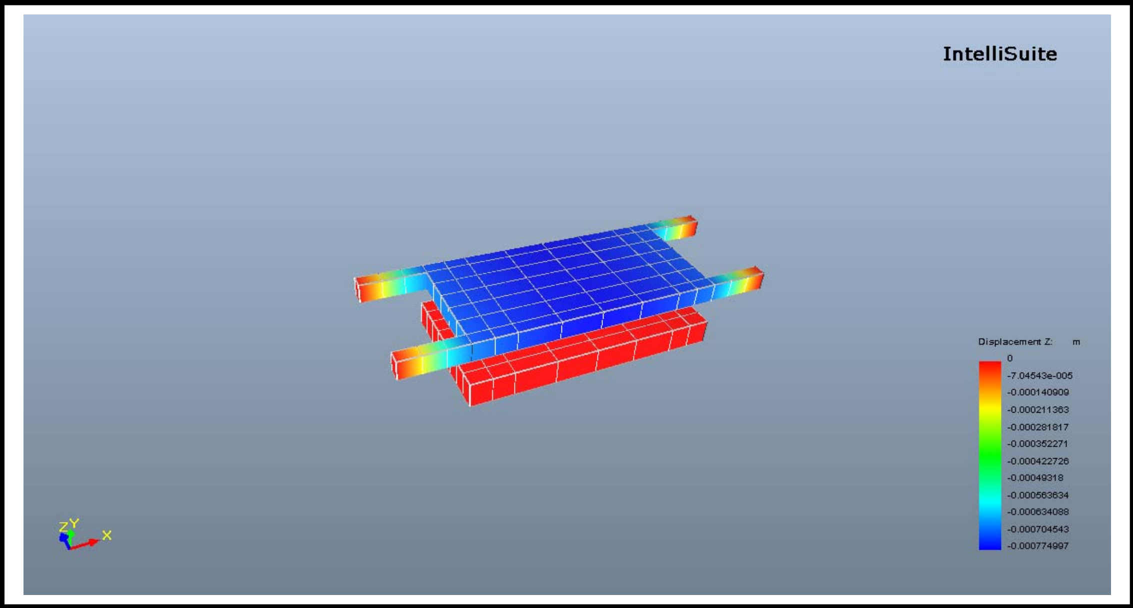
Application Note

Fluid Damping Macromodel Extraction and Validation



Application Note: Fluid Damping Macromodel Extraction and Validation
Version 8.6/PC

Part Number 30-090-101
January 2010

Table of Contents

1. Introduction

Reduced order mechanical and electromechanical macromodels can be extracted from coupled finite element mechanical/boundary element electrostatic (FE/BE) models in IntelliSuite by utilizing the principle of modal superposition [1] [2]. Different from rigid body approximations, modal superposition based reduced order models can reflect the characteristics of MEMS system components.

A MEMS system is initially represented as an FE/BE model with many free nodes. Rather than allowing each node in the model to move freely in any direction, we constrain the motion of the system to a linear superposition of a selected set of modal shape functions, thereby greatly reducing the degrees of freedom of the system. In general, the deformed state and dynamics of a mechanical system can be accurately described as a linear combination of modal shape functions. An equation describing this transformation is given below.

$$\Psi_{ext}(x, y, z, t) = \Psi_{initial}(x, y, z) + \sum q_i(t) \cdot \psi_i(x, y, z) \qquad (1)$$

where Ψ_{ext} represents the deformed state of the structure, $\Psi_{initial}$ the initial equilibrium state (derived from the residual stress without external loads), $\psi_i(x, y, z)$ the displacement vector of the i-th mode, and q_i the coefficient of the i-th mode, which is referred to as the scaling factor for the i-th mode.

Eqn. (1) states that Ψ_{ext} is a linear superposition of a selected set of modal shape functions $\psi_i(x, y, z)$ relative to an equilibrium state $\Psi_{initial}(x, y, z)$.

The modal superposition method can be used to help evaluate system energy terms in an efficient manner. The system energy terms are necessary for generating motion equations for the system using Lagrangian mechanics. An equation of motion can be generated for each modal shape, with each modal shape representing a degree of freedom. The equations of motion can be derived from Lagrangian equations.

$$m_i\, \partial^2 q_i/\partial t^2 + 2\xi_i\omega_i m_i \partial q_i/\partial t + \partial U_m(q)/\partial q_i - \partial U_e(q)/\partial q_i - \sum\psi_i F = 0 \qquad (2)$$

where m_i is the i-th mode generalized mass, q_i the coefficient of the i-th mode, ξ_i the linear modal damping ratio, ω_i the i-th eigenfrequency, ψ_i the i-th modal shape function, $\sum\psi_i F$ the sum over all the modes of the external mode forces weighted by the mode shape, U_m the strain energy, and U_e the electrostatic energy.

The squeezed film damping force can be modeled as an implicit function of the motion.

$$F_{SQFD} = F(\Psi_{ext}) = F(\Psi_{initial}, q_1, q_2, \ldots, \psi_1, \psi_2, \ldots, \partial q_1/\partial t, \partial q_2/\partial t, \ldots) \qquad (3)$$

where F_{SQFD} is the squeezed film damping force.

If the modal cross-talk is negligible, the total force can be simplified to the sum of separate modal forces by ignoring the modal cross-talk.

$$F_{SQFD} = \sum F_i(q_i, \psi_i, \partial q_i/\partial t, \Psi_{initial}) = \sum G_i(q_i, \partial q_i/\partial t, \Psi_{initial})\, \psi_i \qquad (4)$$

Typically, a modal squeezed damping force consists of two terms. One is the viscous effect in fluid flow, referred to as the viscous force. The other reflects the compressibility of the flow medium, referred to as the spring force. It is assumed that the viscous force is proportional to the modal velocity, and the spring force is proportional to the modal displacement.

$$\langle F_i, \psi_i \rangle = C_i\, \partial q_i/\partial t + K_i\, q_i \qquad (5)$$

where $<F_i, \psi_i>$ is the dot product of vectors F_i and ψ_i, and C_i, K_i are the damping coefficient and stiffness coefficient of the i-th mode, respectively.

Typically, C_i and K_i are both nonlinear and frequency dependent.

$$C_i = C_i(q_i, \omega), \quad K_i = K_i(q_i, \omega) \tag{6}$$

But Eqn. (6) can be linearized if the motion is limited in a small neighborhood of a certain value of q_i. In a linearized model, C_i and K_i take on values near the initial state $\Psi_{initial,}$ and they are frequency dependent only.

$$C_i = C_i(q_i=0, \omega) = C_i(\omega), \qquad K_i = K_i(q_i=0, \omega) = K_i(\omega) \tag{7}$$

As can be seen, the damping and stiffness coefficients are still frequency dependent, so they cannot be directly used in macromodel simulation for transient analysis. Methods can be used to transform frequency dependent parameters to frequency independent ones. The transformation method used in this example is referred to [3].

The modal superposition method is an efficient way for computation because only one equation is needed for one mode to describe a coupled system entirely.

An overview of the modal superposition method implementation in fluid damping macromodel extraction is described below. Many aspects of the method are performed automatically and simultaneously without requiring user interaction.

1) Create an FE/BE model

2) Simulate modal contributions

   Perform a frequency analysis to find the natural frequencies and modal shapes of the system. Then perform a non-linear static FEA simulation, or a full non-linear coupled static FEA/BEM relaxation simulation. Solve for the initial deformed shape caused by residual stresses without external loads, and the final deformed shape, with mechanical loads and applied voltages.

   Project the simulated shape deformation onto the space spanned by the calculated modal shapes by performing the operation of dot product as mentioned above. Solve for the coefficients that determine the contribution of each mode to the shape deformation. Select which modes are significant in terms of the magnitude to the shape deformation and should be included in the calculation of the energy domains of the system.

3) Calculate strain energy

   Evaluate the strain energy of the system for a specified number of points over a specified range of motion (valid operating range). The location of each node of the system is determined from the expression derived in step 2) which describes the deformed state of the system as a linear combination of modal shapes. A single mechanical domain FEA analysis is performed at each specified point, and the strain energy is then calculated at each point. Then the strain energy for each relevant mode is plotted.

4) Calculate capacitance (electrostatic energy)

   Similar to step 3, evaluate the capacitance matrix of the system for a specified number of points over a specified range of motion. The electrostatic energy can then be calculated. A single electrostatic domain BEM analysis is performed at each specified point, and the mutual capacitance is then calculated at each point. Then the mutual capacitance for each relevant mode is plotted.

5) Calculate squeezed film damping

   Apply sinusoidal displacement loads at the selected shape modes and perform a frequency analysis. Compute the pressure distribution and project it onto the space spanned by the calculated modal shapes. Then compute the frequency dependent damping and stiffness coefficients. Fit the coefficients into a curve and transform them to frequency independent coefficients.

With calculation results obtained in steps 3, 4 and 5, the user can write the Lagrangian equations for the system.

This example will demonstrate extraction of a reduced order macromodel from coupled finite element mechanical and boundary element electrostatic simulation results and validation of the macromodel. Particularly, the squeezed film damping effect will be modeled during the extraction.

2. Macromodel extraction

2.1. Open the model file

Click *Start ... Programs ... IntelliSuite ... ThermoElectroMechanical*

This opens a window of the TEM Analysis module.

Click *File ... Open*

From the `IntelliSuite\Training\Microfluidic\SQFD\` folder, select the file `plate.save`.

Figure 1 shows the model of a pair of parallel plates.

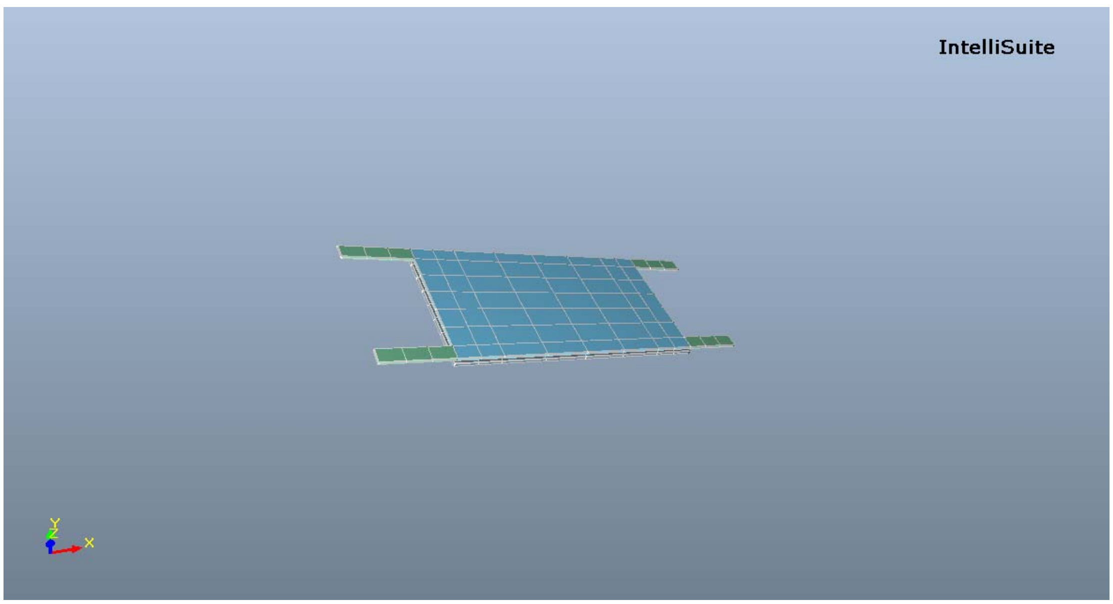


Figure 1 Model of parallel plates

2.2. Simulate modal contributions

2.2.1 Set up simulation

Click *Simulation ... Simulation Setting*

A simulation setting dialog box will appear. Specify the simulation settings as shown in Figure 2.

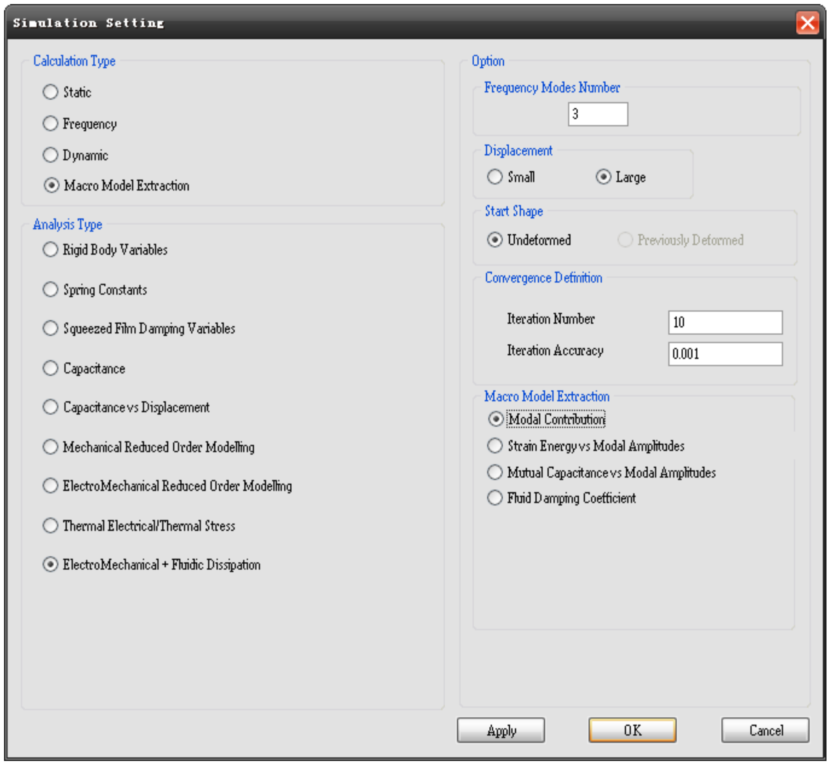


Figure 2 Simulation setting dialog box

Click *View ... Zoom ... Define*

Type 10 as the z-axis zoom factor. Then click on **OK** to close the dialog box. Figure 3 shows the model after zoom-in.

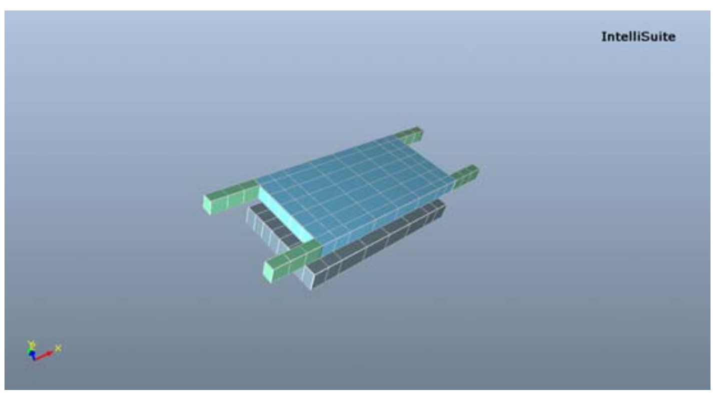


Figure 3 Zoomed model

2.2.2 Edit material properties

Click *Material ... Check/Modify*

Click on the top plate (blue) to select it. Set the Young's modulus to 160 GPa, as shown in Figure 4.

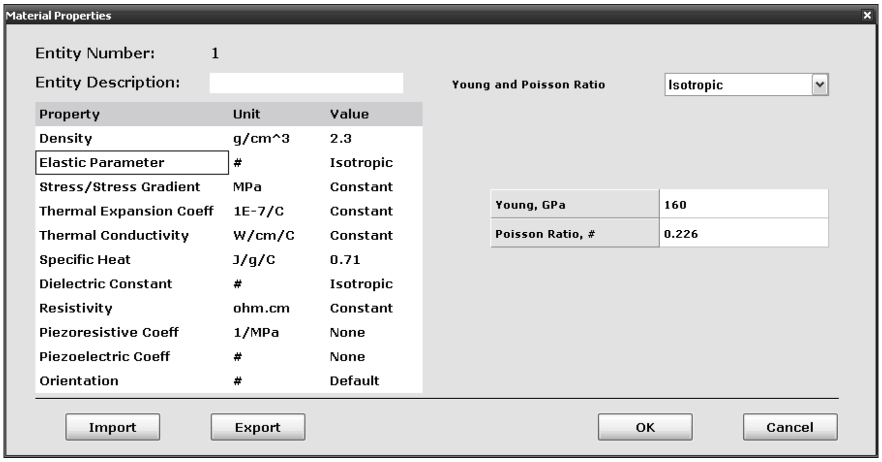


Figure 4 Material properties of top plate

Click to select the beams (green). Set the Young's modulus to 1 GPa, as shown in Figure 5.

