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

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

Printed in the United States of America

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

Restricted Rights Legend

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

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

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

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

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

Table of Contents

1. Intro	. Introduction 1					
2. Macr	omodel	extraction	4			
2.1.	Open the model file					
2.2. Simul		te modal contributions				
	2.2.1	Set up simulation	4			
	2.2.2	Edit material properties	5			
	2.2.3	Apply loads and boundary conditions	6			
	2.2.4	Run analysis	9			
	2.2.5	View simulation results	9			
2.3.	10					
	2.3.1	Set up simulation	10			
	2.3.2	Run analysis	11			
	2.3.3	View simulation results	11			
2.4.	Calculat	te capacitance (electrostatic energy)	12			
	2.4.1	Set up simulation	12			
	2.4.2	Run analysis	12			
	2.4.3	View simulation results	12			
2.5.	Calculat	te squeezed film damping (SQFD)	13			
	2.5.1	Set up simulation	13			
	2.5.2	Run analysis	14			
	2.5.3	View simulation results	15			
3. Macr	omodel	validation	17			
3.1.	Dampin	ng force validation	17			
3.2.	Transient analysis validation					

3.2.1	Macrom	nodel analysis in SYNPLE	18
	3.2.1.1	Open the macromodel file	18
	3.2.1.2	Load the macromodel	19
	3.2.1.3	Set up simulation and run analysis	19
	3.2.1.4	View simulation results and save plots	20
3.2.2	Finite el	ement analysis in TEM	22
	3.2.2.1	Open the finite element model file	22
	3.2.2.2	Set up simulation	23
	3.2.2.3	Edit material properties	24
	3.2.2.4	Apply loads and boundary conditions	25
	3.2.2.5	Run analysis	29
	3.2.2.6	View simulation results and save plots	29
3.2.3	Compar	ison of simulation results in WaveRunner	31
3.2.4	Element	t-based system-level model analysis in SYNPLE	22
	3.2.4.1	Open the element-based schematic model file	22
	3.2.4.2	Edit material properties	23
	3.2.4.3	Set ambient parameters	24
	3.2.4.4	Set up simulation andrun analysis	25
	3.2.4.5	View simulation results and save plots	29
3.2.5	Compar	ison of simulation results in WaveRunner	31
 			40

References

40

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.

$$\Psi_{\text{ext}}(\mathbf{x}, \mathbf{y}, \mathbf{z}, \mathbf{t}) = \Psi_{\text{initial}}(\mathbf{x}, \mathbf{y}, \mathbf{z}) + \sum q_i(\mathbf{t}) \cdot \Psi_i(\mathbf{x}, \mathbf{y}, \mathbf{z})$$
(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), ψ_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 ψ_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.

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

$$F_{\text{SQFD}} = F(\Psi_{\text{ext}}) = F(\Psi_{\text{initial}}, q_1, q_2, \dots, \psi_1, \psi_{2,\dots}, \partial q_1 / \partial t, \partial q_2 / \partial t, \dots)$$
(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.

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

$$\langle \mathbf{F}_{i}, \boldsymbol{\psi}_{i} \rangle = \mathbf{C}_{i} \partial \mathbf{q}_{i} / \partial \mathbf{t} + \mathbf{K}_{i} \mathbf{q}_{i}$$
 (5)

where $\langle F_i, \psi_i \rangle$ 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.

$$C_i = C_i(q_i, \omega), \quad K_i = K_i(q_i, \omega)$$
(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.

$$C_i = C_i(q_i=0, \omega) = C_i(\omega), \qquad K_i = K_i(q_i=0, \omega) = K_i(\omega)$$
(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.

Simulation Setting	
Calculation Type Static Frequency Dynamic Macro Model Extraction Analysis Type Rigid Body Variables Spring Constants Squeezed Film Damping Variables Capacitance Capacitance Capacitance vs Displacement Mechanical Reduced Order Modelling ElectroMechanical Reduced Order Modelling Thermal Electrical/Thermal Stress ElectroMechanical + Fluidic Dissipation	Option Frequency Modes Number 3 Displacement Start Shape Undeformed Previously Deformed Convergence Definition Reration Number 10 Reration Accuracy 0.001 Macro Model Extraction Freid Contribution Freid Contribu
	Apply OK Cancel

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.

