Application Note

Tang Resonator



Application Note: Tang Resonator Version 8.6/PC

Part Number 30-090-101 Jan 2010

© Copyright IntelliSense Software Corporation 2004, 2005, 2006, 2007, 2008, 2009, 2010 All Rights Reserved.

Printed in the United States of America

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

Restricted Rights Legend

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

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

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

IntelliSuite[™], SYNPLE, CleanRoom, Blueprint, Fastfield Multiphysics, Hexpresso, TEM, IntelliMask, IntelliMask Pro, MEMS-Synth, 3DBuilder, IntelliFAB, FABViewer, WaveRunner, EDALinker, are trademarks of IntelliSense Software Corporation. Windows NT, Windows 2000, Windows XP, Vista and Windows 7 are trademarks of Microsoft Corporation.

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

Table of Contents

1. Introduction						
1.1.	Background	1				
2. Desig	an Exploration	3				
2.1.	Frequency Analysis	4				
2.2.	DC Analysis	10				
2.3.	3D Visualization of SYNPLE Model	12				
2.4.	Mask Extraction from SYNPLE	14				
3. Proce	ess Validation/Visualization	15				
3.1.	Layout	15				
3.2.	Process Development with IntelliFab	15				
3.3.	Visualization in FabViewer	18				
4. Extra	ction	20				
4.1.	Creating the Model in 3D Builder	20				
4.2.	Thermoelectromechanical Analysis	22				
	4.2.1 Setting up the model	22				
	4.2.2 Frequency Analysis	26				
	4.2.3 Static Analysis	29				
4.3.	System Model Extraction	33				
	4.3.1 What is System Model Extraction?	33				
4.4.	Extracting the Macromodel	35				
5. Final	Verification	41				
5.1.	Loading the Macromodel	41				
5.2.	Connecting Macromodel to a Circuit	43				
5.3.	Benchmarks	45				

6. Summary		49
5.3.2	DC Voltage vs. Displacement	48
5.3.1	Effects of Ambient Pressure	45

1. Introduction

In this application note, we will discuss the design of a resonator from initial system-level design exploration, through fabrication processing and device-level analysis, to macromodel extraction and verification. We will analyze the resonator we have developed and discuss the benefits of IntelliSuite, a complete Computer Aided Design (CAD) software package offered by IntelliSense, when it comes to MEMS-based resonator design. We will begin the design exploration in SYNPLE, the system-level simulator. Analyses can be performed very quickly in this tool, making it ideal for the initial design stages. We will then discuss the layout and process development of the device. From the mask layout we will automatically generate a 3D meshed model to be used for finite element analysis. After running simulations in the analysis module, we will extract a macromodel to be connected to a circuit and used for verification in SYNPLE.

1.1. Background

Recent advances in fabrication and packaging technologies have allowed for the development of small, high-quality, cost-effective MEMS resonators. Resonators are now an important sector of the MEMS market, with a large potential for growth in the wireless communications and inertial sensing industries. This resonator design discussed in this application note was developed by Dr. William C. Tang, currently of UC-Irvine. A model of the resonator is shown in the figure below.



Figure 1: Resonator Model

One comb drive is used to drive the resonator, and a double-sided single-folded beam structure is used as a spring. The second comb drive is used to sense the displacement of the device. Shown below is a partial SEM micrograph of a Tang resonator.



Figure 2: Tang Resonator

2. Design Exploration

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

To start the program,

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

The SYNPLE window will appear as shown below.

SYNPLE	dellar Teda	D	-h	-		-	2	-	-	2	2	2	2	2	2	2	2		- C
Be cur yew graw gam			ыр <u>Ж.</u> [0	્ •	. Ξ	0		1	•	•	0	Э.					-
		. 0	3 🍐 🖁	*	0.			I.	I Q	e	2	21	2	-	- 🖂	6	1		
Element Library: 4 ×	Schemati	c1																	1 Þ 🗙
🔄 Digital Elements						2.2.2	10.10		2.2	25 25 1			1 1 1	- 12 - S	1000	2.2		0.01	
Electrical Elements	13 11 1333 1																		
General Elements																			
MEMS Elements	18 m 188 m																		
Thermal Elements	10.00.000.00																		<i>3</i>
	10 10 KM 0																		S
	10 10 1024 B																		8
	10 10 1938 12																		2
	212.222.2																		
	1 1 1 10 10 1																		
	10 D 831 0																		
	10 10 1001 10																		0.00
	20.00.000.00																		
	20.00																		=
	10 10 1021 12																		
	18 to 1887 13																		-
	10 10 100 10																		
	10 10 10 10 10																		
	10 10 10 10 10																		
	0.0.000.00																		
	10.00.001.0																		8
	13 11 1833 1																		
	1 1 1 1 1 1																		
	1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1																		
	<						111											1	>
	Messages:																		ąχ
	Velcome to	SYNPLE																	~
																			~
du.												"	mss:1	883.75	180	2.25nH	CA		4 SCD

Figure 3: SYNPLE Window

Various digital, electrical, and MEMS elements can be dragged from the element library on the left into the drawing window on the right. In this section, we'll look at a model of the resonator that has been constructed using these individual elements. To view the model, open the following file:

<Installation Directory>\IntelliSuite\Training\Application_Notes\ Tang_Resonator\Tang_Resonator.ssc

The model will appear as shown below.



Double click on any of the elements to view their individual parameters.

2.1. Frequency Analysis

When you have finished examining the elements,

Click *Simulation...AC Analysis*

For the AC simulation, we will examine the device response to a frequency sweep from 1000 Hz to 1 MHz. The simulation settings are already set. To view the settings,

Click Frequency Sweep

arameter Name: Frequency		
Nominal Value:	1k	
Variation		
Variation Type	Variation Parameters	
	Start value: 1k	
 Linear Log 		
◯ Table	Stop value: 1M	
Gausian Distribution	Number of points: 501	
-Table (value1, value2,, value	eN)	^

Figure 5 Frequency Sweep Selection and Settings

You will see the frequency range we are analyzing.