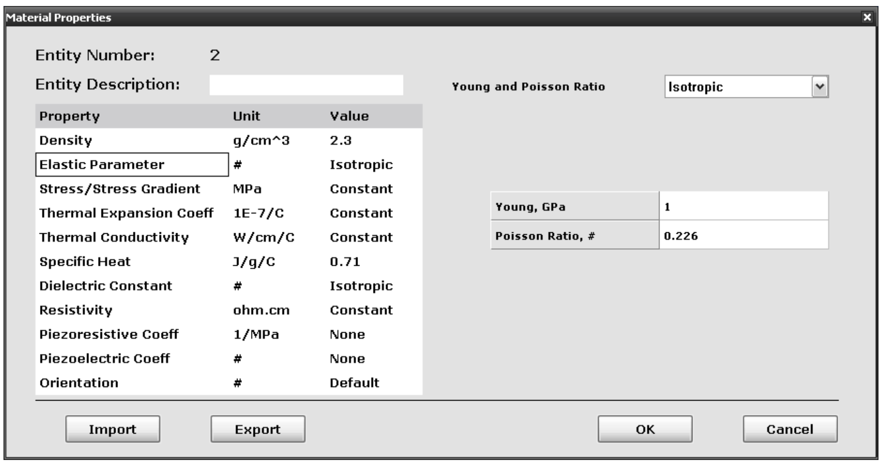


Figure 5 Material properties of beams

2.2.3 Apply loads and boundary conditions

Click *Loads … Selection Mode … Pick on Geometry*

Click *Loads … Amplitude vs Time … None*

Click *Loads … Voltage … Entity*

Select the top plate (blue) and set the voltage to 1 V (Figure 6).

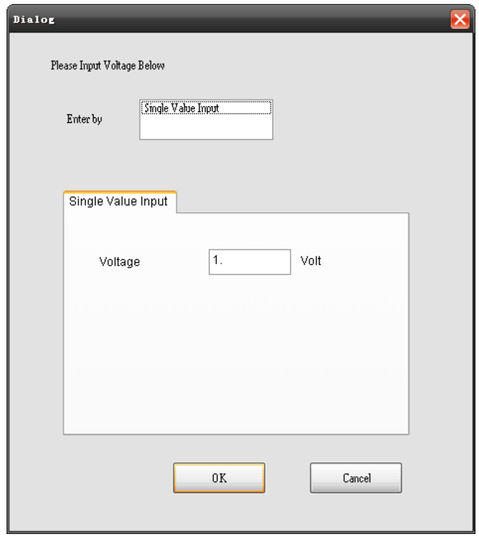


Figure 6 Voltage on top plate

Select the bottom plate (gray) and set the voltage to 0 V (Figure 7).

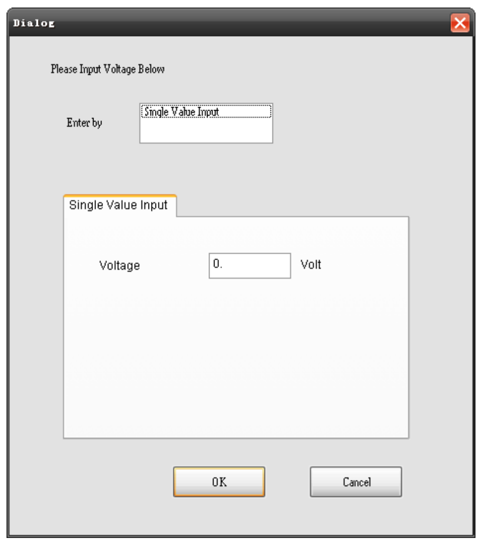


Figure 7 Voltage on bottom plate

Click *Boundary ... Selection Mode ... Pick on Geometry*

Click *Boundary ... Fixed*

Select the end faces of the four beams (faces 14, 9, 26 and 21) and click on **OK** to apply fixed boundary conditions to the faces. Select the top and bottom faces of the bottom plate (faces 2 and 1) and click on **OK** to apply fixed boundary conditions to the faces.

Click *Boundary ... Squeezed Film ... Moving Face*

Select the bottom face of the top plate (face 27).

Click *Boundary ... Squeezed Film ... Fixed Face*

Select the top face of the bottom plate (face 2).

Click *Boundary ... Squeezed Film ... Complete Pair*

A dialog box will pop up, as shown in Figure 8. Make sure that the fluid medium is gas, which means the fluid medium will be compressible.

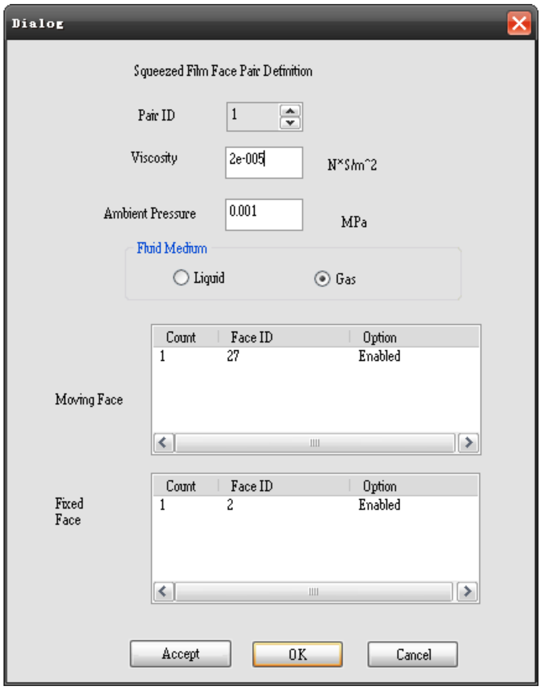


Figure 8 Definition of squeezed film face pair

Click *Boundary ... MacroModel ... Representative Nodes*

Select the center node on the top face of the top plate (Figure 9).

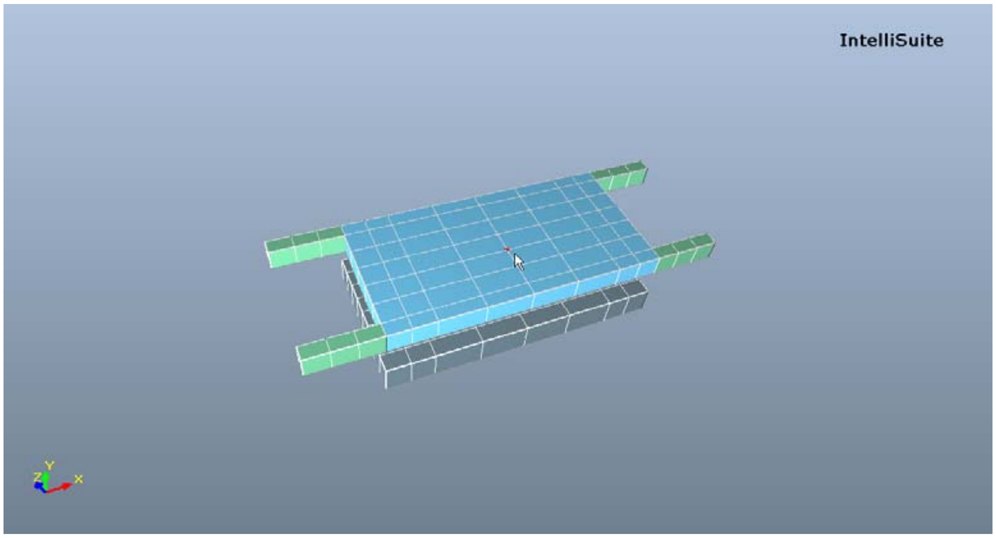


Figure 9 Selection of representative node

A dialog box will pop up. Make sure that node 880 is selected as the representative node (Figure 10).

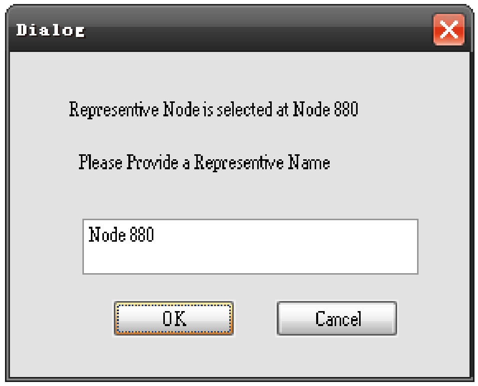


Figure 10 Selected representative node

Click on **OK** to close the dialog box. The representative node information will be saved in the file of `macromodel.out`.

2.2.4 Run analysis

Click *Analysis … Start Extract Macro Model*

2.2.5 View simulation results

When the analysis is complete,

Click *Result … MacroModel … Modal Contribution*.

A dialog box will appear listing the contribution factor of each mode to the shape deformation caused by the static voltage loads.

From the table it can be seen that the contributions from modes 2 and 3 are insignificant compared to that from mode 1. Therefore the contributions from modes 2 and 3 can be ignored for the following analyses. Double click on the 2 and 3 in the Mode column to disable these two modes (Figure 11).

Other than the modes with negligible contribution coefficients, the user should also disable the modes in which he thinks there won't be squeezed film damping from his own experience.

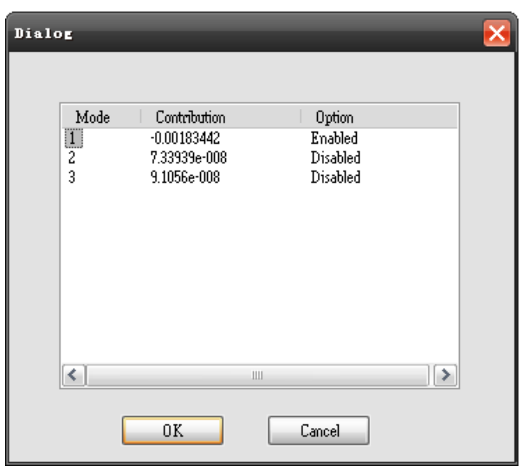


Figure 11 Modal contribution dialog box

Click on **OK** to close the dialog box.

To view animation of a mode,

Click *Result … Modal Animation*

A dialog box will pop up for input of the mode number and scaling factor (Figure 12).

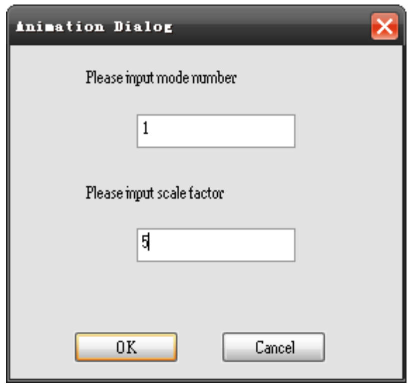


Figure 12 Animation dialog box

Click on **OK**. The animation of mode 1 will be shown. The results are consistent with our experience that there is significant squeezed filming damping in this mode.

2.3. Calculate strain energy

2.3.1 Set up simulation

Click *Simulation … Simulation Setting*

A dialog box will pop up for simulation settings (Figure 13).

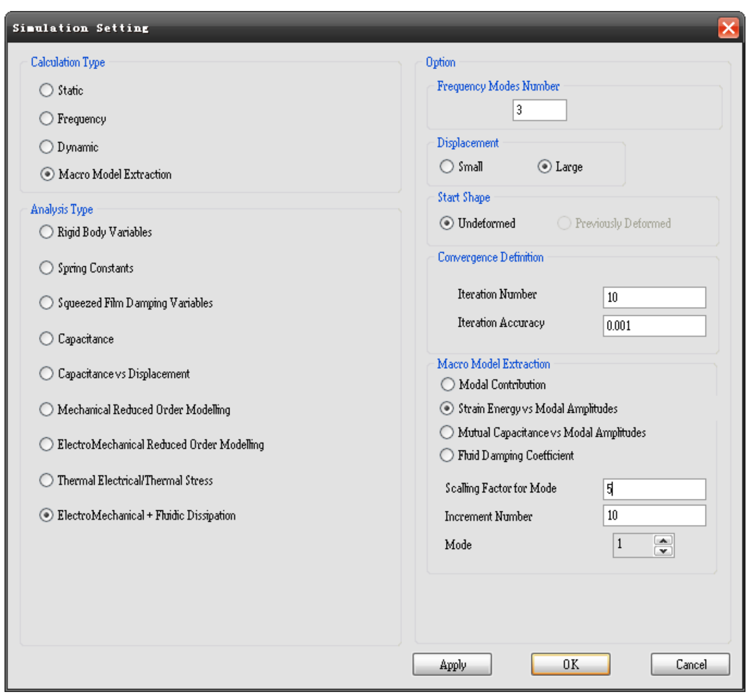


Figure 13 Strain energy simulation settings

Click on **Apply** and then **OK** to close the dialog box.

2.3.2 Run analysis

Click *Analysis ... Start Extract Macro Model*

2.3.3 View simulation results

When the analysis is complete, we can view the results.

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

A dialog box will appear listing the three modes we first obtained. Only mode 1 is enabled in the Option column because we disabled the other two modes.

Double click on 1 in the Mode column to enable the plot option for mode 1 (Figure 14). Then click on **OK**.

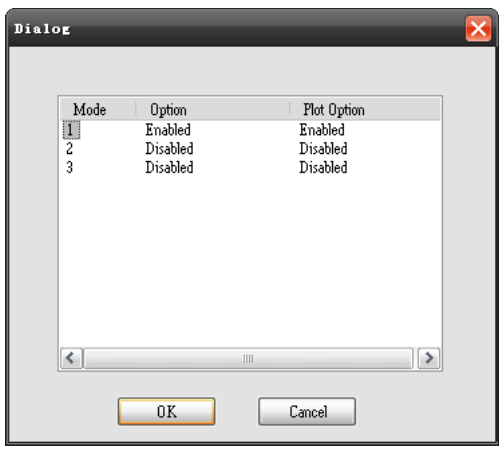


Figure 14 Plot options

The calculated strain energy of mode 1 will be plotted as shown in Figure 15.

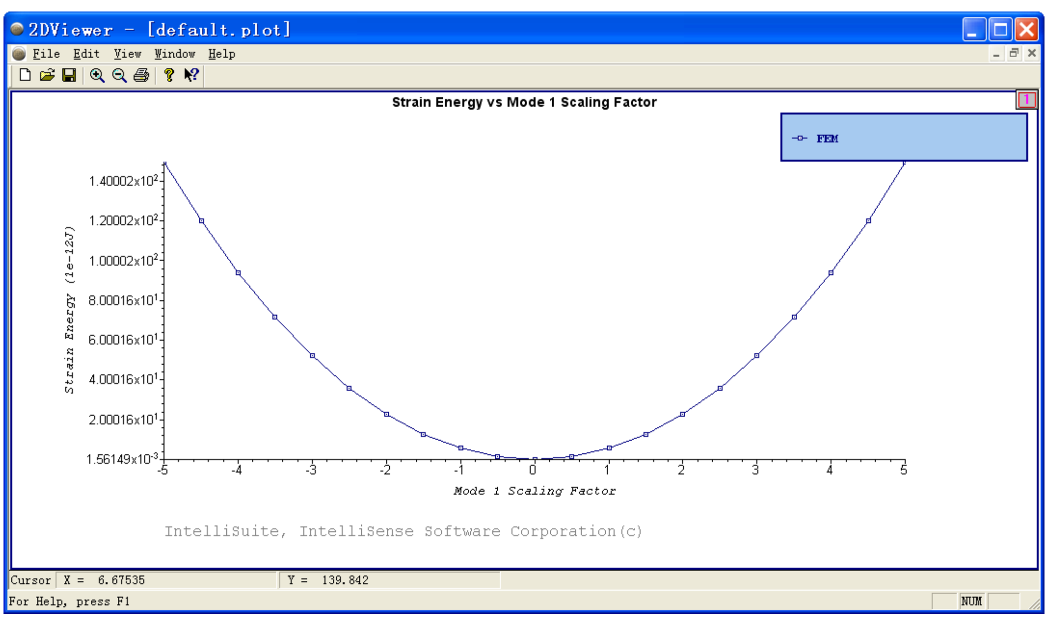


Figure 15 Strain energy of mode 1

2.4. Calculate capacitance (electrostatic energy)

2.4.1 Set up simulation

Click *Simulation ... Simulation Setting*

A dialog box will pop up for simulation settings (Figure 16).