Entity Description:			Young	g and Poisson Ratio	Isotropic	*
Property	Unit	Value				
Density	g/cm^3	2.3				
Elastic Parameter	#	Isotropic				
Stress/Stress Gradient	MPa	Constant				
Thermal Expansion Coeff	1E-7/C	Constant		Young, GPa	160	
Thermal Conductivity	W/cm/C	Constant		Poisson Ratio, #	0.226	
Specific Heat	J/g/C	0.71				
Dielectric Constant	#	Isotropic				
Resistivity	ohm.cm	Constant				
Piezoresistive Coeff	1/MPa	None				
Piezoelectric Coeff	#	None				
Orientation	#	Default				

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.

aterial Properties					
Entity Number: 2 Entity Description:			Young and Poisson Ratio	Isotropic 💌	
Property	Unit	Value			
Density	g/cm^3	2.3			
Elastic Parameter	#	Isotropic			
Stress/Stress Gradient	MPa	Constant			
Thermal Expansion Coeff	1E-7/C	Constant	Young, GPa	1	
Thermal Conductivity	W/cm/C	Constant	Poisson Ratio, #	0.226	
Specific Heat	J/g/C	0.71			
Dielectric Constant	#	Isotropic			
Resistivity	ohm.cm	Constant			
Piezoresistive Coeff	1/MPa	None			
Piezoelectric Coeff	#	None			
Orientation	#	Default			
Import	Export			OK Cancel	

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).

Dislog 🔀
Please Input Voltage Below
Enter by
Single Value Input
Voltage 1. Volt
OK Carrel
Cancer

Figure 6 Voltage on top plate

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

Dialog	_	-	_	-	_	×
Ple	ase Input Voltage	Below				
	Enter by	Single Value	Imput			
	Single Value	Input				
	Voltagi	e	0.	Volt		
			OK		Cancel	

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.

Dialog			_	×
Squeezed Film Face Pair Definition				
	Pair ID			
	Viscosity		N×Shn^2	
Ambi	ent Pressure	0.001	MPa	
	Fluid Medium			
	🔿 Liqu	nid	🖲 Gas	
	Count	Face ID	Option	
	1	27	Enabled	
Moving Face				
	<			>
	Count	Face ID	Ontion	
Fixed	1	2	Enabled	
Face				
	<		1111	>
[Accept		DK Ca	ncel
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).

Dialog	<
Representive Node is selected at Node 880 Please Provide a Representive Name	
Node 880	
0K Cancel	

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.

Dialo	DE		_	X
	Mode	Contribution	Option Enabled	
	3	7.33939e-008 9.1056e-008	Disabled Disabled	
	<		Ш	>
		OK	Cancel	

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).

Animation Dialog 🔀				
Please input mode number				
1				
Please input scale factor				
5				
OK Cancel				

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).

Simulation Setting	X
Simulation Setting Calculation Type O Static O Frequency D Jynamic O Macro Model Extraction	Option Frequency Modes Number 3 Displacement Small O Large
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 ElectroMechanical + Fluidic Dissipation	Start Shape Undeformed Previously Deformed Convergence Definition Iteration Number I0 Iteration Accuracy 0.001 Macro Model Extraction Modal Contribution Strain Energy vs Modal Amplitudes Mutual Capacitance vs Modal Amplitudes Fluid D amping Coefficient Scaling Factor for Mode Increment Number I0 Mode I w
	Apply OK Cancel

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.

Dialo	E	-	_	X
	Mode 1 2 3	Option Enabled Disabled Disabled	Plot Option Enabled Disabled Disabled	
	<	OK	IIII Cancel	

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).

Capacitance Dialog 🔀	Dialog 🔀
Conductor 1 Color Conductor 2 Color Value (microtarads*1e-6) Entity 1 Entity 1 0.7971 Entity 1 Entity 3 -0.7632 Entity 3 Entity 1 0.7972 Entity 3 Entity 3 0.7927	ModeOptionPlot Option1EnabledEnabled2DisabledDisabled3DisabledDisabled
OK Cancel	Cancel

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).

Simulation Setting	$\mathbf{\Sigma}$
Calculation Type Static Frequency Dynamic Macro Model Extraction Analysis Type Rigid Body Variables Spring Constants Squeezed Film D amping Variables Capacitance Capacitance Capacitance vs Displacement Mechanical Reduced Order Modelling ElectroMechanical Reduced Order Modelling Thermal Electrical/Thermal Stress ElectroMechanical + Fluidic Dissipation	Option Frequency Modes Number 3 Displacement Image Start Shape Image Start Shape Image Start Shape Image Image Start Shape Image Image Start Shape Image Imag
	Apply OK Cancel

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.

Fluid Damping Extraction S	etting 🔀
Frequency Range(Hz)	
Starting Frequency	100
Ending Frequency	100000
Number of Frequencies	20
Simulation Points per Period	100
Number of Periods	2
🗌 Add to Existi	ng
ОК	Cancel

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.

