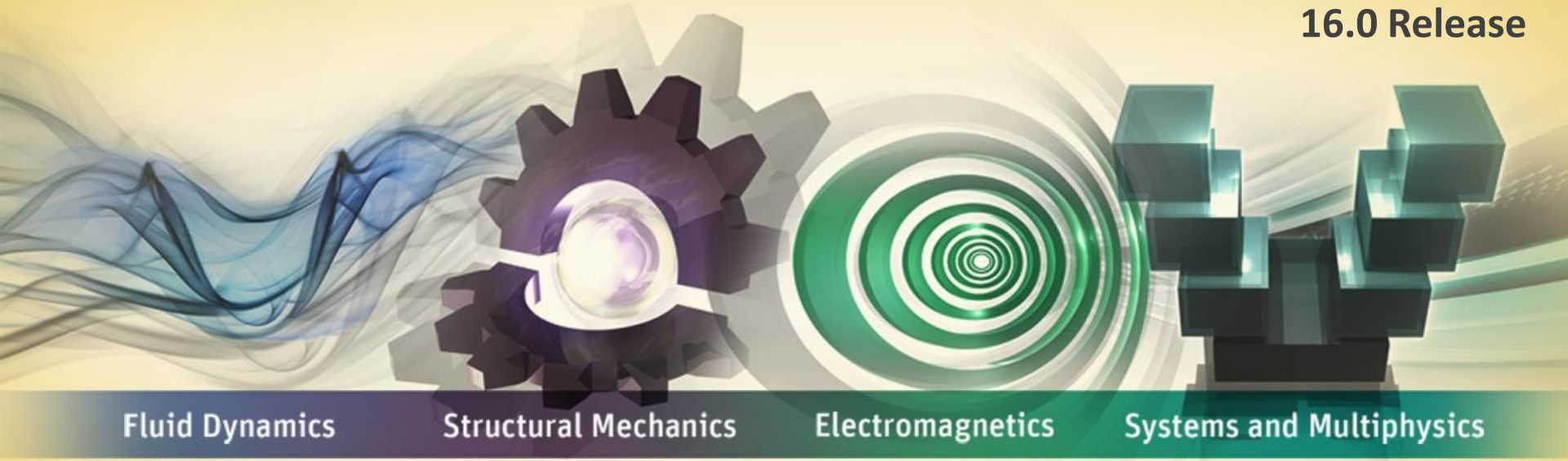
ANSYS

Realize Your Product Promise™

Workshop 3A:
Contact Stiffness Study

16.0 Release

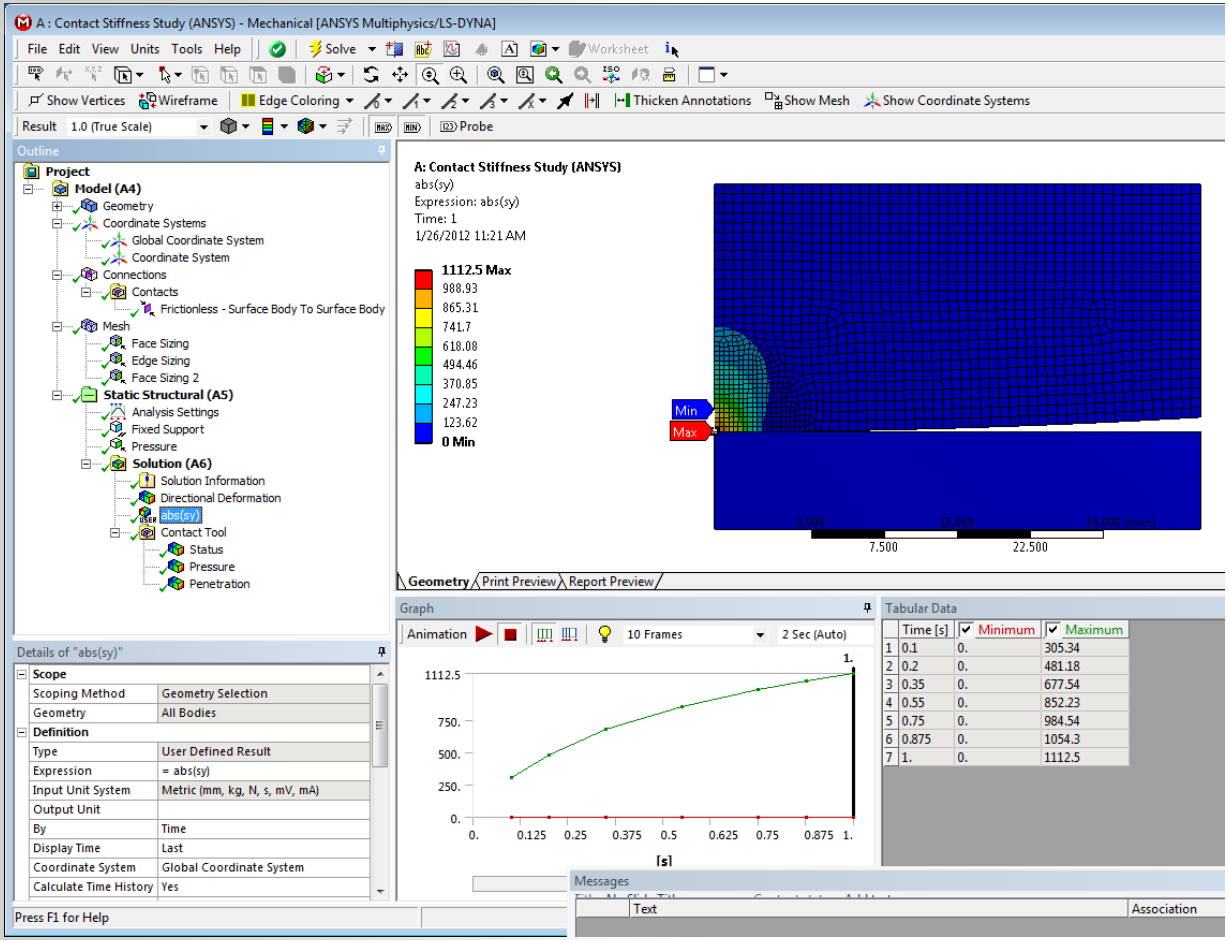


ANSYS Mechanical
Introduction to Structural Nonlinearities

Workshop 3A – Contact Stiffness Study

Goal:

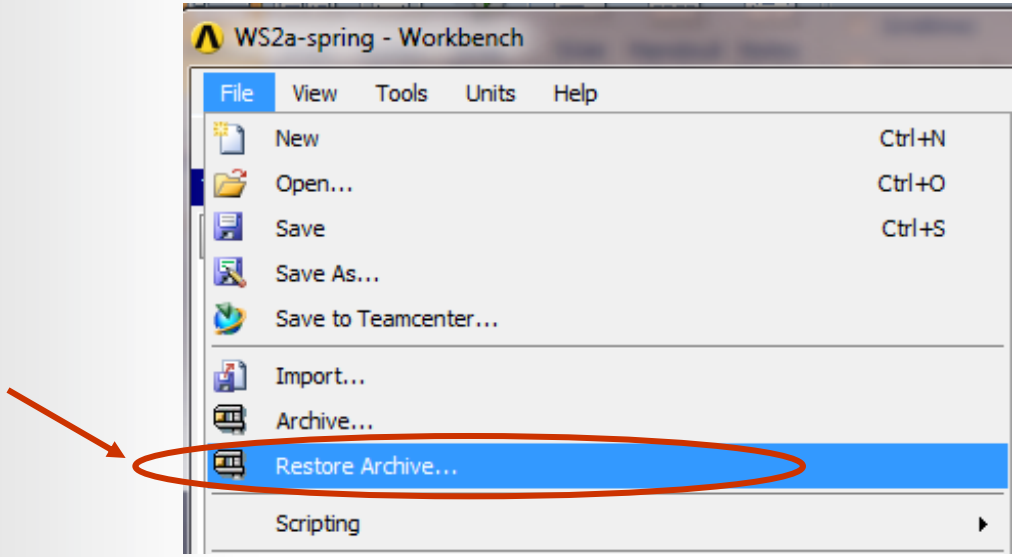