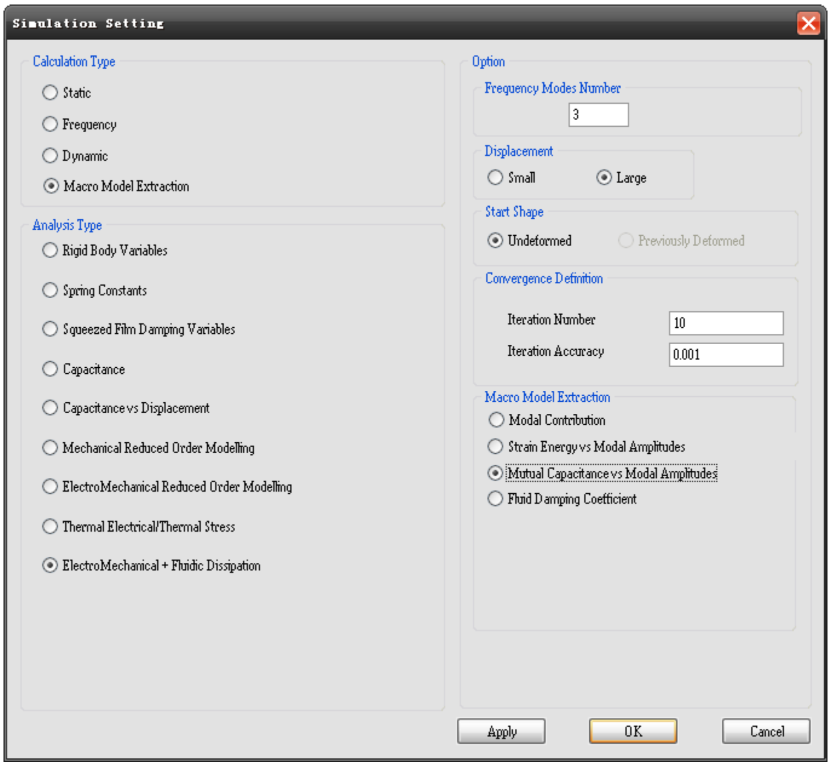


Figure 16 Electrostatic energy simulation settings

Click on **OK** to close the dialog box.

2.4.2 Run analysis

Click *Analysis ... Start Extract Macro Model*

2.4.3 View simulation results

When the analysis is complete, we can view the results.

Click *Result ... MacroModel ... Mutual Capacitance vs Modal Amplitudes*

A dialog box will appear to show the capacitance matrix of the structure.

The first and last capacitances are the capacitances between different parts of the same entities, and the second and third capacitances are the mutual capacitances between different entities. We are only interested in the mutual capacitances because they reflect the capacity of electrostatic energy storage between different entities.

Double click on Entity 1 of the second capacitance below the column of Conductor 1. A new dialog box will pop up (Figure 17).

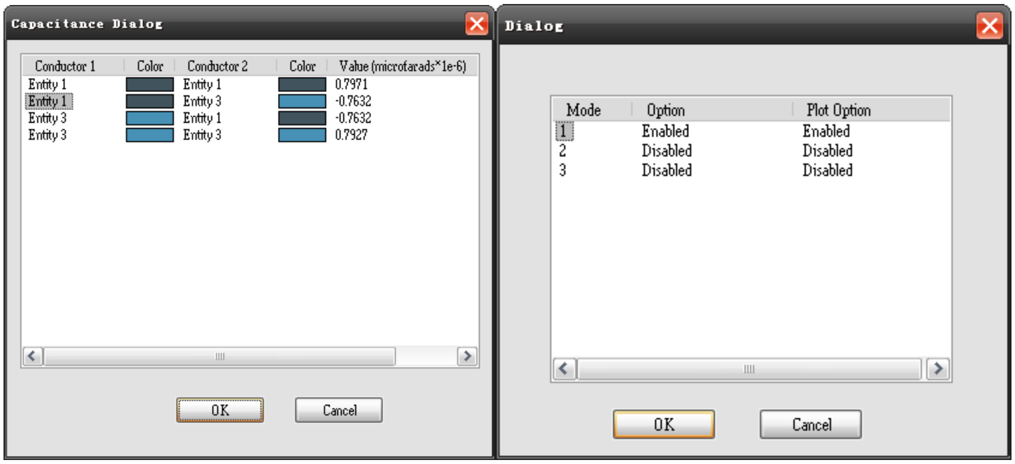


Figure 17 Capacitances and plot options

Double click on 1 in the Mode column to enable the plot option for mode 1. Then click on **OK**. The calculated capacitance of mode 1 will be plotted as shown in Figure 18.

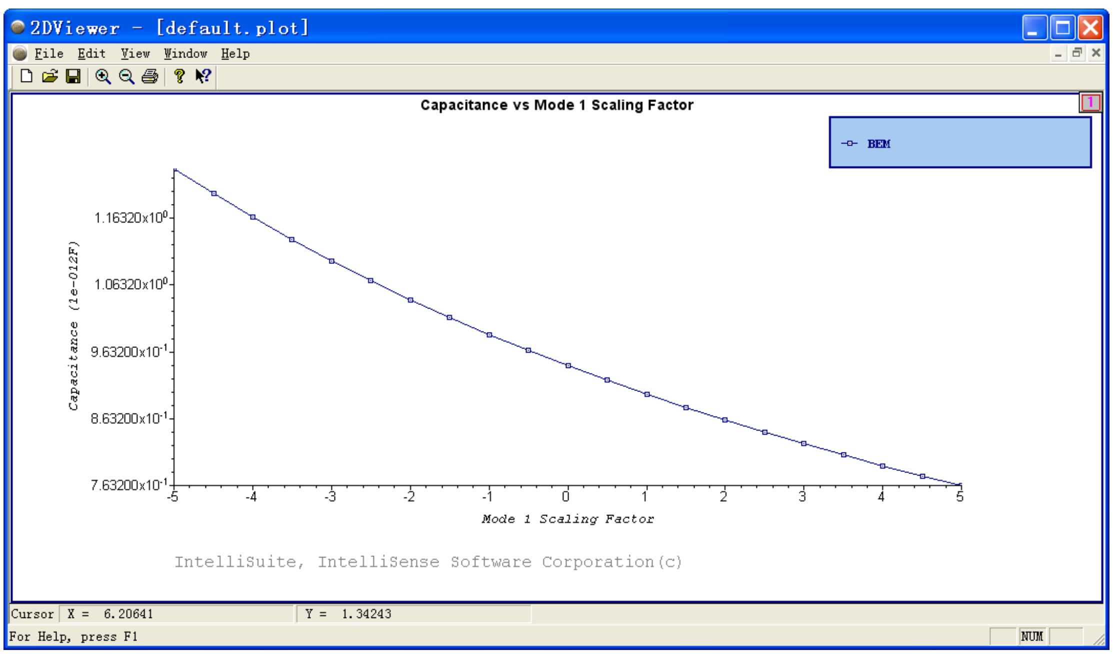


Figure 18 Capacitances of mode 1

The user should plot the mutual capacitances for every enabled mode here because the mutual capacitance values for each selected mode need to be plotted for them to be saved in a format easy to read.

In this example, only mode 1 is enabled.

2.5. Calculate squeezed film damping (SQFD)

2.5.1 Set up simulation

Click *Simulation … Simulation Setting*

A dialog box will pop up for simulation settings (Figure 19).

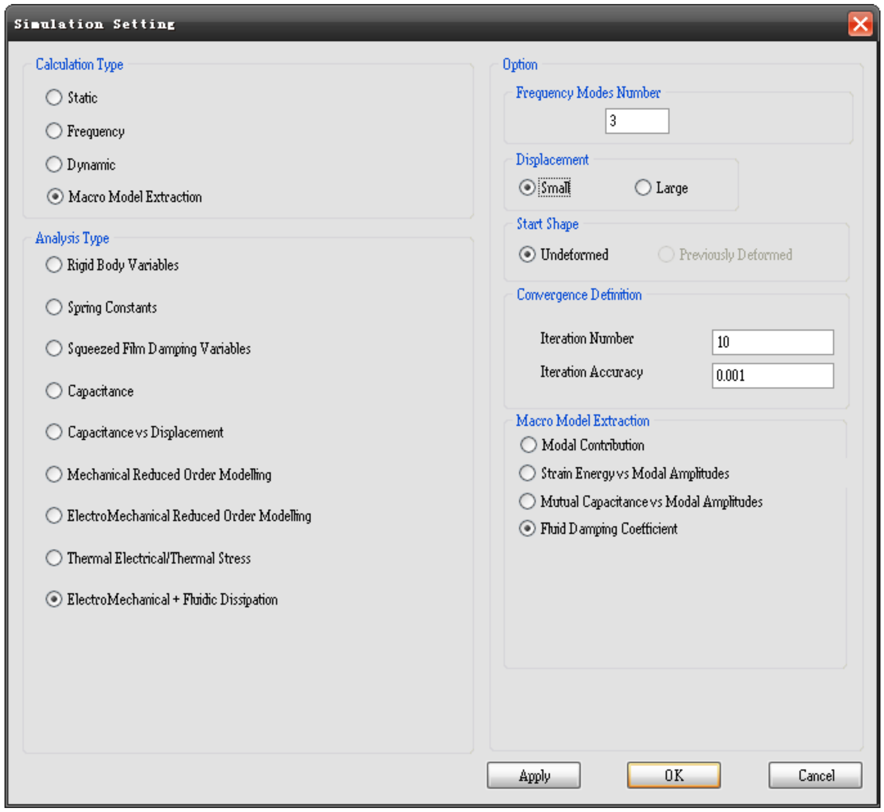


Figure 19 Squeezed film damping simulation settings

Here we treat the system as a linear system. So the box of Small displacement is checked. If the user wants to treat the system as a non-linear one, the box of Large displacement should be checked. Click on **OK** to close the dialog box.

2.5.2 Run analysis

Click *Analysis ... Start Extract Macro Model*

Enter parameter values in the dialog box that pops up, as shown in Figure 20.

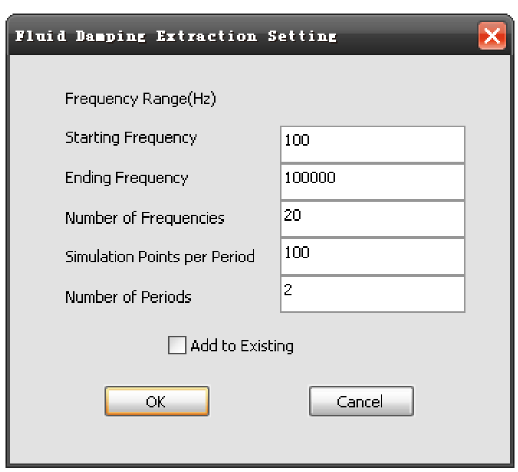


Figure 20 SQFD extraction settings

Click on **OK** to start analysis.

2.5.3 View simulation results

When the analysis is complete,

Click *Result … Fluid Damping vs Frequency*

A dialog box will pop up. Select an exciting mode and a projection mode for the modal based forces/coefficients, as shown in Figure 21.

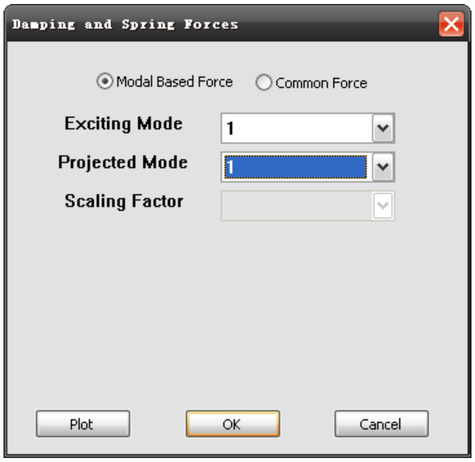


Figure 21 Plot options for damping and spring forces

Click on **Plot**. The damping and spring coefficients/forces will be plotted (Figure 22).

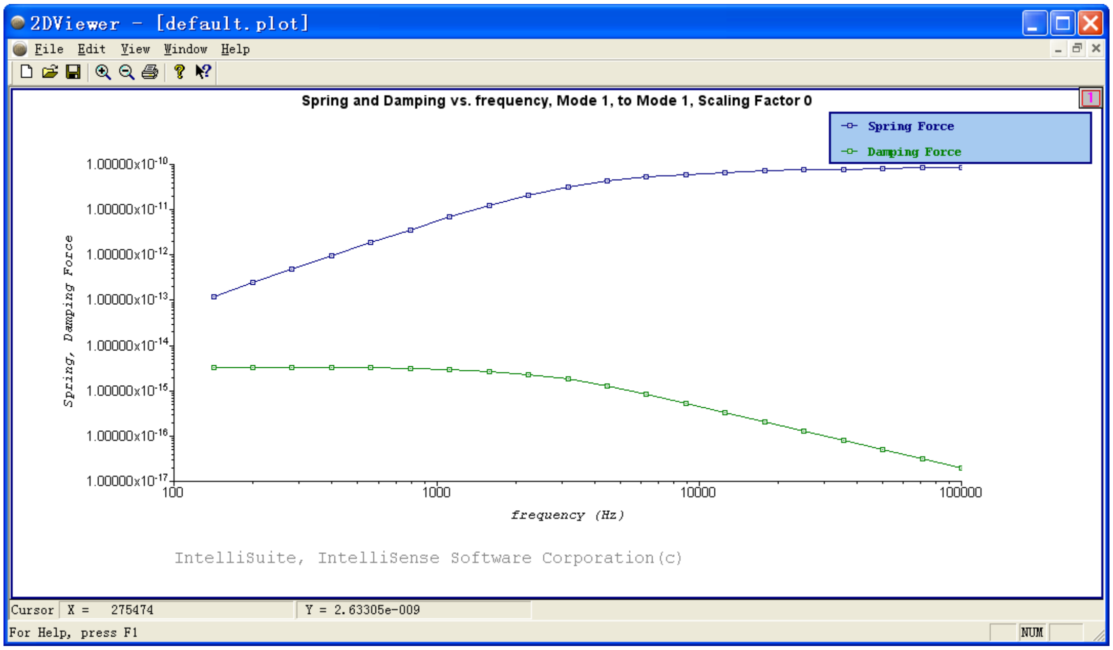


Figure 22 Damping and stiffness coefficients vs frequency

To save the plot,

Click *File ... Save As*

Save the plot as `C_K_Model.plot`.

The user can also plot the actual force on the moving face. For example, on the damping and spring force setting dialog box, the user can check the box of common force (Figure 23).

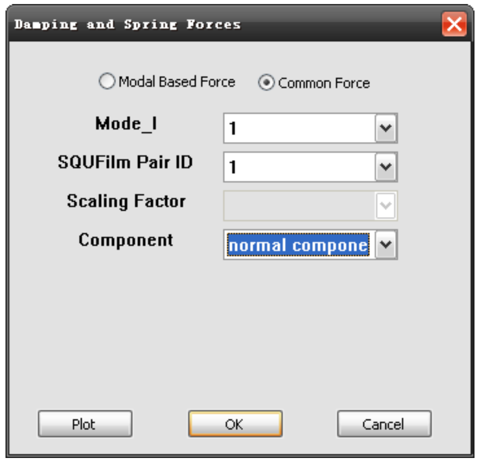


Figure 23 Plot options for damping and spring forces

Clicking on **Plot** will plot a curve of the force acting on the moving face of the squeezed film pair (Figure 24).

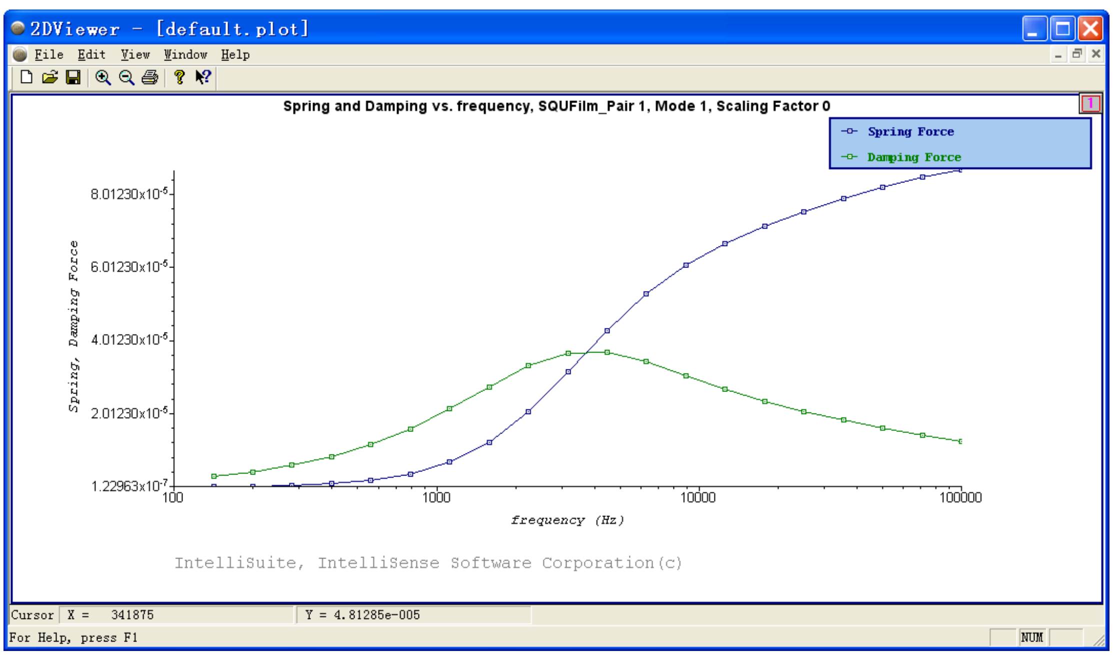


Figure 24 Viscous and spring forces on moving face of mode 1

The two curves show the two terms of the damping force: viscous force and spring force. The unit is Newton per scaling factor. Save the plot as `Force_SQUF_Model.plot`.

