

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

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

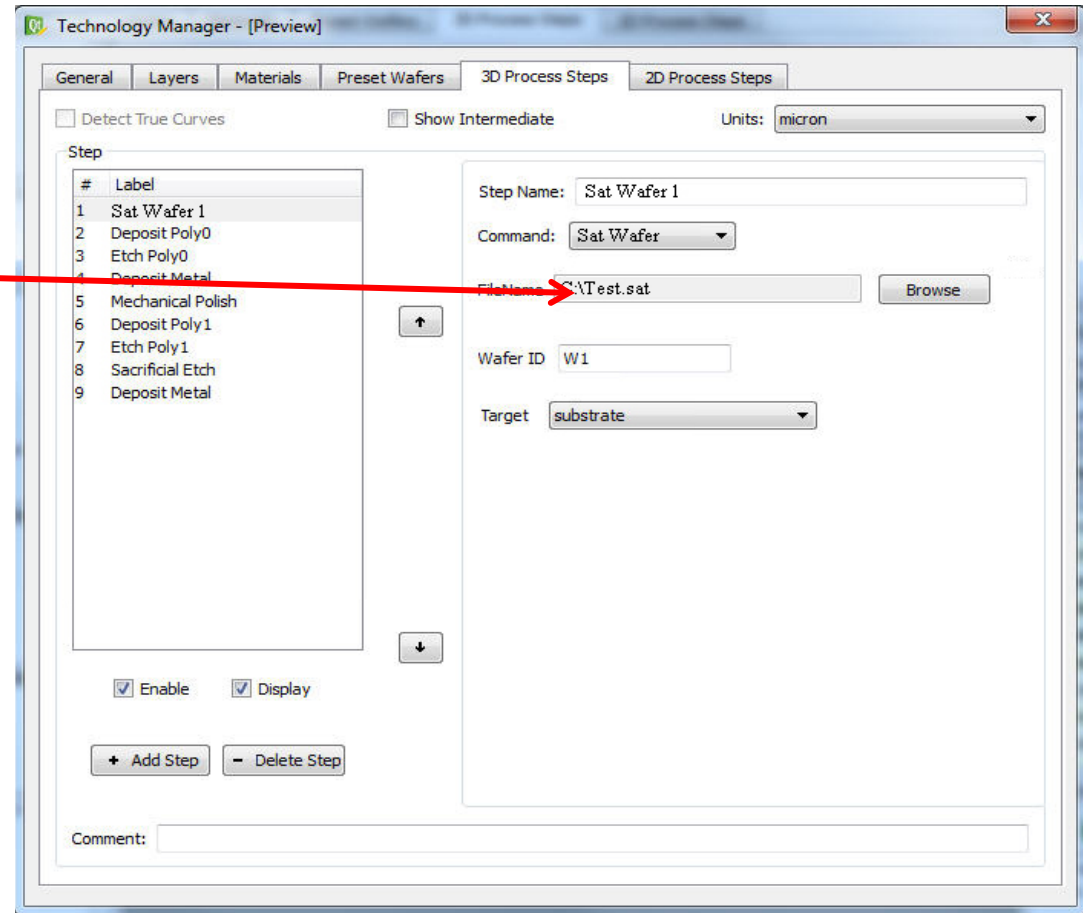
```
//ConsoleProgressHandler consoleProgressHandler;  
ModelGenerator *mg = new ModelGenerator();  
mg->generate3DModel("testGds.gds", "testGds.mpd", "gds.sat", "Cell0");  
return 0;
```