- **Perform a convergence study on contact stiffness**



… Workshop 3A: Contact Stiffness Study

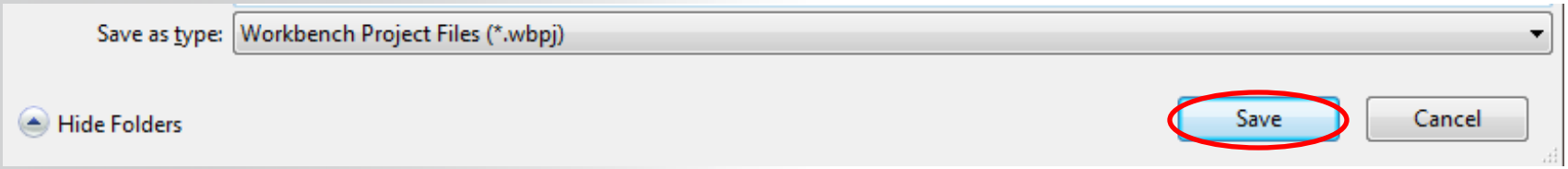
Steps to Follow:

Restore Archive… browse for file “SNL W3a-stiffness.wbpz”



Save as

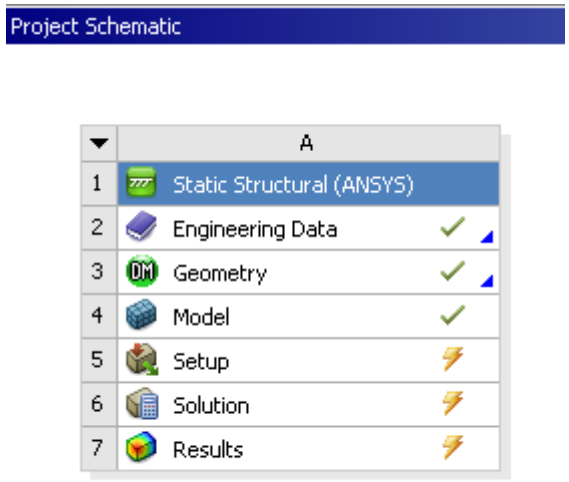
- **File name: ““W3a-stiffness”**
- **Save as type: Workbench Project Files (*.wbpj)**



