase and return to the SYNPLE window.

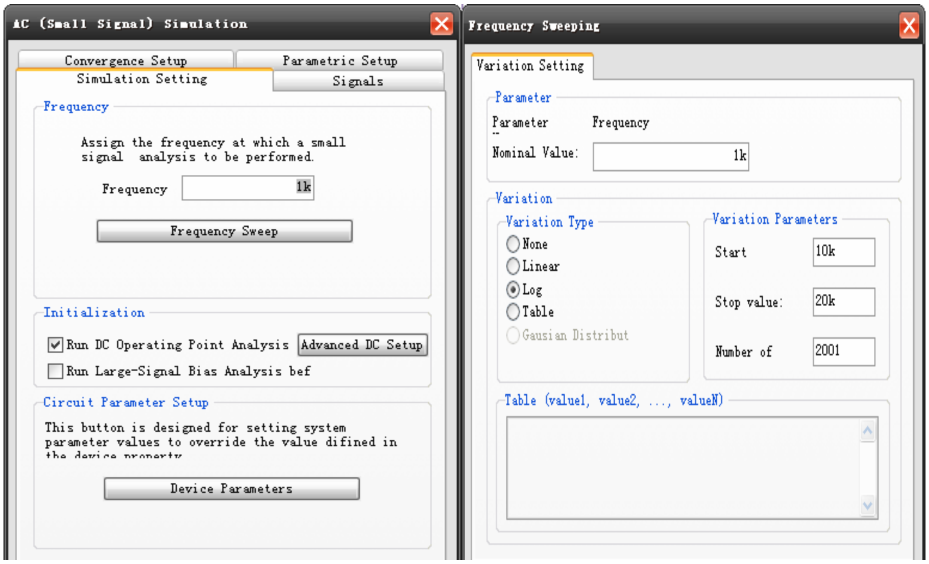
2.2. AC Analysis

We will perform an AC analysis to view the frequency response of the structure.

Click *Simulation…AC Analysis*

The simulation settings are already set. To view the simulation settings,

Click *Frequency Sweep*



Assign the frequency at which a small signal analysis to be performed.

Frequency: 1k

☑ Run DC Operating Point Analysis

☐ Run Large-Signal Bias Analysis bef

This button is designed for setting system parameter values to override the value difined in the device property

Parameter: Frequency

Nominal Value: 1k

● Log

Start: 10k

Stop value: 20k

Number of: 2001

Figure 18 AC Simulation Settings

You can see that we will be analyzing 2001 points between 10 kHz and 20 kHz. To run the AC simulation, return to the AC Simulation window and click *OK*.

When the simulation is completed, a Plot Manager window will appear. Here we can view magnitude and phase plots of the output signals we chose to monitor. The variables represent the x-, y-, and z-displacements of the left proof mass, and *RX*, *RY* and *RZ* correspond to the displacements of the right proof mass. Select *mag_of_LX_ac* and click “Open in WaveRunner” to view the plot as shown below.

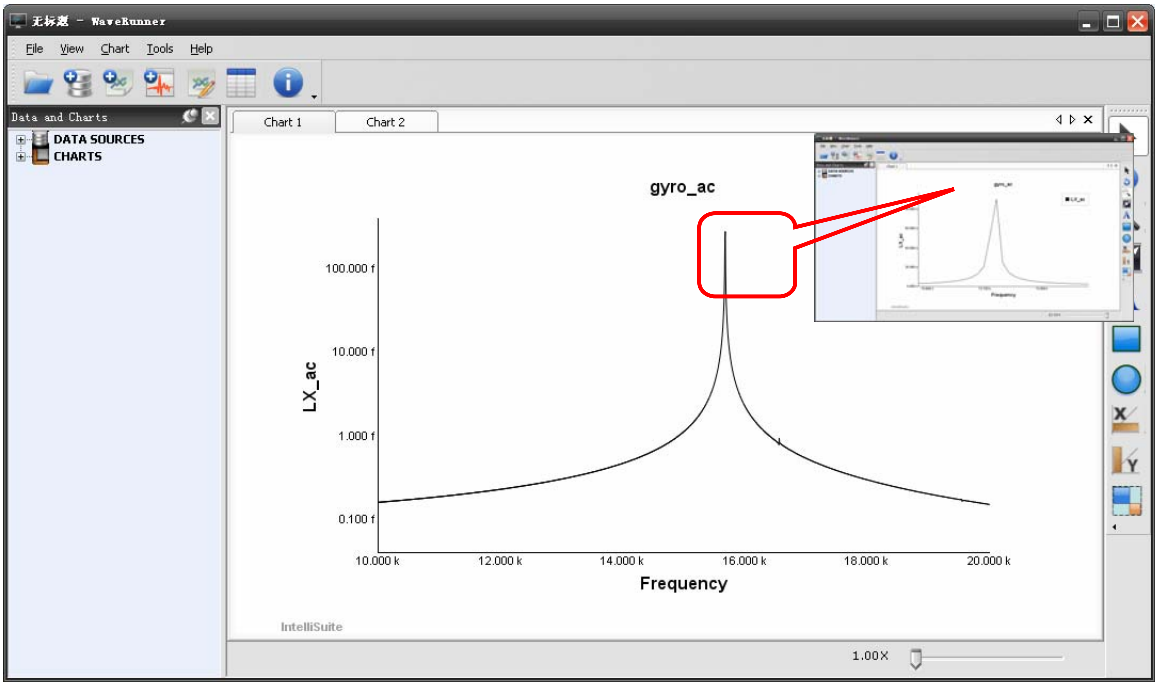


Figure 19 Frequency Response in X-direction

The modal frequency in the x-direction is about 15.7 kHz. To view the frequency response in the y-direction, open *mag_of_LY_ac* back in the Plot Manager window.

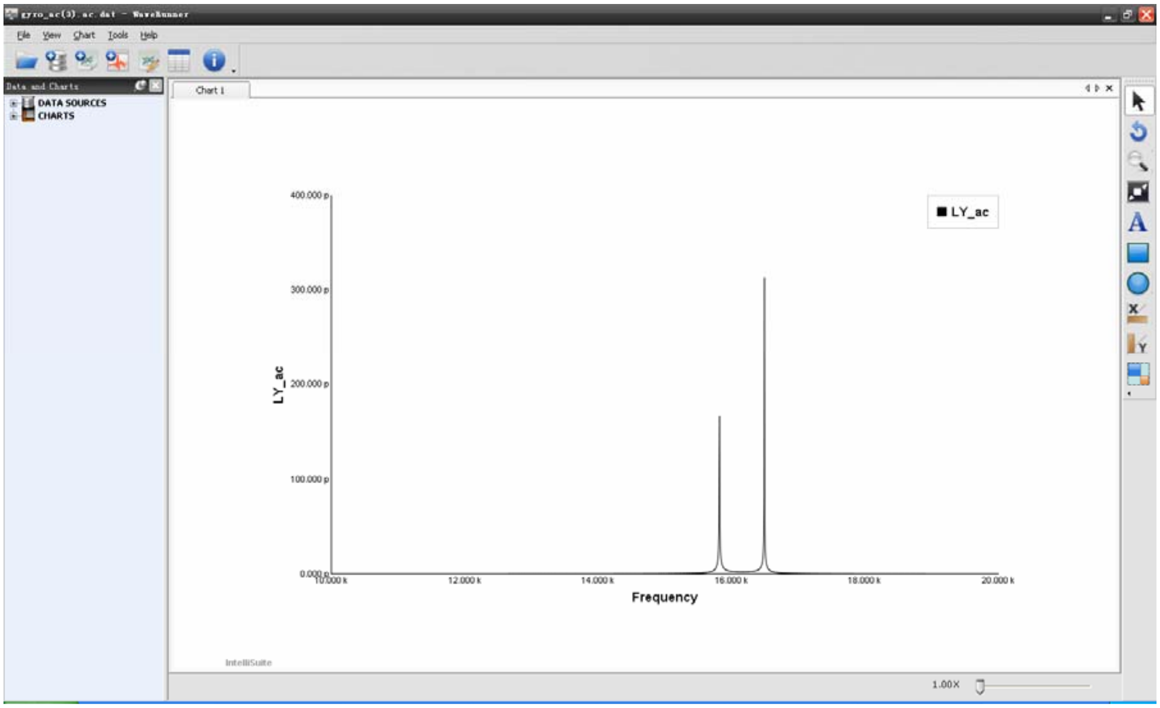


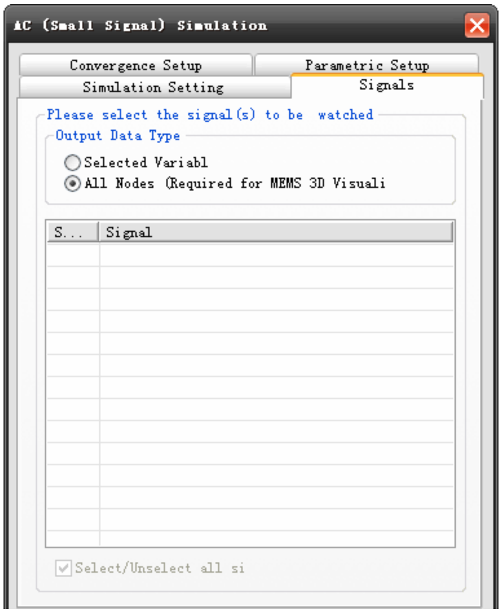
Figure 20 Frequency Response in Y-direction

The modal frequency in the y-direction is about 16.4 kHz.

Mode shapes and result animations can also be viewed after AC and Transient simulations. As long as the *All Nodes* button is selected in the simulation setup, the *MEMS 3D Visualization* menu item will be active after the simulation has run.

Click *Simulation…AC Analysis*
Click *Signals*

We will need results from all of the output signals to get an accurate 3D visualization. Select *All Nodes* under the Signals tab in the DC Simulation dialog.



Please select the signal (s) to be watched
Output Data Type
○ Selected Variabl
● All Nodes (Required for MEMS 3D Visuali
☑ Select/Unselect all si

Figure 21 Select all nodes for MEMS 3D Visualization

Click *OK* to run the simulation. After the simulation is complete, select *MEMS 3D Visualization* in the Results menu.

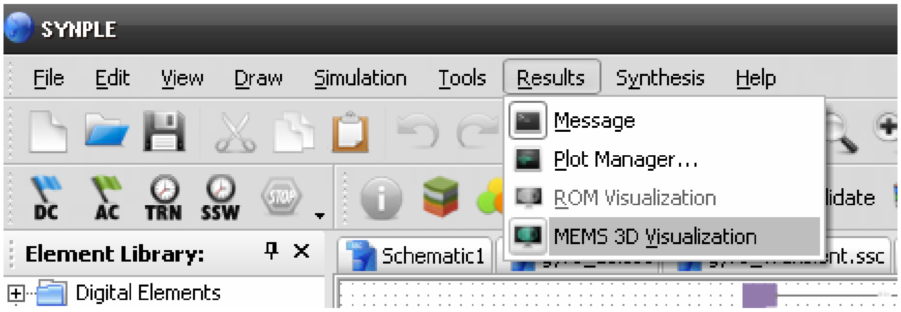
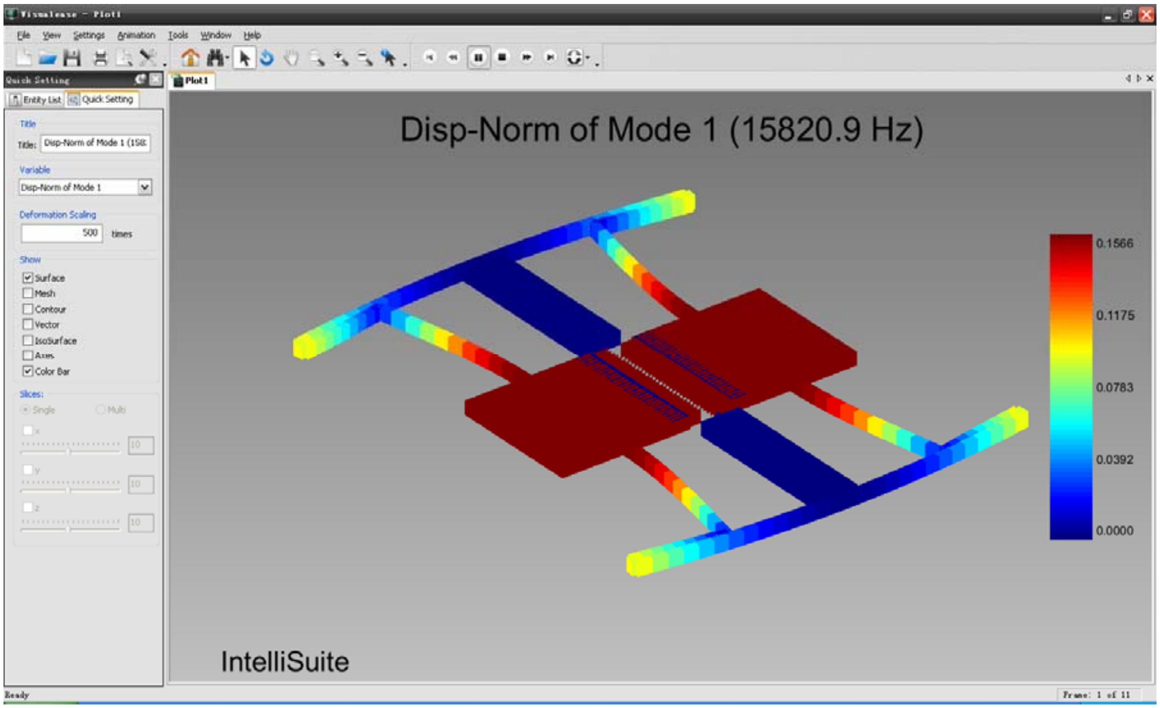


Figure 22: 3D Visualization in Results Menu

When VisualEase opens, choose "*Disp-Norm of Mode 1*" under the "Variable" box to view the mode shape. You can also enter a scaling factor in the Deformation Scaling box to get a better view of the behavior.



Title: Disp-Norm of Mode 1 (158

Variable: Disp-Norm of Mode 1

Deformation Scaling: 500 times

Show:
- ☑ Surface
- ☐ Mesh
- ☐ Contour
- ☐ Vector
- ☐ IsoSurface
- ☐ Axes
- ☑ Color Bar

Slices: ● Single ○ Multi
- ☐ x 10
- ☐ y 10
- ☐ z 10

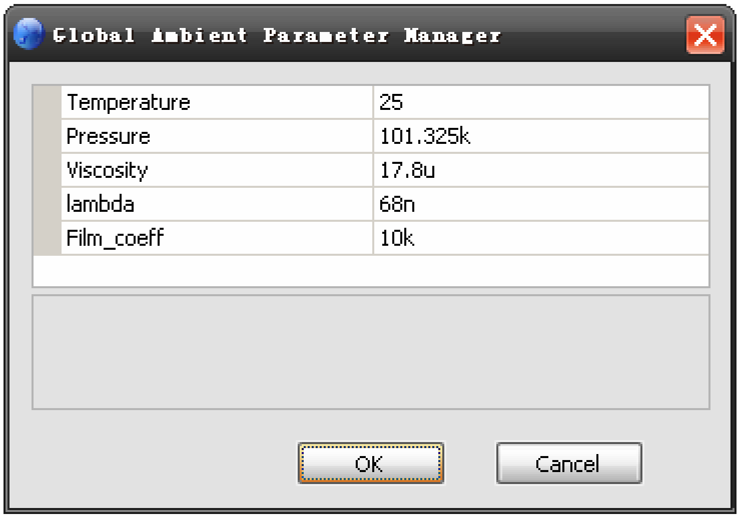
Figure 23: 3D Visualization of Mode 1

2.2.1 Damping and Quality factor

In this section we will determine the damping factor and quality factor for the gyro structure. The interaction of the moving comb fingers and the surrounding fluid is the main source of damping for this structure. The damping factor is dependent on ambient parameters such as pressure, temperature and fluid viscosity. We will first examine the effect of these ambient parameters on the damping factor, then extract the quality factor.

Click *Tools…Ambient Parameter Manager*

The Ambient Parameter Manager with default parameters will appear as shown below.



Temperature	25
Pressure	101.325k
Viscosity	17.8u
lambda	68n
Film_coeff	10k

Figure 24 Ambient Parameter Manager

In this case, we will not set up the ambient parameters in this window. Rather, we will run a parametric analysis to view the device response using multiple different values for the ambient pressure.

Click *Simulation…AC Analysis*

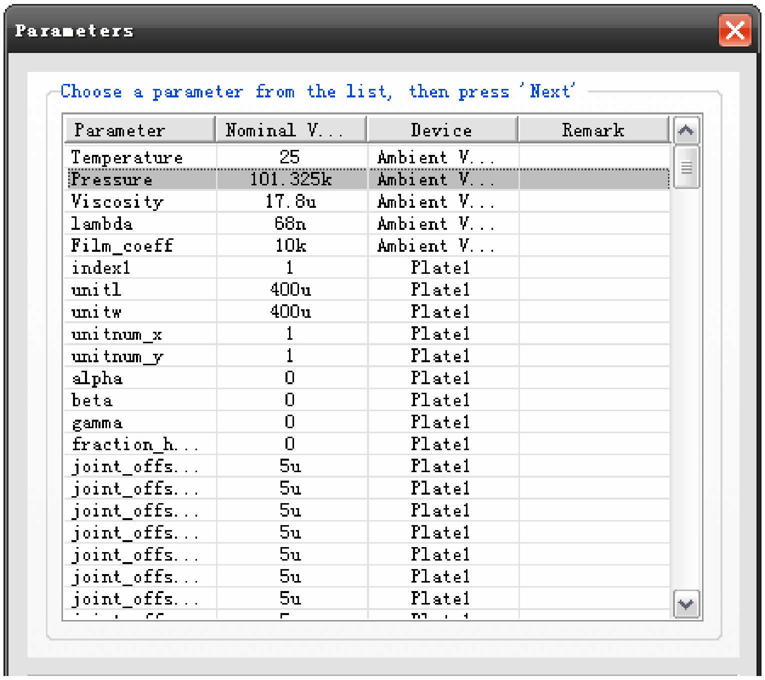
Click *Parametric Setup* tab

Click "+" button to add a parameter

Click *Pressure* as shown in Figure 22

Click *Next*

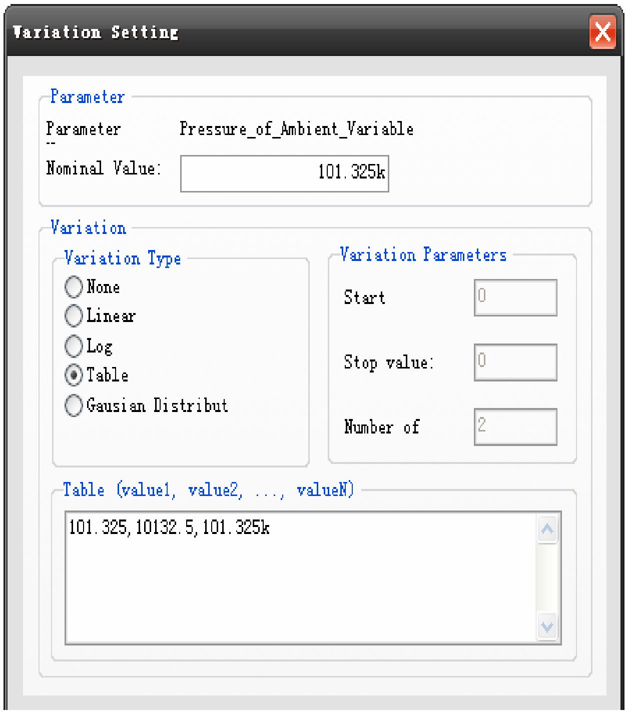
Select *Table* as the Variation Type as shown in Figure 23. Input "101.325, 10132.5, 101.325k" (values in Pa) into the Table dialog box.



Choose a parameter from the list, then press 'Next'

Parameter	Nominal V...	Device	Remark
Temperature	25	Ambient V...	
Pressure	101.325k	Ambient V...	
Viscosity	17.8u	Ambient V...	
lambda	68n	Ambient V...	
Film_coeff	10k	Ambient V...	
index1	1	Plate1	
unitl	400u	Plate1	
unitw	400u	Plate1	
unitnum_x	1	Plate1	
unitnum_y	1	Plate1	
alpha	0	Plate1	
beta	0	Plate1	
gamma	0	Plate1	
fraction_h...	0	Plate1	
joint_offs...	5u	Plate1	
joint_offs...	5u	Plate1	
joint_offs...	5u	Plate1	
joint_offs...	5u	Plate1	
joint_offs...	5u	Plate1	
joint_offs...	5u	Plate1	
joint_offs...	5u	Plate1	

Figure 25 Selecting Parameters



Parameter

Parameter -- Pressure_of_Ambient_Variable

Nominal Value: 101.325k

Variation

Variation Type

○ None

○ Linear

○ Log

● Table

○ Gausian Distribut

Variation Parameters

Start 0

Stop value: 0

Number of 2

Table (value1, value2, ..., valueN)

101.325, 10132.5, 101.325k

Figure 26 Parametric Settings

Click *Finish,* then click *OK* to run the simulation.

After the simulation is complete, select *mag_of_LX_ac* in the Plot Manager and click "Open in WaveRunner".

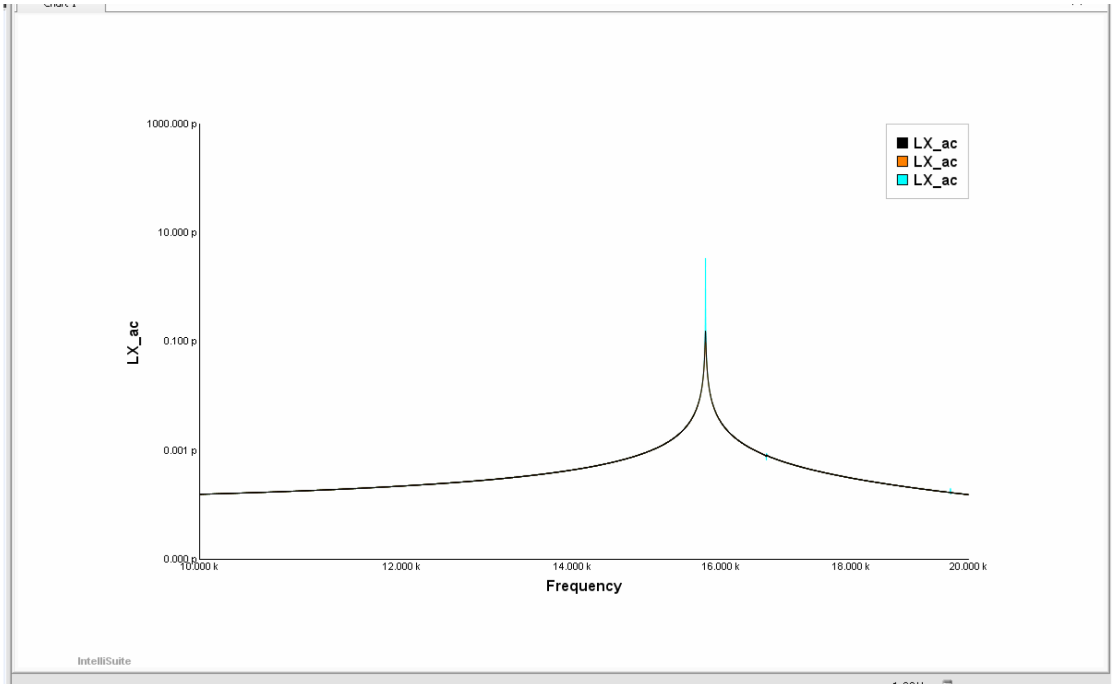


Figure 27 Frequency Response

You can see that the quality factor (peak magnitude divided by DC magnitude) varies with the changes in ambient pressure. You can use the Max-Min button in the toolbar to display maximum and minimum values of each of the curves, and you can use these values to determine the quality factor. At a pressure of 1 atm, the Q factor is about 630. At 0.1 atm, the Q factor is about 1,000, and at 0.001 atm, it is about 22,000.

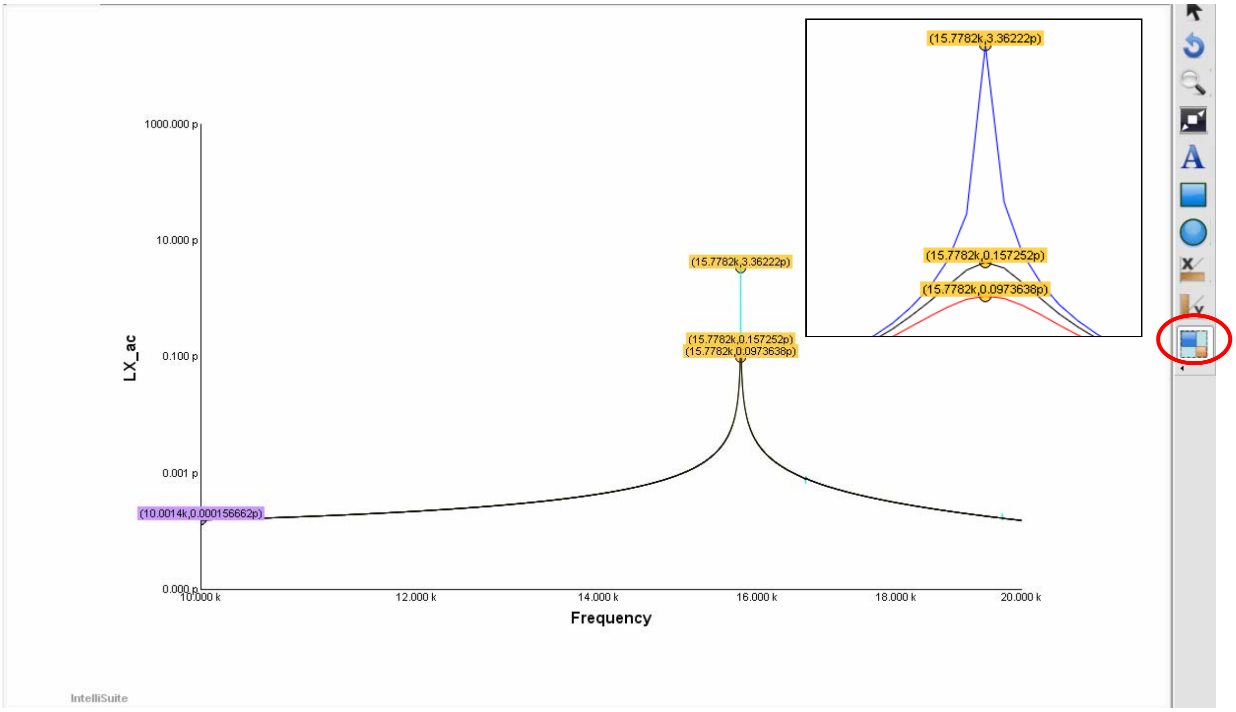


Figure 28 Max/Min Values

2.3. DC analysis

Open the following file: `<InstallationDirectory>/IntelliSuite/Training/Application_Notes/Gyro/SYNPLE_Models/Element_Models/gyro_DC.ssc`.

The DC schematic is shown below in Figure 26.

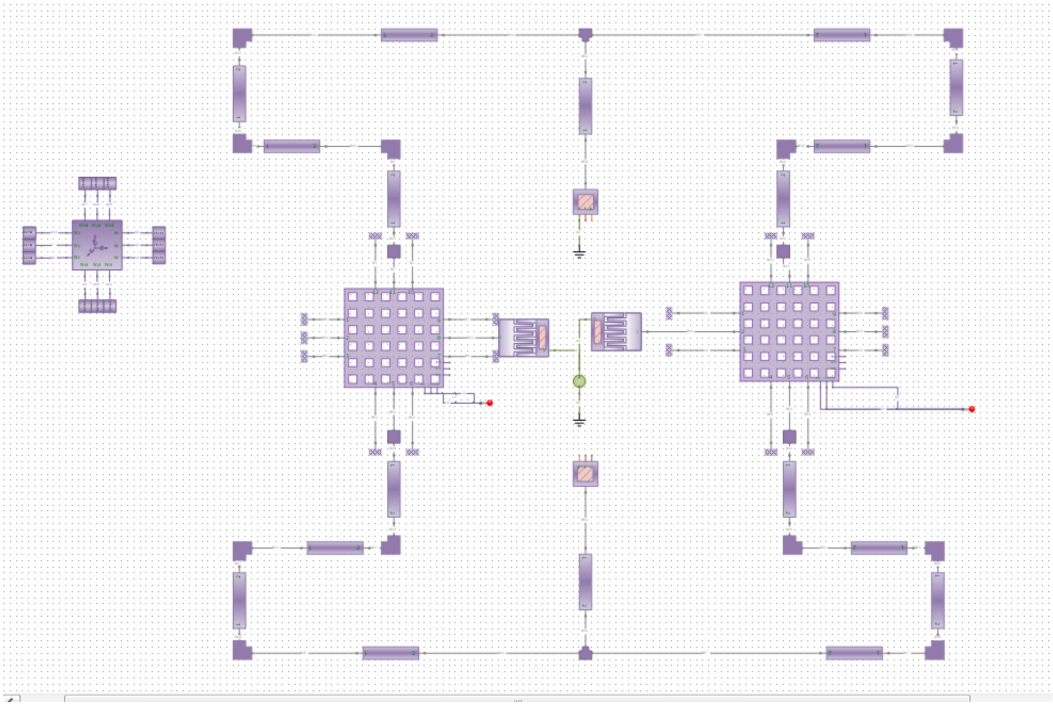
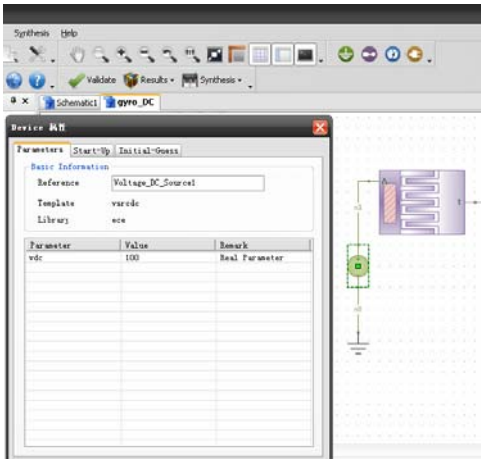


Figure 29 DC Schematic

This is the same schematic that we used for the AC simulations, but a DC voltage load will be applied to the comb drives. Double-click the green Voltage Source element and make sure the load is set to 100 V.



Reference Voltage_DC_Source1
Template vsrcdc
Library ece

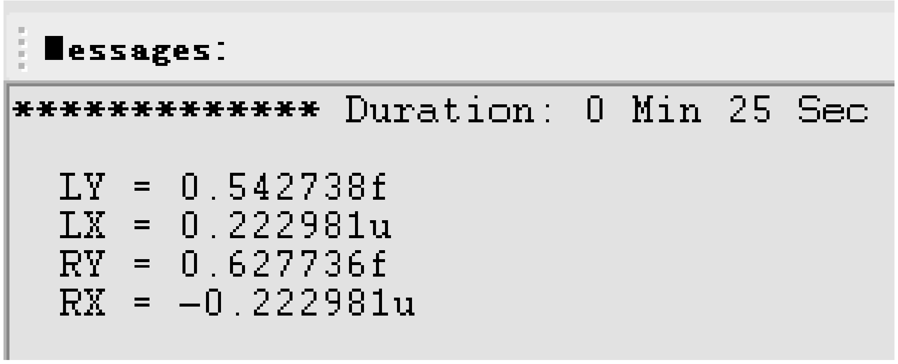
Parameter	Value	Remark
vdc	100	Real Parameter

Figure 30 DC Voltage Source

To run the analysis,

Click *Simulation…DC Analysis*

After the simulation runs, the DC displacement will show up in the message window at the bottom of the screen.



```
************** Duration: 0 Min 25 Sec

  LY = 0.542738f
  LX = 0.222981u
  RY = 0.627736f
  RX = -0.222981u
```

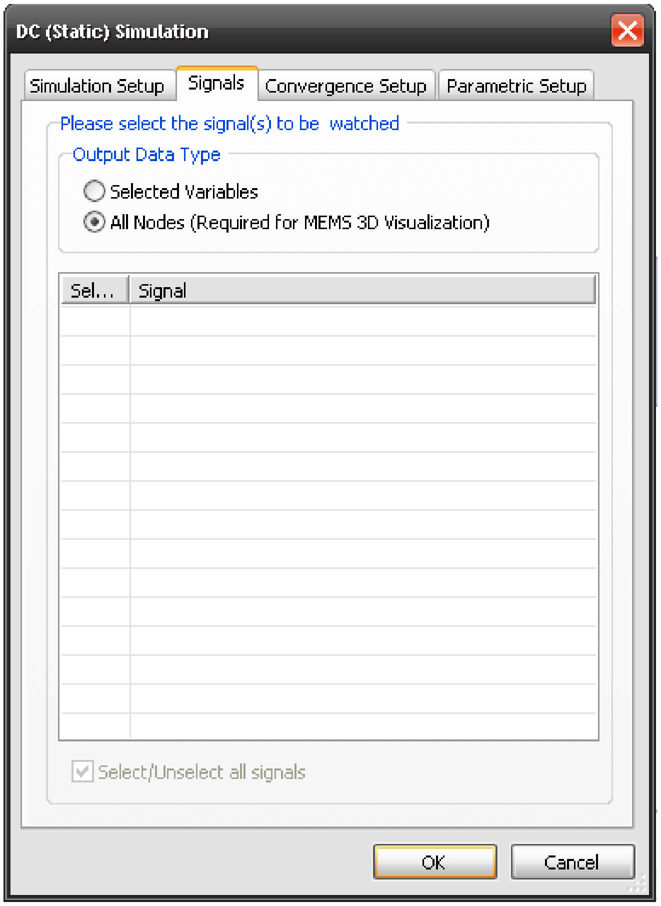
Figure 31 DC Results

The x-displacement *LX* or *RX* is about 0.22 um. Later, we will compare these results to those of our finite element model and reduced-order macromodel.

We can also view simulation results using VisualEase. First, we'll need to run the simulation once again with some different settings.

Click *Simulation…DC Analysis*

We will need results from all of the output signals to get an accurate 3D visualization. Select *All Nodes* under the Signals tab in the DC Simulation dialog.



○ Selected Variables
● All Nodes (Required for MEMS 3D Visualization)

☑ Select/Unselect all signals

Figure 32: DC Simulation Dialog

Click *OK* to run the simulation. After the simulation is complete, select *MEMS 3D Visualization* in the Results menu.

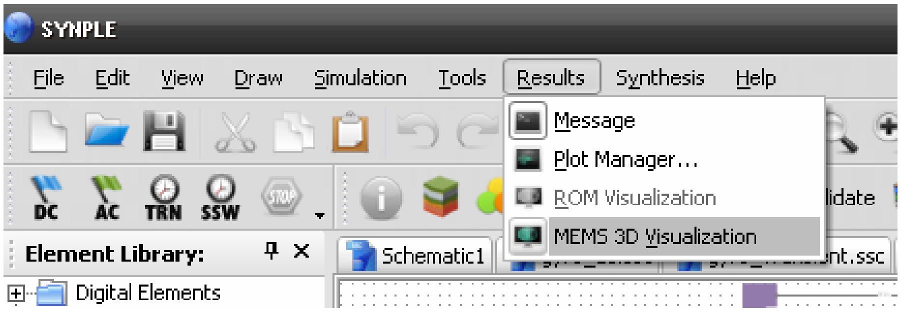
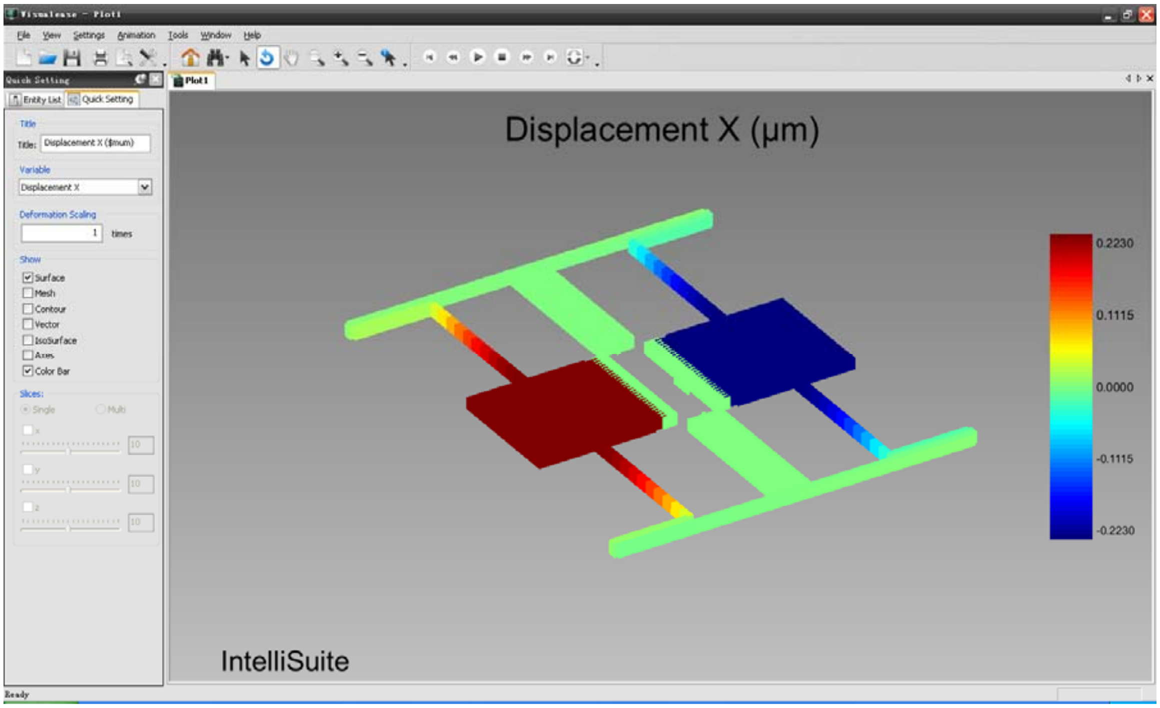


Figure 33: 3D Visualization in Results Menu

When VisualEase opens, choose *Displacement X* under the Variable box to view the x-displacement. You can also enter a scaling factor in the Deformation Scaling box to get a better view of the behavior.