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

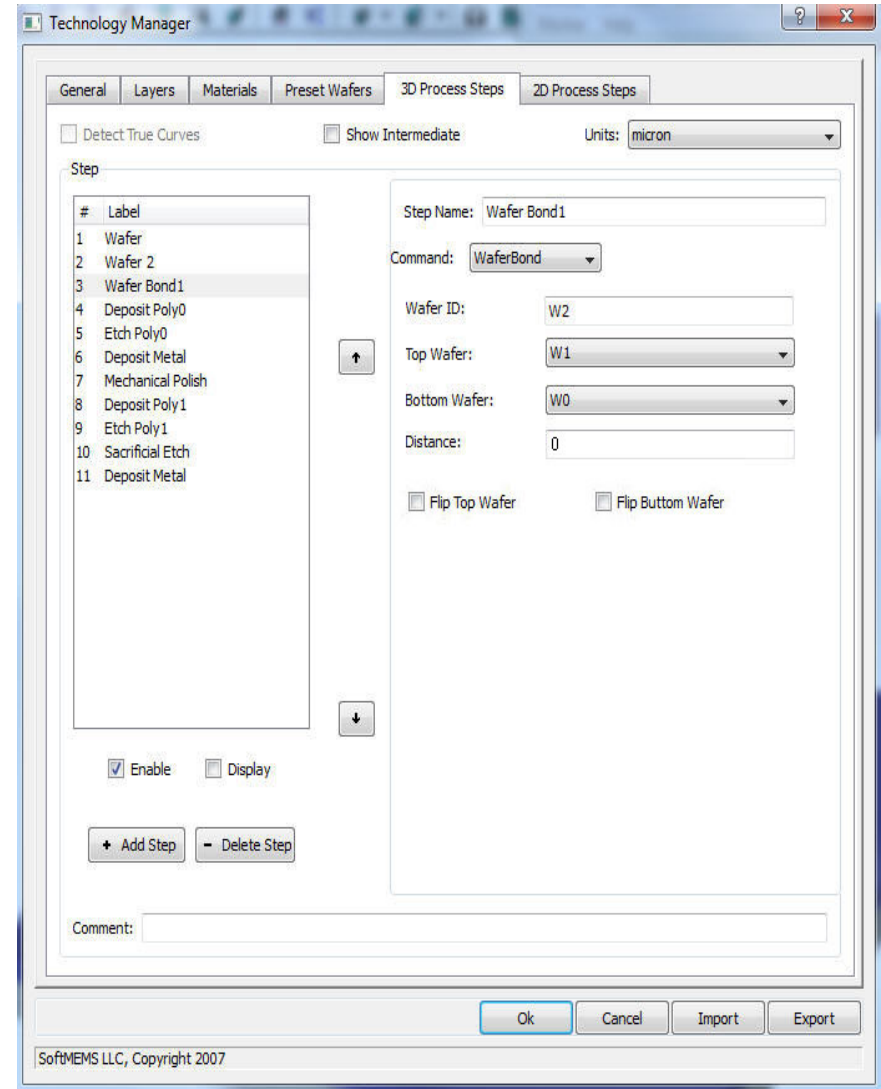
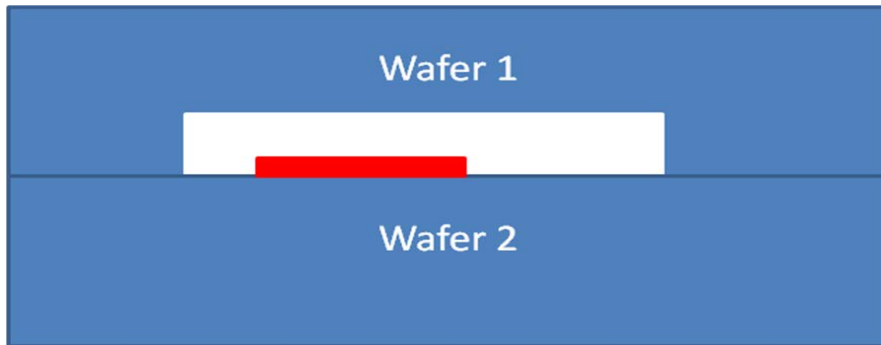
❖ Enhancements to etch

- Rounded corners for etch
- Example: Ga etching



New Wafer Bonding Feature

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

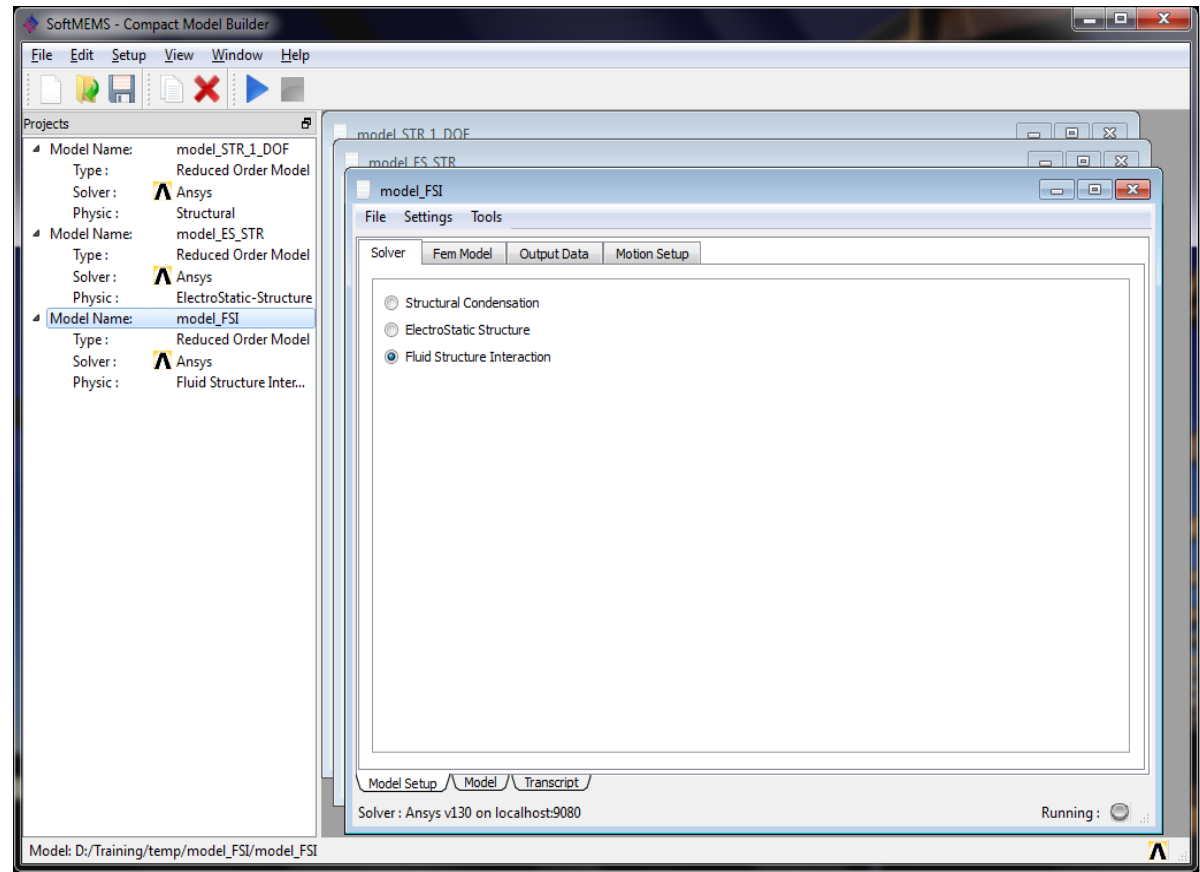


Compact Model Builder v1.0 (v8.4)