So far we have generated all the information required to create a reduced order model. All the information is stored in files. These files will be used in the macromodel simulation in the SYNPLE module.

3. Macromodel validation

In this section, the extracted reduced order macromodel with squeezed film damping will be validated in two ways.

1. Damping force validation. The damping forces simulated using TEM will be compared with results calculated using theoretical equations for the case of parallel plates.

2. Transient analysis validation. Electrostatic loads will be applied to the pair of parallel plates. Then a transient analysis will be performed in both TEM and SYNPLE. The TEM FEA simulation results will be compared with the SYNPLE macromodel simulation results. After that, a transient analysis will be performed in SYNPLE using an element-based system-level model created based on the parallel plates. The element-based system-level model simulation results will be compared with the macromodel simulation results.

3.1. Damping force validation

In [4], an analytical solution for the viscous and spring coefficients of the case that a rigid plate moves with a transverse motion is given by two equations below.

$$C(\Omega) = \frac{64\sigma p_0 A}{\pi^6 d\Omega} \sum_{m=odd} \sum_{n=odd} \frac{m^2 + n^2 c^2}{(mn)^2 \left[\left(m^2 + n^2 c^2 \right)^2 + \sigma^2 / \pi^4 \right]} \tag{8}$$

$$K(\Omega) = \frac{64\sigma^2 p_0 A}{\pi^8 d} \sum_{m=odd} \sum_{n=odd} \frac{1}{(mn)^2 \left[\left(m^2 + n^2 c^2 \right)^2 + \sigma^2 / \pi^4 \right]} \tag{9}$$

where *C(Ω)* is the frequency dependent damping coefficient; *K(Ω)* is the squeeze stiffness coefficient; p_0 is the ambient pressure; *d* is the thickness of the film; *Ω* is the response frequency; *A* is the area of the plate; *c = L/w* is the length to width ratio of the plate; σ is the squeeze number of the system.

$$\sigma = \frac{12\eta L^2 \Omega}{p_0 d^2} \tag{10}$$

L and *w* are the length and width of the plate; *η* is the dynamic viscosity.

The viscous and spring forces normalized by the transverse displacement of the rectangular plate are

$$\begin{aligned} F_{vis} &= C\Omega \\ F_{spr} &= K \end{aligned} \tag{11}$$

In section 2.2, the simulated mode 1 is a transverse motion. The main structure parameters of the plates are:

L = 2e-3 m
w = 1e-3 m
d = 2e-5 m
η = 2e-5 Pa·s
p_0 =1000 Pa

In section 2.5, we calculated the viscous and spring forces. The force values were normalized by the scaling factor and saved in the file of `Force_SQUF_Mode1.plot`. The unit of shape deformation used in simulation of modal shapes was μm. So the force values should be multiplied by 10^6 if the unit of normalized forces is N/m for comparison.

The viscous and spring forces simulated using TEM and analytical equations are plotted as shown in Figure 25. As can be seen, the results obtained from the two methods are close.

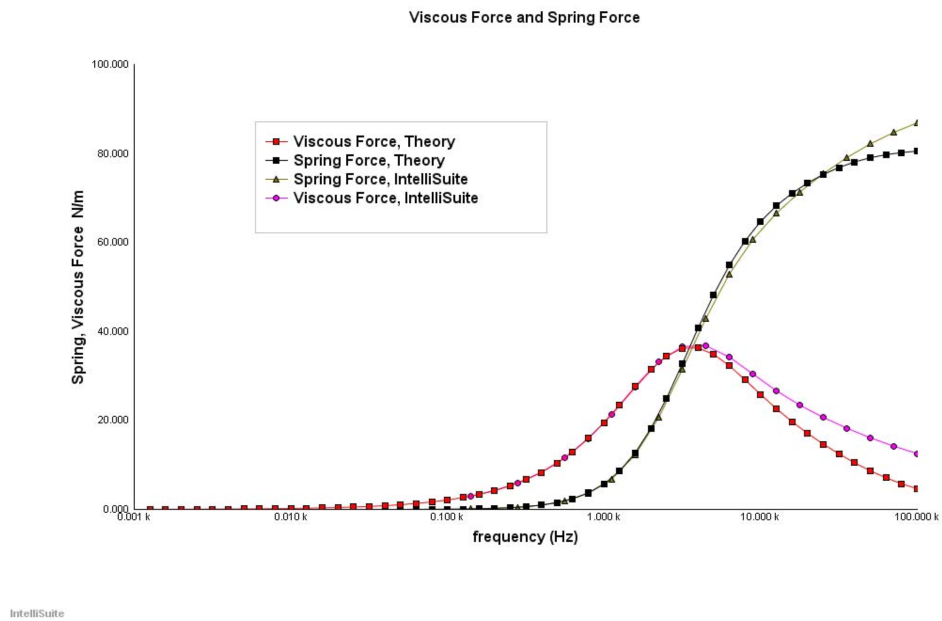


Figure 25 Viscous and spring force comparison

3.2. Transient analysis validation

In this section, a transient analysis will be performed using the generated macromodel and the FEA model in SYNPLE and TEM, respectively. Then the simulated behavior at the representative node using the two methods will be compared in WaveRunner.

3.2.1 Macromodel analysis in SYNPLE

3.2.1.1 Open the macromodel file

Click *Start … Programs … IntelliSuite … SYNPLE*

This opens a window of the SYNPLE module.

During the macromodel extraction, some (temporary) files were generated, three of which are needed for analyses in SYNPLE. These three files are `curr.macmodel`, `str.out` and `macmodel.out`, and they should be copied to the user's SYNPLE simulation work directory. In this example, the three files are already in the work directory of `IntelliSuite\Training\Microfluidic\SQFD\macromodel`.

Click *File … Open*

From the folder of `IntelliSuite\Training\Microfluidic\SQFD\macromodel`, select the file of `plate.ssc`.

Figure 26 shows the schematic of the system.

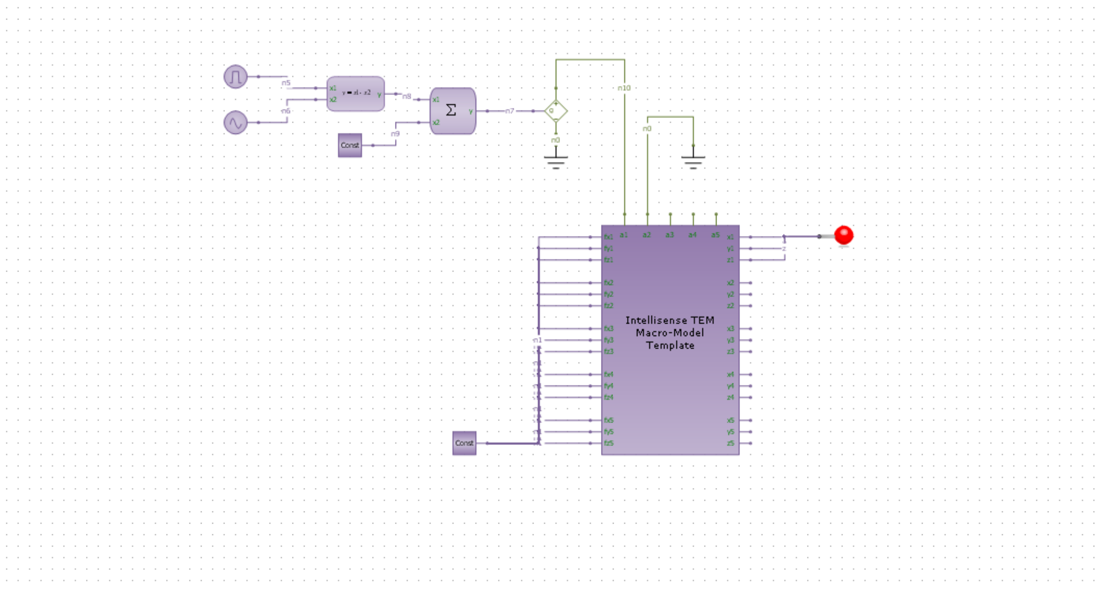


Figure 26 System schematic

3.2.1.2 Load the macromodel

Click *Tools … Load ROM Macro-model*

From the folder of `IntelliSuite\Training\Microfluidic\SQFD\macromodel`, select the file of `curr.macmodel`.

If the macromodel is successfully loaded, a message window will appear as shown in Figure 27.

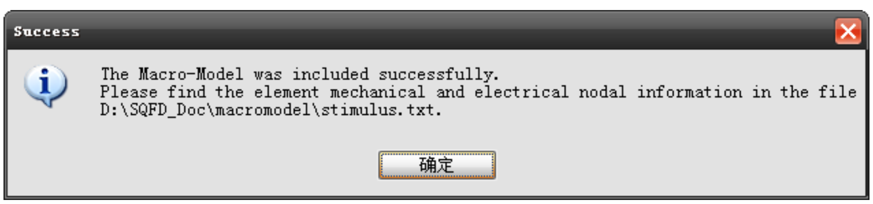


Figure 27 Message window for macromodel loading

3.2.1.3 Set up simulation and run analysis

Click *Simulation … Transient Analysis*

Set the parameters under the tabs Simulation Setup, Signals, Schemes, and Convergence Setup as shown in Figure 28. The tab of Simulation Setup allows the user to specify the transient analysis settings. The tab of Signals offers the options of selecting the signal responses that the user wants to view. The tab of Schemes allows the user to select the type of discretization for the simulation. Refer to *SYNPLE Advanced Guide* for an explanation of each of these discretization schemes. The tab of Convergence Setup lets the user specify the convergence criteria for the simulation. Refer to *SYNPLE Advanced Guide* for an explanation of the N-R convergence parameters.

Click on **OK** to start analysis.

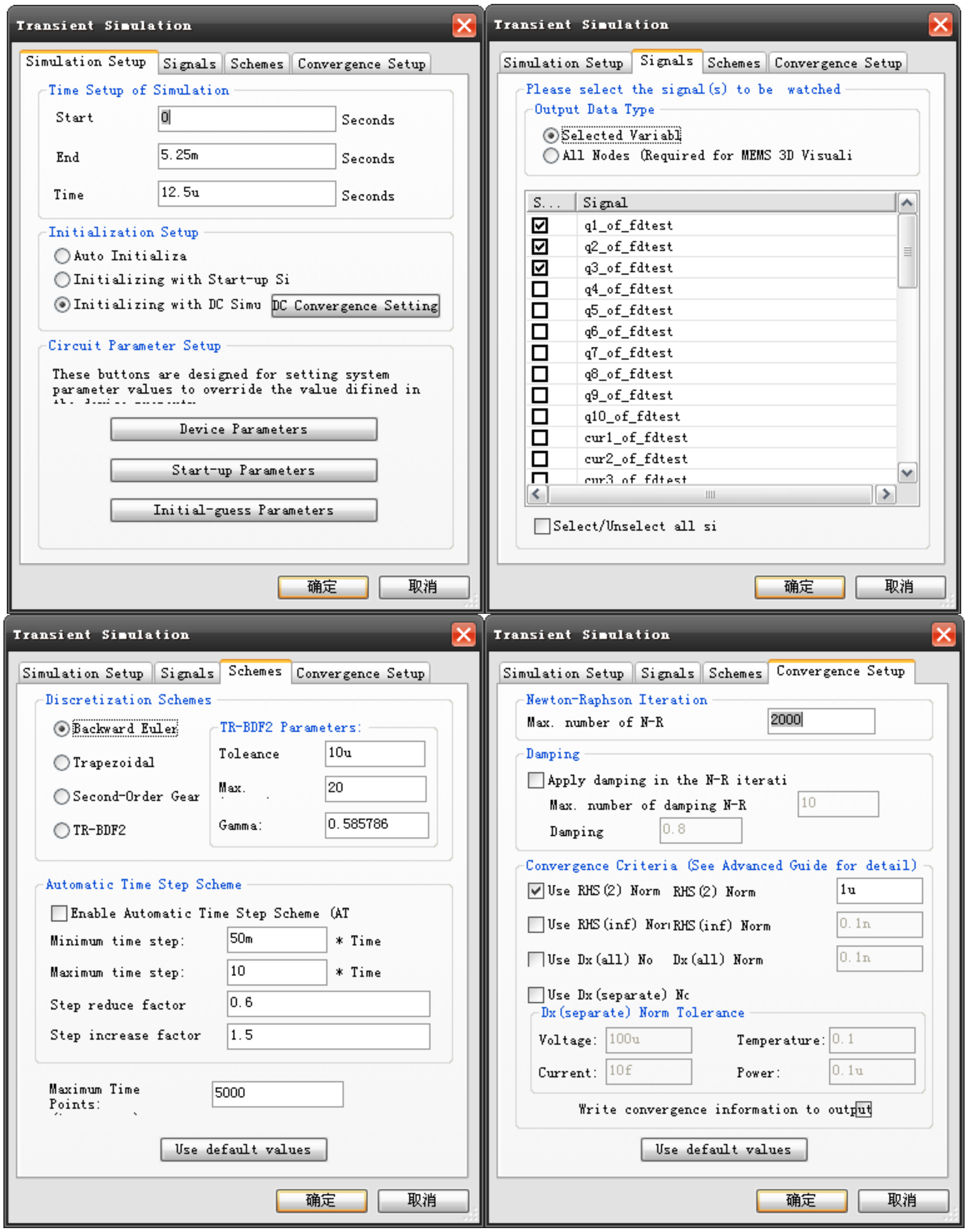


Figure 28 Transient simulation settings

3.2.1.4 View simulation results and save plots

When the analysis is complete, a Plot Manager window will pop up as shown in Figure 29. Double clicking on a signal on the Plot Manager will plot the curve of the signal on a separate window.

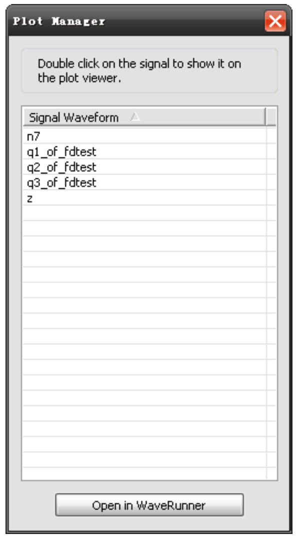


Figure 29 Plot manager

We are interested in the input voltage and displacement at the representative node. Double click on *n7*. The input voltage will be plotted as a curve shown in Figure 30.

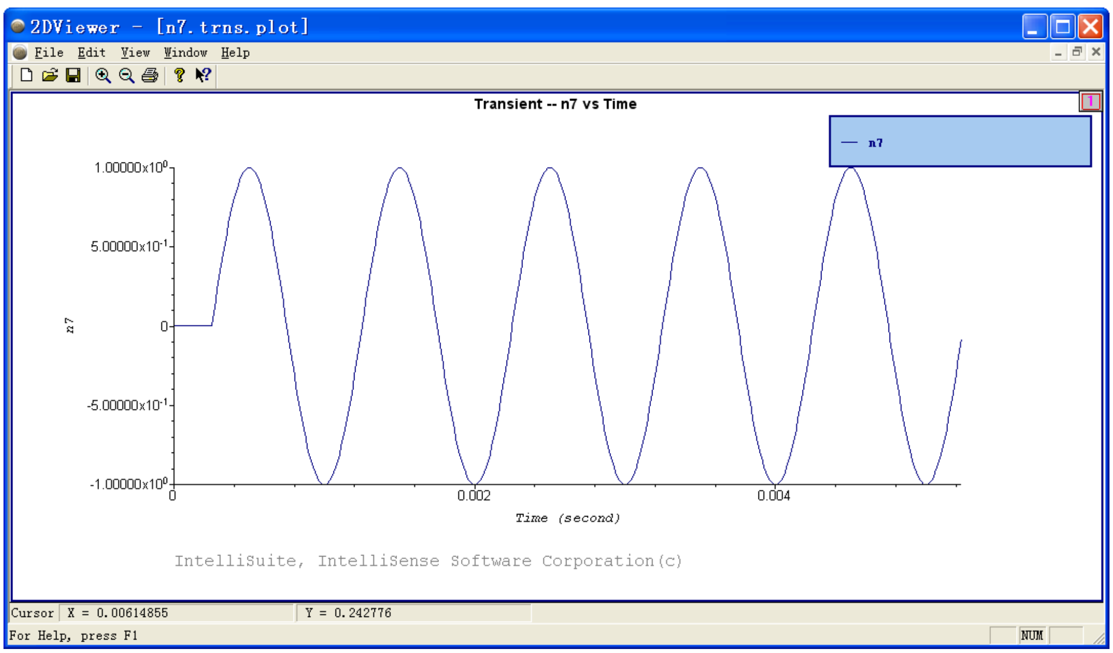


Figure 30 Input voltage at the representative node

Double click on *z*. The displacement at the representative node will be plotted as shown in Figure 31.

