## (Refer Slide Time 00:00)



# Indian Institute of Science भारतीय विज्ञान संस्थान

Indian Institute of Science

(Refer Slide Time 00:04)



NPTEL National Programme on Technology Enhanced Learning

(Refer Slide Time 00:08)



## NPTEL IS OFFERING ONLINE CERTIFICATION COURSES

(Refer Slide Time 00:11)



CERTIFICATES FROM THE IITs & IISc are just a click away..

onlinecourses.nptel.ac.in

(Refer Slide Time 00:15)



NO ENTRANCE EXAMS, NO ENTRY LEVEL CRITERIA JOIN ANY COURSE. ENROL FOR FREE!! https://onlinecourses.nptel.ac.in

(Refer Slide Time 00:19)



Indian Institute of Science, Bangalore NPTEL

Electronic Modules for Industrial Applications using Op-Amps

Dr. Hardik J. Pandya Department Of Electronic Systems Engineering So let me continue, let us continue with the electromechanical and thermal actuator. (Refer Slide Time 00:34)

# <image>

# So till now what we saw was piezoelectric materials, how to model that in COMSOL. We also talked about how to model piezoresistive structures within the COMSOL Multiphysics, how the coupling of structure mechanics with the electrostatics was occurring.

In this example that we are going to talk about, we are going to talk about how the actuation actually happens and what are the physics behind that.

(Refer Slide Time 0:57)

## **Capacitive Pressure Sensor**

- Pressure sensor example designed to provide an introduction to modeling techniques used in MEMS.
- Shows how to set up a coupled structural/electrical problem using important features of the MEMS Module:
  - Electromechanics
  - Thermal Stress
  - · Terminal Boundary Conditions
- Shows how to compute the sensor performance and the effects of thermally induced packaging stresses on the sensor response



One of the example model that comes into picture is Capacitive Pressure Sensor. These are regularly used in our mobile phones as you know and this is kind of touch sensors that people want to model or they want to have it on their mobile phones.

So let me just go ahead and open this particular model. So I go to COMSOL and I go to File, Application Library, and when I go and search for piezo, sorry, not piezo, this is now electromechanical system. So we have done away with the piezoelectric device. Now we are going to move ahead with the electro-mechanical actuations.

So let me go to actuators and I think it should be somewhere over here or it could be a part of structural mechanics module. So if you are not able to search, just search for it, capacitive pressure sensor. Since we are searching for the first time, it may take some time, but then it will take very quickly.

So this is the example which was already available in MEMS module sensors. So, capacitive pressure sensor, right. So let me just open this documentation to help you understand the Physics part.



So, again, that you see over here, this is the actual geometry that you want to model. Okay. So as you can see over here, this is a circular geometry, which is very nicely symmetric in nature and we actually exploit the symmetric nature of your geometry. What we do over here? We model only a quarter part of your geometry instead of the complete spherical or the cylindrical geometry.

So this is the actual domain which is model in COMSOL. So if you take a cross-section over here, you will see something like this, right?

(Refer Slide Time 03:07)

## (Refer Slide Time 2:26)



So you have the sealed chamber in the middle, and then you have insulators on both the sides, and then there are two membranes. One is the bottom part and the other is the top part. Okay. And this actually works as a capacitor because based upon the deformation that is going to happen over here, the change in capacitance is going to occur because capacitance is nothing but  $\epsilon A/d$ . So if the distance is decreasing, your capacitance is going to increase, right?

So let us go to the model. So let me open this model.



(Refer Slide Time 3:46)

So over here again, the first part is to make the geometry. So here you can see that how the geometry has been made, first making the blocks, and then partition domains, and then finally we have this particular domain, right?

File Home Definitions Geometry Material	ng + 1 cap Is Physics Mesh Study Results Developer	acove_pressure_persormpn - concour_munphysics -	
Application Model Model Pi * Variables Parameters * Parameters * Parameters * Parameters Case	e Build ColliveLink + Add Add Material Materials Physics	Build Mesh Mesh 1- Mesh Mesh Mesh Study Add Mesh Study Study Study Study Study Study Results	
Model Builder	- * Settings - *	Graphics Convergence Plot 1	- 1
capacitive_pressure_sensor.mph (root)     Gobal Definitions	Geometry     Build All		N K M
<ul> <li>Parameters 1</li> <li>Common Model Inputs</li> <li>Materials</li> </ul>	Labet: Geometry 1	1 2 mm	
Component 1 (comp1)     E Definitions     M. Generater 1	Scale values when changing units	0	
Materials     Solid Mechanics (solid)	mm • Angular unit	· M.	
Autophysics     Autophysics     Autophysics	Degrees •		-
P the Study 1 P the Study 2	Geometry representation:	-1	
Results	COMSOL kernel •		
Data Sets	Default repair tolerance:	Yes It I	
III Derived Values	Automatic •		
Tables     Gisplacement (solid)     Electric Potential (ec)	Automatic rebuild	Marcaner V Promer Los Table	
<ul> <li>▷ ○ Diaphragm Displacement vs Pressure</li> <li>▷ ○ Model Capacitance vs Pressure</li> </ul>	~	messages riogress cog more	

So we are going to model only a quarter part and then we are going to apply symmetric boundary condition on the right and on the left side. Okay.

Next part is the materials part, very important part. Here we have used the scalar material properties for the silicon as you can see over here and also the coefficient of thermal expansion comes into the picture. So that means this structure is also dealing with the thermal expansion.

(Refer Slide Time 04:30)	llig +   In Printics Math Churks Results	Cap	acitive_pressu	re_sensor.m	ph - COMSOL Multiphysics	- 0	×
Application Application Application Model Model Application	e Ald Geometry	olid Add hanics - Physics Physics	Build M Mesh M	esh Cor	mpute Study Add 3- Study Study Study	Add Plot Group: Unidows Reset Desitop:	
Model Builder	- * Settings Material			- 1	Graphics Convergence Plot 1		••
Capacitive_pressure_sensor.mph (root)     Global Definitions     P Parameters 1     Common Model Inputs	A 5 (overridden) 6		~	> ^		3 1 E 0   E 0 0   4 9 •	
Materials     Component 1 (comp1)     Definitions	Override     Material Properties				. 1	2 mm	
A Geometry 1     Materials     Silicon (mat?)     K 9 Vacuum (mat?)	Material Contents     Property	Variabik Val	ue Unit	Ртори		·	
Steel A/SI 4340 (mat.3)     Solid Mechanics (solid)     Ectrostatics (es)	Young's modulus Poisson's ratio Poisson's ratio Pensity	E 170 nu 0.06 tho 233	(GPa) Pa 6 1 0 kg/m <sup>2</sup>	Basic Basic Basic		0.5	mm
▷ ▲ Multiphysics ▷ ▲ Mesh 1 ▷ ∞ Study 1	Coefficient of thermal expansi Relative permittivity	epsilo 11.1	1/K 1	Basic Basic	1	1 2 3	
> too Study 3 > too Study 3 → Results > _ Data Sets					Y. X	0	
	v ∦.∞.+			-	Messages × Progress Log	Table ×	~ 1
		1.72 GB   1.8	2 G8				
Type here to search	1 0 🚍 🕾 🧔	13 6	•			^ 50 (1 0) ENG 1527	<b>1</b> .

