# **Application Note**

Capacitive Pressure sensor design Application Note: Capacitive pressure sensor design Version 8/PC

Part Number 30-090-101 March 2006

© Copyright IntelliSense Software Corporation 2004, 2005, 2006 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<sup>™</sup> 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

I	INTRO	DUCTION	I
	I.I BAC	KGROUND	1
	I.2 MEN	IBRANE DESIGN	I
2	SAMPL	E PROCESS FLOW	4
	2.1 Sur	FACE MICRO-MACHINED CIRCULAR MEMBRANE DESIGN	4
3	THERM	IO ELECTRO MECHANICAL (TEM) ANALYSIS	7
	3.1 Exp	ORTING TO THE TEM MODULE	7
	3.2 MAN	NIPULATING YOUR VIEW SETTINGS	8
	3.3 Mes	H REFINEMENT	9
	3.4 Ma	ferial Properties, Loads and Boundary conditions	10
	3.4.1	Material properties	10
	3.4.2	Boundary conditions	12
	3.4.3	Loads	13
	3.5 NAT	fural frequency analysis	13
	3.6 STA	TIC BEHAVIOR	15
	3.6.1	Static Stress Analysis with Residual stress effects	15
	3.6.2	Incorporating stress gradient effects into the model	18
	3.6.3	Capacitance vs. Pressure curve	20
	3.6.4	Capacitance vs. Voltage effects	28
	3.6.5	Pull-in and membrane collapse	30
	3.6.6	Overpressure effects (stress effects)	36
	3.7 Dyr	JAMIC BEHAVIOR	39
	3.7.1	Settling time to a step response	39
	3.7.2	Frequency/Spectrum response	42
	3.8 Syst	TEM MODEL EXTRACTION	46
	3.8.1	Dominant and relevant modes	48
	3.8.2	Strain energy capture	50
	3.8.3	Electrostatic energy calculations	54
	3.8.4	Exporting the system model	55
	3.9 Simu	JLATING YOUR MACROMODEL IN SYNPLE	56
	3.9.1	Wiring your circuit	56
	3.9.2	Transient Force vs. displacement simulation	65
	3.9.3	Compatibility with system modeling tools: PSpice and SIMetrix	70
	3.9.3.1	Result Comparison of SYNPLE, PSpice and SIMetrix	70
4	SYSTE	M LEVEL MODELING	78
	4.1 Syst	FEM LEVEL SIMULATION	78
	4.1.1	High level readout circuitry	78
	4.1.2	Transistor level design	86
5	CONCI	USION	93
	5.1 Rev	IEW OF CONCEPTS	93
	5.2 Put	TING IT ALL TOGETHER	93
	5.3 SUM	MARY	93

# 1 Introduction

# I.I Background

Pressure sensors and microphones are among the killer applications of MEMS and are rapidly replacing sensors made with more traditional technologies. In this application note, we discuss in detail the various issues in fabrication, design analysis and system modeling of integrated pressure sensor devices. While IntelliSuite and SYNPLE can be used to model capacitive, piezoresistive and piezoelectric sensors, we will focus on electrostatic/capacitive sensing mechanisms in this note. Once the reader grasps the concepts behind the modeling, we are sure that he/she will be able to device sensors based on other detection technologies as well.

MEMS based pressure sensors and microphones use an elastic plate (also known as a membrane or a diaphragm as the active mechanical element. As the plate deflects due to the applied pressure, the middle surface (or the neutral axis, is located midway between the top and bottom surfaces of the plate) remains unstressed. The pressure introduces bi-axial stresses in the plate. As the plate moves up, straight lines in the plate that were originally vertical remain straight but become inclined; the intensity of either principle stresses at points on any such line is proportional to the distance from the middle surface, and the maximum stresses occur at the outer surfaces of the plate.

Capacitive pressure sensors work by detecting the change in capacitance between a fixed plate and the flexible plate. Piezoresistive pressure sensors work by converting the change in stresses to a change in resistivity of a strategically placed piezoresistor. Piezoelectric sensors work by converting the stresses into a change in electrical potential using a suitable piezoelectric coating.

# I.2 Membrane design

The reader is referred to the classic "Roark's formulas for stress and strain" by W.C. Young and R. G. Budynas (Mc Graw Hill) for detailed analytical formulas for plate design. There are several classic papers that the reader may want to peruse for a good understanding of pressure sensors. These include:

[1] A simulation program for the sensitivity and linearity of piezoresistive pressure sensors. Liwei Lin, JMEMS, December 1999

[2] Solid state capacitive pressure transducers. WH Ko. Sensors and Actuators, vol 10, 1986

Analytical solutions are available for an unstressed membrane. These can be used as a starting point for the design. In reality, full Finite Element/Boundary Element (FE/BE) based calculations are needed to incorporate processing related effects such as residual stresses and strain gradients. These are important effects that significantly effect the membrane deflection and can lead to unwanted effects such as membrane bi-stability (oil-can effect).

As a starting point for the pressure sensor design, consider a circular membrane with a radius R and thickness h. The deflection of the circular membrane is given by the equation:

$$W(r) = \frac{p}{49.6D} (R^{2} - r^{2})^{2}$$
  
where  $D = \frac{Eh^{3}}{12(1 - v^{2})}$ 

The maximum deflection of the membrane is given at the center of the membrane when r = 0 as:

$$W_{\rm max} = \frac{(1 - \nu^2)R^4}{4.13Eh^3}p$$

The maximum stress in the membrane is given as:

$$\sigma_{\max} = \frac{1.25 p R^2}{h^2}$$

For CMOS integrated poly-silicon and silicon pressure sensors, the typical material properties are E = 150-180 GPa, Poisson's ratio = 0.2-0.3. The density of silicon is 2320 kg/m<sup>3</sup>. In most surface micromachined processes the poly-silicon thickness is between 1-2 µm. The capacitor gap in integrated pressure sensors is between 1-2 µm depending upon the choice of sacrificial material.

Since the capacitance between the electrodes is a non-linear function of the gap, most sensor designers like to work in the linear range of pressure-displacement response. The maximum deflection at full scale is chosen such that the maximum deflection does not exceed 25-30% of the capacitance gap. Assuming mean values of membrane thickness of 1.5  $\mu$ m and capacitor gap of 1.5  $\mu$ m, let us limit the maximum deflection of the membrane at full scale to 0.4  $\mu$ m. Based upon this criteria we can estimate the membrane sizes for different full scale pressures using the expression

$$R = 4 \left( \frac{w_{\text{max}} \cdot 4.13Eh^3}{(1 - \upsilon^2)p} \right)$$

The table overleaf gives a sample calculation of the radius of the device for different pressures.

Material Properties		
E	1.69E+11	Pa
Nu	0.22	
Density	2320	kg/m3
Dimensions		
Thickness	1.50E-06	m
Capacitor gap	1.50E-06	m
Maximum stroke	0.28	of gap
Max stroke	4.20E-07	m
Dimensional estimat	tes (radius)	
Max stroke (w_max)	4.20E-07	m
Pressure (bar)	Radius (µm)	Maximum stress (Mpa)
0.1	178.68	18.1
0.2	150.25	25.6
0.5	119.49	40.5
I	100.48	57.2
1.2	96.00	62.7
1 5	90 79	70
1.5	/0.//	

Table I Pressure sensor design

# 2 Sample process flow

## 2.1 Surface micro-machined circular membrane design

- 1. Open the Pressure Sensor Simulation file (ProcessFlow.fab) in IntelliFAB using the default process database. The figure below shows the process flow for the surface micromachined pressure sensor. This can be done by:
- a) Launching IntelliFAB
- b) Clicking on the left hand side of the interface (white) and choosing File > Open Database > matfab.db
- c) Click on the right hand side of the interface (blue) this activates the fabrication portion of the interface open the Fabrication Process flow by choosing File > Open Fab > IntelliSuite\Training\Application Notes\Capacitive Pressure Sensor\Process\ProcessFlow.fab.
- d) You will see the interface shown below.



Figure 1 Process flow for the surface micromachined pressure sensor

2. Explore the process table by double clicking on each of the process steps and looking at the process parameters. In particular, when you open a Lithography step (Definition UV....) make sure you take a look at the mask layout associated with the step. You can do it by clicking the "Layout" button as shown in the figure below.



Figure 2 Spend some time exploring the process parameters. Make sure you take a peek at the Mask layouts associated with the process by clicking the Layout button.

3. Visualize the process by choosing Construct > Visualize. Step through the process visualization using the Start, Previous and Next buttons on the interface.

a) Note that the Last step of sacrificing the silicon wafer was added to isolate the device for ease of simulation, in reality the silicon wafer is never sacrificially etched!



Figure 3 Visualize the device fabrication

# 3 Thermo electro mechanical (TEM) Analysis

# 3.1 Exporting to the TEM module

You can start the simulation by exporting the fabrication sequence into the ThermoElectroMechanical module. You can do this by Clicking Simulate menu and choosing the ThermoElectroMechanical menu entry.

File Process Construct	Simulation Database Help				
🗅 🚅 🔚 👗 🖻 I	ThermoElectroMechanical				
E- Bonding	Anisotropic Etch ki 100				
🖃 Al-Si	2. Deposition SIU2 Thermal Wet				
🛨 Eutectic	3. Definition UV Contact Suss				
- Pvrex	4 Etch SiO2 Wet BOE				

Figure 4 Export to the TEM module

Save the Analysis file (.save) in a convenient directory (make sure that there are no spaces in the folder or file name).

Save Analysis F	ile As:					? 🗙
Savejn:	Processflow-	visualization	•	+ 🗈 💣	•	
	Pressure-sense	or.save				
	File <u>n</u> ame:	Pressure-sensor.save		•		<u>S</u> ave
	Save as type:	Analysis modules' files (*.sav	/e)	•		Cancel

Figure 5 Save the analysis file (.save) in a convenient location make sure that the file, folder and path names do not have spaces in them. Use an underscore character instead of a space.

IntelliSuite will automatically create a Finite Element Meshed model of the structure and open it in the TEM module.

# 3.2 Manipulating your view settings

Since surface micromachined MEMS devices are typically just a few thick, they appear to be very flat in the initial view in TEM. TEM gives you the capability to independently set X, Y and Z default zoom factors.