To return to the simulation settings window, click *OK*. Under the *Signals* tab, you will see that the x- and y-displacement of the resonator will be observed. The settings in the *Convergence Setup* tab should be left on the defaults. To run the AC simulation,

Click OK

After about two minutes, a notification will appear in the message window informing you of the location of the simulation results. The *Plot Manager* window will also appear and will allow you to view a 2D graph of any of the output signals.



Figure 6 Feedback and Signal Manager Windows

Click *mag_of_Y_ac*

Click Open in Waverunner

Waverunner module will launch automatically. This module is new postprocessor of IntelliSuite, especially for MEMS, Analog, mixed signal waveform management.

Ting - Seetlaner		- 🐹
Elle Yew Qhart Iools Help		
📷 😭 😌 🗣 👘	ΤΟ.	
Att ad Carts ATA SOURCES GARTS		
	1.00×	

Figure 7 Waverunner window

Click Chart... Import Data Source

Find the file "Tang_resonator.ac.dat"

Click Open

Click Add Chart

Chart Template Dialog	_	Σ
Chart Type	Data Source	Tang_Resonator
Simple Chart	X-signal	Frequency Y-signal X_ac
Mag-Phase		
X-Y		
XYZ		
Monte Carlo	Scale	Linear
Bus (8-Up)	Processing	None
Displacement (6-Up)	Units	Engineering
		OK Cancel

Figure 8 Select signal to display

Click *Y_ac*

Click OK

The plot will appear as below:



Figure 9 AC Results

If you want to view the graph on a logarithmic scale, double click on the axes and the axis property editor window will appear.

Chart Infor	mation	Dialog 🔀
Chart		Axes Signals
V] <u>T</u> itle	Tang_Resonator
V	Legend	
] <u>B</u> order	
Ur	nits	Engineering
		🗸 ImageMap
		OK Cancel

Figure 10 Chart Information Dialog

Click	Y	
		Chart Information Dialog 🛛 🔀
		Chart Axes Signals
		O× ⊙M ⊖z
		Title Y_ac
		Max 80µ Steps 5
		Min 0µ
		<u>A</u> xis Options
		OK Cancel

Figure 11 axis options

Click Axis Options

Click

Axes

Select Log...Scale

	Chart Informat	ion Dialog	×
	Chart	Axes Signals	
	OX OY C) z	
	Title	Y_ac 🗸	
		1000	
	Max	Steps 5	Y
	Min	1e-005µ	
		<u>A</u> xis Options	
		✓ Show Value Labels	L
		Show Minimum Value	
		Linear Scale	
		✓ Log Scale	
		✓ No Tick Marks	
		Major Tick Marks Inside Major Tick Marks Outside	
		Show Minor Tick Marks	
		Show Major Gridlines	Cancel
L		Show Minor Gridlines	

Figure 12 logarithmic axis

The magnitude curve using logarithmic axes are shown below.



Figure 13 Magnitude Plot

To export the data to Excel or a text editor

Click Chart...Show Data

Dialog			×
SELECT A	LL(Ctrl+A)	COPY(Ctrl	+C)
#	Frequency	Y_ac	^
Row 1	1000	7.94483e-007	
Row 2	1013.91	7.94502e-007	
Row 3	1028.02	7.94521e-007	
Row 4	1042.32	7.9454e-007	
Row 5	1056.82	7.9456e-007	
Row 6	1071.52	7.94581e-007	
Row 7	1086.43	7.94602e-007	
Row 8	1101.54	7.94623e-007	
Row 9	1116.86	7.94646e-007	
Row 10	1132 /	7 04660007	~
	<u>о</u> к	<u>C</u> ar	ncel

Figure 14 data window

Select and *Copy* to copy the data to the clipboard.

You should notice that the resonant frequency is 34.4 kHz.

2.2. DC Analysis

Open the following file:

```
<Installation Directory>\IntelliSuite\Training\Application_Notes\
Tang_Resonator\Tang_Resonator_DC.ssc
```

The model will appear as shown below.



Figure 15 Schematic for DC Analysis

You can see that a DC voltage of 35 V has been applied to the bottom comb drive. To run the simulation,

Click Simulation...DC Analysis

After the simulation is complete, check the message window for the results. You will see that the structure displaces 0.198 microns in the Y-direction.

2.3. 3D Visualization of SYNPLE Model

Results from a simulation in SYNPLE can be visualized in 3D. First, run the simulation again.

Click Simulation...DC Analysis

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

DC (Static) Simulation
Simulation Setup Signals Convergence Setup Parametric Setup Please select the signal(s) to be watched Output Data Type Selected Variables @ All Nodes (Required for MEMS 3D Visualization)
Sel Signal
Select/Unselect all signals
OK Cancel

Figure 16 DC Simulation Dialog

Click *OK* to run the simulation. After the simulation is complete, select *MEMS 3D Visualization* in the Results menu. If you don't see this option, you need to reset your toolbar.

Click *View...Preference...Customize* or



Click *Reset*... in the dialog box that appears. *MEMS 3D Visualization* should now appear as an option in the Results menu.

🌏 sya	IPLE	-	-	-	-	-	-	-	
Eile	<u>E</u> dit	<u>V</u> iew	<u>D</u> raw	<u>S</u> imulation	<u>T</u> ools	<u>R</u> esults	Synthesis	<u>H</u> elp	
1 F.		HI.	X R	s 💼 🔄)e	Mess	age		b. 🖷
	_		a 10			🔳 <u>P</u> lot M	4anager		2
	ĂC	O (2. 💿		Ş 💧	💷 <u>R</u> OM	Visualization		lidate
Elem	ent Li	brary:	д >	< 🍞 Sche	ematic1		5 3D <u>V</u> isualiza	ition	nt.ssc
÷	🕀 🔚 Digital Elements								
	Figure 17 3D Visualization in Results Menu								

IntelliSense's post-processing module VisualEase will open with the 3D visualization of the device. Choose *Displacement Y* under the Variable box to view the y-displacement.