The next part is the vacuum part that happens between the two domains over here. This is the doom that is going to deform to the downward direction and the capacitance is going to change.

File Home Definitions Geometry Materials Ph	rsics Mesh Study Results Developer	
A polication Component Model Pi - Variables Bail	Geometry     Materials     Image: Control of the control of t	Windows Reset Desktop - Layout
Model Builder     - ■       - = t ⊥ = • ■t □⊥ □ •       Image: Second pressure genour ph (noc)       Image: Second physical secon	Settings Material Labet Vacuum Geometric Entity Selection Geometric entity levet Domain Selection: Cavity Cavi	
	Active	-
> ~00 Study 3            > > 20 Data Sets          > ↓ Views          10 Derived Values           10 Tables	Material Contents     Market Variabii, Value     More Unit     Property     Variabii, Value     Unit     Prop     Relative permittivity     epsilo     1     1     Basic     Messages × Progress Log Table	× •

The next part is the steel domain that is in the bottom. This is the major part was steel and this would be again having a very high Young's modulus and very high density. Again, everyone is scalar over here.

The next part is electrostatic, sorry, the next part is the suction mechanism that would be solid mechanics part, and over here we are giving a boundary load as you can see over here.

Application Builder     Component Model     Pi Prameters     Watables* Profestions     Build Build Prameters       Model     Pi Model     Pi Prameters     Pi Prameters     Build Prameters       Model     Build Prameters     Pi Prameters     Build Prameters       Model     Pi Prameters     Pi Prameters     Pi Prameters       Model     Pi Prameters     Pi Prameters     Pi Prameters       Image: Pi Prameters     Pi Pi Pi Pi Pi Pi Pi Pi Pi Pi Pi Pi Pi P	report vetinit: by by materials Ma	wild Mesh Nesh Mesh - 1	pute Study 3- Study Soudy Competence Add Piot Soudy Results Graphics Convergence Piot 1
Model Builder	ngs Jary Load Boundary Load 1	- 1	Graphics Convergence Plot 1
	Boundary Load 1		
Narameters 1     Selectio     Materials     Materials     Materials     Selectio     P = Definitions     P △ Geometry 1			
P ■ Definitions     A Geometry 1	ndary Selection e. Manual		
Active Silicon (mat1)	13	* +	
		_	·
Free 1 P Ow Initial Values 1 P Env	erride and Contribution		mm 1
Symmetry 1 Col	ordinate System Selection		1 2 3
80 Prescribed Displacement 1 Coordin	ate system:		X.L.x mm
Electrostarics (e0	il coordinate system	•	
P ▲ Multiphysics     P ▲ Mesh 1     For     Study 1	ce		Messages × Progress Log Table ×
D role Study 2	×		<u>`</u>
	1.74 GB   1.88 GB		

## (Refer Slide Time 05:08)

So boundary load has been given on this particular boundary. Okay. In this particular boundary, how much pressure has been given? Here it's  $p_0$  pressure that has been given. A  $p_0$  pressure has been defined over here, 20 kilo Pascal. Okay.

I ]	husics Mesh	Shuda	Results Develor	capacitive_pressure_ser	sor.mpl	h - COMSOL	Multiphysi	a		- 0	×
Application Application Application Application Model Pi Pi Pi Pi Variables Parameters P	All Geometry	Add Material	Solid Ad Mechanics - Phys Physics	d Build Mesh Mesh 1- Mesh	Comp	oute Study 3. Study	Add Study	Displacement Add Plot (solid) 1 • Group • Results	Windows R De Layout	LC leset sktop •	
Model Builder	Settings Parameters Labet Parame	eters 1				Graphics Q. Q. d Q. C D Q. C D Q. C D Q. C D Q. C D D Q. C D D D D D D D D D D D D D D D D D D D	Conve	ngence Plot 1			
	** Name 1 p0 2 10 2 Toef 7	bipression O(k/ha) O(degC) O(degC]	Value 20000 Pa 293.15 K 343.15 K	Description Pressure Operating temperature Die Bonding temperat	J.	,	/ //~				
▶         N: Electrostatics (no)           ▶         A: Multiphysics           ▶         A: Multiphysics           ▶         > mis Study 1           ▶         > mis Study 1           ▶         > mis Study 2           ▶         > mis Study 3           ▲         Results           ▶         10 Data Sets					ł	y	mm	0 0	1	2	
Views     In Derived Values     In Tables	к. 1 1 П.	- 11 X. 🖿		_	×	Message	ns × Pr	ogress Log Table			- 1

So we give a particular load in the top and it is going to deform because it's an elastic material. So it's going to deform and the along with the deformation, there is also thermal expansion that is because of the temperature, there is going to be some, some external temperature.

