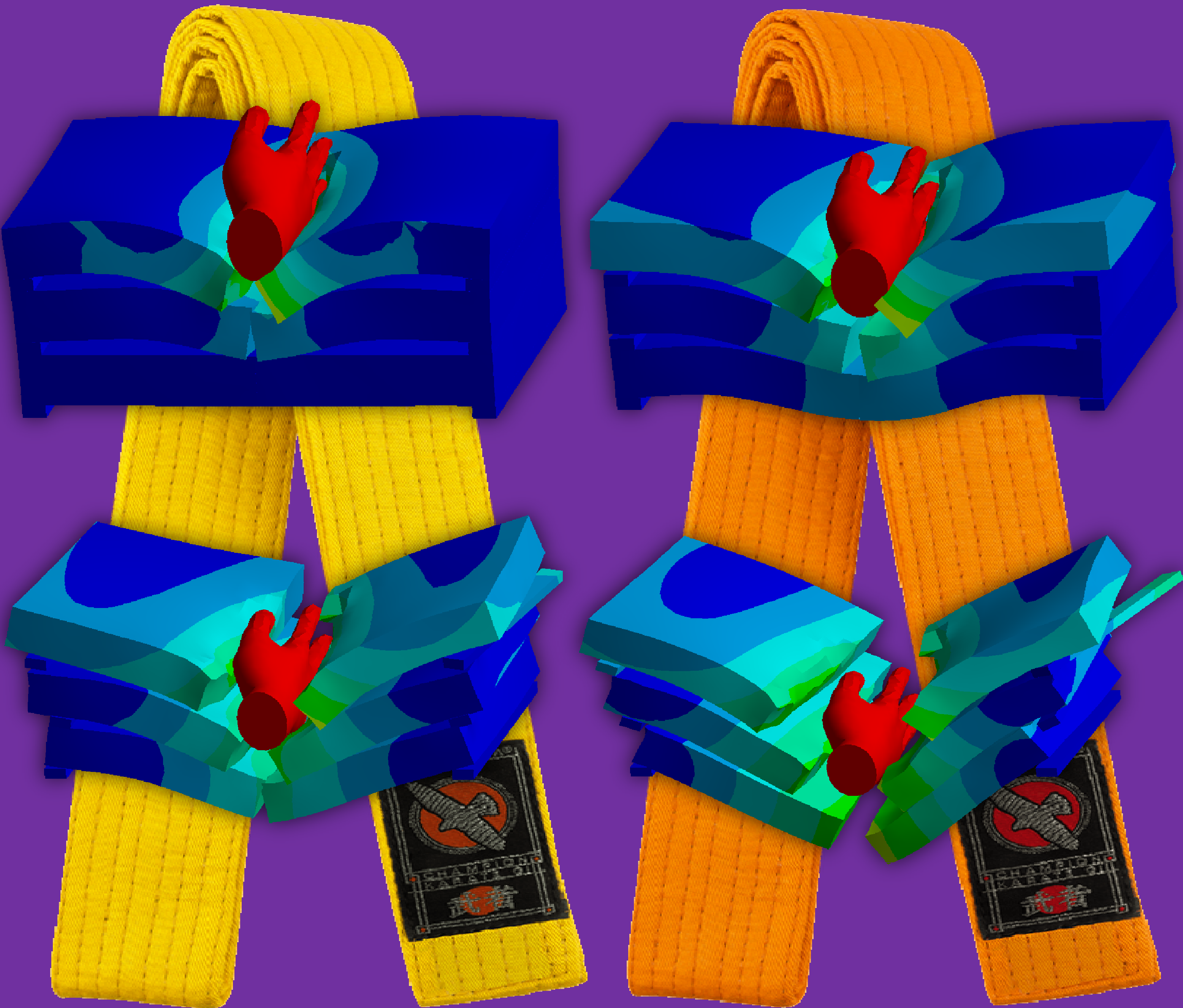
Become a Black Belt in ANSYS Workbench

by Claudiu Danila

Volume I: yellow and orange belt

-20 fast tutorials for smart beginners-



ψ This book is dedicated to Lord Shiva, on the occasion of Shiva Ratri ψ

Foreword

Hi all!

My name is Claudiu and I am in the FEA domain since the year 2000. I began in the faculty as a part of research grants, then I was a mechanical designer for a few good years, but FEA was with my the whole time. In the present I work only as a Finite Element Analyst for a big Automotive company and I get to share via *expertfea.com* and *GrabFEA.com*, some of my experience and expertise with you, people who want to try new things and have a brighter, more exciting future.

I began this book with the yellow belt, skipping the white one, because I assumed that you already have the white one, meaning that, by now, you have already opened ANSYS Workbench, inserted a fixed support and a force on the other end of the beam.

This book is a collection of fast tutorials (only the steps of how to peform the FEA, no extra boring stuff), detailing the FEA scenarios found on the Solved FEA page of *expertfea.com* and *GrabFEA.com*. Many of you asked me to detail the Solved FEA and to present those cases in a step by step approach, so here you have it.

Check the Youtube results movies for the tutorials contained in this book, then take the decision of wether to buy this book or not.

It is best if you are a beginner, because you'll find lots of easy exercises which gradually increase their difficulty in this volume. Only one example, the one with Reinforced Concrete has Geometry modeling detailed! Check the Homework section of each tutorial, in order to expand its options and squeeze more knowledge from each case.

If at first you don't understand what you're doing, no problem, trust me, you'll understand it later. As in Karate, in the beginning it's best if you exercise a lot and regularly, then, in time you will understand why you do what you do. I am telling you now: you do it for yourself, to master your craft, your FEA skills, and to better yourself as a stress engineer!

So, realize that HERE and NOW you have everything at your disposal to improve your career and path in life, and above all, BEGIN YOUR PRACTICE TODAY, REGULARLY! Don't live a day without FEA! (it even rhymes :)

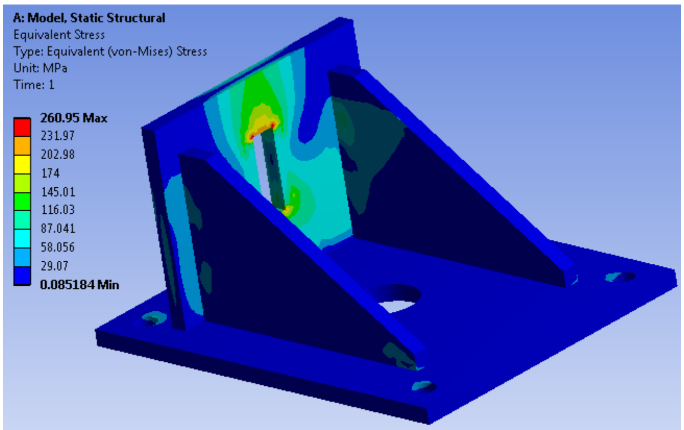
From my experience, I am telling you that, if you practice at least 1 hour each day, you will become very skilled within 1 year. I am doing FEA ~10 hours/day, each day, and I still think I barely scratched the surface of this domain. So, hurry up, train yourself every day and you'll surely become the next Sheldon Imaoka, an ANSYS sensei whom I respect very much and hope to meet in this life!

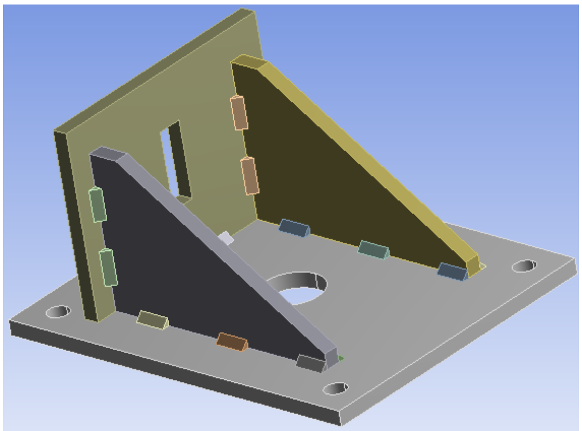
This is my creed: "FEA is the art of skillfully using Geometry, Materials, Mesh, Boundary Conditions and Analysis Settings in order to achieve reliable results in a timely manner". But to do all these very well, you need lots of exercise, a lot of trial and error. There is no training, book or tutorial on how to skip directly to mastership, to skip errors and get the best results! You must walk the path in order to reach the destination; as in real life, usually there are no shortcuts, unless you are one of Bill Gates' children, then this book is not for you. Look on your ID document now: if it doesn't say Jennifer Katharine Gates, Rory John Gates or Phoebe Adele Gates, then let's do together some FEA in ANSYS Workbench! After all, you don't want to be in their shoes, because they probably didn't hear about FEA by now, which is a big loss in their life, maybe their only loss :))

Kidding aside, start with this volume, because more FEA, interesting and challenging will follow in the next volumes, in the next months. Remember also to check other aurhors, trainings and tutorials, because the Absolute Truth is not confined within 1 single scripture, but it can be found in many more...

I wish you all the best and believe in yourself and in your awesome future! **Claudiu, 23rd of September 2016**

<u>**CASE 1:** ANSYS Workbench Static Structural FEA of the verification of a welded structure</u>

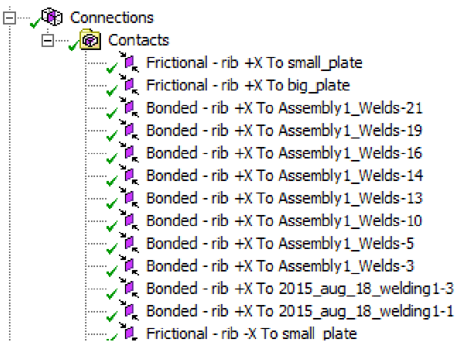




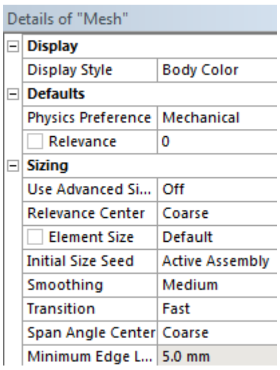
<u>Engineering Data (Materials):</u> All bodies are made of default Structural Steel.

<u>Geometry:</u> 2015_aug_18_welding2.stp

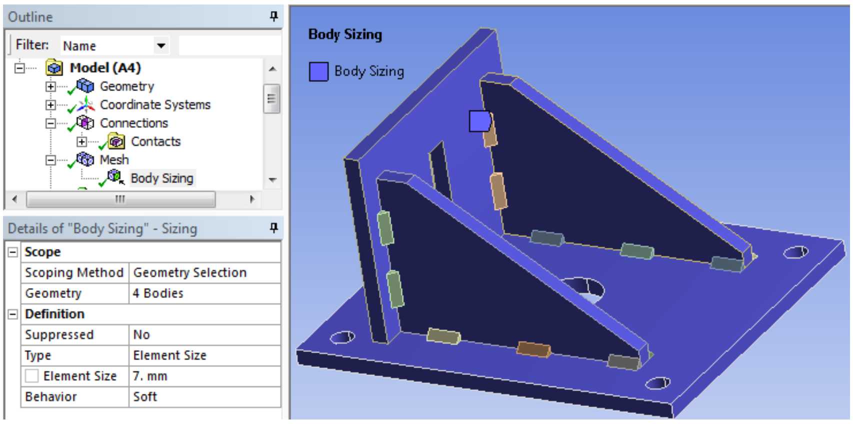
<u>Contacts:</u> All contacts are default Bonded, except the ones between RIB and PLATES, which are Frictional with Friction Coefficient 0.2



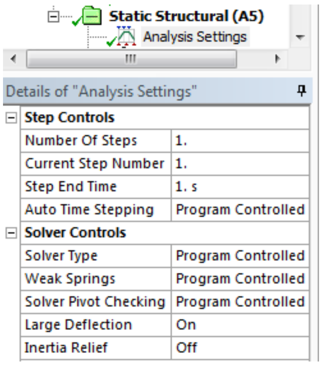
<u>Mesh:</u> The mesh is default, except a Body Sizing of 7 mm on PLATES (not on weldings).