Sebastien Cases

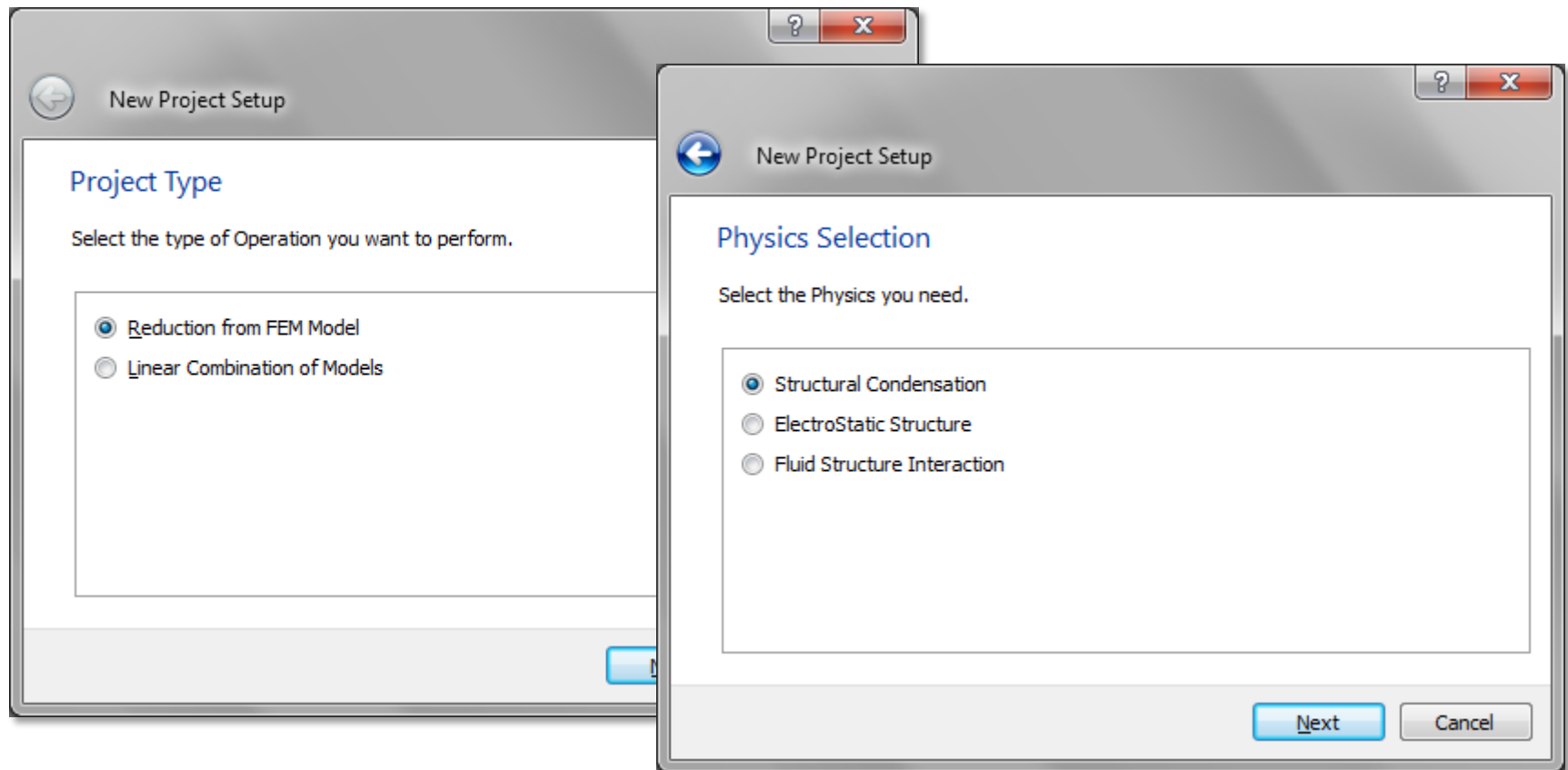
What's New ?

- ❖ New Interface
- ❖ Multi Project
- ❖ Concurrent Simulations
- ❖ Several Solvers Configurations
- ❖ New Features



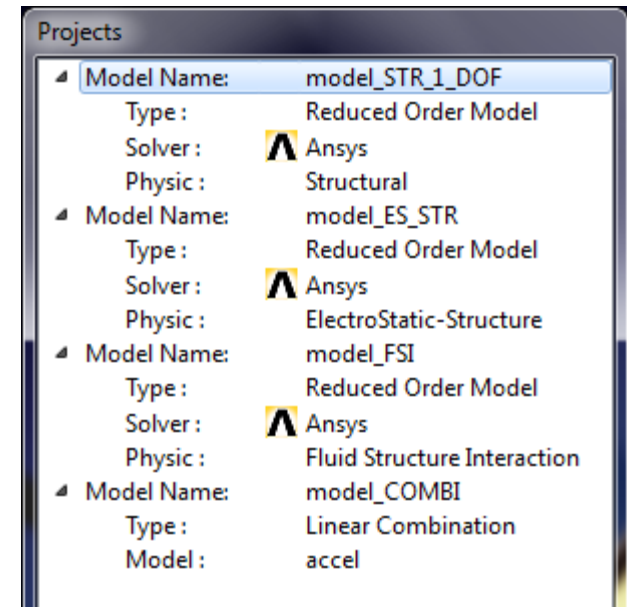
New Interface ..