More Definitions Generativy Matchalls       Payor       Matchalls       Payor       Study       Results       Developer         A Application Builder Underland       Py       **** Study       Py       ************************************	* ○●目記 * ちぐ市市市市美田市・	capar	citive_pressure_sensor.mph - COMSOL Multiphysics	- 0 ×
Model Builder              • • • • • • • • • • • • • • •	Application Model Model Definitions Geometry Materials P	reice Mesh Study Results Developer in import id COLVesinit- d developer Geometry Materials Physics Physics	Buid Meth Meth Meth	Prot p • Usindows Reset • Desktop • Layout
<ul> <li>Capacitive_pressure_sensor.mph (nool)</li> <li>Goobal Definitions</li> <li>Common Model Isputs</li> <li>Component 1 (comp1)</li> <li>E Definitions</li> <li>Common Model Input</li> <li>E Generaty 1</li> <li>Materials</li> <li>Common model Input</li> <li>Model Input</li> <li>Volume reference temperature:</li> <li>T set Common model Input</li> <li>Wourne reference temperature:</li> <li>T set Common model Input</li> <li>T thermal Expansion 1</li> <li>T thermal Expansion 1</li> <li>T thermal Expansion 1</li> <li>Sincent 1</li> <li>Sincent 1</li> <li>Sincent 1</li> <li>Sincent 1</li> <li>Common model Input</li> <li>T thermal Expansion 1</li> <li>T thermal Expansion 1</li> <li>Securit Coefficient of thermal expansion</li> <li>Coefficient of thermal expansion</li> <li>T from material</li> </ul>	Model Builder	Settings Thermal Expansion	Graphics Convergence Plot 1     Q. Q. Q. ⊕ ⊞ ↓ • □ □ □ □	
<ul> <li>Component 1 (comp.)</li> <li>Common model input</li> <li>Model Input</li> <li>Volume reference temperature:</li> <li>Vol</li></ul>		Active a (fot applicable) 4 5 6		
<ul> <li>Volume reference temperature:</li> <li>Volume reference temperature:</li> <li>Temperature:</li> <li>Temperature:</li> <li>Temperature:</li> <li>Timerral Expansion 1</li> <li>Thermal Expansion 1</li> <li>Symmetry 2</li> <li>Spondary 20</li> <li>Prescribed Displacement 1</li> <li>Boundary Load 1</li> <li>Secant coefficient of thermal expansion</li> <li>Coefficient of thermal expansion</li> <li>Coefficient of thermal expansion</li> <li>Temperature:</li> <li>Thermal Expansion</li> <li>Thermal Exp</li></ul>	Component 1 (comp1)     E Definitions     Add (connent 1)     Add (connent 1)     Add (connent 1)	Override and Contribution     Equation     Model Input		
■ Remail Expansion 1       T       User defined       •		Volume reference temperature: T <sub>inif</sub> Common model input Temperature:		
<ul> <li>⇒ prescribed Diplacement 1</li> <li>⇒ Bounday Load 1</li> <li>&gt; Electrostatics (a)</li> <li>&gt; Multiphysics</li> <li>&gt; Multiphysics</li> <li>&gt; Coefficient of thermal expansion:</li> <li>Coefficient of thermal expansion:</li> <li>Coefficient of thermal expansion:</li> <li>∞ Study 1</li> </ul>	Free 1 Fr	<ul> <li>T User defined</li> <li>T0 II</li> <li>Thermal Expansion Properties</li> </ul>		
▷ ▲ Multiphysics     Coefficient of thermal expansion:     ▲       ▷ ¬∞ Study 1     ↓     Grom material	Prescribed Displacement 1 Boundary Load 1 Electrostatics (es)	Input type: Secant coefficient of thermal expansion		
	P → Mutphysics     P → Mesh 1     P 100 Study 1	Coefficient of thermal expansion: α From material	Messages × Progress Log Table	

## (Refer Slide Time 05:43)

So here the temperature external is  $T_0$ . So it's around 20 degrees. This is operating temperature and then we have die bonding temperature that has been given in the model input over here  $T_{ref}$ , right? So here this is while the bonding was taking place at that time, some temperature would have been there and because of that there could have been some thermal expansion of your geometry. So it is not going to be, so this is while the fabrication is going to take place, so because of the fabrication issues also some kind of a deformation that takes into picture because of the thermal expansion.

We definitely want to give symmetric boundary condition because we are modelling only quarter part. So we have the symmetry boundary condition on the right side and on the left side, and then we give prescribed displacement. So what, why we are giving it is we are saying that it's not going to move in the top part. So it's not going to move in that direction in this particular point. So it's kind of we are giving some kind of constrains to this particular device and finally the boundary load from the top side. So it's going to give the load from the top to bottom. So it's going to push it to the bottom.



(Refer Slide Time 06:58)

The next part is the electrostatics part and the electrostatics, of course, we are going to give potentials. So in the case that the terminal boundary condition that is giving a potential of 1 volts. That is the top part. Zoom in and you will see that the top part, the whole domain is giving a, given 1 volts and the bottom part that the, so let me just enable the wireframe rendering and here it is a boundary. So you will notice that this is a boundary. This is not domain. However, the terminal is a domain.

(Refer Slide Time 07:33)

第10日日日、ちさ市市市市業業長・1	capaci	tive_pressure_sensor.mph - COMSOL Multiphysics	– o ×
File Home Definitions Geometry Materials Ph	sics Mesh Study Results Developer		
Application Builder Application Application Model	Geometry Material	Build Mesh Nesh 1- Mesh 2- Mesh 2-	Vindows Reset Desktop - Layout
Model Builder	Settings	- I Graphics Convergence Plot 1	×1
+ - 1 1 ± + 11 11 11 +	Terminal	QQQQH V·EEECON S	
Free 1	* Domain Selection		
<ul> <li>Symmetry 1</li> </ul>			0
Symmetry 2 Precision Displayment 1	Selectory Manual		
Boundary Load 1	(m) 4 (k)	1	
Electrostatics (es)     Change Conservation Solid	Active	8	
Zero Charge 1	\$		
Initial Values 1			
Charge Conservation 2			
Cround 1	Override and Contribution		
Multiphysics Multiphysics Multiphysics	Equation		
> via Study 1	* Terminal		
Study 2 100 Study 3	Terminal name:	2	
🖌 🎘 Results	1	Y X	
Data Sets	Terminal type:		
15 Derived Values	Voltage		
Tables     Displacement (solid)	Voltage:	Messages × Progress Log Table ×	× 1
Electric Potential (es)	V <sub>0</sub> [1	V V	
07:34	1.75 68   1.29 6	35:04	1530
O Type here to search	n 🖸 📴 📴 🧧 📴 🧧	07:34/35:04	10 ENG 27-02-2019 🖏

So how do you signify what is the difference? Here you will see the blue is completely filled, the cylinder and over here the blue is filled only on the surface of the ground, right? So, but in the middle it is still vacuum, right? So you can see over here, right, so in the bottom you have the ground and terminal is in the top, but in the middle it is still left. So let me just go to have a different views. May be that could help. Yeah. So you can see this is the bottom part and this is the top part terminal.

File Home Definitions Geometry Materials	Physics Mesh Study Results Developer	cove pres	in cocio	nanger - Comport manipel	pro-	- 0	
Application Application Application Application	Build Contections - Add Add Add Add Add Add Add Add Add A	Build Mesh	Mesh 1.	Compute Study 3* Study	Displacement (solid) 1 - Group - Results	Windows Reset Desktop - Layout	
Model Builder	Settings	- 1	Graph	Convergence Plo	1 1 2		~ 1
	Terminal		Q Q	魚◆田」と・	A LE LE COM	S BROOK I	
Free 1 Initial Values 1	Domain Selection	-	5.2		ROB 45		
Symmetry 1	Selection Manual						
10 Prescribed Displacement 1	4	+					
Boundary Load 1	18						
Charge Conservation, Solid	Active IC						
Initial Values 1							_
Charge Conservation 2					<u></u>		_
Ground T	Override and Contribution						
Multiphysics A Mesh 1	Equation						
> Study 1	* Terminal						
P No Shudy 2 P No Shudy 3	Terminal name:		z				
Results	1			Y			
> Urews	Terminal type:	_					
Tables	Voltage		Marca	Property L	on Table X		
Displacement (solid)	Voltage:		Messa	iges Progress U	og lable		- 1
Electric Potential (es)	v v0 1	- V	· ·				

Some part is already there which is missing. Okay. That is the vacuum.