Title

Title: Displacement X ($mum)

Variable

Displacement X

Deformation Scaling

1 times

Show

☑ Surface
☐ Mesh
☐ Contour
☐ Vector
☐ IsoSurface
☐ Axes
☑ Color Bar

Slices:

● Single ○ Multi

☐ x 10
☐ y 10
☐ z 10

Figure34: Gyro in VisualEase

2.4. Transient Analysis

Open the following file: `<Installation Directory>/IntelliSuite/Training/ /Application_Notes/Gyro/SYNPLE_Models/Element_Models/gyro_transient.ssc`.

The model will appear as shown below.

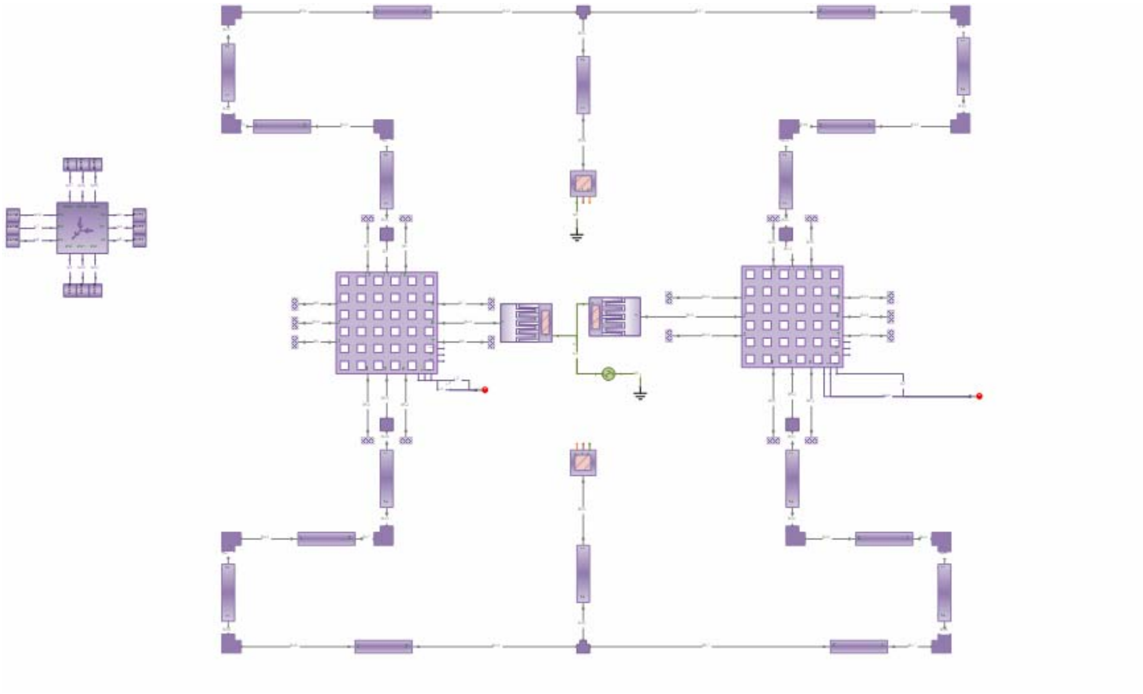
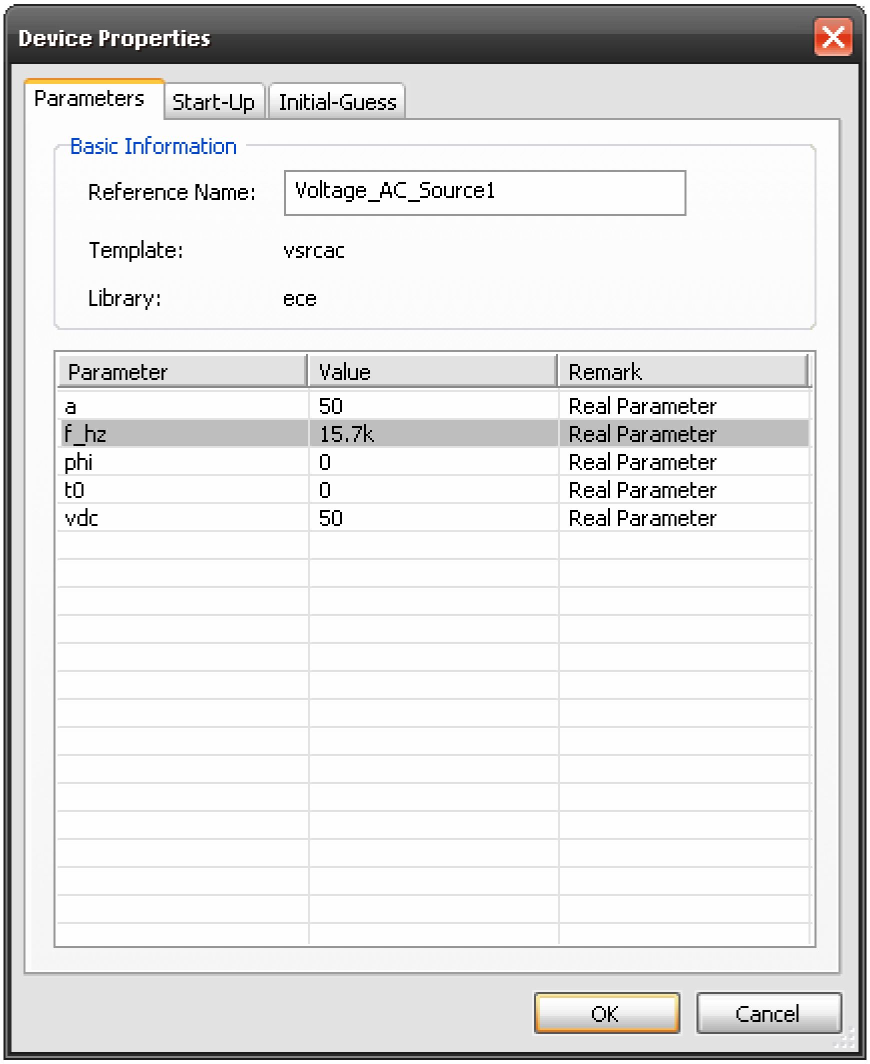


Figure 35 Transient Schematic

You can see that an AC Voltage Source is applied to the comb drives. Double-click on the element to view the parameter settings.



Reference Name: Voltage_AC_Source1

Template: vsrcac

Library: ece

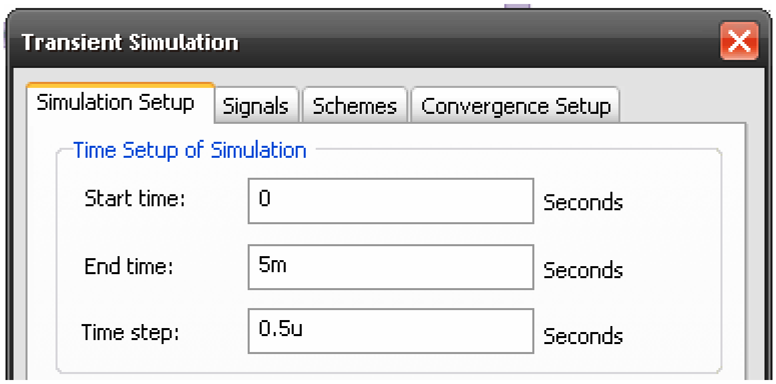
Parameter	Value	Remark
a	50	Real Parameter
f_hz	15.7k	Real Parameter
phi	0	Real Parameter
t0	0	Real Parameter
vdc	50	Real Parameter

Figure 36 AC Voltage Settings

To run the simulation,

Click *Simulation…Transient Analysis*

You will see that the simulation is set up to run for 5 milliseconds with a time step of 0.5 microseconds.



Start time: 0 Seconds

End time: 5m Seconds

Time step: 0.5u Seconds

Figure 37 Transient Settings

Click *OK*

After the simulation is complete, select *LX* in the Plot Manager and click "Open in WaveRunner". A graph of the transient response in the x-direction will appear.

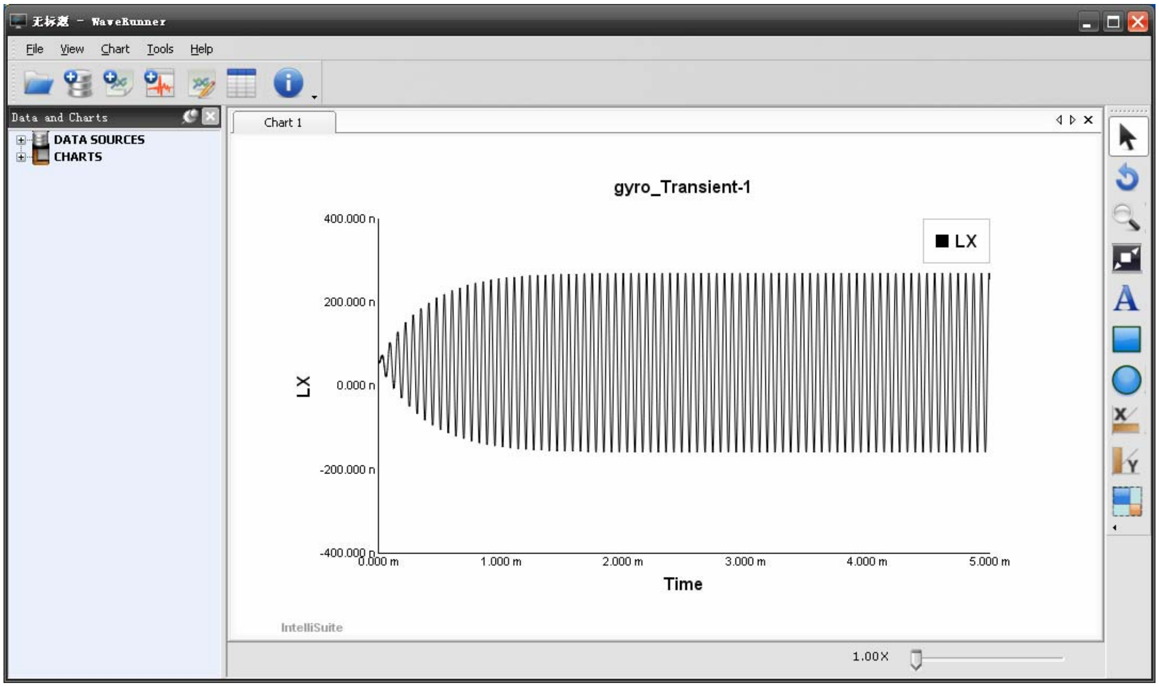


Figure 38 Transient Response - X

Open *LY* to view the response in the y-direction.

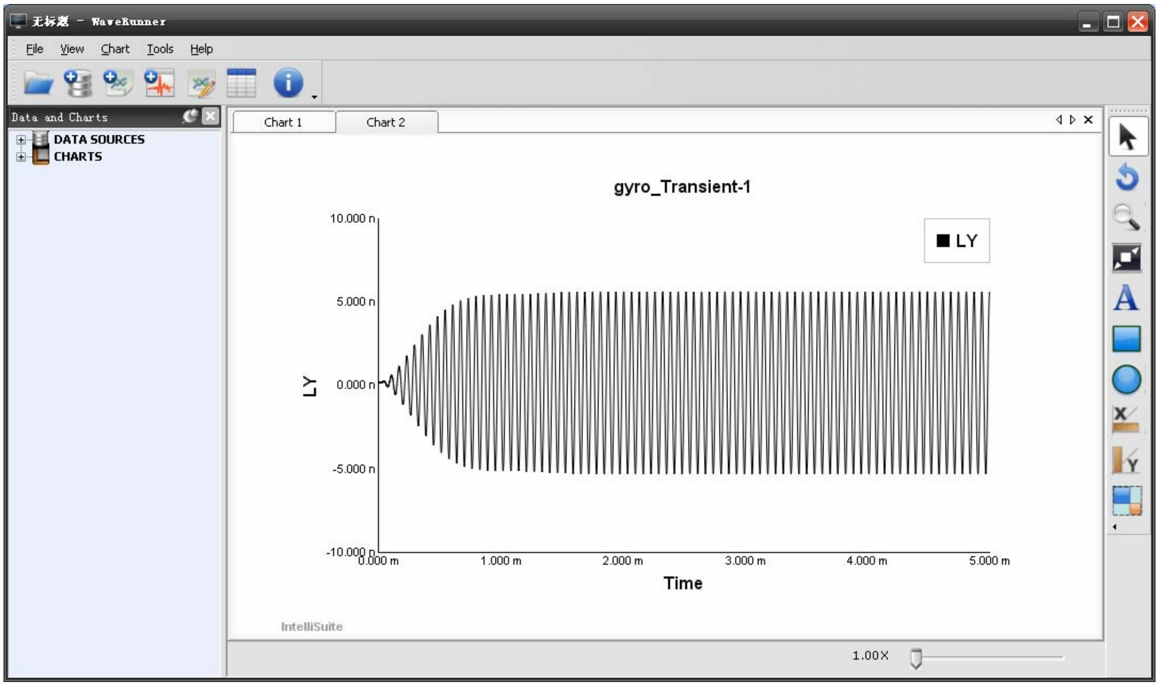


Figure 39 Transient Response - Y

2.5. Coriolis Analysis

A gyroscope is a sensor that measures the rate of rotation of an object. Most micro-machined gyroscopes use vibrating mechanical elements to sense angular velocity. Utilizing vibrating elements to induce and detect Coriolis forces presents many advantages (no rotating parts that require bearings, eliminating concerns about friction and wear). The fundamental operating principle of a vibratory gyroscope relies on the sinusoidal Coriolis force induced by the vibration of a proof-mass and an orthogonal angular-rate input. In this section we will apply an angular velocity to the structure and examine the resulting Coriolis force.

Open the following file: `<Installation Directory>/IntelliSuite/Training/ /Application_Notes/Gyro/SYNPLE_Models/Element_Models/gyro_coriolis.ssc`.

The model will appear as shown below.

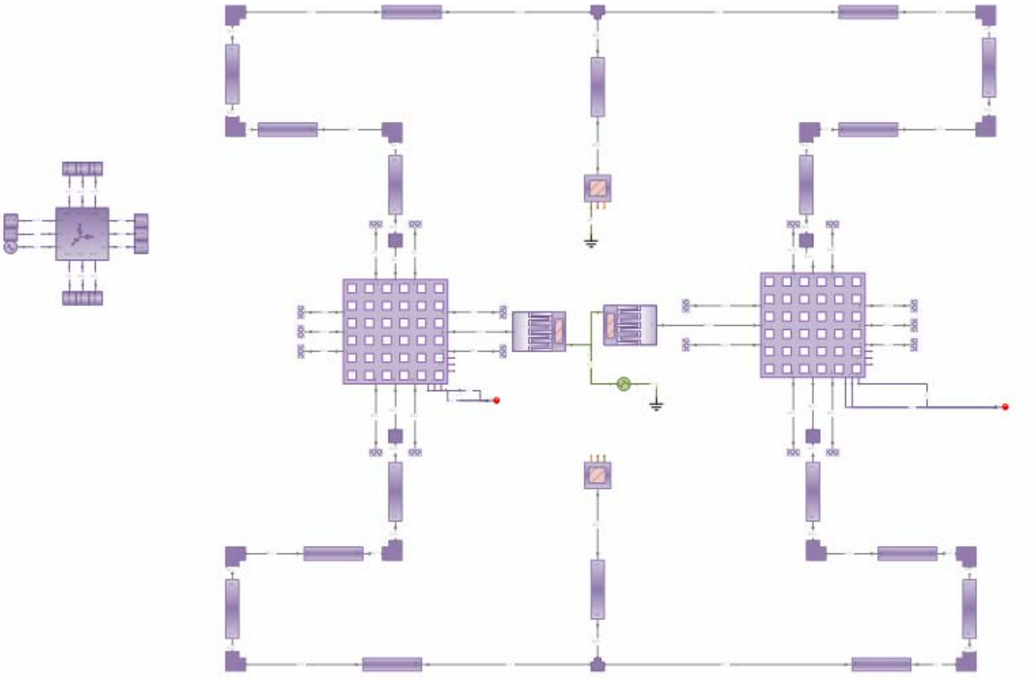


Figure 40 Schematic for Coriolis Analysis

You will see that a rotation input has been applied to the Global Frame element.

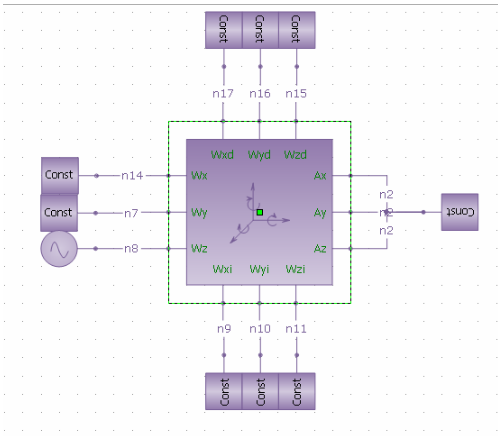
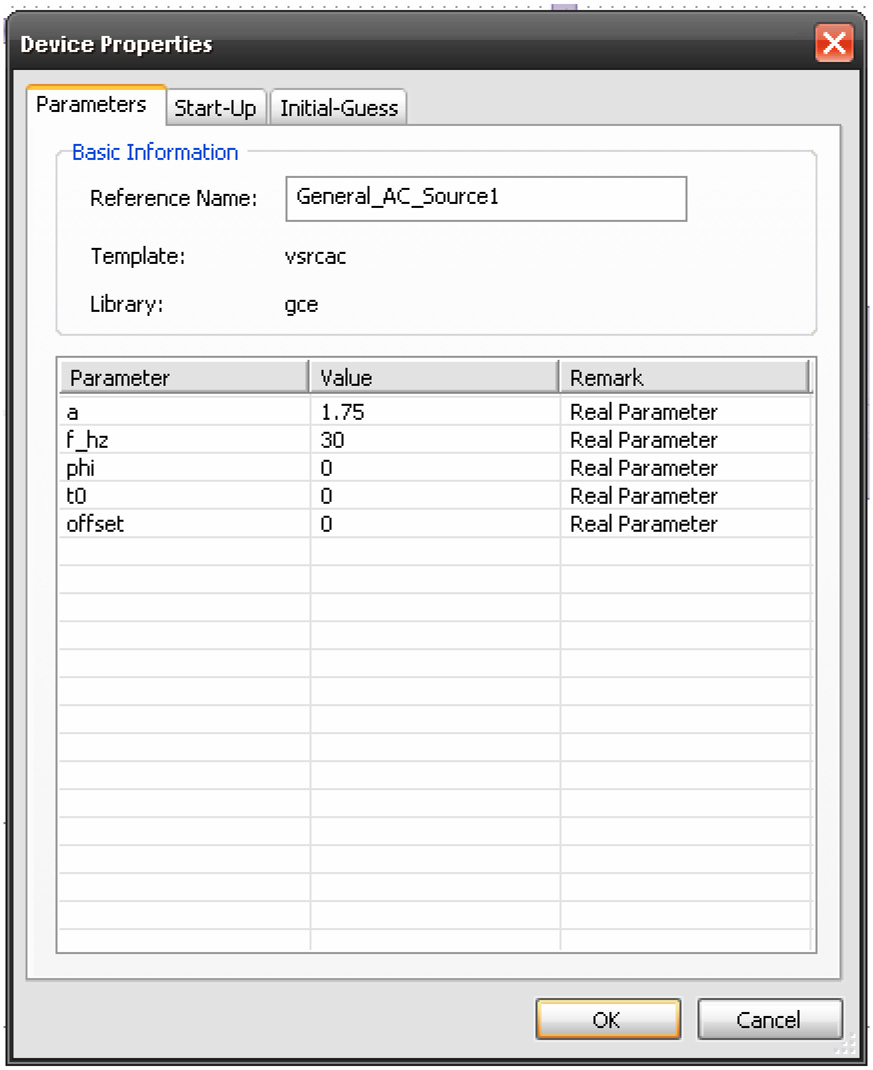


Figure 41 Rotation Input

The General AC Source element is being used to apply a rotation. If you double-click on this element, you will see that the rotation frequency is 30 Hz and the amplitude is 100 deg/sec (or 1.75 rad/sec).



Basic Information

Reference Name: General_AC_Source1

Template: vsrcac

Library: gce

Parameter	Value	Remark
a	1.75	Real Parameter
f_hz	30	Real Parameter
phi	0	Real Parameter
t0	0	Real Parameter
offset	0	Real Parameter

Figure 42 AC Source Parameters

To run the simulation,

Click *Transient Analysis*

Click *OK*

The results of the simulation are shown in the figure below.

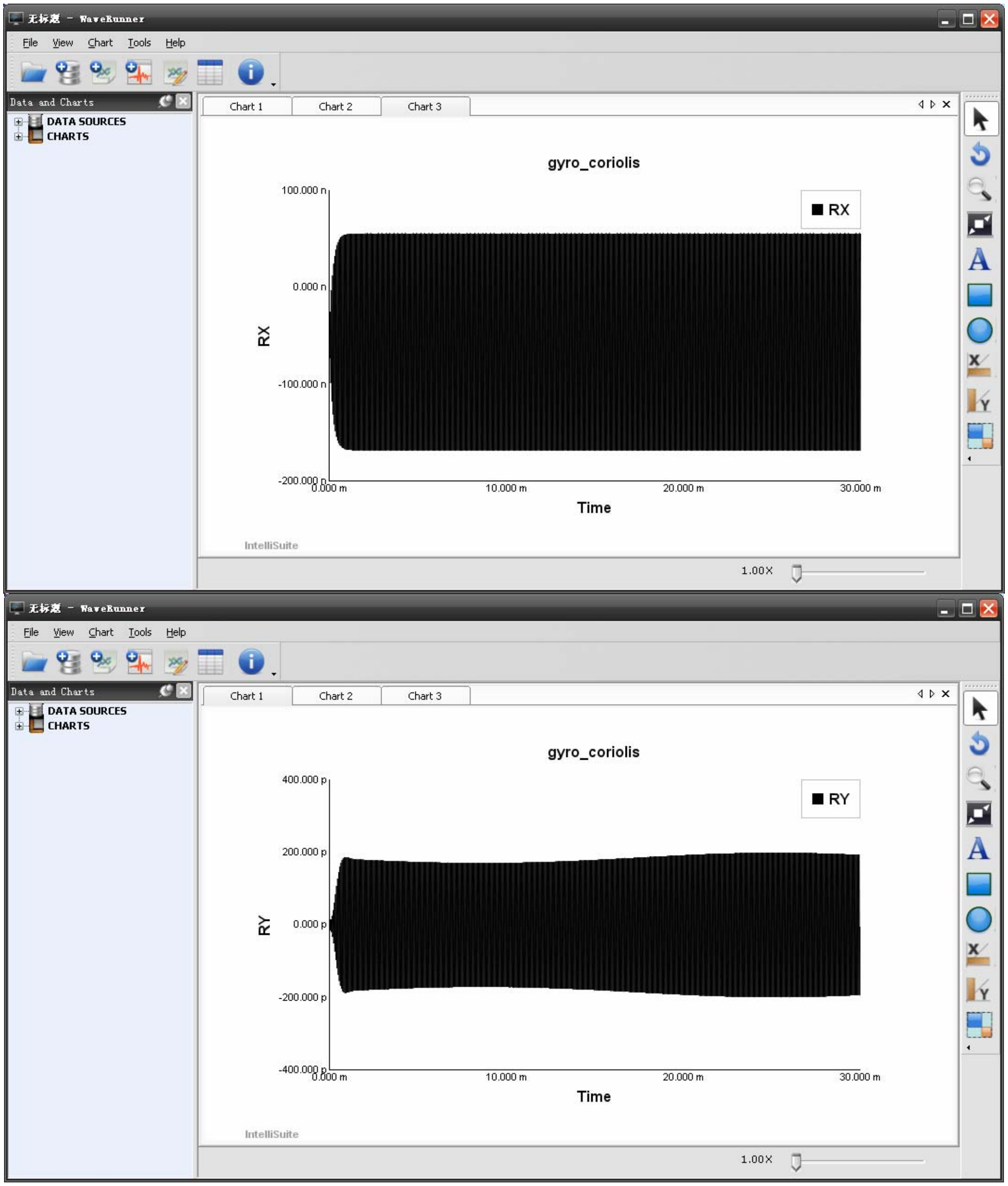


Figure 43 Simulation Results

We see a small result signal because the Coriolis response is very small. If we increase the angular velocity amplitude from 100 deg/sec to 1000 deg/sec, we will see a clearer Coriolis response.

2.6. Quadrature effect

In reality, fabrication imperfections result in non-ideal geometries in the gyroscope structure, which in turn causes the drive oscillation to partially couple into the sense mode. This is called the "quadrature effect." Typically, most of the coupling occurs in the suspension elements. Considering the relative magnitudes of the drive and sense oscillations, even extremely small undesired coupling from the drive motion to the sense mode could completely mask the Coriolis response. In the previous section, we saw that the Coriolis response resulting from a 1.75 rad/sec rotation input is much smaller than the undesired quadrature signal.

Almost all suspension elements in real implementations of vibratory gyroscopes have elastic cross-coupling between their principal axes of elasticity. This phenomenon is called anisoelasticity and is the primary cause of mechanical quadrature error in gyroscopes. In this section we will analyze the quadrature effect using SYNPLE.

A transient plot of the Coriolis response and quadrature signal is shown in the figure below. This Coriolis response is the result of a 500 rad/sec angular velocity input, chosen to obtain a large Coriolis response. The quadrature signal is the transient y-axis displacement result without any rotation input. The drive signal is the same for both curves.

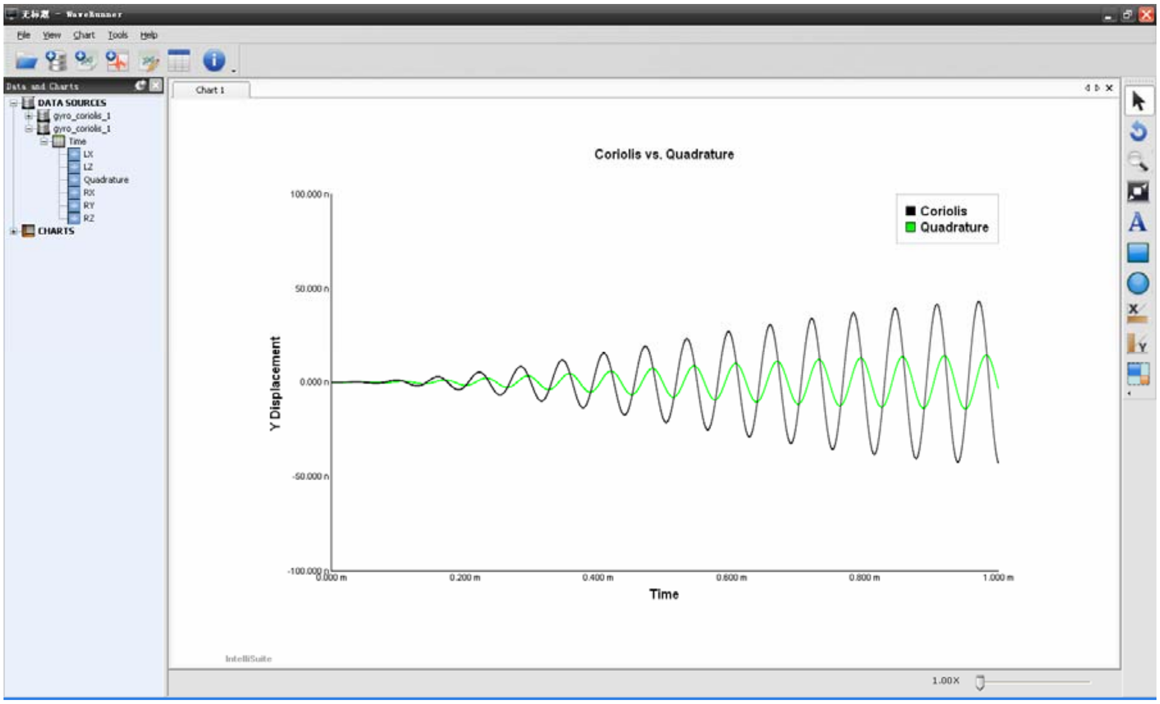


Figure 44 Coriolis and Quadrature Signals

The figure shows a 90° phase shift between the Coriolis signal and the quadrature signal. This result is expected because the Coriolis force is proportional to the drive velocity, *v,* and the quadrature force is proportional to the drive position, *x*. You can use SYNPLE to further analyze the quadrature signal and design quadrature compensation features like mechanical element trimming or an electric loop.

2.7. Mask synthesis

Once you have a schematic created in SYNPLE, it's easy to extract a mask layout from the schematic.

Click *Synthesis…Synthesize Mask Layout in Blueprint*

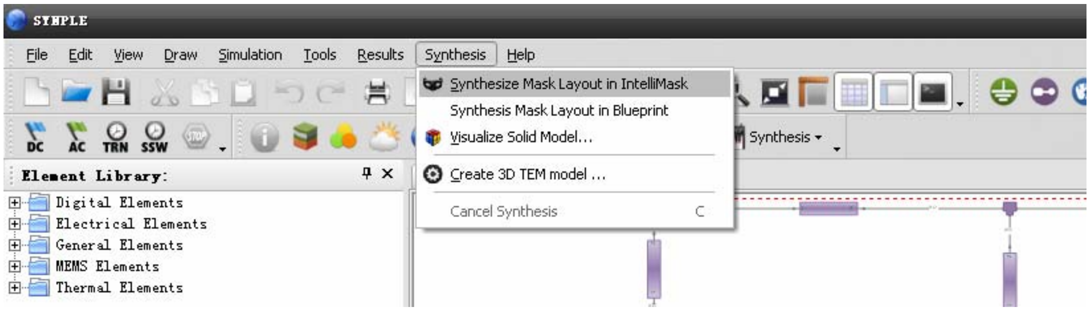
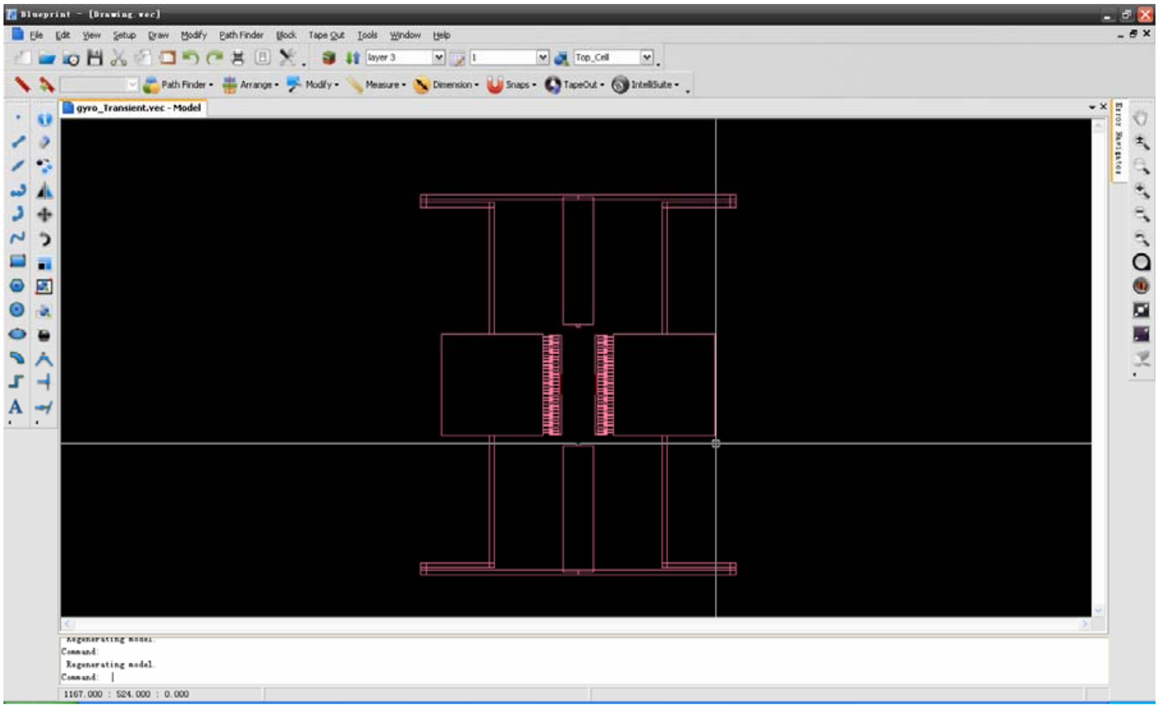


Figure 45 Synthesize Mask Layout

Blueprint, IntelliSense's layout editor, will automatically open with a mask layout for the structure.



```
Regenerating model.
Command:
 Regenerating model.
Command:  |
1167.000 : 524.000 : 0.000
```

Figure 46 Mask Layout

.

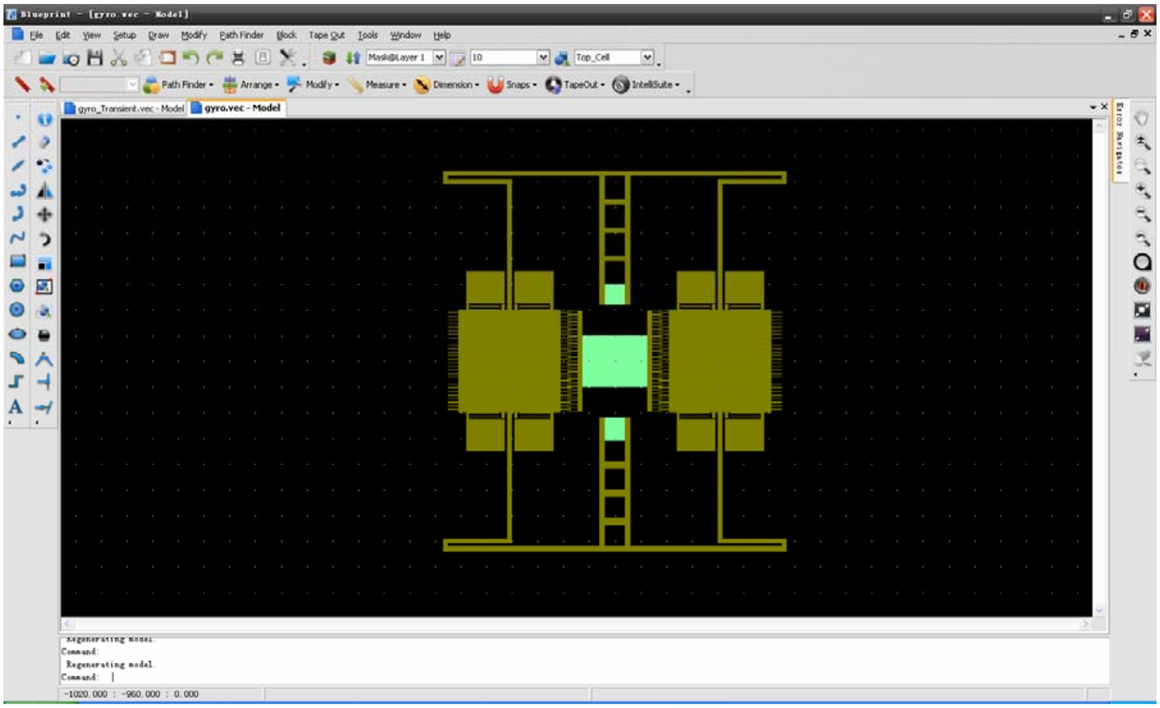
3. Model Construction

3.1. Layout

We will be using a different mask that includes some sense electrodes for the gyro. In Blueprint, open the following mask file:

`<Installation Directory>\IntelliSuite\Training\Application_Notes\Gyro\Mech_System_Model\Gyro.msk`.