… Workshop 3A – Contact Stiffness Study

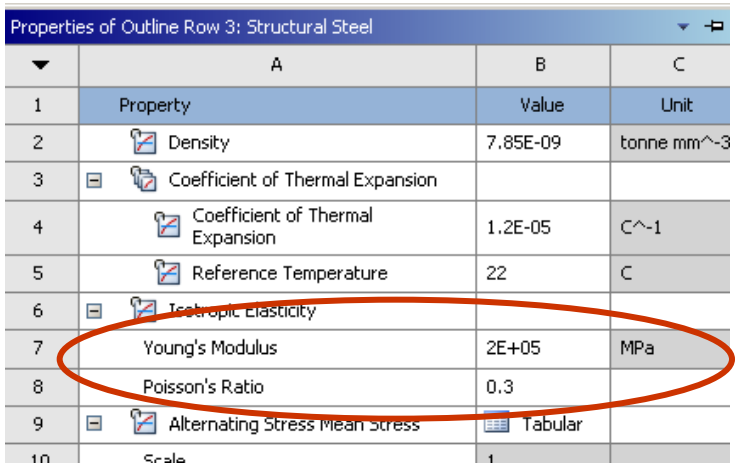
The project Schematic should look like the picture to the right.

- **From this Schematic, you can see that Engineering (material) Data and Geometry have already been defined (green check marks).**
- **It remains to set up and run the FE model in Mechanical**



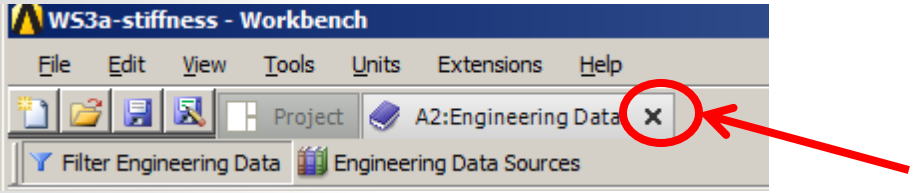
Contact Stiffness Study (ANSYS)

- **Open the Engineering Data Cell (double click on it OR Right Mouse Button (RMB)=>Edit) to verify the linear material properties.**
- **You might have to activate important dialog boxes from Utility Menu > View >…**
  - **Properties**
  - **Outline**
- **Verify that the units are in Metric (Tonne,mm,…) system. If not, fix this by clicking on…**
  - **Utility Menu >Units >Metric(Tonne, mm,…)**