The next is the Multiphysics. So here we move from piezoelectric to electromechanical forces, which covers solid mechanics with the electrostatics. Okay.

#### (Refer Slide Time 08:21) a COMSOL Ð P $\langle \! \circ \! \rangle$ Model Builder Co **T** • The life of 0 CEE Initial Val 0 hescribed D Boundary Load 1 1 (not applicable ostatics (es) (not app Charge Conse Zero Charge 3 (not applicable) nitiai Vali 5 (not applied Charge Cor 6 (not app E Eau A. Br al Forces 1 (eme1) · Coupled Interfac • 10 Data Set • 10 \* Electrostriction Progress Log Table S Displace 1.75 GB | 1.89 GB O Type here **1** •

So that's how it's done. So it has different types of studies. The first one is solves all, all the three of them as you can see over here: the solid mechanics, electrostatics and the moving mesh. Moving mesh is the most important part.

Now let me give you some introduction of moving mesh. So, for example, you have a particular geometry. So let me just go and show it in the making a figure. So, for example, we have a structure like this. So this is what structure that we have. Okay. And then we have one more bottom structure. Okay. And we are going to actually. We are going to apply the force from the top and which is going to make this structure to bend.

For example, it's going to bend like this, right? So it's going to bend to the bottom and we have two different values of d, for example. So this was the actual d value that is the distance between the two plates. This is d\_original and then we have after the displacement that let us define it as d\_final that the position at the final how much is the displacement. So we have d\_original and we have d\_final, right? So the d\_original will give you the original capacitance after deformation.

So the most important thing that is happening over here is, is that after the deformation the mesh of your upper cantilever beam also needs to change, right? So if you are meshing in the first part is the straight one, right, so let me go to the next page.

(Refer Slide Time 10:49)



Yeah. So if we talk about two parts, so my first part is going to be the straight part. This is just an explanatory diagram. This is not what is meshed over here. For example, over here we have this kind of meshing. So meshing is like we divide the actual geometry into small, small parts, right? So this is the original geometry. So let me write original.





After deformation, once the load is given, so let me write after deformation, the structure is going to actually deform. So by deform I mean the bottom structure is going to remain the same because we have given a kind of a constraint in the bottom, but the top structure is going to move like this. Okay. So you will see that your mesh also needs, needs to get deform, right?

So here you will find that there are more compressed structure over here as compared to the more relaxed versions on the left side. So here it would be more relaxed. Here it would be more relaxed mesh and over here it would be more compressed mesh.





So the meaning of the point that I am saying is that the mesh is going to get change and the way that I want to show you is done using the moving mesh that you can see over here, right? So the mesh that we were discussing before, that has to be taken into consideration using the moving mesh. Okay. The mesh needs to be actually changing and only then we give the potential on the changed geometry that is a deformed geometry.

So we need to work with two different geometries. The one is the original geometry that you saw in the, in the figure and then after the deformation what is going to happen once you give the electrostatic signal. So what is the capacitance of that? So you have, this is the capacitance that the original geometry that this will give and this is capacitance that the deformed geometry will give is going to change.

(Refer Slide Time 13:32)



So it's very important that the mesh actually change so that it will get reflected as a form of the capacitance. Okay. So let us see how does that varies.

So over here they do a parameter sweep of different, different pressure. However, let us go with a very simplified way. Let us give 0 and 25,000 just to see how the pressure varies.



So you can also see the convergence plot over here. Usually, the errors go below 10<sup>-3</sup> to get it resolved. Okay. So we have the results. So what I wanted to showcase you are the mesh. So I would just disable this. I will first plot the mesh. So you can see that mesh is also there. If you want only mesh the, and we want to visualise the mesh. So I just make a volume mesh and this is the actual mesh, right, that is coming from the mesh over here.



So this is the actual mesh that we have done over here that is using the map mesh. So we actually have the size node. We first mesh the bottom part and then we sweep it to the top part. Okay.

So let us go with the mesh of this particular geometry. So in the first one also we have two values. So you can see that this is for 0 and then this is for 25,000 kilo, sorry, Pascals of pressure. So you can see some deformation that was there, right? So you can see the change in the mesh and this change in the mesh is then taken into consideration while we do the simulation.

If you want to see it side by side that also is possible. Just duplicate it and have some kind of deformation to shift it towards the right side that is in order of 1 mm. So I just move it to the right side, so I can visualise both of them together. So in the mesh 2, I just write it as 25,000, right?

(Refer Slide Time 15:56)



So if you want to just maximise it, so you can see that on the left side is for no deformation and then on the right side it's with the deformation. So both are having a different mesh. Okay.





So you have the potential plot over here and then the deformation plot.

(Refer Slide Time 16:24)





And then you can also vary the pressure from 0 to 25,000 and then see how much the change in displacement is going to occur at a particular point. Okay. You can also compare the capacitance with the analytical capacitance that is available as an equation and then see how much is it compared with this, right?

(Refer Slide Time 16:51)



So the complete simulation, so now I have done only for the quarter part, but if you want to simulate for the other part or the complete structure, you can also mirror the dataset. To mirror the dataset, you can just go to dataset and use 3D mirrors.

#### (Refer Slide Time 17:11) COMSOL M σ E Mesh $\otimes$ Pi - Variables -The Import 200 200 A ..... 88 = IC Its Functions • Build ne Study Model Builder Settings hics Convergence Plot 1 1 T . T . . . . . . . . QQAH V·EEEC 0 H E 2 AL Size · Plot 0 Mapped Swept 1 Label: Study 2/54 1 Study 1 Solution Step 1: Statio Solutio • 10 Study 2 Study 3 Compc • Frame Spatial (x, y, · Study 1/Solution 1 (sol1) Study 2/Solution 2 (sol2) Mirror 3D 1 Study 3/Solution 3 (sol.2) Mesh 2 Defo lectric Potential (es) × Progress Log Table × om Dis 4 🖸 🔚 🕂 🌍 🕥 ~ 10 (1) ENG 27-02 O Type here to search

So if this is my actual geometry, this is my actual geometry that you can see over here, I can use the mirror to actually make it half of the geometry. I can again use the mirror. So I can just use 1 more mirror. So let me go ahead and add one more mirror and use my study 2 study mirror, this one, and then I have to choose this zx-plane and I get the complete solution.

So I actually model only a quarter part, but I get the result for the complete simulation of the complete switch. For example, now if I want to see the complete switch, I can just change it

to my mirror 3D 2 dataset that is nothing but this dataset. Just use this one and just plot it and I get the result for the complete structure where I have only modelled for a quarter of the structure.



Okay. So this is how the pressure sensor work. You give a particular load which changes the geometry of your structure that means the meshing in the case of numerical software such as COMSOL and then on that changed geometry we give electro, electrical physics and from there we understand what is the capacitance.