- ❖ ... Based on Wizard to ease new data capture



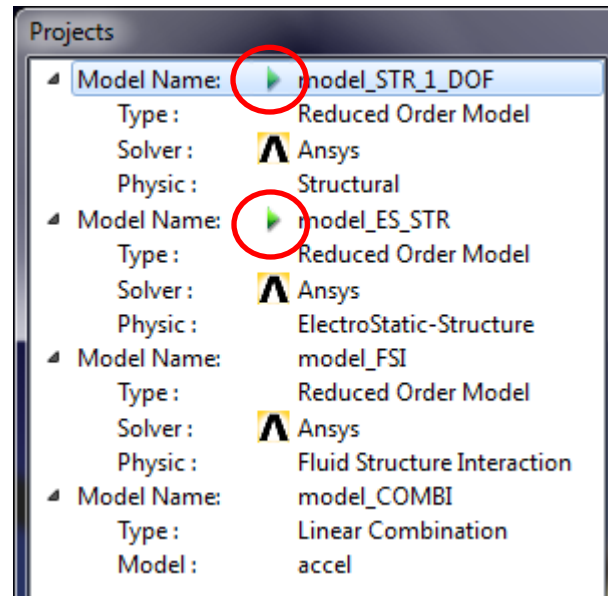
Multi Project

- ❖ 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



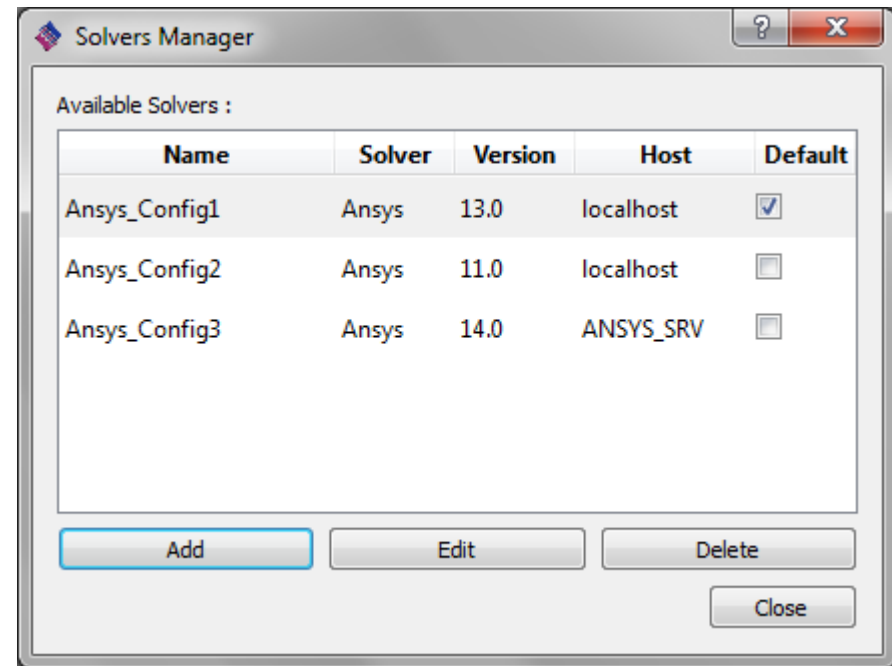
Concurrent Simulations

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



New Features

❖ Structural Systems :

- Non-Linear Stiffness extraction
- Support of Pre-stressed geometries

❖ Electrostatic-Structural Systems :

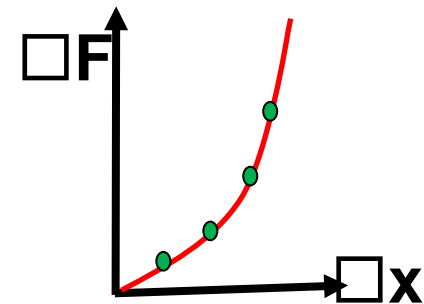
- Non-Linear Electrostatic effects extraction

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 $k(x) = k_1 \cdot x + k_2 \cdot x^2 + k_3 \cdot x^3 + \dots$

Non-Linear Stiffness

The image shows two overlapping windows from the Ansys software. The background window is titled 'model_STR_1_DOF' and has tabs for 'Solver', 'Fem Model', 'Output Data', 'DOFs', and 'Load Cases'. The 'Stiffness' section is checked, and 'Non-Linear Structural Stiffness Extraction' is selected. A red box highlights this section, with a large red arrow pointing to the foreground window.