Details of "Mesh"

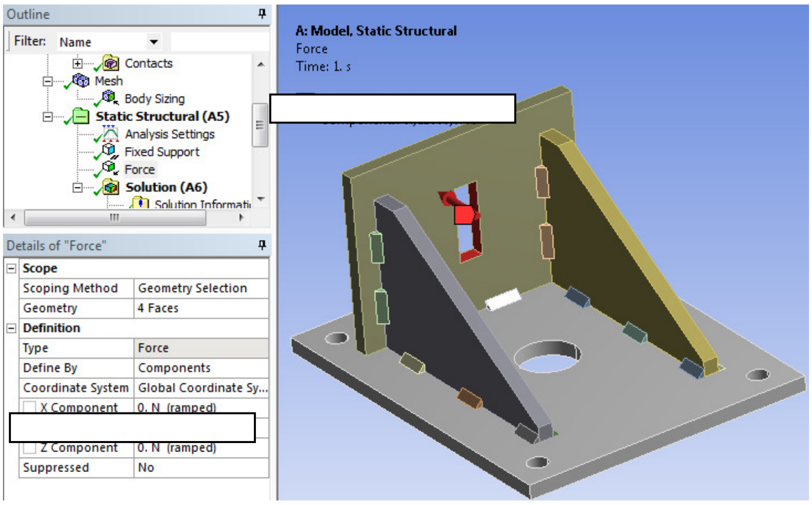
Display	
Display Style	Body Color
Defaults	
Physics Preference	Mechanical
Relevance	0
Sizing	
Use Advanced Si...	Off
Relevance Center	Coarse
Element Size	Default
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Fast
Span Angle Center	Coarse
Minimum Edge L...	5.0 mm



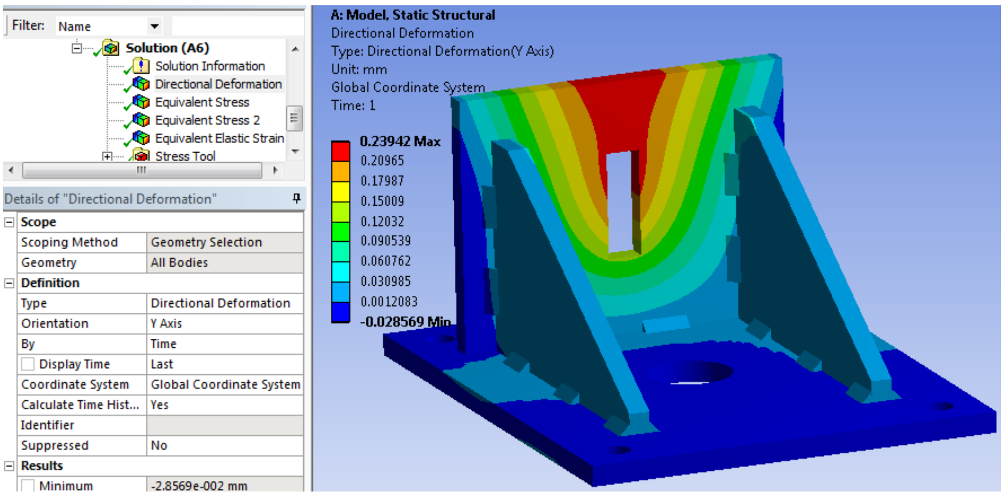
Analysis Settings: Default values, 1 step, as seen here.



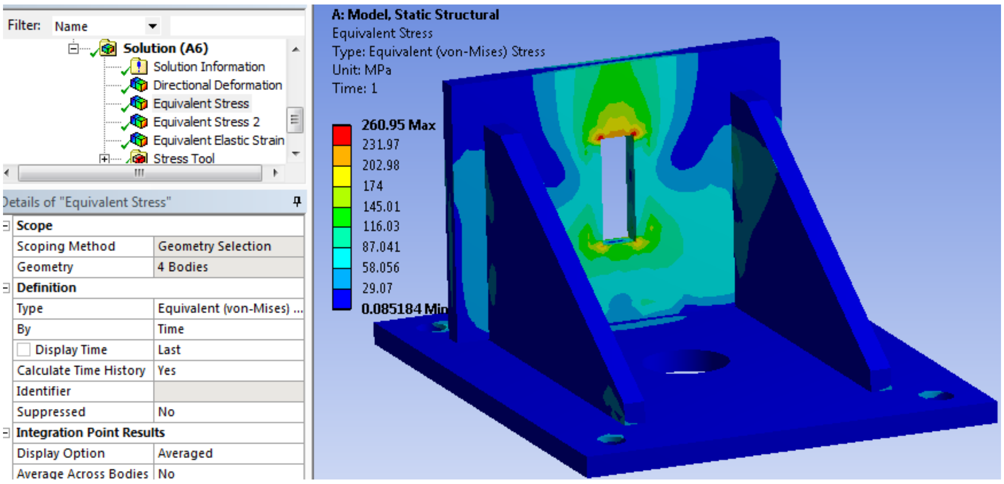
Force: 12.000 N on Y axis, applied inside the rectangular hole. Right click Solution, Solve.



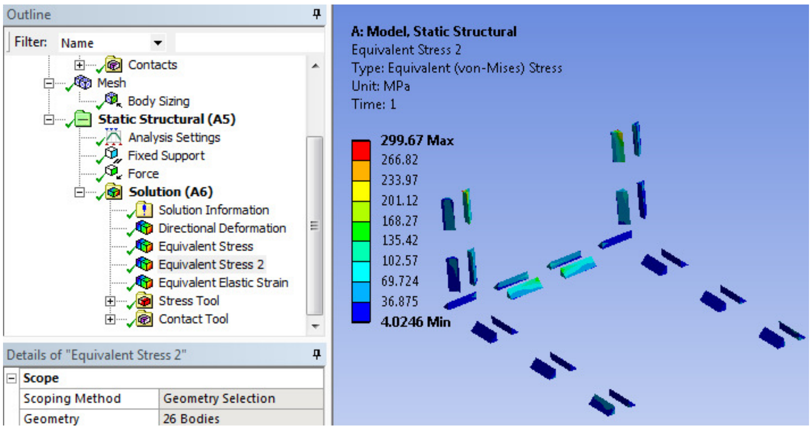
Solution: Directional Deformation on Y axis.



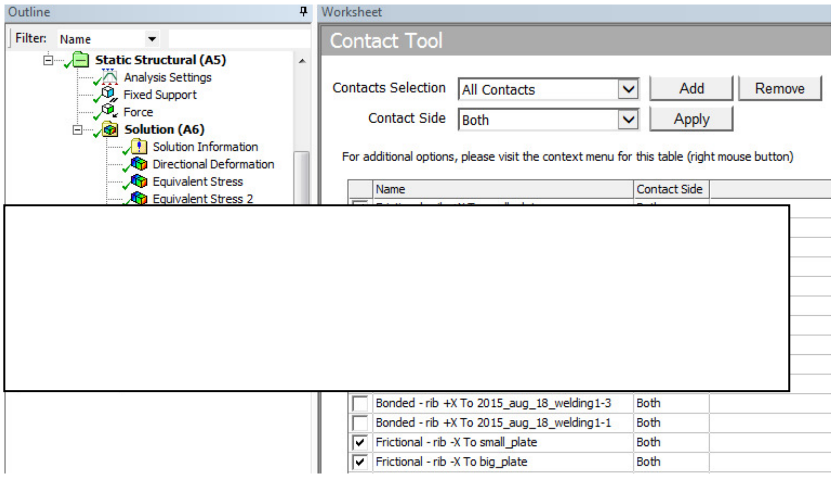
Default Equivalent Stress.



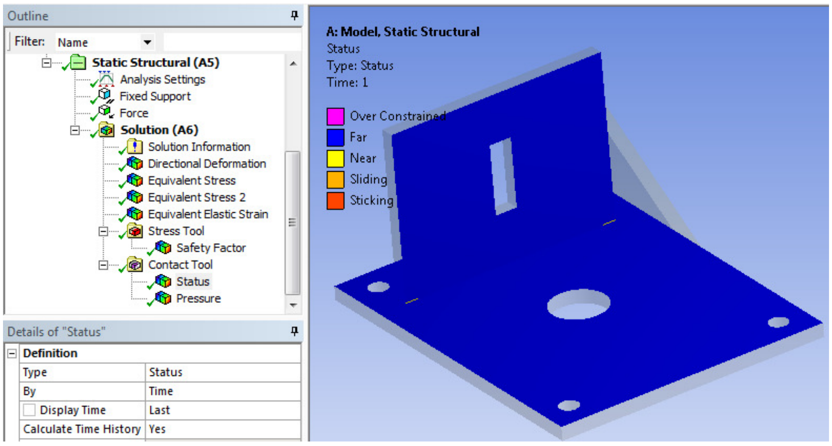
Equivalent Stress only in the weldings, as seen here.



Contact Tool only in the Frictional Contacts, as seen here.



Default Contact Tool, Status, as seen here.



<u>Further homework:</u>

- increase the mesh Relevance 100, solve, draw the conclusions

- increase the Force 10 times, solve, draw the conclusions

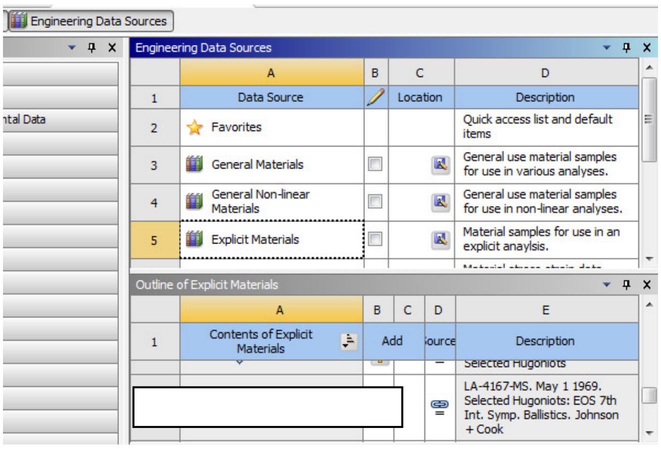
- change the material of the ribs to Structural Steel NL, solve, draw the conclusions

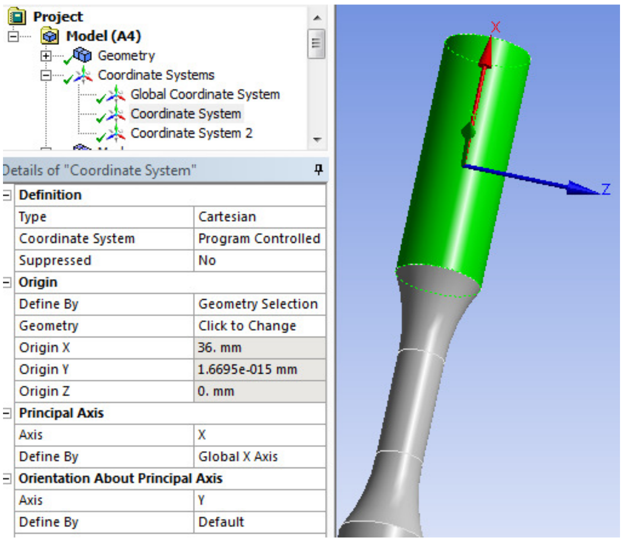
- This page, intentionally left blank -

CASE 2: ANSYS Workbench Explicit Dynamics FEA of tensile and torsion tests on steel specimens

A) TENSILE simulation

Engineering Data (Materials): Press the Engineering Data Sources button and click the yellow plus sign in the B column, to add it to our materials

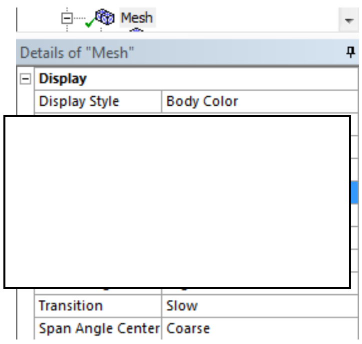




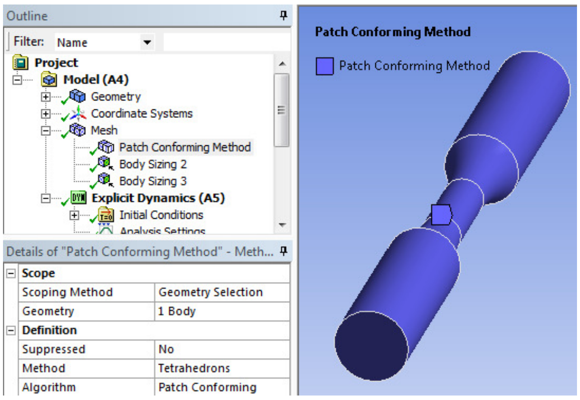
Create another Coordinate System on the opposite cylindrical face.

Contacts: None

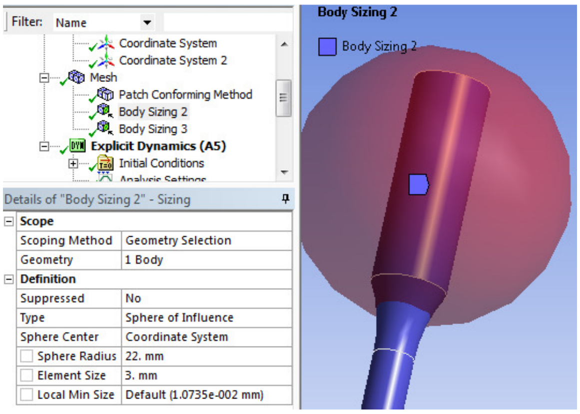
Mesh: Use these options.



Insert Meshing Method, Tetrahedrons.

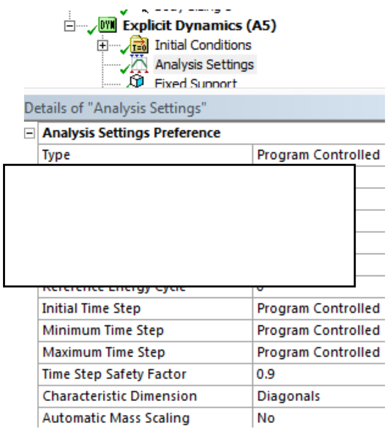


Insert a Mesh Sizing using Type, Sphere of Influence. For Sphere Center, select the previously created Coordinate System. Insert these values.