Figure 18 Resonator in VisualEase

2.4. Mask Extraction from SYNPLE

Mask layouts can be extracted from your SYNPLE schematic. Select Synthesize Mask Layout in Blueprint from the *Synthesis* menu as shown in the figure below.



Blueprint will automatically open with a mask layout for the structure.

E Blue	🕞 šlograt - (šroning red																																																					
D De	Edt	Yerr	Setu	0	-	Node	y Ł	wh F	inder	59	lock .	Yas	×25	1 1	pols	Ж	ndov	+ t	do)																																		- 6	# ×
10		H	26	5		-)	0	×	1		×.		3	11	90	uctur	al .	1						×	3	Top	, Cel	٢.	3	1.																								
1				12	Fath	Finde		=,	Inn		-	Mot	¥y •		Mea	oure			men	sion	. 8	j s	NIDE		01	ape0	ut•	6	Inte	ASUE																								
		Lana S	eson	dor	DC.v		tooled										-							2220				~				• 240																			1.4	×		
								1.11	-		-		-						-								-		-					-						-	-						-		-	-			101	Q.
11																																																					May I	*
1.																																																					Ex to	a
																																																					<u> </u>	2
-																																																						3
2 4	2																																																					3
~ .																																																						3
																-							L.	1	1																													0
																T																																						
0																																																						-
0																																																						
•																																																						- 1
21	< 1																																																					X
5 -	ŧ. 1																																																					•
Α -																																																						
	-				-											-																																			-			
	7.4	CAD ALL AL	112 8	Lear	-	-	-	-	-	-	_	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	_	-	2		
	Com	sand:	ine -																																																			
	Com	and:	1																																																			
	114	.000 :	117.0	00 :	0.00	0																																													T			

Figure 20 Resonator Layout in Blueprint

3. Process Validation/Visualization

There are two ways to build a model in IntelliSuite. A model based on the layout and actual fabrication processes can be constructed in IntelliFab and visualized in FabViewer. In 3D Builder, a user can generate a meshed 3D model of a device without worrying about the device processing. The first method will be discussed in this section, and the creation of a model in 3D Builder will be discussed in Section 4.

3.1. Layout

The first step in building the resonator model is to generate the layout for the device. In Figure 21 you will see the layout used for the resonator.



Figure 21 Resonator Layout

To view the mask, you must run IntelliMask.

Click Start...Programs...IntelliSuite...Blueprint

Once you have started Blueprint you can open the mask.

Click File...Open.

You must select the file from: <Installation Directory>\IntelliSuite\Training\ 3DBuilder\Non-Manhattan_Isotropic\MUMPS\TangResonator.msk

When you are finished inspecting the mask, exit the software.

Click File...Exit.

3.2. Process Development with IntelliFab

Once the layout is designed, the process must be developed. This is done using IntelliFab.

Click Start...Programs...IntelliSuite...IntelliFab.

The IntelliFab window will appear. The database on the left contains all of the available process steps. Double-clicking on a process step will add it to the process flow on the right.

🚳 Untitled - IntelliFab		_		_	_ 🗆 🔀
File Edit View Database Setup Int	elliSuite Help				
B 🖻 💾 🕹 B 🗋	⊁. ♥ 🚽 🔒 (OD P.	Setup 🖌 Verify Pro	cess 🞯 Layout + 🇯	Wisualize
Data Base 🧷 🗵	Drag a column header here to grou				
H H + -	# 🖻 🗹 Type	Material	Process	Process ID	Process Option
Bonding Deposition Deposition Etch			There are no items to show.		
Properties ab Data Base	1				
Filter Enter filter text here					
For Help, press F1					CAP NUM SCRL

Figure 22 IntelliFab window

To view the resonator process flow that uses the MUMPs template,

Click File...Open Fab...

Select the file from the following location: <Installation Directory>\IntelliSuite\ Training\FabViewer\TangResonator\MUMPS_with_release_holes.fab.

A setup window will appear where you can select the mask file and set the die size. Leave everything as is and click *OK*. The process flow will appear as shown below.