The foreground window is titled 'model_STR_1_DOF*' and has the same tabs. It shows the 'DOF Requirements' section with a table:

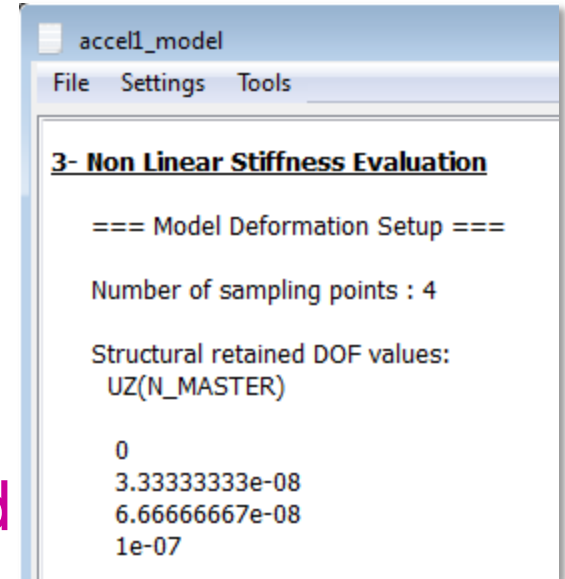
Node	Label	Min	Max	Sampling Points
N_MASTER	UZ	0.0	5e-07	4

Below the table are 'Add', 'Delete', and 'Clear' buttons. At the bottom, the 'Non-linear Stiffness' section shows 'Number of Sampling Points : 4' with an information icon. The status bar at the bottom indicates 'Solver : Ansys v130 on localhost:9080' and 'Running : [status icon]'.

□
- Number of sampling points

Non-Linear Stiffness

- ❖ According to the DOF definition the sampling points are reported
- ❖ Then Ansys generates the deformed geometries



```
accel1_model
File Settings Tools

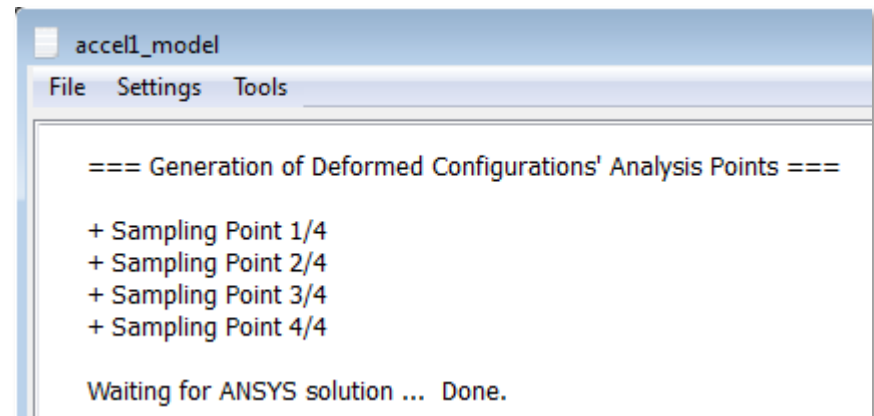
3- Non Linear Stiffness Evaluation

=== Model Deformation Setup ===

Number of sampling points : 4

Structural retained DOF values:
UZ(N_MASTER)

0
3.33333333e-08
6.66666667e-08
1e-07
```



```
accel1_model
File Settings Tools

=== Generation of Deformed Configurations' Analysis Points ===

+ Sampling Point 1/4
+ Sampling Point 2/4
+ Sampling Point 3/4
+ Sampling Point 4/4

Waiting for ANSYS solution ... Done.
```

Non-Linear Stiffness

- ❖ Finally the analysis are performed on each deformed geometry
- ❖ The force is extracted and the Stiffness Matrices computed

```
accel1_model
File  Settings  Tools

=== Non-Linear Structure Analysis ===

--- Analysis 1/4 ---
Waiting for ANSYS solution ... Done.

Stiffness Matrix:
[[ 32.9634]]

--- Analysis 2/4 ---
Waiting for ANSYS solution ... Done.

Stiffness Matrix:
[[ 32.9633]]

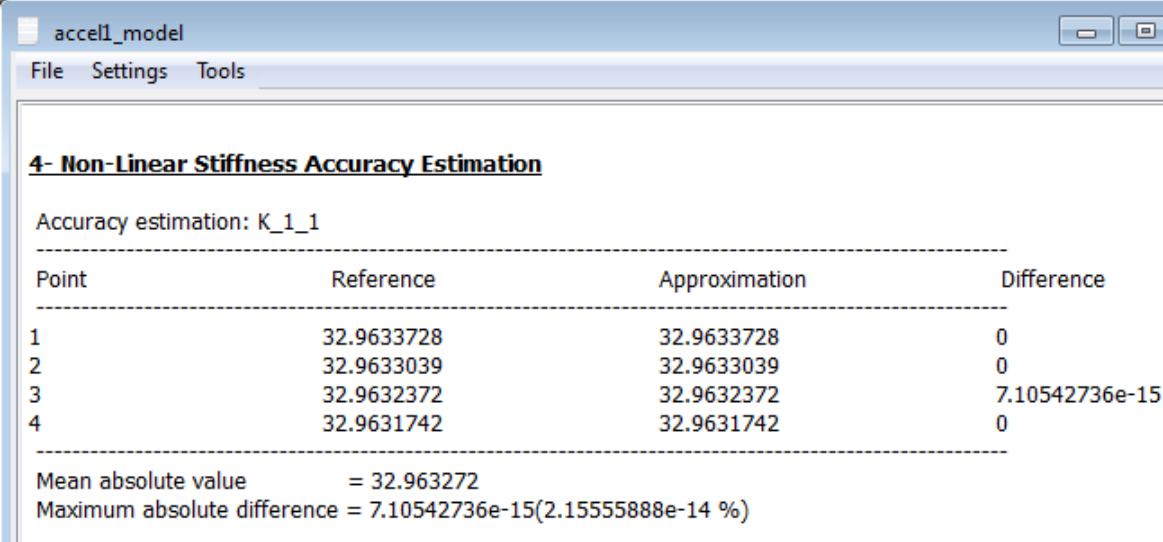
--- Analysis 3/4 ---
Waiting for ANSYS solution ... Done.

Stiffness Matrix:
[[ 32.9632]]

--- Analysis 4/4 ---
Waiting for ANSYS solution ... Done.
```