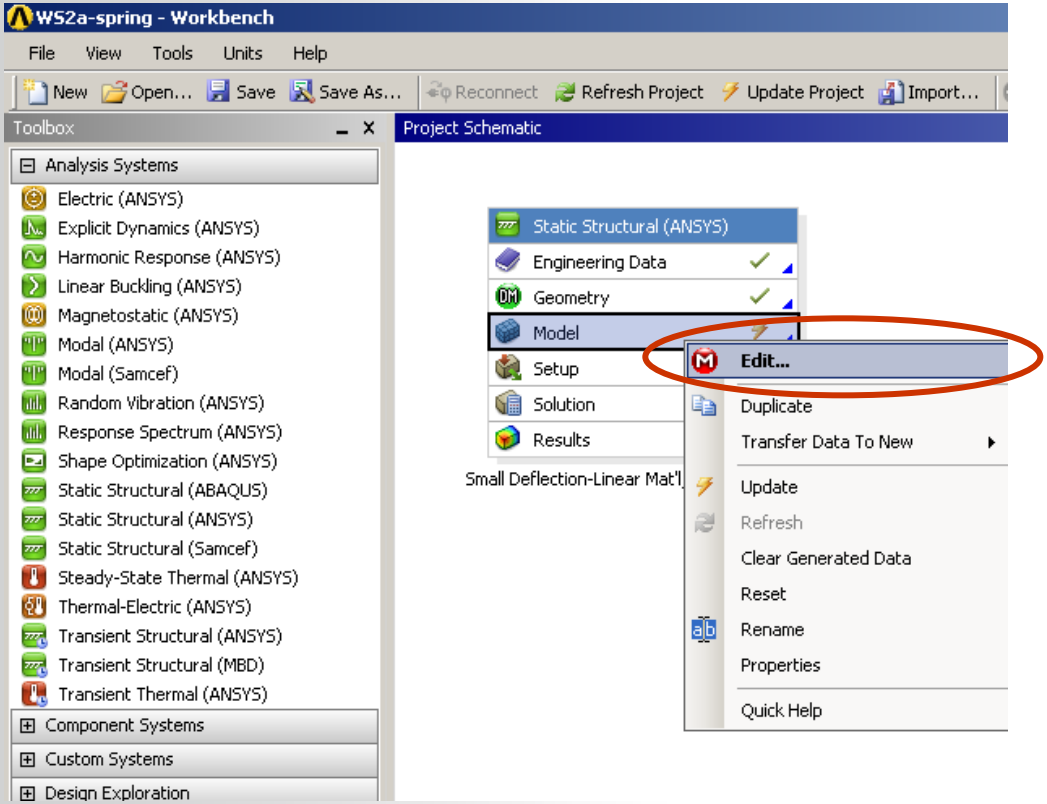


… Workshop 3A – Contact Stiffness Study

- Close Engineering Data…



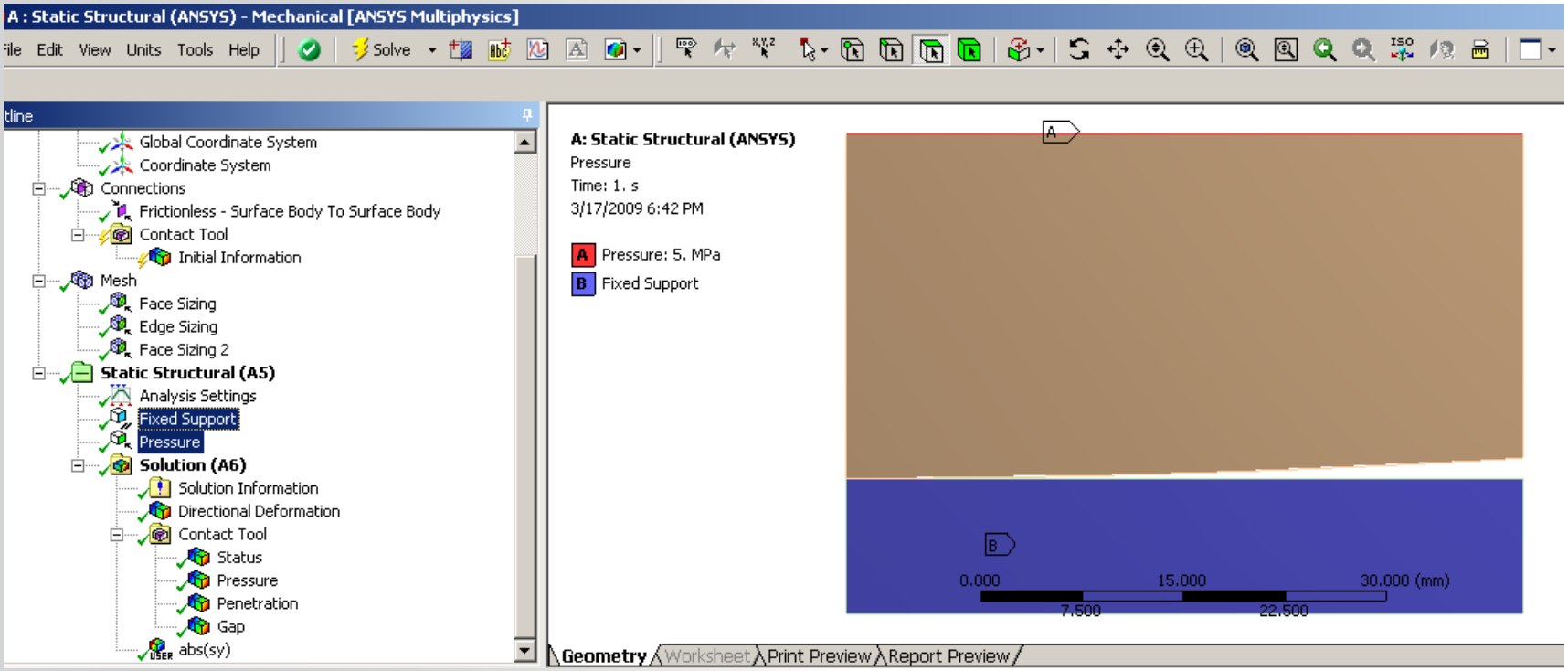
- Double click on the Model Cell to open the FE Model (Mechanical Session) (or RMB=>Edit…)



… Workshop 3A – Contact Stiffness Study

Geometry is 2D Axisymmetric. Lower plate is rigidly constrained. Upper plate is a flexible body with a crowned contour along bottom face. The upper plate is under a 5MPa pressure load acting downward.

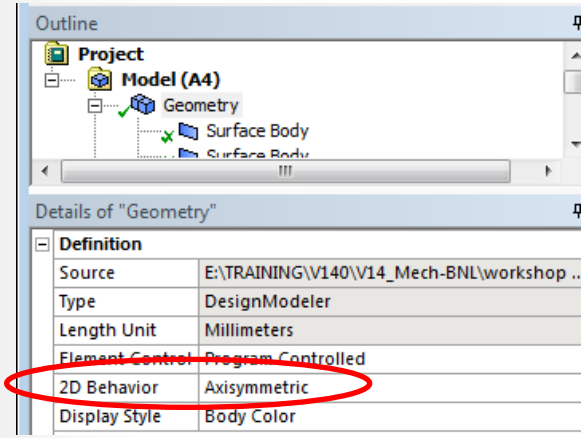
Material: Both plates are default linear elastic structural steel.