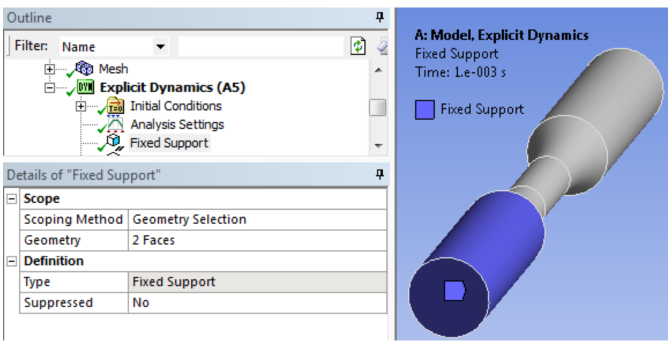


Repeat for the other side and for the other Coordinate System.

<u>Explicit Dynamics, Analysis Settings:</u> Default values and End Time =



<u>Fixed Support:</u> Apply it on one end.

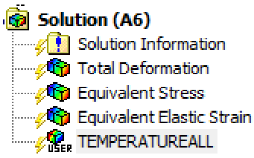


<u>Displacement:</u> Apply mm on X axis, applied on the faces opposite to the ones fixed above.

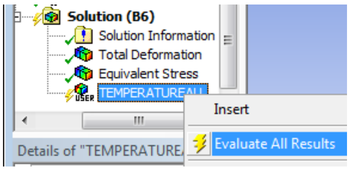
Solution: Default Total Deformation.

Default Equivalent Stress.

Default Equivalent Elastic Strain. Right click Solution, Solve.

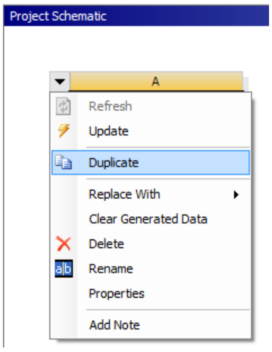


Right click any Solution item, Evaluate All Results.

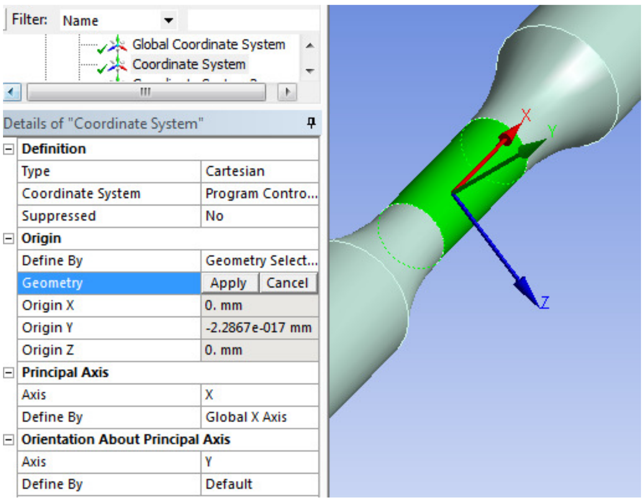


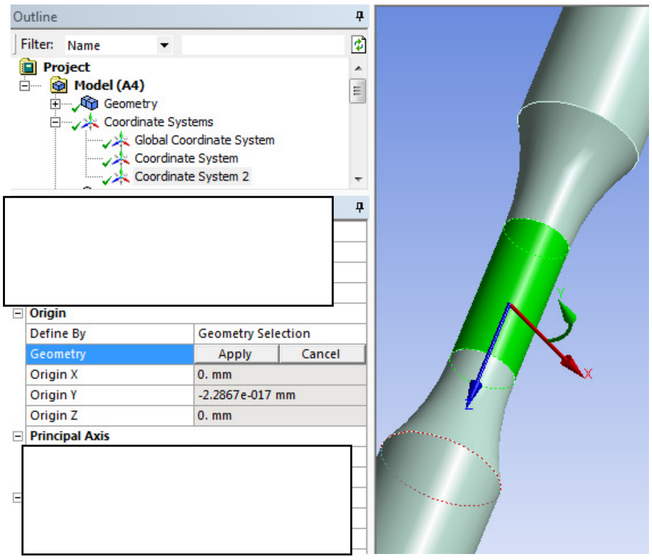
B) TORSION simulation

In Project Schematic window, right click the previous Explicit Dynamics case, Duplicate.

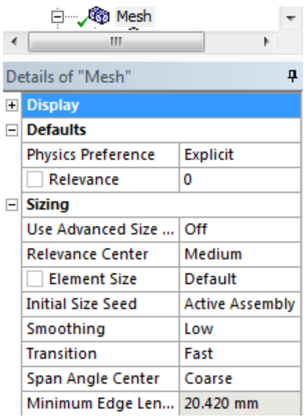


Coordinate System: Select this face, Apply.

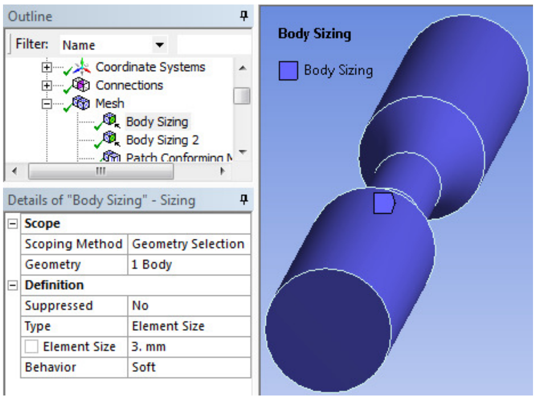


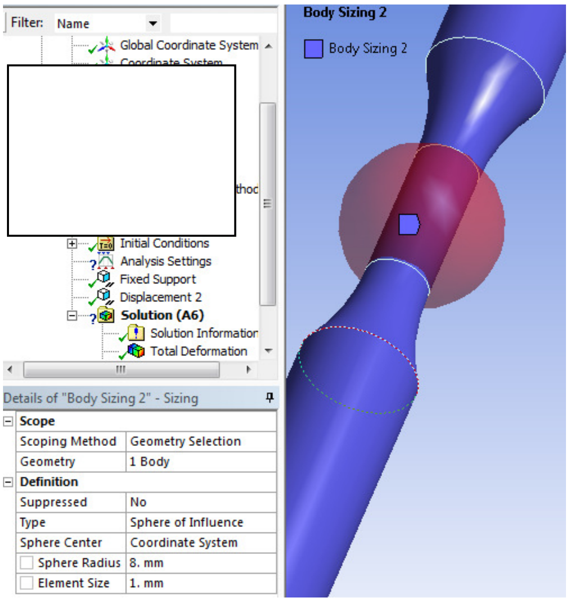


<u>Mesh:</u> Use these options.

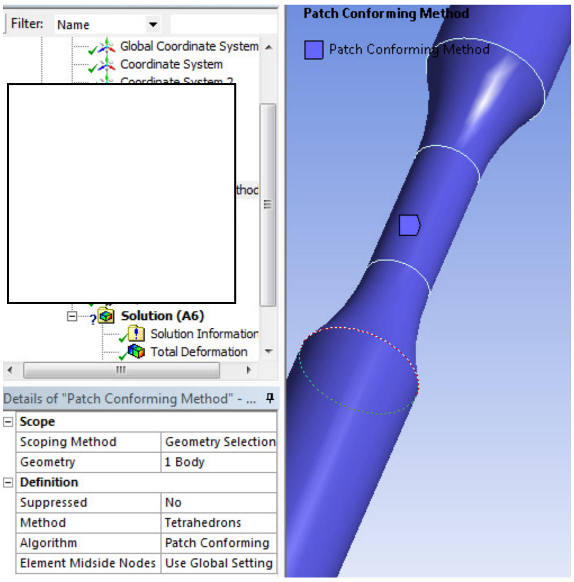


Insert a Body Sizing of 3 mm.

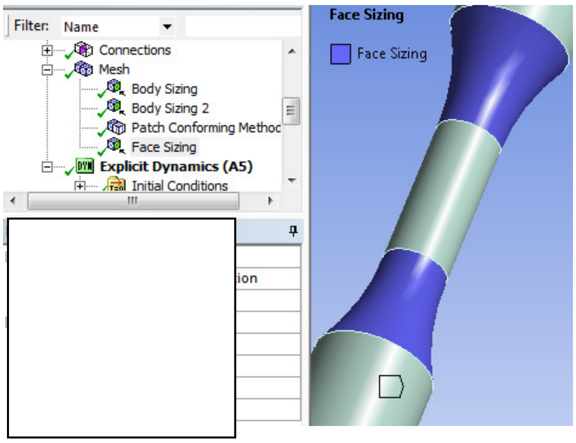




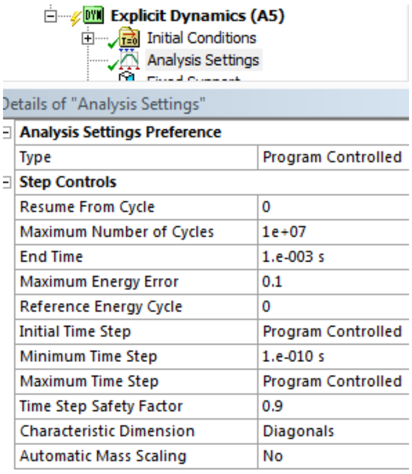
Insert Meshing Method, Tetrahedrons.



Insert a Mesh Sizing of 1.5 mm on these 2 faces.

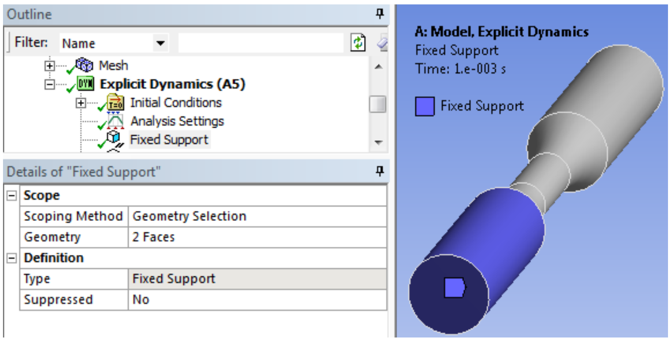


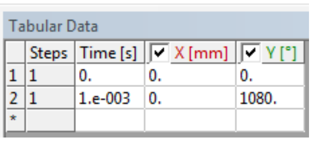
<u>Explicit Dynamics, Analysis Settings:</u> Insert these values for End Time and Minimum Time Step.



Analysis Settings Preference	
Type	Program Controlled
Step Controls	
Resume From Cycle	0
Maximum Number of Cycles	1e+07
End Time	1.e-003 s
Maximum Energy Error	0.1
Reference Energy Cycle	0
Initial Time Step	Program Controlled
Minimum Time Step	1.e-010 s
Maximum Time Step	Program Controlled
Time Step Safety Factor	0.9
Characteristic Dimension	Diagonals
Automatic Mass Scaling	No

<u>Fixed Support:</u> Apply it on one end.





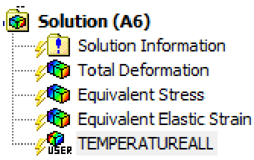
	Steps	Time [s]	X [mm]	Y [°]
1	1	0.	0.	0.
2	1	1.e-003	0.	1080.
*				

<u>Solution:</u> Default Total Deformation.

Default Equivalent Stress.

Default Equivalent Elastic Strain. Right click Solution, Solve.

Default TEMPERATUREALL, applied after solving, as seen for TENSILE case..



<u>Further homework:</u>

- refine the mesh more, solve, draw the conclusions

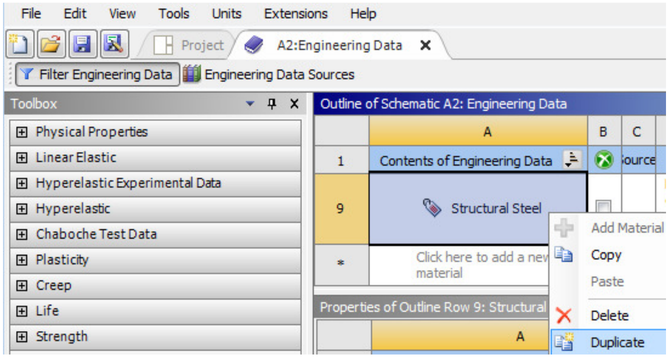
- change the default material to Structural Steel, then Structural Steel NL, solve, draw the conclusions

- change the End Time to 1e-2, solve, draw the conclusions

- This page, intentionally left blank -

CASE 3: ANSYS Workbench Explicit Dynamics FEA of a rubber part falling on metal pins

Engineering Data (Materials): Right click the default Structural Steel, Duplicate.