Non-Linear Stiffness

- ❖ 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.



4- Non-Linear Stiffness Accuracy Estimation

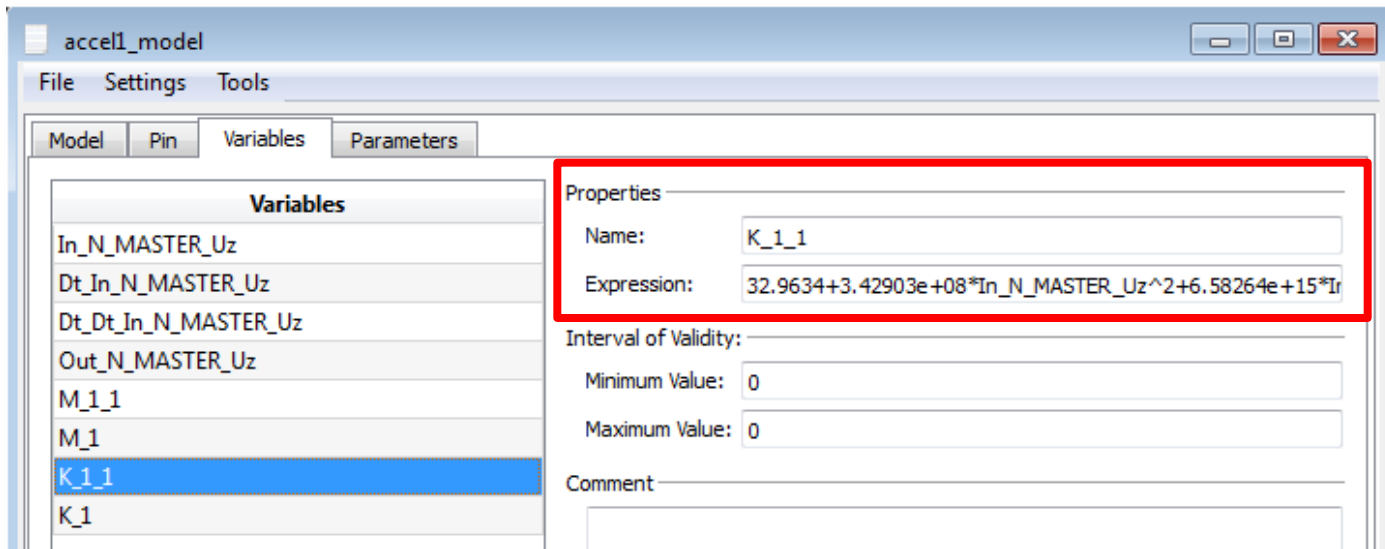
Accuracy estimation: K_1_1

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

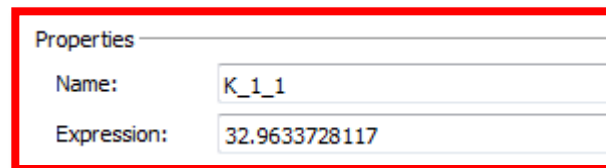
Mean absolute value = 32.963272
Maximum absolute difference = 7.10542736e-15 (2.15555888e-14 %)