Damping and Spring For	ces 🔀
Modal Based Fo	rce 🔿 Common Force
Exciting Mode	1 💌
Projected Mode	1
Scaling Factor	
Plot	OK Cancel

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).

Damping and Spring Ford	ces 🔀
O Modal Based Fo	rce 💿 Common Force
Mode_l	1
SQUFilm Pair ID	1
Scaling Factor	~
Component	normal compone 🖌
Plot	OK Cancel

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.

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

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

where $C(\Omega)$ is the frequency dependent damping coefficient; $K(\Omega)$ 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

$$F_{vis} = C\Omega \tag{11}$$

$$F_{sor} = K$$

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
 $\eta = 2e-5$ 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_Model.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.



IntelliSuit

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.



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.

Transient Simulation 🛛 🔀	Transient Simulation
Simulation Setup Signals Schemes Convergence Setup	Simulation Setup Signals Schemes Convergence Setup
Time Setup of Simulation	Please select the signal(s) to be watched
Start 🛛	Output Data Type
End 5.25m Seconds	All Nodes (Required for MEMS 3D Visuali
Time 12.5u Seconds	S Signal
Initialization Setup	✓ q1_of_fdtest
🔿 Auto Initializa	a of fdtest
○Initializing with Start-up Si	□ q4 of fdtest
● Initializing with DC Simu DC Convergence Setting	g5_of_fdtest
Circuit Peremeter Setun	□ q6_of_fdtest
offetter a aneces becap	□ q7_of_fdtest
These buttons are designed for setting system	☐ q8_of_fdtest
it. J	☐ q9_of_fdtest
Device Parameters	qlU_of_fdtest
	Curl_of_fdtest
Start-up Parameters	
Initial-guess Parameters	
	Select/Unselect all S1
一 确定	一 确定 取消
Transient Simulation 🔀	Transient Simulation 🛛 🔀
Transient Simulation X	Transient Simulation
Transient Simulation X Simulation Setup Signals Schemes Convergence Setup Discretization Schemes	Transient Simulation
Transient Simulation X Simulation Setup Signals Schemes Convergence Setup Discretization Schemes OBackward Euler TR-BDF2 Parameters:	Transient Simulation Simulation Setup Signals Schemes Convergence Setup Newton-Raphson Iteration Max. number of N-R 2000
Transient Simulation X Simulation Setup Signals Schemes Convergence Setup Discretization Schemes Backward Euler Trapezoidal TR-BDF2 Parameters: Toleance 10u	Transient Simulation X Simulation Setup Signals Newton-Raphson Iteration Max. number of N-R Damping Description
Transient Simulation Simulation Setup Signals Schemes Convergence Setup Discretization Schemes Backward Buler, TR-BDF2 Parameters: Toleance 10u Second-Order Gear Max. 20	Transient Simulation X Simulation Setup Signals Schemes Convergence Setup Newton-Raphson Iteration Max. number of N-R Damping Apply damping in the N-R iterati
Transient Simulation Simulation Setup Signals Schemes Convergence Setup Discretization Schemes Discretization Schemes TR-BDF2 Parameters: Toleance 10u Second-Order Gear TR-BDF2 Gamma: 0.585786	Transient Simulation Simulation Setup Signals Schemes Convergence Setup Newton-Raphson Iteration Max. number of N-R Damping Apply damping in the N-R iterati Max. number of damping N-R Desping
Transient Simulation Simulation Setup Signals Schemes Convergence Setup Discretization Schemes	Transient Simulation Simulation Setup Signals Schemes Convergence Setup Newton-Raphson Iteration Max. number of N-R Damping Apply damping in the N-R iterati Max. number of damping N-R Damping 0.8
Transient Simulation Simulation Setup Signals Schemes Convergence Setup Discretization Schemes • Backward Euler TR-BDF2 Parameters: • Trapezoidal Second-Order Gear • TR-BDF2 TR-BDF2 • Automatic Time Step Scheme • Scheme	Transient Simulation Simulation Setup Signals Schemes Convergence Setup Newton-Raphson Iteration Max. number of N-R Damping Apply damping in the N-R iterati Max. number of damping N-R Damping Convergence Criteria (See Advanced Guide for detail) V Use RHS (2) Norm
Transient Simulation Simulation Setup Signals Schemes Convergence Setup Discretization Schemes • Backward Euler TR-BDF2 Parameters: • Trapezoidal Second-Order Gear • TR-BDF2 • Automatic Time Step Scheme Enable Automatic Time Step Scheme (AT	Transient Simulation Simulation Setup Signals Schemes Convergence Setup Newton-Raphon Iteration Max. number of N-R Damping Apply damping in the N-R iterati Max. number of damping N-R Damping Convergence Criteria (See Advanced Guide for detail) V Use RHS (2) Norm RHS (2) Norm Iu Use RHS (inf) Norr PHS (inf) Norm
Transient Simulation Simulation Setup Signals Schemes Convergence Setup Discretization Schemes Backward Euler Trapezoidal Second-Order Gear TR-BDF2 Automatic Time Step Scheme Enable Automatic Time Step Scheme (AT Minimum time step: 50m * Time	Transient Simulation Simulation Setup Signals Schemes Convergence Setup Newton-Raphson Iteration Max. number of N-R Damping Apply damping in the N-R iterati Max. number of damping N-R Damping Convergence Criteria (See Advanced Guide for detail) V Use RHS (2) Norm RHS (2) Norm Iu Use RHS (inf) Norn RHS (inf) Norm Use RHS (inf) Norn RHS (inf) Norm