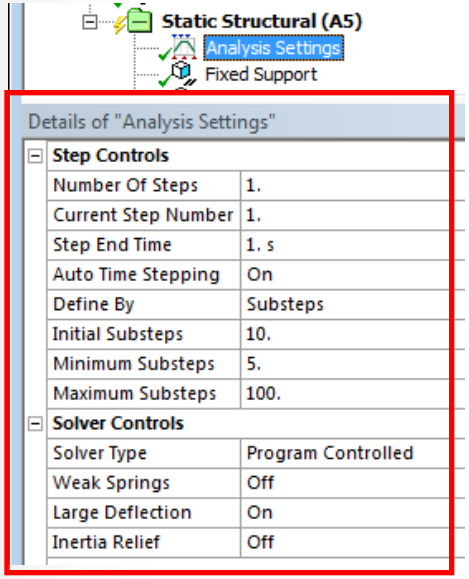
… Workshop 3A – Contact Stiffness Study

Open the folders beneath the model branch to become familiar with the model set-up.

Highlight “Geometry” and refer to the details window to verify that this is a 2D axisymmetric model.

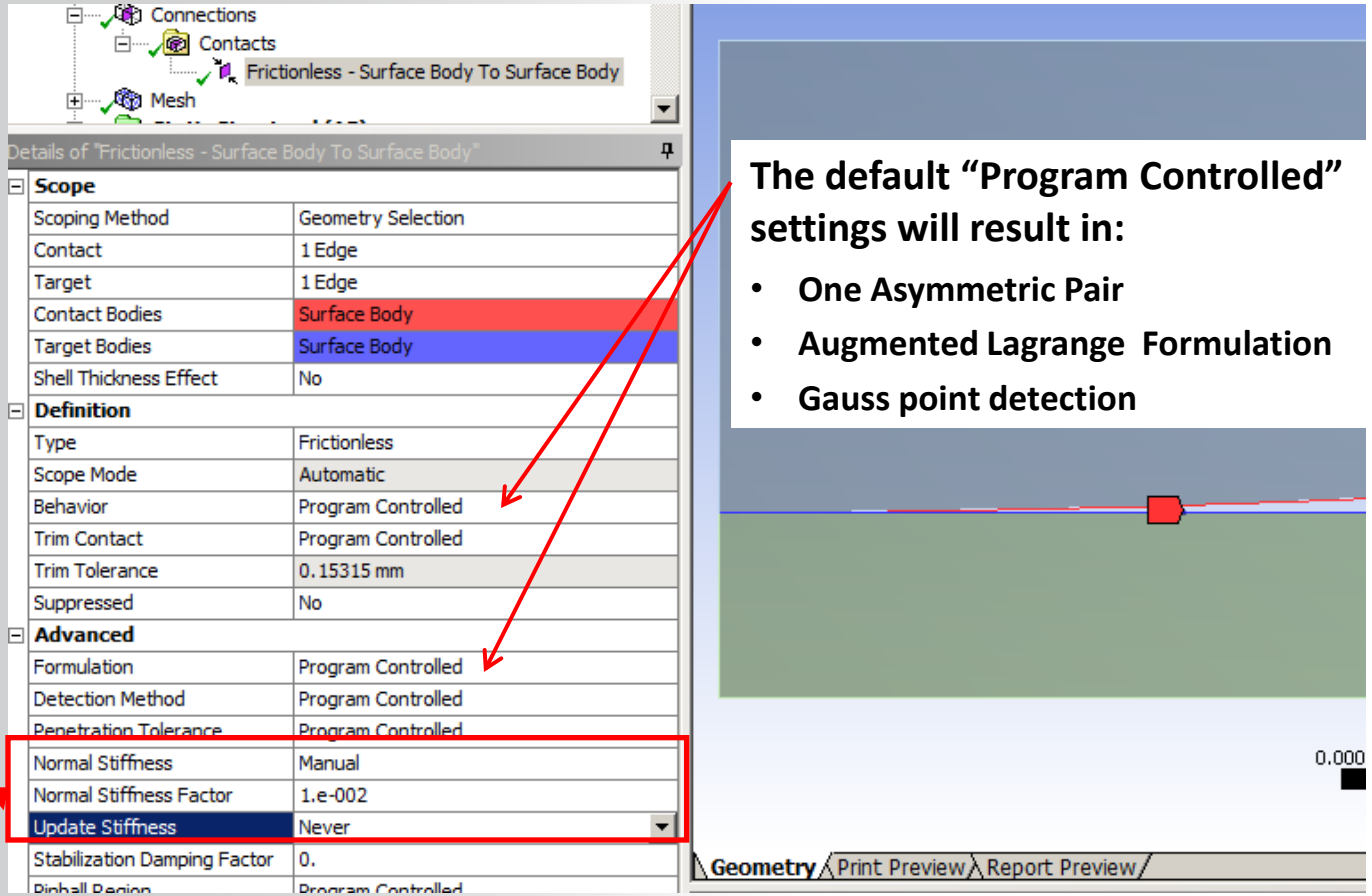


Inspect the Analysis Settings.
Autotime stepping = ON
Initial substeps = 10
Max substeps = 100
Large deflection = ON



… Workshop 3A – Contact Stiffness Study

Review the Frictionless contact set up and specifications. A single contact pair has already been set up with the following specifications:



This workshop will focus on a study of the contact stiffness and its influence on results (surface pressure and penetration).