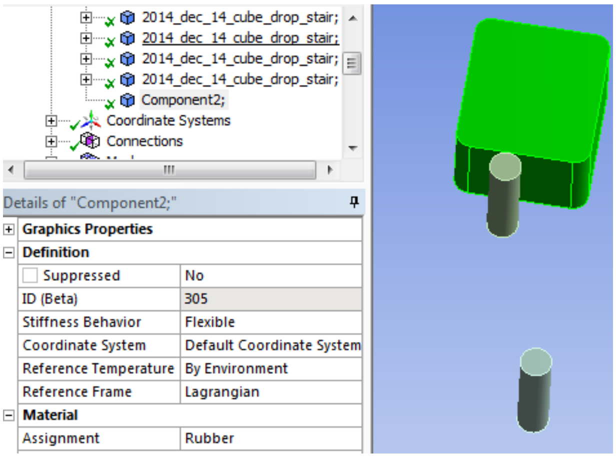


Rename the duplicate material as Rubber, then insert these values.

Geometry: 2014_dec_14_cube_drop_pins2.x_t

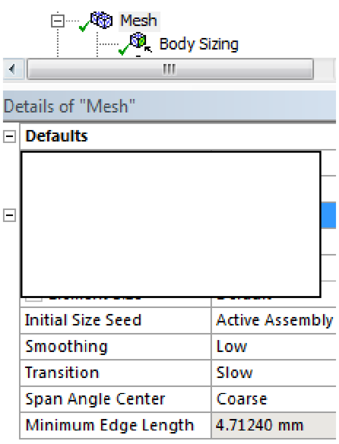
Select all bodies (seen green here), except the last one, then change their

For the rectangular body (green here), change Material, Assignment to Rubber.

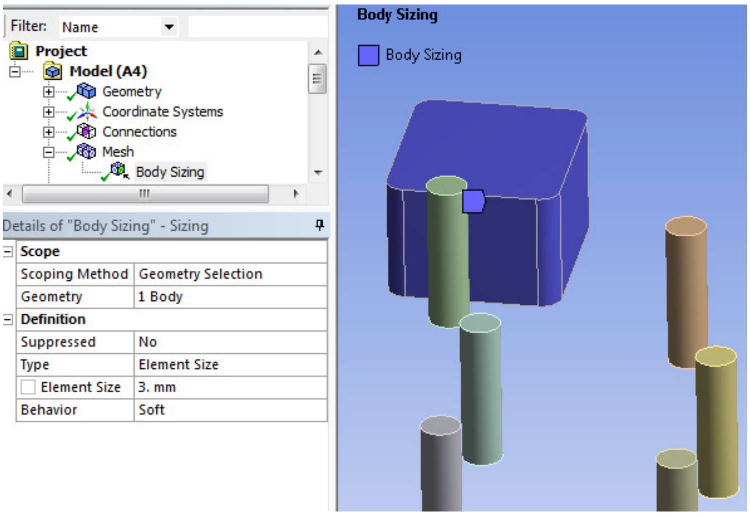


Connections: Delete any existing Contacts and keep the default Body Interactions

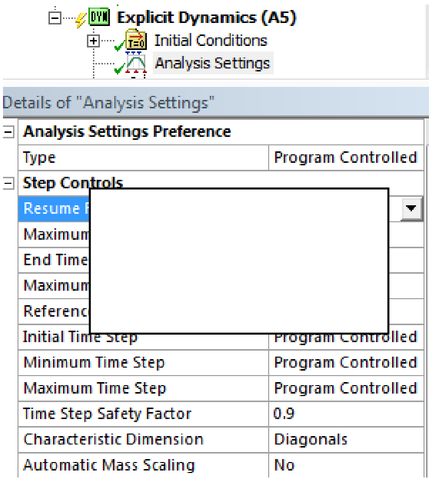
Mesh: Use these options.



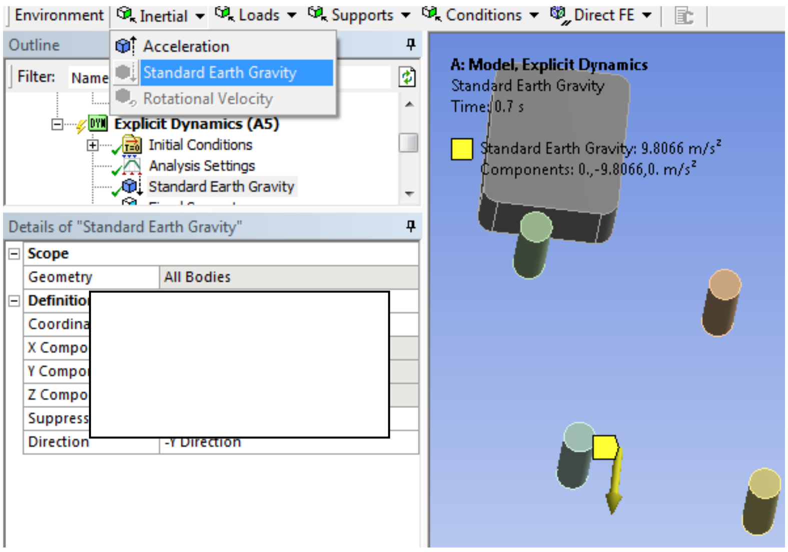
Insert a Mesh Sizing of 3 mm on the falling part (blue here).



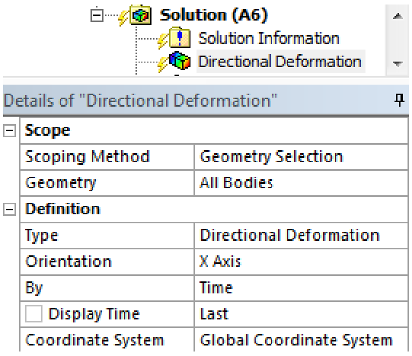
<u>Explicit Dynamics, Analysis Settings:</u> Default values and End Time =



<u>Standard Earth Gravity:</u> Apply it on -Y Direction.

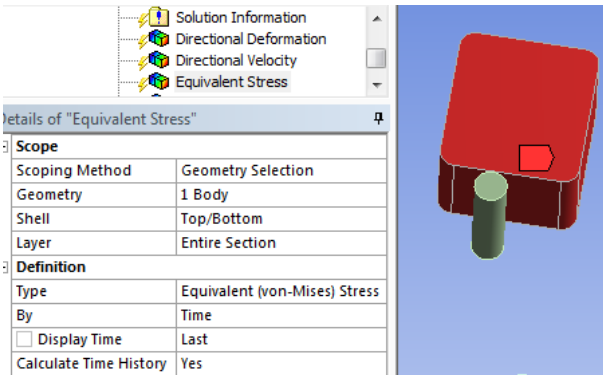


<u>Solution:</u> Default Directional Deformation on X axis, all bodies.

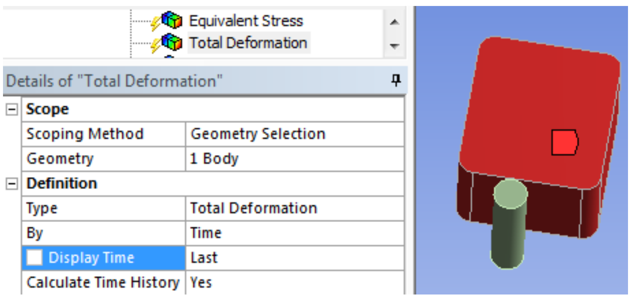


Directional Velocity on Y axis, only on the rubber body, red here.

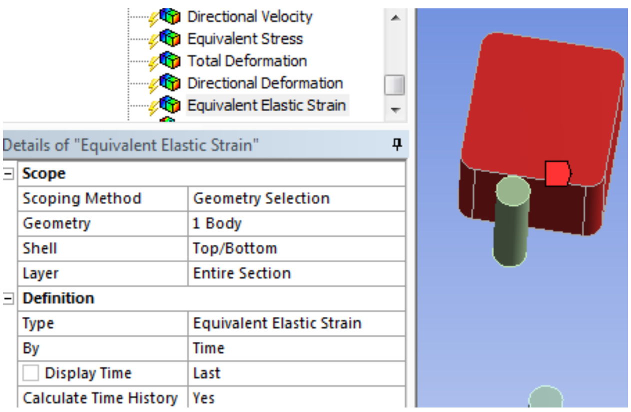
Equivalent Stress, only on the rubber body, red here.



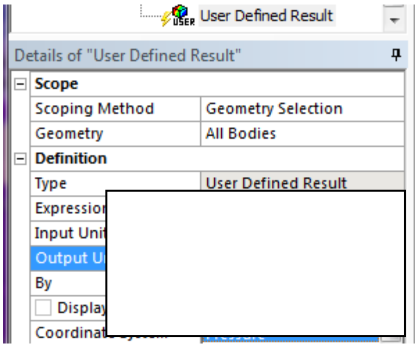
Total Deformation, only on the rubber body, red here.



Equivalent Elastic Strain, only on the rubber body, red here.



On Definition, Expression, write PRESSURE, Enter. Output Unit = Pressure. Right click Solution, Solve.



<u>Further homework:</u>

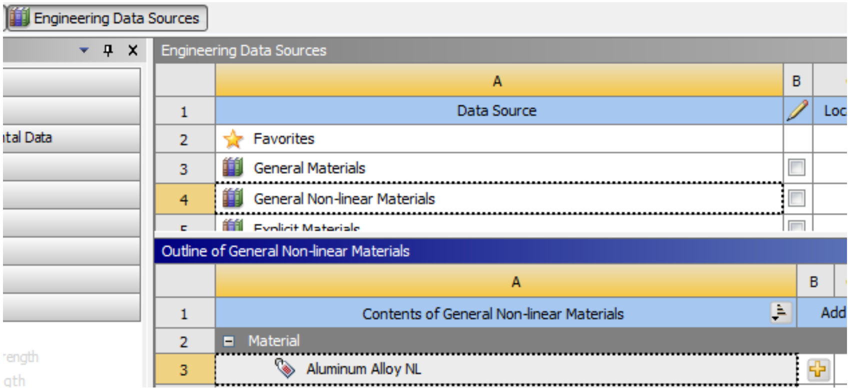
- change Young's Modulus to 0.02 MPa, solve, draw the conclusions

- refine the mesh more, solve, draw the conclusions

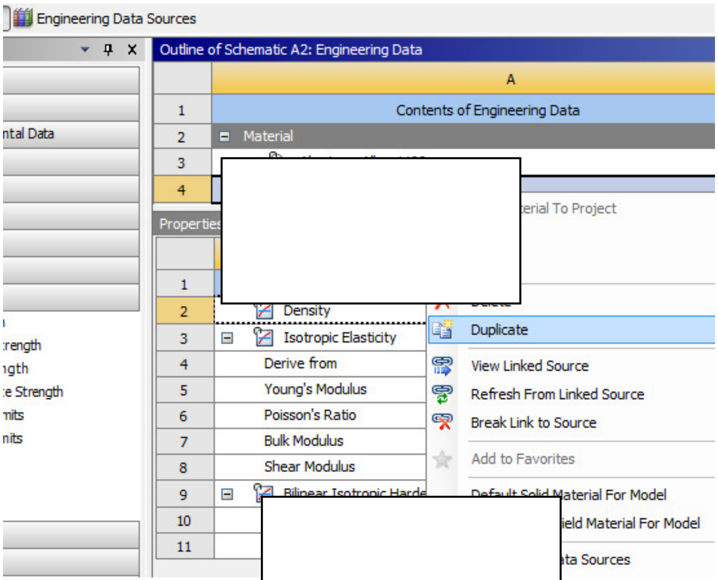
- make the pins behavior as Flexible, solve, draw the conclusions

CASE 4: ANSYS Workbench Static Structural FEA of an Aluminum sheet bending

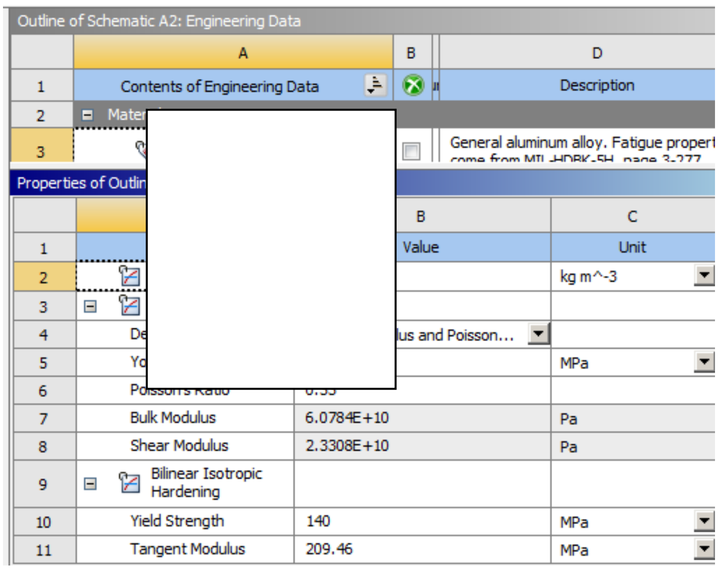
Engineering Data (Materials): Press the Engineering Data Sources button. Go to Aluminum Alloy NL from Geleral Non-Linear Materials and click the yellow plus sign in the B column, to add it to our materials.