MUMPS_with_	release_holes.f	ab - IntelliFa	b		-					⊐ 🔀
File Edit Vie	ew Database	Setup Int	elliSuite:	Help						
	4 🔏 🖞	6	×	2	• 🕂 🔶		🔪 🛄 Setup 🧹 Ve	rify Process 🞯 Layout •	Visualize 🕕	
Properties		,¢ 🗵	Drag							^
<u>₿</u> 2.			# 1	I Ty	ne .	Material	Process	Process ID	Process Option	
😑 Basic Informa	tion				Definition	The contain	Contact	Succ	Trocos option	-
Process Option	Sacrifice	~	10		Etch	PSC	Concacc	Generic	Partial Etching	
Side	Тор	~	11		Definition	FDG	Contact	Succ	Factorictoning	_
Parameters			12		Etch	DEC	Contact	Generic	Etch Through	
T_oper(deg_C)	20		12		DOLV1 Processi	PDG	Generic	Generic	Etter mrough	- 11
Properties			12		Denesitien	DebuGi	L D C V D	C314	Conformal Department	
t_film(nm)	2000		13		- Deposition	POlybi	Contact	Suga	Conformal Deposition	_
t_etch(nm)	2000		17		- Deminicion	DebuGi	Contact	SUSS	Thek Thursonk	
Information			15		- EUUI	PUIyoi	Dry	oro-ridsilid	Etter mrough	_
Description			16		- Derinition	UV D.L.C.	Contact	SUSS	et la standa	
			17			POIYSI	Dry	5F6-Plasma	Etch Inrough	_
			G		Second Sacrific	ial				
			18		- Deposition	PSG	Generic	Generic	Conformal Deposition	
			19		- Definition	07	Contact	SUSS		
			20	×	– Etch	PSG	Generic	Generic	Etch Through	
			21		 Definition 	UV	Contact	Suss		
			22	V	L Etch	PSG	Generic	Generic	Etch Through	
			G	M 🗉	POLY2 Processi	ng				
			23	\checkmark	 Deposition 	PolySi	LPCVD	SiH4	Conformal Deposition	
			24	$\overline{\mathbf{v}}$	 Definition 	UV	Contact	Suss		
			25	\checkmark	– Etch	PolySi	Dry	SF6-Plasma	Etch Through	
			26	V	 Definition 	UV	Contact	Suss		
			27	\checkmark	L Etch	PolySi	Dry	SF6-Plasma	Etch Through	
			G	☑ =	METAL Pads					
			28		 Deposition 	PR-53800	Spin	53810		
			29	V	 Definition 	UV	Contact	Suss		
			30	V	– Etch	PR-53800	Wet	1112A	Etch Through	
			31		 Deposition 	Al	Sputter	Ar-Ambient	Conformal Deposition	
			32	V	 Definition 	UV	Contact	Suss		
			33	V	– Etch	Al	Wet	PAN	Etch Through	
			34	V	L Etch	PR-53800	Wet	Lift-off	Partial Etching	
Properties	Data Base		35		Etch	PSG	Generic	Generic	Sacrifice	~
Filter Enter f	ilter text here	;								
For Help, press F1									CAP NUM SO	IRL 🤞

Figure 23 MUMPs Process Used for the Resonator

You can click on any of the process steps and the associated parameters will appear in the Properties window on the left. If you select any of the mask steps (Definition UV Contact Suss) you can click the "Layout" button to open IntelliMask with the mask layout being used for that step.

3.3. Visualization in FabViewer

Once the process has been fully developed, the device can be visualized in FabViewer.

Click IntelliSuite...Visualize in FabViewer

FabViewer will open as shown below.



Figure 24 FabViewer Window

Click Setup...Resolution

The resolution setup window will appear as below. Here you can input scaling factors for both the surface layers and the substrate. You can also select from a list of preset resolution values or input your own resolution settings. Set the resolution to Best as shown below.

Resolution and scaling
Z-Scaling Surface layers 5
Substrate 1/ 10
Resolution
Resolution (number of voxels)
O Use presets
O Custom (voxels) X 278 Z 130
VRAM estimated at current resolution: 24 MB
(Available VRAM: 60 MB)
Current die size 591 µm x 379 µm x 13 µm (556 x 357 x 130 voxels)
Advanced
Use Defaults Cancel OK

Figure 25 Resolution settings

To view a simulation of the process and the resulting device, click the play button at the top of the screen. The figure below shows the final structure when the resolution is set to "Best". The simulation will take about 5 minutes to run at this resolution setting.



Figure 26 Resonator Model in FabViewer

4. Extraction

4.1. Creating the Model in 3D Builder

Building a model based on a process flow in IntelliFab (discussed in Section 3) is one way to create a model in IntelliSuite. The second method uses a program called 3D Builder to generate a meshed 3D model of a device based solely on the mask layout. The benefit of the second method is that it allows the user to forgo the process development part of MEMS design and skip straight to device level analysis, leaving process development to later.

To open the program,

Click Start...Programs...IntelliSuite...3D Builder.

The 3D Builder window will appear. Select Automesh from mask layout in the Mesh drop-down menu.

🕮 3DBuilder - [3DB2]											
🗄 Eile Edit View Draw Modify	Mesh Iools Window Help	_ 8 ×									
四層圖 从所自居	Automesh from mask layout 🛛 Ctrl+M 🔽 📰 Entity 1 💌 🎤 🛛 🥕 AutoMesh 🔱 Snap + 🥔 Validate + 🔳										
k	Irace from mask layout) Gild Mesh	5									
 ★ ★	III Multiple Slices Ctrl+Shift+L · · Spider Web Ctrl+Shift+W · · · Spider Web Ctrl+Shift+F · · · Zpper Ctrl+Shift+F · · · Zpper Ctrl+Shift+Z · · · Zpper by Two Nodes · · · ·	0 et 0									
	Yalidate Mesh Connectivity Validate Mesh Conformity Draw Mesh Outline	2									
5											
	****	•									
	V.m.										
		_									
	Level # Bot Lev: Top Lev: Heral Mathematic Add Level Level 0 0.000 1.000 1.000 Deleta Level Level 1 1.000 2.000 1.000 Deleta Level Level 2 2.000 3.000 1.000 Level Level 3 3.000 4.000 1.000 Level Level 4 4.000 5.000 1.000 Level Level 5 5.000 6.000 1.000 Level Level 6 6.000 7.000 1.000 Modily Height										
Level: 0 Side: Bottom											
	Snap to Grid / Rotation CAP NUM	SCRL									

Figure 27 Automeshing in 3D Builder

In the dialog box that appears,

Click Browse...

Select the following file: <Installation Directory>\IntelliSuite\Training 3DBuilder\Non-Manhattan_Isotropic\MUMPS\TangResonator.msk

Click the *Advanced Setup* button. This will bring up a dialog box where you can choose which process steps to include.

Simplify polygons (arcs Meshing tolerance Ignore polygons with a Automatically assign e	s, circles etc) whose edges exc areas smaller than ntities	0.1 0.01	ım sq.um	
del Details Process Modeling	🔿 3D Modeling	ı		
Layer # in Mask	Layer Name	Processing	Thickness	Bottom Elevation
13	POLYO	Ground Plane	0.5	0.6
43	ANCHOR1	Anchor	2	1.1
45	POLY1	Structural(Conformal)	2	3.1
47	POLY1_POLY2_VIA	Structural Via	0.75	5.1
49	POLY2	Structural(Conformal)	1.5	5.85
50	DIMPLE	Dimple	0.75	2.35
51	METAL	Structural(Conformal)	0.5	7.35

Figure 28 Advanced Layer Setup

This particular device will be generated based on the MUMPS process. Leave all of the boxes checked, and click OK to return to the previous dialog box. Leave the maximum mesh size at 30 um and click OK in the Meshing Type dialog box. The program will then begin creating the automeshed structure.