The model will appear as shown below.



```
 Regenerating model.
Command:
 Regenerating model.
Command:  |
-1020.000 : -960.000 : 0.000
```

Figure 47 Gyro Layout

When you are finished inspecting the mask, close the program.

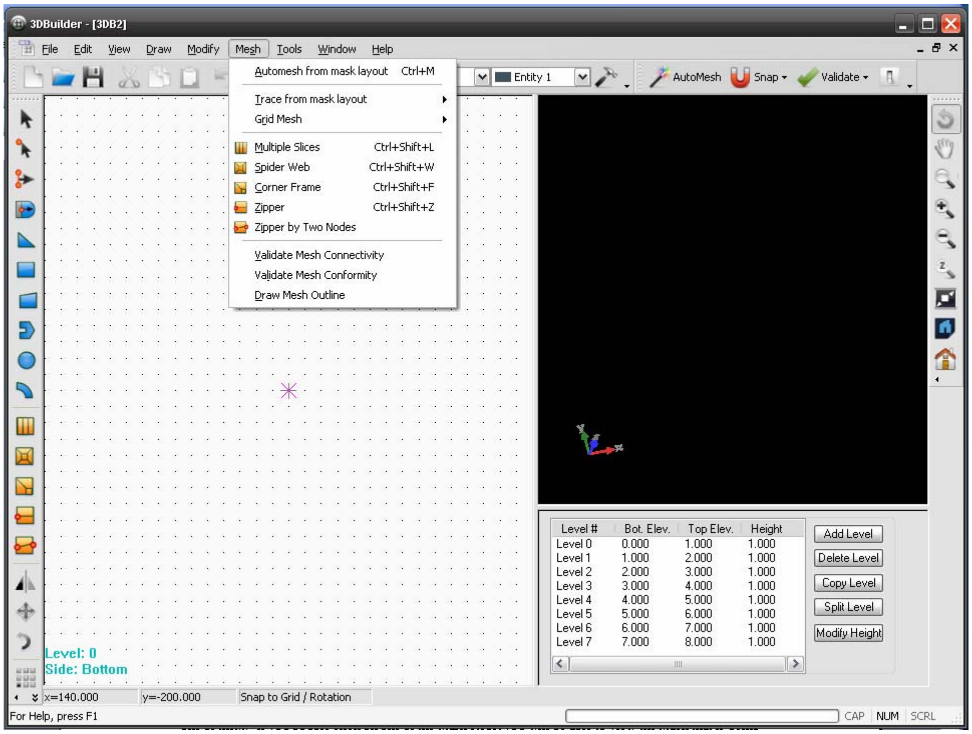
3.2. Solid Structure Construction with 3D Builder

To create a 3D meshed model from the mask layout for use in the analysis module, we will use a program called 3D Builder.

To open the program,

Click *Start…Programs…IntelliSuite…3DBuilder.*

The 3DBuilder window will appear. The left side of the window is used to manipulate the 2D layout of each layer. The right side of the window is used for 3D visualization and layer management. Select *Automesh from mask layout* in the Mesh drop-down menu.



Level #	Bot. Elev.	Top Elev.	Height
Level 0	0.000	1.000	1.000
Level 1	1.000	2.000	1.000
Level 2	2.000	3.000	1.000
Level 3	3.000	4.000	1.000
Level 4	4.000	5.000	1.000
Level 5	5.000	6.000	1.000
Level 6	6.000	7.000	1.000
Level 7	7.000	8.000	1.000

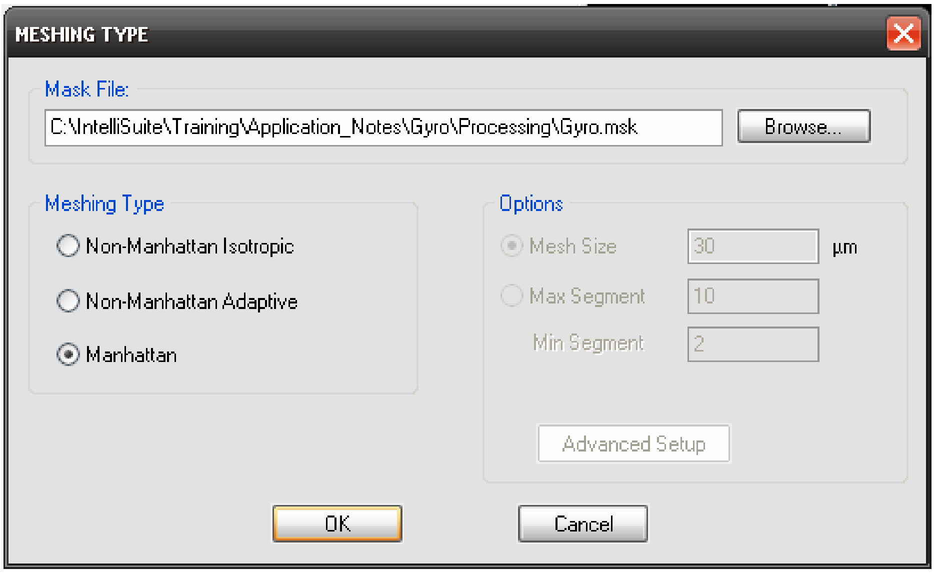
Figure 43 3D Builder Window

In the dialog box that appears,

Click *Browse…*

Select the following file: `<Installation Directory>\IntelliSuite\Training\Application_Notes\Gyro\Mech_System_Model\Gyro.msk`.

Because this structure is composed solely of straight lines and right angles, we will select the *Manhattan* mesh type.



Mask File: C:\IntelliSuite\Training\Application_Notes\Gyro\Processing\Gyro.msk

Meshing Type

○ Non-Manhattan Isotropic

○ Non-Manhattan Adaptive

● Manhattan

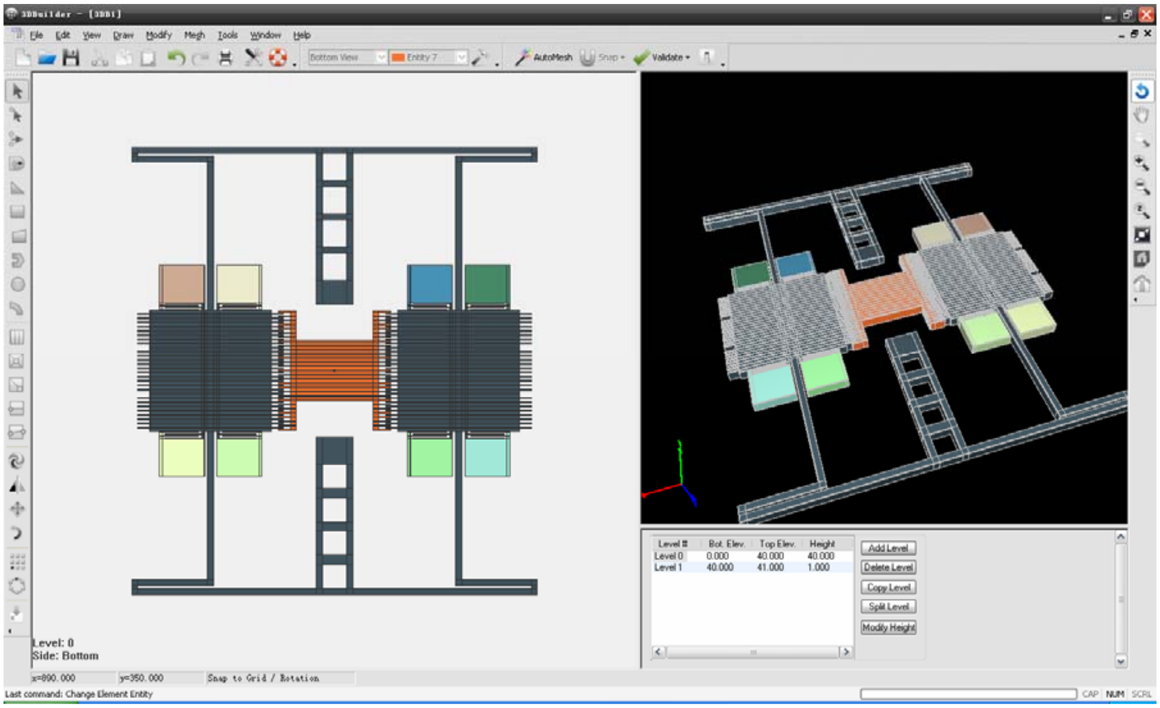
● Mesh Size 30 µm

○ Max Segment 10

Min Segment 2

Figure 48 Automesh Setup

Click *OK,* and after a few seconds, the automeshed structure will appear. In the Levels Manager on the bottom right, you will see that Layer 0 is the device layer and Layer 1 is the anchor layer.



Level #	Bot. Elev.	Top Elev.	Height
Level 0	0.000	40.000	40.000
Level 1	40.000	41.000	1.000

Level: 0
Side: Bottom

x=890.000 y=350.000 Snap to Grid / Rotation

Last command: Change Element Entity

Figure 49 Automeshed Structure

Click the *Modify Height* button and input a value of 40 um. The structure will then appear as below.

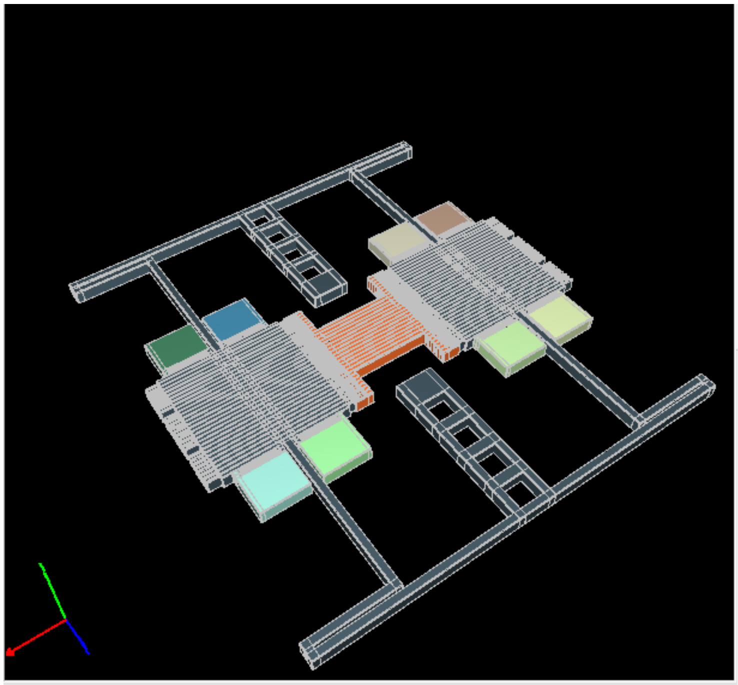
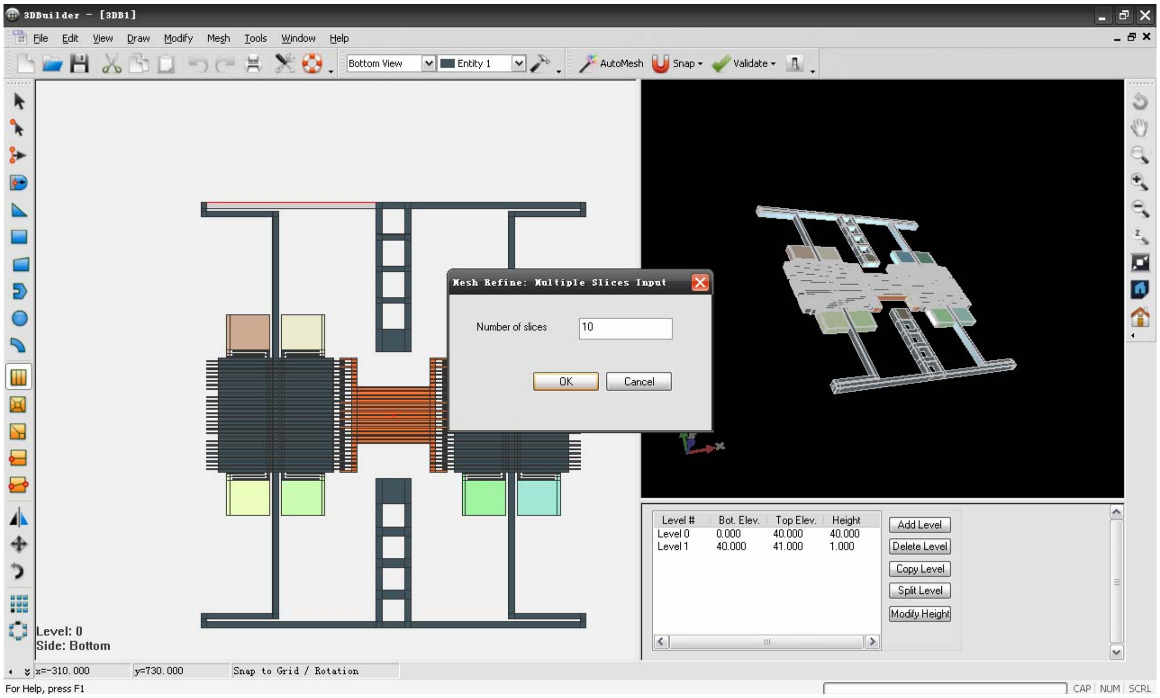


Figure 50 Solid Structure

Click *Mesh…Multiple Slices*

Select the beam highlighted in gray in the figure below, and input “10” in the dialog box that appears.



Mesh Refine: Multiple Slices Input

Number of slices: 10

Level #	Bot. Elev.	Top Elev.	Height
Level 0	0.000	40.000	40.000
Level 1	40.000	41.000	1.000

Level: 0
Side: Bottom

x=-310.000 y=730.000 Snap to Grid / Rotation

For Help, press F1

Figure 51 Slicing Function

Click *OK*

The beam will then be divided into 10 parts. Repeat the operation on each beam. Slice the longer beams into 10 segments and the shorter beams into 5 segments. When you are finished, the model should appear as shown below.

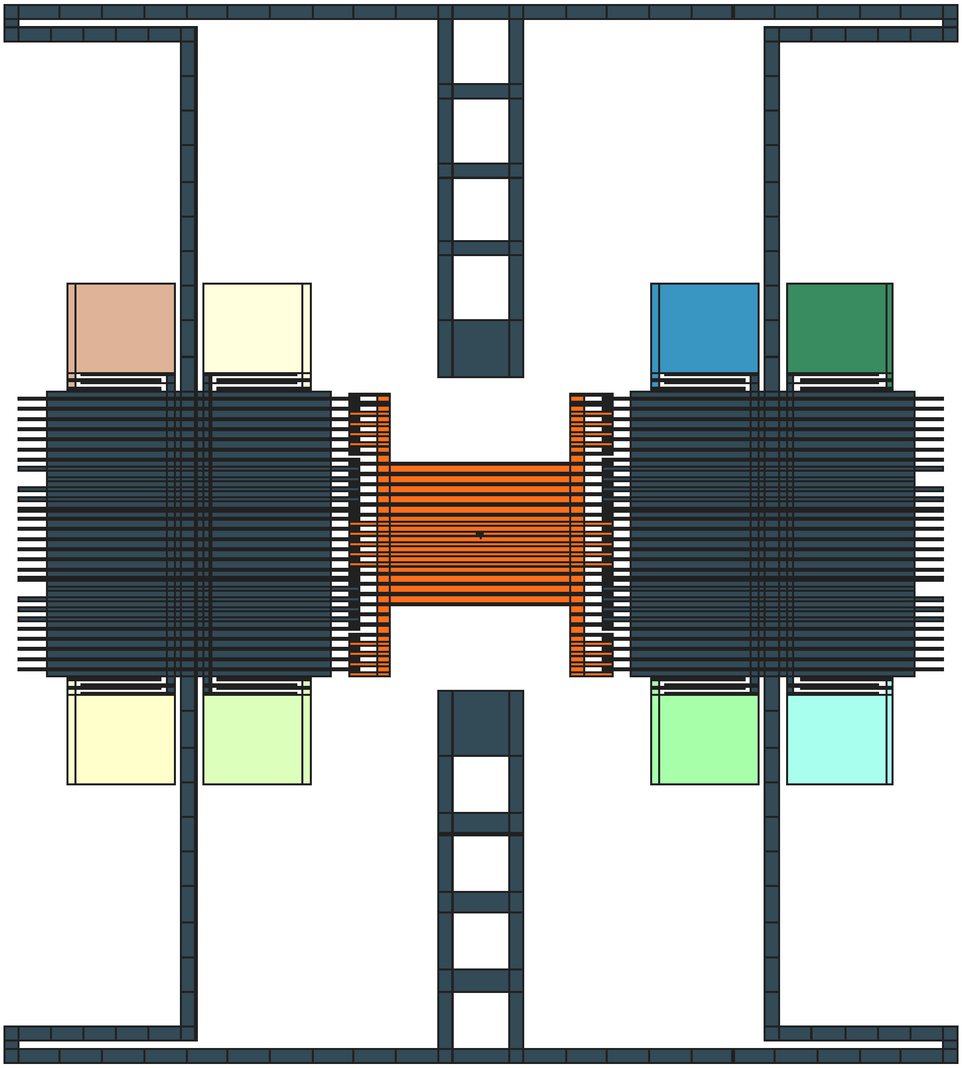
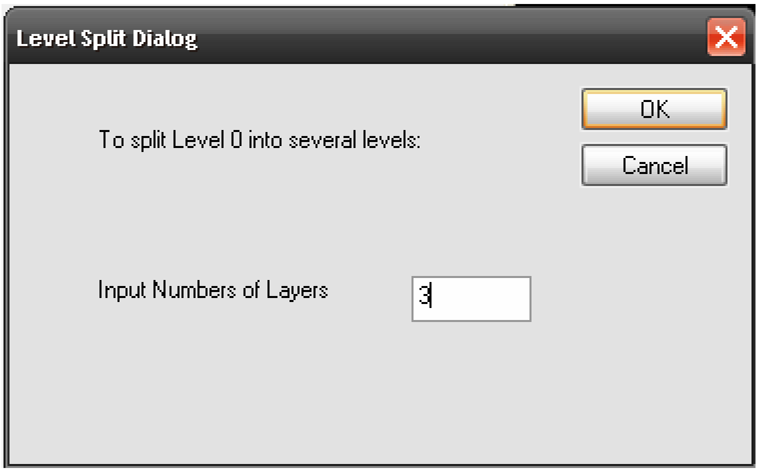


Figure 52 Model with Refined Mesh

In the Levels Manager on the bottom right, click the *Split Level* button. Input “3” in the dialog that appears.

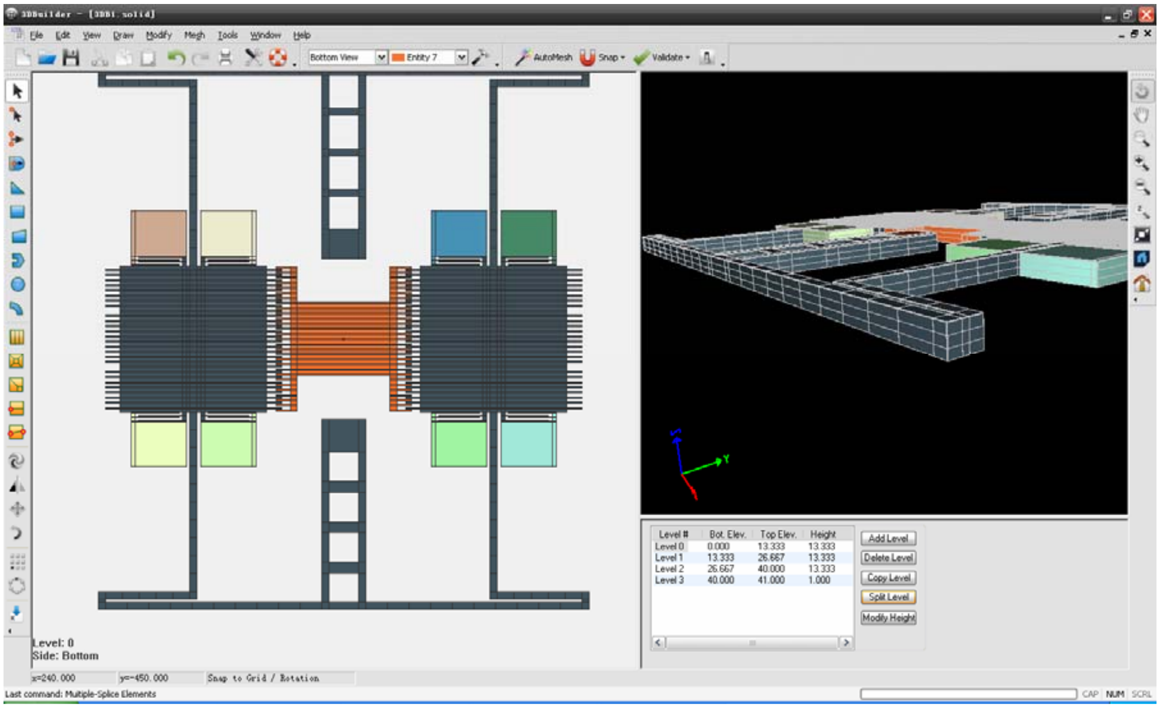


To split Level 0 into several levels:

Input Numbers of Layers 3

Figure 53 Split Level Function

The model will then appear as shown below.



Level #	Bot. Elev.	Top Elev.	Height
Level 0	0.000	13.333	13.333
Level 1	13.333	26.667	13.333
Level 2	26.667	40.000	13.333
Level 3	40.000	41.000	1.000

Level: 0
Side: Bottom

x=240.000 y=-450.000 Snap to Grid / Rotation

Last command: Multiple-Splice Elements

Figure 54 3D Solid Model

Now we are ready to send the structure to the analysis module. You will first need to save the 3D Builder file in a location of your choosing. Then,

Click *File…Export to Analysis Module…*

Click *Continue without Check* and select the ThermoElectroMechanical analysis module. Save the analysis file, making sure not to use spaces in the file or folder names.

4. ThermoElectroMechanical (TEM) Analysis

The ThermoElectroMechanical Analysis Module (TEM) is the device-level Finite Element Analysis application developed by IntelliSense. This module allows you to incorporate material properties, loads, and boundary conditions to fully analyze a device in the static, frequency, and dynamic domains. One additional feature unique to IntelliSuite's TEM is the ability to create N-DOF (N-degree-of-freedom) system models that can be imported into a system-level simulator like SYNPLE. These models retain the accuracy of finite element-based models, but can be quickly simulated under multiple loading conditions in the system-level solver. This functionality also allows you to co-simulate your CMOS control circuitry with your finite element-based MEMS device. This is a very powerful capability only offered by IntelliSense that allows our users to fully develop their device from initial design exploration all the way through to full integration of the MEMS device with their circuit.

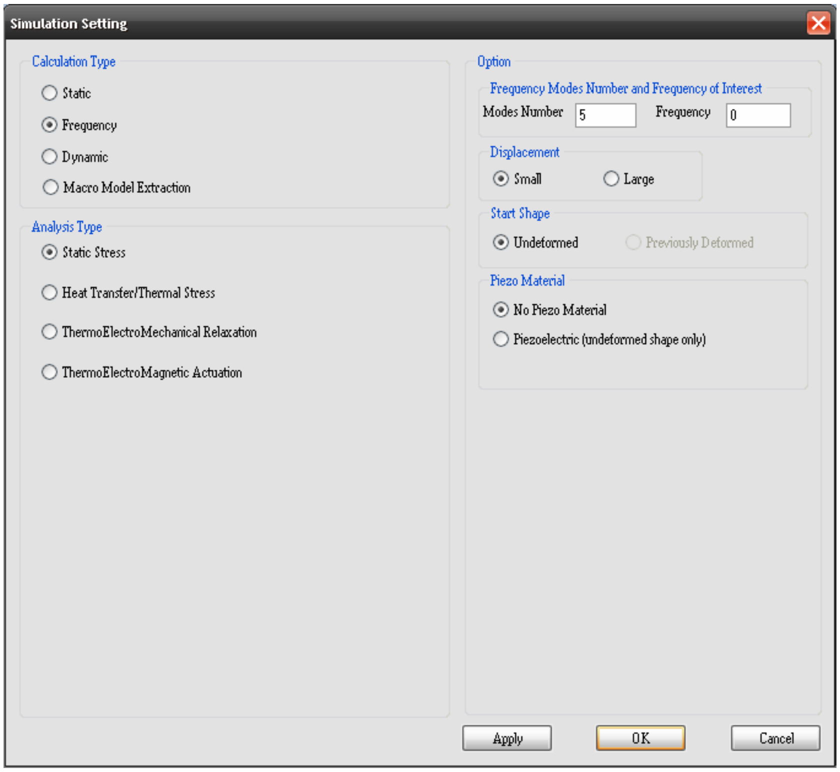
In this section we will discuss the device-level simulation of the gyro. We will set up the model and run Frequency, Static, Dynamic, and System Model Extraction simulations. We work in this order so that we perform the simpler, faster simulations first (frequency and static). Once we understand our model better, we will perform the more lengthy simulations (dynamic and SME).

4.1. Setting up the model

The first thing to do once we have our finite element model in the TEM is to choose a simulation setting and check the material properties.

Click *Simulation…Simulation Setting*.

A window will appear showing all of the simulation types and their respective options.



Calculation Type
○ Static
● Frequency
○ Dynamic
○ Macro Model Extraction

Analysis Type
● Static Stress
○ Heat Transfer/Thermal Stress
○ ThermoElectroMechanical Relaxation
○ ThermoElectroMagnetic Actuation

Option

Frequency Modes Number and Frequency of Interest
Modes Number 5
Frequency 0

Displacement
● Small ○ Large

Start Shape
● Undeformed ○ Previously Deformed

Piezo Material
● No Piezo Material
○ Piezoelectric (undeformed shape only)

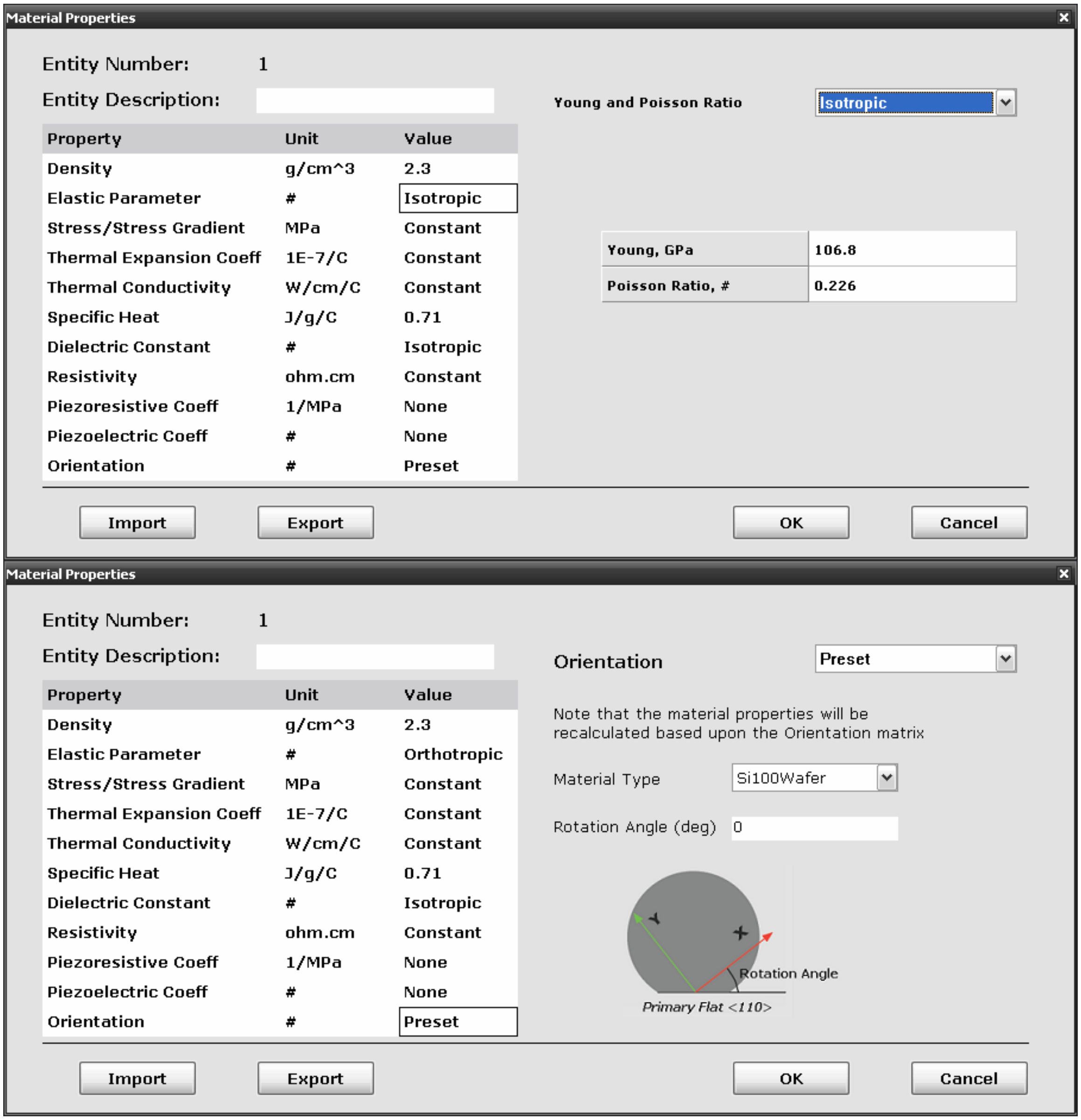
Figure 55 Simulation Setting Window

In the first simulation we will run is a Natural Frequency Analysis, so set the *Calculation Type* to *Frequency*, and the *Analysis Type* to *Static Stress*. Set the simulation to analyze the first 6 modes using small scale displacement theory, an undeformed start shape, and no piezomaterial.

Once we have the simulation set up, we can work on our material properties, loads, and boundaries.

Click *Material…Check/Modify*.

Select the large entity that makes up the moving structure and it will turn red. The Material Properties dialog will appear as shown below with default settings for bulk silicon. You will see that you can choose to simulate isotropic, orthotropic, or anisotropic materials in the *Elastic Parameter* field. The simulator will also account for the effect of the wafer orientation on the material properties in the *Orientation* field. In this case we will leave the settings as the default and assume an isotropic material.



Entity Number: 1

Entity Description:

Property	Unit	Value
Density	g/cm^3	2.3
Elastic Parameter	#	Isotropic
Stress/Stress Gradient	MPa	Constant
Thermal Expansion Coeff	1E-7/C	Constant
Thermal Conductivity	W/cm/C	Constant
Specific Heat	J/g/C	0.71
Dielectric Constant	#	Isotropic
Resistivity	ohm.cm	Constant
Piezoresistive Coeff	1/MPa	None
Piezoelectric Coeff	#	None
Orientation	#	Preset

Young and Poisson Ratio: Isotropic

Young, GPa	106.8
Poisson Ratio, #	0.226

Entity Number: 1

Entity Description:

Property	Unit	Value
Density	g/cm^3	2.3
Elastic Parameter	#	Orthotropic
Stress/Stress Gradient	MPa	Constant
Thermal Expansion Coeff	1E-7/C	Constant
Thermal Conductivity	W/cm/C	Constant
Specific Heat	J/g/C	0.71
Dielectric Constant	#	Isotropic
Resistivity	ohm.cm	Constant
Piezoresistive Coeff	1/MPa	None
Piezoelectric Coeff	#	None
Orientation	#	Preset

Orientation: Preset

Note that the material properties will be recalculated based upon the Orientation matrix

Material Type: Si100Wafer

Rotation Angle (deg): 0

Figure 56 Material Properties

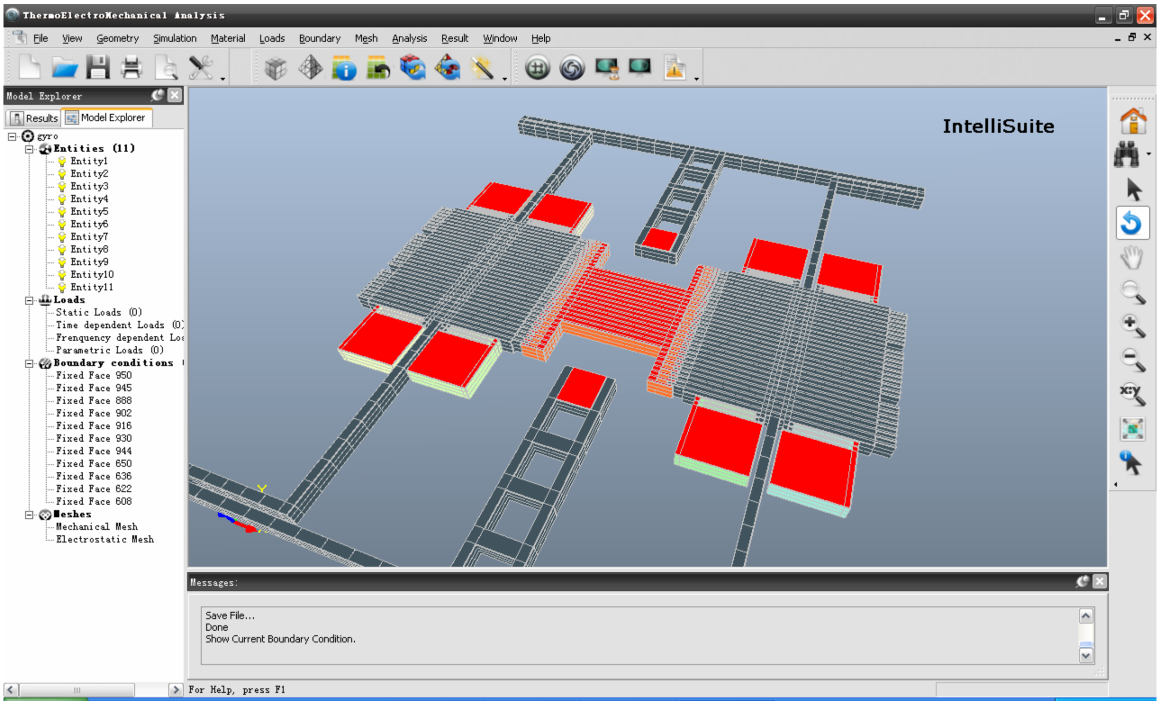
Click *OK*

The material properties of the rest of the entities do not matter in this case because they will be used for electrostatic purposes only; none of them will move.

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

Click *Boundary…Selection Mode…Pick on Geometry*
Click *Boundary…Fixed*

Select each of the red faces shown in the figure below.



Save File…
Done
Show Current Boundary Condition.

Figure 57 Fixed Faces

When you have finished applying the fixed boundary conditions, check to see that they have all been applied.

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

The software should go through a command prompt, and all of the fixed faces will appear in bright red as shown in the above figure.

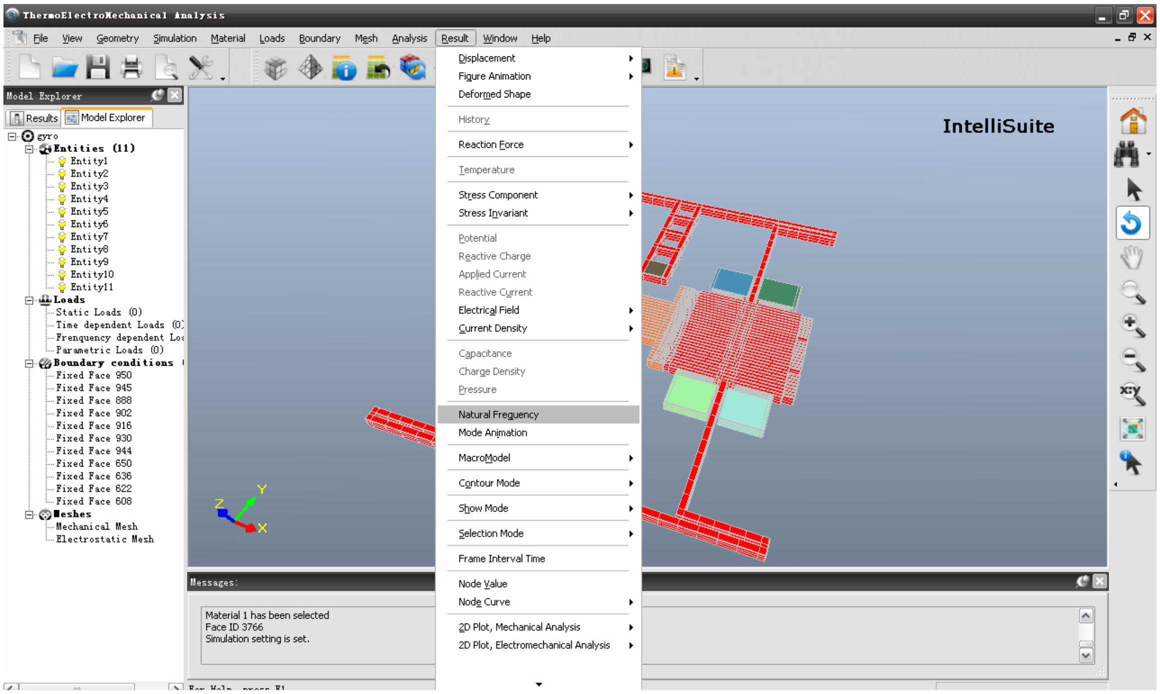
The final step in setting up a model in TEM would be the application of loads. In the first simulation, we will only be running a natural frequency analysis, so we do not have any loads to apply at this point. When we begin performing the electrostatic simulation, we will apply our voltage loads.