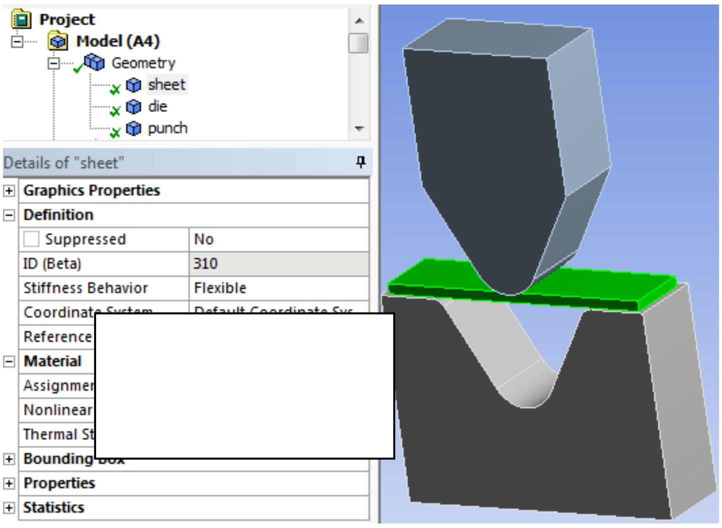
Press again the Engineering Data Sources button to access our materials. Select t ht click, Duplicate.



Name the material to change its properties to the ones shown here. Feel free to change also the units from the ▼ symbol.

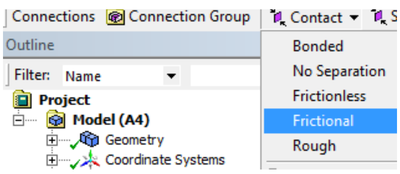


Geometry: bending.x_t

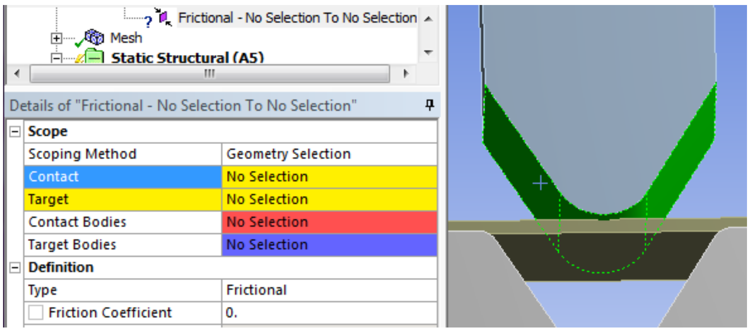


Connections, Contacts: Delete the default contacts.

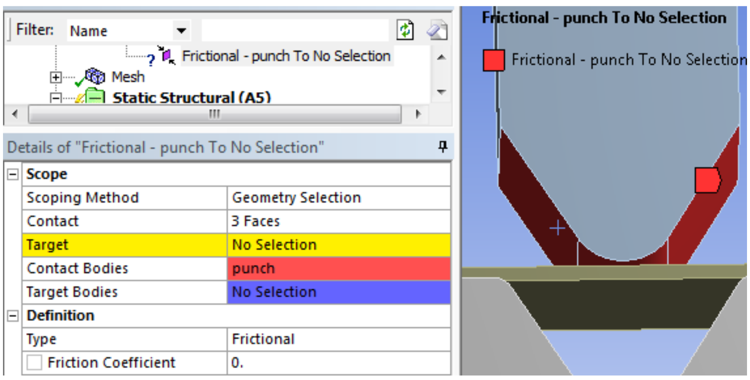
Go to the corresponding toolbar and choose Contact, Frictional.



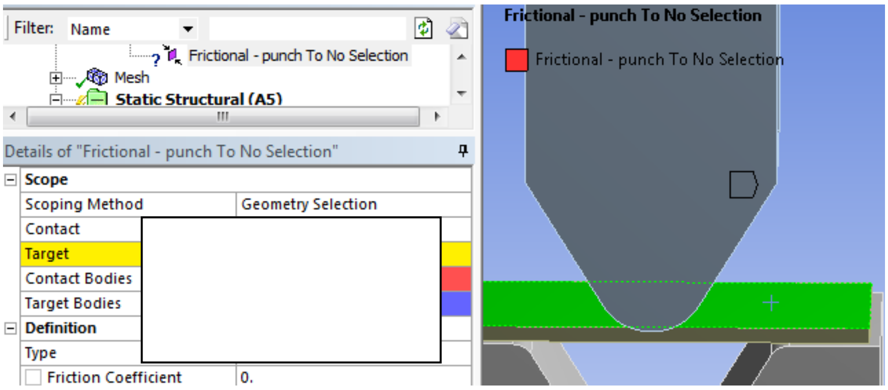
On the Punch, select the faces seen green here, go to Scope, Contact, No Selection, click, Apply.



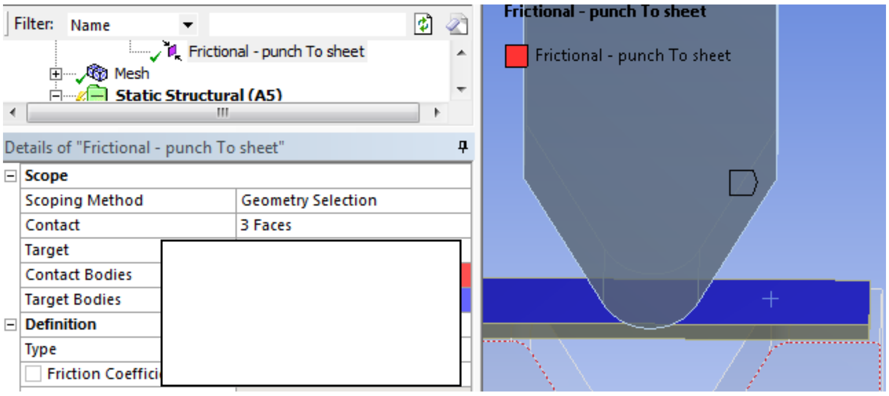
This is the result.



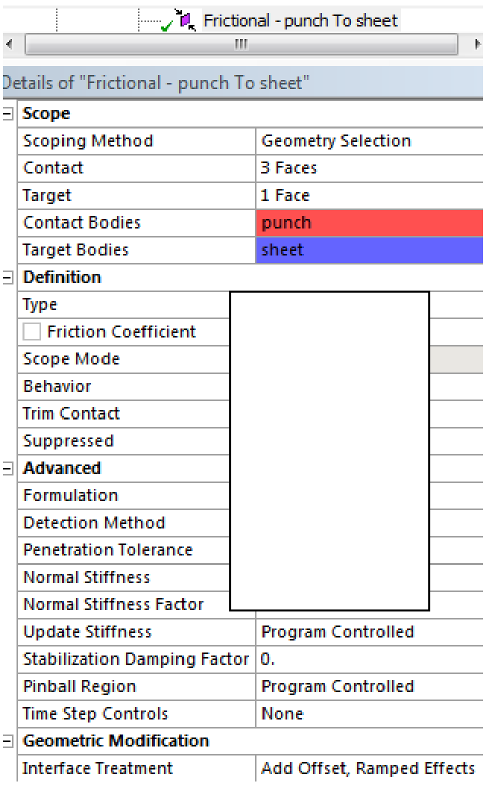
On the Sheet, select the face seen green here, go to Scope, Target, No Selection, click, Apply.



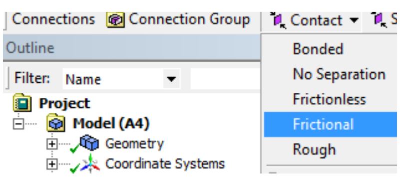
This is the result.



Assign these details.

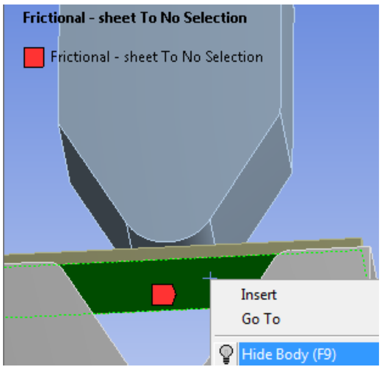


Go to the corresponding toolbar and choose Contact, Frictional.

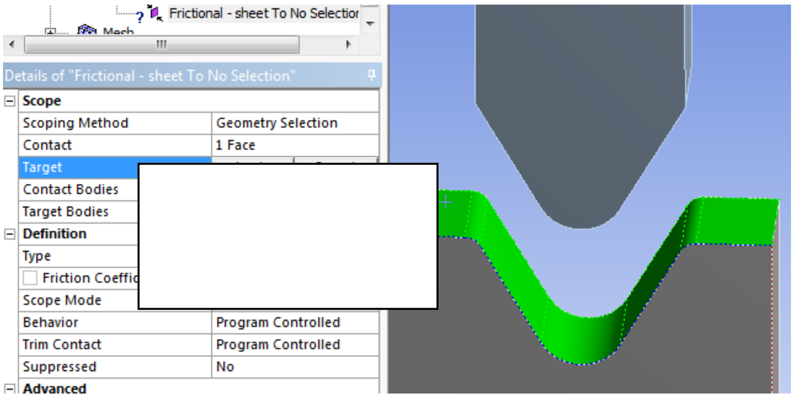


On the Sheet, select the faces seen green here, go to Scope, Contact, No Selection, click, Apply.

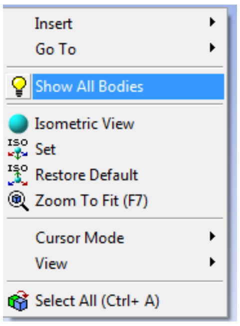
Select the Sheet, right click, Hide Body.



Select these green faces on the Die, go to Scope, Contact, No Selection, click, Apply.

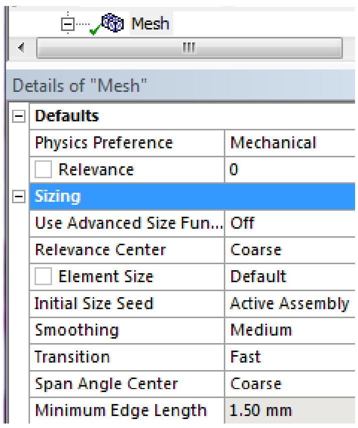


Right click anywhere, Show All Bodies.

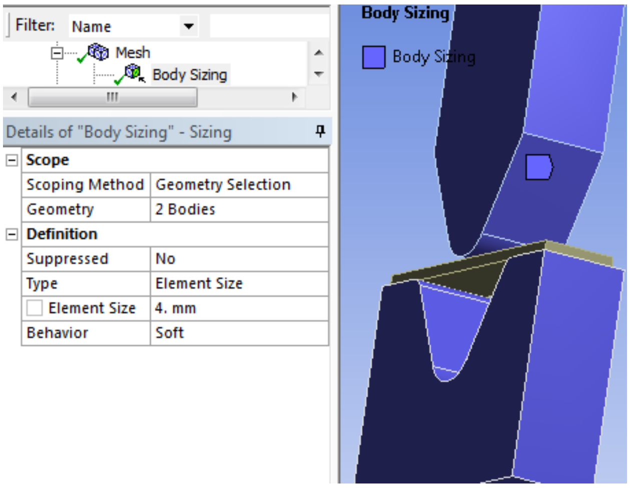


Assign these details.

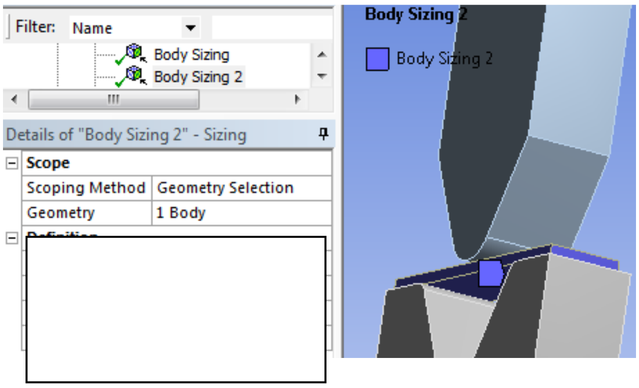
<u>Mesh:</u> It should have these details.



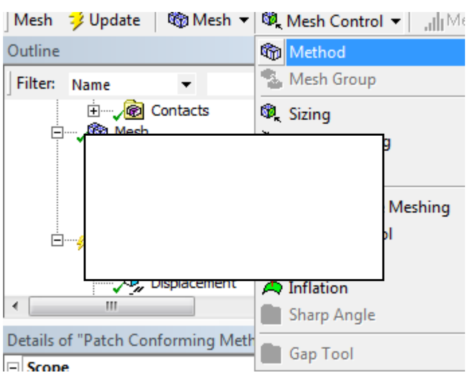
On the Punch and Die, blue here, apply a Mesh Sizing



On the Sheet, blue here, apply a Mesh Sizing of



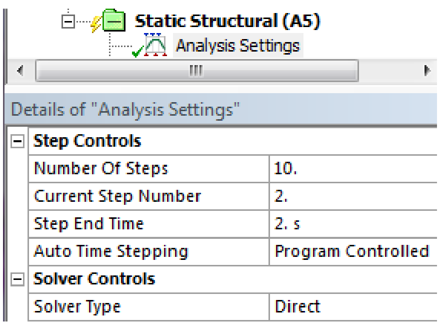
Go to Mesh Control, Method.