Transient Simulation Simulation Setup Signals Schemes Convergence Setup Discretization Schemes Discretization Schemes Discretization Schemes TR-BDF2 Parameters: Toleance 10u Max. 20 Gamma: 0.585786 Automatic Time Step Scheme Enable Automatic Time Step Scheme (AT Minimum time step: 50m * Time Maximum time step: 10 * Time	Transient Simulation Simulation Setup Signals Schemes Convergence Setup Newton-Raphson Iteration Max. number of N-R Damping Apply damping in the N-R iterati Max. number of damping N-R Damping O.8 Convergence Criteria (See Advanced Guide for detail) V Use RHS (2) Norm RHS (2) Norm Use RHS (inf) NorrRHS (inf) Norm Use Dx (all) No Dx (all) Norm
Transient Simulation Simulation Setup Signals Schemes Convergence Setup Discretization Schemes Discretization Schemes Discretization Schemes TR-BDF2 Parameters: Toleance 10u Max. 20 Gamma: 0.585786 Automatic Time Step Scheme Enable Automatic Time Step Scheme (AT Minimum time step: 50m * Time Maximum time step: 10 * Time Step reduce factor 0.6	Transient Simulation Simulation Setup Signals Schemes Convergence Setup Newton-Raphson Iteration Max. number of N-R Damping Apply damping in the N-R iterati Max. number of damping N-R Damping 0.8 Convergence Criteria (See Advanced Guide for detail) V Use RHS (2) Norm RHS (2) Norm Use RHS (inf) NorrRHS (inf) Norm Use Dx (all) No Use Dx (separate) N: Use Dx (separate) N:
Transient Simulation Simulation Setup Signals Schemes Convergence Setup Discretization Schemes Discretization Schemes Discretization Schemes TR-BDF2 Parameters: Toleance 10u Max. 20 Gamma: 0.585786 Automatic Time Step Scheme Enable Automatic Time Step Scheme (AT Minimum time step: 50m * Time Maximum time step: 10 * Time Step reduce factor 0.6 Step increase factor 1.5	Transient Simulation Simulation Setup Signals Schemes Convergence Setup Newton-Raphson Iteration Max. number of N-R Damping Apply damping in the N-R iterati Max. number of damping N-R Damping O.8 Convergence Criteria (See Advanced Guide for detail) V Use RHS (2) Norm RHS (2) Norm Use RHS (inf) NortRHS (inf) Norm Use Dx (all) No Dx (separate) Norm Tolerance Valtage: Valtage:
Transient Simulation Simulation Setup Signals Schemes Convergence Setup Discretization Schemes • Backward Euler TR-BDF2 Parameters: Toleance 10u O Trapezoidal Second-Order Gear Max. 20 20 O TR-BDF2 Gamma: 0.585786 Automatic Time Step Scheme Second - Step Scheme 10 Brable Automatic Time Step Scheme 10 * Time Maximum time step: 50m * Time Step reduce factor 0.6 5 Step increase factor 1.5 5	Transient Simulation Simulation Setup Signals Schemes Convergence Setup Newton-Raphson Iteration Max. number of N-R Damping Apply damping in the N-R iterati Max. number of damping N-R Damping O.8 Convergence Criteria (See Advanced Guide for detail) V Use RHS (2) Norm RHS (2) Norm Use RHS (inf) NornRHS (inf) Norm Use Dx (separate) Nor Tolerance Voltage: 100u Theorem Tournet: Converging 0.1n Vse Dx (all) Nor Tolerance Voltage: 100u Temperature: 0.1
Transient Simulation Simulation Setup Signals Schemes Convergence Setup Discretization Schemes	Transient Simulation Simulation Setup Signals Schemes Convergence Setup Newton-Raphson Iteration Max. number of N-R Damping Apply damping in the N-R iterati Max. number of damping N-R Damping O.8 Convergence Criteria See Advanced Guide for detail) V Use RHS (2) Norm RHS (2) Norm Use RHS (inf) Norn RHS (inf) Norm Use Dx (separate) Nc Dx (separate) Nc Voltage: 100u Temperature: 0.1u
Transient Simulation	Transient Simulation Simulation Setup Signals Schemes Convergence Setup Newton-Raphon Iteration Max. number of N-R Damping Apply damping in the N-R iterati Max. number of damping N-R Damping 0.8 Convergence Criteria See Advanced Guide for detail) V Use RHS (2) Norm RHS (2) Norm Use RHS (inf) NorrRHS (inf) Norm 0.1n Use Dx (all) No Dx (separate) Nc Dx (separate) Nc Voltage: 100u Write convergence information to outrue
Transient Simulation Simulation Setup Signals Schemes Convergence Setup Discretization Schemes Discretization Schemes Discretization Schemes TR-BDF2 Toleance 10u Max. 20 Gamma: 0.585786 Automatic Time Step Scheme Camma: 0.585786 Automatic Time Step Scheme Step reduce factor 0.6 Step increase factor 1.5 Maximum Time Points: Use default values	Transient Simulation Simulation Setup Signals Schemes Convergence Setup Newton-Raphon Iteration Max. number of N-R Damping Apply damping in the N-R iterati Max. number of damping N-R Damping 0.8 Convergence Criteria (See Advanced Guide for detail) V Use RHS (2) Norm RHS (2) Norm Use EMS (inf) NortRHS (inf) Norm Use Dx (all) No Dx (separate) Nc Dx (separate) Nc Voltage: 100u Write convergence information to outroits Vise default values