4.2. Frequency Analysis

Because we have already set up the simulation with the correct material properties and boundaries, the frequency analysis is easy to complete.

Click *Analysis…Start Frequency Analysis*

The frequency analysis will take 2-3 minutes. While it is running, a command prompt will be visible on your computer screen which will disappear when the analysis has completed. After the simulation, the results available to you will be the list of natural frequencies and animations of each mode shape. These can be found next to each other in the *Result* menu.

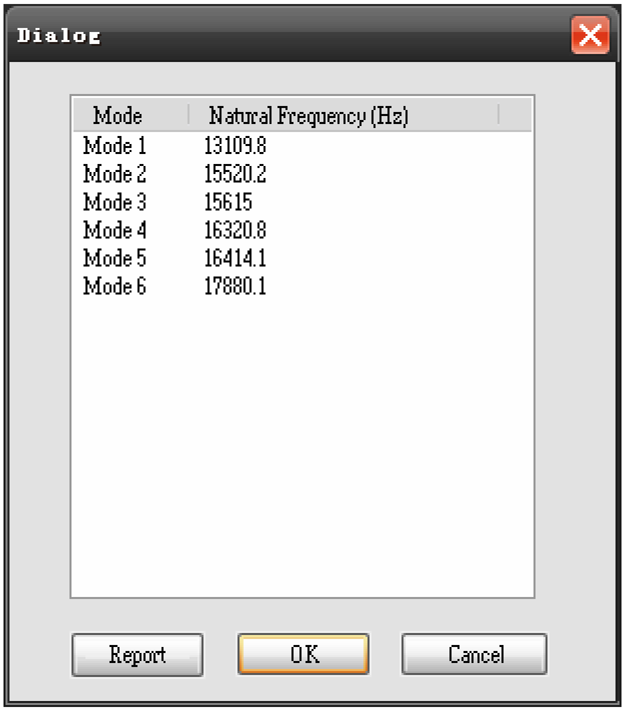


Material 1 has been selected
Face ID 3766
Simulation setting is set.

Figure 58 Result Menu

Click *Result…Natural Frequency*

You will see the list of natural frequencies shown below.



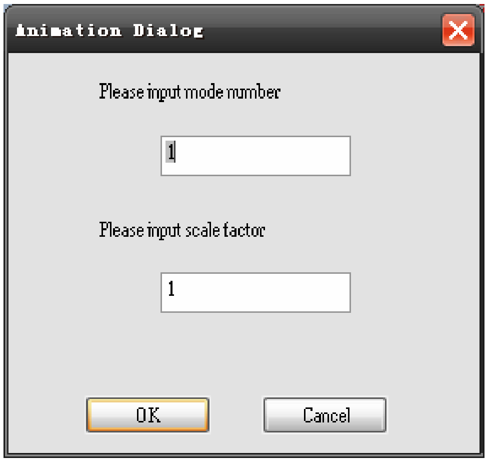
Mode	Natural Frequency (Hz)
Mode 1	13109.8
Mode 2	15520.2
Mode 3	15615
Mode 4	16320.8
Mode 5	16414.1
Mode 6	17880.1

Figure 59 Natural Frequency Results

If you want to view the animation of one of the natural frequencies,

Click *Result…Mode Animation*

This will bring up a window that will ask you for the mode number of the natural frequency you want to animate and ask you to provide a scaling factor.



Please input mode number

1

Please input scale factor

1

Figure 60 Mode animation selection box

Select 3 for the mode number and 40 for the scaling factor. When the mode animation begins, you will see both of the masses of the gyro moving out of phase in the x-direction. If you select mode 2, you will see them moving in phase. The first six mode shapes are shown below.

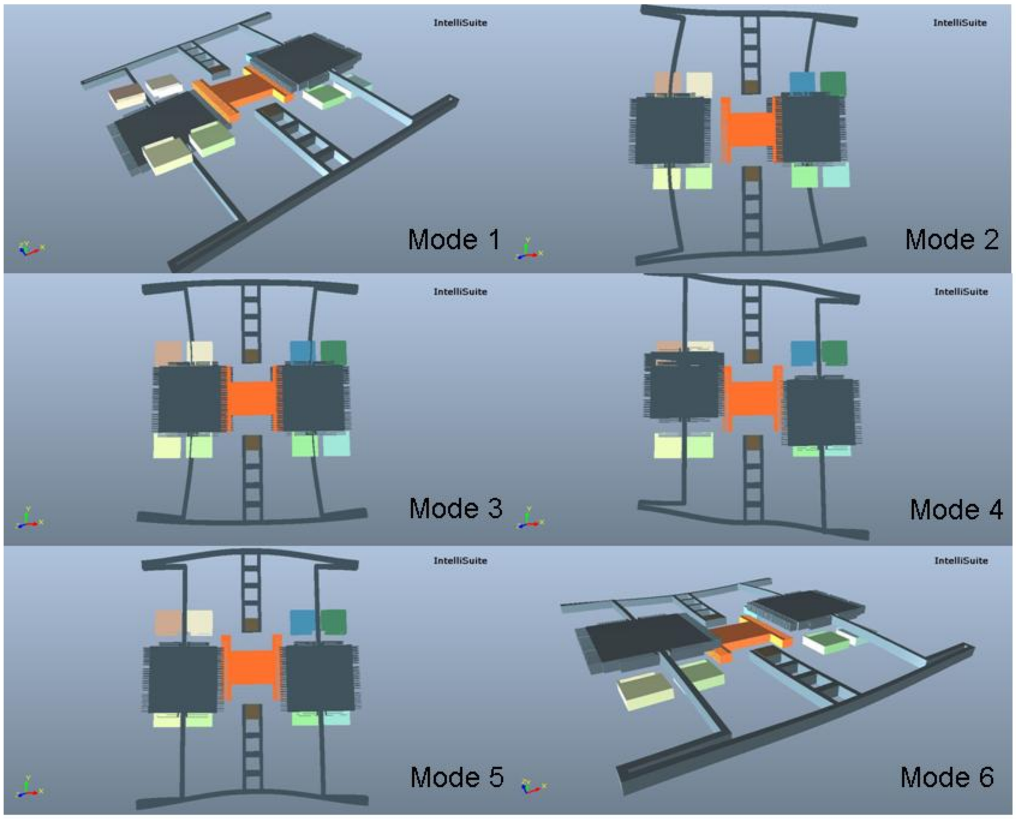


Figure 61 Mode animation

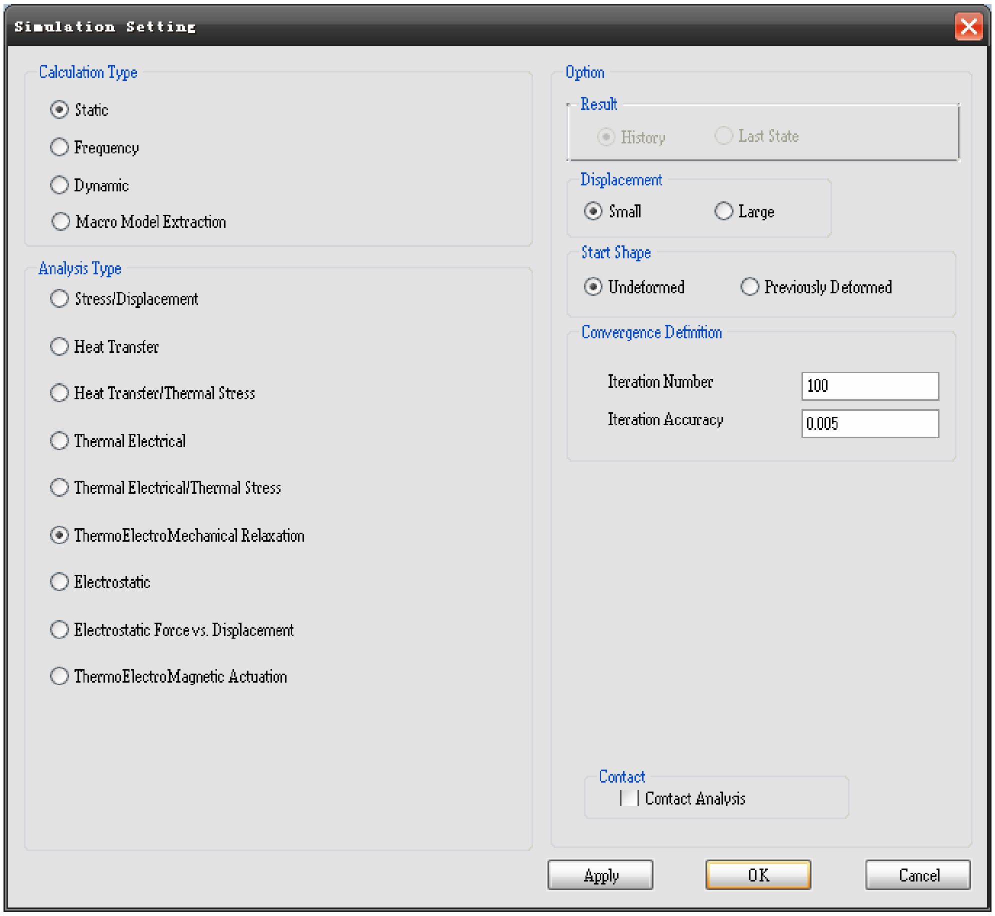
Modes 1 and 6 correspond to the movement of the masses in the z-direction. Modes 2 and 3 correspond to the movement of the masses in the x-direction. Modes 4 and 5 correspond to the movement of the masses in the y-direction. We will be driving the gyro in the x-direction and the resultant movement y-direction will be used to sense changes in the capacitances. We will use mode 3 as the drive mode and mode 4 will be our sense mode. Now that we understand the important modes of the gyro, and we have confirmed that our boundary conditions are correct, we can move on to static simulations.

4.3. Static Simulation

For a static simulation we will perform a simple voltage-induced displacement analysis where we actuate the drive combs to and determine the displaced shape of the gyro. We have already created a file for you that you can use to run the simulation.

Open the following file: `<Installation Directory>/IntelliSuite/Training/`
`/Application_Notes/Gyro/Device_Models/Gyro.save`

Make sure the simulation settings are set up as shown in the figure below.


Simulation Setting

Calculation Type
- ● Static
- ○ Frequency
- ○ Dynamic
- ○ Macro Model Extraction

Analysis Type
- ○ Stress/Displacement
- ○ Heat Transfer
- ○ Heat Transfer/Thermal Stress
- ○ Thermal Electrical
- ○ Thermal Electrical/Thermal Stress
- ● ThermoElectroMechanical Relaxation
- ○ Electrostatic
- ○ Electrostatic Force vs. Displacement
- ○ ThermoElectroMagnetic Actuation

Option

Result: ● History ○ Last State

Displacement: ● Small ○ Large

Start Shape: ● Undeformed ○ Previously Deformed

Convergence Definition
Iteration Number: 100
Iteration Accuracy: 0.005

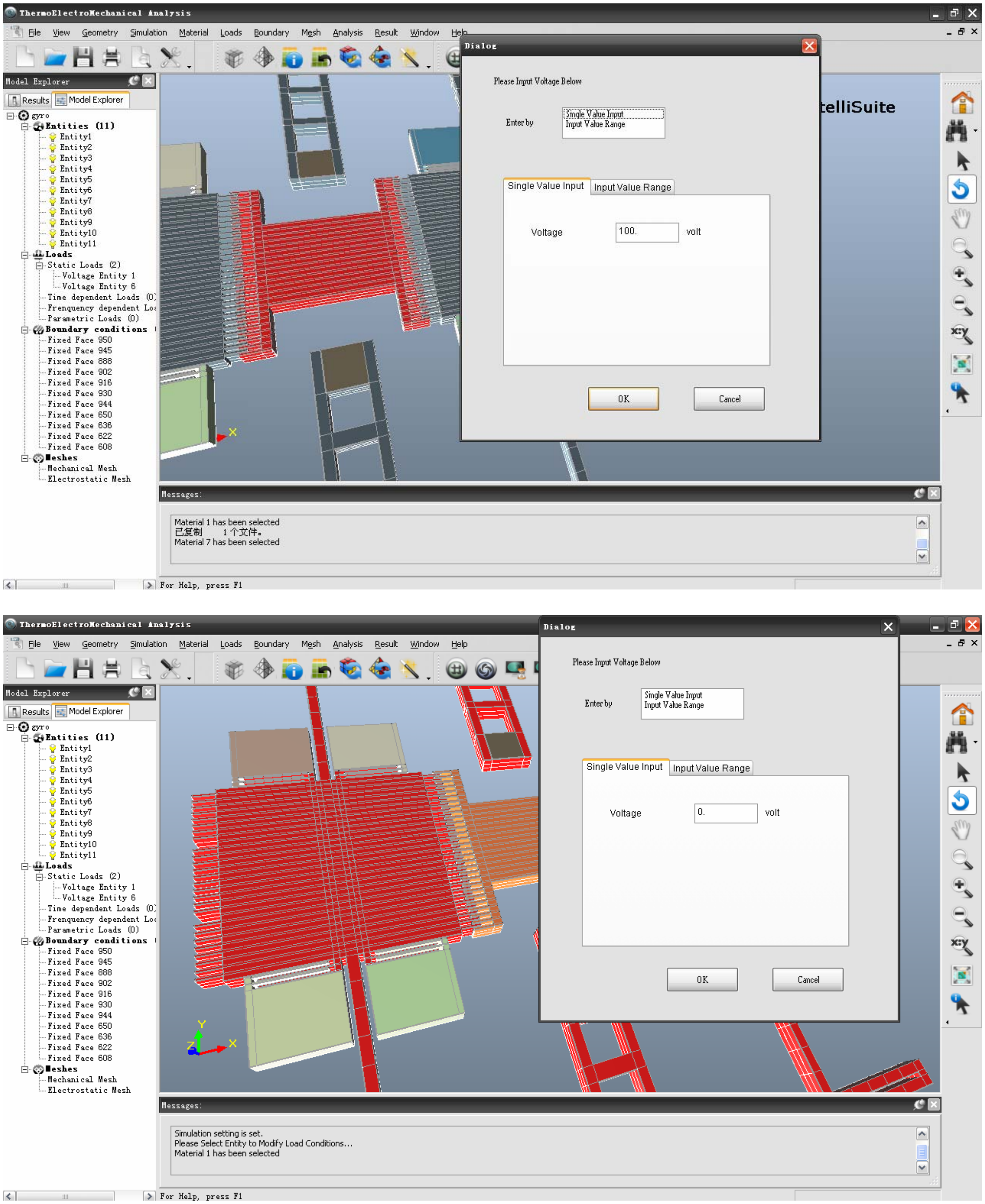
Contact: ☐ Contact Analysis

Figure 62 Static Simulation Settings

For this analysis, we will need to apply voltage loads to three of the conductors. Later when we are looking at the transient case and looking at the Coriolis force response of the sense capacitances, we will define other conductors. In this simulation, we only care about the drive combs and the moving mass. To apply a Voltage load,

Click *Loads…Selection Mode…Pick on Geometry*
Click *Loads…Voltage…Entity*

Apply a 100 V voltage load on fixed side of drive combs and 0 V on the moving mass as shown in the figure below.



Please Input Voltage Below

Enter by: Single Value Input

Voltage 100. volt

Messages:
Material 1 has been selected
已复制 1 个文件。
Material 7 has been selected

Please Input Voltage Below

Enter by: Single Value Input

Voltage 0. volt

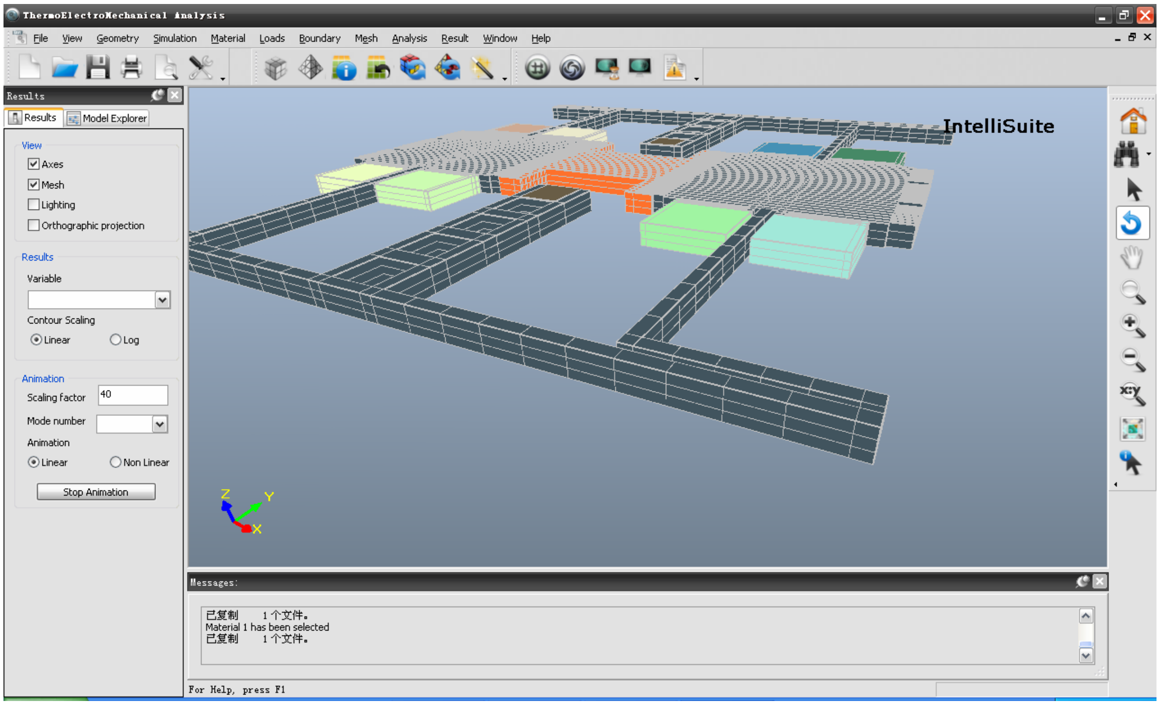
Messages:
Simulation setting is set.
Please Select Entity to Modify Load Conditions...
Material 1 has been selected

Figure 63 Voltage Loads

Now the simulation is set, the material properties are correct, the boundaries are defined, and the loads have been applied. The final step is the mesh refinement.

With all finite element solvers, a solution is more accurate with a more refined mesh. One thing that is special to IntelliSuite and its TEM is the ability of the user to decouple the mechanical and electrostatic meshes. This allows the user to define where the important mechanical elements are and where the important electrostatic elements are. In this particular case, the important mechanical elements are the spring supports and the important electrostatic elements are the comb drives.

For this device, the mechanical mesh refinement has already been done in Section 3.2. Each part of the spring was broken up into multiple segments.



View
☑ Axes
☑ Mesh
☐ Lighting
☐ Orthographic projection

Results
Variable
Contour Scaling
● Linear ○ Log

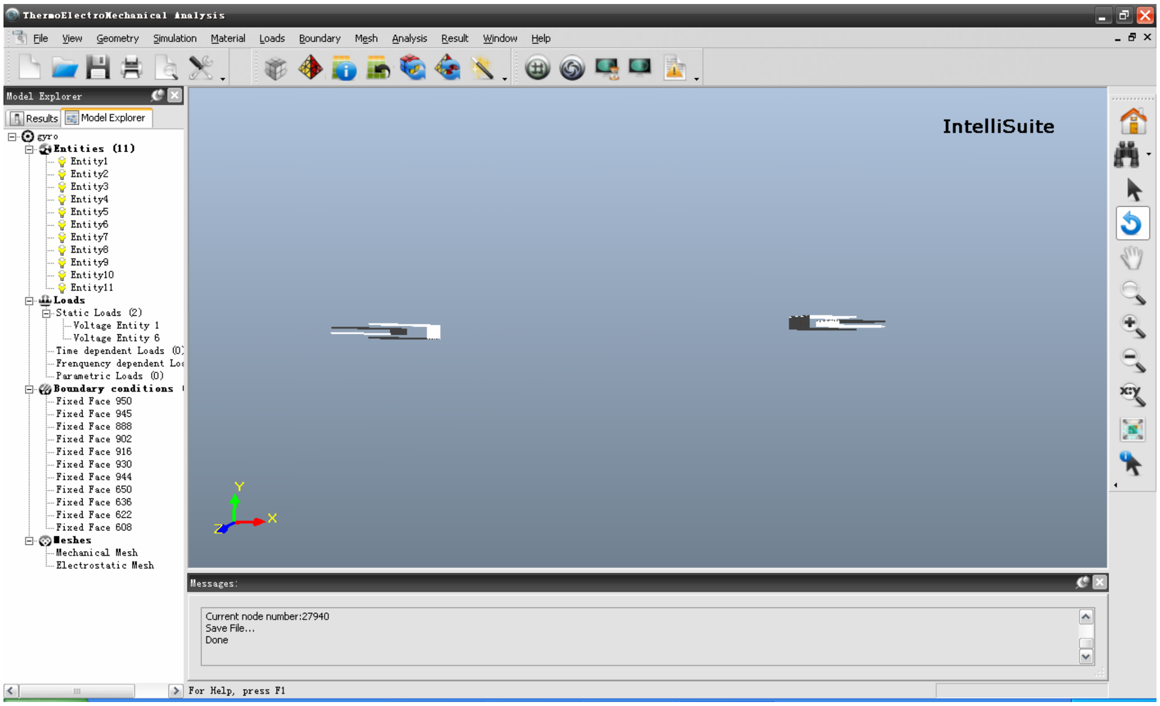
Animation
Scaling factor 40
Mode number
Animation
● Linear ○ Non Linear

```
Messages:
已复制      1 个文件。
Material 1 has been selected
已复制      1 个文件。
For Help, press F1
```

Figure 64 Refined Mechanical Mesh

To check the electrostatic mesh,

Click *Mesh…Selection Mode…Check Only*
Click *Mesh…Elec_Mesh*



```
Current node number:27940
Save File...
Done
```

Figure 65 Electrostatic Mesh

One thing that you will notice about the electrostatic mesh is that most of the device is being ignored (even most of the drive combs!). This is done because the only parts of the device that will be interesting electrostatically are the drive combs and the sense electrodes. In this particular example, we will only worry about the drive combs. If you were to perform a noise analysis, you would want to examine more of the sense electrodes.

In setting up this electrostatic mesh, we are taking advantage of a feature called Multi-bank Exposed Face Meshing (MEFM). The electrostatic forces are generated by the electrostatic mesh and then applied to the mechanical mesh. Each identical comb finger will see the same electrostatic forces, and because of this, we can simulate one finger and tell the software how many other duplicates there are. The software will apply the electrostatic pressures to the device accordingly. This is yet another feature specific to IntelliSuite that allows users to accurately simulate their electromechanical devices with far greater efficiency than our competitors.

Once you have checked out the simulation settings, material properties, loads, boundaries, and mesh, it's time to run the simulation.

Click *Analysis…Start Static Analysis*

When the simulation is running, you will see a command prompt. This prompt will display some data about the status of the simulation (what iteration step it's on, the maximum displacements in the x, y, and z-directions). The simulation should take about 3-5 mins to run. Once the command prompt disappears, the simulation is complete and you can view the results.

Click *Result…Displacement…X*

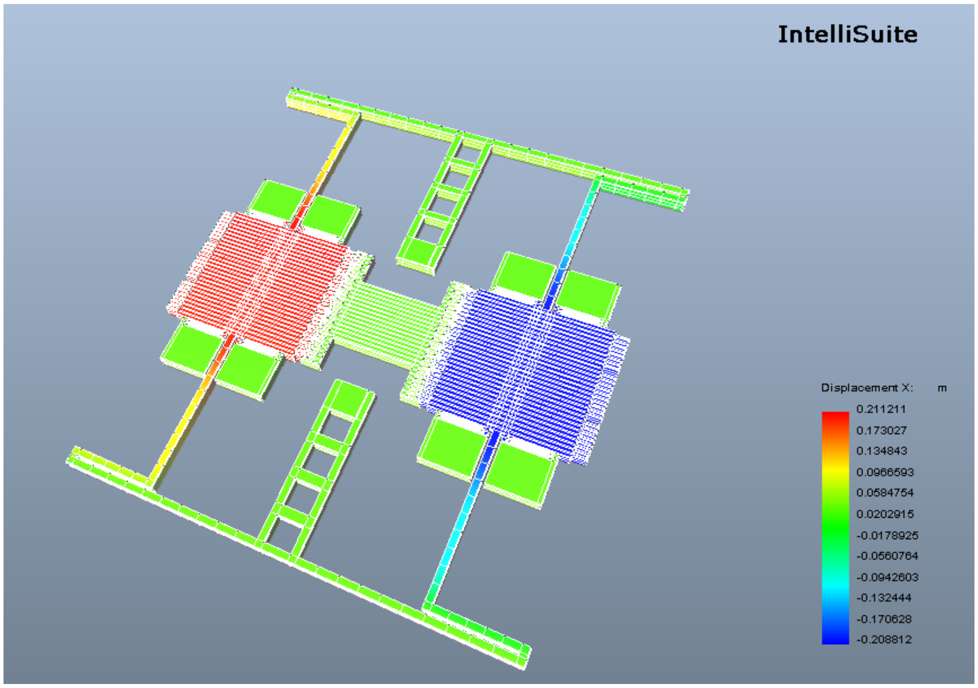


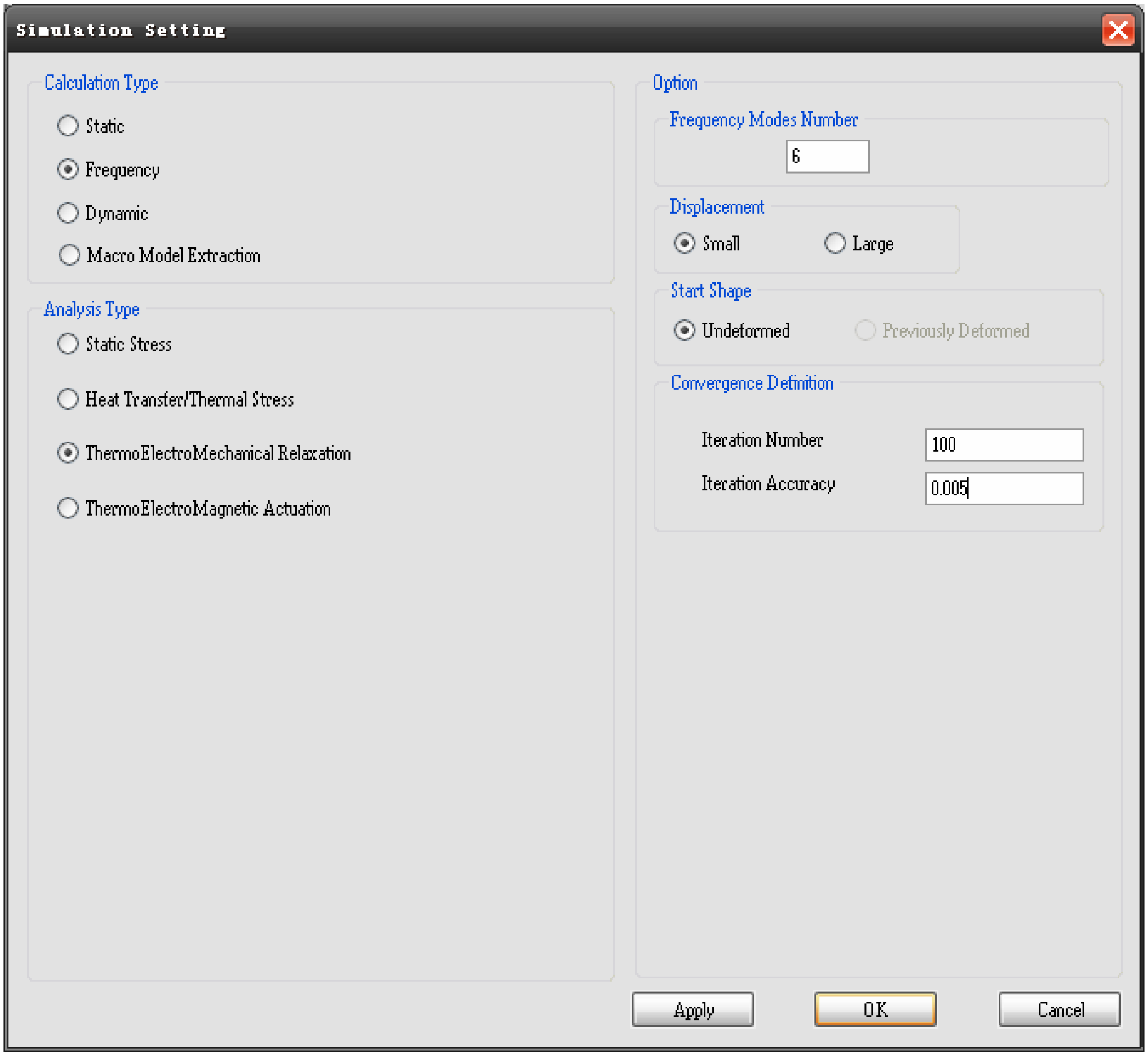
Figure 66 Static Results in the X-direction

You can also view capacitance matrices, charge densities, electrostatic pressures, stresses, etc. Feel free to explore the results menu.

4.4. Spring softening effect

Electrostatic forces resulting from an external DC voltage source will change the stiffness of a structure and reduce the resonant frequency. This effect is called electrostatic spring softening, and we will examine its effect on this gyro model.

Set up the simulation to run a frequency analysis as shown in the figure below.



Simulation Setting

Calculation Type
○ Static
● Frequency
○ Dynamic
○ Macro Model Extraction

Analysis Type
○ Static Stress
○ Heat Transfer/Thermal Stress
● ThermoElectroMechanical Relaxation
○ ThermoElectroMagnetic Actuation

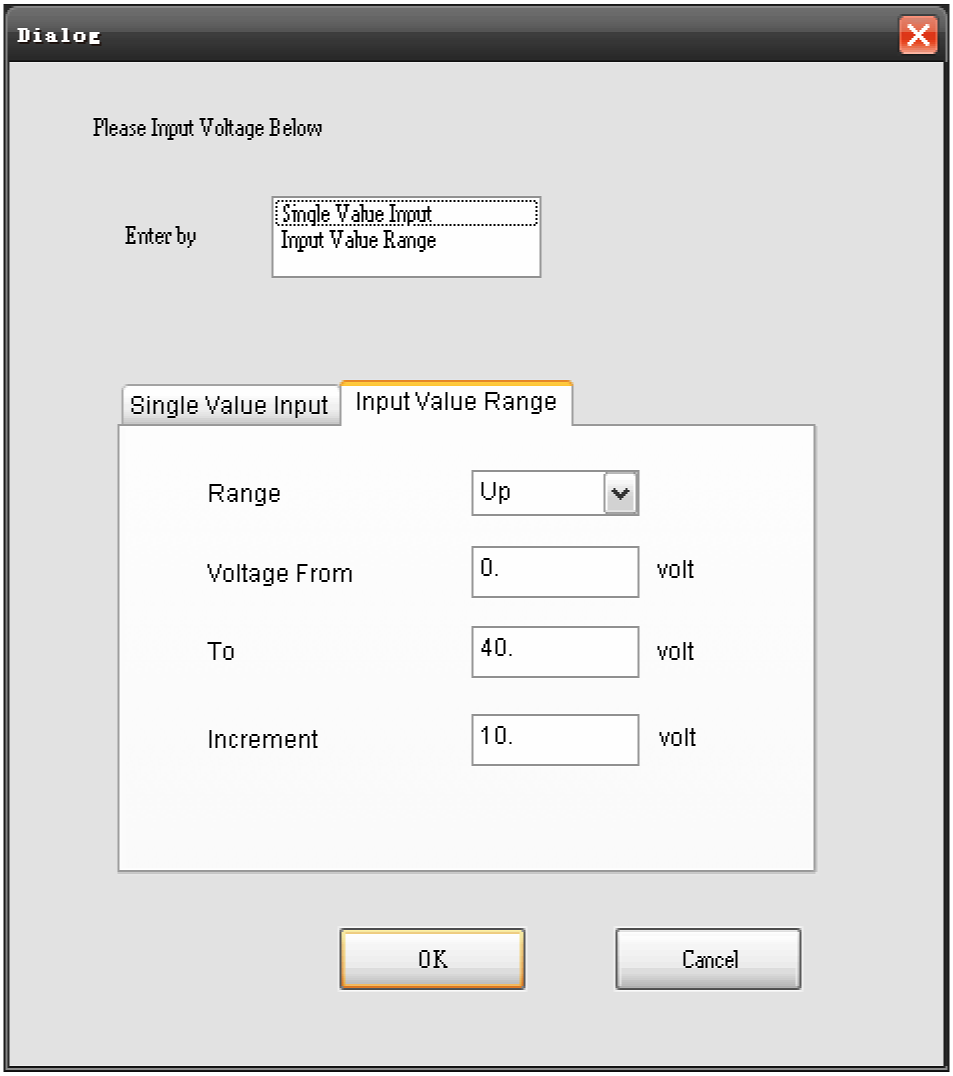
Option
Frequency Modes Number: 6
Displacement: ● Small ○ Large
Start Shape: ● Undeformed ○ Previously Deformed
Convergence Definition
Iteration Number: 100
Iteration Accuracy: 0.005

Figure 67 Simulation Settings

Click *OK*

Click *Loads…Voltage…Entity*

The voltage on the moving entity will remain at 0 V. Click on the fixed comb drive element. Click the “Input Value Range” tab in the dialog that appears, and set the voltage to sweep from 0 to 40 V as shown below.



Range: Up

Voltage From: 0. volt

To: 40. volt

Increment: 10. volt

Figure 68 Voltage Settings

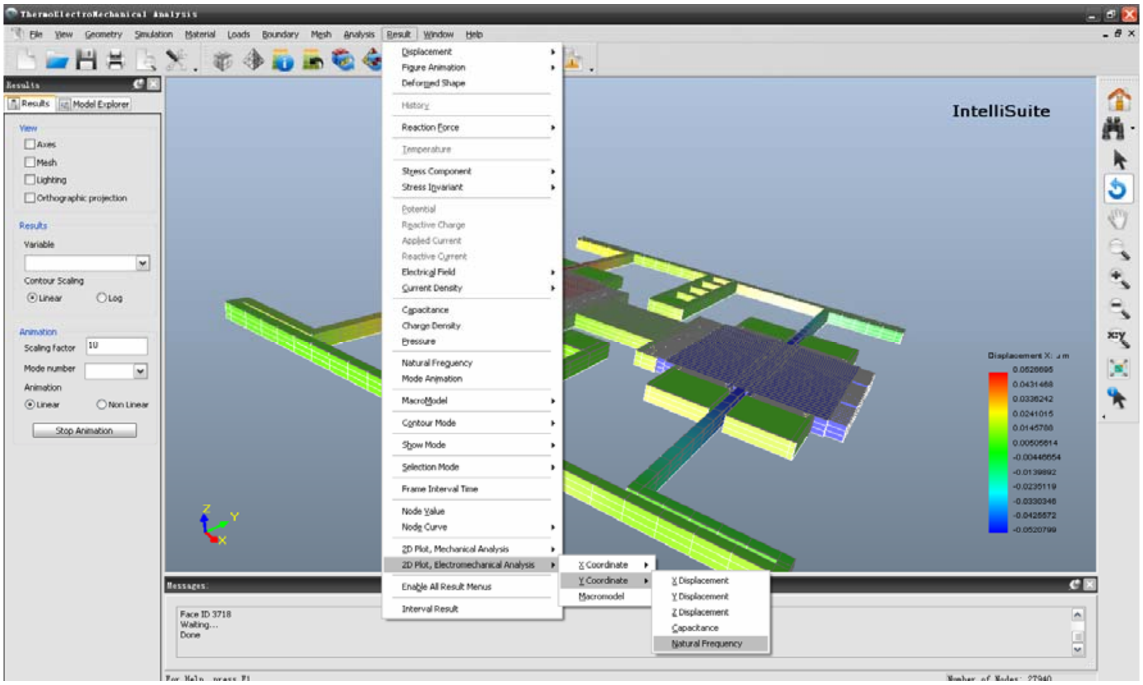
Click *OK*

To run the simulation,

Click *Analysis…Start Frequency Analysis*

When the simulation is complete,

Click *Result…2D Plot, Electromechanical Analysis… Y Coordinate…Natural Frequency*



```
Face ID 3718
Waiting...
Done
```

```
For Help, press F1                                    Number of Nodes: 27940
```

Figure 69 Results Menu

In the window that appears, select modes 3 and 4 (drive and sense modes). The results will appear as shown below.