Figure 29 Automeshed structure

For a better view of the structure, click inside the 3D window, then select *View...Define View Z Scaling*. Set the Z Scale Factor to 5.

Open the following file:

<Installation Directory>\IntelliSuite\Training\Application_Notes\ Tang_Resonator\Tang_Resonator.solid

This structure is the same as the automeshed structure you just created, but the comb drives have been set as different entities. This will be necessary for applying voltage loads in the finite element analysis tool.

To send the structure to the finite element analysis module,

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

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

4.2. Thermoelectromechanical Analysis

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

4.2.1 Setting up the model

The *Export to Analysis Module* function in 3D Builder will automatically open the model in the TEM module.



Figure 30 Resonator in TEM Module

To set the Z scaling factor,

Click View...Zoom...Define

Set the Zoom Factor in the z-direction to 5.

The first step in the TEM module is to choose a simulation setting and check the material properties.

Click *Simulation*...*Simulation Setting*.

The window shown in Figure 31 will appear. In this window you will find all of the simulation types and their respective options.

Simulation Setting	
Calculation Type Static Frequency Dynamic Macro Model Extraction Analysis Type Static Stress Heat Transfer/Thermal Stress ThermoElectroMechanical Relaxation ThermoElectroMagnetic Actuation 	Option Frequency Modes Number and Frequency of Interest Modes Number g Frequency 0 Displacement Small Large Start Shape O Undeformed Previously Deformed Piezo Material No Piezo Material Piezoelectric (undeformed shape only)
	Apply OK Cancel

Figure 31 Simulation Setting Window

In the first simulation we will run is a natural frequency analysis, so set the *Calculation Type* to *Frequency*, and the *Analysis Type* to *Static Stress*. Set the simulation to analyze the first 5 modes using small scale displacement theory, an undeformed start shape, and no piezomaterial.

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

Click Material...Check/Modify.

Select the entity that includes the beams and comb drives and makes up the moving structure. The entity will turn red and the Material Properties dialog will appear.

Entity Description:			Young	j 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 32 Material Properties

You will find that this entity has the material properties of poly silicon. The material properties of the rest of the entities do not matter in this case because they will be used for electrostatic purposes only, none of them will move. Click *OK* to exit this dialog.

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

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

Select the faces shown in Figure 30. All fixed faces are displayed in red.



Figure 33 Fixed Faces displayed in red

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

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

The software should go through a command prompt, and all of the fixed faces will appear in bright red. Note that you would have to select *Boundary*...*Pick on Geometry* before you could define any more boundary conditions.

4.2.2 Frequency Analysis

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

Click Analysis...Start Frequency Analysis

The frequency analysis will take 2-3 minutes. A command prompt similar to the one shown below will appear on the screen while the simulation is running.



Figure 34 Natural Frequency Results

When this window disappears, the simulation is complete. After the simulation, the results available to you will be the list of natural frequencies and animations of each mode shape. These can be found next to each other in the *Result* menu (Figure 35).

C ThermoElectroMechanical Analy	sis			. 🗆 🔀
Re View Geometry Simulat	tion Material Loads Boundary Mesh Analysis	Result Window Help		- 0 ×
- C. 👝 🖬 🖶 C.	👽 🛞 🙈 🔜 🚔 📥	Displacement	• 3.	
	🗠 , 🕸 🕸 🖬 📾 🧟 🧟	Figure Animation	• 📥 •	
Model Explorer 🗩 🗵		Deformed Shape		
🖪 Results 🔣 Model Explorer		History	IntelliCuite	
⊡ ⊙ TangRes.save		Desition Eason	Internoute	.9.8.
B- 2i Entities (9)		Person Person		8.8
Entry2		Temperature		.
- 💡 Entity3		Stress Component	•	2
- ¥ Entity4	1	Stress Invariant	•	3
- V Entry6	1	Potential	-	
- 💡 Entity7		Reactive Charge	1 martine 1	87
- V Entity8		Applied Current		
E- I Loads		Reactive Current		9
E-Static Loads (1)		Electrical Field		
Voltage Entity 5		Current Density		- 5
Voltage Entity 6	all and a second s	Capacitance		0
- Frenquency dependent Loads		Charge Density		-
Parametric Loads (0)		Pressure		xxy
Boundary conditions (8) Fixed Face 9		Maker of Procession	-	
-Fixed Face 1		Nada Animating		
-Fixed Face 3		Pidde Animation	-	0
- Fixed Face 369		MacroModel	<u>^ N</u>	7
-Fixed Face 155		Contour Mode		•
- Fixed Face 456		Show Mode	-	
⊡ @ Meshes			-	
- Mechanical Mesh		Selection Mode	· 🗾	
- Electrostatic Mesh		Frame Interval Time		
	•	Node Value		
	1	Node Curve	,	
		2D Plot, Mechanical Analysis		
		2D Plot, Electromechanical Analysis		
		Fachie All Date & Manual	-	
	Messages:	Lindund Hit Kesülk Pierkus	.¢ 🛛	
		Interval Result		
	1 Me(s) copied. Show Current Boundary Condition.		<u> </u>	
	Please Select Entity			
			×	
	For Malo, nears E1		Number of Noder - 19728	1
			Walluer or Nodes: 19728	

Figure 35 Natural Frequency Results

Click Result...Natural Frequency

You will see the list of natural frequencies shown in Figure 36.

