# What's new in MEMS Pro V8.2 and v8.4 Highlights

so

# **Platform Support**

#### MEMS Pro v8.2 supported OS:

- ≻ Windows:
  - *XP*
  - Windows 7
  - Windows 8
- ➢ OS types
  - 32 bits
  - 64 bits

## 64 bit engine

Larger models can now be made

> > 4GB can be address

Engine has no memory limit



### SoftMEMS API

- 3D Models can be made from inside a "C" program or inside Matlab
  - ➢ Supports GDS read in
  - ➢ Fabrication process read in from a file
  - ➢ 3D Model generation
  - Cross-section generation

//ConsoleProgressHandler consoleProgressHandler; ModelGenerator \*mg = new ModelGenerator(); mg->generate3DModel("testGds.gds", "testGds.mpd", "gds.sat", "Cell0"); return 0;



#### **SoftMEMS Batch mode**

- 3D Models can be run from a batch script on UNIX, LINUX or PC to generate large models offline
  - ➢ Supports GDS read in
  - ➢ Fabrication process read in from a file
  - ➢ 3D Model generation
  - Cross-section generation

# Import of model from previous processing

 Users can read in a file containing geometry and continue processing it with \_ adding more process steps



#### **Process enhancements**

#### Enhancements to etch

- Rounded corners for etch
- ➢ Example: Ga etching





# **New Wafer Bonding Feature**

Wafer bonding now
 supports overlapped
 wafers for bonding
 surface features to
 cavities



Genera	l Layers	Materials	Preset Wafers	3D Process Steps	2D Process Steps	
De	tect True Curv	es	Show	Intermediate	Units: micron	×
Step						
#	Label			Step Name: Wafer	Bond1	
1	Wafer			Commands Wie Com		
2	Wafer 2		_	Command: Waterbo	and 👻	
3	Deposit Poly0			Wafer ID:	W2	
5	Etch Poly0		_			
6	Deposit Metal		Ť	Top Wafer:	W1	•
7	Mechanical Po	ilish		Bottom Wafer:	.w/o	
8	Deposit Poly 1 Etch Poly 1			bottom water.	140	•
10	Sacrificial Etch	1		Distance:	0	
11	Deposit Metal					
				Flip Top Wafer	Flip Buttom V	Vafer
			•			
2						
	V Enable	📃 Display	0			
	Add Step	- Delete S	step			
						i
Com	nent:					
Comr	nent:					
Comr	ment:					

# Compact Model Builder v1.0 (v8.4)

soft

Sebastien Cases

#### What's New ?

#### New Interface

- Multi Project
- ConcurrentSimulations
- Several Solvers
   Configurations

SoftMEMS - Compact Model Builder		
<u>File E</u> dit <u>S</u> etup <u>V</u> iew <u>W</u> indow <u>H</u> elp		
Projects 🗗	model STR 1 DOF	
Projects     Image: Project series <ul> <li>Model Name:</li> <li>Reduced Order Model Solver:</li> <li>Ansys</li> <li>Physic:</li> <li>Structural</li> <li>Model Name:</li> <li>model_ES_STR</li> <li>Type:</li> <li>Reduced Order Model Solver:</li> <li>Ansys</li> <li>Physic:</li> <li>ElectroStatic-Structure</li> <li>Model Name:</li> <li>model_FSI</li> <li>Type:</li> <li>Reduced Order Model Solver:</li> <li>Ansys</li> <li>Physic:</li> <li>ElectroStatic-Structure</li> <li>Model Name:</li> <li>model_FSI</li> <li>Type:</li> <li>Reduced Order Model Solver:</li> <li>Ansys</li> <li>Physic:</li> <li>Fluid Structure Inter</li> <li>Physic:</li> <li>Fluid Structure Inter</li> <li>Solver:</li> <li>Ansys</li> <li>Solver:</li> <li>Model Name:</li> <li>Fluid Structure Inter</li> <li>Physic:</li> <li>Fluid Structure Inter</li> <li>Physic:</li> <li>Physic Solver:</li> <li>Ansys</li> <li>Physic Solver:</li> <li>Solver:</li> <li>Ansys</li> <li>Physic Solver:</li> <li>Ansys</li> <li>Physic Solver:</li> <li>Solver:</li> <li>Ansys</li> <li>Physic Solver:</li> <li>Solver:</li> <li>Ansys</li> <li>Physic Solver:</li> <li>Physic</li></ul>	model SIR 1 DOE         model FS STR         model FSI         File Settings Tools         Solver Fem Model Output Data Motion Setup         Structural Condensation         ElectroStatic Structure         Pluid Structure Interaction	
Model: D:/Training/temp/model_FSI/model_FSI		Λ

soft

New Features

#### **New Interface** ...

#### Based on Wizard to ease new data capture

	?
New Project Setup	? <mark>* × •</mark>
Project Type         Select the type of Operation you want to perform. <ul> <li>Reduction from FEM Model</li> <li>Linear Combination of Models</li> </ul>	New Project Setup         Physics Selection         Select the Physics you need. <ul> <li>Structural Condensation</li> <li>ElectroStatic Structure</li> <li>Fluid Structure Interaction</li> </ul> <ul> <li>Fluid Structure Interaction</li> </ul> Next Cancel