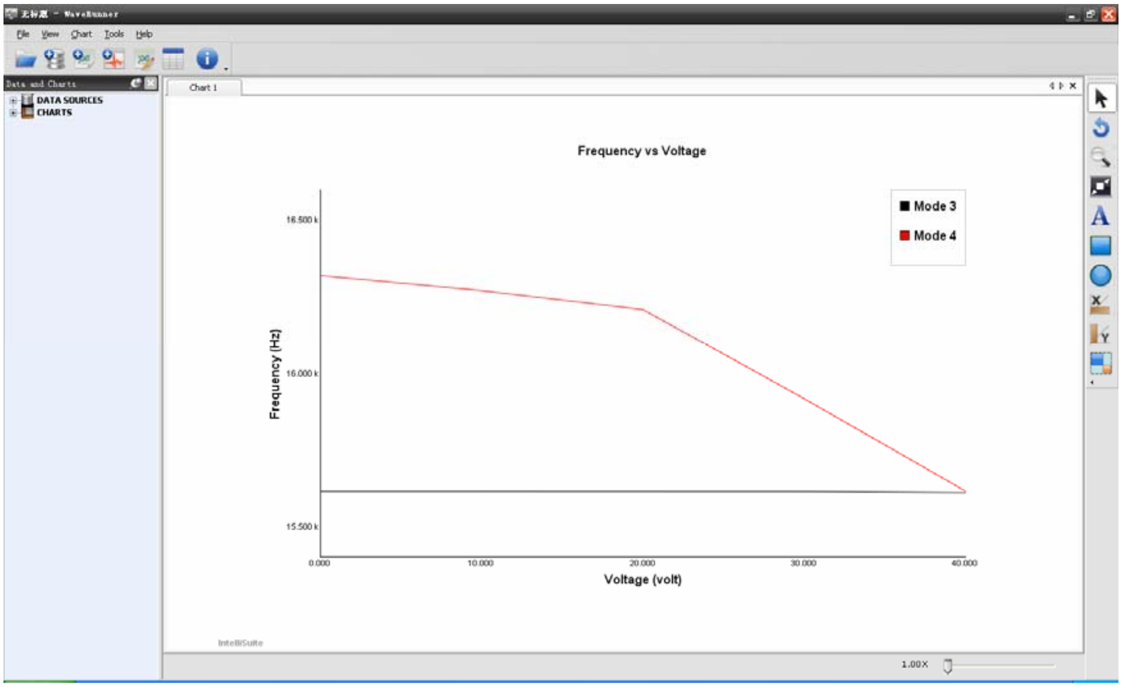


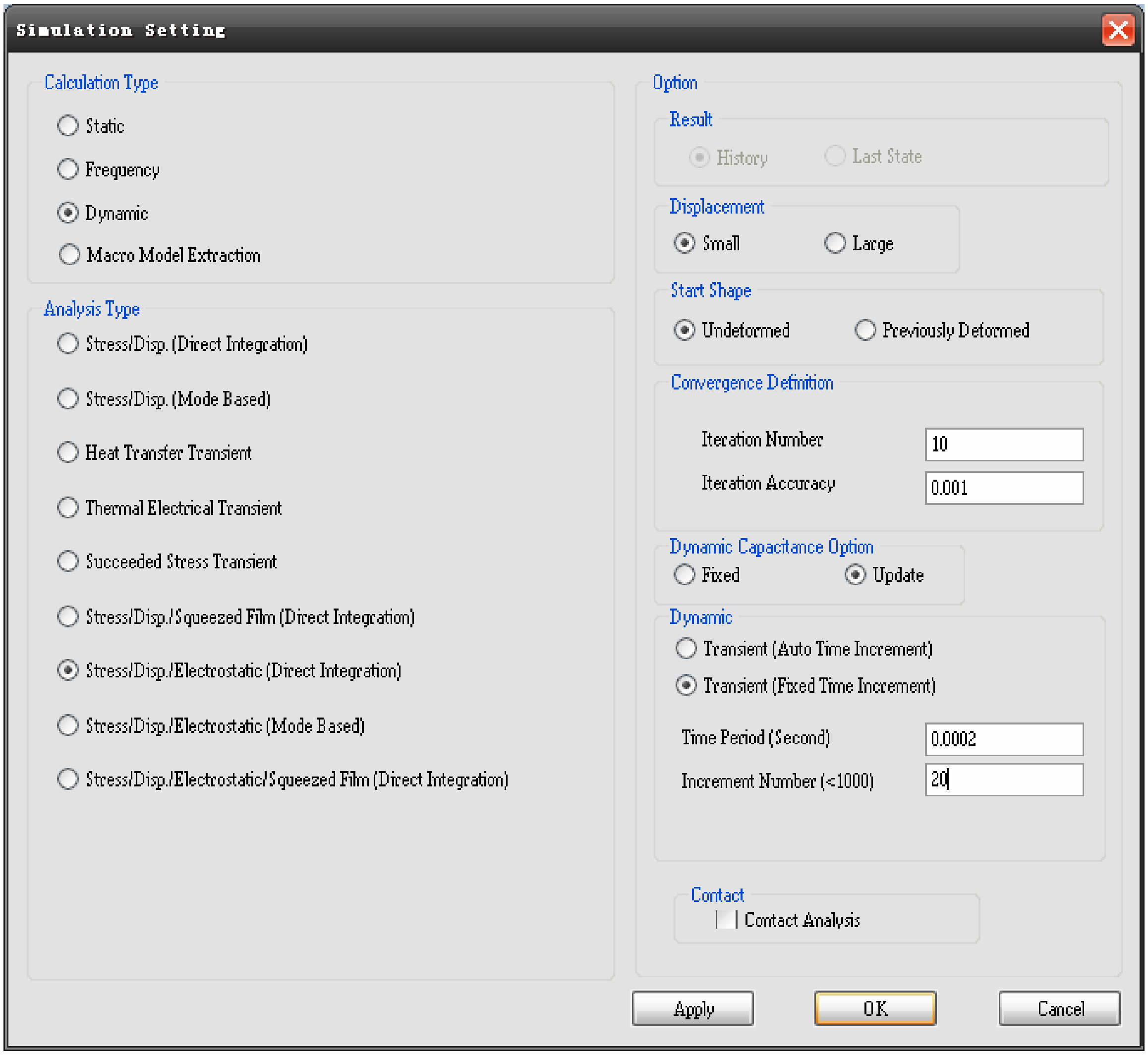
Figure 70 Frequency vs. Voltage

In the above figure we see that under a certain bias voltage, the resonant frequency of the sense mode is near the resonant frequency of the drive mode. This kind of gyro is called a mode-matched tuning-fork gyroscope (M^2-TFG). The greatest advantage of an M^2-TFG is that the rotation-induced Coriolis signal is amplified by the mechanical quality factor of the sense mode. Due to the Q factor amplification, an M^2-TFG operated under the mode-matched configuration will offer a higher sensitivity and better resolution.

4.5. Dynamic Simulation

The static simulation is complete, now it is time to run the device under a dynamic load. How will we actuate the device? We will use the two drive combs, but we have to determine the frequency of the AC voltage source. We want to actuate the gyro at one of the natural frequencies of the device. With this gyro, the third mode (when the masses move in opposite directions along the x-axis) is the drive mode. This mode has a frequency of 15,615 Hz. Because we want to have a sampling rate of about 10-15 points per cycle we need to have a sampling rate of ~200,000 Hz. We will examine the first five cycles, so we will look at the first 200 microseconds and examine 20 steps for the startup.

Set the simulation as shown below.



Simulation Setting

Calculation Type
○ Static
○ Frequency
● Dynamic
○ Macro Model Extraction

Analysis Type
○ Stress/Disp. (Direct Integration)
○ Stress/Disp. (Mode Based)
○ Heat Transfer Transient
○ Thermal Electrical Transient
○ Succeeded Stress Transient
○ Stress/Disp./Squeezed Film (Direct Integration)
● Stress/Disp./Electrostatic (Direct Integration)
○ Stress/Disp./Electrostatic (Mode Based)
○ Stress/Disp./Electrostatic/Squeezed Film (Direct Integration)

Option

Result
● History ○ Last State

Displacement
● Small ○ Large

Start Shape
● Undeformed ○ Previously Deformed

Convergence Definition
Iteration Number: 10
Iteration Accuracy: 0.001

Dynamic Capacitance Option
○ Fixed ● Update

Dynamic
○ Transient (Auto Time Increment)
● Transient (Fixed Time Increment)
Time Period (Second): 0.0002
Increment Number (<1000): 20

Contact
☐ Contact Analysis

Figure 71 Dynamic Simulation Settings

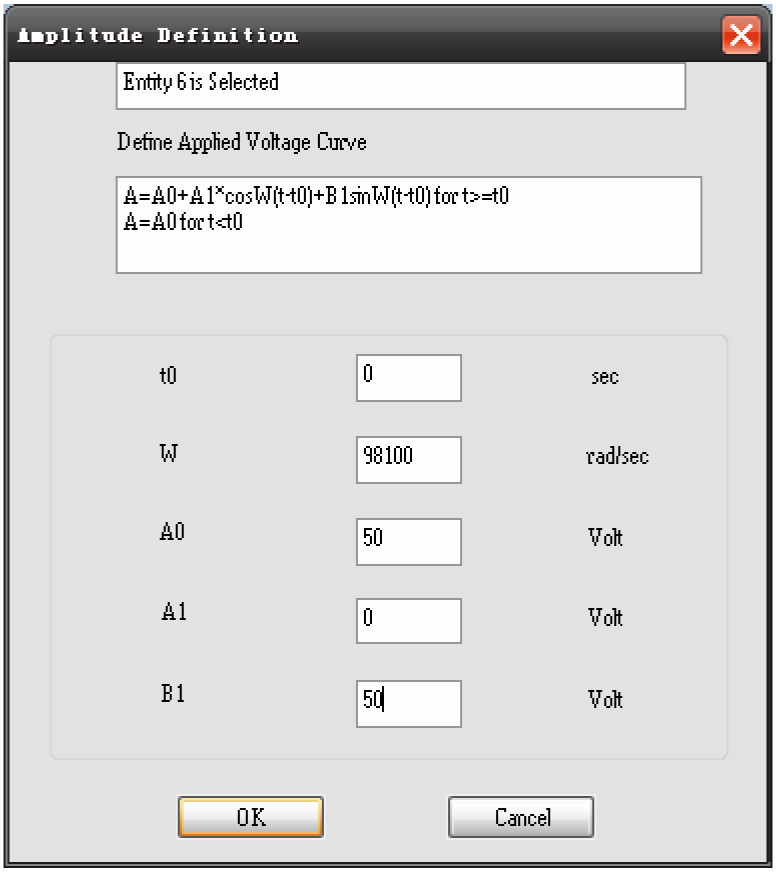
First, we have to reset the static voltages on the drive combs.

Click *Loads…Voltage…Entity*

Select the fixed comb drive entity and change the voltage from 100V to 0V. The next step is to apply the AC voltage.

Click *Loads…Amplitude vs. Time…Periodic*
Click *Loads…Voltage…Entity*

Select the fixed comb drive entity and set the voltage as shown in ***Figure 72***.



Amplitude Definition

Entity 6 is Selected

Define Applied Voltage Curve

A=A0+A1*cosW(t-t0)+B1sinW(t-t0) for t>=t0
A=A0 for t<t0

t0: 0 sec
W: 98100 rad/sec
A0: 50 Volt
A1: 0 Volt
B1: 50 Volt

Figure 72 Dynamic Voltage Settings

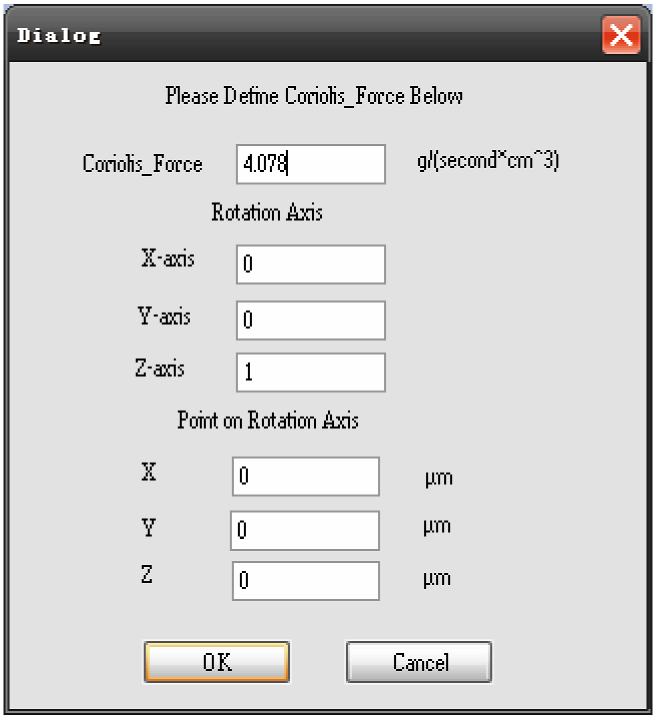
We also have to set up the dynamic damping definitions.

Click *Material…Damping Definition*

Select the moving entity and input 78 for the *Mass_damping* factor and 0 for the *Stiffness_damping*. This corresponds to a damping ratio of 0.005 for both the drive (15.6 kHz) and sense (16.3 kHz) natural frequencies. (See the TEM Help in the *Help* menu for more information on how to calculate these values.) Once the damping definitions have been set, we need to set up the Coriolis force resulting from the rotation of the device.

Click *Loads…Amplitude vs. Time…None*
Click *Loads…Coriolis Force*

Select the moving mass (large entity) and input the below values for the Coriolis force.



Please Define Coriolis_Force Below

Coriolis_Force 4.078 g/(second*cm^3)

Rotation Axis

X-axis 0

Y-axis 0

Z-axis 1

Point on Rotation Axis

X 0 μm

Y 0 μm

Z 0 μm

Figure 73 Coriolis Force Settings

The magnitude of the Coriolis force load is defined as the mass density multiplied by the angular velocity. For the current structure, the density is 2.33 g/cm^3. If the angular velocity is 1.75 rad/sec (100 deg/sec), then the Coriolis force can be defined as 4.078 g/(second* cm^3). The z-axis (0, 0,1) is the axis of rotation.

Finally, it is time to run the transient simulation.

Click *Analysis…Start Dynamic Analysis*

The dynamic analysis will take about 24 hours, possibly more. The results are displayed below. If you want to view the results on your computer without having to run the simulation, send us an email (support@intellisense.com) to request the result file. This file is quite large (~250MB) which is why it is not included in your installation.

The important dynamic results, the displacements in the x- and y- directions of the two moving masses, are shown below.

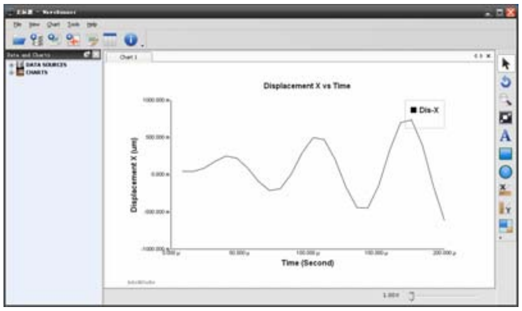


Figure 74 X-Displacement on Mass 1

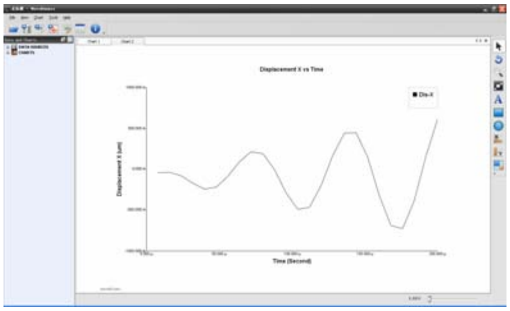


Figure 75 X-Displacement on Mass 2

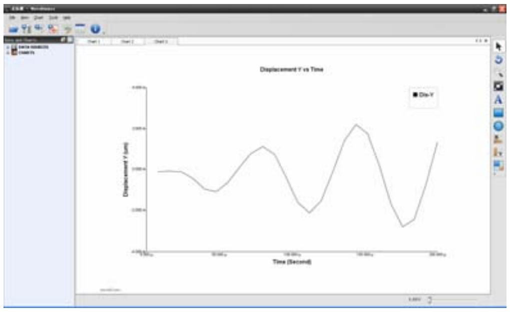


Figure 76 Y-Displacement on Mass 1

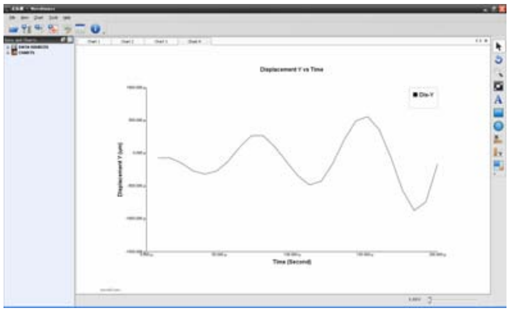


Figure 77 Y-Displacement on Mass 2

You'll notice that the two masses move opposite each other along the x-axis and in the same direction along the y-axis. It is also important to notice that we have only run the simulation for three cycles. This is too short of a time frame to determine the full start-up effect of the gyro. We may need to run fifty cycles or more to see the full response. With the simulation of three cycles taking 24 hours to complete, you can easily see why system modeling using a reduced-order model is so important. With the System Model Extraction feature, we can generate a system model of the device in less than 2 hours. We can then import that model into SYNPLE and run a full transient analysis of the start-up response in a matter of seconds. In the next section we will demonstrate how to extract a system model and incorporate that model into SYNPLE.

4.6. System Model Extraction

System Model Extraction is a means by which a full three-dimensional meshed numerical model of a multi-conductor electromechanical device without dissipation can be converted into a reduced-order analytical macromodel. This can then be inserted as a black-box element into a mixed signal circuit simulator. This process is based upon the energy method approach, in that we construct analytical models for each of the energy domains of the system and determine all forces as gradients of the energy.

The energy method approach has the advantage of making this process modular, enabling us to incorporate other energy domains into our models in the future. Another beneficial side effect of energy methods is that the models we construct are guaranteed to be energy conserving, because each of the stored energies is constructed as an analytical function, and all forces are computed directly from analytically computed gradients. The SME process also has the advantage of being able to be performed almost entirely automatically, requiring the designer only to construct the model, run a few full three-dimensional numerical computations, and set a few preferences a priori. Above all, this process has the ultimate benefit of constructing models that are computationally efficient, allowing their use in a dynamical simulator.

Our first task is to reduce the degrees of freedom of the system. Rather than allow each node in a finite element model to be free to move in any direction, we constrain the motion of the system to a linear superposition of selected set of deformation shapes. This set will act as our basis set of motion. The positional state of the system will hence be reduced to a set of generalized coordinates, each coordinate being the scaling factor by which its corresponding basis shape will contribute. Next, we must construct analytical macromodels of each of the energy domains of the system. In the case of conservative capacitive electromechanical systems, these consist of the electrostatic, elastostatic, and kinetic energy domains. These macromodels will be analytical functions of the generalized coordinates. (As we will see in the section on Using Mode Shapes as a Basis Set, some of these energy domains will be determined as a byproduct of modal analysis, avoiding the need for explicit calculation.) We can then use Lagrangian mechanics to construct the equations of motion for the system in terms of its generalized coordinates. Finally, we can translate these equations of motion into an analog hardware description language, thereby constructing a black-box model of the electromechanical system that can be inserted into an analog circuit simulator.

The Figure below gives the Flow Chart for the conversion of an FEA model into an equivalent system level mode.

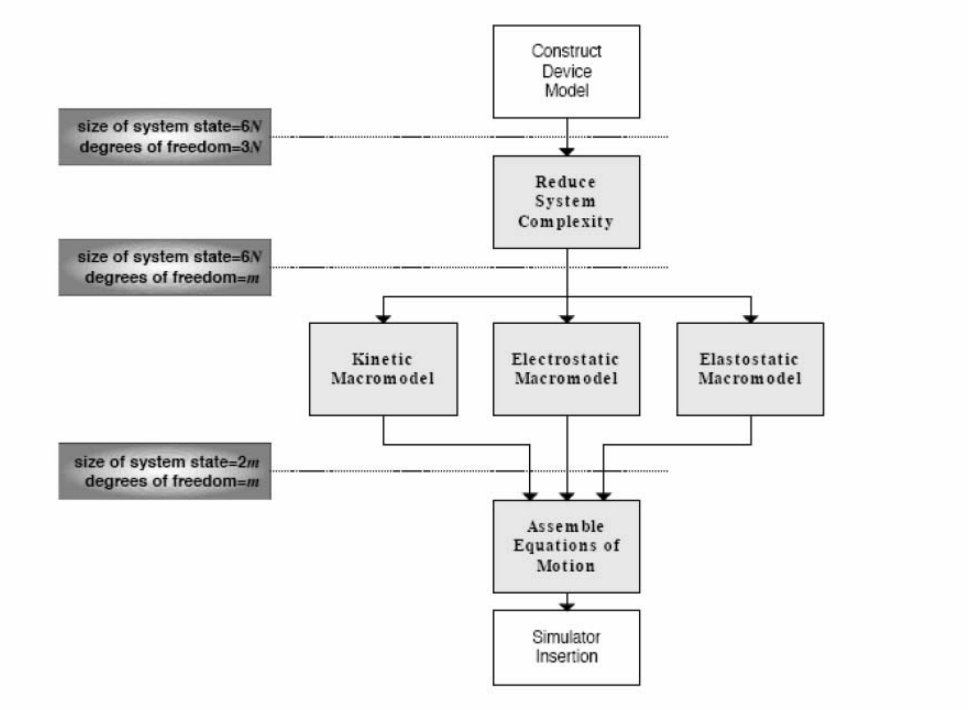


Figure 78 Creation of a Macromodel

Some of the key equations used for the conversion process are discussed in this section. In general, the deformation state and dynamics of mechanical system can be accurately described as the linear combination of mode shape function or modal superposition.

$$u_i(t, x_i, y_i, z_i) = u_{eq} + \sum_{j=1}^{m} q_j(t) \cdot \Phi_j(x_i, y_i, z_i) \tag{1}$$

Where u_i represents the deformed state of the structure, u_{eq} represent the initial equilibrium state (derived from the residual stress conditions without external loads), Φ_j represents the displacement vector for the jth mode, q_j represents the coefficients for the jth mode, which is referred as "scaling factor for mode j". In general, (Eq. 1) describes a coordinate transformation of finite element displacement coordinates to modal coordinates of the macromodel. The deformation state of the structure given by n nodal displacements $u_i(i=1,2,\ldots,n)$ is now represented by a linear combination of m mode weighted by their amplitudes $q_j(i=1,2,\ldots,n)$ where $m<<n$. The governing equation of motion describing the ROM of electrostatic actuated MEMS structures in modal coordinates is given by:

$$m_j \ddot{q}_j + 2\xi_j \omega_j m_j \dot{q}_j + \frac{\partial W_{st}}{\partial q_j} = \frac{1}{2} \sum_r \frac{\partial C_{ks}}{\partial q_j} \cdot (V_k - V_s)^2 + \sum_{i=1}^{n} \Phi_j^i \cdot F_i \tag{2}$$

Where m_j is the modal generalized mass, ω_j the modal eigenfrequency, ξ_j the linear modal damping ratio, W_{st} the modal strain energy function, C_{ks} the modal capacity-stroke function, r the number of capacities involved for Microsystems with multiple electrodes, V_{ks} the electrode voltage applied between electrode k and s, and F_i a local force acting at the *i-th* node. The modal strain energy and capacity-stroke functions are derived from a series of FE runs at various deflection states in the operating range.

The modal superposition method is efficient since just one equation per mode and one equation per conductor is necessary to describe the coupled system entirely, which can be applied to both linear and nonlinear geometry.

The modal superposition based reduced order modeling procedure includes the following steps:

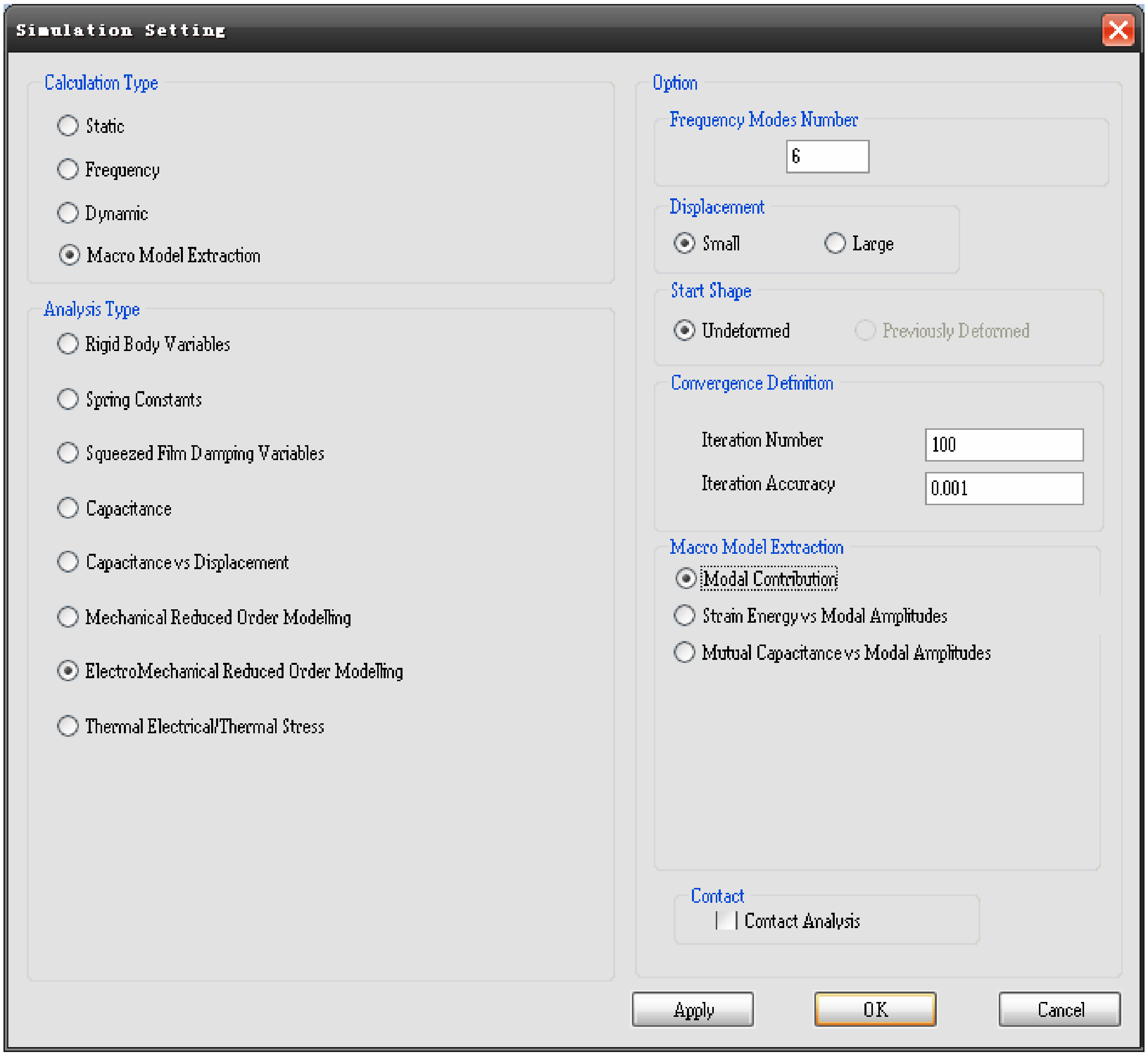
- Determine the “Modal Contribution”. In this step, the software performs the standard electromechanical relaxation analysis and solves the initial deformed state (derived from the residual stress without external loads) and the final deformed state (with mechanical loads and applied voltages). It then uses the QR factorization algorithm to determine the modal contribution for the deformed state.
- Calculate the relationship of “strain energy vs modal amplitudes” for each mode.
- Calculate the relationship of “mutual capacitance vs modal amplitudes”.
- From step 2 and 3, the user will obtain $\partial W_{st}(q)/\partial q_j$ and $\partial C(q)/\partial q_j$ respectively.

4.6.1 Extracting the Electromechanical Macromodel in TEM

We will be performing a Macromodel Extraction so set the simulation as such.

Click *Simulation…Simulation Setting*

Set up the simulation as shown below.



Simulation Setting

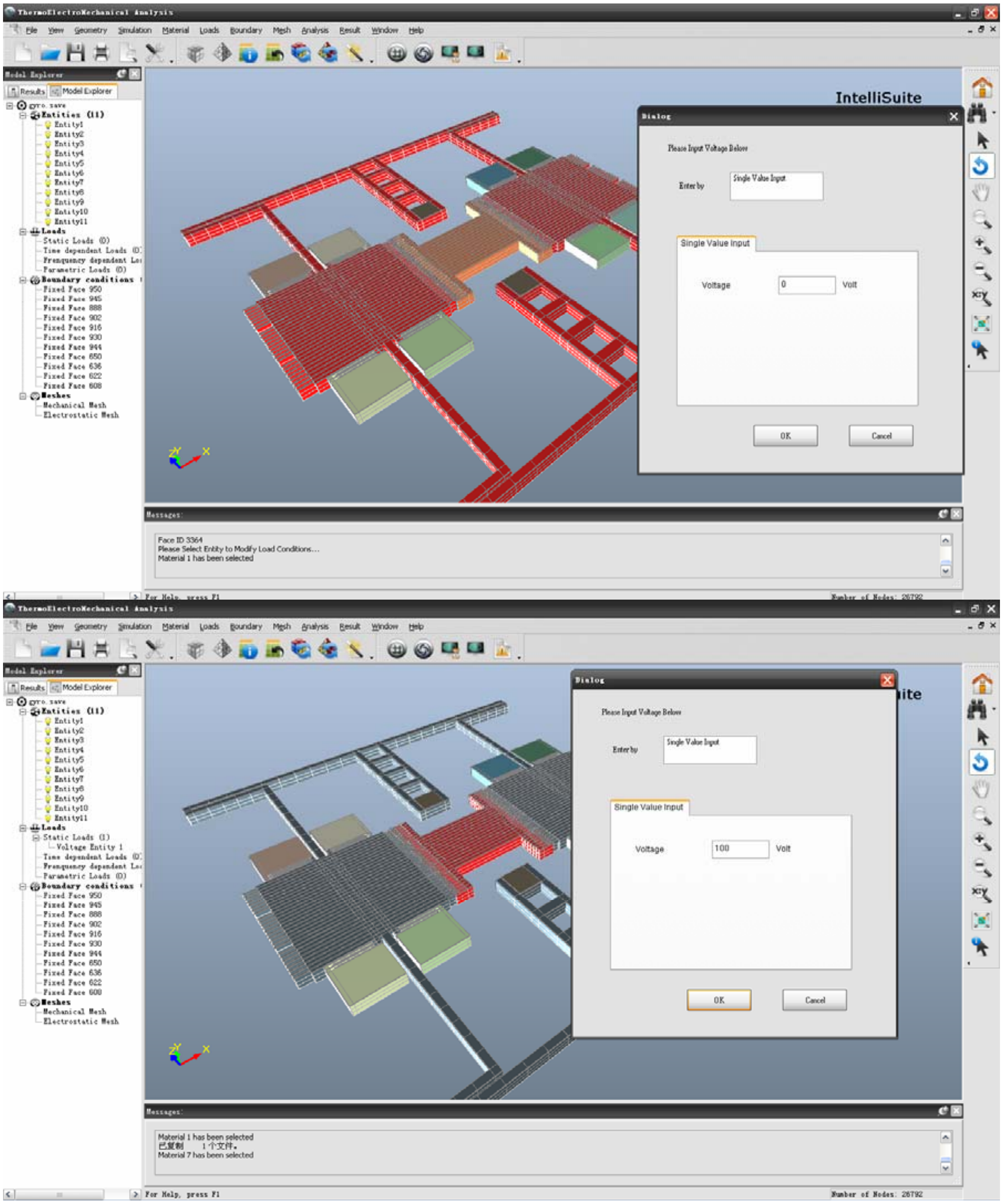
Calculation Type
○ Static
○ Frequency
○ Dynamic
● Macro Model Extraction

Analysis Type
○ Rigid Body Variables
○ Spring Constants
○ Squeezed Film Damping Variables
○ Capacitance
○ Capacitance vs Displacement
○ Mechanical Reduced Order Modelling
● ElectroMechanical Reduced Order Modelling
○ Thermal Electrical/Thermal Stress

Option
Frequency Modes Number: 6
Displacement: ● Small ○ Large
Start Shape: ● Undeformed ○ Previously Deformed
Convergence Definition
Iteration Number: 100
Iteration Accuracy: 0.001
Macro Model Extraction
● Modal Contribution
○ Strain Energy vs Modal Amplitudes
○ Mutual Capacitance vs Modal Amplitudes
Contact
☐ Contact Analysis

Figure 79 ElectroMechanical Macromodel Extraction Simulation Settings

Now that the simulation is set up, check that the material properties and boundaries are properly set as described in the static simulation setup on page 38. We will apply a voltage load while calculating the modal contribution. This is especially useful if you do not know exactly which modes will be important. If you apply loads that approximate the loading conditions you expect to see in the actuation of the device, you will be able to determine which modes contribute most to the deformation caused by the actuation loads. This allows you to run your macromodel extraction while focusing on the correct modes. Apply a voltage load of 100 V to the comb drive entity and 0 V to the moving mass.



Please Input Voltage Below

Enter by: Single Value Input

Single Value Input

Voltage 0 Volt

Messages:

Face ID 3364
Please Select Entity to Modify Load Conditions…
Material 1 has been selected

Please Input Voltage Below

Enter by: Single Value Input

Single Value Input

Voltage 100 Volt

Messages:

Material 1 has been selected
已复制 1 个文件。
Material 7 has been selected

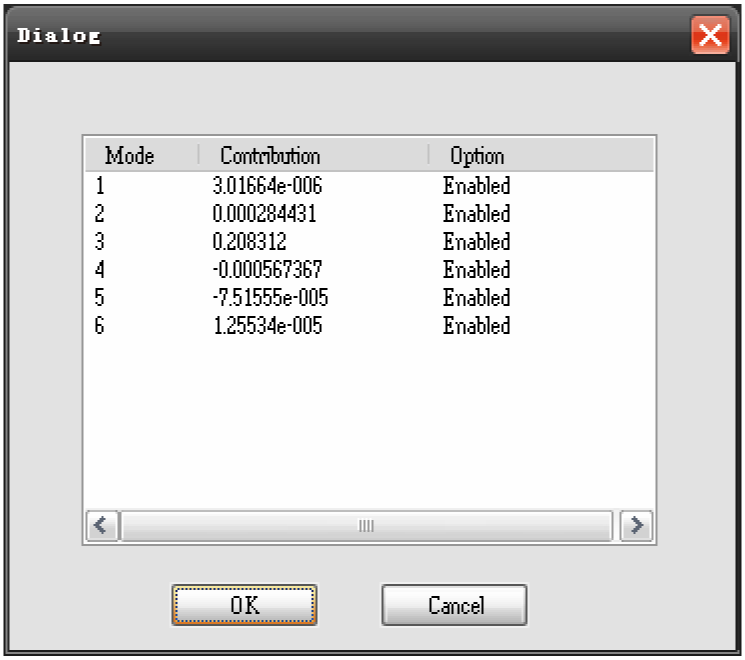
Figure 80 Voltage Loads

Once the loads have been set, run the analysis.

Click *Analysis…Start Extract Macromodel*

When the analysis is complete (it should take only a few minutes) you can view the contribution of each mode to the deformation of the device.