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.

?lo	t Manager 🔀
	Double click on the signal to show it on the plot viewer.
Si	gnal Waveform 🛆
n7	,
qt	_of_fdtest
q2	2_of_fdtest
q3	3_of_fdtest
-	
Ŀ	
Ŀ	
-	
Ŀ-	
E.	
	Open in WaveRunner

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.

Calculation Type Option Static Frequency Dynamic Bistory Macro Model Extraction Small Analysis Type Stress/Disp. (Direct Integration) Stress/Disp. (Mode Based) Undeformed Heat Transfer Transient Iteration Number Stress/Disp./Squeezed Film (Direct Integration) Iteration Accuracy Stress/Disp./Squeezed Film (Direct Integration) Fixed Stress/Disp./Electrostatic (Mode Based) Transient (Auto Time Increment) Stress/Disp./Electrostatic/Squeezed Film (Direct Integration) Totasient (Auto Time Increment)	Simulation Setting	
Stress/Disp./Squeezed Film (Direct Integration) Stress/Disp./Electrostatic (Direct Integration) Stress/Disp./Electrostatic (Mode Based) Stress/Disp./Electrostatic/Squeezed Film (Direct Integration) Stress/Disp./Electrostatic/Squeezed Film (Direct Integration)	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	Option Result History Last State Displacement Small Large Start Shape O Undeformed Previously Deformed Convergence Definition Iteration Number 10 Iteration Accuracy 0.001 Dynamic Capacitance Option Fixed
	Stress/Disp./Squeezed Film (Direct Integration) Stress/Disp./Electrostatic (Direct Integration) Stress/Disp./Electrostatic (Mode Based) Stress/Disp./Electrostatic/Squeezed Film (Direct Integration)	Dynamic Transient (Auto Time Increment) Transient (Fixed Time Increment) Time Period (Second) 0.00525 Increment Number (<1000) 420

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.

Entity Description:			Young	ı and Poisson Ratio	Isotropic	~
Property	Unit	Value				
Density	g/cm^3	2.3				
Elastic Parameter	#	Isotropic				
Stress/Stress Gradient	MPa	Constant				
Thermal Expansion Coeff	1E-7/C	Constant		Young, GPa	160	
Thermal Conductivity	W/cm/C	Constant		Poisson Ratio, #	0.226	
Specific Heat	J/g/C	0.71				
Dielectric Constant	#	Isotropic				
Resistivity	ohm.cm	Constant				
Piezoresistive Coeff	1/MPa	None				
Piezoelectric Coeff	#	None				
Orientation	#	Default				

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.