## **Multi Project**

Multiple projects can be opened at the same time

 Projects Explorer allowing to have a quick summary of the model and allowing to easily switch between them

Projects	
Model Name:	model_STR_1_DOF
Type :	Reduced Order Model
Solver :	\Lambda Ansys
Physic :	Structural
Model Name:	model_ES_STR
Type :	Reduced Order Model
Solver :	\Lambda Ansys
Physic :	ElectroStatic-Structure
Model Name:	model_FSI
Type :	Reduced Order Model
Solver :	\Lambda Ansys
Physic :	Fluid Structure Interaction
Model Name:	model_COMBI
Type :	Linear Combination
Model :	accel

#### **Concurrent Simulations**

Multiple simulations can be ran at the same time

Projects Explorer reporting the current activity





## **Solvers Managements**

Different Ansys Configurations can be setup enabling user to easily switch between them in 1 click.

SOTT

 Allows configuration of
 Multiple versions of Ansys
 Ansys running on another computer/server with higher memory and CPU

🔷 Solvers Manager						
Available Solvers :						
Name	Solver	Version	Host	Default		
Ansys_Config1	Ansys	13.0	localhost			
Ansys_Config2	Ansys	11.0	localhost			
Ansys_Config3	Ansys	14.0	ANSYS_SRV			
Add Edit Delete						
				Close		

#### **New Features**

- Structural Systems :
  - Non-Linear Stiffness extraction
  - Support of Pre-stressed geometries
- Electrostatic-Structural Systems :
  - Non-Linear Electrostatic effects extraction

#### **Non Linear Stiffness Extraction**

- Non-Linear Stiffness extraction, also called Spring Hardening Effect
  - Generation of deformed structures in Ansys (NL static solutions) under predefined displacements
  - Resultant Force extracted from Ansys for each sampling point allowing to get an expression of the force in function of displacement  $F = k(x) \cdot x$



• where 
$$k(x) = k_1 \cdot x + k_2 \cdot x^2 + k_3 \cdot x^3 + hot$$

model_STR_1_DOF	
File Settings Tools	
Solver Fem Model Output Data DOFs Load Cases	
✓ Inertia and Damping	model_STR_1_DOF*
No Transient Quality Evaluation	File Settings Tools
Transient Quality Evaluation	Solver Fem Model Output Data DOFs Load Cases
Mass Tuning	DOF Requirements
V Stiffness	Node Label Min Max Sampling Po
Model Setup (Model)	Add     Delete     Clear       Non-linear Stiffness     Number of Sampling Points :     4 i
Number of sampling points	Model Setup     Model     Transcript       Solver : Ansys v130 on localhost:9080     Running

 According to the DOF definition the sampling points are reported

Then Ansys generates the deformed geometries

accel1_model		
File Settings Tools		
3- Non Linear Stiffness Evaluation		
=== Model Deformation Setup ===		
Number of sampling points : 4		
Structural retained DOF values: UZ(N_MASTER)		
0		
3.333333339-08 6.66666667e-08		
1e-07		



 Finally the analysis are performed on each deformed geometry

SOTT

 The force is extracted and the Stiffness Matrices computed

ł	File Cettiene Teele
	File Settings loois
	=== Non-Linear Structure Analysis ===
	Analysis 1/4 Waiting for ANSYS solution Done.
l	········
	Stiffness Matrix:
	[[ 32.9034]]
	Analysis 2/4 Waiting for ANSYS solution Done
	Walking for ANSTS solution Done.
ľ	Stiffness Matrix:
	[[ 32.9633]]
	Analysis 3/4 Waiting for ANSYS solution — Done
	Walking for ANSTS solution Done.
	Stiffness Matrix:
	[[ 32.9632]]
	Analysis 4/4
l	Waiting for ANSYS solution Done.