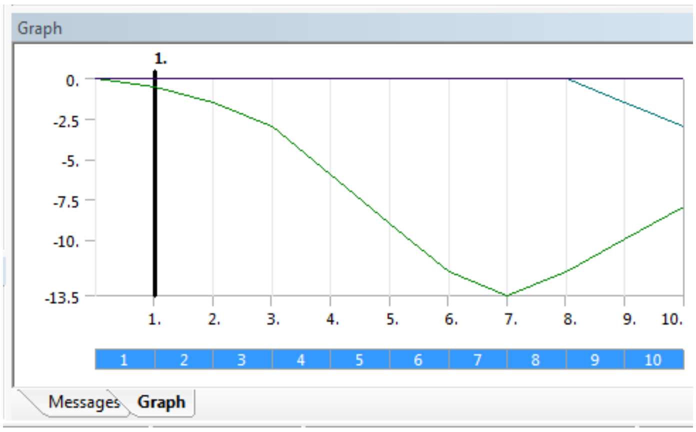
Make Definition, Method, Tetrahedrons, on the Sheet.

After you generate the Mesh, it should look like this.

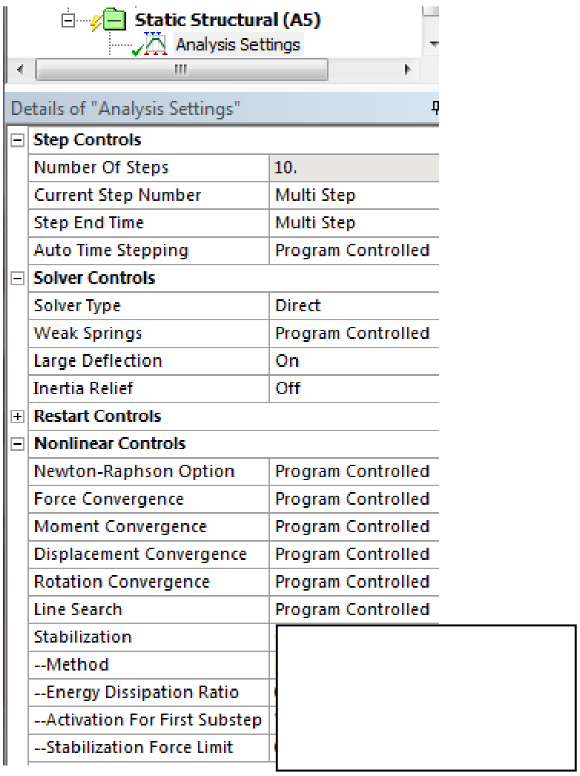
<u>Static Structural, Analysis Settings:</u> Make Number of Steps = 10 and Solver Type = Direct.



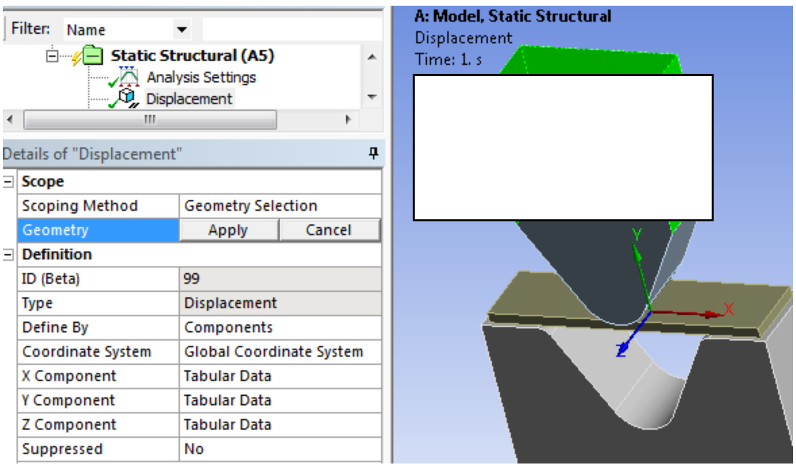
In the Graph tab select with Shift or Ctrl pressed all 10 steps, thus making the numbering cells blue.



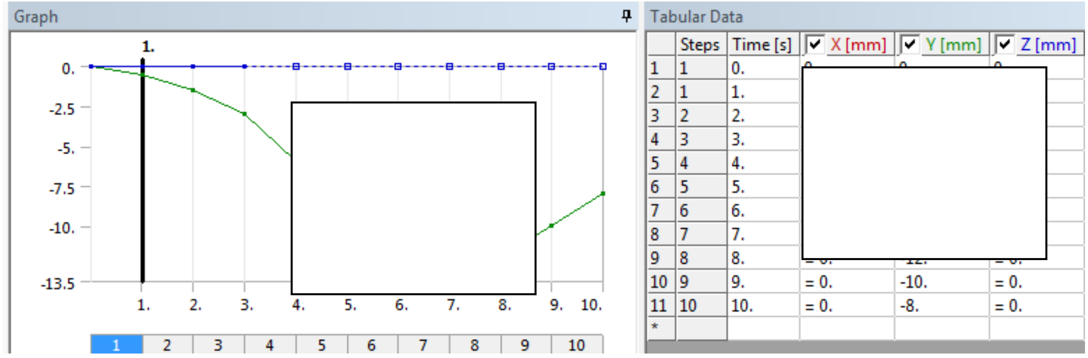
Insert these details for all 10 steps in the same time. *Remember the Stabilization values, because they will help you converge almost any FEA scenario in the future!*



Static Structural: Insert a Displacement on these 3 green faces on the punch, Apply.

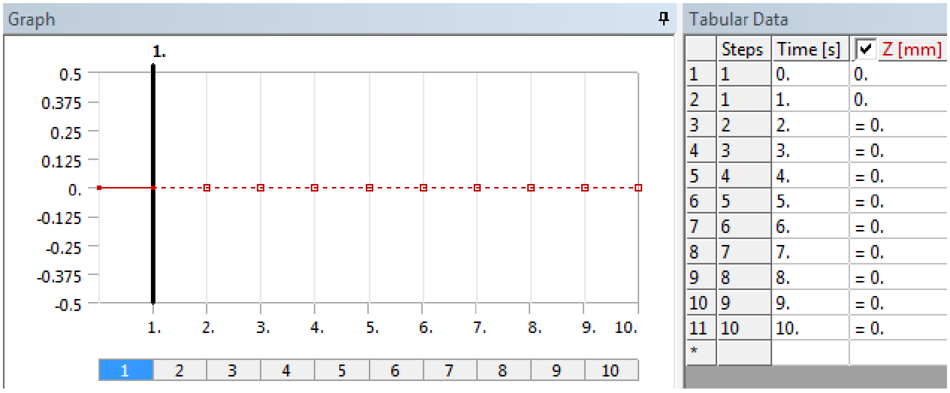


The Tabular Data and Graph should look like here.

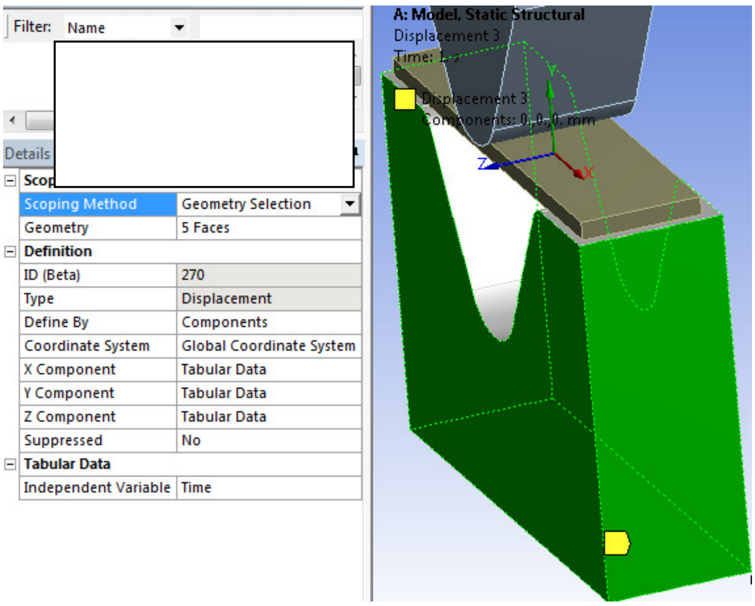


Insert a Displacement on the 2 lateral faces (normal to Z axis), of the Sheet, green here, Apply.

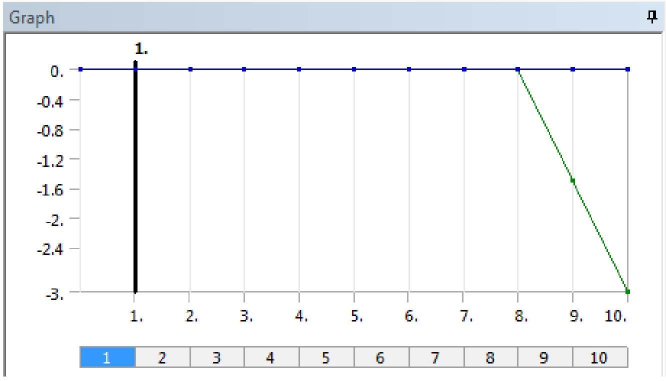
The Tabular Data and Graph should look like here.

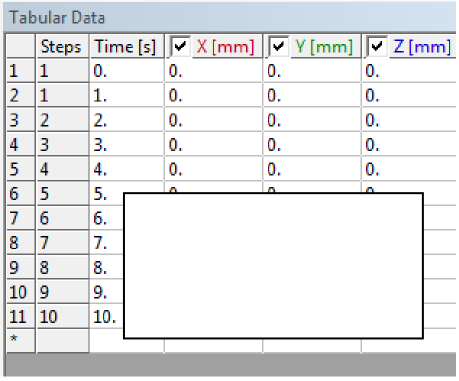


Insert a Displacement by selecting all the outer faces, green here, of the Die, except the ones inside, where the Sheet will be bent and touch the Die, Apply. Make sure that you have selected 5 faces.



The Tabular Data and Graph should look like here.

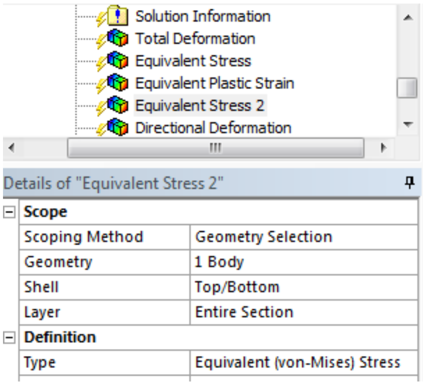


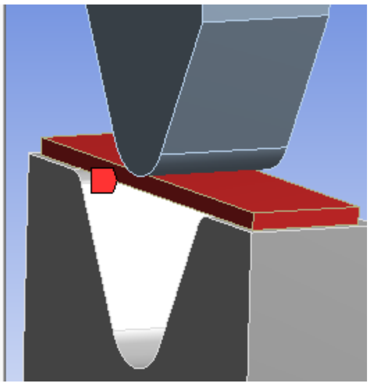

Tabular Data

	Steps	Time [s]	X [mm]	Y [mm]	Z [mm]
1	1	0.	0.	0.	0.
2	1	1.	0.	0.	0.
3	2	2.	0.	0.	0.
4	3	3.	0.	0.	0.
5	4	4.	0.	0.	0.
6	5	5.			
7	6	6.			
8	7	7.			
9	8	8.			
10	9	9.			
11	10	10.			
*					

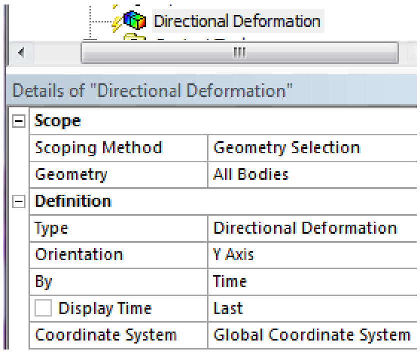
Solution: Insert default Total Deformation, Equivalent Stress and Equivalent Plastic Strain on all bodies.

Insert an Equivalent Stress only in the Sheet.





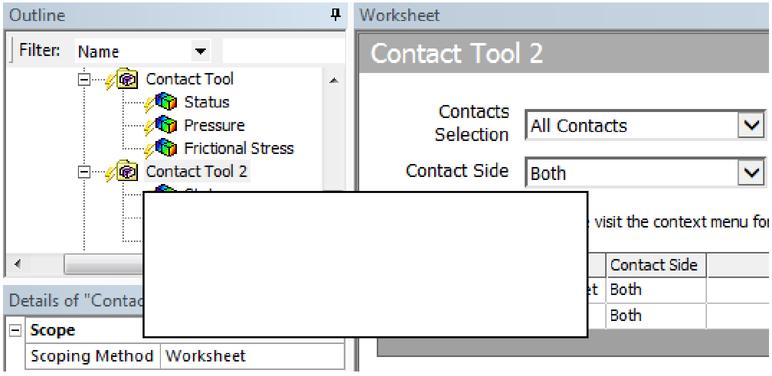
Insert Directional Deformation about the Y axis, on all bodies.