The way to evaluate capacitance is very simple. Just right click on the Derived Values, Global Evaluation and just search for capacitance. You can see that you can get the capacitance with the value of esC11. So we solve for two pressures.

(Refer Slide Time 18:56)



So as you can see over here, the capacitance for two pressure is changing. I can make it as a unit of picofarads and then again evaluate. So as, as expected the d value is going to that the distance between the two plates is reduced because of more pressure and it is inversely proportional to the capacitance. That's why the capacitance is increasing as the pressure is increased.

You can also go to have a line graph of how it represents the change in capacitance based upon the pressure as you can see over here.



(Refer Slide Time 19:34)

(Refer Slide Time 19:40)

## Biased Resonator (2D and 3D)

- In this sequence of tutorials, an electrostatically actuated MEMS resonator is simulated. The device is biased with a DC voltage. And then driven by a smaller AC voltage. A series of tutorials shows how to compute:
- · The biased displacement
- · The pull-in voltage
- · The biased resonant frequencies
- The frequency domain response



The next example is biased resonator. This is an actuator beam kind of an example where we give a particular pull-in voltage and because of that pull-in voltage how much deformation or displacement or actuation of that cantilever beam is going to take place has been modelled.

So if you want to see that particular model, you can just go over here. I think it's available in MEMS module somewhere. So let me just go through. Yeah. In the MEMS module, in the actuators, yeah. So we have made different kinds of analysis on biased resonators. So you can have particular frequency models where you want to know what kind of modes with which it is going to fluctuate.

We have pull-in of this voltage, I mean, set a particular voltage where it's going to touch the bottom plate kind of thing analysis. Then we have 3D structures again, right? So you can actually open any of that model. First, go to the documentation and then open the model. Okay.

So I will just open that particular model, one of the model and explain it.

(Refer Slide Time 20:51)

◎ [] ● ■ ■ + ちさ用的用き業業長・[			bias	ed_resonator_3d_p	ull_in.mph - COMSOL Multiphysics		- O	×
File Home Definitions Geometry Materials P	hysics Mesh	Study	Results Developer					
Application Builder Application Application Model Model Pir * Variables Parameters Profession Parameters Profession Parameters Profession Parameters Profession Parameters Profession Parameters Profession Parameters Profession Parameters Parameters Profession Parameters Param	Geometry	Add Material	Solid Add Mechanics - Physics Physics	Build Mesh Mesh Mesh	Compute Stationary Study	Displacement Add Plot (solid) 1 • Group • Results	Windows Reset Desitop	
Model Builder	Settings		* 1	Graphics Co	onvergence Plot 1 ×	N Geos		- 1
Biased_resonator_ld_pull_in.mph (root)     @ Global Definitions     P. Parameters 1     @ Materials	Label: Compo	ment 1		* 🖸 🖾 🗄	ο (1) (1) (1) (1) (1) (1) (1) (1) (1) (1)		° .	0
Polit Component 1 (comp1) Polin Stationary Prom Pull In 1 28 Results	General Unit system: Same as glob	al system			.10		3 1 μm -1	
	Spatial frame First	coordinates Second	Third	y and a				
	x Material fram	y e coordinate	1					
	First X	Second	Third	Messages ×	Progress Log Table			~ 3
	Geometry fra	me coordinat	les	(Feb 27, 2019 3.2	6 PM] Formed union of 2 solid object	cts.		~
	First	Second	Third	(Feb 27, 2019 3:2 (Feb 27, 2019 3:3	edges, and 45 vertices.	and 45 vertices.		
	Mesh frame o	coordinates	, cy	(Feb 27, 2019 3:3 (Feb 27, 2019 3:3	riements.			
	First	Second	Third	[Feb 27, 2019 3:37 PM] Complete mesh consists of 1600 domain elements, 1750 boundary elem [Feb 27, 2019 3:43 PM] Opened file: C\Program Files\COMSOL\COMSOLS#(Multiphysics_copy1)			0 boundary elements, and httphysics_copy1/application	507 ons'
	Xm Farmetry share	Ym order	,Zm	¢				5
			1.84 GB   2.03	GB				
Type here to search	0	P5	(3) (3) (3)			^ <b>10</b> A	41) ENG 1543 27-02-2019	5

Again, the geometry part is been important. Then the material part. So here we have used polycrystalline silicon over here. So if I enable the wireframe rendering, it's polycrystalline silicon that you can see over here and then we have silicon nitride in the bottom part and silicon oxide over here and the rest of the domain is air domain. Okay.

In structural mechanics or solid mechanics, we give a particular fixed constrain in the bottom part and symmetry on this part and then on electrostatics we give a particular voltage and then ground on the bottom and terminal on this particular boundary. This is little bit more complex. To start with, I will suggest you to go to the first example of 2D resonators. Okay.

(Refer Slide Time 21:53)

## Surface Micromachined Accelerometer

- This tutorial shows how to simulate a capacitively actuated surface micromachined accelerometer, using the Electromechanics Interface. It also demonstrates how to build up a complicated geometry from a number of individual geometry subsequences (linked from an external file).
- The example used is based on a case study from the book Microsystem Design by Stephen D. Senturia (Kluwer Academic Publishers, 5th Edition, 2003, pages 513-525).



So Let us go ahead and this is an example model of an accelerometer. So this is also available in the Application Library of COMSOL. So if you want, you can just go. So this kind of devices are usually used to understand the acceleration of a particular device, maybe your mobile phones, even, so there are many games where which accelerometer devices have been used in cars, in vehicles, in aircrafts, accelerometers have been used a lot and now they are also moving from accelerometer to SATNAC (inaudible), so which actually works on the basis of optics, but in this case it's purely (inaudible) mechanical approach.

So let me just go ahead and open this particular model. I have this model over here somewhere. Yeah. So we have an example of surface micromachined accelerometer the geometry of which is little bit complex.



So we have the proof mass, the folded spring, the building blocks that is the electrode arrays, both the electrode arrays and then the analysis part. So let me just open this model.

(Refer Slide Time 23:10)



So this is the structure that you can see over here. Geometry part has been modelled over here. So it has been introduced from the geometric parts.

So there are many structure that are already available in COMSOL. So if you want to understand what are those materials, you can just right click on the Global Definition and go to Geometric Parts and go to Part Libraries. So if you go to Part Libraries, there are many structures that are already available.

For example, you are looking for lens, a different some kind of a lens. Then you can actually have the lens introduced from here directly from geometry and based upon the focal point, the complete design of lens is going to change. If you are talking about some kind of beams and bolts, you can also look, try to install from here. Okay. You have codes, magnetic codes, different kinds of codes, magnetic, that is also possible. Different kinds of coils are also possible as you can see over here.