- The polynomial expression is extracted and a report for each sampling point is displayed.
- The reference stiffness (from Ansys) is compared to the Approximated one (polynomial value) at the same point.

accel1_model					
File Settings T	ools				
4- Non-Linear S	tiffness Accuracy Estimation				
Accuracy actimat	Hony K 1 1				
Accuracy estimation	uon: K_1_1				
Point	Reference	Approximation	Difference		
1	32.9633728	32.9633728	0		
2	32.9633039	32.9633039	0		
3	32.9632372	32.9632372	7.10542736e-15		
4	32.9631742	32.9631742	0		
Mean absolute v	alue = 32.963272				
Maximum absolute difference = 7.10542736e-15(2.15555888e-14 %)					



# The Behavioral model is finally generated with the stiffness set as a polynomial expression

accel1_model		
File Settings Tools		
Model Pin Variables Parameters		
Variables	Properties	
In_N_MASTER_Uz	Name:	K_1_1
Dt_In_N_MASTER_Uz	Expression:	32.9634+3.42903e+08*In_N_MASTER_Uz^2+6.58264e+15*Ir
Dt_Dt_In_N_MASTER_Uz	Interval of Validity	
Out_N_MASTER_Uz	Minimum Value:	
M_1_1	Minimum value.	
M_1	Maximum Value:	0
K_1_1	Comment	
K_1		

 To compare to the expression of the Linear Stiffness Coefficient

Properties	
Name:	K_1_1
Expression:	32.9633728117



#### **Initial Stress Extraction**

- Initial Stress can be applied onto geometrie's material through the 'Use Initial Stress' box
- The values can be entered in the interface for each material or, when it exists, an Ansys ISTRESS file can be loaded

model_str*	model_str*
File Settings Tools	File Settings Tools
Solver     Fem Model     Output Data     DOFs     Load Cases       Use Load Steps Files	Solver     Fem Model     Output Data     DOFs     Load Cases       Use Load Steps Files
Image: Stepsy to the stepsy	From :       10 :       Step by :         Image: Use Initial Stress       Image: Use Stress File         Image: Define Stress Values       Image: Use Stress File         ISFILE File :       Image: Browse
Sxy: -30e6         Syz: 0.0         Sxz: 0.0           Remove         <<< Add	Model Setup /\ Model /\ Transcript /
Solver : Ansys v90 on localhost:9080 Running : 🔘	Solver : Ansys v90 on localhost:9080 Running : 🔘 🔬
SOj	* t M E พ う 27

#### **Initial Stress**

- First, an non-linear static analysis is performed to generate the pre-stressed struture
- Then the Substructuring stage is started from the previous analysis
- $\Rightarrow$  No stress
- Eigen Frequency
- $\Rightarrow$  **Pre-stressed**
- Eigen Frequency

- --- Computing Substructure Eigen Frequencies ---
- Bhv Eigen Frequencies: 861.932426
- --- Computing Substructure Eigen Frequencies ---

Bhv Eigen Frequencies: 1161.20085





 Non-Linear Electrostatic effect, also called Spring Softening Effect

✤ In FEM formulation, the structural model is defined as follow:

$$M\ddot{x} + D\dot{x} + Kx = \sum_{k} F_{k} + F_{coupling}(x, \{V\})$$

The non linear coupling loads term Fcoupling is defined by :

$$\{F_{coupling}(x, \{V\})\} = \frac{1}{2} \{V\}^{T} [\partial_{x}(C(x))] \{V\}$$

\* Where the capacitance matrix [C] depends only on the structural configuration : [C(x)]

softMEMS



New option to allow the non-linear Electrostatic stiffness extraction based on non inverted polynomial expression of the Capacitance

 $C(x) = C_0 + C_1 \cdot x + C_2 \cdot x^2 + C_3 \cdot x^3 + hot$ 

es_beam	- • ×					
File Settings Tools						
Solver Fem Model Output Data DOFs DOFs Sweep Electrical Data Load Cases						
Inertia and Damping						
No Transient Quality Evaluation						
Transient Quality Evaluation						
Mass Tuning						
V Stiffness						
Non-Linear Structural Stiffness Extraction						
Coupled Loads and Capacitance						
Non-Linear Electro-Static Stiffness Extraction						
Structural Loads						

- In both Cases (Linear and non-linear) the deformed geometries are generated.
- Then, Ansys extracts the Capacitance values between the conductors for each deformed geometry.
- The difference is mde only at the behavioral model generation step where, the polynomial expression is defined as being inverted (better static results) or noninverted (spring softening capabilities)

#### Inverted Capacitance expression results:

Accuracy estimation: C_1_1						
Point	Reference	Approximation	Differenc			
1	0.0233312298	0.0233312298	0			
2	0.022135	0.022135	0			
3	0.0211047433	0.0211047433	0			

#### Non-inverted Capacitance expression results:

4- Capacitances Accuracy Estimation Accuracy estimation: C_1_1					
Point	Reference	Approximation	Difference		
1	0.0233312298	0.0233312298	0		
2	0.022135	0.022135	-3.4694469		
3	0.0211047433	0.0211047433	-3.4694		
Mean absolute v Maximum absolu	alue = 0.0221903244 ute difference = 3.46944695e-18(1	.56349537e-14 %)			
	coft MEMS				