Click *Result…MacroModel…Modal Contribution*



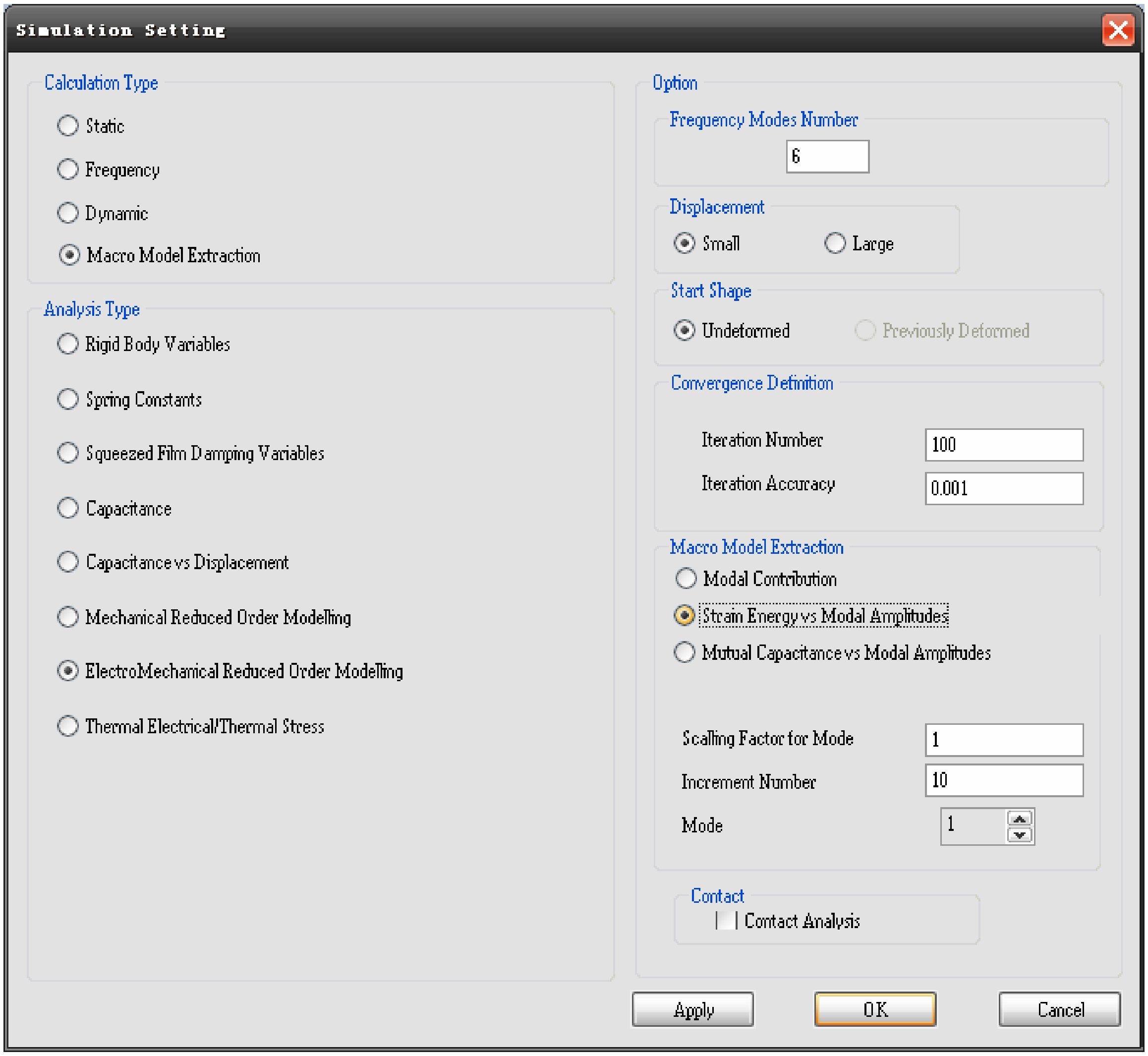
Mode	Contribution	Option
1	3.01664e-006	Enabled
2	0.000284431	Enabled
3	0.208312	Enabled
4	-0.000567367	Enabled
5	-7.51555e-005	Enabled
6	1.25534e-005	Enabled

Figure 81 Modal Contributions

In this example we will use all six of the modes we have examined in our strain energy calculation step. If you wanted to reduce the time needed for the strain energy step, you could easily disable any of the modes so they would be ignored during the strain energy simulation.

Click *Simulation...Simulation Setting*

Set up the strain energy step as shown below.



Calculation Type

○ Static
○ Frequency
○ Dynamic
● Macro Model Extraction

Analysis Type

○ Rigid Body Variables
○ Spring Constants
○ Squeezed Film Damping Variables
○ Capacitance
○ Capacitance vs Displacement
○ Mechanical Reduced Order Modelling
● ElectroMechanical Reduced Order Modelling
○ Thermal Electrical/Thermal Stress

Option

Frequency Modes Number: 6

Displacement: ● Small ○ Large

Start Shape: ● Undeformed ○ Previously Deformed

Convergence Definition

Iteration Number: 100
Iteration Accuracy: 0.001

Macro Model Extraction

○ Modal Contribution
● Strain Energy vs Modal Amplitudes
○ Mutual Capacitance vs Modal Amplitudes

Scalling Factor for Mode: 1
Increment Number: 10
Mode: 1

Contact

☐ Contact Analysis

Figure 82 Strain Energy Simulation Settings

Leave the scaling factor and increment number as the default values for modes 1, 2, 4, 5 and 6. For mode 3, set the scaling factor to 5 and the increment number to 25 as shown above. This will examine the strain energy for a wider range of motion for mode 3. Because mode 3 is the drive mode, it will be the mode that will experience the largest range of motion under actuation conditions. Once you have set up the simulation for the strain energy calculation, you are ready to run the analysis, as all of the material properties, loads, and boundaries have already been set up.

Click *Analysis…Start Extract Macromodel*

When the analysis is complete you can see the strain energy vs. modal amplitudes graph for each mode.

Click *Result…MacroModel…Strain Energy vs Modal Amplitudes*

To show a plot of the strain energy vs. scaling factor for a particular mode, double-click on the mode number and click OK.

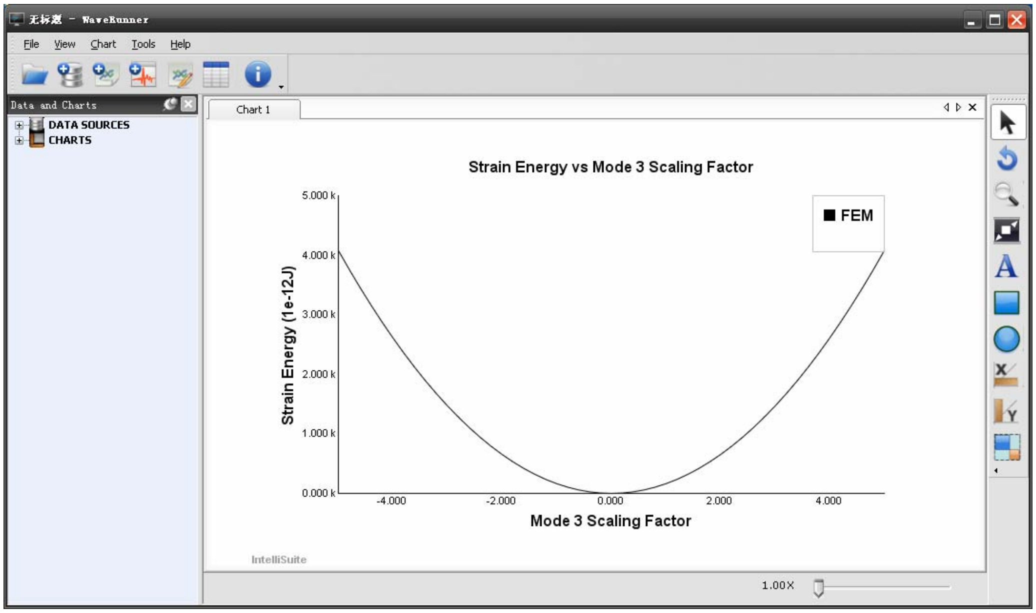
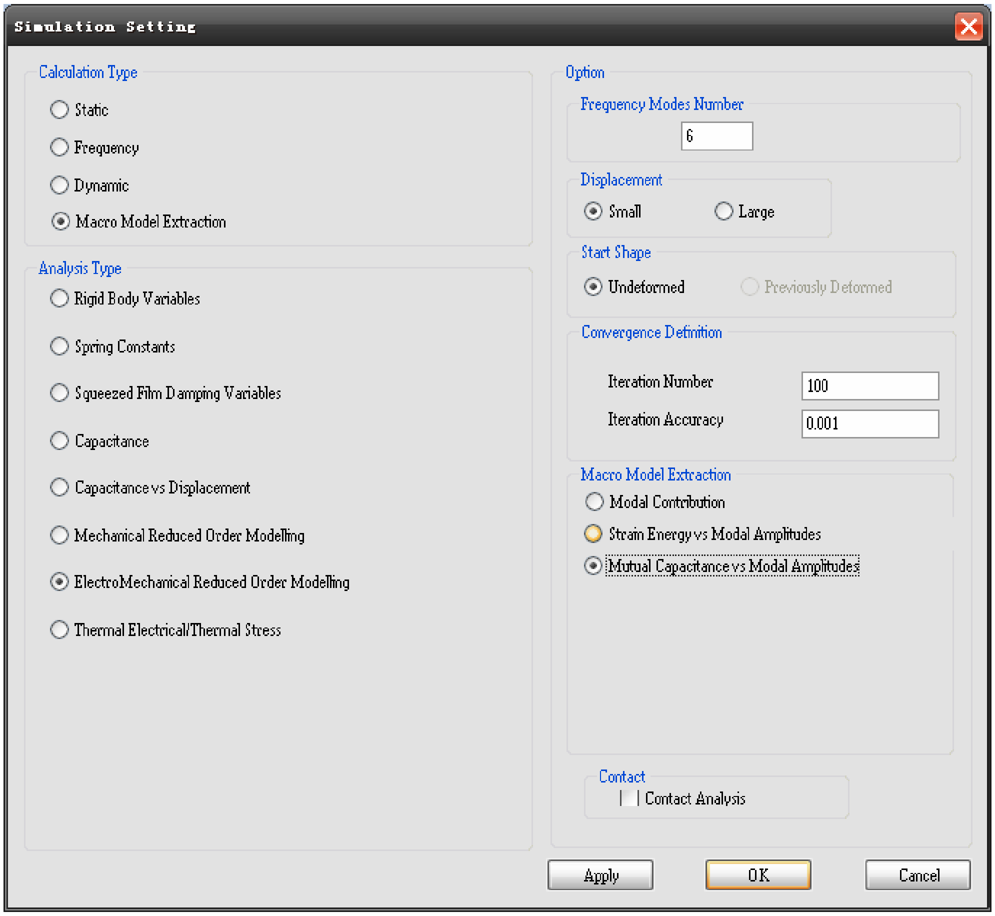


Figure 83 Strain Energy vs. Mode 3 Scaling Factor

We will now set up the final simulation step. Click *Simulation…Simulation Setting* and set up the simulation as shown below.


Simulation Setting

Calculation Type
- ○ Static
- ○ Frequency
- ○ Dynamic
- ● Macro Model Extraction

Analysis Type
- ○ Rigid Body Variables
- ○ Spring Constants
- ○ Squeezed Film Damping Variables
- ○ Capacitance
- ○ Capacitance vs Displacement
- ○ Mechanical Reduced Order Modelling
- ● ElectroMechanical Reduced Order Modelling
- ○ Thermal Electrical/Thermal Stress

Option

Frequency Modes Number: 6

Displacement: ● Small ○ Large

Start Shape: ● Undeformed ○ Previously Deformed

Convergence Definition
- Iteration Number: 100
- Iteration Accuracy: 0.001

Macro Model Extraction
- ○ Modal Contribution
- ○ Strain Energy vs Modal Amplitudes
- ● Mutual Capacitance vs Modal Amplitudes

Contact
- ☐ Contact Analysis

Figure 84 Capacitance Simulation Settings

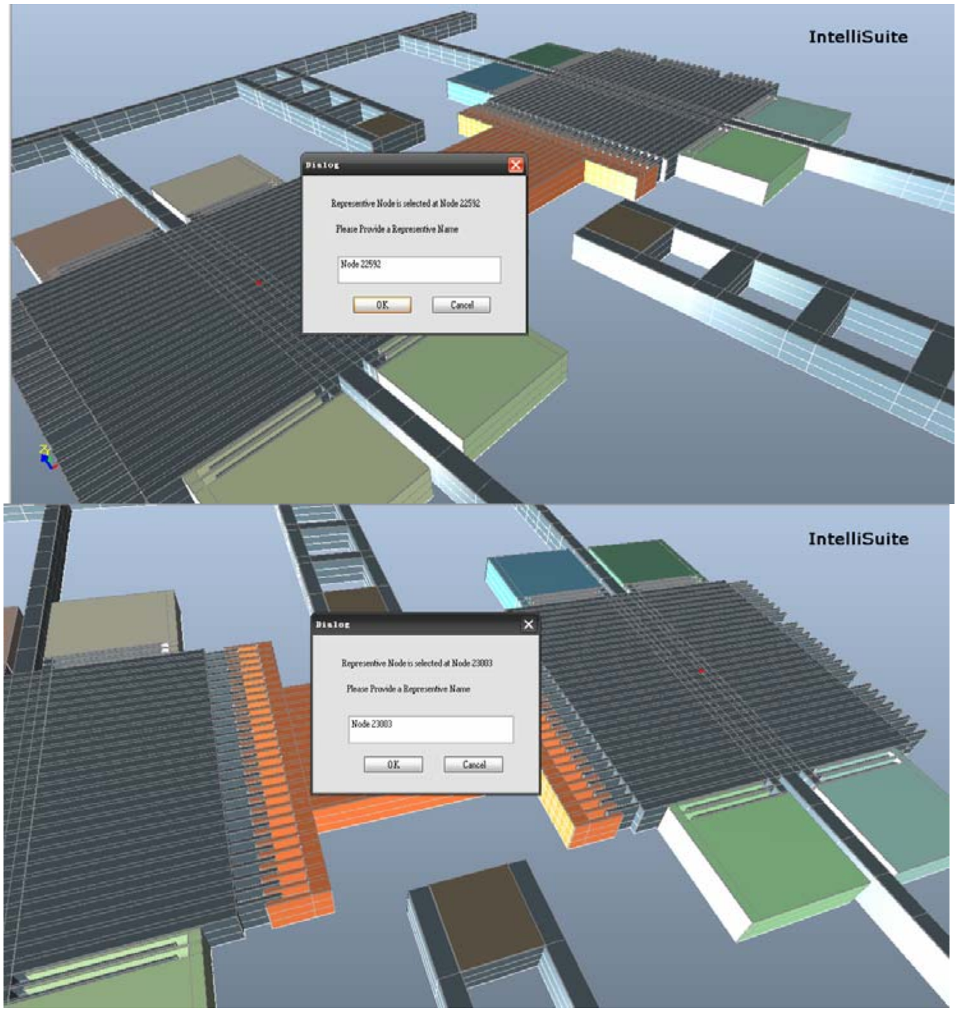
Click *OK*

Click *Analysis…Start Extract Macromodel*

After the simulation has run, we need to select the representative nodes that will be used to view the output displacements and apply any mechanical loads that we need. In this case we will look at the center points of each of the two proof masses.

Click *Boundary…Macromodel…Representative Nodes*

Select the nodes shown in the figure below. The representative node information will be saved in the file named "macmodel.out".



Representive Node is selected at Node 22592

Please Provide a Representive Name

Node 22592

Representive Node is selected at Node 23003

Please Provide a Representive Name

Node 23003

Figure 85 Representative Nodes

When you have selected the representative nodes the macromodel extraction is complete. Copy the "curr.macromodel," "macromodel.out," and "str.out" files from the active directory and place them in a new directory for use with SYNPLE. Be sure to move these files to a new folder, as they will be overwritten in the current directory if you run another simulation.

5. Macromodel Simulation in SYNPLE

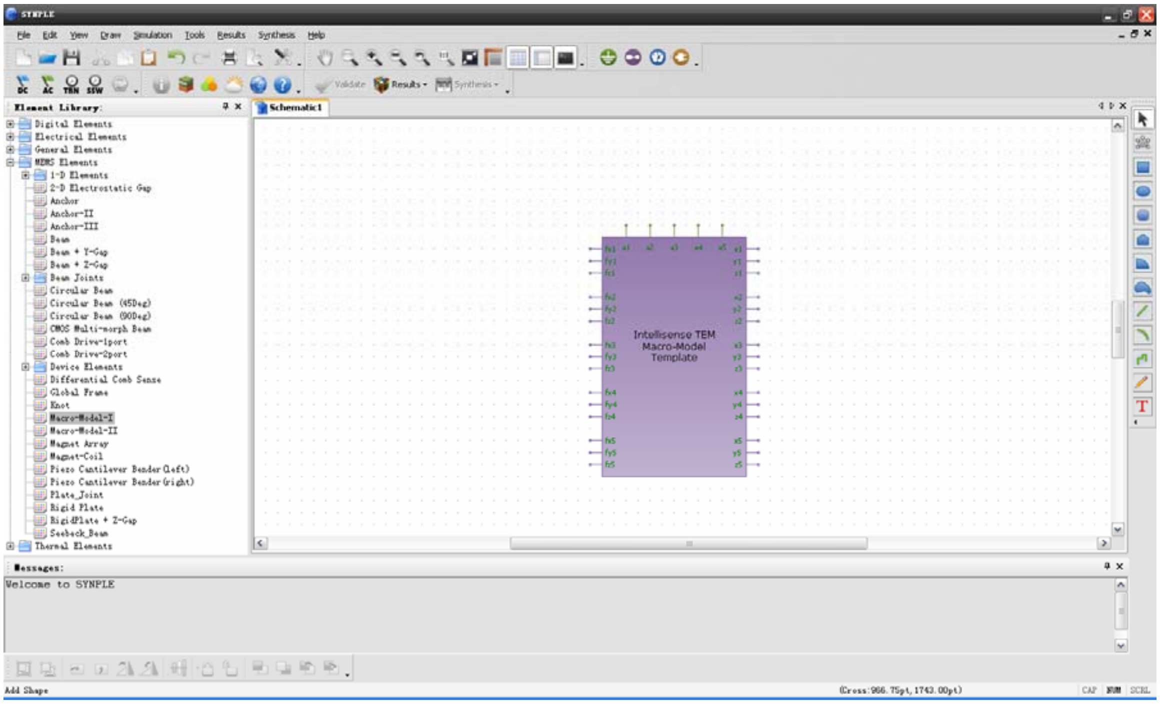
You'll remember from earlier that the dynamic simulation using finite element methods took multiple hours to complete. The system model extraction when run efficiently should take you less than one hour. You'll see from this section that constructing a SYNPLE schematic and running a simulation takes very little time as well.

5.1. Macromodel Setup

First, open SYNPLE.

Click *Start…Programs…IntelliSuite…SYNPLE*

Click and drag the *Macro-Model-I* element from the *MEMS Devices* element library on the left into the drawing window.



Welcome to SYNPLE

Figure 86 Macromodel Element

The next step is to link the macromodel element to the macromodel file we have extracted. Click on the Macromodel element to highlight it.

Click *Tools…Load ROM Macro-model*

To load the macromodel, find the location of your macromodel files, select the "curr.macmodel" file, and click *Open*. This will bring up a verification dialog box to let you know that the macromodel has been properly loaded and linked to the element. You can also use the macromodel file located at

`<Installation Directory>\IntelliSuite\Training\Application_Notes\Gyro\`
`Macromodel\curr.macromodel`.

To validate the macromodel extracted from TEM module, we need to perform a pure mechanical simulation and compare the results to the results obtained in the TEM module. We will perform two analyses here, one AC and one DC. The first step is to build and verify the mechanical system without the electrical components. After we have verified the mechanical system, we will run a transient analysis and then begin to incorporate a Coriolis feedback loop and electrical components for drive and sense.

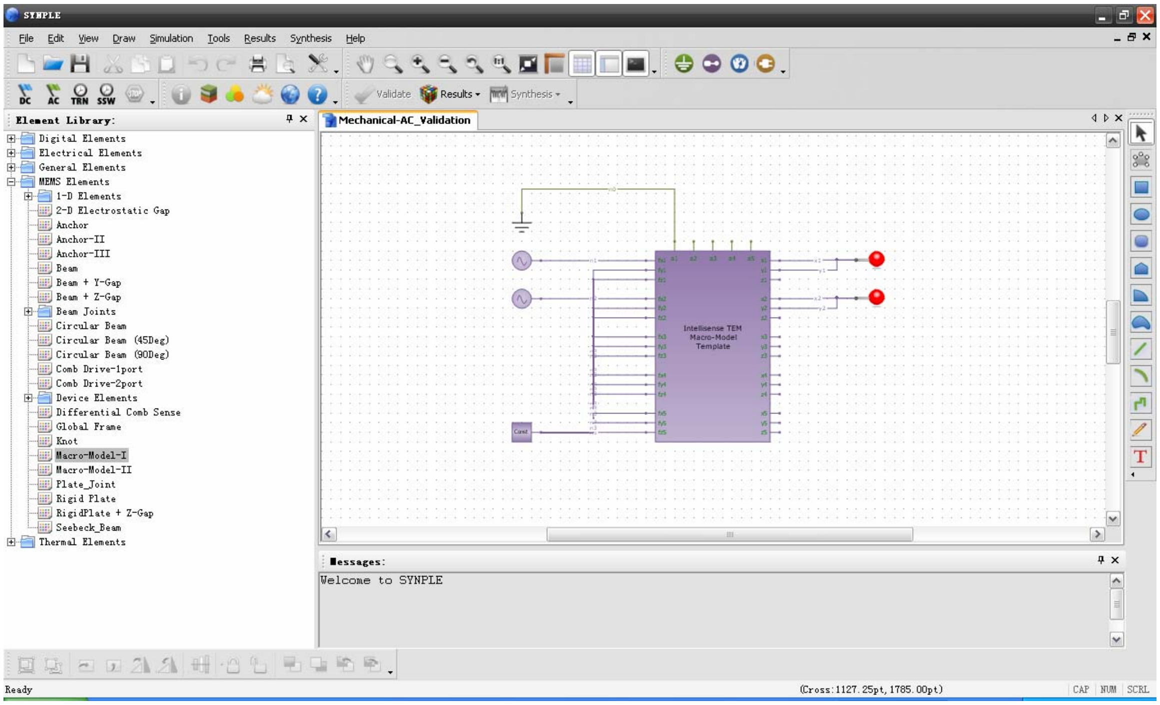
5.2. AC Validation

You will recall from the TEM Natural Frequency simulation that the drive frequency is 15.6 kHz. This can be easily duplicated in SYNPLE. With SYNPLE open,

Click *File…Open*

Open the following file: `<Installation Directory>\IntelliSuite\Training\Application_Notes\Gyro\SYNPLE_Models\Mechanical-AC_Validation.ssc`.

This file should appear as shown below.



Welcome to SYNPLE

Figure 87 SYNPLE Schematic for AC Simulation

You can play around with any of the elements by double clicking on them. You will see an *Electrical_Ground* element used to set the electrical reference node of the macromodel element. Two *General AC Source* elements are used to set up the x-direction drive forces at the representative nodes. One *Constant* element is used to set the forces in non-interesting direction and non-used nodes as zero. Finally, two *Output Probe* elements are included to monitor the displacement of the representative nodes

First, load the macromodel file as discussed earlier.

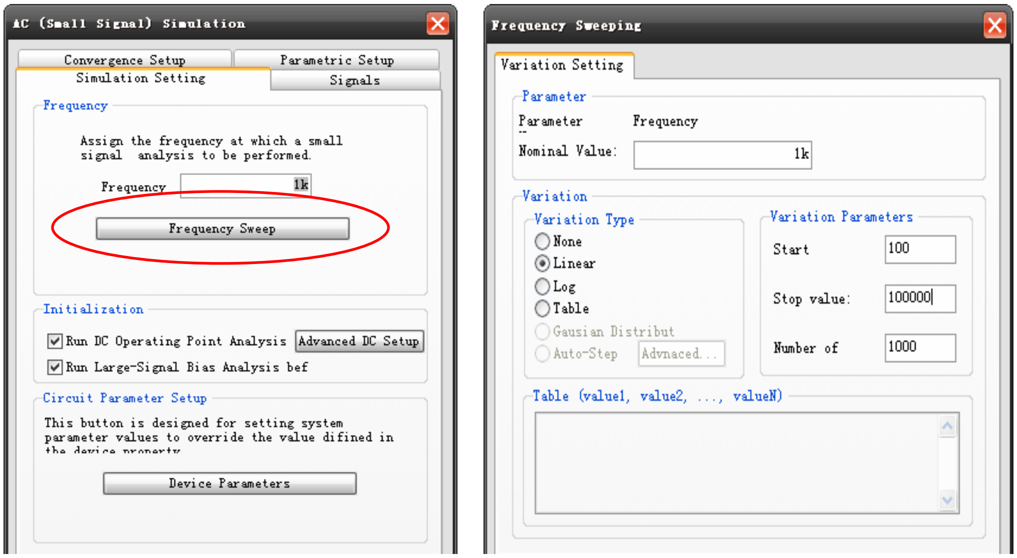
Click *Tools…Load ROM Macro-model*

Load the following macromodel file: `<InstallationDirectory>\IntelliSuite\Training\Application_Notes\Gyro\Macromodel\curr.macromodel`.

Click *Simulation…AC Analysis*

The simulation settings are already set. To view the simulation settings,

Click *Frequency Sweep*



Assign the frequency at which a small signal analysis to be performed.

Frequency: 1k

☑ Run DC Operating Point Analysis

☑ Run Large-Signal Bias Analysis bef

This button is designed for setting system parameter values to override the value difined in the device property

Parameter: Frequency

Nominal Value: 1k

○ None
● Linear
○ Log
○ Table
○ Gausian Distribut
○ Auto-Step Advnaced...

Start: 100

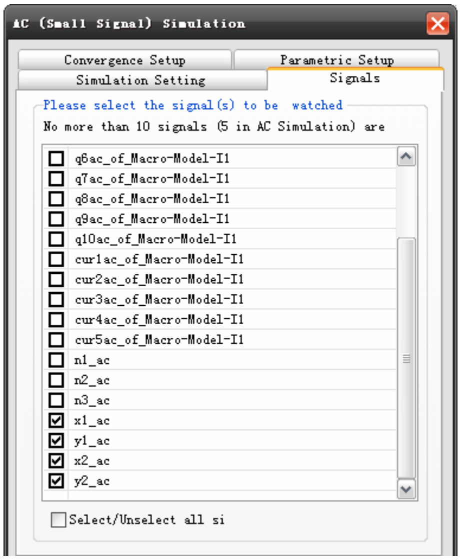
Stop value: 100000

Number of: 1000

Figure 88 Frequency Sweep Selection and Settings

You can see that we will be analyzing 1000 points between 100 Hz and 100,000 Hz.

In the Signals tab, check that we are analyzing the x1 and x2 signals, representing the x-displacement of our representative nodes.



Please select the signal(s) to be watched

No more than 10 signals (5 in AC Simulation) are

☐ q6ac_of_Macro-Model-I1
☐ q7ac_of_Macro-Model-I1
☐ q8ac_of_Macro-Model-I1
☐ q9ac_of_Macro-Model-I1
☐ q10ac_of_Macro-Model-I1
☐ cur1ac_of_Macro-Model-I1
☐ cur2ac_of_Macro-Model-I1
☐ cur3ac_of_Macro-Model-I1
☐ cur4ac_of_Macro-Model-I1
☐ cur5ac_of_Macro-Model-I1
☐ n1_ac
☐ n2_ac
☐ n3_ac
☑ x1_ac
☑ y1_ac
☑ x2_ac
☑ y2_ac

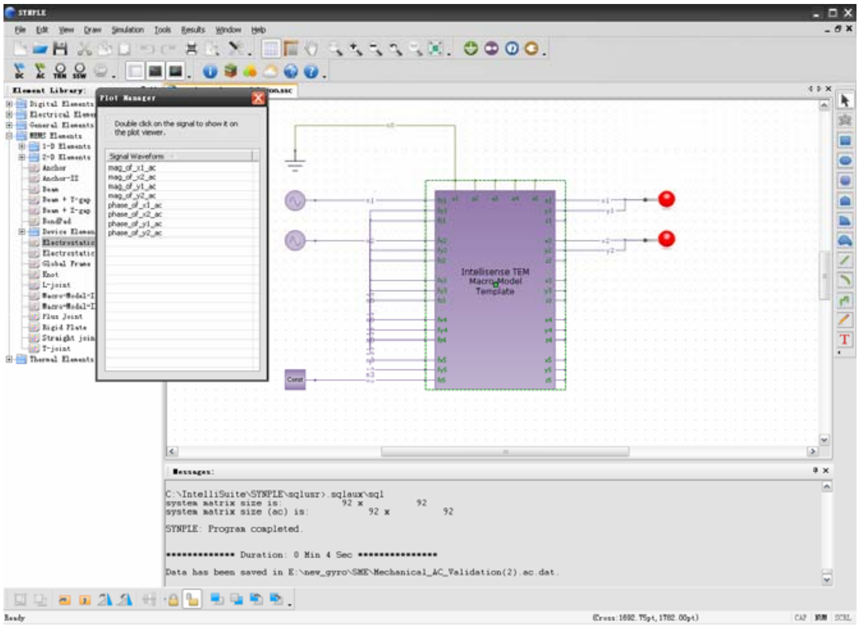
☐ Select/Unselect all si

Figure 89 AC Simulation Signals

You can check the convergence criteria, but they should be left as the default. Run the AC simulation.

Click *OK*

When the simulation is complete, the Plot Manager window will appear with the simulation results.



```
C:\IntelliSuite\SYNPLE\sqlusr>.sqlaux\sql
system matrix size is:          92 x          92
system matrix size (ac) is:          92 x          92

SYNPLE: Program completed.

************* Duration: 0 Min 4 Sec ***************

Data has been saved in E:\new_gyro\SME\Mechanical_AC_Validation(2).ac.dat.
```

Figure 90 Plot Manager

Open *mag_of_x2_ac* or *mag_of_x1_ac* to view the x-displacement results.

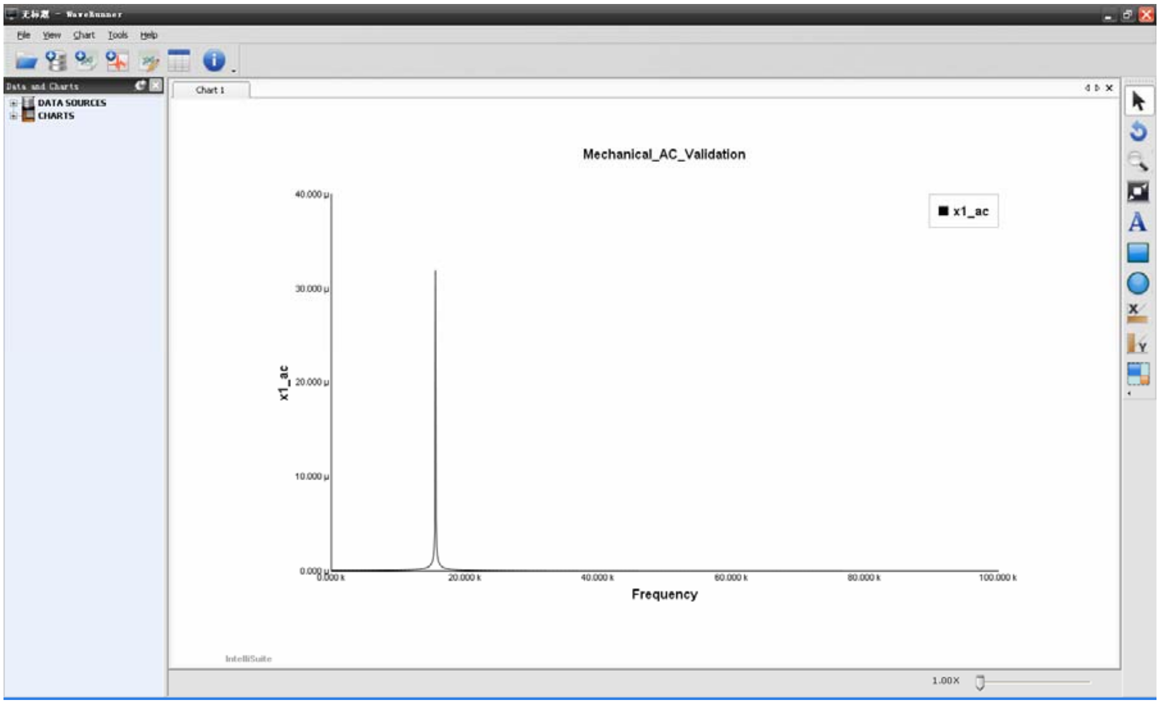


Figure 91 AC Results

You should notice that the natural frequency appears at 15.6 kHz. The AC simulation in SYNPLE produces the same results as the TEM simulation, thus the mechanical model is translating properly. The next step is to check the DC simulation results when we use a comb drive.

5.3. DC Validation

If you look back at the TEM simulation, the x-displacement of the moving masses under a static load of 100 V on the drive combs is ~ 0.22 microns. We want to see how accurately we can duplicate this result with the macromodel simulation in SYNPLE. Open the DC simulation schematic.

Click *File…Open*

Open the following file: `<Installation Directory>/IntelliSuite/Training/ /Application_Notes/Gyro/SYNPLE_Models/Mechanical-DC_Validation.ssc`.

The schematic is shown below.

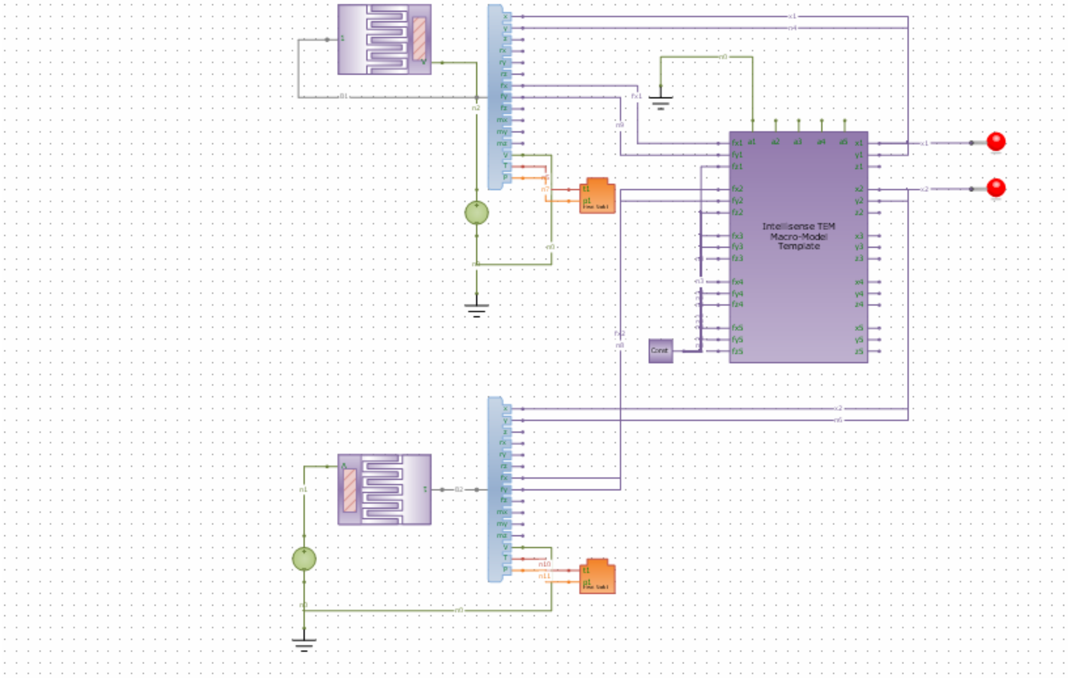


Figure 92 DC Schematic

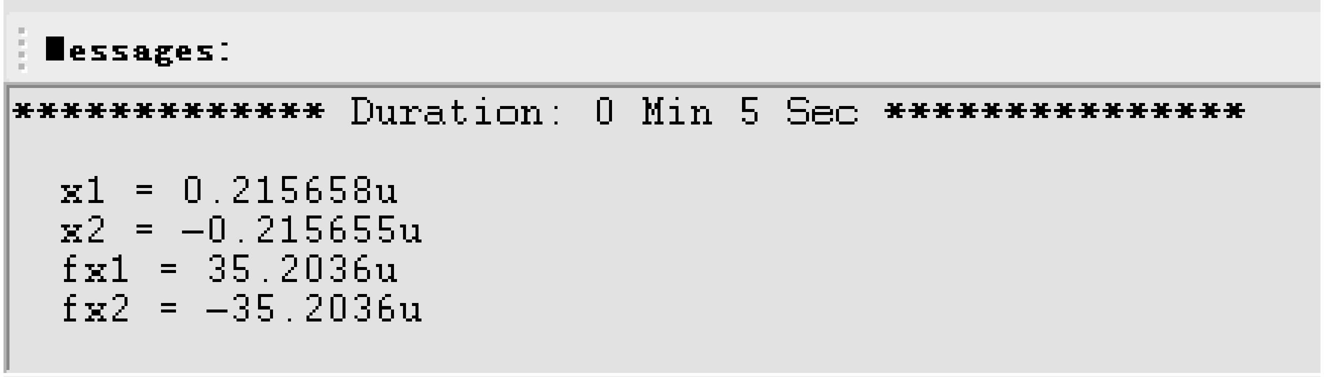
You will see that the *Comb Drive* and *Voltage DC Source* elements are connected to the macromodel using the *Splitter* element (found in the *Draw* menu). You can double-click on any of the elements to view the parameters and find more information in the Elements Documentation file under the *Help* menu.