Dialo	g		×
Dialo	g Mode 1 Mode 2 Mode 3 Mode 4 Mode 5	Natural Frequency (Hz) 32577.4 34744.3 38771.7 52488.8 58087.3	×
	Report	OK Cancel	

Figure 36 List of Natural Frequencies

Click OK to exit this dialog. If you want to view the animation of one of the natural frequencies,

Click Result...Mode Animation

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

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

Figure 37 Mode animation selection box

Select 2 for the mode number and 1 for the scale factor. When the mode animation begins, you will see the mass moving in the y-direction. This is the mode that we are interested in (Mode 1 corresponds to the movement of the mass in the z-direction). The natural frequency of mode 2 is 34744 Hz.

4.2.3 Static Analysis

Close the frequency analysis model and open the following file:

```
<Installation Directory>\IntelliSuite\Training\Application_Notes\
Tang Resonator\TEM\Resonator DC.save
```

In the Model Explorer on the left, you will see that static voltage loads have been applied to the model (applied using the *Voltage...Entity* option under the *Loads* menu). Double-click on *Voltage Entity 4* and you will see that a 35 V load has been applied to the top comb drive. The folded-beam structure and opposite comb drive have been set to 0 V.



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

To refine the electrostatic mesh,

Click Mesh...Group Surfaces for Elec mesh...Group Surfaces

The Group Mesh Dialog will appear. Fill in the values in Figure 36 and click OK.

Group	Mesh Dialog	
	Refine Electrical M	lesh within Two Nodes
	Factor N (0-30) Element Number	3
	First Node X	179
	First Node Y	121.5
	First Node Z	3.1
	Second Node X	261
	Second Node Y	137
	Second Node Z	5.1
	OK	Cancel

Figure 39 Mesh Coordinates for Comb 1

Click Mesh...Group Surfaces for Elec_mesh...Group Surfaces

Fill in the values in Figure 37 and click OK.

Group	Mesh Dialog		×
	Refine Electrical 1	vlesh within Two Nodes	
	Factor N (0-30) Element Number	3	
	First Node X	179	
	First Node Z	3.1	
	Second Node Y	238.5	
	OK	Cancel	

Figure 40 Mesh Coordinates for Comb 2

Click Mesh...Update Electrical Mesh

The electrostatic mesh will appear as shown below.



Figure 41 Electrostatic Mesh

Click Simulation ... Simulation Setting to view the simulation settings for this model.

Simulation Setting	
Calculation Setting Calculation Type Static Frequency Dynamic Macro Model Extraction Analysis Type Stress/Displacement	Option Result History Last State Displacement Start Shape Undeformed Previously Deformed
 Heat Transfer Heat Transfer/Thermal Stress Thermal Electrical Thermal Electrical/Thermal Stress ThermoElectroMechanical Relaxation Electrostatic Electrostatic Force vs. Displacement ThermoElectroMagnetic Actuation 	Convergence Definition Iteration Number 10 Iteration Accuracy 0.001
	Contact I Contact Analysis Apply OK Cancel

Figure 42 Static Simulation Settings

The material properties and boundary conditions are the same as defined in the previous model. After you have finished examining the model,

Click Analysis...Start Static Analysis

When the command prompt disappears, the simulation is complete. To view the y-displacement results,

Click *Result...Displacement...Y*

The model will appear as shown below.



Figure 43 Y-Displacement Results

To view the displacement of a certain node,

Click Result...Node Value

Click the center of the proof mass, for example. You will see that the y-displacement is 0.199 microns. The value obtained earlier from SYNPLE is within 0.5% of this value. For more information on the comparison of static simulations in SYNPLE and TEM, see Section 5.3.2.

4.3. System Model Extraction

4.3.1 What is 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. This can then be inserted as a black-box element into a mixed signal circuit simulator. This process is based upon the energy method approach, in that we construct analytical models for each of the energy domains of the system and determine all forces as gradients of the energy.

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

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

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



Figure 44 Creation of a Macromodel

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

$$u_{i}(t, x_{i}, y_{i}, z_{i}) = u_{eq} + \sum_{j=1}^{m} q_{j}(t) \cdot \Phi_{j}(x_{i}, y_{i}, z_{i})$$
(1)

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

$$m_{j}\ddot{q}_{j} + 2\xi_{j}\omega_{j}m_{j}\dot{q}_{j} + \frac{\partial W_{st}}{\partial q_{j}} = \frac{1}{2}\sum_{r}\frac{\partial C_{ks}}{\partial q_{j}} \cdot (V_{k} - V_{s})^{2} + \sum_{i=1}^{n}\Phi_{j}^{i} \cdot F_{i}$$
(2)

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

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

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

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

4.4. Extracting the Macromodel

The following procedure describes the steps to extract an electromechanical macromodel of the resonator.

Click Simulation...Simulation Setting

Set the simulation as shown in the figure below.

Simulation Setting	\mathbf{X}
Calculation Type	Option
🔿 Static	Frequency Modes Number
O Frequency	5
🔘 Dynamic	Displacement
 Macro Model Extraction 	🔿 Small 💿 Large
Analusis Type	Start Shape
○ Rigid Body Variables	Undeformed Previously Deformed
O Spring Constants	Convergence Definition
Squeezed Film Damping Variables	Iteration Number 15
Capacitance	Iteration Accuracy 0.005
Capacitance vs Displacement	Macro Model Extraction
O Mechanical Reduced Order Modelling	Strain Energy vs Modal Amplitudes
Sector Mechanical Reduced Order Modelling	Trancoal Cabacitance vs ratonal Ambienties
O Thermal Electrical/Thermal Stress	
ElectroMechanical + Fluidic Dissipation	
	Contact Contact Analysis
	Apply OK Cancel

Figure 45 Modal Contribution Simulation Settings

Click OK

To start the analysis,

Click Analysis...Start Extract Macro Model

Once the analysis is complete,

Click Result...Macromodel...Modal Contribution

This will show the contribution of each mode to the deformation of the device.