- 1. You can use the Shift + Up arrow and Shift + Down arrow to zoom in and out. Ctrl + Up arrow and Ctrl + Down can be used to rotate the device. You can also use your mouse to manipulate the device in the 3D space.
- 2. You can use the View > Zoom > Define to set your view setting in X, Y and X directions.

[9] Thermoellectromechanical Analysis - [1551-same]	
File         Veter         Generative         Mathematical Mathematicae Mathematicae	- # X
Konch      Translate     Zown in     Rest      Sam Out	
Mesh     John Kindew     Johns     Johns     Johns	
<ul> <li>✓ Toober</li> <li>✓ Status Ber</li> </ul>	
3D Reviewa * Bedgmand Calar Hen Calar	
Communication and edition of the second seco	
Define acon factors for the x, y, and z dimensions	. NUM

Zoom Dialo	g		×
	Zoom Factor (#)		
	X	1	
	Y	1	
	Z	10	
	08	Cancel	

Figure 6 Zoom Define Dialog allows you to set independent X, Y and Z zoom factors. Choose a Z zoom factor of 100 in this case.

3. Feel free to explore the different view options available and make yourself comfortable with the keyboard shortcuts and mouse movements to manipulate the device the in 3D space.

4. The view of the pressure sensor should look similar to the figure below

📅 ThermoElectroMechanical Analysis - [TLST.save]	E 🗟 🛛
Be yew Geometry Smulaton Baterial Loads Buundary Mgch Analysis Beault Window Belo	- # ×
Meranda de maria da de para la UNTI A 11120, AS Maranes ande de para l'ANSI 11420, AS Maranes ande de para l'ANSI 11420, AS Maranes ande ande ande de la UNTI A 11120, AS Maranes ande ande ande de la UNTI A 11120, AS Maranes ande de para l'ANSI 1453, IASA Maranes ande de para l'ANSI 1453, IASA Maranes ande de la UNTI A 11120, AS Maranes ande de la UNTI	
Fer Heb, pres P1	NM

Figure 7 Initial view of the device after setting the appropriate zoom factors

# 3.3 Mesh refinement

Click...Mesh...Auto

ThermoElectroMechanical Analysis - [TEST.save]		E 8 🛛
File Vew Geometry Simulation Material Loads Boundary	Meth Analysis Result Window Help	_ # X
	Information	
1	Auto	
	Load Previous Mesh	
	Fement Structure Number Definition	
	Remark Bergreah	
	Group Surfaces for Elec_mesh	
	Medt_Mesh	
	Extract Resetur Peak-	
	Selection Mode	
Intervation		
Current rock number 4140 Minimum rock and 0.277711 0.111205 0.5		
Maximum meth mar (co.m) 146387, 146387, 15		
The original involtor releaded?		
Maximum much imp (2.9.2) 0277711.0111276.05		
Carrent rode marker \$752		
The inspiral methic reliaded		
Refine the mesh globally		N.M.

Enter...10...in the maximum mesh size dialogue



# 3.4 Material Properties, Loads and Boundary conditions

In order to perform analyses in the TEM module, you can follow the menus sequentially from View to Result. The actions that are to be performed are laid out in a logical progression. The menus will allow you to choose your simulation settings, check and modify material properties, apply loads and boundary conditions, mesh the device, and explore the simulation results.

## 3.4.1 Material properties

The material properties of the device can be set by Selecting the Materials menu. Materials > Check/Modify. You can click on an entity to set the material properties.



# Figure 8

Use Material > Check/Modify to modify the material properties



Modify DENSITY Value								
Constant Variable	,							
Row # Temperature, deg DENSITY, g/cm3								
2								
4								
5								
7								
9	OK							
10	Cancel							

Figure 9 Selected entities are highlighted in red. For instance clicking on the word density will bring the modify density dialog which allows you to specify either a constant or a variable (with temperature) density

Modify STRESS Value		Modify STRESS Value	
Constant   Variable	(	Constant Variable	
Original Value 20 MPa New Value 1 MPa	OK Cancel	Row #         Order         Value, MPa           1	OK Cancel

Figure 10 You can apply residual stresses (constant) or high order variable stress gradient profiles.

#### 3.4.2 Boundary conditions

You can set the boundary conditions by selecting the appropriate degree of freedom of a particular entitiy and clicking on the appropriate boundary.

Make sure you fix all of the fixed boundaries of the device as shown in the figures below.





Figure 11 Fixed boundary conditions for the pressure sensor. The bottom electrode and the surface in contact with the silicon are fixed

Click...Boundary....SelectionMode...CheckOnly

Click...Boundary...Fixed

All the Fixed Boundaries will be highlighted

#### 3.4.3 Loads

IntelliSuite allows you to apply a large number of loads to the device. These range from forces, pressures, Coriolis forces, temperatures and other stimuli. The loads can be constant loads, time varying loads or frequency varying loads.

We will revisit the load application in static analysis.

# 3.5 Natural frequency analysis

Let us explore the first 5 modes of the device.

Frequency analysis allows you to quickly check your model setup and mesh convergence information. Since AC/Frequency analysis results can be performed quickly, they are often used to make sure of the model accuracy.

1. Set the simulation settings by choosing Simulation > Simulation Settings

🗓 Thei	ThermoElectroMechanical Analysis - [Pressure-sensor.save]												
📄 File	View	Ge	ometry	Simulation	Material	Loa	ads	Boundary	Mesh	Analysis	Result	Window	Help
🗋 🖻		X		Simulatio	n Setting		1						
	Piezoresistive		×										

#### Figure 12 Accessing Simulation settings

2. Set the simulation settings as below. Click Apply and OK

Simulation Setting	
Calculation Type Static Frequency Dynamic Macro Model Extraction Analyss Type Static Stress Heat Transfer/Thermal Stress ThermoElectroMechanical Relaxation	Option Frequency Modes Number and Frequency of Interest Modes Number 5 Displacement Small C Large Start Shape Oundeformed C Previously Deformed Piezo Material No Piezo Material Piezoelectric (undeformed shape only)
	Apply OK Cancel

Figure 13 Simulation settings for frequency analysis

- 3. Start the simulation by choosing Analysis > Start Frequency Analysis. Wait for the analysis to complete this should take 1-5 minutes depending upon your machine
- 4. Explore the results by choosing Result > Natural Frequency

Dialo	)g		×
	Mode	Natural Frequency (Hz)	
	Mode 1	2.57476e+006	
	Mode 2	5.24579e+006	
	Mode 3	5.48563e+006	
	Mode 4	8.64889e+006	
	Mode 5	8.91309e+006	
	1		
	- ·	07 Cm 1	
	Keport		

Natural Frequency

5. View mode animations by choosing Result > Mode Animation

🔀 ThermoElectroMechanical Analysis - [TEST, save]	
Bie Verw Gennetry Smulation Baterial Lands Boundary Migh Analysis Benuit Window Help	. # X
D S B X P B B F F	1971 - 1971 - 1971 - 1971 - 1971 - 1971 - 1971 - 1971 - 1971 - 1971 - 1971 - 1971 - 1971 - 1971 - 1971 - 1971 -
X	
Memodani         C           Para Guert Roadon Cultona, Para Guert Roadon Cultona, Para Guert Roadon Cultona, Para Guert Roadon, Standan Andreja M.         C           Standan Andreja M.         Standan Andreja M.           Standan Andreja M.         Standan Andreja M.	
for table second #1	24.04

# 3.6 Static behavior

## 3.6.1 Static Stress Analysis with Residual stress effects

To model the effect of a Residual Stress of 20Mpa on the diaphragm, first change the simulation settings as shown in the Figure below

Simulation Setting	X
Calculation Type	Option Result Thistory C Last State Displacement Small Large Start Shape Undeformed Previously Deformed Piezo Material No Piezo Material Piezoelectric (undeformed shape only) Piezoresistive - Transducer Assembly
	Contact Contact Analysis Apply OK Cancel



Click...Material...Check/Modify



Select the Diaphragm and a dialogue will appear as shown in the Figure below.

ThermoElectroMechanical Analysis - [TEST.save]		
🗍 Pie Vew Geonetry Seulaton Material Loads Boundary Heah Analysis Result Window Help		- 0 ×
D # B ( 1 % B) # ? #	Check Modify Material Property	3]
	Norm         Norm         Yan           110000         Mith         9           110000         Mith         4           110000<	
Alexandrea Server Alexandrea Alexandrea Tandalau enderge uit Tandalau en		
Fur Help, press F1		NUM

Double Click...Stress

Modify STRESS Value		
Constant   Variable		
Original Value 0 New Value	MPa	
20	MPa	O K Cancel

Enter...20 for the New Value of STRESS (Tensile stress of 20Mpa)

Click...OK twice to close both the dialogues.

Click...Analysis...Start Static Analysis



Click...Results...Z Displacement

# 3.6.2 Incorporating stress gradient effects into the model

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

Click on the diaphragm

Double Click...Stress

Click...Variable

Enter the stress values as shown in the Figure below

Thermoldlectroldechanical Analysis - [1137.save]	6	
The Year Georetry Seuditor Hateria Lundo Boundary Hech Analysis Result Wedge Hes	Check/Modify Material Property	×
	Import         Dat         Yes           CTBQ         Data         Yes           CTBQ         Data         Yes           CTBQ         Data         Yes           CTBQ         Data         Yes           TTBQ         Data         Data           TTBQ         Data         Data <t< td=""><td></td></t<>	
Mennehan (Feren 12) IKS Maradi (2) Na kenikatal Jana 12: IKS Maradi (2) Na kenikatal Jana (2) Na kenikatal Jana (2) Na kenikatal Jana (2) Na kenikatal	86	
For Help, press F1		
Constant Variable Row # Order 1 2 3 4 5 6 7 8 9 10	Value, MPa 1 20 2 40 3 60 0K Cancel	

Click...Analysis.....Start Static Analysis

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

ThermoElectroMechanical Analysis - [TIST.save]	
Be Vere Generaty Smulation (Salaria) Ladas Baurdary Mash Analysis Beaut Windon Heb	- # ×
	рл 0.00156525 0.00100000 0.000434905 0.000635441 0.00126661 0.00126661 0.00239295 0.00235213 0.00455165
Menular U Ang Dee Part Stat Part Stat U Ang Dee Dee Dee Dee Dee Dee Dee De	
For Meg. (res. F.	N.M.