The simulation is already set up, so

Click *Simulation…DC Analysis*

Click *OK*

When the simulation is complete, you will see the results in the Messages window. This window will display the results of the DC simulation for each signal that was selected. The important signals here are the forces applied by the comb drives, f1 and f2, and the displacements of the two moving masses, x1 and x2. The results are shown below.



```
************* Duration: 0 Min 5 Sec ***************

  x1 = 0.215658u
  x2 = -0.215655u
  fx1 = 35.2036u
  fx2 = -35.2036u
```

Figure 93 DC Simulation Results

When you compare the results from the SYNPLE analysis to the results from the TEM analysis, you see about a 5% difference. This is because there are fringe field effects that get taken into account in the TEM that are not taken into account in SYNPLE. The comb drives that we use have an overlap to thickness ratio of 0.4; this is very low, and thus 3D effects will dominate. The 2D comb drive generator cannot take this into account, and thus the results show some inaccuracy. However, this inaccuracy is not extremely large, and we can still generate a very good model for transient simulation that we can use to optimize our structure and controls.

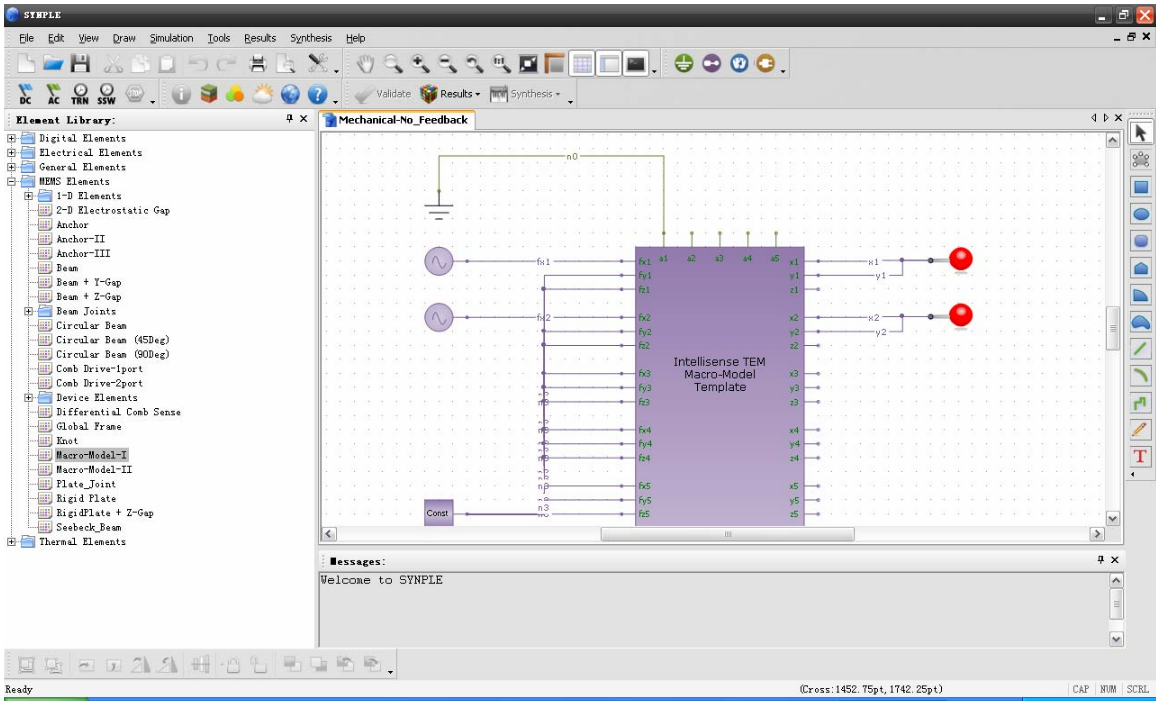
5.3.1 Benchmarks

Both the AC and DC results match up very well between the SYNPLE element-based model, TEM finite element model, and SYNPLE macromodel. Results for the natural frequency of the drive mode and the DC static displacement from each of the models are listed below.

	Element-based model	3D FEA (TEM)	Macromodel
Drive mode (Hz)	15.7 K	15.6 K	15.6 K
DC displacement	0.22 um	0.21 um	0.22 um

5.4. Mechanical Macromodel Transient Simulation

Open the following file: `<Installation Directory>/IntelliSuite/Training/ /Application_Notes/Gyro/SYNPLE_Models/Mechanical-No_Feedback.ssc`.



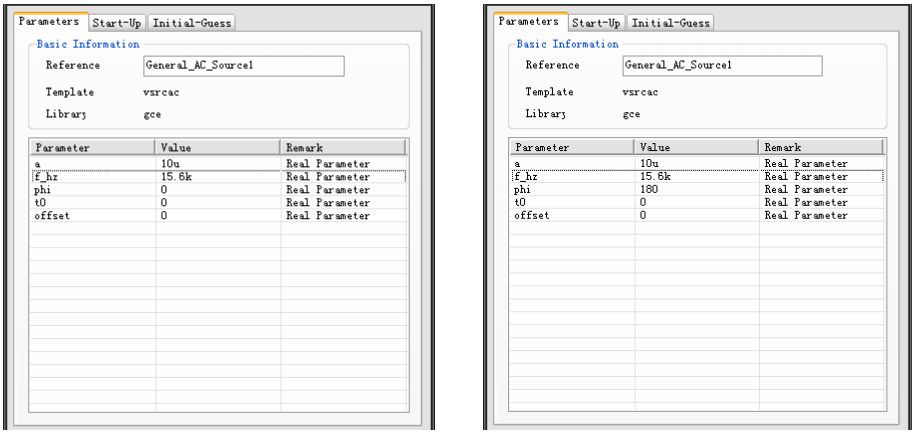
Welcome to SYNPLE

Ready

(Cross:1452.75pt,1742.25pt)

Figure 94 Mechanical Macromodel Schematic

If you double click on the any of the symbols, for instance the *General AC Source*, you can view the element parameters as shown below. The outputs of these two sources have the same amplitude and frequency but a 180 degree phase difference so that we actuate mode 4.



Reference General_AC_Source1
Template vsrcac
Library gce

Parameter	Value	Remark
a	10u	Real Parameter
f_hz	15.6k	Real Parameter
phi	0	Real Parameter
t0	0	Real Parameter
offset	0	Real Parameter

Reference General_AC_Source1
Template vsrcac
Library gce

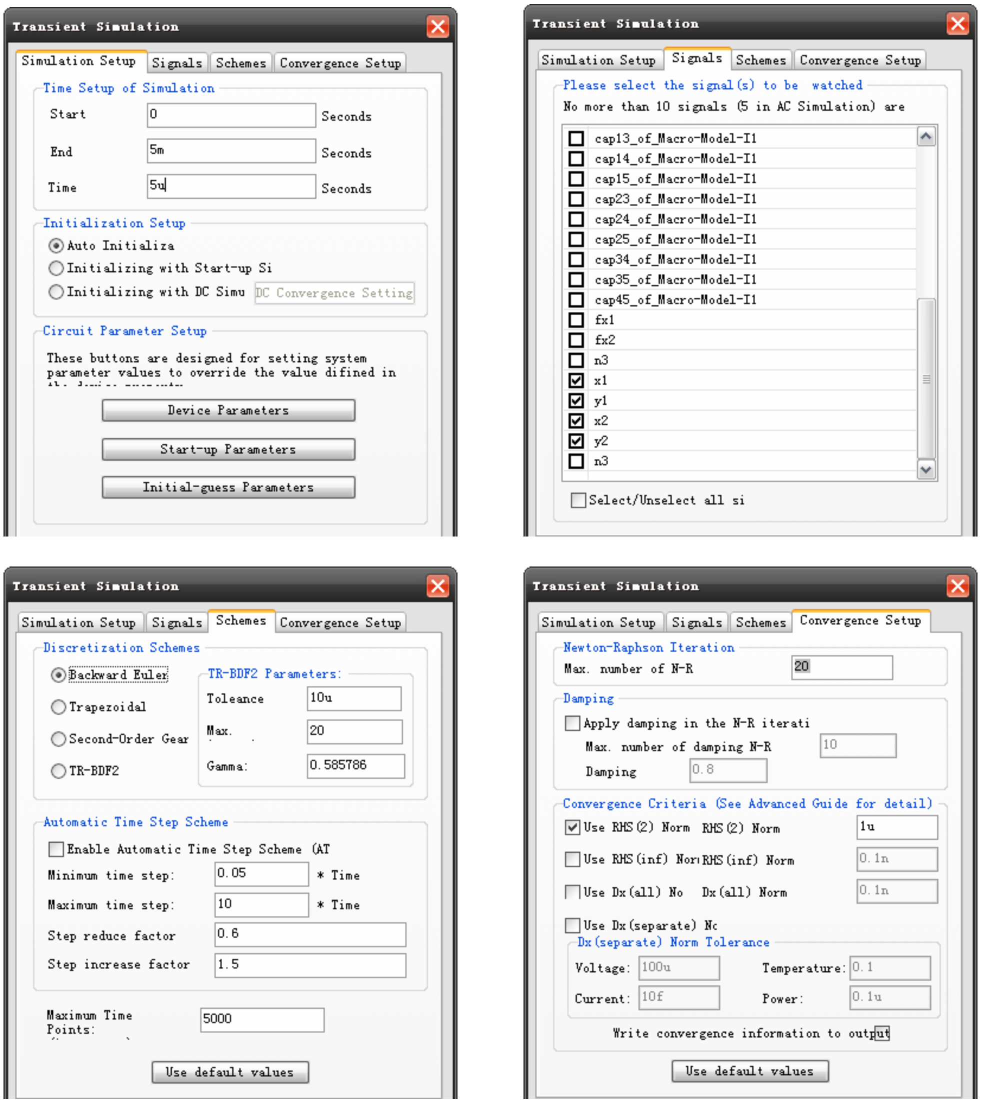
Parameter	Value	Remark
a	10u	Real Parameter
f_hz	15.6k	Real Parameter
phi	180	Real Parameter
t0	0	Real Parameter
offset	0	Real Parameter

Figure 95 General AC Source Parameters

Once you have finished checking out the elements, run the transient analysis.

Click *Simulation…Transient Analysis*

Make sure the simulation parameters are set up as shown below.



Time Setup of Simulation
Start: 0 Seconds
End: 5m Seconds
Time: 5u Seconds

Initialization Setup
● Auto Initializa
○ Initializing with Start-up Si
○ Initializing with DC Simu

Circuit Parameter Setup
These buttons are designed for setting system parameter values to override the value difined in

Please select the signal(s) to be watched
No more than 10 signals (5 in AC Simulation) are
☐ cap13_of_Macro-Model-I1
☐ cap14_of_Macro-Model-I1
☐ cap15_of_Macro-Model-I1
☐ cap23_of_Macro-Model-I1
☐ cap24_of_Macro-Model-I1
☐ cap25_of_Macro-Model-I1
☐ cap34_of_Macro-Model-I1
☐ cap35_of_Macro-Model-I1
☐ cap45_of_Macro-Model-I1
☐ fx1
☐ fx2
☐ n3
☑ x1
☑ y1
☑ x2
☑ y2
☐ n3
☐ Select/Unselect all si

Discretization Schemes
● Backward Euler
○ Trapezoidal
○ Second-Order Gear
○ TR-BDF2

TR-BDF2 Parameters:
Toleance: 10u
Max.: 20
Gamma: 0.585786

Automatic Time Step Scheme
☐ Enable Automatic Time Step Scheme (AT
Minimum time step: 0.05 * Time
Maximum time step: 10 * Time
Step reduce factor: 0.6
Step increase factor: 1.5
Maximum Time Points: 5000

Newton-Raphson Iteration
Max. number of N-R: 20

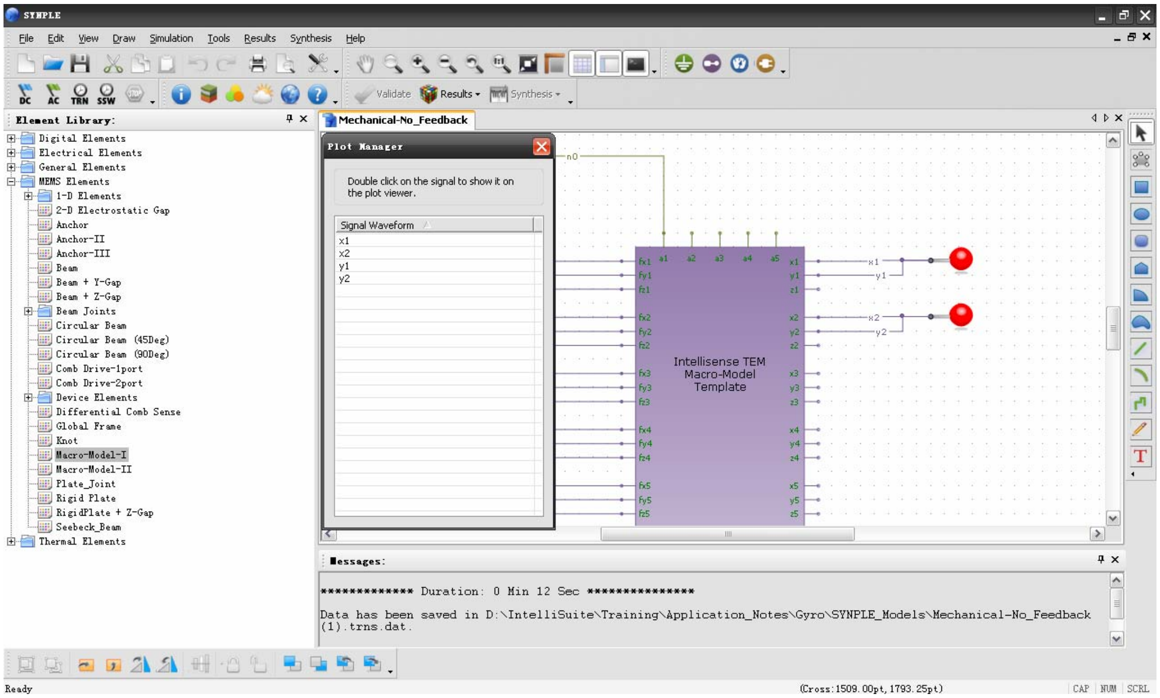
Damping
☐ Apply damping in the N-R iterati
Max. number of damping N-R: 10
Damping: 0.8

Convergence Criteria (See Advanced Guide for detail)
☑ Use RHS(2) Norm — RHS(2) Norm: 1u
☐ Use RHS(inf) Nor — RHS(inf) Norm: 0.1n
☐ Use Dx(all) No — Dx(all) Norm: 0.1n
☐ Use Dx(separate) Nc
Dx(separate) Norm Tolerance
Voltage: 100u
Temperature: 0.1
Current: 10f
Power: 0.1u
Write convergence information to output ☐

Figure 96 Transient Simulation Setup

Click *OK*

Once the analysis is complete, the Plot Manager will appear with the selected signals. The message window will also show you where the simulation result data has been saved. All SYNPLE results are saved in a format that can easily be opened by Microsoft Excel or any other database editor.



Double click on the signal to show it on the plot viewer.

```
************* Duration: 0 Min 12 Sec ***************

Data has been saved in D:\IntelliSuite\Training\Application_Notes\Gyro\SYNPLE_Models\Mechanical-No_Feedback
(1).trns.dat.
```

Figure 97 Completed Simulation with Plot Manager

Double click on any of the signals to view the plot of the results.

The plot of x1 vs. time will appear as shown below.

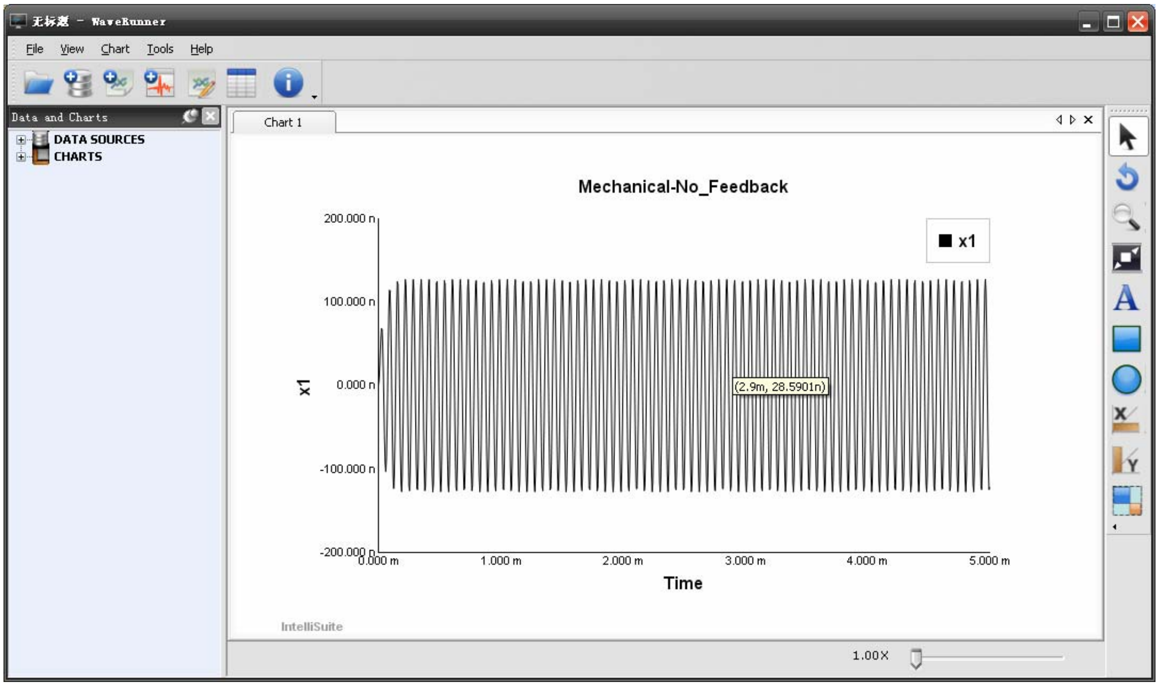


Figure 98 Plot of x1 vs. Time

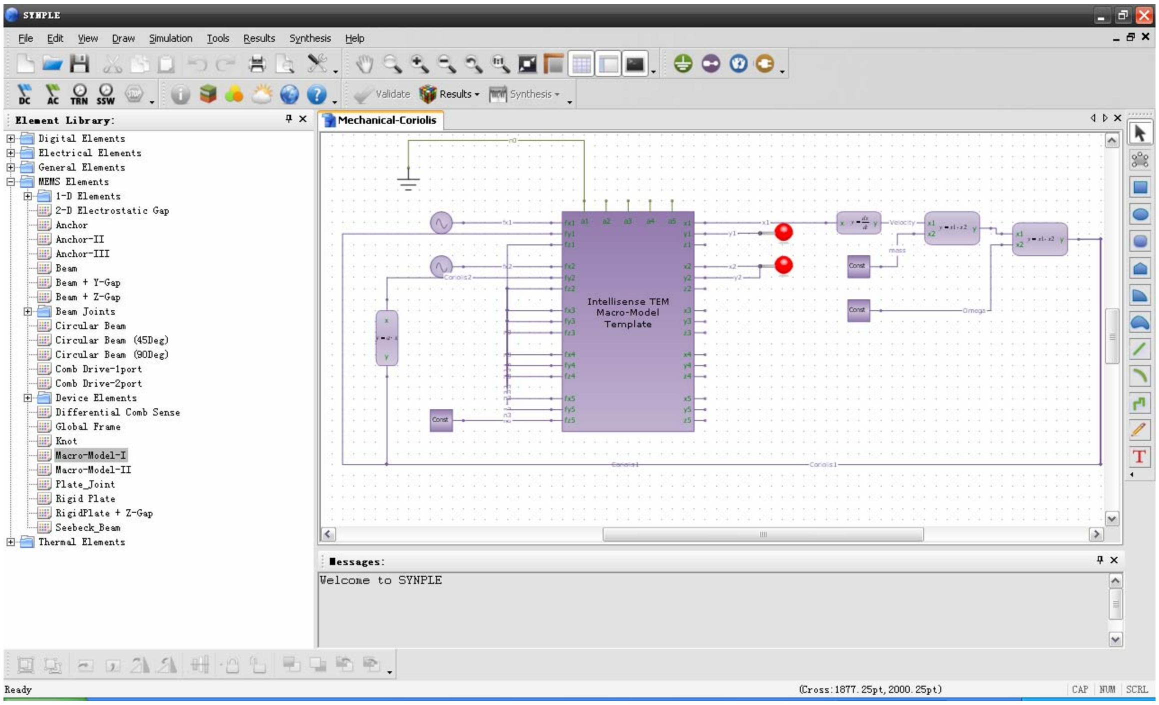
5.5. Incorporating the Coriolis Feedback

To simulate the Coriolis effect in the gyro design, we can apply a Coriolis force feedback to the representative node in the y-direction. The Coriolis force can be expressed by the following equation:

$$F_{Coriolis} = -m(\vec{\omega}\times\vec{v}) = -m(2\pi f\frac{dx}{dt})$$

Open the Coriolis feedback schematic at `<Installation Directory>/IntelliSuite/Training/Application_Notes/Gyro/SYNPLE_Models/Mechanical-Coriolis.ssc`.

The schematic will appear as shown below.



Welcome to SYNPLE

Figure 99 Coriolis Feedback Schematic

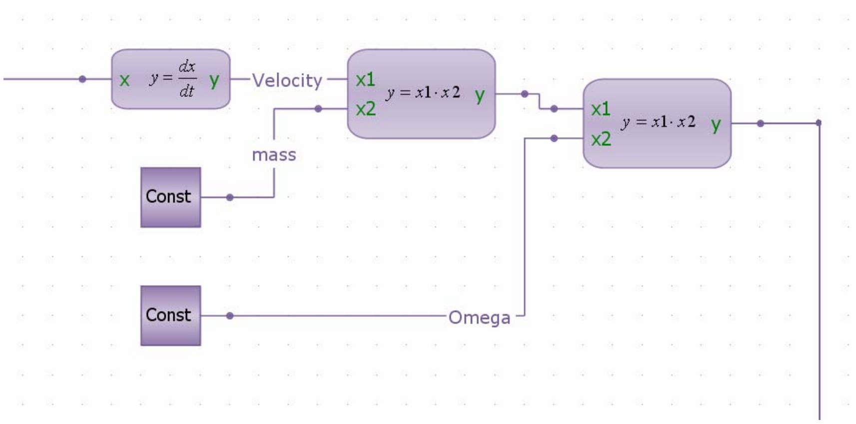
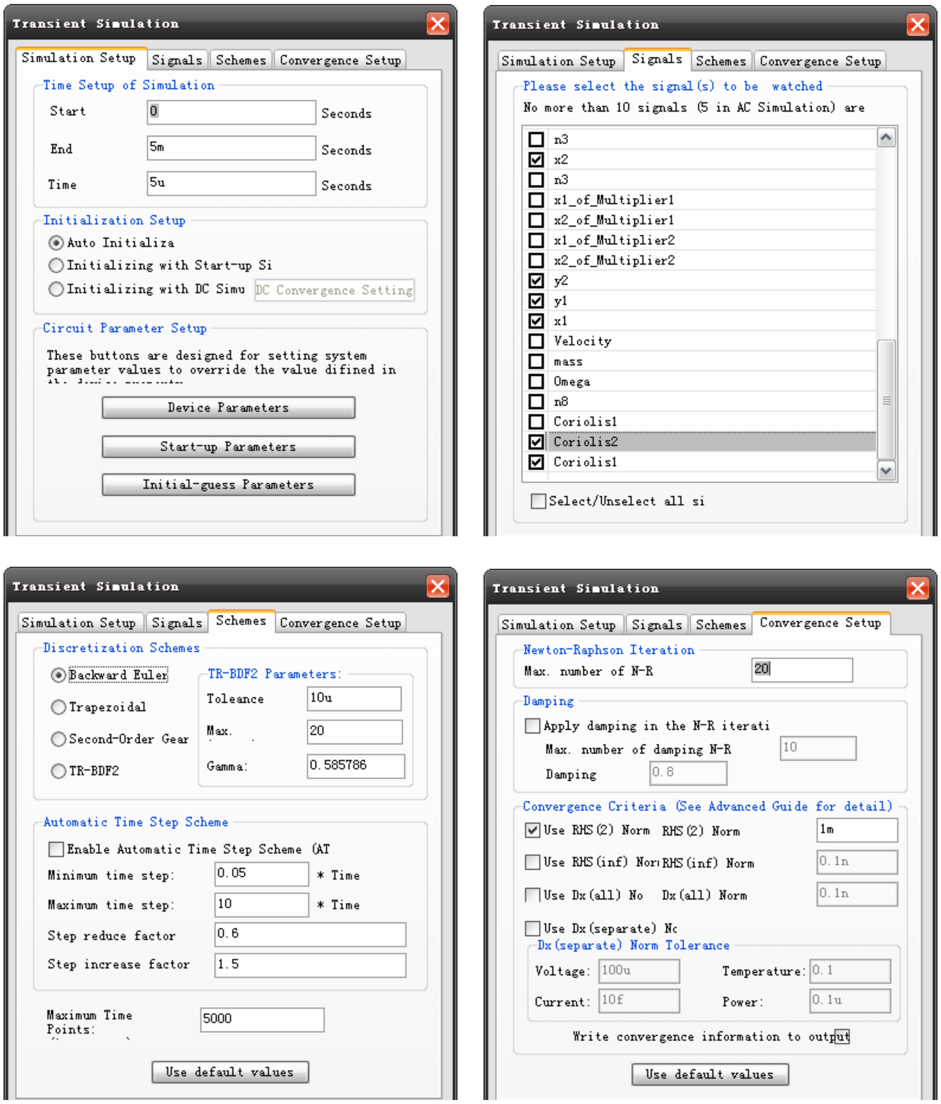


Figure 100 Coriolis Force Feedback Loop

You will see that *Derivative*, *Multiplication*, and *Multiplier* elements (in the Math sub-folder in the General Elements library) are used to apply the Coriolis force. When you are finished inspecting the elements, you can run the simulation.

Click *Simulation…Transient*

Set the simulation parameters as shown below: (Make sure that the signals *y1* and *y2* have been selected as well)



Time Setup of Simulation

Start 0 Seconds

End 5m Seconds

Time 5u Seconds

Initialization Setup

● Auto Initializa

○ Initializing with Start-up Si

○ Initializing with DC Simu DC Convergence Setting

Circuit Parameter Setup

These buttons are designed for setting system parameter values to override the value difined in

Please select the signal(s) to be watched

No more than 10 signals (5 in AC Simulation) are

☐ n3

☑ x2

☐ n3

☐ x1_of_Multiplier1

☐ x2_of_Multiplier1

☐ x1_of_Multiplier2

☐ x2_of_Multiplier2

☑ y2

☑ y1

☑ x1

☐ Velocity

☐ mass

☐ Omega

☐ n8

☐ Coriolis1

☑ Coriolis2

☑ Coriolis1

☐ Select/Unselect all si

Discretization Schemes

● Backward Euler

○ Trapezoidal

○ Second-Order Gear

○ TR-BDF2

TR-BDF2 Parameters:

Toleance 10u

Max. 20

Gamma: 0.585786

Automatic Time Step Scheme

☐ Enable Automatic Time Step Scheme (AT

Minimum time step: 0.05 * Time

Maximum time step: 10 * Time

Step reduce factor 0.6

Step increase factor 1.5

Maximum Time Points: 5000

Newton-Raphson Iteration

Max. number of N-R 20

Damping

☐ Apply damping in the N-R iterati

Max. number of damping N-R 10

Damping 0.8

Convergence Criteria (See Advanced Guide for detail)

☑ Use RHS(2) Norm RHS(2) Norm 1m

☐ Use RHS(inf) Nor RHS(inf) Norm 0.1n

☐ Use Dx(all) No Dx(all) Norm 0.1n

☐ Use Dx(separate) Nc

Dx(separate) Norm Tolerance

Voltage: 100u Temperature: 0.1

Current: 10f Power: 0.1u

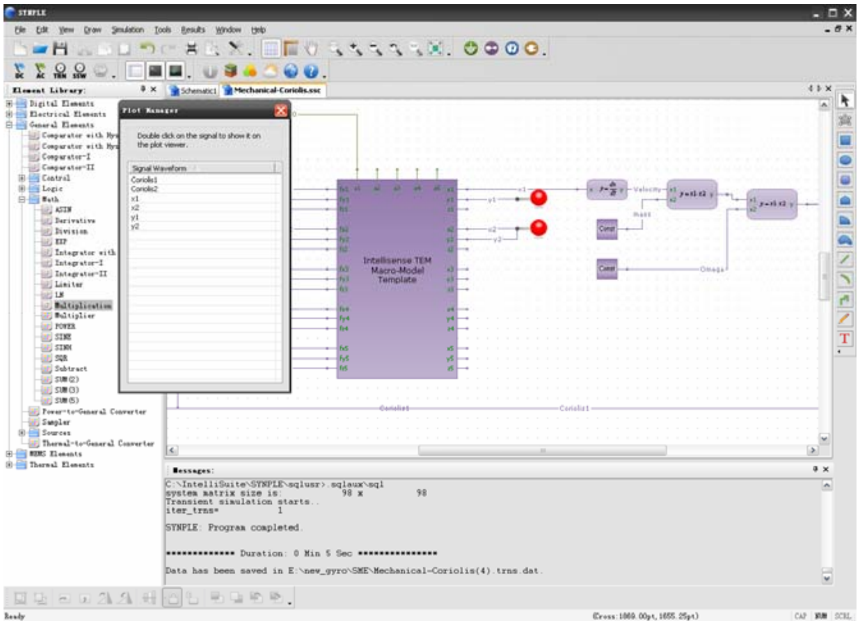
Write convergence information to output

Figure 101 Transient Coriolis Force Simulation Settings

To start the analysis,

Click *OK*

Once the analysis is complete, the signal manager with the selected signals will appear.



```
C:\IntelliSuite\SYNPLE\sqlusr>.sqlaux\sql
system matrix size is:          98 x          98
Transient simulation starts..
iter_trns=          1

SYNPLE: Program completed.


************* Duration: 0 Min 5 Sec ***************

Data has been saved in E:\new_gyro\SME\Mechanical-Coriolis(4).trns.dat.
```

Figure 102 Coriolis Simulation Results

The displacement and Coriolis force results are shown below.