… Workshop 3A – Contact Stiffness Study

Highlight the contact region and set the following:

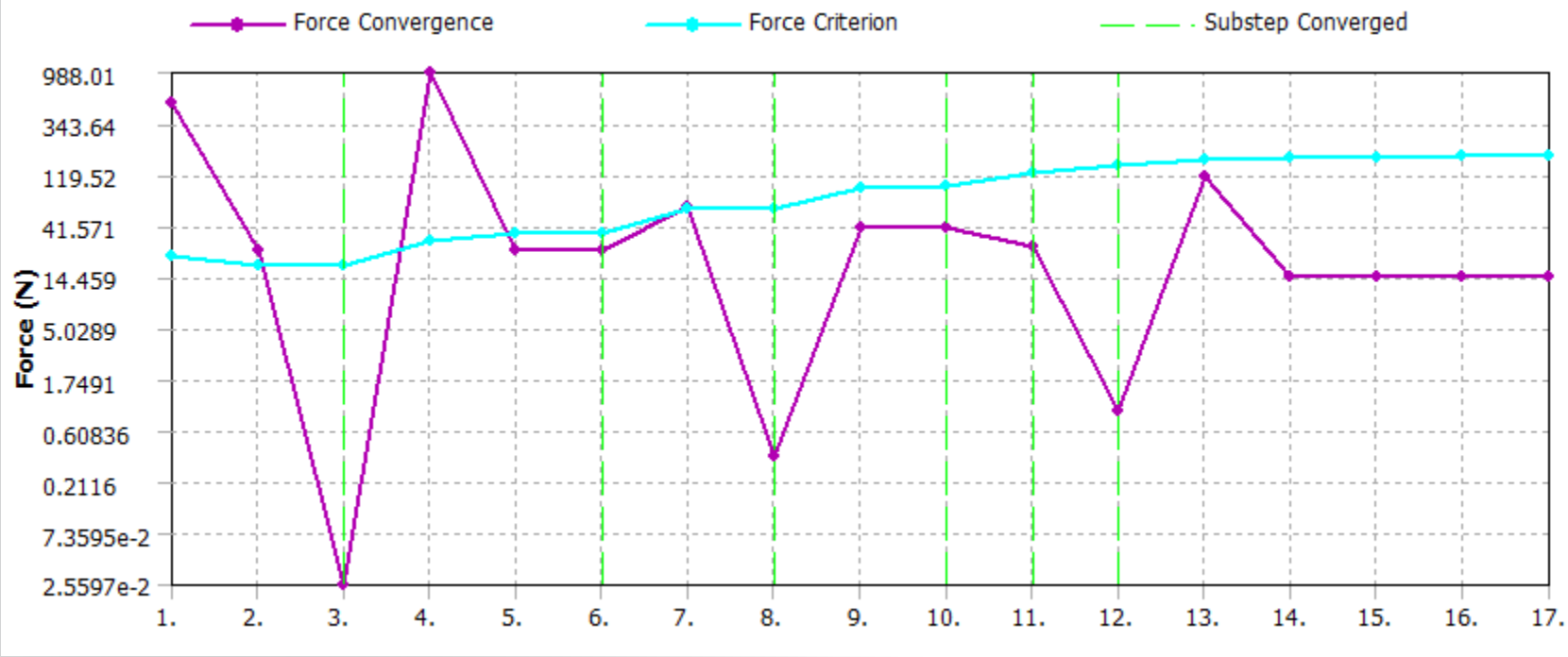
Normal Stiffness =“Manual”

Normal Stiffness Factor = 1e-002.

Update Stiffness = Never

Execute the Solve

Advanced	
Formulation	Program Controlled
Detection Method	Program Controlled
Interface Treatment	Add Offset, No Ramping
Offset	0. mm
Normal Stiffness	Manual
Normal Stiffness Factor	1.e-002
Update Stiffness	Never
Stabilization Damping Factor	0.
Pinball Region	Program Controlled
Time Step Controls	None



… Workshop 3A – Contact Stiffness Study

Highlight the Solution Information Branch and scroll up the Solver Output to find the contact specifications.