## 3.6.3 Capacitance vs. Pressure curve

Click...Simulation...Simulation Setting

Simulation Setting X - Calculation Type Option Static - Result -C History 🖲 Last State C Frequency Displacement C Dynamic Large 🔿 Small C Macro Model Extraction Start Shape -- Analysis Type - Undeformed O Previously Deformed Stress/Displacement Convergence Definition 🔘 Heat Transfer Iteration Number 10 G Heat Transfer/Thermal Stress Iteration Accuracy 0.001 C Thermal Electrical C Thermal Electrical/Thermal Stress ThermoElectroMechanical Relaxation C Electrostatic C Electrostatic Force vs. Displacement Contact Contact Analysis Apply 0K Cancel

Change the simulation settings as shown in the Figure below.

Click...Apply...OK

We will remove the stress gradient in the model and include a constant residual tensil stress of 20Mpa.

Click...Material...Check/Modify

Click on the Diaphragm

Double Click...Stress...Constant

Change the value as shown in the Figure below

Modify ST	TRESS Value		×
Constant	Variable		
	Original Value		
		MPa	
	New Value		
	20	MPa	
			Cancel

Click...Loads....SelectionMode...Pick on Geometry

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

🗄 ThermoElectroMechanical Analysis - [TEST.save]	E C 🔀
Re Vew Geometry Serulation Naterial Loods Roundary Hesh Analysis Result Window Help	- 8 ×
Image: Description of the second of the s	
Menunia     C       Plans Solid Fas.     C       I fani Libit     C       Varia, interview     C	
Links authors had to the antity	No. M.

# Select the Diaphragm

Enter 0V as shown in Figure below

ThermoElectroMechanical Analysis - [TEST, save]		
File Vew Geometry Simulation Material Loads Boundary Mesh Analysis Result Window Help	Dialog	- #×
	Please light Voltage Below	
	Enter by Dryce Range	
	the second se	
	Single Value Input Value Range	
	Votege E. vot	
	OX: Cancel	
Testi 104		
Pean felest Entry's Mode Load Conditions		
Polanternal Zhao bomi selaudinal Polanter Selaut Enthylin Modity Load Canadhinay		
R devid Charleen plastel		
For Help, press F1		14,04

Dialog	
Please Input Voltage Below	
Enter by Single Value Input Input Value Range	
Single Value Input Value Range	1
Voltage 0 volt	
0K Cancel	

Click...OK

Click on the Bottom Doped Electrode

Enter 0V for the Voltage as shown in Figure below

ThermoElectroMechanical Analysis - [TEST.save]	Dialog	
File View Geometry Simulation Material Loads Boundary Mesh Analysis Result Window Help		- # X
	Please Input Voltage Delow	
	Enterby Dept Lines	
	and the second	
	Single Value Input Value Range	
	Votage 0 vot	
	in the second seco	
	OK Cancel	
blanda		
Hutting,		
Down Unroduction performant		
Please Select Rotty to Modey Load Conditions Material 2 has been selected		
Presis Faleet Tethylis Moldy Load Condition Material Diachaen salaetad		
Material That been interted		
For Heb, press F1		N.M

Click...OK

Click....Loads...Pressure...Face

Select the top face of the diaphragm

Enter 0 MPa for the Pressure as shown in the Figure below

ThermoElectroMechanical Analysis - [TEST.save]		🖬 🖻 🔛
File View Geometry Simulation Material Loods Boundary Mesh Analysis Result Window Help		_ # ×
Def E 1 De E E Y P Tenestre		
Pressure Pace	Dialog	X
Displacement • Node		
Print Lannacion Hand Thui	Please Input Pressure Below	
Rendered to a contract of the second s		
Acceleration	Single Value Input	
Consis Porce	Enter by	
Voltage •		
Current *		
Overge Density	Manufacture and a second se	
Amplitude vs Time	Single Value Input	
Amplitude vis Frequency •		
Selection Mode		
	Pressure 0 MPa	
X / I / I I I I I I I I I I I I I I I I		
	OX Cancel	
· · · · · · · · · · · · · · · · · · ·		
· · · · · · · · · · · · · · · · · · ·		
blanstee		
Base of Scores State		
Security of the second se		
School and a long in a product and a constraint.		
There Select Tempor Model Load Condenau		
Schered to be which a		
Please Salest Facet Modely Load Condense. Face 37 be here solected		
		Annual of Second
Apply a pressure load to a surface		N.M

Dialog
Please Input Pressure Below
Enter by Single Value Input
Single Value Input
Pressure 0 MPa
0K. Cancel

# Click...OK

Click...Analysis...Start Static Analysis

Once the analysis is complete,

Click...Result...Displacement...z

ThermoElectroMechanical Analysis - [TEST.save]	L 2 X
🗌 Bjer Vjew Geometry Sjmulation Baterial Loads (Bundary Medin Analysis Benult Window Help	_ # X
	pm 8.022394-005 9.004194-005 9.000260314 9.0005080857 9.0006771129 9.0008941401 9.00191167 9.001145222 9.00152249 9.00152249
birned him binded     C       Mand him binded     C       Mand him binded     C       Source     C       Does     C	

The displacement should be due to the residual stress in the diaphragm.

С	apacitance Di	alog				K
	Conductor 1 Entity 1 Entity 1 Entity 2 Entity 2 Entity 2		Conductor 2 Entity 1 Entity 2 Entity 1 Entity 2 Entity 2	Color	Value (nanofarads*1e-6) 20.55 -20.03 -20.03 23.61	
	<				>	ļ
	Report		OK		Cancel	

Click...Result...Capacitance

This is the initial capacitance between the diaphragm and the electrode. The capacitance result is in the form a capacitance matrix. The capacitance between Entity 1 and Entity 2 or vice versa is the capacitance of interest. The capacitance results are in the form of a matrix. Entity1 – Entity 1 capacitance is C11, which is the capacitance of Entity 1 with respect to infinity. Entity 1-Entity2 /Entity2-Entity1 are the capacitances of Entity 1 w.r.t Entity 2. Entity2-Entity2 is the capacitance of Entity 2 w.r.t infinity.

Increase the pressure on the diaphragm and find the change in capacitance:

Click...Loads...Pressure...Face

Select the top face of the diaphragm as shown in the Figure below

Enter a Pressure value of 0.3447 MPa

	المتاركت والما
File Ven Geometry Simulation Material Loads Boundary Mesh Analysis Result Window Help	- # X
	-
Peed by D Trans 3 dive Tere by Super Value by at Description DESCRIPTION	
Identified     Control       Control     Control       First First hours should     Control       Variang     Control       First First hours should     Control	

Click...Analysis...Start Static Analysis

 $Click...Result...Displacement \ Z$ 



Click...Result...Capacitance

C	apacitance Di	alog			
	Conductor 1 Entity 1 Entity 1 Entity 2 Entity 2	Color	Conductor 2 Entity 1 Entity 2 Entity 1 Entity 2	Color	Value (nanofarads*1e-6) 24.96 -23.54 -23.54 -23.54 27.12
	<				
	Report		<u>(0K</u> )	(	Cancel

Repeat the above steps for different values of Pressure and Plot the Pressure vs. Capacitance values to characterize the response of the capacitive pressure sensor. Please compare the results with the plot below.



#### Capacitance vs. Pressure



Please note the Z-displacement for each of the Pressure values and compare the results with the plot below

#### 3.6.4 Capacitance vs. Voltage effects

Remove the Pressure loads on the Diaphragm.

Click...Loads...SelectionMode...Delete All

Click...Loads...Pressure...Face

Click...Loads...SelectionMode...Pick on Geometry

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

Select the diaphragm (yellow entity)

Enter 10V

We will retain 0V on the Doped bottom electrode (green entity)

Click...Analysis....StartStaticAnalysis

Once the simulation is complete

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

Click...Result...Capacitance

C	apacitance Di	alog				×
	Conductor 1 Entity 1 Entity 1 Entity 2 Entity 2	Color	Conductor 2 Entity 1 Entity 2 Entity 1 Entity 2 Entity 2	Color	Value (nanotarads*1e-6) 20.55 -20.03 -20.03 23.61	
	<					>
	Report		OK	(	Cancel	

Repeat this simulation by varying the voltage on the yellow entity (diaphragm)

The results form this information can be used to arrive at the C vs. V response. Please match the results from the simulation with the results in the Figure below.



# Capacitance vs. Voltage

The capacitance should not change for the pressure range 0-50 Psi.

#### 3.6.5 Pull-in and membrane collapse

For Pull-in analysis, we will need to perform a Thermo-Electro-Mechanical-Relaxation analysis with Contact. The contact faces would be the bottom face of the yellow entity (diaphragm) and the top face of the green entity (bottom doped electrode).

Click...Simulation...Simulation Settings

Change the simulation settings as shown in the Figure below

Calculation Type	Option
<ul> <li>Static</li> </ul>	- Result
C Frammer	C History 👁 Last State
C Prequency	- Displacement
O Dynamic	C Small © Large
Macro Model Extraction	
Analysis Type	Start Shape
Stress/Displacement	<ul> <li>Undetormed</li> <li>Freviously Detormed</li> </ul>
C Heat Transfer	Convergence Definition
	Iteration Number
G Heat Transfer/Thermal Stress	There at time A command
C Thermal Electrical	Tretation Accuracy [0.001
C Thermal Electrical/Thermal Stress	
_	
<ul> <li>ThermoElectroMechanical Relaxation</li> </ul>	
C Electrostatic	
C Flortnot tio Ferrouur Dicelsonment	
S Electrostatic Porce VS. Displacement	
	Contact
	2 - XMANARAJARIANA

Click...Apply...OK

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

Select the Yellow entity (diaphragm)

Click...InputValueRange

Enter the values as shown in the Figure below:

Dialog					
Ple	ease Input Voltage Below	ą			
	Enter by Inpu	le Value Input t Value Range			
	Single Value Input	Input Value R	ange		1
	Range		Up 💌		
	Vottage Fro	m	300.	volt	
	То		1000.	volt	
	Increment		100.	volt	
		OK		Cancel	

Click...Boundary...SelectionMode...PickonGeometry Click...Boundary...Contact...FacePairDefinition...Face A