Entity Description:			Young and Poisson Ratio	Isotropic 💌
Property	Unit	Value		
Density	g/cm^3	2.3		
Elastic Parameter	#	Isotropic		
Stress/Stress Gradient	MPa	Constant		
Thermal Expansion Coeff	1E-7/C	Constant	Young, GPa	1
Thermal Conductivity	W/cm/C	Constant	Poisson Ratio, #	0.226
Specific Heat	J/g/C	0.71		
Dielectric Constant	#	Isotropic		
Resistivity	ohm.cm	Constant		
Piezoresistive Coeff	1/MPa	None		
Piezoelectric Coeff	#	None		
Orientation	#	Default		

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.

Check	/Modify Material	Property	_	×
	P .		TT 1	
	Property	Umit Umit	Vatue	
	Mass_damping	1/second	0.0	
	Sourcess_damping	second	0.0	
	•			/
	E dit Property			
			OK	Cancel

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).

Amplit	nde Definition		$\mathbf{\overline{N}}$			
	Entity 3 is Selected					
	Define Applied Voltage Curve					
	A=A0+A1×cosW(t+t0)+B1sinW(t+t0) for t>=t0 A=A0 for t <t0< th=""></t0<>					
	t0	0.00025	sec			
	W	6280	rad/sec			
	A0	0	Volt			
	Α1	0	Volt			
	B1	1	Volt			
	OK		Cancel			

Figure 38 Voltage on top plate

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

Amp	litude Definition			×			
	Entity 1 is Selected	Entity 1 is Selected					
	Define Applied Voltage Cu	Define Applied Voltage Curve					
	A=A0+A1 [×] cosW(t+0)+B A=A0 for t <t0< td=""><td colspan="6">A=A0+A1*cosW(t+0)+B1sinW(t+0) for t>=t0 A=A0 for t<t0< td=""></t0<></td></t0<>	A=A0+A1*cosW(t+0)+B1sinW(t+0) for t>=t0 A=A0 for t <t0< td=""></t0<>					
	t0	0	sec				
	W	0	rad/sec				
	AO	0	Volt				
	A1	0	Volt				
	B1	0	Volt				
	OK		Cancel				

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



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



Figure 42 Moving face of squeezed film pair



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.

Dialog	_	_		X
:	Gqueezed Film	Face Pair Definiti	on	
	Pair ID	1]	
7	Viscosity		N×Shn^2	
Ambie	mt Pressure	0.001	0.001 MPa	
	The 2 h V - 21			
L C	Fluid Medium			
	🔿 Lig	uid	💿 Gas	
	Count	Face ID	Option	
	1	27	Enabled	
Moving Face				
	<		Ш	>
	Count	Face ID	Option	
Fixed Face	1	2	Enabled	
	<			
	<u></u>			
[Accept	0	K Cancel	

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 μ m, 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.

Add new signal	
	Count of Signals 2
Data Source	Transient z vs Time 💌
×	Time (second)
У	z
z	z 🗸
u	z
New Signal Pro	perties
Name	Dis-Z, Macmodel
Expression	y*1000000
Use For e	mathematical expression to create the new signal. xample: (x+y) or (x*sin(y)) or (x-y) etc.
	Cancel Add new signal

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.

Chart Template Dialog		×
Chart Type		
×	Data Source	Transient z vs Time
Simple Chart	X-signal	Time (second) Y-signal Z Dis-Z, Macmodel
Mag-mase X-Y		
XYZ		
	Scale	Linear
Bus (8-Up)	Processing	None
	Units	Engineering
Displacement (6-Up)		
		OK Cancel

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.

Chart Information Dialog 🔀	Chart Information Dialog 🛛 🔀
Chart Axes Signals	Chart Axes Signals
☑ Ittle Dis-Z vs Time	
✓ Legend	Title Time (second)
Border	Max 6m Steps 5
	Min Om
Units Engineering	<u>Axis Options</u>
✓ ImageMap	
OK Cancel	OK Cancel
Chart Information Dialog	Chart Information Dialog
Chart Axes Signals	Chart Axes Signals
Title Dis-Z (Micron)	Signal Symbol Line
Max 0.5m Steps 5	V Dis-Z, TEM - Solid
Min -1.5m	
Axis Options	
	< >
OK Cancel	OK Cancel

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.