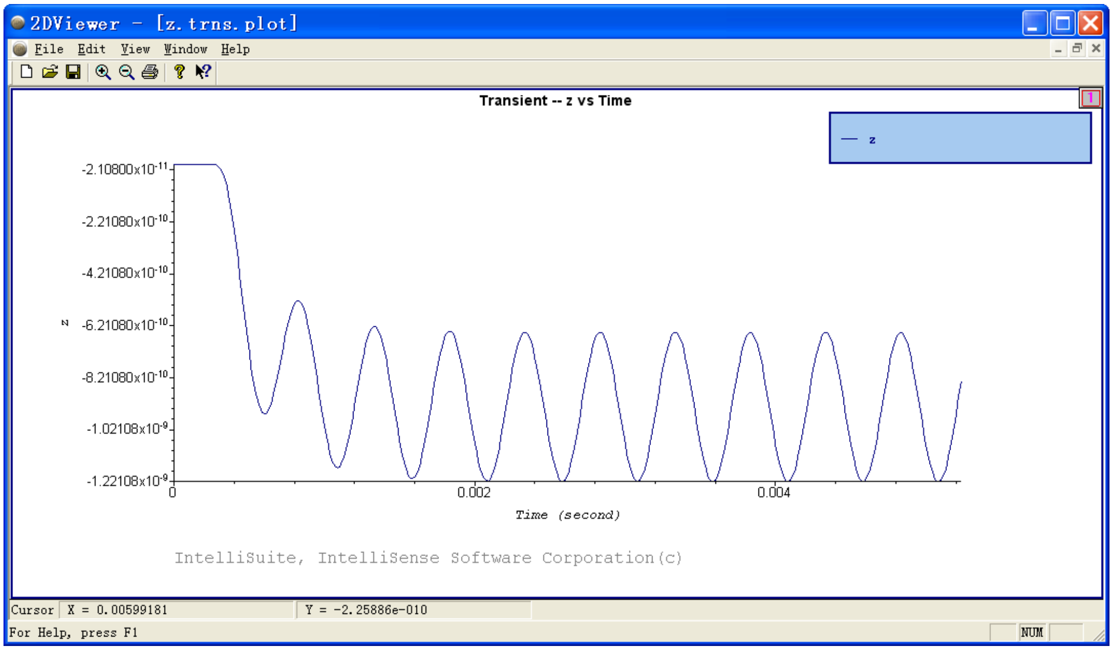


Figure 31 Z-displacement at the representative node

To save the plot, on 2DViewer,

Click *File ... Save As.*

Then save the plot as `z_mac.plot`.

3.2.2 Finite element analysis in TEM

3.2.2.1 Open the finite element model file

Click *Start ... Programs ... IntelliSuite ... ThermoElectroMechanical*

A window of the TEM module will open.

Click *File ... Open*

From the folder of `IntelliSuite\Training\Microfluidic\SQFD\fem`, select the file of `plate.save`.

Figure 32 shows the finite element model of the parallel plates.

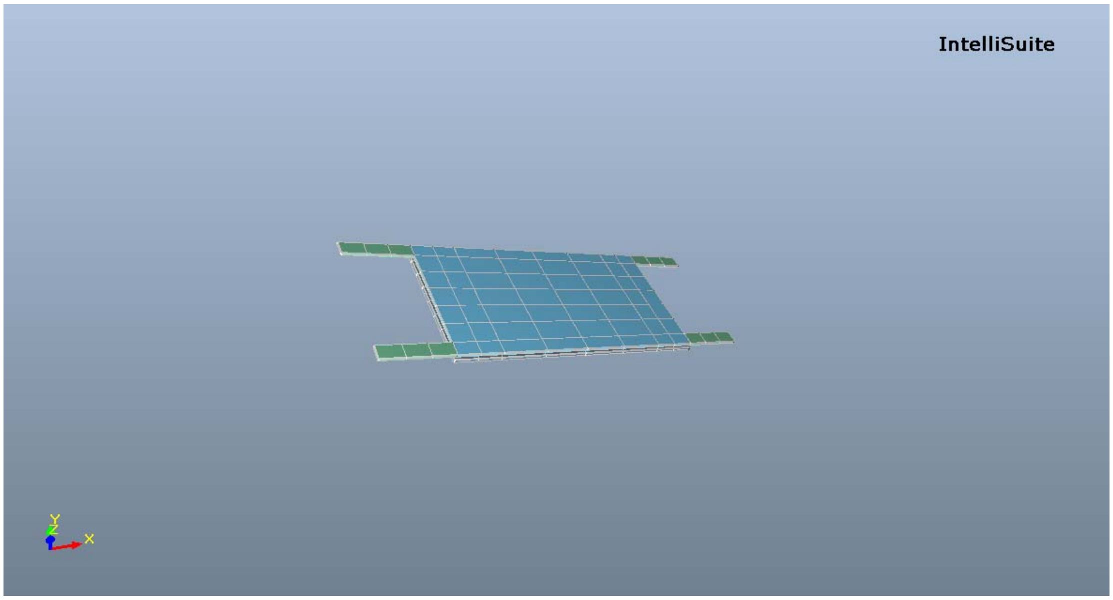


Figure 32 Finite element model of parallel plates

3.2.2.2 Set up simulation

Click *Simulation … Simulation Setting*

A simulation setting dialog box will appear. Specify the simulation settings as shown in Figure 33.

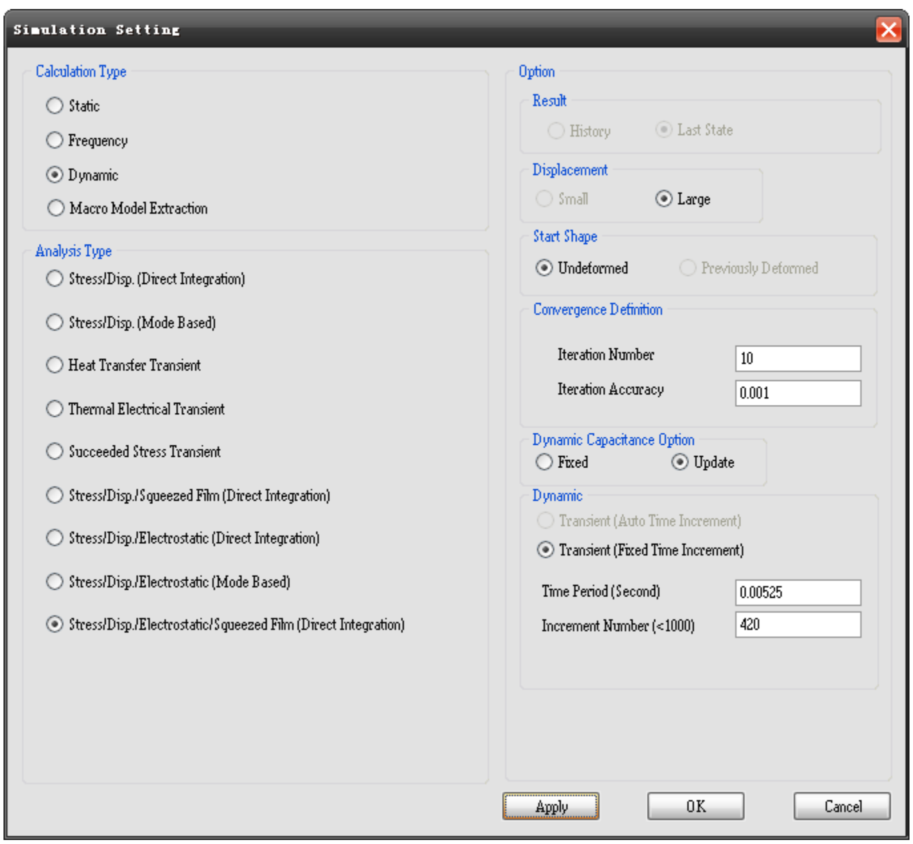


Figure 33 Simulation setting dialog box

Click *View ... Zoom ... Define*

Type 10 for the z-axis zoom factor. Then click on **OK** to close the dialog box. Figure 34 shows the model after zoom-in.

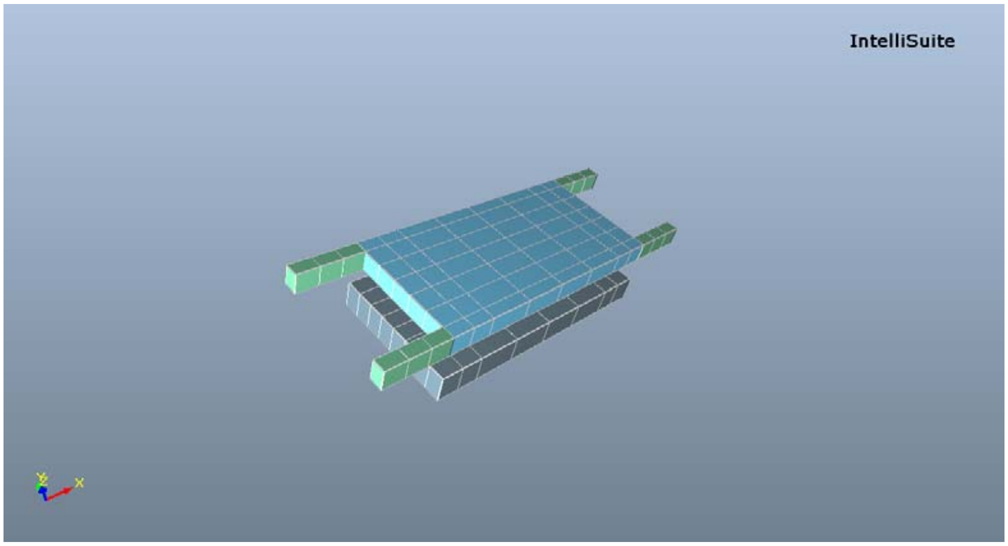


Figure 34 Zoomed model

3.2.2.3 Edit material properties

Click *Material ... Check/Modify*

Click on the top plate (blue) to select it. Set the Young's modulus to 160 GPa, as shown in Figure 35.

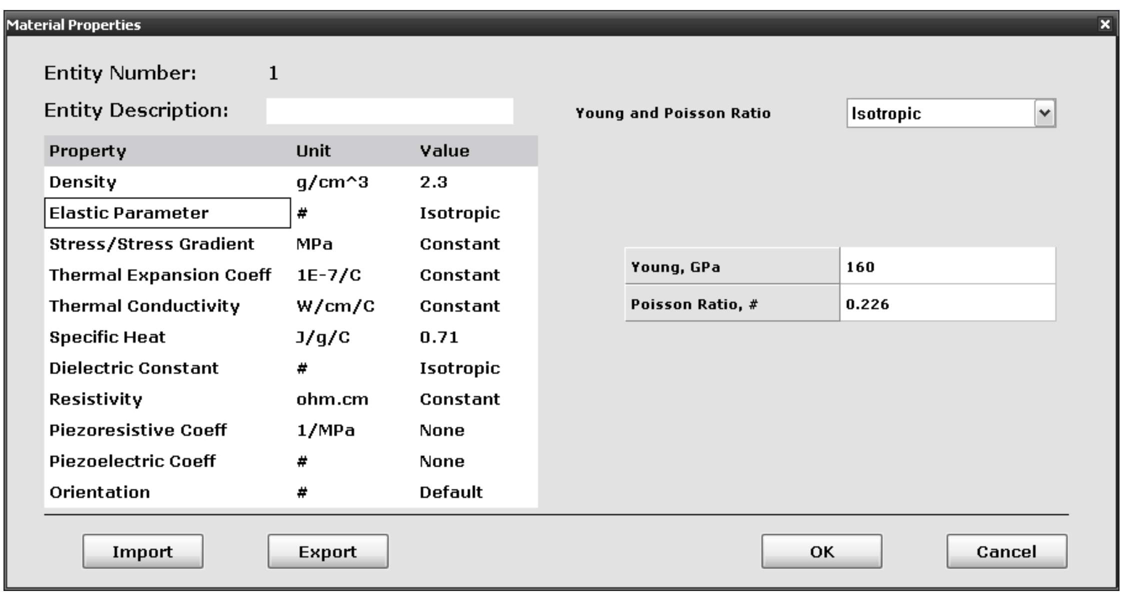


Figure 35 Material properties of top plate

Click to select the beams (green). Set the Young's modulus to 1 GPa, as shown in Figure 36.

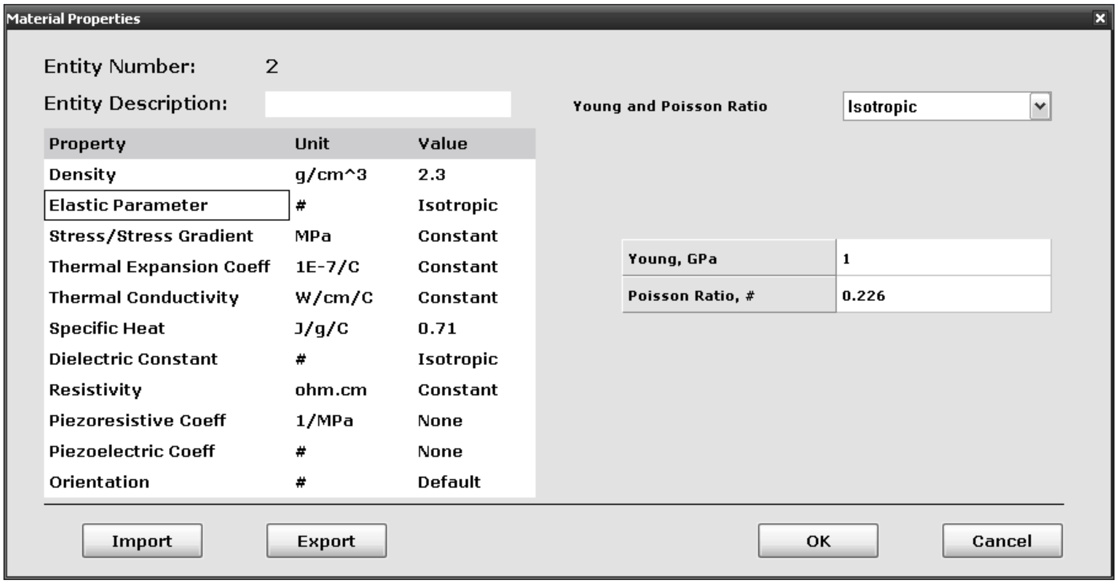


Figure 36 Material properties of beams

Click *Material ... Damping Definition*

Click on the top plate (blue) to select it. Leave the Mass_damping and Stiffness_damping as 0, as shown in Figure 37.

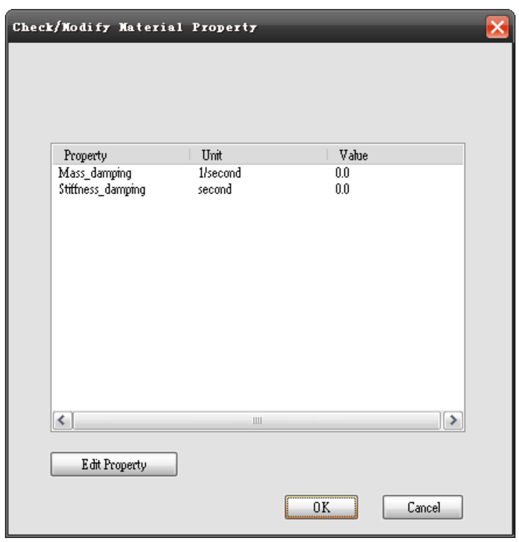


Figure 37 Material damping definition

3.2.2.4 Apply loads and boundary conditions

Click *Loads ... Selection Mode ... Pick on Geometry*

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

Click *Loads ... Voltage ... Entity*

Select the top plate (blue) and set the voltage to the same as in SYNPLE simulation (Figure 38).

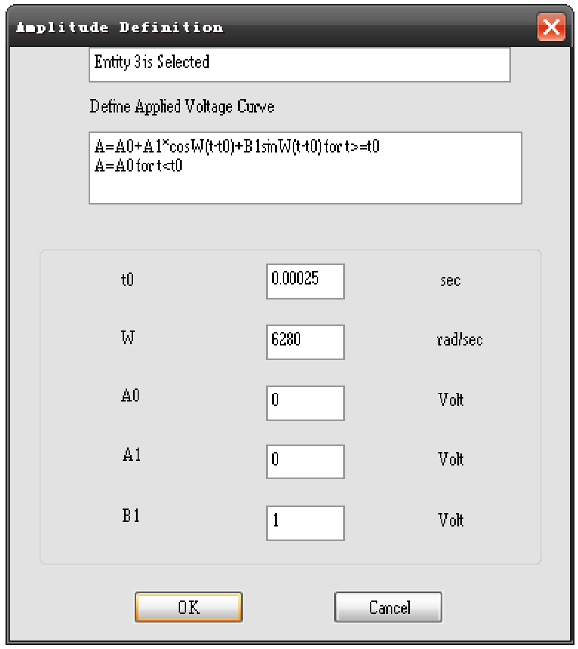


Figure 38 Voltage on top plate

Select the bottom plate (gray) and set the voltage to 0 V (Figure 39).