ThermoElectroMechanical Analysis - [TEST.save]		
File Vew Geometry Simulation Material Loads boundary, Mesh Analysis Result V	Indone Help	_ # X
Dok D 1 D St /6 9 M2 Fiel		
YZ Fixed		
KZ Fixed		
XY Fixed		
X Pixed		
Y Ford		
2 Fixed		
Free		
All Peor Fixed		
Group Surfaces for Fixed Boundary		
Rotate Define		
Attach Spring		
Attach Dashpot		
Attach Tie		
E CONTROL OF	Retro Res Defenses a	
Shanned Stre	From the Definition + 1 by Energia	
Marvelloter	Ence a second seco	
Havenue	Conciste Par	
Selection Mode	Verify Contact Paris	
	/	
	* · · · · · · · · · · · · · · · · · · ·	
	1	
Identifier .		
Material 2 has been selected		
Samulation participas part.		
til aine		
Down 1		
ferral ation setting is set.		
Please Select Entity.		
Show Current Contact Defention		



Select the bottom face of the diaphragm (yellow entity) as shown in the Figure below

Click...OK

Click...Boundary...Contact...FacePairDefinition...FaceB...Rigid

Select the top face of the Bottom electrode (green entity)




for tests, press #1 Click...OK

 $Click...Boundary....Contact...Face Pair Defintion...Complete\ Pair$ 

DoubleClick....Disabled

To Enable Face 89 and 2

Contact Pair	Dialog							
	Contact Pair Definition							
Pair ID 1 -								
Friction F	actor	0						
	Count	Face ID	Option	_				
Face A	1	89	Enabled					
	<		]	>				
	Corrot	Face ID	Ontion	_				
Face B Rigid	1	2	Enabled	_				
	<			>				
Face B Deformable	Count	Face ID	Option	-				
	<			>				
A	CCEPT	OK	Cancel					

Click...Accept...OK

Click...Boundary...Contact...FacePairDefinition...VerifyContactPairs

Dial	og			$\mathbf{\times}$
	Cor	ntact Pair		
	Pair ID	Option	~	
	1	Activated		
			~	
	<		>	
	( OK		Cancel	
			Cancel	

Contact Pair 1 should be activated

Click...OK...to activate the pair

Click...Analysis....StartStaticAnalysis

Click...Result...2DplotElectroMechanicalAnalysis...XCoordinate...Voltage Click...Result...2DplotElectroMechanicalAnalysis...YCoordinate...Z-displacement

A 2D plot will appear as shown in the Figure below



The voltage sweep was from 300V to 1000V. This can be fine tuned to 0 V to 650V with 50V increment. The simulation will take longer as more points need to be computed.

#### **3.6.6** Overpressure effects (stress effects)

Click...Simulation...SimulationSetting

Reset the simulation settings as shown in the Figure below

Simulation Setting	X
Calculation Type            • Static             • Frequency             • Dynamic             • Macro Model Extraction             Analysis Type             • Stress/Displacement             • Heat Transfer             • Heat Transfer             • Heat Transfer            • Heat Transfer/Thermal Stress             • Thermal Electrical            • ThermoElectrolMechanical Relaxation             • Electrostatic             • Electrostatic Force vs. Displacement	Option Result
	Contact

Click...Apply...OK

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

Select the yellow entity (diaphragm)

Click...SingleValueInput

Enter 0V for the voltage input

Dialog						
Ple	ase Imput Voltage	Below				
	Enter by	Single Value I Imput Value R	imput lange			
	Single Value Ir	nput   Input V	/alue Range			
	Voltage		이	volt		
					1	
			OK		Cancel	

Click...Loads...Pressure...Face.

Enter 6.894 MPa (1000 psi) for the pressure load.

ThemselectroMechanical Analysis - [TEST-saw]	E 🖸 🔀
De per Servery Spulator Spiral Load Spurdary Mph Brakes Spiral S	1 8 ×
	pm 7.03121c-605 40.1723074 40.515275 40.680532 40.680542 41.52594 41.52594 41.52595 41.527145 41.527145
	10.0

С	apacitance Di	alog				×
	Conductor 1 Entity 1 Entity 1 Entity 2 Entity 2	Color	Conductor 2 Entity 1 Entity 2 Entity 1 Entity 2	Color	Value (nanofarads*1e-6) 62.38 -61.87 -61.87 65.47	
	<					Σ
	Report		<u>0K</u>	0	ancel	

# 3.7 Dynamic behavior

# 3.7.1 Settling time to a step response

We will perform a dynamic Stress/Displacement (Direct Integration) analysis to determine the settling time for the pressure sensor and the influence of stiffness/mass damping on the settling time

Click...Simulation....SimulationSettings (to reset the simulation settings as shown in the Figure below)

zulation Type	Option
Static	_ Result
Frequency	⊂ History 🍜 Last State
Dvnamic	Displacement
Macro Model Extraction	C Small 🔍 Large
lucia Tana	Start Shape
Stress/Disp. (Direct Integration)	<ul> <li>Undeformed</li> <li>Previously Deformed</li> </ul>
Strate (Made David)	Piezo Material
Stressruisp. (Mode Blased)	💿 No Piezo Material
Heat Transfer Transient	<ul> <li>Piezoelectric (undeformed shape only)</li> </ul>
Thermal Electrical Transient	Piezoresistive - Transducer Assembly
Succeeded Stress Transient	Dynamic Steady State Option
·	• No Yes
Stress/Disp./Squeezed Film (Direct Integration)	Dyname
Stress/Disp./Electrostatic (Direct Integration)	Transient (Fixed Time Increment)
Stress/Disp./Electrostatic (Mode Based)	Time Period (Second)
	Time renod (Second)
stressrpisbity lectrostaticis daeesea tiim (pittect tiiteditation)	Increment Number (< 1000)   36
	Contact
	Contact Analysis

Click...Apply...OK Click...Material...DampingDefinition (Define Mass and Stiffness damping according to the Figure below) for a damping factor of 0.01

c/Modify Material	Property		
Property	Unit	Value	
Mass_damping	1/second	1.256e-9	
Stittness_damping	second	3.45e4	
<			>
Edit Property			
	_		
		07	Consel

Click...Apply...OK Click...Loads...Amplitude vs. time....Tabular

Click...Loads...Pressure...Face

Select the top face of the diaphragm (yellow entity)

Complete the Time vs. Pressure table as shown in the Figure below

Amplituide Definition			×
Face 37 is	elected		
Define Applied I	essure Curve		
Data Number	6		
Time(Sec	nd)	Pressure(MPa)	
		0	
3 1e-006		-0.3447	
4 4e-006		-0.3447	
5 4.1e-006		0	
E	it OK	Cancel	

Click...OK

Click...Analysis...StartDynamicAnalysis

Click...Result...2DPlotMechanicalAnalysis....Maximum....Displacement Z

The 2D plot will appear as shown in the Figure below.



#### 3.7.2 Frequency/Spectrum response

The spectrum analysis is a frequency sweep analysis. This gives the displacement results for the model over a specified frequency range. The Spectrum analysis results should match the static frequency analysis results.

Click....Simulation....SimulationSetting

Change the simulation settings as shown in the Figure below

Simulation Setting	
Calculation Type Static Frequency Macro Model Extraction Analysis Type Stress/Disp. (Direct Integration) Stress/Disp. (Mode Based) Heat Transfer Transient Thermal Electrical Transient	Option Result Mistory C Last State Displacement Small C Large Start Shape Undeformed Previously Deformed Piezo Material No Piezo Material Piezoelectric (undeformed shape only)
<ul> <li>Succeeded Stress Transient</li> <li>Stress/Disp./Squeezed Film (Direct Integration)</li> <li>Stress/Disp./Electrostatic (Direct Integration)</li> <li>Stress/Disp./Electrostatic (Mode Based)</li> <li>Stress/Disp./Electrostatic/Squeezed Film (Direct Integration)</li> </ul>	Dynamic         Image: Steady State Dynamics         Image: Transient Modal Dynamics         Freq. Range, from (Hz)         2000000         Freq. Range, to (Hz)         9000000         Point Number         26
	Apply OK Cancel

Click...Loads....Selection Mode....Pickon Geometry

Click...Loads...Amplitudevs.Time...Tabular

Set the loads according to the Figure below

Amp	lituide Definitio	n				×
		Face 37 is Selected	]			
	Defi	ne Applied Pressure	Curve			
	Da	ita Number	2			
	-	4	1			-
		Frequency(Hz)		Pressure(MP	'a)	_
	1	2000000.		1.e-003		
	2	7000000.		1.e-003		
					1	
	<				>	J.
		Edit	OK	Cancel		

(DoubleClick on 1 / 2 to Enter the Frequency and Pressure)

Click....OK

Click...Material...ModeDamping

Check if the ModeDamping is set to 0.01 for Mode 1.

Click...OK

Click....Analysis....StartDynamicAnalysis

Once the Analysis is complete

Click...Result...2DPlotMechanicalAnalysis...Maximum...Displacement...Z

The 2D plot should appear as shown in Figure below



Compare this value with the static Frequency results (shown in Figure below)

Dial	og		×
	Mode Mode 1 Mode 2 Mode 3 Mode 4 Mode 5	Natural Frequency (Hz) 2.57476e+006 5.24579e+006 5.48563e+006 8.64889e+006 8.91309e+006	
	Report	OK Cancel	]

# 3.8 System model extraction

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

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

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

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



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

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

where  $\Psi_{ext}$  represents the deformed state of structure,  $\Psi_{initial}$  represent the initial equilibrium state (derive from the residual stress without external loads),  $\psi_i(x,y,z)$  represents the displacement vector for the i<sup>th</sup> mode,  $q_i$  represents the coefficients for the i<sup>th</sup> mode, which is referred as "scaling factor for mode i". The modal superposition based reduce order modeling method is to solve each equation

$$m_i \partial^2 q_i / \partial t^2 + 2\xi_i \omega_i m_i \partial q_i / \partial t + \partial U_m(q) / \partial q_i - \partial Ue(q) / \partial q_i - \sum \psi_i F_j = 0$$

where  $m_i$  is the i<sup>th</sup> mode generalized mass,  $\xi_i$  is the linear modal damping ratio,  $\omega_i$  is the i<sup>th</sup> eigenfrequency,  $\psi_i$  is the ith modal shape function

(the displacement vector for the ith mode).  $\Sigma \psi_i F_j$  is sum over all the nodes of the external node force weighted by the mode shape.  $U_m$  is the strain energy,  $U_e$  is the electrostatic energy,  $U_e$  can be described as

 $U_e = 1/2 \sum (C^* V^2)$ 

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

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

- Find out "Modal Contribution". In this step, perform the standard electromechanical relaxation analysis and solve the initial deformed state (derive from the residual stress without external loads) and the final deformed state (with mechanical loads and applied voltage). Then use the QR factorization algorithm to determine the mode contribution for the deformed state.
- Calculate the relationship of "Strain Energy vs modal amplitudes". In this step, calculate the selected mode "Strain Energy vs modal amplitudes".
- Calculate the relationship of "mutual capacitance vs Modal amplitudes". In this step, calculate the selected mode "mutual capacitance vs modal amplitudes".
- From step 2 and 3, user can obtain  $\partial U_m(q)/\partial q_i$  and  $\partial Ue(q)/\partial q_i$  respectively.