Dialo	ÞE			×
	Mode	Contribution	Option	
	1	-0.000721301	Enabled	
	3	0.194781 -2.78719e-005	Enabled Enabled	
	4	0.00113942	Enabled	
	5	0.000264754	Enabled	
	<			>
		(
		OK	Cancel	

Figure 46 Modal Contribution

If you wanted to reduce the time necessary to complete the strain energy calculation (the next step), you could disable any of the modes by double-clicking on them. In this case we will use all five modes.

Next, we will run the strain energy calculation.

Click Simulation...Simulation Setting

Set the simulation as shown in the figure below.

Simulation Setting	×				
Calculation Type	Option				
🔿 Static	Frequency Modes Number				
O Frequency	5				
O Dynamic	Displacement				
Macro Model Extraction	🔿 Small 💿 Large				
Analusis Time	Start Shape				
Rigid Body Variables	Undeformed Previously Deformed				
	Convergence Definition				
Spring Constants	Thurstein Marshan				
🔘 Squeezed Film Damping Variables	15				
Capacitance	Iteration Accuracy 0.005				
Canacitance vs Displacement	Macro Model Extraction				
	O Modal Contribution				
Mechanical Reduced Order Modelling	Strain Energy vs Modal Amplitudes				
ElectroMechanical Reduced Order Modelling	 Mutual Capacitance vs Modal Amplitudes 				
Thermal Electric all Thermal Stress					
	Scalling Factor for Mode				
C Electro Mechanical + Fluidic Dissipation	Increment Number 10				
	Mode 1				
	Contact Contact Analysis				
	Apply OK Cancel				

Figure 47 Strain Energy Simulation Settings

Click OK

To start the analysis,

Click Analysis...Start Extract Macro Model

Once the analysis is complete,

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

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



Figure 48 Strain Energy vs. Mode 1

Mont	coloot the	Mastal	Compositornos	Model	Amelitydae	hutton in t	ha Cimulation	Cattings .	
inexi,	select the	Mutual	Capacitance v	s. modal	Amphiludes	button in t	ne Simulation	Settings v	window.

Simulation Setting	N 1997 - 1997 - 1997 - 1997 - 1997 - 1997 - 1997 - 1997 - 1997 - 1997 - 1997 - 1997 - 1997 - 1997 - 1997 - 1997
Calculation Type Static Static Static Static Dynamic Macro Model Extraction Analysis Type Rigid Body Variables Spring Constants Squeezed Film Damping Variables Capacitance Capacitance Capacitance Capacitance ElectroMechanical Reduced Order Modelling Thermal Electrical/Thermal Stress ElectroMechanical + Fluidic Dissipation	Option Frequency Modes Number 5 Displacement Small Large Start Shape O Undetormed Previously Deformed Convergence Definition Iteration Number 15 Iteration Accuracy 0.005 Macro Model Extraction Modal Contribution Strain Energy vs Modal Amplitudes Image: Modal Amplitudes Image: Mutual Capacitance vs Modal Amplitudes Image: Modal Amplitudes Image: Contact Contact Analysis
	Apply OK Cancel

Figure 49 Capacitance Simulation Settings

Click OK

Click Analysis...Start Extract Macro Model

After the analysis is complete,

Click Result...Macromodel...Mutual Capacitance vs. Modal Amplitudes

A dialog showing the capacitance between each of the voltage entities will appear as below.

Capacitance)	Dialog	_	-		×
Conductor 1 Entity 4 Entity 4 Entity 4 Entity 5 Entity 5 Entity 5 Entity 6 Entity 6 Entity 6		Conductor 2 Entity 4 Entity 5 Entity 6 Entity 4 Entity 5 Entity 6 Entity 4 Entity 5 Entity 6		Value (picofarads*1e-6) 4025. -3109. -63.48 -3109. 7530. -3111. -63.48 -3111. 4027.	
<		0K		Cancel	>

Figure 50 Capacitance Results

The final step is to select the representative nodes that will be used to view the output displacements.

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

Select the node in the center of the top plate of the moving structure as shown in the figure below.



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

5. Final Verification

Simulations, especially dynamic simulations, can take several hours to complete in the TEM module. Extracting a macromodel and running the simulation in SYNPLE can save a great deal of time without sacrificing the accuracy of the results. You will also be able to connect the macromodel to the circuitry that will be used to control the device.

5.1. Loading the Macromodel

To open SYNPLE,

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

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



Figure 52 Schematic with Macro-Model Element

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

Click Tools...Load ROM Macro-model

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

In this case, we will examine a schematic that has already been created. Open the following file:

```
<Installation Directory>\IntelliSuite\Training\Application_Notes\
Tang_Resonator\Resonator_macromodel.ssc
```

The file will appear as shown in the figure below.



Figure 53 Macromodel Schematic for DC Analysis

The macromodel file has already been loaded into the template. A voltage load of 35 V has been applied to the pin corresponding to one of the comb drives. The folded-beam structure and opposite comb drive have been connected to a ground. The applied forces on the structure have been set to zero.

- Click Analysis...DC Analysis
- Click OK

After the simulation runs, you will see that the y-displacement is 0.188 microns, within 5% of the values obtained from the SYNPLE element model and the TEM model. Notice that the simulation only takes about 10-15 seconds, about 10x faster than the SYNPLE element model and 50-100x faster than the TEM simulation.

5.2. Connecting Macromodel to a Circuit

Open the following file:

```
<Installation Directory>\IntelliSuite\Training\Application_Notes\
Tang Resonator\Macromodel circuit.ssc
```

The schematic will appear as shown below.