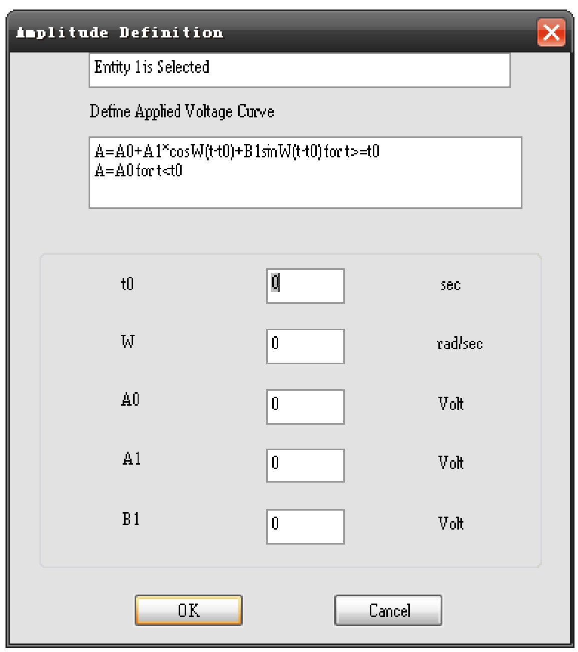


Figure 39 Voltage on bottom plate

Click *Boundary … Selection Mode … Pick on Geometry*

Click *Boundary … Fixed*

Select the end faces of the four beams (faces 14, 9, 26 and 21) and click on **OK** to apply fixed boundary conditions to the faces (Figure 40).

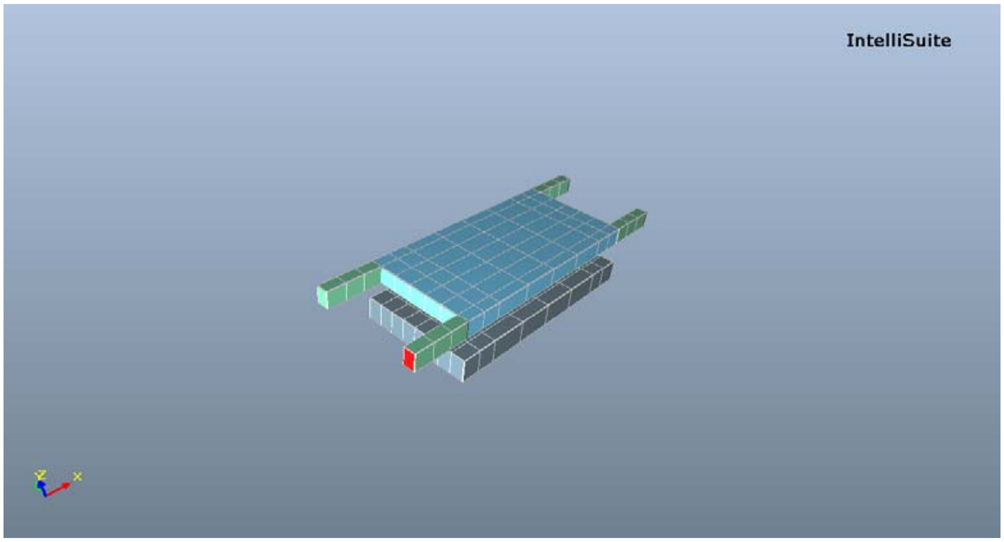


Figure 40 Fixed boundary conditions on beams

Select the top and bottom faces of the bottom plate (faces 2 and 1) and click on **OK** to apply fixed boundary conditions to the faces (Figure 41).

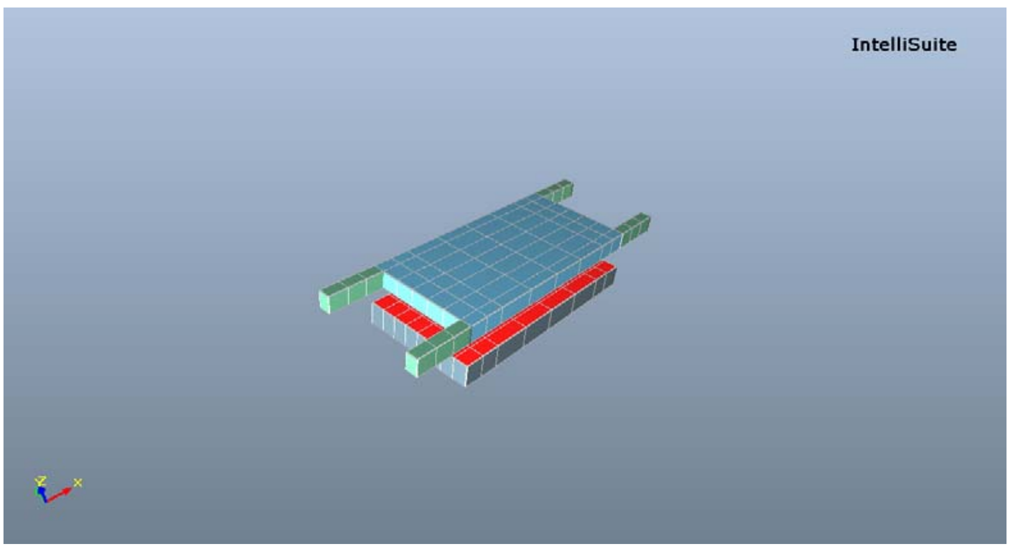


Figure 41 Fixed boundary conditions on bottom plate

Click *Boundary … Squeezed Film … Moving Face*

Select the bottom face of the top plate (face 27) as shown in Figure 42.

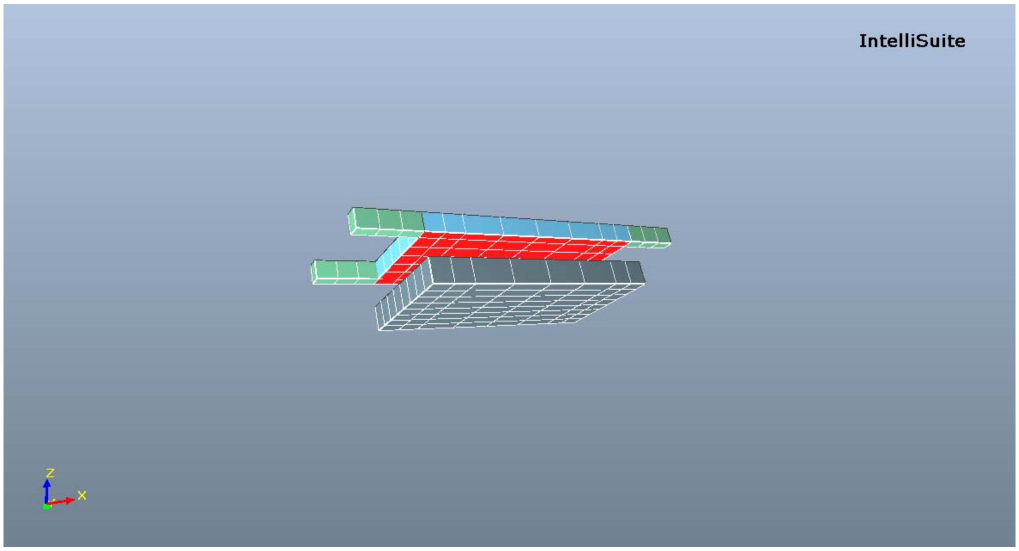


Figure 42 Moving face of squeezed film pair

Click *Boundary … Squeezed Film … Fixed Face*

Select the top face of the bottom plate (face 2) as shown in Figure 43.

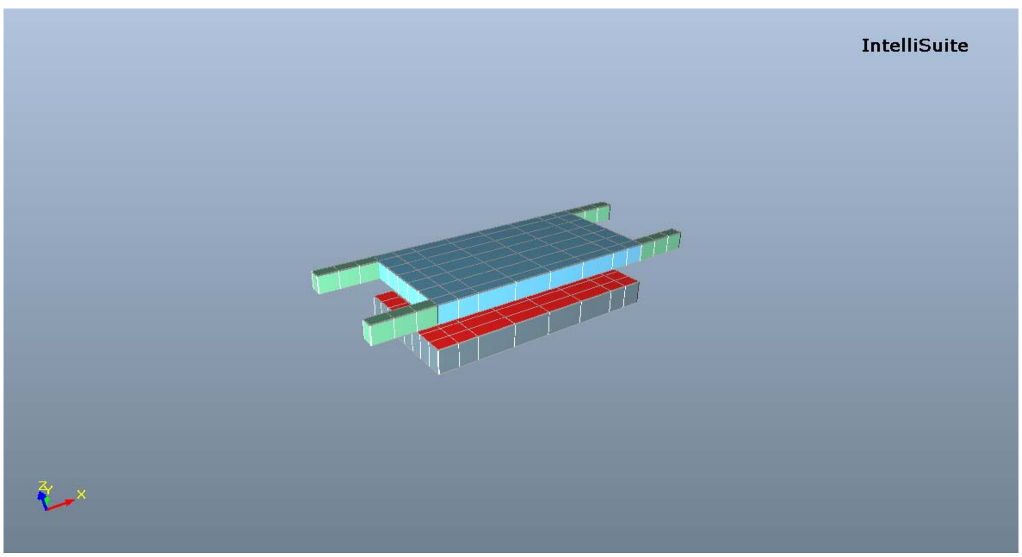


Figure 43 Fixed face of squeezed film pair

Click *Boundary … Squeezed Film … Complete Pair*

A dialog box will pop up, as shown in Figure 44. Make sure that the fluid medium is gas, which means the fluid medium will be compressible.

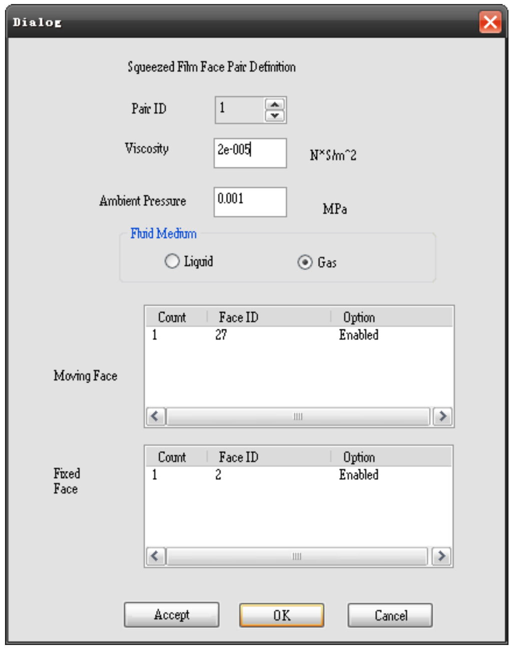


Figure 44 Definition of squeezed film face pair

3.2.2.5 Run analysis

Click *Analysis … Start Dynamic Analysis*

3.2.2.6 View simulation results and save plots

The simulation will take more than five hours. When the analysis is complete, we can view the displacement of the nodes on the plates along z-axis.

Click *Result … Displacement … Z*

Figure 45 shows the displacement of the nodes on z-axis.

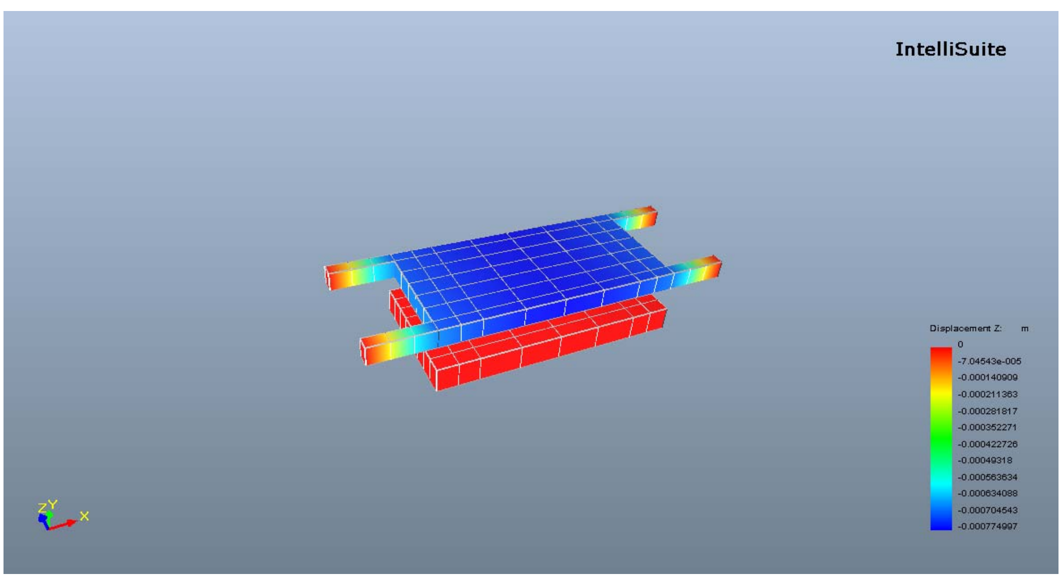


Figure 45 Z-displacement of nodes on plates

To view the z-displacement at the representative node,

Click *Result ... Node Curve ... Magnitude.*

Click to select the representative node (Node 880), which will then be highlighted, as shown in Figure 46.

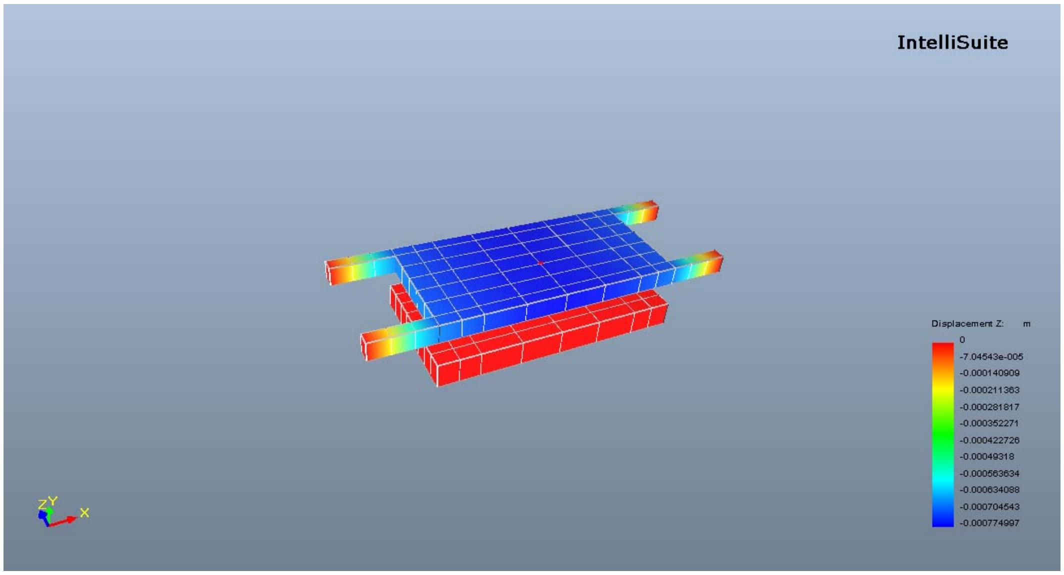


Figure 46 Selecting representative node

The z-axis displacement at the representative node (Node 880) will be plotted (Figure 47).

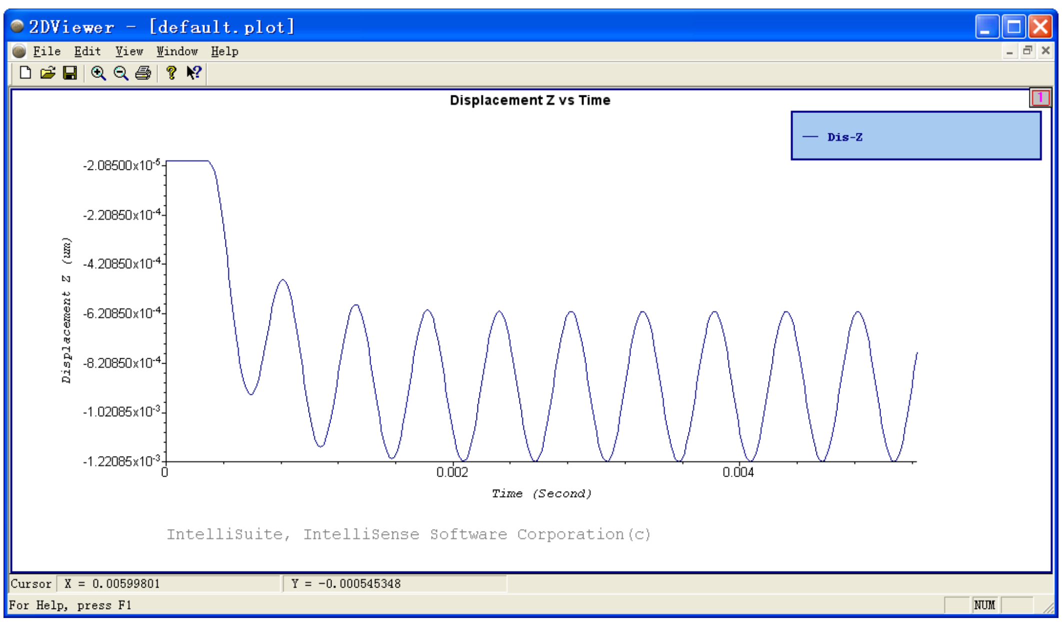


Figure 47 Z-displacement at the representative node

To save the plot, on 2DViewer,

Click *File ... Save As.*

Then save the plot as `z_tem.plot`.