For the current pressure sensor device, the following procedure describes the steps to extract the macromodel.

#### 3.8.1 Dominant and relevant modes

Click...Simulation...SimulationSettings

Reset the Simulations according to the Figure below

Simulation Setting	$\mathbf{X}$
Simulation Setting Calculation Type Static Frequency Dynamic Macro Model Extraction Analysis Type Rigid Body Variables	Option Frequency Modes Number 5 Displacement Small © Large Start Shape © Undeformed © Previously Deformed
<ul> <li>Spring Constants</li> <li>Squeezed Film D amping Variables</li> <li>Capacitance</li> <li>Capacitance vs Displacement</li> <li>Mechanical Reduced Order Modelling</li> <li>ElectroMechanical Reduced Order Modelling</li> </ul>	Convergence Definition         Iteration Number       15         Iteration Accuracy       0.005         Macro Model Extraction       •         • Modal Contribution       •         • Strain Energy vs Modal Amplitudes       •         • Mutual Capacitance vs Modal Amplitudes         • Mutual Capacitance vs Modal Amplitudes         • Contact         • Contact Analysis
	Apply OK Cancel

# Click...Material...ModeDamping Definition

Enter 5 for the mode number



Click...Analysis...Start Extract Macro Model

ThermoElectroMechanical Analysis - [TEST, save]	
Re Vew Geometry Simulation Material Loads Boundary Mesh Analysis Result Window Help	_ # X
D Set 2 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1	
Tay Frequency Analysis	
Salar Logitanti, Anargun Bhat Libin and Bases Mandal	
and Latitude Production	
Auto Setup for Memory	
- Manufacture	the second se
Webcare to befolk that is a second a	

Once the analysis is complete

Click...Results...Macromodel...ModalContribution

Dialo	g				
	Mode	Contribution	Option		
	1	-0.55784	Enabled		_
	2	-0.000232702	Enabled		
	3	0.00233432	Enabled		
	5	-0.0043514	Enabled		
	ľ	0.0100000	Fugoica		
	1				>
				,	<u> </u>
	[				
	L	OK j	Cancel		

Since Mode 1 has the maximum contribution (which is the case for most cases), the rest of the Modes can be disabled (Double Click on the Mode number to disable/enable a mode) but we will retain all the 5 modes for this case.

#### Click...OK

From the Modal contribution results, we notice that the contribution from Mode 1 is the maximum. For the Strain Energy Capture and the Mutual Capacitance capture, the scaling factor needs to be selected in the simulation settings. The default scaling factor for all the modes is 1, which will work for most of the cases. The only criterion for selecting the scaling factor is that the scaling factor should be greater than the modal contribution for a given mode. For a modal contribution "-0.55784", the scaling factor can be chosen as "1" and the scaling factor will range from "-1" to "1".

#### 3.8.2 Strain energy capture

Click...Simulation...Simulation Settings

Change the simulation settings as shown in the Figure below

Simulation Setting	X
Calculation Type	- Option
C Static	Frequency Modes Number
C Frequency	5
C Dynamic	Displacement
Macro Model Extraction	🔿 Small 💽 Large
	Start Shape
Analysis Type	Undeformed     C Previously Deformed
	Convergence Definition
C Spring Constants	
Squeezed Film D amping Variables	Iteration Number 15
C Canacitance	Iteration Accuracy 0.005
	Macro Model Extraction
Capacitance vs Displacement	C Modal Contribution
C Mechanical Reduced Order Modelling	Strain Energy vs Modal Amplitudes
ElectroMechanical Reduced Order Modelling	Mutual Capacitance vs Modal Amplitudes
	Scalling Factor for Mode 1
	Increment Number 10
	Mode 1 -
	Contact
	Apply OK Cancel

Click...Apply...OK.

Click...Material...Mode Damping Definition.

Enter 5 for the Mode number

Specify the Mode Damping as 0.01 for the 5 modes.

Click...Simulation...Start Extract Macromodel

Once the simulation is complete

Click...Result...Macromodel....Strain Energy vs. Modal Amplitudes

Double Click on "1" in the Model column to enable the plot option for mode 1 and then click OK.

A plot of the strain energy for mode 1 over the operating range will appear as shown in Figure below



Similarly, plots for Mode 2, 3, 4 and 5 can be generated using the same procedure.









Click...File ...Exit

Click...OK

# 3.8.3 Electrostatic energy calculations

For the Mutual Capacitance vs. Modal Amplitudes calculations:

Reset the Simulation settings as shown in the Figure below:

Click...Simulation....Simulation Settings

Click...Apply...OK

Click...Simulation...Start Extract Macromodel

Once the simulation is complete

Click...File...Save

Click...Boundary...Macromodel...Representative Nodes.



Select the node as shown in the Figure below (Highlighted node) Node 5715

We will apply node forces on the selected node during the simulations in SYNPLE. The "macmodel.out" file stores the information on the representative node.

### 3.8.4 Exporting the system model

The four files "str.out", "curr.macmodel", "macmodel.out" and "TEST.save" in the current working directory provide all the information required for the macro model generation. User can manually copy the four files to a different directory for use with SYNPLE module and exit the Thermal electro-mechanical module, or keep the TEM module open and start the SYNPLE module

Run the SYNPLE module and incorporate the macromodel information generated from the TEM module

# 3.9 Simulating your Macromodel in SYNPLE

# 3.9.1 Wiring your circuit

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

In the SYNPLE simulator, on the left side you have a list of available elements, categorized into Electrical Elements, MEMS Elements....

On the right side you have the 2D grid for your schematic as shown in the Figure below

SYNPLE - [Schematic1]	
Bie Edt yew Schematic Sir	mulaton Results jundom tiete
D 0 11 2 14 1	
52001	
PDC TRAN NOW IN	
Developing to the second secon	
北当月4日的四日9	a 15 16 16 16 16 16 16 16 16 16 16 16 16 16
Ready	0.0 4494, 31 74,54

Click...MEMSDevices....Click the Macromodel for TEM element and drag the element to the grid on the right side as shown in Figure below

	SYNPLE - [Schematic1]	
	Ple Edt Vew Schematic Sim.	Aston Results Window Help
	1 3 H & 1	
	2R22 1	
N 10 10 1 10 10 10 10 10 10 10 10	Construction     C	
	1 4 4 4 4 1 1 1 1 1 0 10	0 84 87 84

Click... the Macromodel template on your right

Click...Schematic...Include Reduced Order Macromodel

Select the "curr.macmodel" file (This file is saved in IntelliSuite\Training\Application\_Notes\Capacitive\_Pressuer\_Sensor\SystemModeling)

Click...Open

SYNULE - (Schematic1)			_ C 🛛
The fill Year Schematic Ser	datan Results Window Help		- F X
DOLLAN R			
52001	<b>1</b> 平因		
Pot man and			
		Including Macro Model File	
H Electrical Devices			
🗄 🏭 General Devices		Look in: 🔛 Macronodelnew 💽 🖝 🔝 😷 🛄+	
H - MEMS Devices		Curr Imode, macroodel	
Anghor Al Elli Beam (vint		2 Darr metrodel	
+ Compatible Elements			
Device Element		Documents Date Michford: 3/27/2006 11:13 AM	
Electrostatic Comb Drive		Size: 1.13MB	
Electrostatic Gap =/ Cac	· · · · · · · · · · · · · · · · · · ·		
Porce/Moment Khot	a at V1 V2 V2 V4 V2 at a	Tauleus	T
Global Frame	needed to be a second	reacto	
Knot			
Macromodel of TEM	· · · · · · · · · · · · · · · · · · ·		
- Non-Linear Beam	MEMO 2	Mr. Documents	
Pressure Sensor Capacit	Macro-Model	my bocuments	
Repuisve Horce (for tes Build Plate	Tomplata		T T T T ATA T T T T T T T T T T T T T T
TEM Macromodel (15 No			
2-Gap (for testing)		Normater Descent	
Mixed-Signal Devices	· · · · · · · · · · · · · · · · · · ·	ry conjuctor rite rame. Jour macroote	
		Pike of type. Macro Hodel File (macmode) Cencel	
12	C		8
🛦 🎿 🗛 📢 🖄 🗃 🔒	C & G		
			Land and land and Land and

To complete the circuit for the static analysis, we will need the following electrical and general elements:

- A "DC source" element from the Electrical Devices library
- A "Ground" element from the Electrical Devices library
- 2 "Constant" and 4 "Output terminal" elements from the General devices library

Please wire the elements as shown in the Figure below

If the wires are connected correctly, a name will automatically appear for the wire. The name can be changed by double clicking on the wire. Select the wire button as shown in the Figure below (arrow) Click and Drag between the nodes on the respective elements to connect them with a wire.

Please save the file at regular intervals

Please refer the wiring section in the "Getting Started with SYNPLE" manual for more instructions on wiring and common errors.



Once the wiring is complete

Click...File...Save/Save As

NOTE: Please save file in the same folder as the other files (curr.macmodel, macmodel.out, str.out) All system modeling files are saved in IntelliSuite\Training\Application Notes\Capacitive Pressuer Sensor\SystemModeling

Double Click the Macromodel template/element on the schematic

Set the properties of the macromodel as shown in the Figure below.