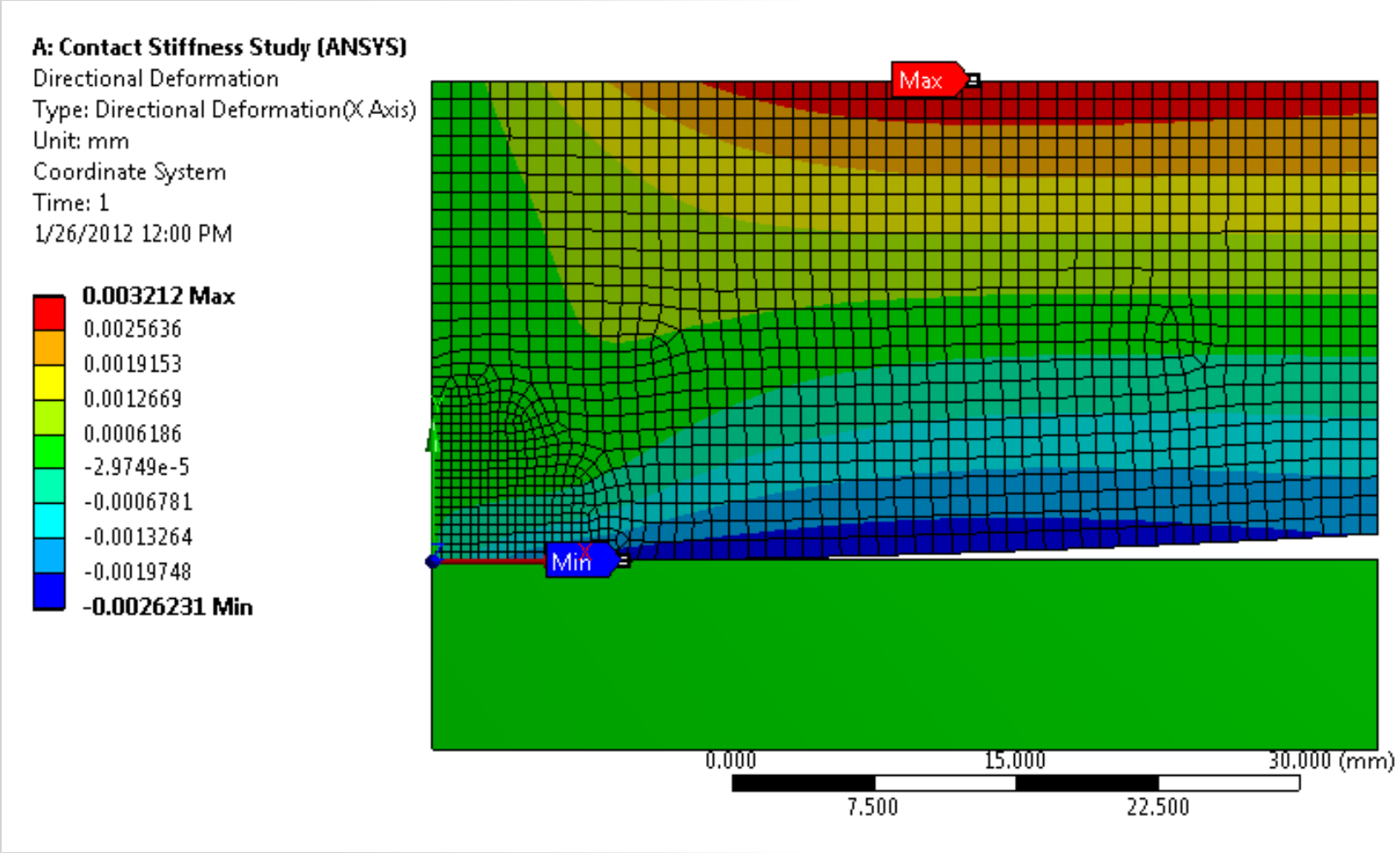
- **Confirm the auto-asymmetric behavior, Augmented Lagrange formulation, gauss point detection and the user defined Normal Stiffness (FKN) value with no mention of automatic stiffness updating.**

```
*** NOTE ***                            CP =       3.338   TIME= 11:57:18
Symmetric Deformable- deformable contact pair identified by real
constant set 3 and contact element type 3 has been set up.  The
companion pair has real constant set ID 4.  Both pairs should have the
same behavior.
ANSYS will keep the current pair and deactivate its companion pair,
resulting in asymmetric contact.
Contact algorithm: Augmented Lagrange method
Contact detection at: Gauss integration point
Contact stiffness factor FKN                0.10000E-01
The resulting contact stiffness              2666.7
Default penetration tolerance factor FTOLN   0.10000
The resulting penetration tolerance          0.84371E-01
Frictionless contact pair is defined
Average contact surface length               0.80694
Average contact pair depth                   0.84371
User defined pinball region PINB             0.15315
*WARNING*: Initial penetration is included.

*** NOTE ***                            CP =       3.338   TIME= 11:57:18
Max.  Initial penetration 4.974007539E-02 was detected between contact
element 1584 and target element 1647.
You may move entire target surface by : x= 1.278077453E-04, y=
-4.973991119E-02, z= 0,to reduce initial penetration.
*****************************************
```