Material: 👘 📉 🛧 🛧	↓ SQFD_beam		^
Delucilicop	Density (g/cm^3)	2.3	
Polysiicon Debuiliese Deeed	Young (GPa)	1	
Polysilicon_Doped	Poisson ()	0.22	
5<100>	Resistivity (ohm*m)	2E-5	
Si<110>	thermal_coeff_expansion ()	2.33E-6	
Si<111>	residual_stress (Pa)	0	=
SiO2	strain_gradient (um/m)	0	
Si3N4_LPCVD	Thermal_conductivity (W/C*r	125	
Al	TCR1 (1/K)	8.3E-4	
Au	TCR2 (1/K^2)	5E-7	
Ni	PI_11 (1/Pa)	6.6E-11	
Cu	PI_12 (1/Pa)	-1.1E-11	
PZT general	PI 44 (1/Pa)	138.1E-11	~
unkown01	Young (GPa)		
SOED heam	Young's modulus of material		
54. 5 <u>-</u> 500			

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.

Temperature	25
Pressure	1k
Viscosity	76u
ambda	68n
Film_coeff	28.6k

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, $\lambda_0 = 68n \cdot \eta_{eff}$ 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.

Transient Simulation	X	ransient	Simulation 🔀
Simulation Setup Signals Schemes Conve	ergence Setup	Simulation	Setup Signals Schemes Convergence Setup
Time Setup of Simulation		Please	select the signal(s) to be watched
Start 0 Seconds		Output Data Type	
		⊚Se	lected Variabl
End 5.25m	Seconds	○ Al:	l Nodes (Required for MEMS 3D Visuali
Time 12.5u	Seconds	S	Signal
-Initialization Satur			B15. Temp
Auto Initializa			B15. Power
O Initializing with Start-up Si			element_model_z
Thitislizing with DC Simu DC Converse	rannan Satting		n6
	rgence secting		n7
-Circuit Parameter Setup		H	n8
These buttons are designed for setting	g system	H	x1 of Multiplication1
parameter values to override the value	e difined in	H	x2 of Multiplication1
		H	i1 of General Variable Controlled Vol
Device Parameters			v1_of General Variable Controlled Vol
Struton Providence			pwr_elec_of_General_Variable_Controll
Start-up rarameters			ni
Initial-guess Parameters			v
		Sel	ect/Unselect all si
确定	取消		确定 取消
	Ų		
Transient Simulation	т 🔀	ransient	Simulation 🔀
Transient Simulation	ergence Setun	ransient Simulation	Simulation X
Transient Simulation	ergence Setup	ransient Simulation	Simulation X
Transient Simulation Simulation Setup Signals Schemes Converting Discretization Schemes	ergence Setup	ransient Simulation Newton- May nu	Simulation X Setup Signals Schemes Convergence Setup Raphson Iteration Whar of W-R 2000
Transient Simulation Simulation Setup Signals Schemes Conver Discretization Schemes © Backward Euler TR-BDF2 Paramet	ergence Setup	ransient Simulation Newton- Max. nu	Simulation X Setup Signals Schemes Convergence Setup Raphson Iteration mber of N-R 2000
Transient Simulation Simulation Setup Signals Schemes Conver Discretization Schemes © Backward Euler Trapezoidal Toleance	ergence Setup S ters:	ransient Simulation Newton- Max. nu Damping	Simulation X Setup Signals Schemes Convergence Setup Raphson Iteration mber of N-R 2000
Transient Simulation Simulation Setup Signals Discretization Schemes	ergence Setup	ransient Simulation Newton- Max. nu Damping Appl	Simulation Setup Signals Schemes Convergence Setup Rephson Iteration umber of N-R 2000 y damping in the N-R iterati
Transient Simulation Simulation Setup Signals Discretization Schemes	ergence Setup	ransient Simulation Newton- Max. nu Damping Appl Max.	Simulation Setup Signals Schemes Convergence Setup Raphson Iteration umber of N-R 2000 y damping in the N-R iterati number of damping N-R 10
Transient Simulation Simulation Setup Signals Schemes Convert Discretization Schemes OBackward Euler TR-BDF2 Paramet Toleance Incleance Incleance OTR-BDF2 Gamma: Incleance	ergence Setup S ters: 10u 20 0.585786	ransient Simulation Max. nu Damping Appl Max. Damp	Simulation
Transient Simulation Simulation Setup Signals Schemes Convert Discretization Schemes OBackward Euler TR-BDF2 Paramet Toleance Imax Imax OTrapezoidal Second-Order Gear Imax OTR-BDF2 Gamma: Imax	ters: 10u 20 0.585786	ransient Simulation Max. nu Damping Appl Max. Damp Converge	Simulation