Device	е вторе	rues			
- Dev	ice:			_	
Tor	Tanalay mana antimese				
Ter	npiate:	mems_rom romocop			
Ref	. Name:	prssnr1mode	V	Shown on schematic	
Levi	an blass		-		
Lay	er No.:	ļ	1	Use Layer Definition	
Vi	Name	Valu	Je		
	xi1		1		
	xi2		0		
1	xi3		0		
i Fi	xi4		0		
H	vi5		0		
H	vi6		n		
H	vi7		0		
H			0		
			0		
분	XI3 		0		
님	XITU 4		0		
븓	qmax1		0		
븓	qmax2		U		
님	qmax3		U		
님	qmax4		U		
브	qmax5		0		
	qmax6		0		
	qmax7		0		
	qmax8		0		
	qmax9		0		
	qmax1	)	0		
	alpha1	100	М		
	alpha2	100	М		
	alpha3	100	М		
	alpha4	100	М		
	alpha5	100	М		
	alpha6	100	м		
	alpha7	100	М		
	alpha8	100	М		
	alpha9	100	М		
	alpha1	0 100	М		
	beta1		3		
	beta2		3		
	beta3		3		
	beta4		3		
10	beta5		3		
Ē	betaß		3		
Ē	beta7		3		
	beta8		3		
<b>I</b>	beta9		3		
1	beta10		3		
H	nf		1		
H	n coel		5		
H	indev1		1		
H	flag of		'n		
H	flag_p	)	n		
님	flag_p/	-	ň		
H	flag at	,	0		
금	flag =		0		
	nag_p;		0		
				OK Cancel	

Click...OK

Double Click the DC source and set "VDC as 5". This voltage has to be less than the pull-in voltage. We will need a minimum DC voltage

Double Click the "Constant" element connected to pin Fz1 and modify the properties as shown below

D	evice	Properti	es			
	-Devic Temp Ref. Laye	e: blate: co Name: [6 r No.: [	inst ICE			Shown on schematic Jse Layer Definition
	Vi	Name		Va	alue	
		с		65	50u	
		c1			0	
		c2			0	
	,		OK		ancel	]

We are defining a  $650 \mu N$  load in the z-direction

For the other "Constant" element leave the default values (0) for c, c1 and c2.

Click...DC

The EIR Year Schematic Senders Register (Mission (Mission)		
1 1 1 1 2 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1		
Ma / A A A M M F3		
De velle elle 🥿 🔢 📖 David		
Dervices Box x		
Beautry (D)		
Beautry with Tennyature		
Short		
Single Pole Double Throw Switch (SPC		
Switched-capacitor Elements		
Trangle Valtage Source	**************************************	
- Volt Meter		
Voltage Control Current Source (VCC		
Voltage Control Voltage Source (VCV	The Australia (Contraction of Contraction of Contra	
Voltage Pulse (10)		
Voltage Pulse (20p)	DC Setup   Sweek   N.B Jacobio Setup	
- Voltage Source (AC)	2.5 2. A service [advant   use manon samp]	
Voltage Source (AM) (for testing)		
Voltage Source (DC)		
Voltage Source (DC/AC)	A REAL PROPERTY AND ADDRESS OF AD	
Voltage Source (PH) (for testing)	Assign the time at which a UC simulation to be performed.	
Voltage-to-General Variable Converts		
🗟 🚟 General Devices	Contraction Contraction Contraction	
Clock Pube Source (0/1)	3.1 (4)	
Cox Source		
Conparator		
Comparator_1	A REAL AND A	
Conparator_with_hysteresis		
Conderator with Preterese 1		
Consume the second	and a second	
Derivative (un tesung)		
District	Annual Instance Berlin (Marca)	
Ceneral & Source	Parametric variation/ settings (JC sweep)	
General Dukes Source (In)		
- General Puite Source (200)	Set Stat up Parameters	
General Samler	a a ser a	
General Source (AM) (for testing)	Set Initial-guess Parameters	
- General Source (PH) (for testing)		
- Integration	Set Temperature	
- Integration1		
- Integrator with Reset		
- Linter	· · · · · · · · · · · · · · · · · · ·	
- Multiplication	OK Cancel	
- Multiplicator		
- Notch Rejector		
- Output Terminal		
- Output Terminal		
- Pole of Order 1		
- Pole of Order 2		
- SDN()		
- \$QR0		
= 5LM(2)		
- \$LM(3)		
=		
E Triangle Sources		
- VCO 🖌		

We will now specify a DC sweep (Parametrize the applied force) for the force/loading condition.

Click...Parametric Variation Settings (DC Sweep)

Click...Add (in the parametric variation window)

Select the parameter "c\_of\_force"

Double Click on the Parameter and change the settings as shown in the Figure below

Parameter Name: c_of_force	
lominal Value:	0.00065
Variation Type	Variation Parameters
O None	Start value:
C Log	Stop value: 0.001
C Gausian Distribution	
C Auto-Step Advnaced	Number of points: 51
Table (value1, value2,, valueN)	
0, 2e-005, 4e-005, 6e-005, 8e-005	5, 0.0001, 0.00012, 0.00014,
0.00016, 0.00018, 0.0002, 0.0002	22, 0.00024, 0.00026, 0.00028, 36, 0.00038, 0.0004, 0.00042,
0.00044, 0.00046, 0.00048, 0.000	J5, 0.00052, 0.00054, 0.00056, S4, 0.00066, 0.00068, 0.0007,
]	

We are sweeping the force linearly from 0N to 0.001N with the average value being 650  $\mu$ N

Click...Done Click...Close Leave the DC Time as 0

Click...Signals Select the Signals shown in Figure below

DC Analysis				
DC Setup Signals N-R Iteration Setup				
Please select the signal(s) to be watched: No more than 10 signals are allowed in one simulation.				
□       n3         ☑       z1         ☑       y1         ☑       x1         ☑       z2         □       n2         □       n4				
q2_of_prssnr1mode         q3_of_prssnr1mode         q4_of_prssnr1mode         q5_of_prssnr1mode         q6_of_prssnr1mode         q7_of_prssnr1mode         q7_of_prssnr1mode				
□       q8_of_prssnr1mode         □       q9_of_prssnr1mode         □       q10_of_prssnr1mode         ☑       cur1_of_prssnr1mode         ☑       cur2_of_prssnr1mode         ☑       cur3_of_prssnr1mode				
cur4_of_prssn1mode         cur5_of_prssn1mode         pwr_elec_of_vsrcdc_1         i1_of_vsrcdc_1         v1_of_vsrcdc_1				
Select/Unselect all signals	Cancel			



The first 7 signal names are names of the wires shown in Figure below



DC Analysis				
DC Setup   Signals N-R Iteration Setup				
N-R Iteration Max. N-R iterations: 2000 Damping Damp in the N-R iterations Max. damp N-R iter.: 10 Damping factor: 0.8				
Convergence Criteria				
RHS(2) Norm Tolerance: 100u				
RHS(infinite) Norm Tolerance: 0.1n				
delt(x)_all Norm Tolerance: 0.1n				
☐ delt(x)_separate Norm				
Voltage Tol.: 100u Temperature Tol.: 0.1				
Current Tol.: 10f Power Tol.: 0.1u				
Write convergence information to output file.				
Use default values				
OK Cancel				

Click...OK to start the analysis

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



Double Click the signals to view the plot/result

# Eg: Double click on Z1

The plot of Z1 vs. Force will appear as shown in the Figure below



This simulation file has been saved as "Pressure\_sensor1.ssc" and is present in IntelliSuite\Training\Application\_Notes\Capacitive\_Pressure\_Sensor\SystemModeling.

## 3.9.2 Transient Force vs. displacement simulation

We will perform a transient analysis for with the same macromodel

We will need to remove the "constant" element assigned to "Fz1" and replace it with a "General Pulse Source (1p) as shown in Figure below

This element is available in the General Devices category

	PLE pressure_sense	rttrans.sc)
	Edt yew Schematic S	mulaton Results Window (belo
	2024	
	0001	1 # IZ
	DC TALLES BOW	
	s Bax 3	
	Digital Devices	
	Electrical Devices	
	General Devices	
	MEMS Devices	
	Mixed-Signal Devices	
	Thermal Devices	
		· · · · · · · · · · · · · · · · · · ·
		Macro-Model x2 +
		et les Lerminte et
		bit = 1 = 1 = 1 = 1 = 1 = 1 = 1 = 1 = 1 =
A STREET A A B B B B B B		(S)
A CALLER AND A CALLER A	A LIP PLAT	- 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1
		WY THE A

Double click on the General Pulse element and change the properties as shown in the Figure below

D	Device Properties				
	-Devio Temp Ref. Laye	be: plate: pulse1 Name: <mark>pulse1_1</mark> rNo.:	F	Shown on schematic Use Layer Definition	
	Vi	Name	Value		
		tl	1u		
		t2	3u		
		val1	0		
		val2	650u		
		val3	0		
	1				
		ОК	Cance		

We are applying a 2µsec pulse of 650µN

Click...OK

Click...Transient

Reset the simulation settings as shown in the Figures below:

Transient Analysis					
Transient Se	Transient Setup   Signals   Schemes   N-R Iteration Setup				
Setup of	Transient simulation				
Start tim	e: 🚺	Seconds			
End time	e: 10u	Seconds			
Time ste	p: 10n	Seconds			
E Run	Run Startup Analysis prior to the Transient Analysis.				
	Set Start-up Pa	arameters			
1	Set Initial-guess I	Parameters			
	Set Temper	ature			
		ОК	Cancel		

Transient Analysis				
Transie	nt Setup Signals Schemes N-R Iteration Setup			
Please	e select the signal(s) to be watched:			
No	more than 10 signals are allowed in one simulation.			
	n3			
	z1			
	y1			
	x1			
	z2			
	n2			
	n4			
	q1_of_prssnr1mode			
	q2_of_prssnr1mode			
	q3_of_prssnr1mode			
	q4_of_prssnr1mode			
님	q5_of_prssnr1mode			
님	q6_of_prssnr1mode			
님	q/_of_prssnr1mode			
븜	q8_of_prssnr1mode			
님	q9_ot_prssnr imode			
님				
님	curl_or_pressrimode			
H				
H	cur4 of presn1mode			
H	curb of presn mode			
H	nwr elec of vsrodo 1			
H	i1 of vsrcdc 1			
ō	v1 of vsrcdc 1			
, ☐ Select/Unselect all signals				
	OK Cancel			