Non-Linear Stiffness

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

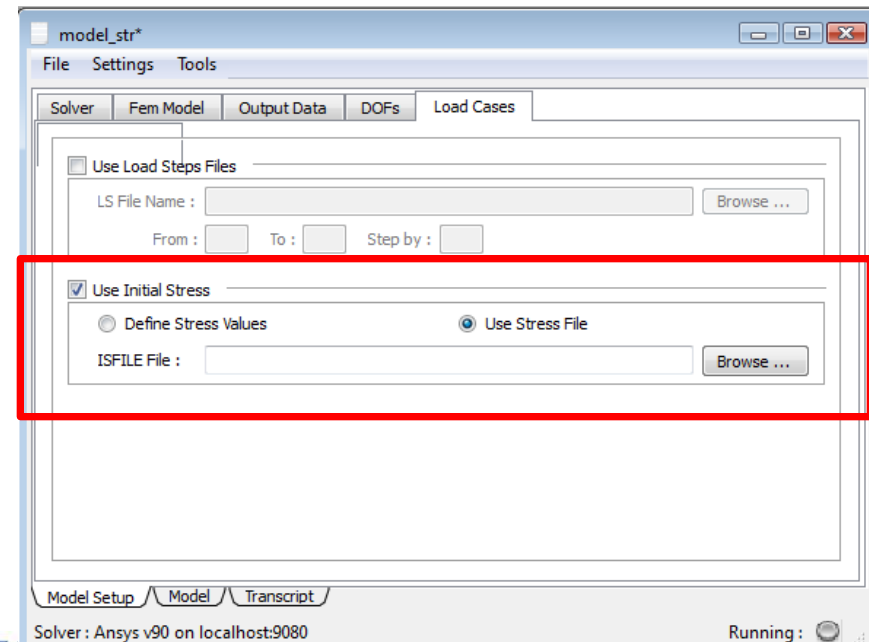
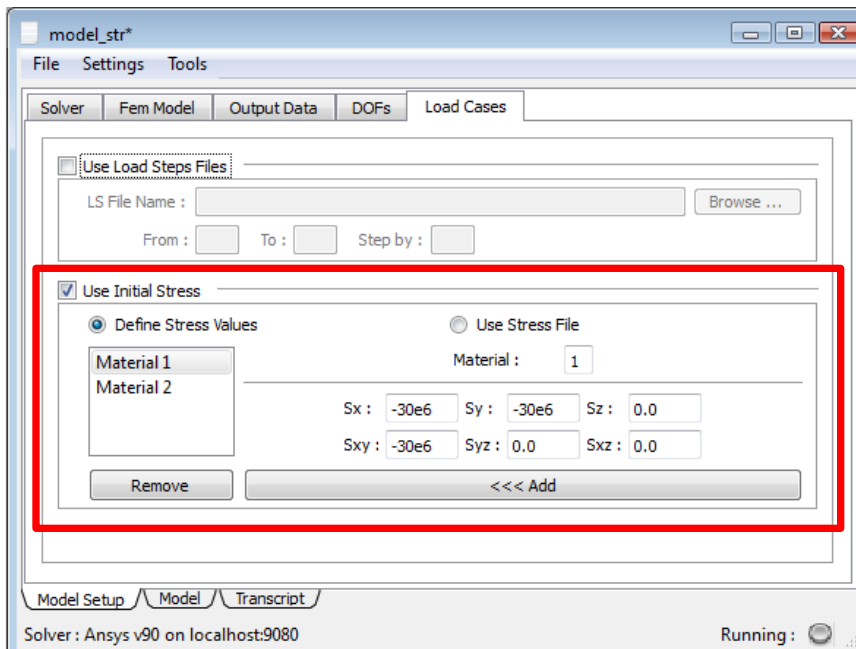


- - To compare to the expression of the Linear Stiffness Coefficient



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



Initial Stress

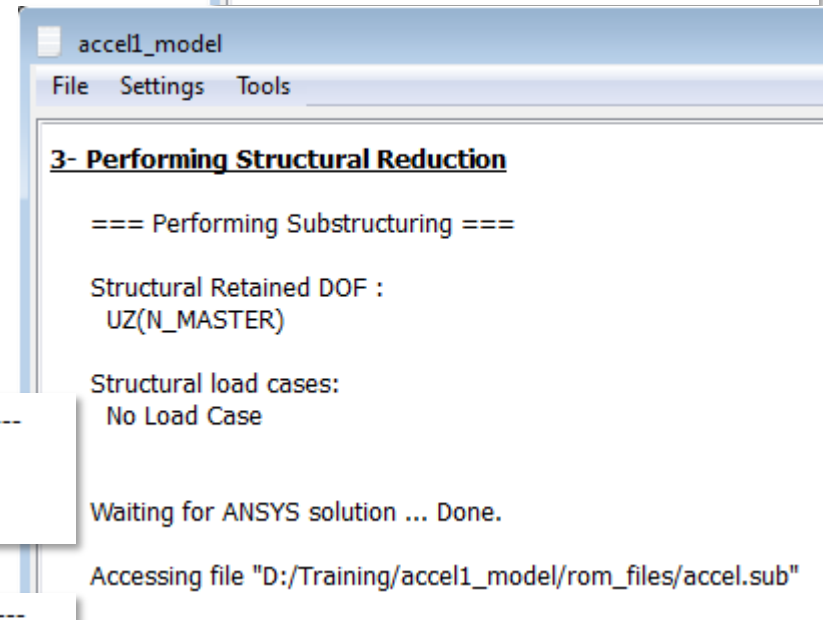
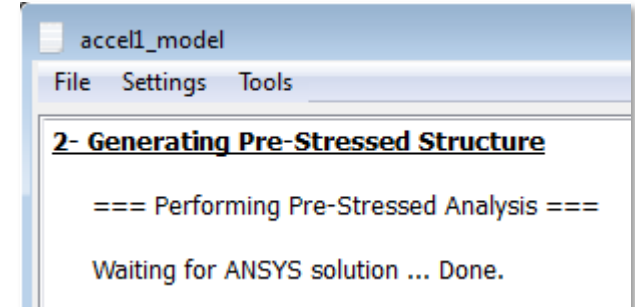
- ❖ First, a non-linear static analysis is performed to generate the pre-stressed structure
- ❖ Then the Substructuring stage is started from the previous analysis

⇒ No stress

Eigen Frequency

⇒ 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 Effects

- ❖ 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 $F_{coupling}$ 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 :
 $[C(x)]$