3.2.3 Comparison of simulation results in WaveRunner

Now we compare the z-displacement results at the representative node obtained from TEM and SYNPLE in WaveRunner.

Click *Start ... Programs ... IntelliSuite ... Utilities ... WaveRunner*

This will open a main window of the WaveRunner module.

Click *Chart ... Import Data Source*

Select the file of `z_mac.plot` from where it was saved previously.

Click *Chart ... Import Data Source*

Select the file of `z_tem.plot` from where it was saved previously.

Click to expand the tree on the panel of Data and Charts (Figure 48).

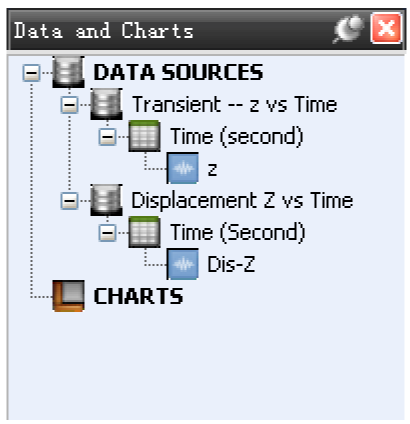


Figure 48 Data sources and charts

The units of the numbers in `z_mac.plot` are SI units, e.g., the unit of length is m. We will convert the numbers in m to numbers in µm, the unit used in `z_tem.plot`.

Click *Chart ... Add Signal*

A dialog box will pop up for new signal adding, which lets the user create new data and signals based on the originally input data by performing mathematic operations on the original data. Here, we create a new signal 10^6 times the original signal z, as shown in Figure 49.

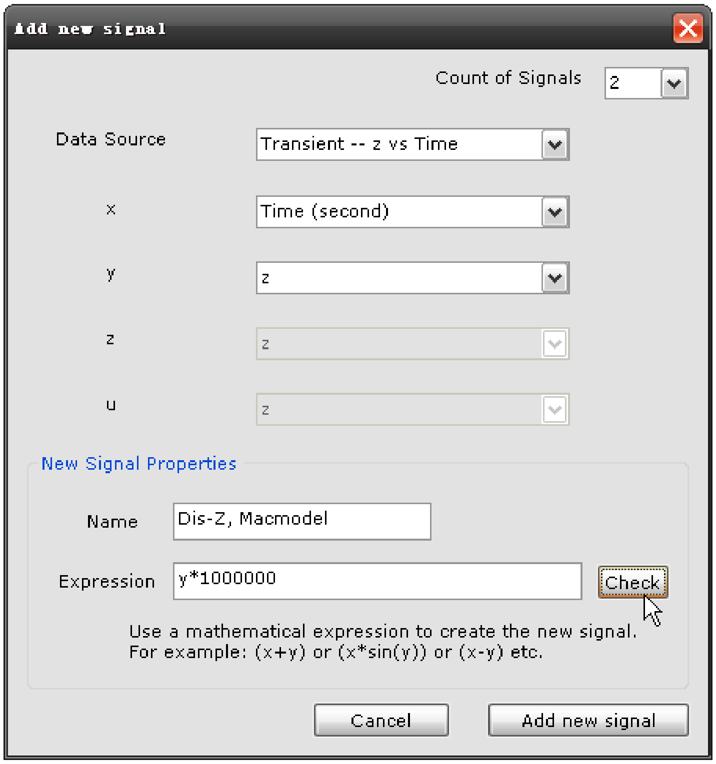


Figure 49 Adding new signal

Click on **Check** to verify if the expression exists in the database. If the expression exists, the button of **Add new signal** will be enabled. Click on **Add new signal** to add the new signal to the data source (Figure 50).

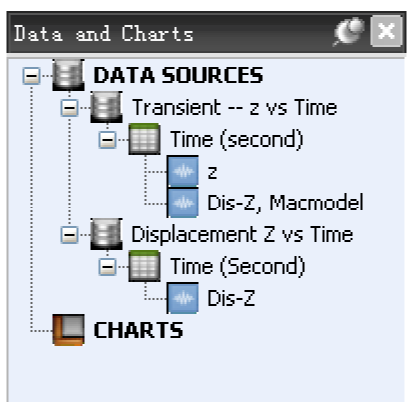


Figure 50 Data sources and charts

Click *Chart ... Add Chart*

A chart template dialog box will open, letting the user select charts for plotting in the 2D plot area of the main window. Specify the settings as shown in Figure 51.

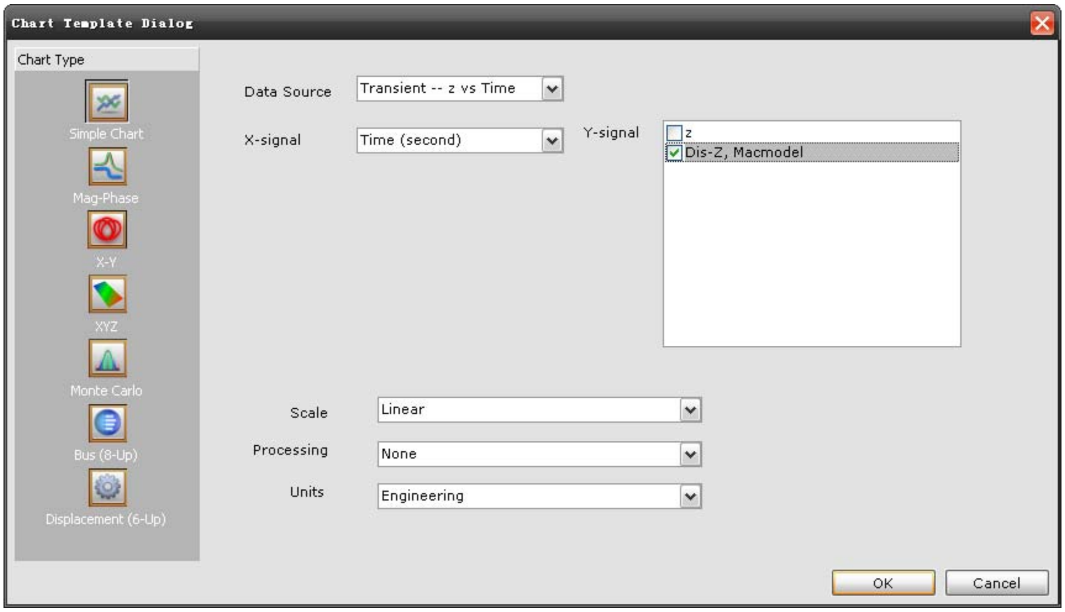


Figure 51 Chart template dialog box

Click on **OK** to plot the curve (Figure 52).

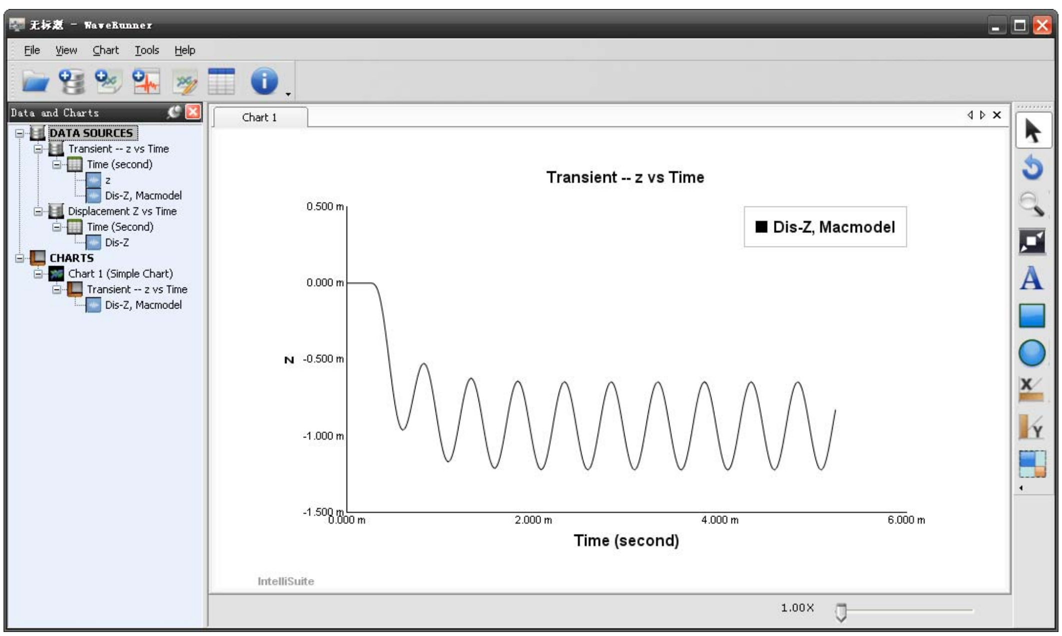


Figure 52 Plotting curve

To add the signal of *Dis-Z* to the chart, the user can drag the signal from the Data and Charts panel to the 2D plot area. Figure 53 shows the plot of the two signals.

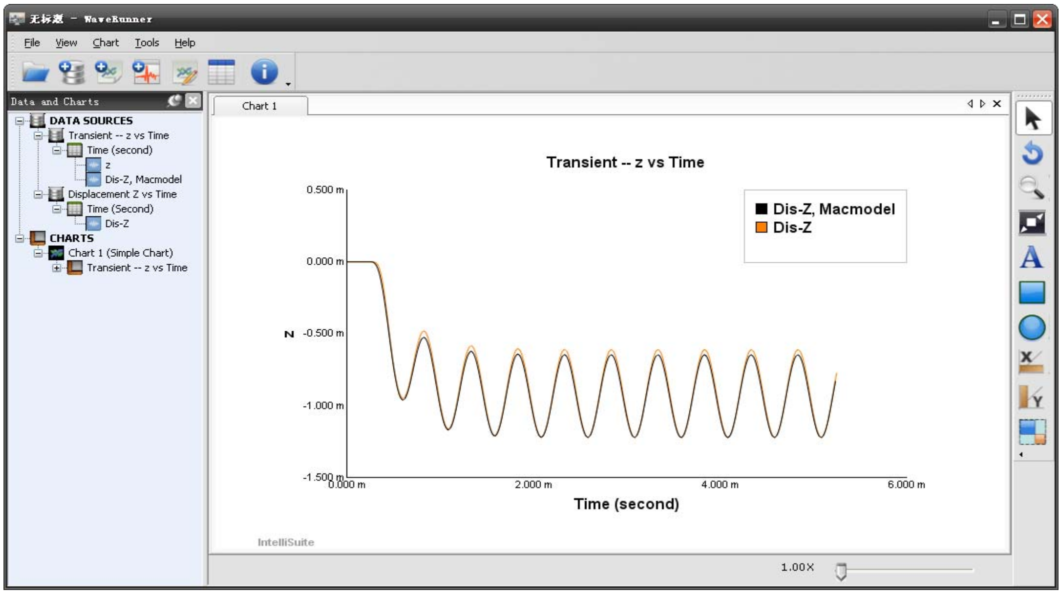


Figure 53 Plot of two signals

Modify the text (titles, labels, legends, etc.) on the chart, as shown in Figure 54.

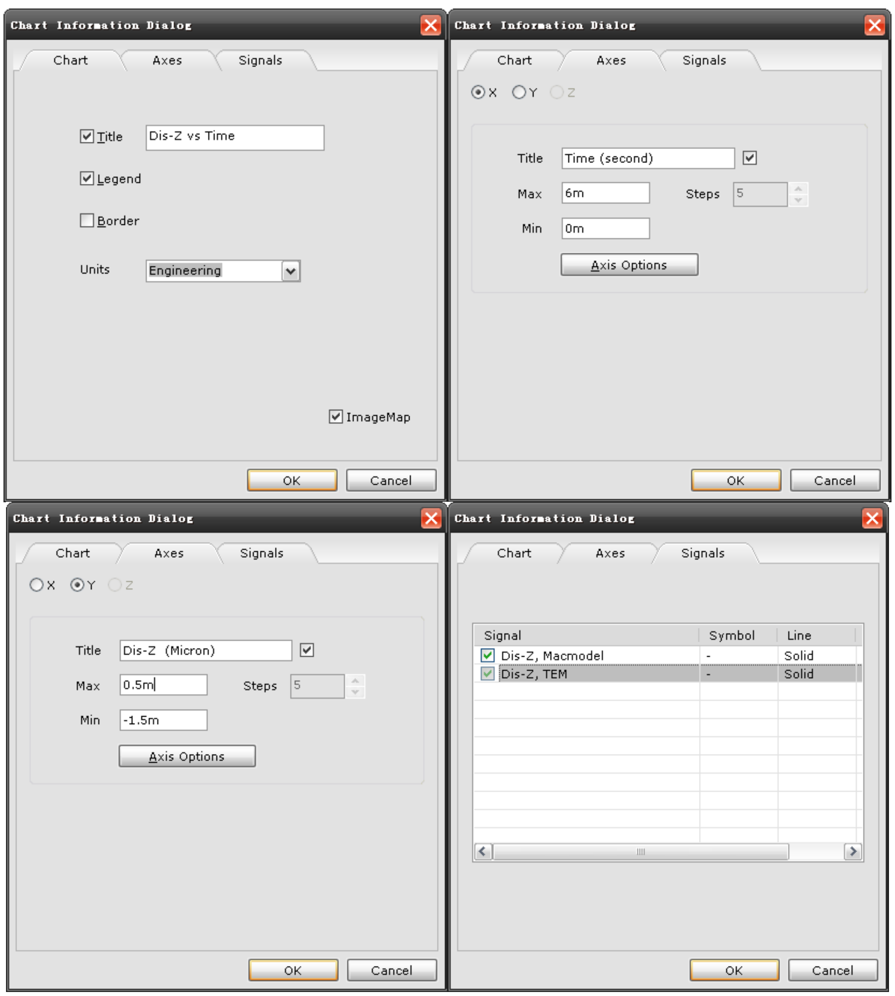


Figure 54 Editing text on chart

Figure 55 shows the final chart of the simulation results obtained using macromodel analysis and finite element analysis in SYNPLE and TEM, respectively.

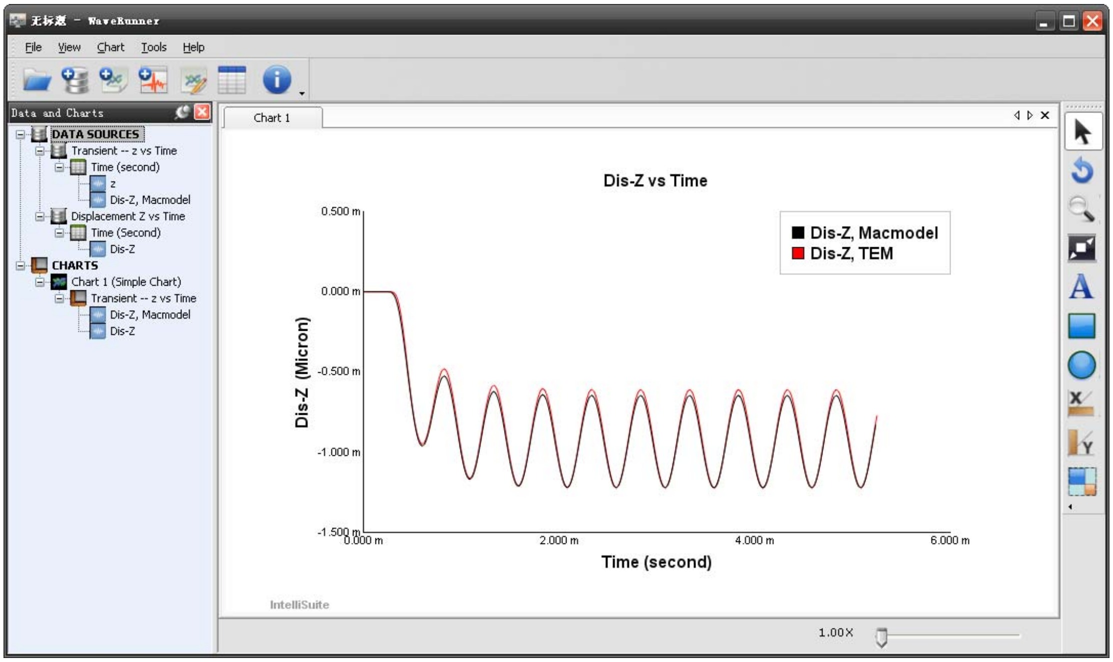


Figure 55 Macromodel analysis and finite element analysis results

As can be seen, the simulation results from the SYNPLE macromodel analysis and the TEM finite element analysis are very close.