Transient Analysis	Transient Analysis
Transient Setup   Signals   Schemes   N-R Iteration Setup	Transient Setup   Signals   Schemes N-R Iteration Setup
Discretization Schemes     TR-BDF2 Parameters:     Toleance:     Toleance:	N-R Iteration Max. N-R iterations: 200 Damping
C Second-Order Gear     Max. iterations: 20	Max. damp N-R iter.: 10 Damping factor: 0.6
© TR-BDF2 Gamma: 0.585786	Convergence Criteria
Automatic Time Step Scheme	
Enable Automatic Time Step Scheme (ATSS)	RHS(infinite) Norm Tolerance: 0.1n
Minimum time interval: 50m * Time Step	delt(x)_all Nom Tolerance: 0.1n
Step reduce factor (<1): 0.6	delt(x)_separate Norm
Step increase factor (>1): 1.5	Voltage Tol.: 100u Temperature Tol.: 0.1
Maximum Time Points: 50000 (itmax_tms)	Current Tol.: 10f Power Tol.: 0.1u
	Write convergence information to output file.
Use default values	Use default values
OK Cancel	OK Cancel

Click...OK

To start the simulation

Once the simulation is complete

Double Click...on signals in the Signal manager to view the transient results

SYNPLE - [pressure_sensor1trans.ssc]		
File Edit View Schematic Simulation Resul	a Window Help	. 6
🗋 🤌 🖬 📚 🐒 🗋 🖻 🖻	I I I I I I I I I I I I I I I I I I I	
5200 / m #0		
Digital-to-Agains Converter(1 bt)/0, Vie		
and and a start well	z.wel/2 reedback	
Signal Marager		
[ management of the second sec	6 P. 1	- 12
Double click on the signal to show it on the plot viewer	+ 21	
The second se		
21	- 2	
21	* ng mar	
12	auforda	· · · · · · · · · · · · · · · · · · ·
12	end_solve	
	end,d T	
901)	Purning simulation.	
	system matrix size iz 33 x 53	MEMS
DCS)	Transient simulation starts.	
D(S)	fe_tns= 1	• tis Template is •
Voltage Pulse (1p)	SYNPLE: Program completed.	
Voltage Puse (20p)		→ 14
Voltage Source (AN) (for testion)	Simulation finished.	
<ul> <li>Voltage Source (DC)</li> </ul>	Data has been saved in C-W.w/icoDemo/trainino/Tutoria/ViorOma/Macromodelnew/pressure_sensor1transDitms.data	
Voltage Source (DC/AC)	v • •	
Voltage Source (PM) (for testing)		
General Devices		
Clock Pulse Source (0/1)		
Clock Source		
Comparator		
- Comparator_1		
Comparator with hysteresis 1		
Constant		
Derivative (for testing)		
Difference		
General MC Source		
General Pulse Source (1p)		
General Pulse Source (20p)		
General Sampler		
General Source (AM) (for testing)		
Integration		
Integration 1		
Integrator with Reset		
Liniter		
Multiplication		
Noteh Beleritor		
	× *	
		2
:요사 쉐 哈 집 음 일 ] 백 백 백	1 %	
ty .		0.0 4494.31 NUM

Eg: Double click on the Z1 signal to view the transient displacement results:



Click...File...exit

The transient simulation file is saved as

 $IntelliSuite \ Training \ Applicaton \ Notes \ Capacitive \ Pressure \ Sensor \ System \ Modeling \ pressure \ sensor \ 1 trains. ssc$
### 3.9.3 Compatibility with system modeling tools: PSpice and SIMetrix

### 3.9.3.1 Result Comparison of SYNPLE, PSpice and SIMetrix

The structure has 2 electrodes, which are connected to a 5 volts DC source and ground respectively. An external pulse force acts at z-direction on the reference node. We are monitoring the z-direction displacement in AC Sweep, DC Sweep and Transient analysis.

AC Sweep: Swept frequency from 100 Hz to 100 MHz

DC Sweep: Swept the external force from 0 to 1 mN

Transient: Applied a 650 µN pulse force as shown on the plot.



#### **DC Sweep:** SYNPLE



#### SIMetrix



# Transient:



#### SIMetrix:



The SIMETRIX circuit files are saved in IntelliSuite\Training\Application\_Notes\Capacitive\_Pressuer\_Sensor\SystemModeling \EDALinker\SIMetrix

The EDALinker.exe file is present in

IntelliSuite\Training\Application\_Notes\Capacitive\_Pressuer\_Sensor\SystemModeling\EDALinker

AC Analysis:

# SYNPLE





SIMet	rix												
ហ <sub>a</sub>	c1 (S	ichemat	rics Sub	circu	it *) (S	electe	d)					_	
Eile	<u>E</u> dit	⊆ursors	Annotate	Cur	<u>v</u> es A <u>x</u> e	es Vie <u>v</u>	<u>M</u> easu	ure <u>P</u> lo	ŀt				
89		<b>e</b> Q	🔇 🖉	<b>**</b> *	•🗄 👹	H E	= 💹	$\int \mathbb{R}$		5 <u>3DB</u> 3 <u>DB</u>	<u>.</u>		
	🔲 F	hase(:dz)						. 🗖 :	dz				
ac1	phas	e(:dz)											
		10m-											
							+			<u>ہ</u>			
		1m-											
											NE		
	2	100ս									X		
+	20.2	10.5											
		100										$\mathbb{N}$	
		1œ											
		100 n											
		-20-											
		-40-											
	~	-60-											
1	(Z)	-80-											
	ase	-100-								+			
	5	-120-											
		-140-					+ +						
		-160-					+ +				++		
		10	)0 40r	 ] 1k	: 2k 4ł	 : 10k	40k	 100k	400k	 1M 2M 4	м 10M	401	1100M
			-	, ,									
			requen	су/Н	Hertz								
x: 129	9.309	MegHertz	y: 7.851	94uV									

# 4 System level modeling

4.1 System level simulation

### 4.1.1 High level readout circuitry

Sigma Delta Modulator:

This example is for a second order Sigma delta modulator. The input to the modulator is a sine wave and the output is a digital output carried by a clock signal. This Sigma Delta Converter can be interfaced with the MEMS Macromodel to simulate the complete control circuit for the MEMS device.

Start the SYNPLE module.

Click on File....Open....C:/IntelliSuite/SYNPLE/Examples/ElectricalCircuitExamples/First order sigmadelta modulator/sigdel5.ssc



The Sigma-delta modulator file

The circuit should appear as shown in Figure below with the operational amplifiers, switching circuits and capacitors.



Second Order Sigma-Delta Modulator

We will perform a transient analysis with the modulator with a sine wave input.



AC Input Signal

Double click on the AC input signal to view the amplitude and frequency of the signal as shown in Figure 3 and Figure below.

Device	Properties		
Devia Tem Ref. Laya	ce: plate: vsrcac Name: <mark>vin</mark> er No.:		Shown on schematic Use Layer Definition
Vi	Name	Value	
	а	0.8	
	f_hz	3k	
	phi	0	
	tO	0	
	vdc	0	
	OK	Cancel	

AC input signal

Click OK to close the dialog.

Click on the Transient Analysis button



The Transient Analysis Dialog will show up as shown in Figure below.

Tra	Transient Analysis 🛛 🛛 🗙					
Т	Transient Setup   Signals   Schemes   N-R Iteration Setup   Setup of Transient simulation					
	Start tin	ne:	0	Second	3	
	End tim	e:	400u	Second	3	
	Time ste	ep:	5n	Second	3	
	Run DC Analysis prior to the Transient Analysis.     Run Startup Analysis prior to the Transient Analysis.     Set Element Parameters     Set Start-up Parameters					
	Set Initial-guess Parameters					
	Set Temperature					
			0	к	Cancel	

Transient Analysis settings

Please reset the Simulation settings as shown in Figures below.

Transie	nt Analysis	×			
Transier	Transient Setup Signals Schemes N-R Iteration Setup				
Please No	Please select the signal(s) to be watched: No more than 10 signals are allowed in one simulation.				
	vin+	^			
	pwr_elec_of_vin				
	v1_of_vin				
	i1_of_vin				
	f				
	VX+				
	phi4	_			
님	pwr_elec_of_s3	_			
님	i1_of_s3	_			
님	v1_of_s3	_			
님	ni 	_			
님	pwr_elec_or_c1				
님	v1 of c1				
H	obi1				
H	pwr elec of s2				
H	i1 of s2				
	v1_of_s2	~			
, 	Select/Unselect all signals				
	OK Ca	ncel			
Signals	s D				

Transient Analysis					
Transient Setup   Signals   Schemes   N-R Iteration Setup					
Discretization Schemes					
Backward Euler	TR-BDF2 Para	meters:			
C Trapezoidal	Toleance:	10u			
C Second-Order Gear	Max. iterations:	20			
C TR-BDF2	Gamma:	0.585786			
Automatic Time Step Sche	eme e Step Scheme (A	ATSS)			
Minimum time interval:	50m	* Time Step			
Maximum time interval:	10	* Time Step			
Step reduce factor (<1):	0.6				
Step increase factor (>1):	1.5				
Maximum Time Points: 5000000 (tmax_tms)					
Use default values					
	Oł	Cancel			

Discretization Schemes

Transient Analysis
Transient Setup   Signals   Schemes N-R Iteration Setup
N-R Iteration Max. N-R iterations: Damping Damp in the N-R iterations Max. damp N-R iter.: 10 Damping factor: 0.8
Convergence Criteria
I▼ RHS(2) Norm Tolerance: 1u
RHS(infinite) Norm Tolerance: 0.1n
C delt(x)_all Nom Tolerance: 0.1n
C delt(x)_separate Nom
Voltage Tol.: 100u Temperature Tol.: 0.1
Current Tol.: 0.1u
Write convergence information to output file.
Use default values
OK Cancel

N-R Iteration Setup



Simulation completes and the Signal Manager appears.

Double Click on "Vin" in the Signal Manager and the input sine wave signal will appear as shown in Figure below.