Non Linear Electrostatic Effects

❖ Principle of Capacitance Formulation

❖ Generate a set of model configurations

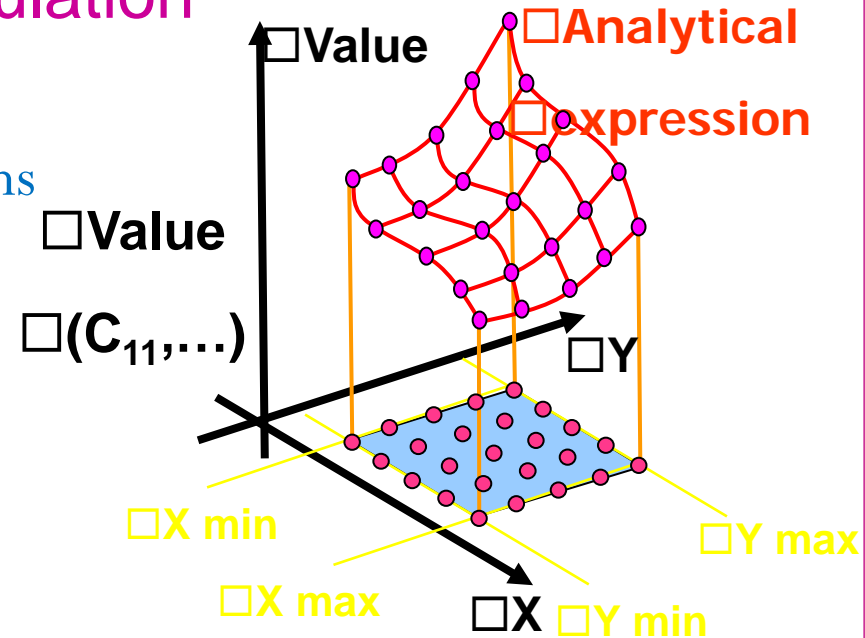
- Each Configuration corresponds
- to a x value

❖ In each of the configuration

- compute $[C(x_k)]$

$$C(x) = \frac{1}{p_0 + p_1 \cdot x + p_2 \cdot x^2 + p_3 \cdot x^3 + \dots}$$

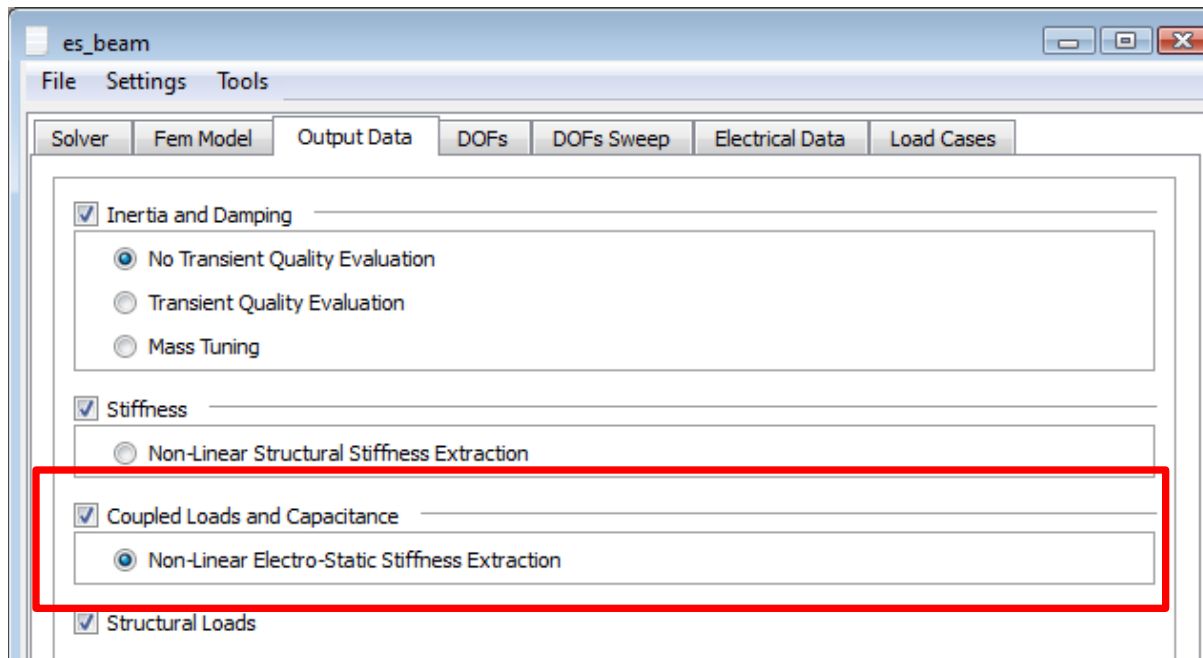
- ❖ Extract analytical expression for each value using a polynomial fitting



Non Linear Electrostatic Effects

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

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



Non Linear Electrostatic Effects

- ❖ 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 made only at the behavioral model generation step where, the polynomial expression is defined as being inverted (better static results) or non-inverted (spring softening capabilities)

Non Linear Electrostatic Effects

❖ 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	0
3	0.0211047433	0.0211047433	0

Mean absolute value = 0.0221903244
Maximum absolute difference = 0(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	-3.4694469e-14
3	0.0211047433	0.0211047433	-3.4694469e-14

Mean absolute value = 0.0221903244
Maximum absolute difference = 3.46944695e-18(1.56349537e-14 %)