Transient Simulation Simulation Setup Signals Schemes Convergence Discretization Schemes OBackward Euler TR-BDF2 Paramet Toleance Toleance OTR-BDF2 Gamma: Automatic Time Step Scheme The Step Scheme	ters: 10u 20 0.585786	ransient Simulation Newton- Max. nu Damping Appl Max. Damp Max. Damp Y Use	Simulation
Transient Simulation Simulation Setup Signals Schemes Convert Discretization Schemes TR-BDF2 TR-BDF2 Parameter O Trapezoidal Second-Order Gear Max. E O TR-BDF2 Gamma: Gamma: E Automatic Time Step Scheme Enable Automatic Time Step Scheme Second Scheme	ergence Setup S ters: 10u 20 0.585786 (AT (AT	ransient Simulation Newton- Max. nu Damping Appl Max. Damp Vax Vse Vse	Simulation
Transient Simulation Simulation Setup Signals Schemes Converting Discretization Schemes OBackward Euler TR-BDF2 Parameter Trapezoidal Second-Order Gear TR-BDF2 Gamma: Automatic Time Step Scheme Enable Automatic Time Step Scheme Minimum time step: Son	Image: constraint of the second sec	ransient Simulation Max. nu Damping Appl Max. Damp Converg Vise Vise	Simulation
Transient Simulation Simulation Setup Signals Schemes Converting Discretization Schemes	ergence Setup S ters: 10u 10u 20 0.585786 0 (AT * Time * Time * Time	Annsient Simulation Max. nx Damping Appl Max. Damp Converg Vuse Use	Simulation
Transient Simulation Simulation Setup Signals Schemes Converting Schemes	Image: constraint of the second sec	Annsient Simulation Max. nx Damping Appl Max. Damp Converg Vise Use Use	Simulation
Transient Simulation Simulation Setup Signals Schemes Converting Schemes	Image: constraint of the second sec	Annsient Simulation Max. nu Damping Appl Max. Damp Converg Vse Use Use Dse Dse	Simulation
Transient Simulation Simulation Setup Signals Schemes Converting Schemes	<pre></pre>	Ansient Simulation Newton- Max. nu Damping Appl Max. Damp Converg Vise Use Use Use Vse Vse Vse Vse	Simulation
Transient Simulation Simulation Setup Signals Schemes Converting Discretization Schemes O Backward Euler TR-BDP2 Paramet Toleance Imax O Trapezoidal Max O TR-BDF2 Gamma: Automatic Time Step Scheme Step reduce factor In maximum time step: 10 Step reduce factor 0.6 Step increase factor 1.5	<pre></pre>	Ansient Simulation Newton- Max. nu Damping Appl Max. Damp Vax. Use Use Use Use Use Use Vota Currer	Simulation
Transient Simulation Simulation Setup Signals Schemes Converting Discretization Schemes O Backward Euler TR-BDF2 Parameter Trapezoidal Second-Order Gear Max. Gamma: O TR-BDF2 Gamma: Automatic Time Step Scheme Enable Automatic Time Step Scheme Maximum time step: 50m Maximum time step: 10 Step reduce factor 0.6 Step increase factor 1.5 Maximum Time 5000 Points: 5000	<pre></pre>	Ansient Simulation Max. nu Damping Appl Max. Damp Converg VUse Use Use Use Vuse Vota Currer	Simulation
Transient Simulation Simulation Setup Signals Schemes Converting Discretization Schemes O Trapezoidal Second-Order Gear O TR-BDF2 Automatic Time Step Scheme Enable Automatic Time Step Scheme Maximum time step: 10 Step reduce factor 0.6 Step increase factor 1.5 Maximum Time Points: Vse default values	<pre></pre>	Ransient Simulation Newton- Max. nu Damping Appl Max. Damp Vose Use Use Use Use Use Volta Curren	Simulation
Transient Simulation Simulation Setup Signals Schemes Converting Discretization Schemes O Backward Euler TR-BDP2 Paramet Taspezoidal Taleance Second-Order Gear Max. TR-BDF2 Gamma: Automatic Time Step Scheme Enable Automatic Time Step Scheme Maximum time step: 10 Step reduce factor 0.6 Step increase factor 1.5 Maximum Time Points: 5000 Use default values 5000	<pre></pre>	Ansient Simulation Newton- Max. nu Damping Appl Max. Damping Converg VUse Use Use Use Use Use Currer	Simulation

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.

Plot Manager 🔀
Double click on the signal to show it on the plot viewer.
Signal Waveform
element_model_z
Open in WaveRunner

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