Click on the Transient Analysis Button again and reset the settings on the dialog as shown in Figure below

Transient Analysis	Transient Analysis	Transient Analysis
Transient Setup       Signals       Schemes       N-R iteration Setup         Please select the signal(s) to be watched:       No more than 10 signals are allowed in one simulation. <b>Vine</b> pwr_elec_of_vin              vin_of_vin                 vi.of_vin               vi.of_vin                 vi.of_vin               vi.of_vin                 vi.of_vin               vi.of_vin                 vi.of_vin                   vi.of_vin               vi.of_vin                 vi.of_vin                   vi.of_vin                   vi.of_vin                   vi.of_vin                   vi.of_s3                   vi.of_s2                   pwr_elec_of_s2                   vi.of_s2                   vi.of_s2	Transient Setup       Signals       Schemes       N-R teration Setup         Please select the signal(s) to be watched: No more than 10 signals are allowed in one simulation.       Image: Constraint of the simulation of the simulatint of the simulatint of the simulation of the simulatint of the s	Transient Setup       Signals       Schemes       N-R iteration Setup         Please select the signal(s) to be watched:       No more than 10 signals are allowed in one simulation. <ul> <li>pwr_elec_of_c_3</li> <li>i1_of_c_3</li> <li>i1_of_c_3</li> <li>i1_of_c_3</li> <li>i1_of_c_switch5_8</li> <li>i1_of_switch5_8</li> <li>i1_of_c_4</li> <li>pwr_elec_of_cpamp1_2</li> <li>d-</li> <li>pwr_elec_of_varcac_1</li> <li>i1_of_varcac_1</li> <li>i1_of_varcac_1</li> <li>i1_of_varcac_1</li> <li>j.j.g/switch5_8</li> </ul>
OK Cancel	OK Cancel	OK Cancel

Unselect "Vin" and Select "y" and "yb"

Click OK to start the Analysis

Once the analysis completes, Click on the signals "y" and "yb" in the signal manager window.

The Digital Outputs should appear as shown in Figures below.



Digital Output y



y and yb can be located on the circuit and the outputs are inverse of each other.

This example was for a second order sigma-delta modulator. The "FirstOrderSigmaDeltaModulator" folder has more examples.

## 4.1.2 Transistor level design

This capacitive pressure sensor was built in SYNPLE using a variety of MEMS and electrical elements available in the SYNPLE library. The circuit is designed to amplify the capacitance signal of the pressure sensor. The circuit output is an amplified "Analog" signal for an applied pressure pulse. The Analog signal can be sent through a Sigma Delta Converter for an equivalent digital output.

Click...File...Open..." balanced\_demodulator\_BSIM.ssc"

File is located in IntelliSuite\Training\Application\_Notes\Capacitive\_Pressuer\_Sensor\SystemModeling



Double Click any Bsim to check the properties of the transistor

Device Properti	ies			×
Device: Template: bs Ref. Name: M	im1_n 15		Shown on schematic	:
Layer No.:		1	Use Layer Definition	
Vi Name		Value		^
		5u		
□ 😡 w		5u		
ad ad		0		
as		0		
D pd		0		
D ps		0		
Vfb		-0.7		
□ Ivfb		-40m		
wvfb		50m		
🗖 phi		0.84		
🔲 lphi		0		
🔲 wphi		0		
🗖 k1		0.78		
□ lk1		-800u		~
	OK	Cance		

Variables "1" and "w" are highlighted as they are global variables for all Bsim s' in the circuit.



Click...Schematic...Global Variable Manager



Click...Add

G	lobal Va	riable Manager			X
	The globa Double cli	I definitions for the ck on variable nam	system. e to assign to syste	em parameters	
	Туре	Variable	Value		
	real	nl	5u		
	real	nw	5u		
	real	GMIN_INTELLI	1p		
	,				
	Add	Modify	Assign	Remove	Close

Define the value and the variable name

Global Variable Definition	
Variable Type: C Integer C Real Variable Name: Variable Value:	OK Cancel

Click...OK...Close

# Bsim simulation contd...

Double click on the Pressure input to the sensor



The properties of the input signal should be changed according to the Figure below

D	evice	Properties		×
	-Devio Tem Ref. Laye	ce: plate: triangle_1 Name: <mark>triangle_1_1</mark> rrNo.:		Shown on schematic
	Vi	Name	Value	
		t1	1200	1
		t2	1	
		tO	0	)
		g_high	120	)
		g_low	(	)
		epsl	1r	1
		iO	1	
		ОК	Cano	el

We are defining a 120Pa pressure pulse from 0 sec to 120µsec. (The input is in the form of a triangular pulse and the loading condition is during the linear increase (first half "t0" to "t1" of the triangular pulse)

Click...OK

Click...Transient

Set the simulation settings as shown in the Figures below

Transient Analysis		Transient Analysis
Transient Setup   Signals   Schemes   N-R Iterat	tion Setup	Transient Setup Signals Schemes N-R Iteration Setup
Setup of Transient simulation		No more than 10 signals are allowed in one simulation.
Start time: 0 Se	econds	□ v1_of_Cr1 ▲ □ n6
End time: 120u Se	econds	pwr_elec_of_Cr2
Time step: 0.2u Se	econds	□ v1_of_Cr2 □ n4
✓ Run DC Analysis prior to the Transient Anal	lysis.	□ n24 ☑ Pressure
E Run Startup Analysis prior to the Transient	Analysis.	□ i1_of_Cs1 □ v1_of_Cs1
Set Element Parameters		
Set Startuin Parametere		n1 pwr.elec. of va
Set letter Parenter		□ i1_of_va □ v1_of_va
		□ n10 □ pwr_elec_of_Cdc1
Set Temperature		
		J Select/Unselect all signals
OK	Cancel	OK Cancel

Transient Analysis	Transient Analysis
Transient Setup       Signals       Schemes       N-R Iteration Setup         Please select the signal(s) to be watched:       No more than 10 signals are allowed in one simulation.         ibulk_of_M21       ibulk_of_M21         pwr_elec_of_clock_2       i1_of_clock_2         i1_of_clock_2       i1_of_clock_2         n19       pwr_elec_of_LP1         iigate_of_LP1       ii_of_c_1         iil_of_c_1       iil_of_c_1         pwr_elec_of_c_1       iil_of_c_1         iv1_of_c_1       iil_of_c_1         pwr_elec_of_LP2       iil_of_LP2         iil_of_LP2       iil_of_LP2         iil_of_LP2       iil_of_LP2         iil_of_c_2       iil_of_c_2         iil_of_c_2       iil_of_c_2         iil_of_c_2       iil_of_c_2         iil_of_c_2       iil_of_c_2         iil_of_c_2       iil_of_c_2         iv1_of_c_2       iil_of_c_2         iv1_of_c_2       iil_of_c_2         select/Unselect all signals	Transient Setup       Signals       Schemes       N-R Iteration Setup         Discretization Schemes       Image: Constraint of the setup       TR-BDF2 Parameters:       Toleance:       Toleance:
OK Cance	OK Cancel

Transient Analysis	×
Transient Setup Signals Schemes N-R Iteration Setup	
N-R Iteration	
Max. N-R iterations: 2000	
Damping Damp in the N-R iterations	
Max. damp N-R iter.: 10 Damping factor: 0.2	
Convergence Criteria	
I RHS(2) Norm Tolerance: 1u	
RHS(nfinite) Norm Tolerance: 0.1n	
delt(x)_all Norm Tolerance: 0.1n	
☐ delt(x)_separate Norm	
Voltage Tol.: 1m Temperature Tol.: 0.1	
Current Tol.: 1n Power Tol.: 0.1u	
Write convergence information to output file.	
Use default values	
OK Cancel	

Click...OK

Once the Simulation is complete

Double click on Pressure in the Signal Manager window

The Pressure input to the sensor will be displayed as shown in Figure below



Double Click...Cap\_of\_Cs1

The capacitance output of the pressure sensor will be displayed as shown in Figure below



# 5 Conclusion

## 5.1 Review of concepts

A surface micro-machined circular capacitive pressure sensor was designed and simulated in both the device level and the system level. The layout was designed in IntelliMask and the process simulation and the 3D structure were realized in IntelliFab. The device level simulation performed on the device can be categorized into frequency analysis, static analysis, dynamic analysis and system model extraction. The static analysis involved residual stress analysis, stress gradient analysis, capacitance vs. pressure analysis, capacitance vs. voltage analysis, pull-in analysis, and overpressure analysis. The dynamic analyses involved dynamic pressure analysis and spectrum analysis. System model extraction involved extraction of relevant modes, strain energy and electrostatic energy. A Transient Force vs. displacement analysis was performed on the system model of the pressure sensor in SYNPLE and the results were compared with the results from SIMETRIX and PSPICE. System level analysis involved sigma delta modulator analysis and transistor level design. The sigma delta modulator could be potentially connected to the macromodel output to convert the analog output into an equivalent digital signal. The Transistor level design was for a pressure sensor designed using a collection of MEMS and Electrical elements including BSims (Transistors). The properties of the transistor can be varied according to the information available from the process flow.

## 5.2 Putting it all together

We will now review the results from each of the analyses and the significance of the results. The natural frequency analysis was done to determine the natural frequencies for the first five modes. These results were further used to validate the results from the frequency/spectrum analysis. The static stress and residual stress analysis were performed to determine the effects of these stresses on the device behavior. The Capacitance vs. Pressure analysis was performed to determine the change in capacitance for applied pressure and characterize the capacitive response of the device. The Capacitance vs. Voltage analysis was performed to determine the displacement and the resulting capacitance caused by varying the voltage on the device. The pull-in and membrane collapse analyzed to determine the device behavior at very high pressure loads. The dynamic analysis was performed on the device to determine the settling time for the sensor for a specific force/pressure pulse and damping factor. The system model of the pressure sensor was extracted to perform a system level analysis in SYNPLE and in EDA tools such as SIMETRIX and PSPICE. The results from SYNPLE, SIMETRIX and PSICE were compared for a transient Force vs. Displacement analysis.

## 5.3 Summary

A Capacitive pressure sensor was designed successfully both at the device level and the system level. The methodical approach to design a surface micromachined capacitive pressure sensor from Layout through process simulation, Frequency Analysis, Static Analysis, Dynamic Analysis, System Model Extraction, SPICE extraction, System level simulation in SYNPLE, transistor level simulation in SYNPLE and comparison of results between SYNPLE.SPICE and SIMETRIX has been discussed in detail in this application note.