3.2.4 Element-based system-level model analysis in SYNPLE

3.2.4.1 Open the element-based schematic model file

Click *Start ... Programs ... IntelliSuite ... SYNPLE*

A window of the SYNPLE module will open. Because a `.ssc` schematic model file has been created for the parallel plate system,

Click *File ... Open.*

From the folder of `IntelliSuite\Training\Microfluidic\SQFD\element_model`, select the file of `plate.ssc`.

Figure 56 shows the schematic model.

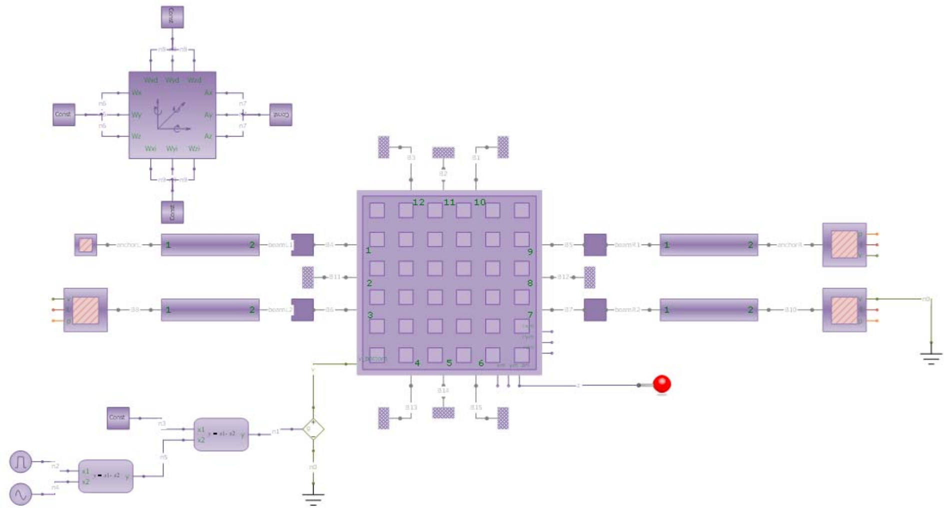


Figure 56 Schematic of parallel plates

3.2.4.2 Edit material properties

Double click on the rigid plate gap element. Make sure the material of plate is set to polysilicon. But the beams will be defined as a new material.

Click *Tools ... Material Property Manager*

Click *Material ...* 

Type `SQFD_beam` as the material name, and set the Young's modulus to 1 GPa and the Poisson ratio to 0.22, as shown in Figure 57. Then click on **Close**. Double click on the other three beams and set their materials to `SQFD_beam`.

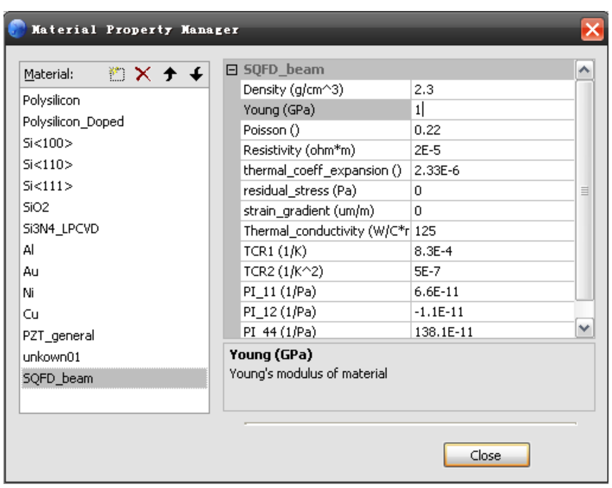


Figure 57 Material properties of beams

3.2.4.3 Set ambient parameters

Click *Tools ... Ambient Parameter Manager*

Set the pressure is 1k, and the viscosity to 76u.

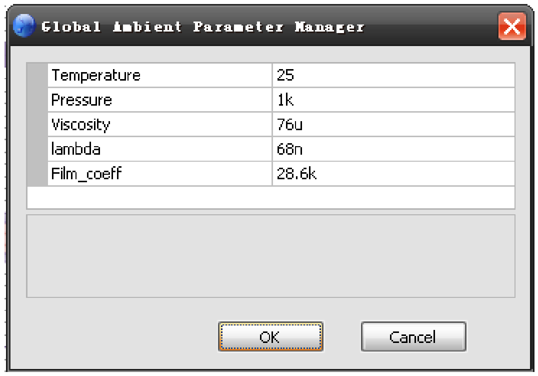


Figure 58 Ambient parameter settings

In TEM, the effective viscosity is set by the user directly, e.g., $\eta_{eff} = 20u$ in this case. But in SYNPLE, the user only inputs the parameters of the pressure and viscosity, and the module will automatically calculate the effective viscosity for simulation.

In [5], the effective viscosity is given by

$$\eta_{eff} = \frac{\eta}{1 + f(K_n)} \tag{12}$$

where $K_n = \lambda / d$ is the Knudsen number and η the viscosity of the air. d is the gap between the parallel plates. λ is the molecular mean free path of the air, which is inversely proportional to the pressure, P.

$$\lambda = \frac{P_0}{P} \lambda_0 \tag{13}$$

where λ_0 is the molecular mean free path of the air under a reference pressure P_0.

Here, we use Veijola's equation to approximate the effective viscosity.

$$\eta_{eff} = \frac{\eta}{1 + 9.638 K_n^{1.159}} \tag{14}$$

In this case, $\lambda_0 = 68n$. η_{eff} is 20u, the same as in the TEM simulation. Therefore, η is 76u by calculation using Eqn. (14).

3.2.4.4 Set up simulation and run analysis

Click *Simulation … Transient Analysis*

Set the parameters under the tabs Simulation Setup, Signals, Schemes, and Convergence Setup as shown in Figure 59.

Click on **OK** to start analysis.

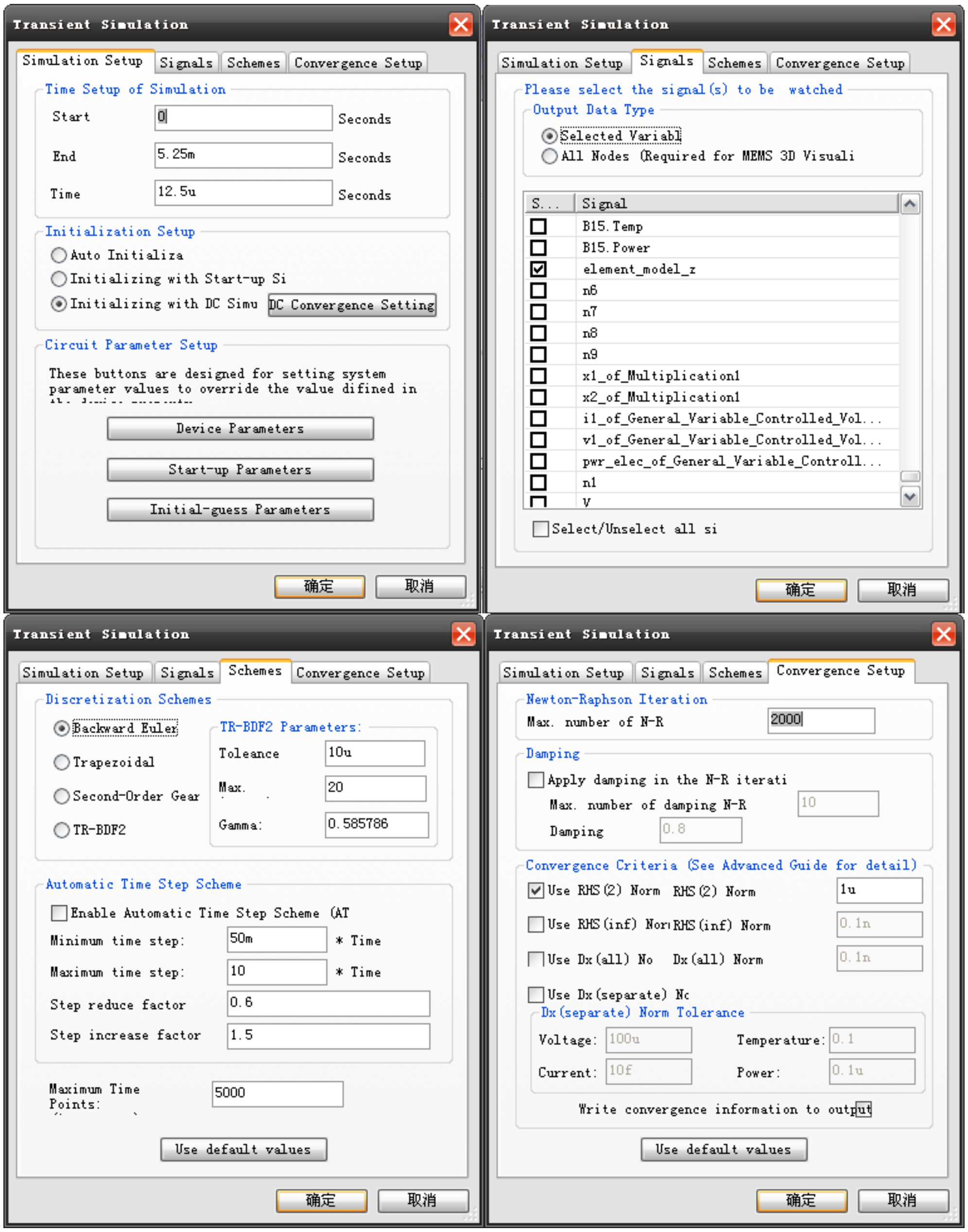


Figure 59 Transient simulation settings

3.2.4.5 View simulation results and save plots

When the analysis finishes, a Plot Manager window will pop up as shown in Figure 60. Double click on the signal of `element_model_z`, the displacement at the center point on the top plate will be plotted as shown in figure 61.

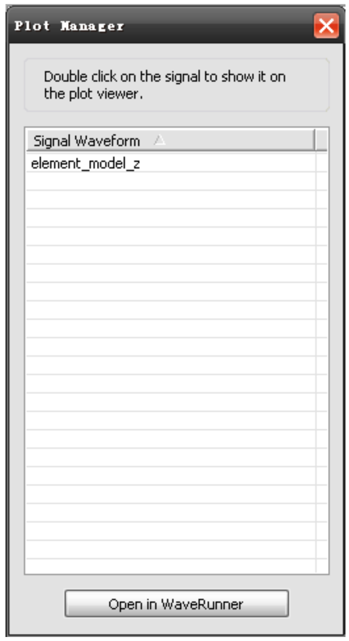


Figure 60 Plot manager

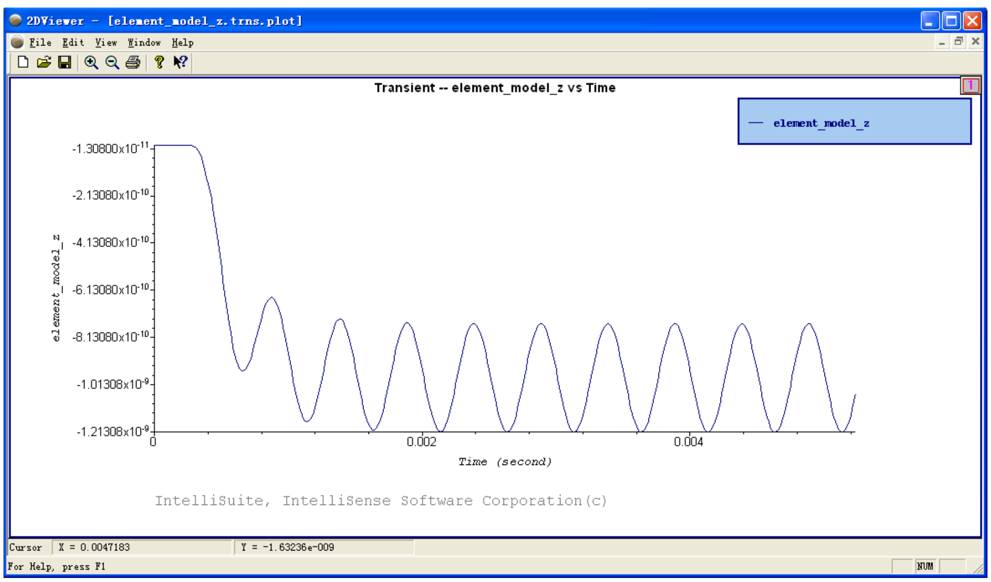


Figure 61 Z-displacement at the center point of top plate

To save the plot, on 2DViewer,

Click *File … Save As.*

Then save the plot as `z_element.plot`.

3.2.5 Comparison of simulation results in WaveRunner

Repeat the procedure as described in section 3.2.3 to open WaveRunner, import data sources, add signals and add charts, and we can compare the z-displacement results at the representative node obtained from the macromodel simulation and the results at the center point on the top plate in WaveRunner, as shown in Figure 62.

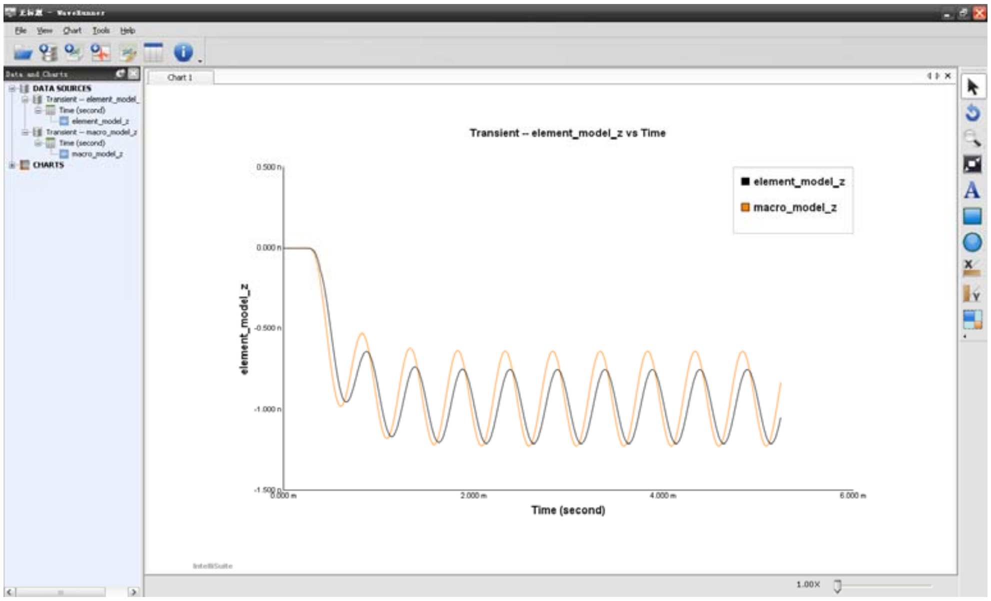


Figure 62 Macromodel analysis and element-based model analysis results

As can be seen again, the simulation results from the SYNPLE macromodel analysis and the element-based system-level model analysis are close. The biggest z-displacement difference is about 10%.

Therefore, based on the TEM FEA simulation results, the SYNPLE macromodel simulation results and the SYNPLE element-based system-level model simulation results, we can draw a conclusion that the macromodel extraction of fluid damping in TEM is efficient and accurate.

References

[1] Gabby, L., "Computer Aided Macromodeling for MEMS", Doctor of Philosophy Thesis, Department of Electrical Engineering and Computer Science, Massachusetts Institute of Technology, Boston MA, June 1998

[2] J. Mehner, L. Gabby and S. D. Senturia, "Computer-aided Generation of Nonlinear Reduced-order Dynamic Macromodels", *J. Microelectromech. Syst.*, Vol. 9, June 2000, pp. 262-278

[3] Jan E. Mehner, Wolfram Doetzel, Bernd Schauwecker and Dale Ostergaard, "Reduced Order Modeling of Fluid Structural Interactions in MEMS based on modal projection techniques", *Transducers'03*, pp 1840-1843 vol. 2, 2003

[4] J. J. Blech, "On isothermal squeeze films", *Journal of Lubrication Technology*, Vol. 105, pp. 615-620, 1983

[5] T. Veijola, H. Kuisma, J. Lahdenpera and T. Ryhänen, "Equivalent-circuit model of the squeezed gas film in a siliconaccelerometer", *Sensors and Actuators A: Physical*, 1995, A48:239-248

IntelliSense

Total MEMS Solutions

www.intellisense.com