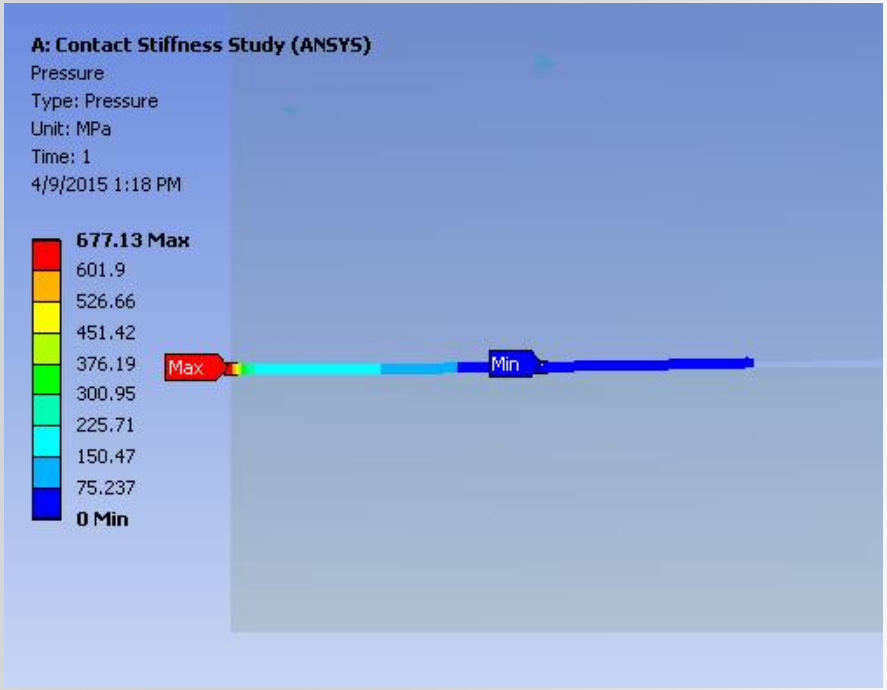
… Workshop 3A – Contact Stiffness Study

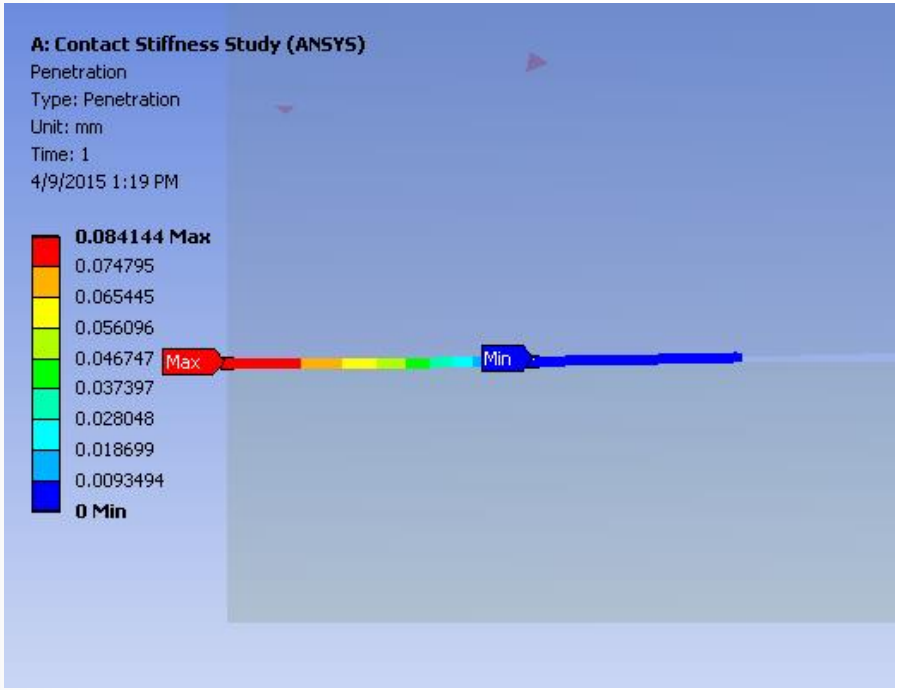
- **Post process the Deformation:**



… Workshop 3A – Contact Stiffness Study

- **Post process the contact results:**
  - **Contact Pressure**
  - **Contact Penetration**





… Workshop 3A – Contact Stiffness Study

- Record the results in Table below.
  - Note: You can cut-paste electronically from results cell.
- Repeat analysis with Normal Stiffness factors, FKN= 0.1, 1.0, 10.0, 100.0
- For further comparison, change the contact formulation to Normal Lagrange.

Contact Formulation	FKN	Total Deform	Contact Pressure	Penetration	# of iterations
Aug Lagrange	0.01				
Aug Lagrange	0.1				
Aug Lagrange	1				
Aug Lagrange	10				
Aug Lagrange	100.0				
Norm Lagrange	N/A				

- Experiment also with Update Stiffness between iterations @ FKN=100 for comparison.
- Which combination offers the best results (in terms of accuracy and # of iterations)?

… Workshop 3A – Contact Stiffness Study

- **Conclusions**

  Notice that, in general, as stiffness increases, contact penetration decreases while maximum contact pressure increases. Notice also the general trend toward more iterations and longer run times leading eventually to convergence troubles at FKN=100.

  Note the benefit of using the automatic stiffness updating tool to overcome convergence trouble at FKN=100.

  Note also that regardless of which stiffness value is used, the overall total displacement of the assembly changes very little. This underscores the need to know your engineering objectives. If localized contact related results are unimportant, then the program controlled defaults might be acceptable.

  Specifying the right contact stiffness is highly problem dependent.

  The "correct" answer depends on the engineering objectives and is always a balance between quality (accuracy) and cost (run time).