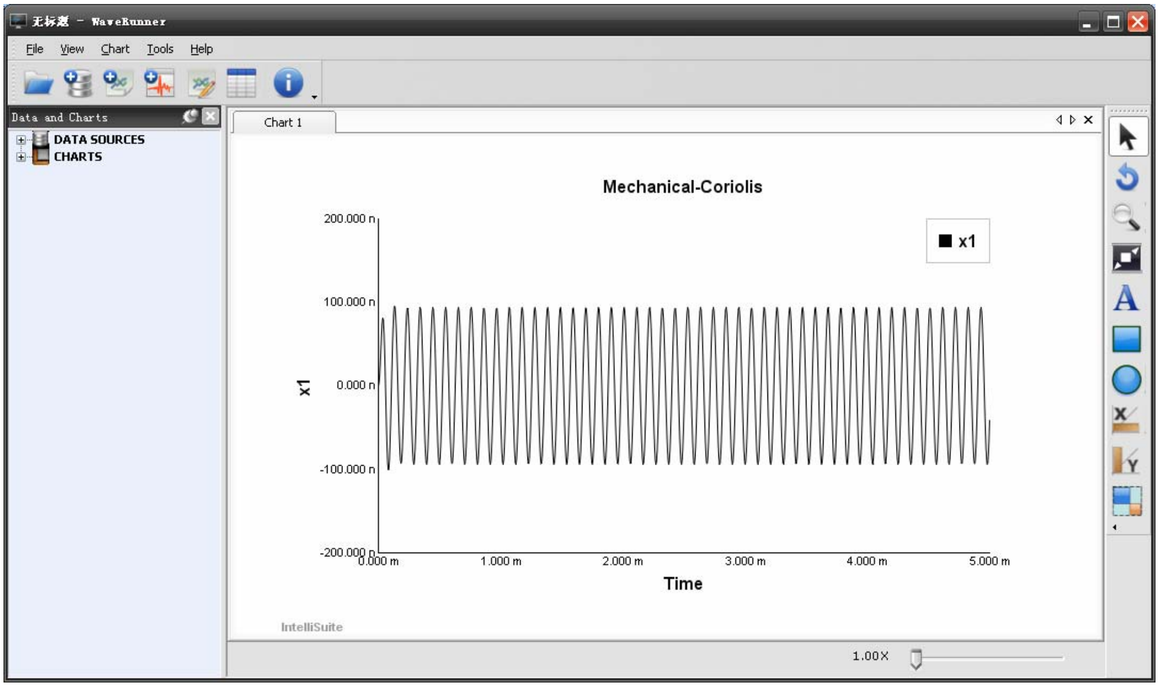


Figure 103 Transient Plot of x1 vs Time

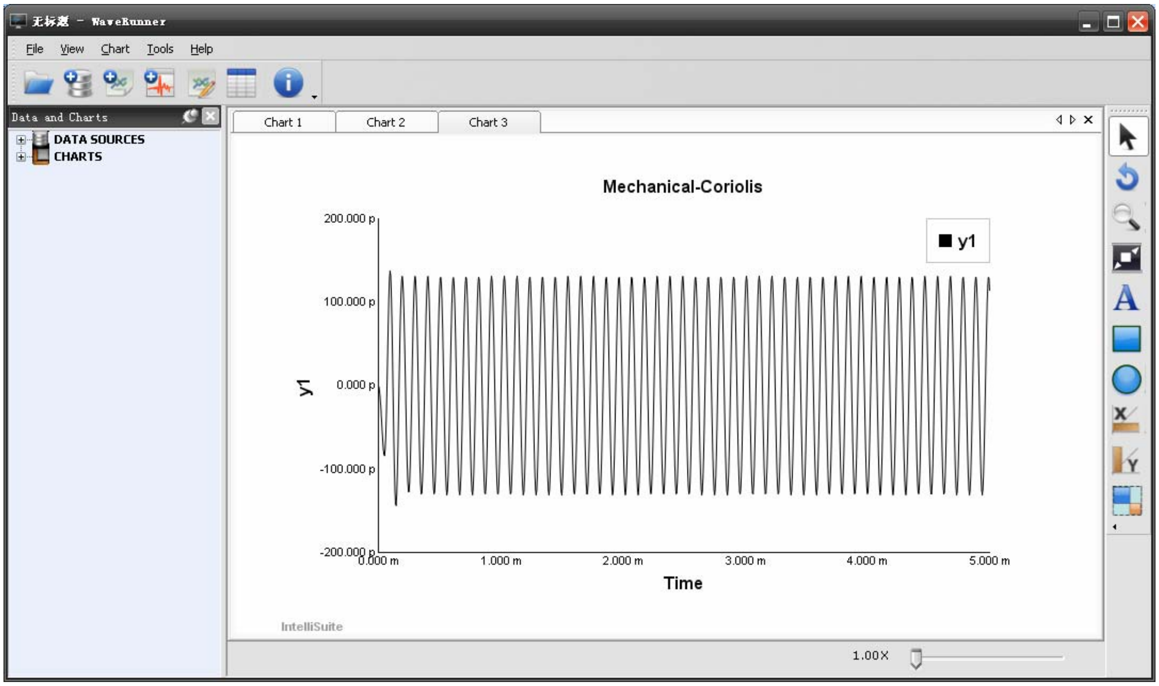


Figure 104 Transient Plot of y1 vs Time

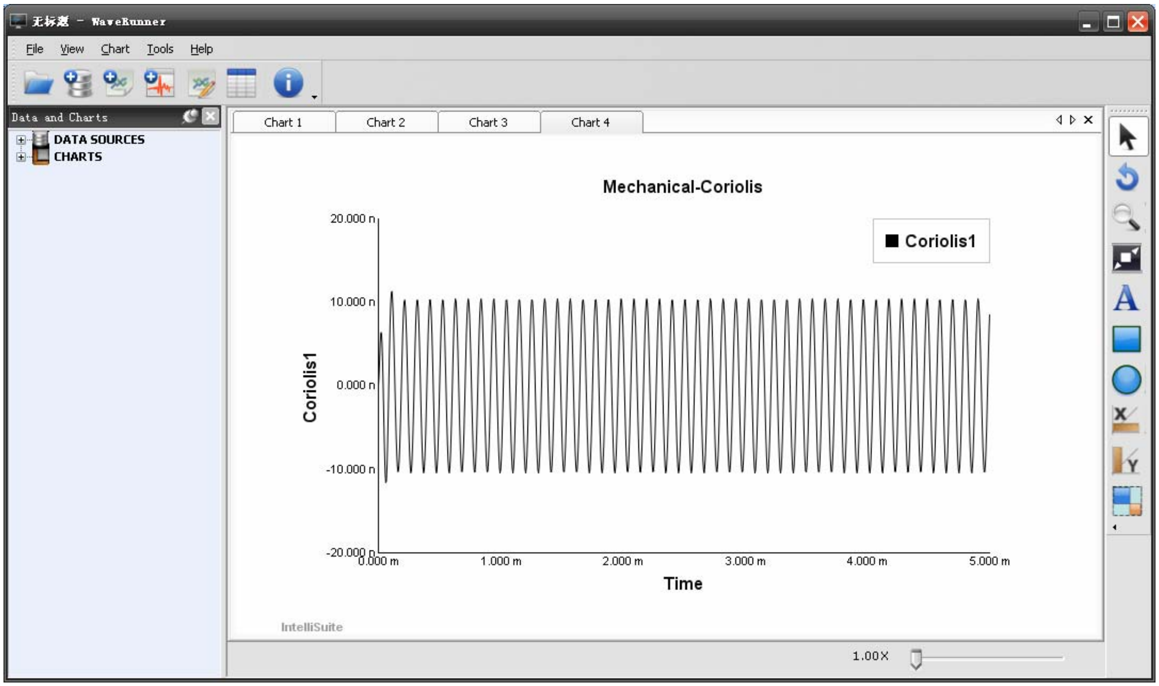


Figure 105 Transient Plot of Coriolis_1 vs Time

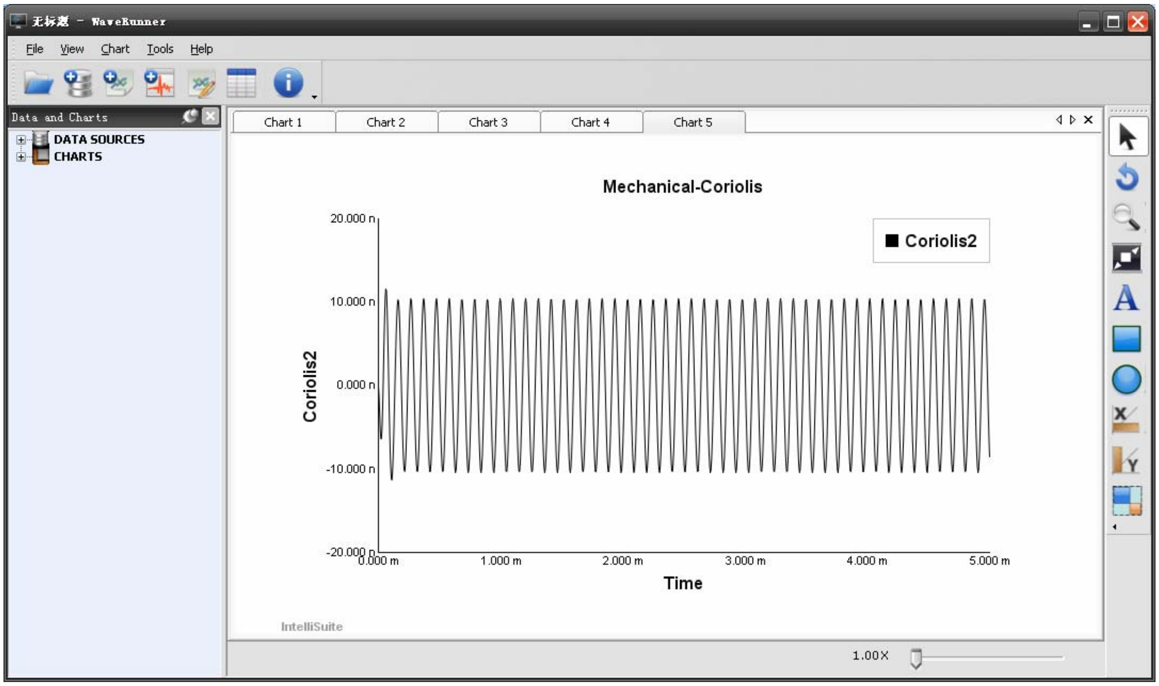


Figure 106 Transient Plot of Coriolis_2 vs Time

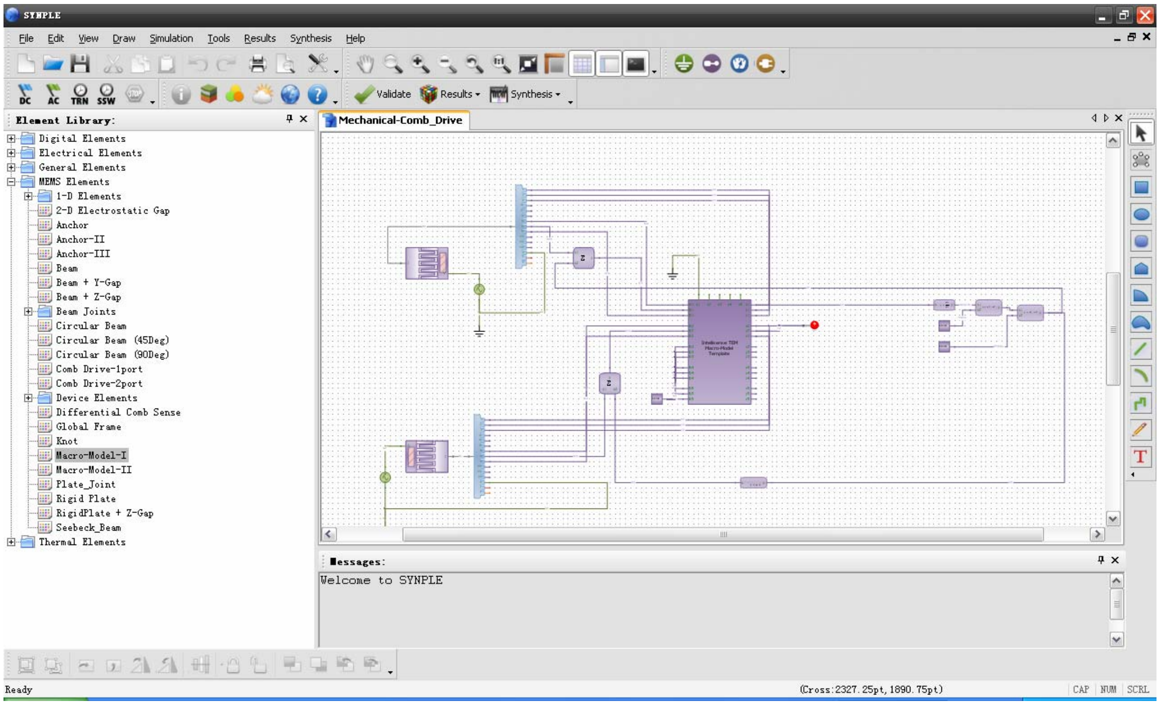
You will notice that the force and the response are in the opposite direction for points 1 and 2. This is because the two masses are always moving in opposite directions.

5.6. Incorporation of the Drive Combs

The proof masses of the gyro are driven by two comb drives. We can add *Comb Drive* elements to the schematic instead of simply applying a voltage on a node. This will allow us to run parametric analysis on the comb drive parameters to determine the optimal comb drive. We can consider questions like: How many fingers do we want to use? What gap do we want to use? How long should the fingers be?

A schematic which includes comb drives is located here:

`<Installation Directory>/IntelliSuite/Training/Application_Notes/Gyro/ /SYNPLE_Models/Mechanical-Comb_Drive.ssc`



```
Welcome to SYNPLE
```

Figure 107 Schematic with Drive Combs

This file is ready for simulation.

Click *Simulation…Transient*
Click *OK*

The simulation results are shown below.

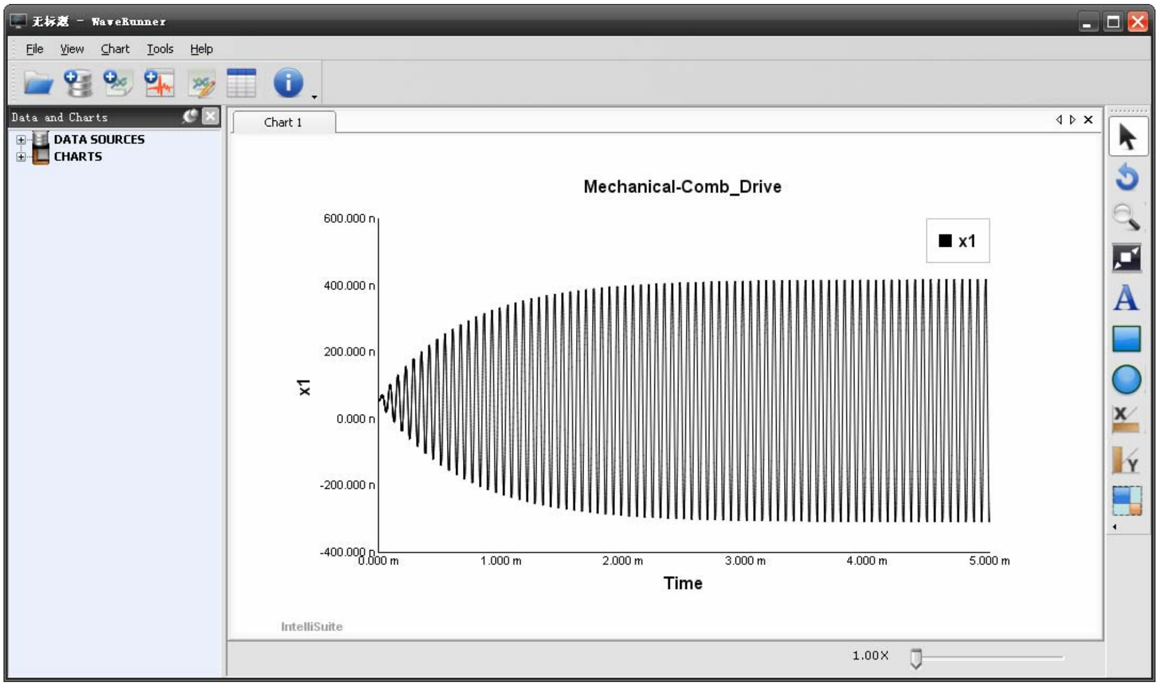


Figure 108 Transient Result of x1

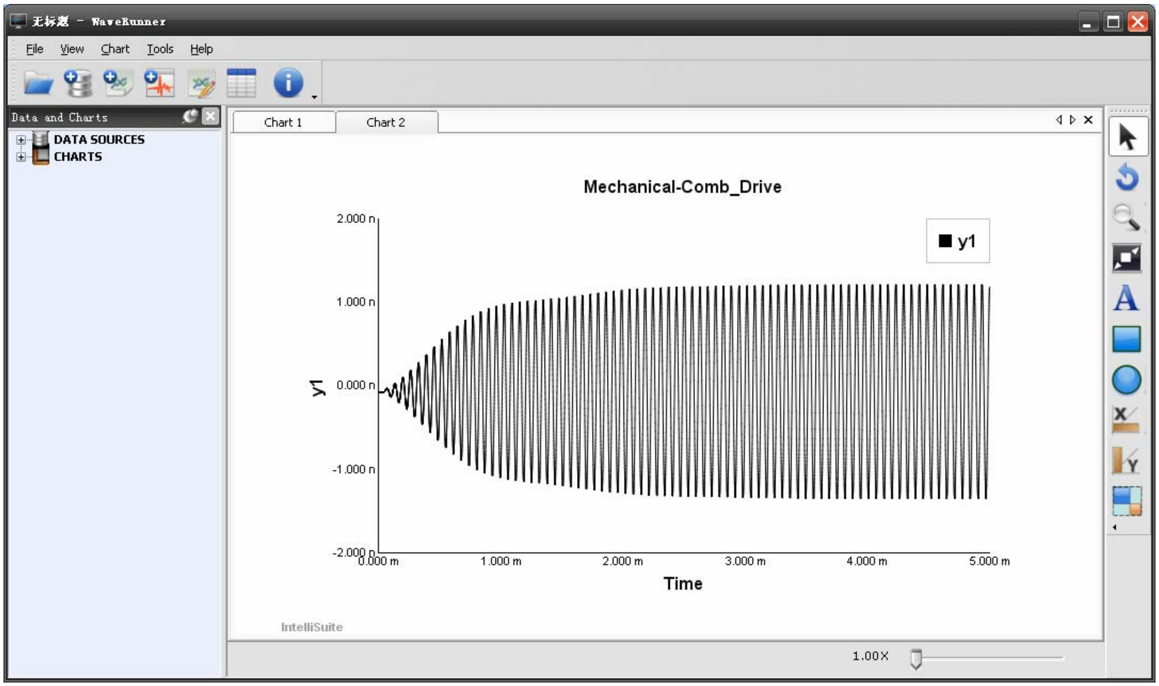


Figure 109 Transient Result of y1

5.7. Including the Sense Electrodes

The displacement of proof masses will be sensed by two electrostatic gaps. The *Electrostatic Gap* elements take the displacements in the x-y plane as input variables. There are many parameters of the *Electrostatic Gap* element which are described in more detail in the “Getting Started with SYNPLE” or “Elements Documentation” manuals.

Open the file:

```
<Installation Directory>/IntelliSuite/Training/Application_Notes/Gyro/
/SYNPLE_Models/Mechanical-Constant.ssc
```

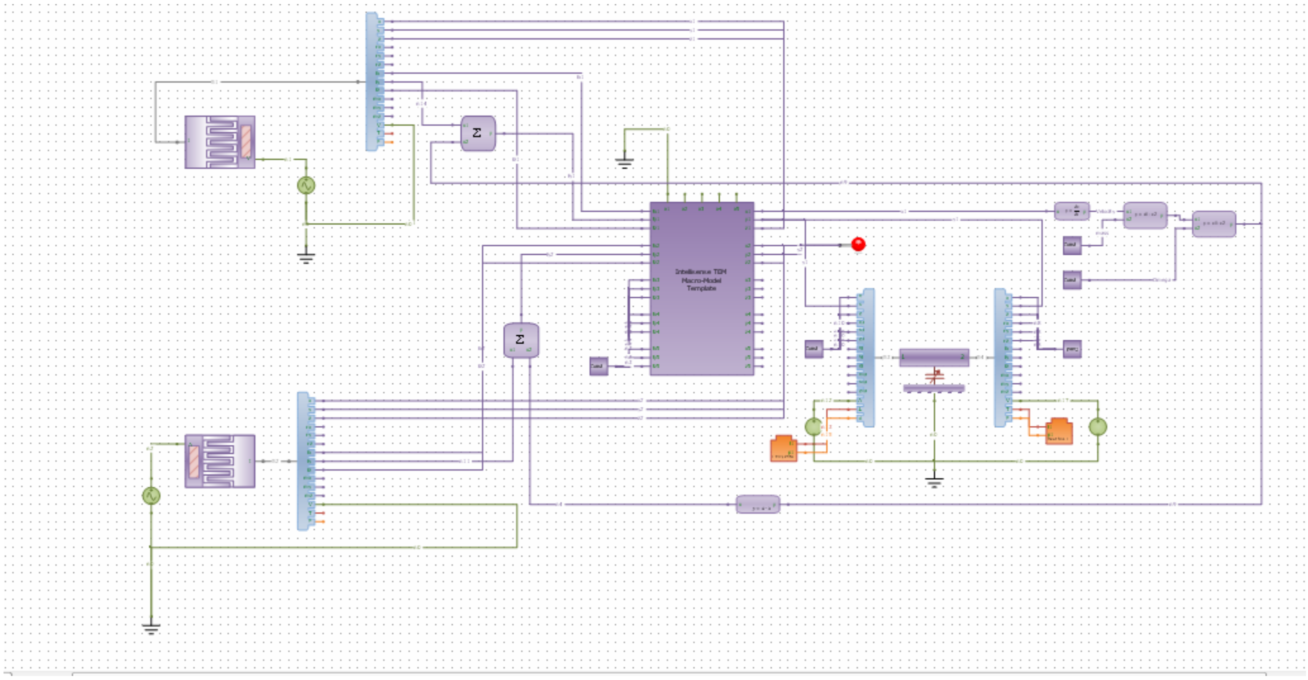


Figure 110 Full Schematic with Sense Gaps

This file is ready for simulation.

Click *Simulation…Transient*
Click *OK*

We have schematics set up for two loading conditions: one for a constant rotation and one for a sinusoidal rotation. Feel free to examine them. You can examine the different wave sources by double clicking on them in the schematic (they have been applied as the rotation in the Coriolis feedback loop) and their value has been selected as the output signal “omega”. They are located here:

```
<Installation Directory>\IntelliSuite\Training\Application_Notes\Gyro\
SYNPLE_Models\Mechanical-Constant.ssc
```

```
<Installation Directory>\IntelliSuite\Training\Application_Notes\Gyro\
SYNPLE_Models\Mechanical-Sine.ssc
```

The results are shown below.

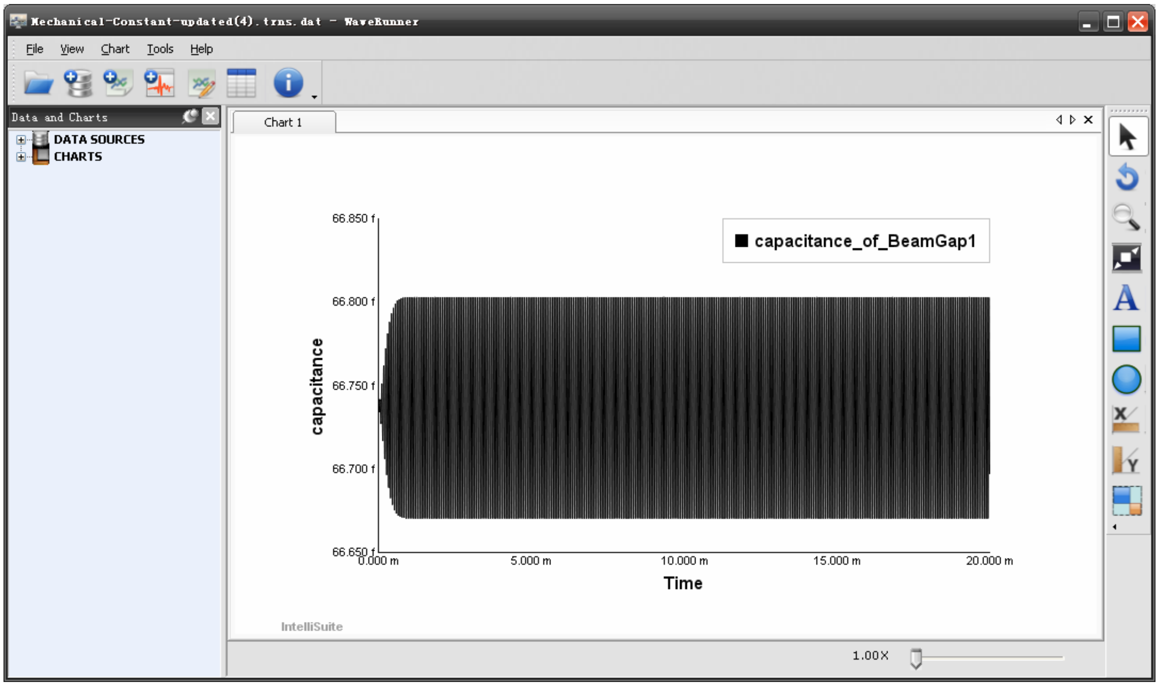


Figure 111 Capacitance Response to a Constant Rotation input

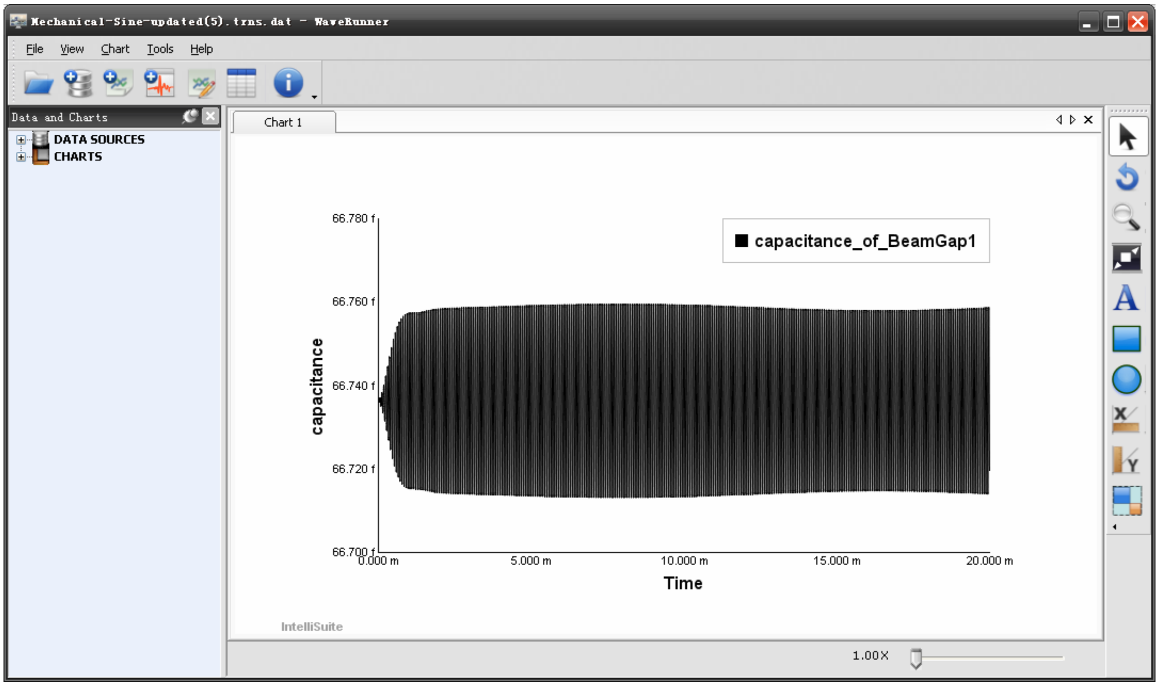
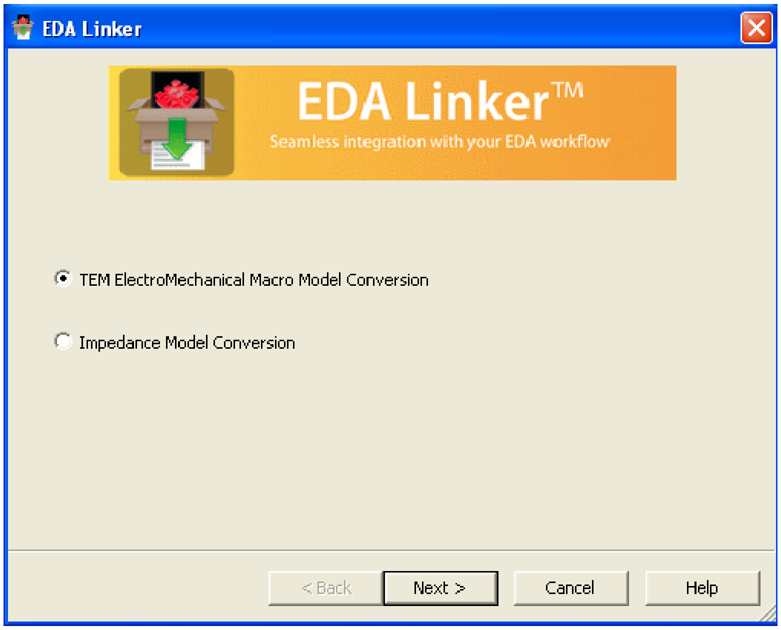


Figure 112 Capacitance Response to a Sine Wave Rotation input

6. EDA Linker

In this section we will use EDA Linker to convert our macromodel to other HDLs (hardware description languages). EDA Linker provides a convenient link between IntelliSuite's TEM module and system level simulators like VHDL-AMS, SPICE, and Simulink.

Click *Start…Programs…IntelliSuite…Utilities… EDA Linker*

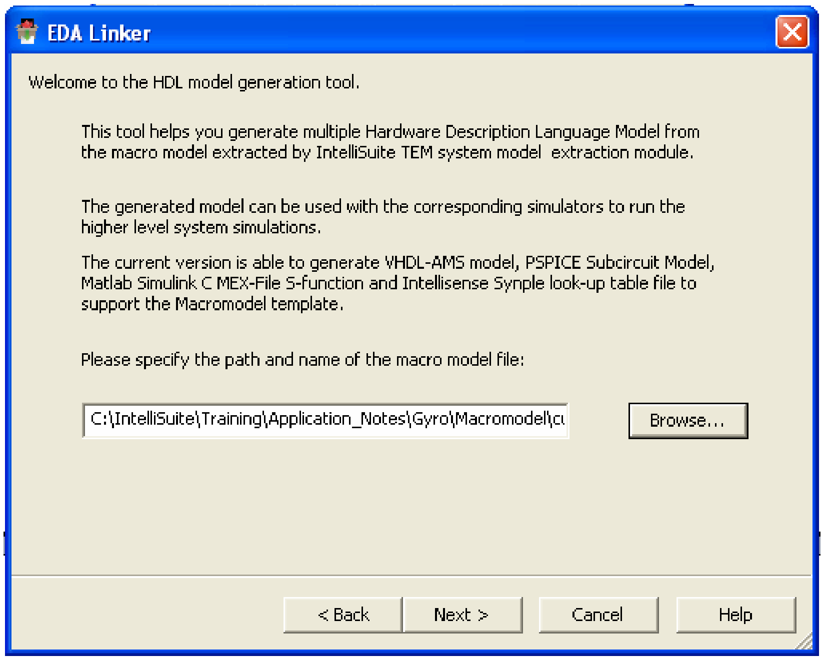


● TEM ElectroMechanical Macro Model Conversion

○ Impedance Model Conversion

Figure 113 EDA Linker Window

Click *Next*



Welcome to the HDL model generation tool.

This tool helps you generate multiple Hardware Description Language Model from the macro model extracted by IntelliSuite TEM system model extraction module.

The generated model can be used with the corresponding simulators to run the higher level system simulations.

The current version is able to generate VHDL-AMS model, PSPICE Subcircuit Model, Matlab Simulink C MEX-File S-function and Intellisense Synple look-up table file to support the Macromodel template.

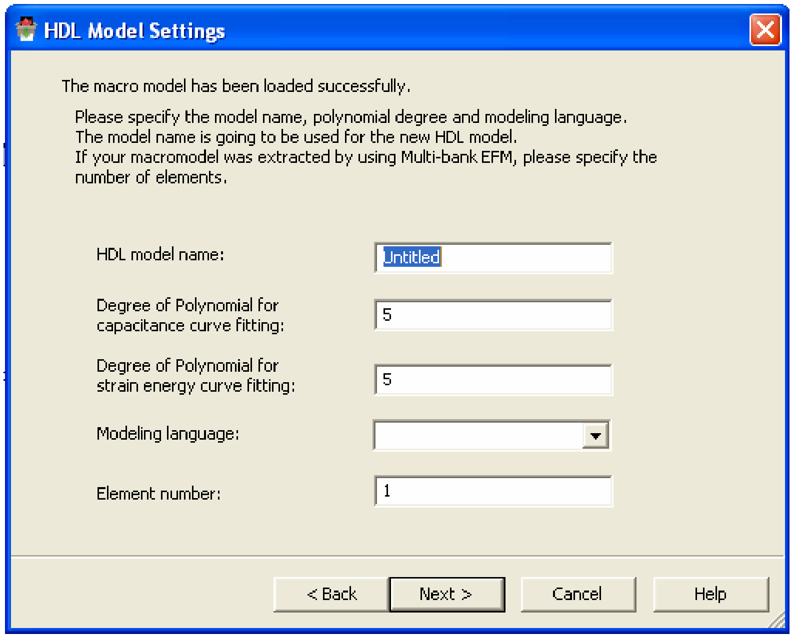
Please specify the path and name of the macro model file:

C:\IntelliSuite\Training\Application_Notes\Gyro\Macromodel\cu

Figure 114 Macromodel File Selection

Select the desired macromodel file, then

Click *Next*



The macro model has been loaded successfully.

Please specify the model name, polynomial degree and modeling language.
The model name is going to be used for the new HDL model.
If your macromodel was extracted by using Multi-bank EFM, please specify the number of elements.

HDL model name: Untitled

Degree of Polynomial for capacitance curve fitting: 5

Degree of Polynomial for strain energy curve fitting: 5

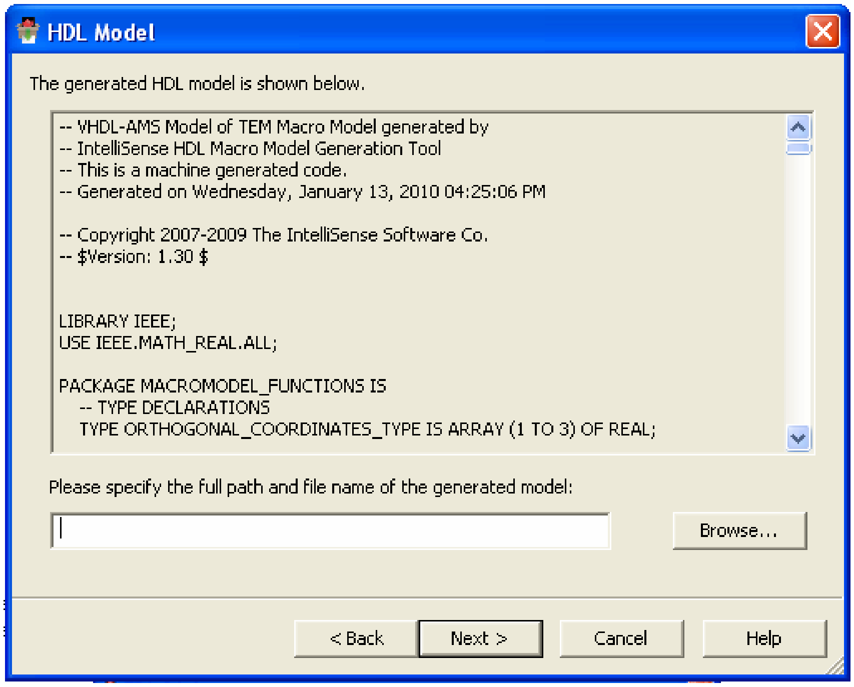
Modeling language:

Element number: 1

Figure 115 HDL Model Generation

Enter a name for the HDL model and specify the degree of polynomial for curve fitting. Note that in most cases, four- or five-degree polynomials are sufficient; ten-degree polynomials are the maximum. If you find that the resulting model is not sufficiently accurate, you may use a higher order polynomial. The last step is to select a modeling language. In the current version of EDA Linker, the following formats are supported: VHDL-AMS, PSPICE/SIMetrix, HSPICE, Simulink MEX S-Function. More languages will be available in future versions. After filling all the blanks,

Click *Next*



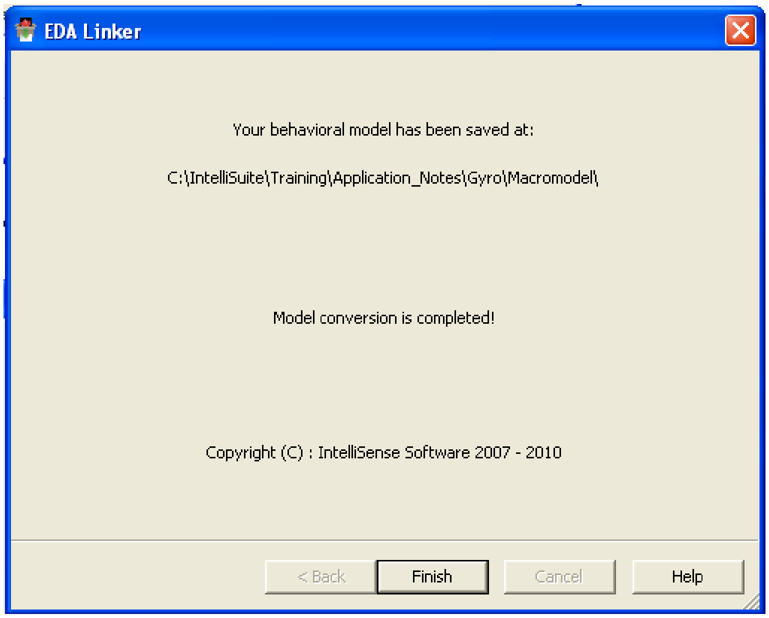
The generated HDL model is shown below.

```
-- VHDL-AMS Model of TEM Macro Model generated by
-- IntelliSense HDL Macro Model Generation Tool
-- This is a machine generated code.
-- Generated on Wednesday, January 13, 2010 04:25:06 PM

-- Copyright 2007-2009 The IntelliSense Software Co.
-- $Version: 1.30 $


LIBRARY IEEE;
USE IEEE.MATH_REAL.ALL;

PACKAGE MACROMODEL_FUNCTIONS IS
   -- TYPE DECLARATIONS
   TYPE ORTHOGONAL_COORDINATES_TYPE IS ARRAY (1 TO 3) OF REAL;
```

Please specify the full path and file name of the generated model:

Figure 116 Save HDL Model

The source code of the generated HDL model is displayed but cannot be edited in this tool. Specify a path and a filename for the HDL file and click **Next** to save it.



Your behavioral model has been saved at:

C:\IntelliSuite\Training\Application_Notes\Gyro\Macromodel\

Model conversion is completed!

Copyright (C) : IntelliSense Software 2007 - 2010

Figure 117 Model Conversion

Click **Finish** to quit EDA Linker. You can use this procedure to create HSPICE, PSPICE and Simulink/Matlab models in addition to VHDL models. We have already created these HDL models for the gyro; you can find them here: `<Installation Directory>\IntelliSuite\Training\ \Application_Notes\Gyro\HDL_model\.`

7. Summary

IntelliSense provides tools that integrate through all levels and stages of the MEMS design cycle. Users can develop their device from the bottom up using a SYNPLE element model and mask synthesis or from the top down using System Model Extraction and circuitry integration.

SYNPLE is a great tool for initial design exploration. Parametric analysis and optimization features allow a designer to explore a large design space very quickly. Once a schematic has been constructed, it is easy to automatically extract a mask layout or meshed model to use for further analysis.

At the device level we provide a fully capable finite element solver, the ThermoElectroMechanical Analysis module (TEM). The TEM uses Exposed Face Meshing methods to de-couple the mechanical and electrostatic meshes of devices. This allows our users to perform mechanical mesh refinement in areas with high stress gradients and electrostatic mesh refinement in areas of high charge density. This greatly simplifies the mesh needed for the device without sacrificing any accuracy. This alone can save an order of magnitude on time.

IntelliSuite is the perfect tool for gyro design. Gyros are, by nature, coupled systems. The TEM is optimized for mixed domain analysis, and models can be exported to system level solvers for quick transient analysis as well as CMOS integration. System Model Extraction and SYNPLE modeling saves an incredibly large amount of time when compared to our own finite element solver, while our own FE solver saves time over our competitor's. When you compare the time required to run a full dynamic finite element analysis to the time required to run a system model extraction and dynamic analyses in SYNPLE, SME with SYNPLE is orders of magnitude faster. Running optimization analyses with SME and SYNPLE can take a few hours, while running multiple optimization analyses with a FE solver would take days, weeks, or possibly months depending on the extent of the optimization.

Because gyros are complex structures with many degrees of freedom, it is absolutely necessary to utilize the N-Degree of Freedom (N-DOF) System Models that the TEM produces. 6-DOF models will not produce results anywhere near the accuracy of the N-DOF models for gyro devices. These N-DOF models give users a much better idea of how their device is going to react to its control circuitry.

For more information on the unique advantages of IntelliSuite when it comes to gyro design, feel free to contact us at info@intellisense.com.

IntelliSense

Total MEMS Solutions

www.intellisense.com