(Refer Slide Time 24:09)



So over here the model has introduced these kind of domains from the Part Libraries directly and from there they have intro, that have called in this particular interface and then applied the material properties. There is air and polycrystalline in the bottom and then finally the physics part, that is the solid mechanics part that talks about a particular fixed constant. So we give a fixed constant at the bolts that you can see over here, the blue part over here, right?

I I II	(別代本・) surface_micro Anterials Physics Mesh Study Results Developer	machined_accelerometer.mph - COMSOL Multiphysics	- 0 X
A Component Pi Pi ** Variabit Builder Model Parameters ** Parameters *** Parameters ************************************	s • Build Convertex • Add Add Material Ceconverty Materials • Physics	Image: Add Mesh         =         ~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~	Indows Reset Desktop - Layout
Nodel Builder	Settings	Graphics Convergence Plot 1	
A Geometry Parts     B Proof mass with Ingers     B Proof Proo	Label: Fixed Constraint 1 Boundary Selection Selection Anchor plane   Active  Active  Active  Active  Contrapplicable  78  97  119  Override and Contribution  Equation	Mesages Progress Log Table	

So these are the different kinds of fixed constant. Here there are many. You can see over here. So all, all of them are bolts. That's why fixed constants have been given. The blue part signifies the fixed constant.

Then the symmetry part because we have modelled only half of the accelerometer, not the complete accelerometer. That's why we have given you a symmetry boundary condition over here. So it will in the post-processing we can actually reflect what is on the left side to the right side with the same results.

(Refer Slide Time 25:2	26) ( 19 12, - ) surface, m	sicromachi	ined_accelerometer.mph - COMSOL Multiphysics	- 0 ×
Application Model Definitions Geometry M Application Component Model Definitions	Laterials Physics Mesh Study Results Developer Is* Is* Build ColliveLink+ All Converty Materials Developer Materials Developer Add Materials Developer Physics Physics Physics	Build Mesh Me	Meth 1- Compute Study 2: Add Set Flet: Study Study Compute Study 2: Add Compute St	t Windows Reset Desktop - Layout
Model Builder	Settings Electrostatics Labet: Electrostatics Name: es Domain Selection Selection: All domains 1 Active: 3 4 5 6 Equation Manual Terminal Sweep Settings Electrostatics	* * *		200 200 200 200 200
	Zee [Solonn]     Activate manual terminal sweep     Physics-Controlled Mesh     Enable		Messages × Progress Log Table ×	*4
Tune here to search	1.86 GB   2.06	G8	A 50	at di) ENG 1548 💽.

Then the electrostatics part. Then we have sense terminal and floating potential as we discussed before to withdraw a particular voltage. Okay. Then we have Multiphysics coupling with couples the solid mechanics with the electrostatics, and then we have two types of study. Both are stationery, but, however, first is normal operation and second is for self-testing the details of which you will find it in the documentation.

(Refer Slide Time 25:54)

# Thermal Actuator

This tutorial example of a two-hot-arm thermal actuator couples three different physics phenomena: electric current conduction, heat conduction with heat generation, and structural stresses and strains due to thermal expansion.



Okay. So this is the last example for the actuator is thermal actuator. In this it works on the principle of thermal expansion. So let me just go and open this particular example. So I just search for thermal actuator and you can see over here there are two types of thermal actuators and along with a particular tutorial model also.

So let me just open this documentation. This is what demos that we will do in this session.



This is a thermal actuator that you can see over here and these are the anchors. This is the cold arm. The dimples are there with which it's going to get attached with. These are the main two hot arms are where the temperature would be rising because of Joule heating and these are the boundary conditions.

## (Refer Slide Time 26:45)



We give a particular voltage and ground it on the other side. We also give some temperature boundary condition that is fixed temperature on these boundaries and this boundaries and fixed temp at the dimples that you can see over here.

### (Refer Slide Time 26:58)



And then we give roller boundary condition so that it can actually roll while it's moving along laterally like up and down in this case.

Okay. So just open this model. We are also going to do a demo on this model. That would be after the session and here you can see it's made of polysilicon, right?

Aglication whether is the state in the state is a spectra whether is a spectra whether is the state is a spectra whether is a spectra whether is the state is a spectra whether is the spectra whether i	こ ● 日 日 ・ つ C* 1例 IC 用 日 国 Hic • Home Definitions Geometry M	」號 掲 ・ l faterials Physics Mesh Study	Results Develop	mal_actuator_sir	implified.mph - COMSOL Multiphysics	- 0 X
Model Builder     Settings     Material     Image: Common Model Inputs	A Component Pi * Variable Reserved Pi * Variable Reserved Parameters * Reserved Paramete	s - Build CaliveSalk - Ad Al Geometry Mate	Ad enial enial enial enial	Build M Mesh 1	Lesh Compute Study Add 1 Study Study Results	dd Plot Windows Reset Destop - Layout
Active     Material     Ma	vlodel Builder	Settings		~ 1	Graphics Convergence Plot 1	
Parameters 1   Common Model Inputs   Materials   Dermal Actuator (compl)   Definitions   Connective entity level:   Dermal Actuator (compl)   Definitions   Def	thermal_actuator_simplified.mph (root)     Global Definitions	Material Labet: Polysilicon				
Geometric entity level Domain  Geometric entity  Geometric entity level Domain  Geometric ent	Parameters 1     Gommon Model Inputs     Materials	Geometric Entity Selection				a
<ul> <li>Monthey 1</li> <li>Materials</li> <li>Popolatics Currents (ex)</li> <li>Material Transfer in Solids (ht)</li> <li>Material Transfer in Solids (ht)</li> <li>Material Properties</li> <li>Override</li> <li>Material Properties</li> <li>Material Contents</li> <li>Metal C</li></ul>	Thermal Actuator (comp1)     E Definitions     Vi Geometry 1	Geometric entity level: Domain Selection: All domains				
Image: Solid Mechanics (not)       Image: Solid Mechanics (not)	Geometry 1      Geometry 1	Active 1		100		
▲ Meih 1       >       Override       150       µm         ▲ Weih 1       >       Override       µm       150       µm         ▲ Beautis       >       Material Properties       >       150       µm         ▲ Bit Stables       >       Material Contents       >       µm       µm         ▲ Bit Stables       >       Material Contents       >       µm       µm         ▲ Bit Stables       >       Material Contents       >       µm       µm         ▲ Bit Stables       >       Bestice permittivity signal, Sign	Heat Transfer in Solids (ht)     Solid Mechanics (solid)     A Multichusics	ALC: N		\$	0	50 100
Image: Second	Mesh 1	1. Override				150 µm
<ul> <li></li></ul>	A 🕷 Results	Material Properties			40 20 0 200	
In Tables     **     Property     Variable Value     Unit     Property       Image: Section (Detential (ec))     **     Property     Variable Value     Unit     Property       Image: Section (Detential (ec))     **     Property     Variable Value     Unit     Property       Image: Section (Detential (ec))     **     Property     Sigma (f)     Sim     Basic       Image: Section (Detential Conductivity)     Sigma (f)     Sim     Basic       Image: Section (f)     Sim     Sim     Basic       Image: Section (f)     Sim     Sim     Sim       Image: Section (f)     Sim     Sigma (f)     Sim       Image: Section (f)     Sim     Sigma (f)     Sim <td><ul> <li>It Derived Values</li> </ul></td> <td>* Material Contents</td> <td></td> <td></td> <td>μm</td> <td></td>	<ul> <li>It Derived Values</li> </ul>	* Material Contents			μm	
Constant press     Cp     Creation press     Cre	M Tables     ME Electric Potential (ec)     M Temperature (ht)	Property  Electrical conductivity	Variable Value Uni sigma sigma(T) S/r	it Prop	A	
	Stress (solid)     Sport     Sport	Heat capacity at constant pres     Relative permittivity     Denvity	Cp 678(J//k J/(k epsilo 4.5 1 ma 2320/km kn/	Ig-K) Basic Basic Im <sup>4</sup> Rasic	Messages × Progress Log Table 2 ×	-