Figure 54 Macromodel and Circuit for AC Analysis

The folded beam structure has been connected to a DC voltage. An AC voltage has been attached to one comb drive, and the inverted signal has been attached to the other comb drive. The frequency of the AC source has been set to 35 kHz to activate mode 2. To run the simulation,

Click Analysis ... AC Analysis

Click OK

After the simulation runs, double click mag of y1 ac in the Plot Manager.

Plot Manager 🔀
Double click on the signal to show it on the plot viewer.
Signal Waveform
mag_of_Vac+_ac
mag_of_Vacac
mag_of_Vac_in_ac
mag_of_Vdc_ac
mag_of_Vdc_in_ac
mag_of_x1_ac
mag_or_yr_ac
phase of Vac+ ac
phase_of_Vacac
phase_of_Vac_in_ac
phase_of_Vdc_ac
phase_of_Vdc_in_ac
phase_or_x1_ac
phase_of_y1_ac
phase_or_zr_de
· · · · · · · · · · · · · · · · · · ·
Open in WaveRunner

Figure 55 Plot Manager

This will bring up a plot of the AC response in the y-direction as shown below. Note that the axes in this curve have been set to Logarithmic Scale as discussed earlier in Section 2.1.



Figure 56 AC Response

Since mode 2 is being activated, this curve only shows the response corresponding to mode 2 and not any of the other modes. The peak occurs at 34.6 kHz, within 0.5% of the value from TEM.

5.3. Benchmarks

5.3.1 Effects of Ambient Pressure

The quality factor for the resonator will be highly dependent upon the packaging pressure for the device. In this section we will observe how the AC response changes for different packaging pressures.

The ambient parameter settings can be accessed through the Ambient Parameters button in the Managers toolbar.



The same frequency analysis as in Section 2.1 was run for varying values of the ambient pressure. Shown below is a Bode plot of the y-displacement at ambient pressures of 0.01 atm and 1 atm. You can see that the natural frequency does not change with pressure, but the Q factor decreases as the pressure increases.



Figure 58 Bode Plot for Pressure = 0.01 atm



Figure 59 Bode Plot for Pressure = 1 atm

The results from SYNPLE were compared with an empirical model developed by Xia Zhang and William C. Tang. This model predicts the Q factor of such a resonator for various packaging pressures and is defined as

$$\frac{1}{Q} = \frac{\mu}{\sqrt{2E\rho A_m (w/l)^3}} \left(\frac{\alpha A_q}{dh} \left(1 + \frac{d}{\delta}\right) + \frac{A_c}{d_c h}\right)^{-1}$$
(1)

where μ is the effective viscosity of the ambient fluid, *E* is the Young's modulus of the structure, ρ is the density, A_m is the effective area, *w* and *l* are the width and length of the beams, α is an experimental scaling factor, A_q is the damping-related effective area, *d* is the air film thickness, *h* is the structure thickness, δ is the penetration depth, A_c is the area of comb finger overlap, and d_c is the comb finger gap.

The effective viscosity is calculated using the following equation:

$$\mu = \frac{\mu_{stp}}{1 + 2K_n + 0.2K_n^{0.788} e^{-K_n/10}}^2,$$
(2)

where μ_{stp} is the viscosity of the fluid at standard temperature and pressure and K_n is the Knudsen number, given by

$$K_n = \frac{\lambda}{d} \,.$$

 λ is the mean free path and *d* is the air gap in this case.

The Q factor obtained from SYNPLE was compared to Q obtained from Equation 1. The results are shown below.



Figure 60 SYNPLE/Empirical Comparison

As you can see, the results from SYNPLE match up very well with the empirical predictions.

5.3.2 DC Voltage vs. Displacement

Varying static voltages were placed on one of the comb drives in the SYNPLE element model, TEM finite element model, and SYNPLE macromodel. The displacement of the central mass was obtained from each of the models, and the results are plotted in the figure below.



Figure 61 DC Voltage vs. Displacement

As you can see, the results from the three models match up almost exactly.

6. Summary

In this application note we discussed the development of a Tang resonator from the initial design stages all the way through the design process to final verification and control circuitry integration. We first began the design exploration in SYNPLE. The capability to run simulations that are much faster than FEA simulations makes this a great tool for early design optimization. Furthermore, the 3D visualization and mask extraction features make it easy to jump right into the next stage of the design process.

From a mask layout in IntelliMask and process flow in IntelliFab we were able to create a 3D model based on processing effects in FabViewer. We also used the automeshing feature in 3D Builder to create a 3D model from the mask layout with one click.

From there the model was exported to the ThermoElectroMechanical Analysis model for a more in-depth analysis. We discussed how to set up the simulation and learned about IntelliSuite's unique mesh refinement capabilities. The TEM uses Exposed Face Meshing methods to de-couple the mechanical and electrostatic meshes of devices, allowing users to perform mechanical mesh refinement in areas with high stress gradients and electrostatic mesh refinement in areas of high charge density. This greatly simplifies the mesh needed for the device without sacrificing any accuracy and can save an order of magnitude on time.

A macromodel was then extracted to SYNPLE to be used for quick dynamic analyses and control circuitry integration. System Model Extraction and SYNPLE modeling saves an incredibly large amount of time when compared to our own finite element solver, while our own FE solver saves time over our competitors'. Running optimization analyses with SME and SYNPLE can take a few hours, while running multiple optimization analyses with a FE solver would take days, weeks, or possibly months. Also, because resonators are complex structures with many degrees of freedom, it is absolutely necessary to utilize the N-Degree of Freedom (N-DOF) System Models that the TEM produces. 6-DOF models will not produce results anywhere near the accuracy of the N-DOF models for gyro devices. These N-DOF models give users a much better idea of how their device is going to react to its control circuitry.

Finally, we discussed how SYNPLE had been benchmarked against experimental models as well as against finite element simulations. High accuracy, unique features, and fast, easy-to-run simulations make IntelliSuite the perfect tool for resonator designers.



IntelliSense

Total MEMS Solutions

www.intellisense.com