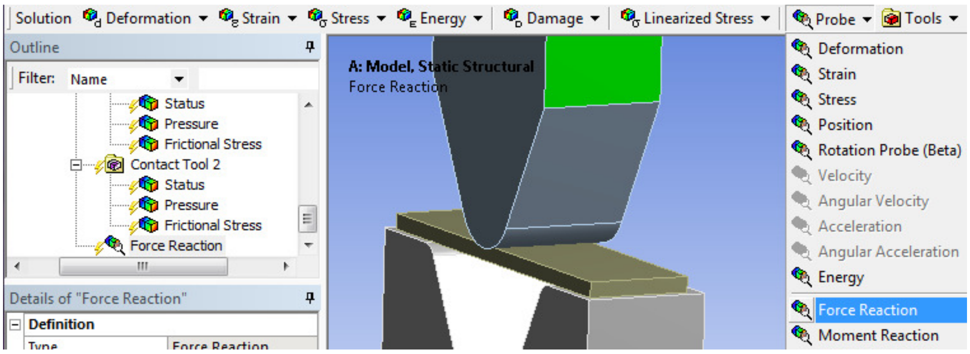
From the Solution toolbar, insert Tool, Contact Tool.

Right click Contact Tool and Insert Frictional Stress and Pressure. Right click Contact Tool, Duplicate.

Click the Contact Tool 2 and uncheck the 2nd contact, to keep the 1st, as seen here.



From the Solution toolbar, insert probe, Force Reaction.



Make Boundary Condition = Displacement (the one we created on the punch). Solve.

<u>Further homework:</u>

- increase the Friction Coefficient to 0.19, solve, draw the conclusions

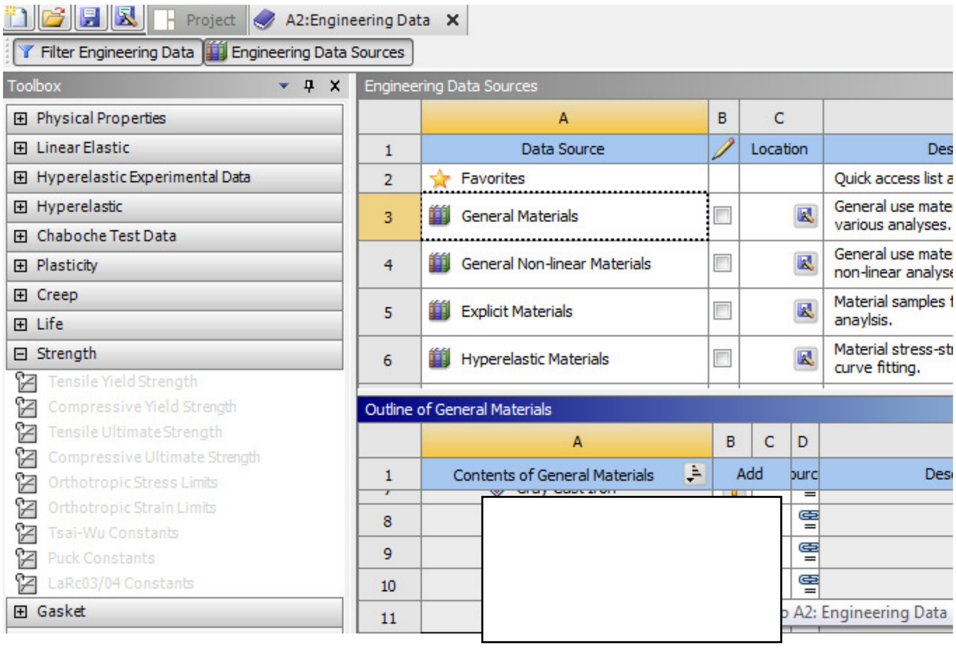
- refine the mesh of the sheet metal part, solve, draw the conclusions

- make the Die and Punch behavior as Rigid, solve, draw the conclusions

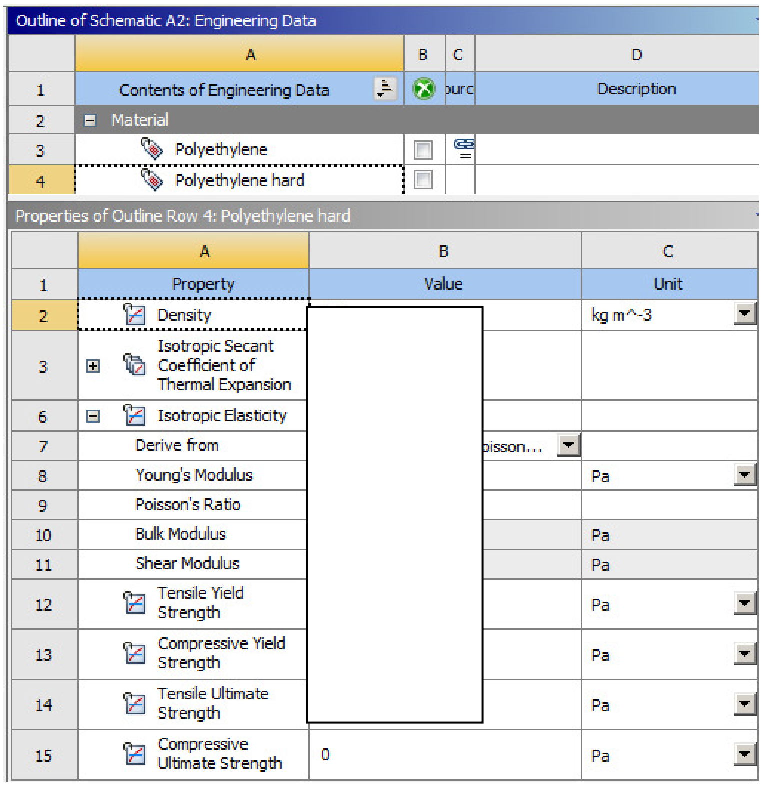
- change the material of the sheet metal part to Copper Alloy NL, solve, draw the conclusions

CASE 7: ANSYS Workbench Biomechanics FEA of the treatment of a tibial plateau

Engineering Data (Materials): Press Engineering Data Sources then go to General Materials and click the yellow plus sign from column B at

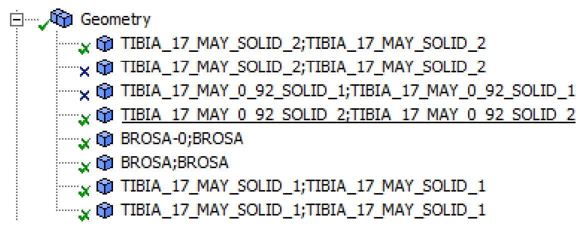


Assign these values for



Geometry: 2015_jul_27_tibia_brose1.x_t

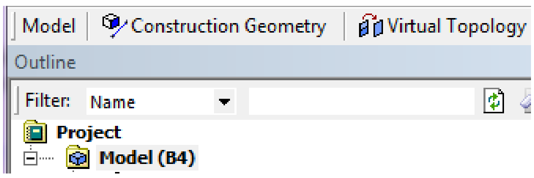
Suppress the 2nd and 3rd body. You should have this configuration.

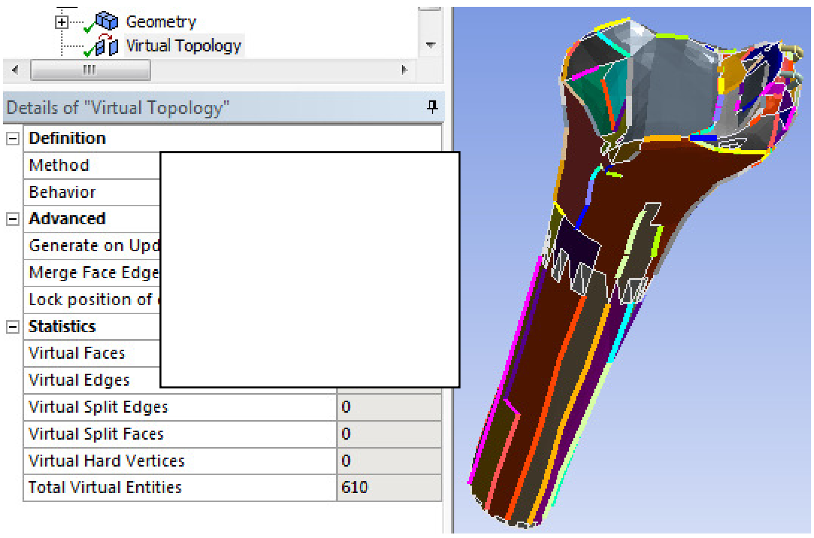


Assign to the 1st and 7th body, highlighted here in grey, the Polyethylene hard.

Assign to the 4th and 8th body, highlighted here in grey, the Polyethylene soft.

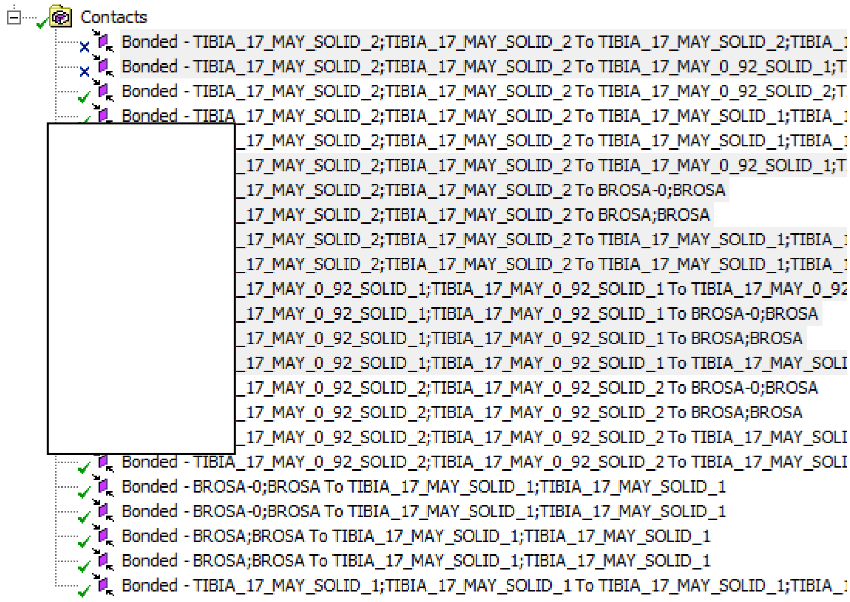
Select Model, Virtual Topology.





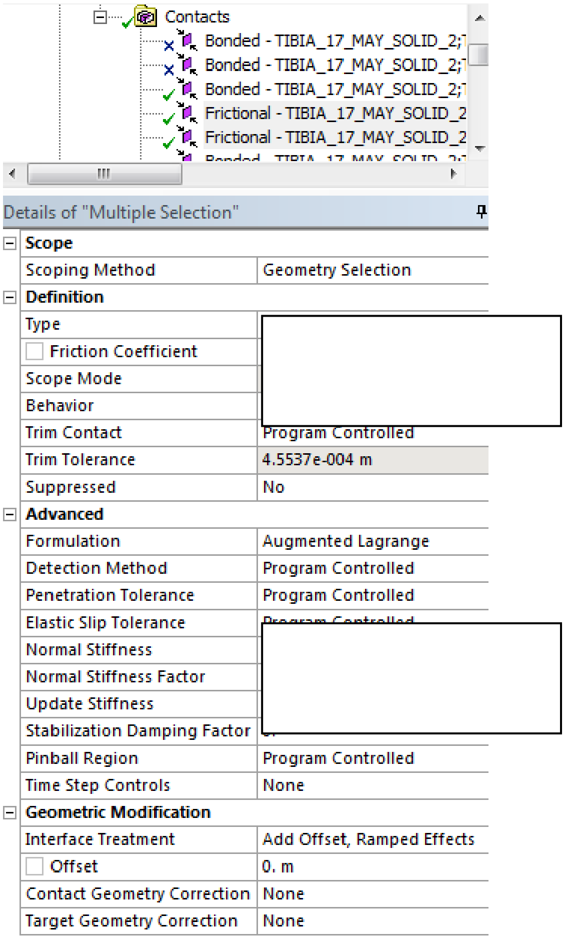
<u>Connections, Contacts:</u> Right click, Rename Based on Definition.

Suppress the first 2 contacts, then from 6th to 14th. They should look like here.



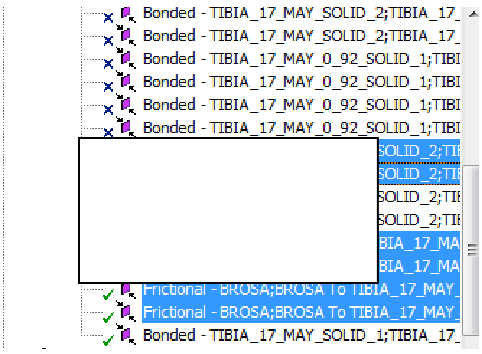
For the 3rd contact make Advanced, Formulation, MPC.