And then we give, so here there are three physics. Earlier we used to work with only two physics. Now it's addition of the heat transfer physics also to take into account the temperature rise and because of the temperature rise, the thermal expansion. So this is the electric current interface with which we give a particular voltage in this particular boundary, okay, and then ground at this particular boundary. Okay.

(Refer Slide Time 27:46)	-   them	al_actuator_sin	mplified.mph - COMSOL Multiphysics	- a ×
Home         Definitions         Geometry         Materials           Application Builder         Component Model         Pi Parameters         In Functions - In Functions - Definitions           Application         Model         Pi Parameters         In Functions - In Functions - Definitions           Model         Builder         In Functions         In Functions           Model         Parameters         In Functions         In Functions           Model         Parameters         In Functions         In Functions           Model         Parameters         In Functions         In Functions           Image: Parameters         In Functions         In Functions         In Functions           Image: Parameters	Physics Meth Study Results Develope Build Columnities Geometry Materials Physics Settings Ground	Build Ma Mesh 1 Mesh	sh Compute Study Add 1. Study Study Competence Plot 1 Competence	Windows Reset Desktop Layout
	Boundary Selection Selection Ground I      Active Override and Contribution	+ + = 36 0	μm 100 200 0 0	
Crown 1  Cr	Equation 5		A A A A A A A A A A A A A A A A A A A	
27:45 Express Scri COMSOL Exampl	Google - Google			≏ 🛱 atl 10:24 PM Tuesday

And then in the heat transfer we give temperature boundary condition that is the room temperature at the, these points and then we give heat flux. So it will be also getting cooled with a particular conviction, so convictive cooling. So that is defined as HTS, HTC\_S. This

HTC\_S had been defined somewhere over here as .04 W per millikelvin over here, right? So this is heat transfer co-efficient.

So that is the upper part and the bottom part is also getting cooled up with a different heat transfer co-efficient that is HTC\_US where HTC\_US has been defined over here. So that's divided by 2. Okay. Let me just open it. Yeah. So it's 0.04, the upper part, .04/2 micrometers and the bottom part is 0.04/100 micrometers so the top part is more as is getting cooled quickly, quickly as compared to the bottom part.

The last part is the solid mechanics part. Of course, we are going to give some kind of fixed constant. It's even somewhere over here as you can see over here in the bottom.



This is the three places where fixed constants have been given and then the electromagnetic heating is taking into consideration. That is this exactly couples the electric currents with the heat transfer in solids. Okay. And you can just go to study and get the results. The results actually looks like this.

(Refer Slide Time 29:28)



So you will see that the deformation is actually going to takes place like this. We have given a roller boundary condition. That is why it cannot move up and down, but it is allowed to move in Y direction, right? It is not allowed to move in Z direction, but it is allowed to move in Y direction. That is laterally it can move, right? This is the actual geometry that you can see in the black lines and the deformed geometry that you can see is at the red structure that you can see. Okay.

(Refer Slide Time 30:05)

# Fluid structure interactions



The next part is the fluid-structure interaction. So we talked about thermal. We talked about structural. We also talked about electrical a lot, but what about coupling fluid, fluid dynamics along with the structural mechanics?

# **Fluid-Structure Interaction**

This example shows how to model fluid-structure interactions (FSI) using the MEMS Module. Viscous forces and the system's pressure impose forces to the surface of a structure. The deformation in the soft structure is not small and the fluid regime will therefore change. This means that changes in the structure and the fluid dynamics are coupled.



So we have a couple of examples to do that. One of the example is a very simple flow of fluid through a channel from left to right, and there is an elastic material as you can see over here and it is going to bend. So, again, when we talk about the bend, we also need to take into account the change in the geometry with the help of change in the meshing. So you can see that the meshing is also going to change in this particular domain.

So let me just go and open this particular model. So let me just go over here and I go to the MEMS. I think it's available somewhere over here. Fluid-structure interaction, yes, it's available over here. There are two examples. One is fluid-structure interaction. Another is micropump.

(Refer Slide Time 31:04)



In the Fluid-structure interaction, if you just open this documentation, you will see the Physics part and this is the most important part that I want to signify is the change in the mesh.

(Refer Slide Time 31:16)



So you can see this particular pillar is having a different mesh as compared to the original mesh which was actually vertical. So this is not any more vertical. It's actually tilted with a particular angle and that would only happen if you apply moving mesh or a deformed mesh.

So just quickly go through the physics behind this. So this is the laminar flow. This is the particular structure. We have two materials. One is the may be water I guess and the second is the structural material over here. This is the pillar and we give an inlet fluid from the left side

with the particular flow velocity and outlet on the right side. So the fluid is going to move from the left to the right and the forces are going to get exerted by this pillar.

Home Definitions Geometry Materials	Flysics Mesh Study Results Developer	action.mph - COMSOL Multiphysics - O
A plication Component Model Pi ** Variables ************************************	Stringort All         Stringort Add         Stringort Add         Stringort Solid Methnics         Stringort Physics         Stringort Build Methnics         Stringort Methnics         Stringort Methnics <th< th=""><th>Mesh Compute Study Add 1- Study Study Add Poto Study Study Add Poto Study Study Add Poto Study Study Add Poto Results Layout</th></th<>	Mesh Compute Study Add 1- Study Study Add Poto Study Study Add Poto Study Study Add Poto Study Study Add Poto Results Layout
odel Builder	Settings - •	Graphics Convergence Plot 1
Gobal Definitions     Parameters 1	Fixed Constraint Label: Fixed Constraint 1	
Variables 1     Materials	▼ Boundary Selection	μm 70
Component 1 (comp1)     Definitions	Selection: Manual •	60
Geometry 1     A Geometry 1     A      A	5 5 ÷	40
<ul> <li>Material 1 (mot1)</li> <li>Material 2 (mot2)</li> </ul>	Active	30
<ul> <li>Laminar Flow (sp0)</li> <li>Fluid Properties 1</li> </ul>	*	10
Initial Values 1 Wull 1	<ul> <li>Durable and Contribution</li> </ul>	
iniet 1     Outlet 1	Equation	-20"
<ul> <li>Solid Mechanics (solid)</li> <li>Linear Elastic Material 1</li> </ul>		-30"
<ul> <li>Free 1</li> <li>Initial Values 1</li> </ul>		-50"
Fixed Constraint 1     Multiphysics		-60 <sup>-1</sup> 50 100 150 200
A Mesh 1     Study 1		Messages × Progress Log Table ×
E & Results		1

Such a mechanics is used to model the deformation of this pillar. A fixed constant has been given in the bottom part, so it cannot move. This particular boundary cannot move. The other part are free to move, right? The deformation is given using moving mesh. This is the most important part that, that needs to be taken into consideration. That is the move, the mesh is going to change, right?

(Refer Slide Time 32:34)

# Micropump Mechanism

This example shows how to model fluid-structure interactions (FSI) using the MEMS Module. Viscous forces and the system's pressure impose forces to the surface of a structure. The deformation in the soft structure is not small and the fluid regime will therefore change. This means that changes in the structure and the fluid dynamics are coupled.



Fluid flow and von Mises stress within a passive microfluidic flow rectification system. A pumping mechanism is drawing fluid up into the vertical shaft from the horizontal channel. The channel contains two tilted flaps which respond to the fluid flow by bending. In this case, when fluid is drawn into the vertical channel, asymmetric bending of the flaps results in a much larger flow from the left hand channel than from the right channel. The next example is going to be the micropump mechanism example in which case we have an inlet on this left side and there is a pump on the top, and it's going to, the main importance of micropump is that it allows the flow to in only a particular direction, right?

So over here there are two pillars that you can see which are actually bending, and based upon that it's going to store the fluid at the top and then it's going to again refill. So the fluid is going to be filled on the top and then again it's going to move towards the right side, but the back flow is kind of restrained.

So let me just open this particular model, how to this model, a very similar approach to what we saw before. So this is again the documentation. So I would suggest you to go through the documentation. This is the actual part.



So this is the actual input and we allow, you want to flow, have the flow in a particular direction, right? Again, the mesh is what is going to change at every instant.

So let me just open this particular model. So you can see this is the time dependent simulation. So at each time step the mesh is going to be changed. Again, a laminar flow to understand the flow of fluid. Inlet is given over here. Outlet is given on the both the sides, and then the structural mechanics to actually have the fixed constant on the bottom part and the two pillars, which are having a particular angle is modelled using a particular value as you can see over here.

(Refer Slide Time 34:15)

●   □ ● ■ ■ ● + ちさ出り用き用意味	micropum	p_mechanism.mph - COMSOL Multiphysics	- 0 ×
File + Home Definitions Geometry Materials	Physics Mesh Study Results Developer		
Application Builder Application Application Application	Callivelinik - Add Geometry Material Geometry Physics	Build Mesh Nesh 1+ Mesh Study Study Study	Windows Reset - Desktop - Layout
Model Builder	Settings Material	Graphics Convergence Plot 1     Q. Q. Q. ⊕ ⊕ □ ↔ Ø ⊕ ■ ⊖ ⊖ №	- 1 ( ) ) ) ( ) ( ) ( ) ( ) ( ) ( ) ( ) ( )
Global Definitions     Parameters 1     Component 1 (comp1)	Labet: Fluid Geometric Entity Selection		
<ul> <li>▷ Definitions</li> <li>▷ A Geometry 1</li> <li>▲ ● Materials</li> </ul>	Geometric entity level: Domain Selection: All domains	• 250"	-
Polace (mart)     Polace (mart)     Polace (mart)     Laminar Flow (pp)     Polace Provide (mart)     Polace Provide	Active 3	+ 150"	-
Initial Values 1     Wall 1     Ginet 1	4 (overridden)	50-	
Solid Mechanics (solid)     Solid Mechanics (solid)	Di Override	0	
<ul> <li>Enter Classic, Haterian F</li> <li>Free 1</li> <li>Initial Values 1</li> </ul>	Material Properties     Material Contents	-50	-
Fixed Constraint 1     Global ODEs and DAEs (pe)     Multiphysics	Property Variabik Value	Unit -100 -200 -100 0	μm 100 200
P ▲ Mesh 1     Study 1     N. Step 1: Time Dependent	Dynamic viscosity mu visc	Pas Messages × Progress Log Table ×	×.1
	1.88 GB   2.05	58	
Type here to search	J O 🔚 🕾 🌍 🗐 🤮	~ *	● _4 _4 0 ENG _27-02-2019 5

The rest of them is the fluid, which is defined by dense and visc, which is defined in the parameters. So you see the dense and the visk, that is the velocity and the density have been defined over here. This is to understand with what velocity is the fluid is moving and then finally the mesh that you can see over here.

	micropump_mechanism.mph - COMSOI, Multiphysics - Ø X
Home         Definitions         Geometry         Materials           Application Builder         Component Model         Pri Parameters         ** Variables           Pri * Parameters         ** Interfere Case Definitions	Global ODEs and DALs     Meth     Study     Results     Developer       Image: Contractions - Add Meterials     Add Meterials     Image: Contractions - Add Meterials     Image: Con
Model Builder     - i       ★ = 1 + # ★ If #↓ #↓ ★       ▲ @ Component 1 (comp.l)       ▶ Definitions       ▶ % Geometry 1       ▲ @ Materialis	Settings Convergence Piot 1 Casethies Converg
P ● Fluid (mmt2)     Solid (mmt2)     Solid (mmt2)     Earlinar Flow (pp)     Fluid Properties 1     Fluid Properties 1     Wall 1     Outlet 1     Outlet 1     Outlet 1     Outlet 1     Fired Tourstant 1     Fired Constraint 1     Global OpEs and DAEs (pp)     Global Equations 1     Matter 1     Global Equations 1     Matter 1	Sequence type: User-controlled mesh
All Suze 1 Free Triangular 1	
a too proof i	1.88 GB ( 2.05 GB
Type here to search	↓ □ 🛅 📴 🌍 😰 🤮 🚺 🚼

This is the actual mesh and then the deformed geometry using the moving mesh. Okay. And then you do a time dependence simulation to understand if it is moving with a particular, what is the flow rate that you can see from here. Okay.

(Refer Slide Time 34:53)



So next is the demo on thermal actuator. So I think that's how the time I have right now, right, till 4 o'clock. So I think I will stop.