For the 4th and 5th contact insert these details.



Make the next available contacts as Frictional (except the last one), with the same details as above.

Assign to these contacts highlighted in blue a Friction Coefficient of 0.2.



For the last contact make Advanced, Formulation = MPC.

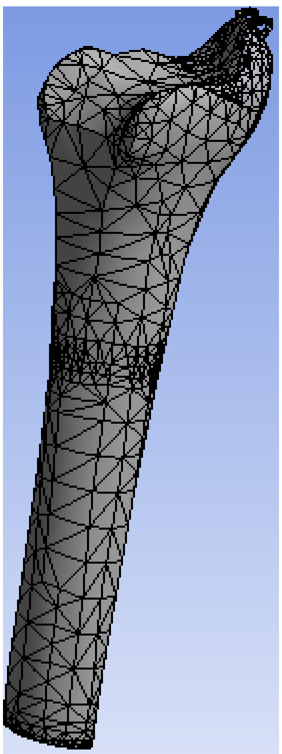
<u>Mesh:</u> It should be default, with these details.

Mesh

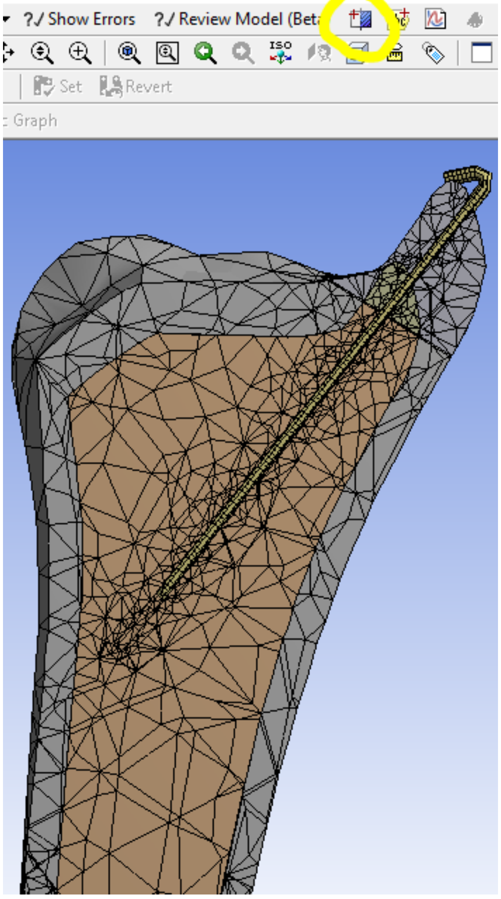
Details of "Mesh"

Defaults	
Physics Preference	Mechanical
Relevance	0
Sizing	
Use Advanced Size Fun...	Off
Relevance Center	Coarse
Element Size	Default
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Fast
Span Angle Center	Coarse
Minimum Edge Length	1.4391e-005 m

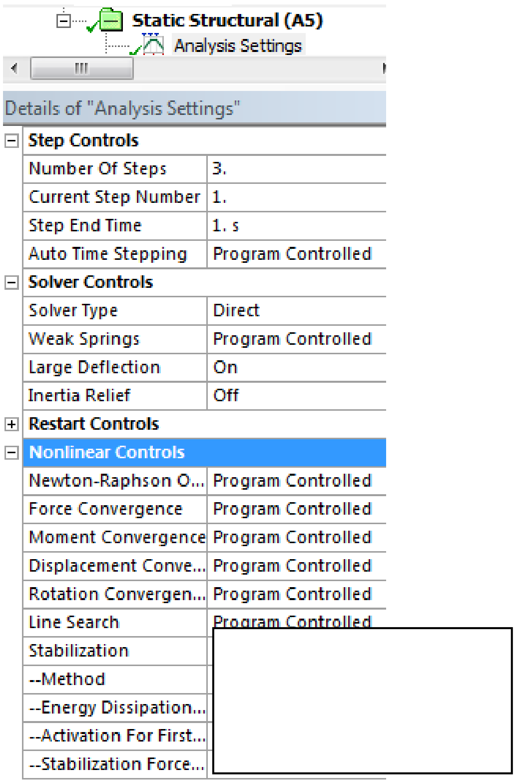
After you generate the mesh, it should look like here.



If you create a Section Plane (yellow circle) thru one of the broaches (wires), the mesh should look like here.



Static Structural, Analysis Settings: Make Number Of Steps = 3 then insert these details for each step.

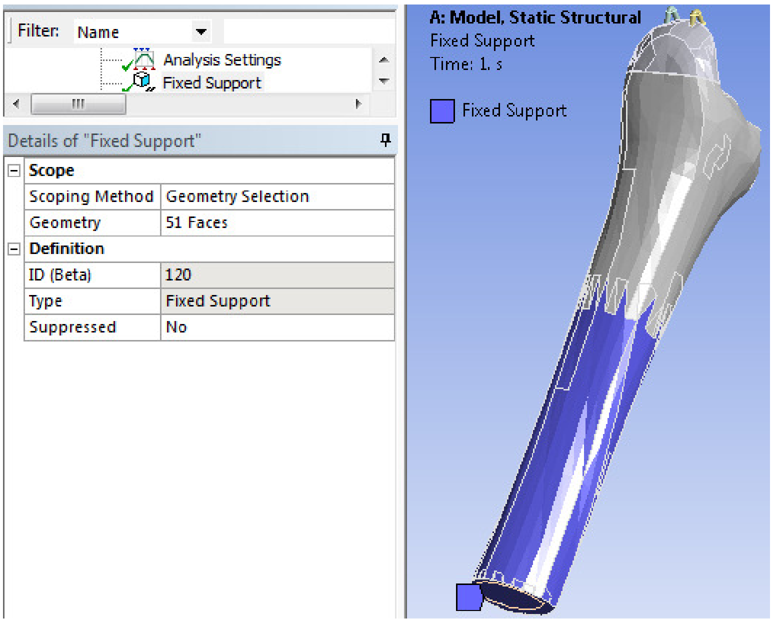


Static Structural (A5)
Analysis Settings

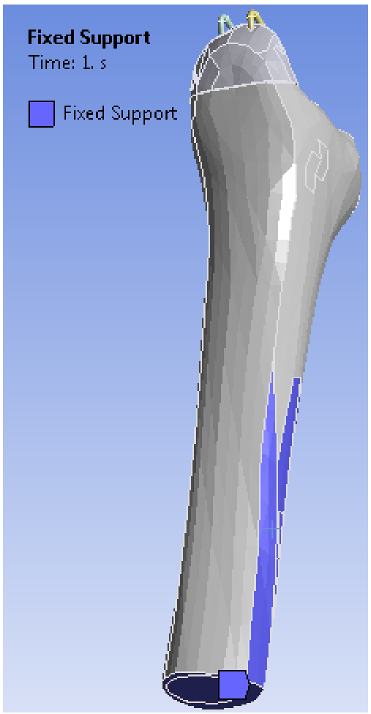
Details of "Analysis Settings"

Step Controls	
Number Of Steps	3.
Current Step Number	1.
Step End Time	1. s
Auto Time Stepping	Program Controlled
Solver Controls	
Solver Type	Direct
Weak Springs	Program Controlled
Large Deflection	On
Inertia Relief	Off
Restart Controls	
Nonlinear Controls	
Newton-Raphson O...	Program Controlled
Force Convergence	Program Controlled
Moment Convergence	Program Controlled
Displacement Conve...	Program Controlled
Rotation Convergen...	Program Controlled
Line Search	Program Controlled
Stabilization	
--Method	
--Energy Dissipation...	
--Activation For First...	
--Stabilization Force...	

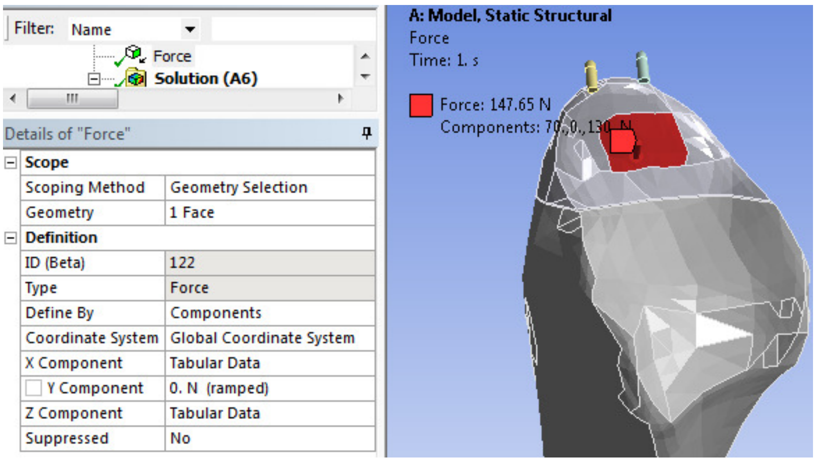
Insert a Fixed Support with approximation on the lower half of the assembly, as seen here in blue. Or use similar faces, depending on how the Virtual Topology simplified the model.



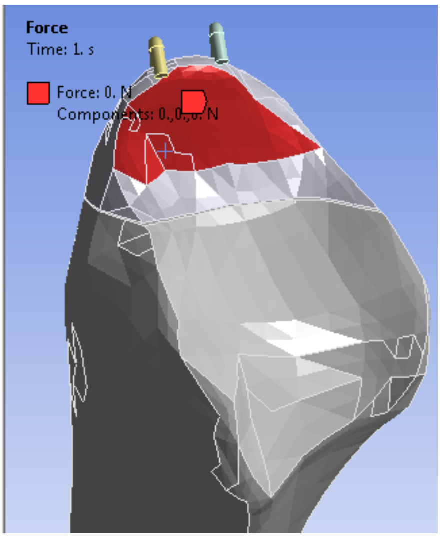
Selection with a different topology of faces.



Insert a force on this area, red here, depending on the available faces.



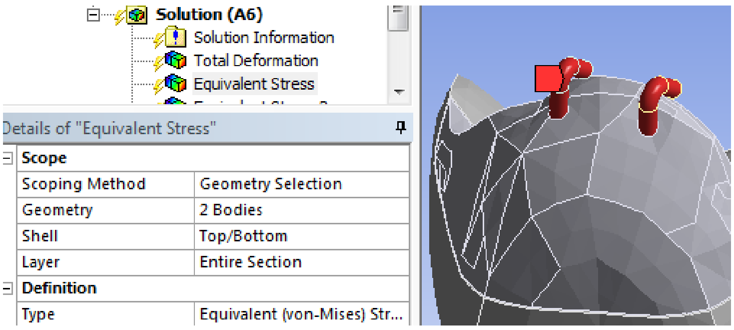
Another possibility, depending on your topology.



The Tabular Data of the Force should look like here.

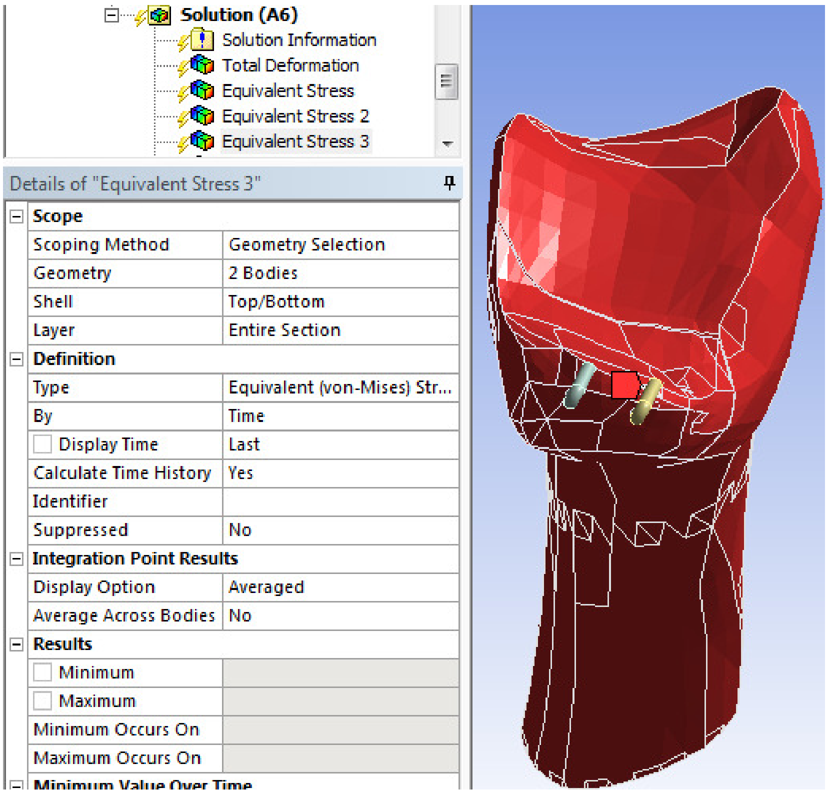
<u>Solution:</u> Default Total Deformation of all bodies.

Equivalent Stress only in the 2 broaches, red here.

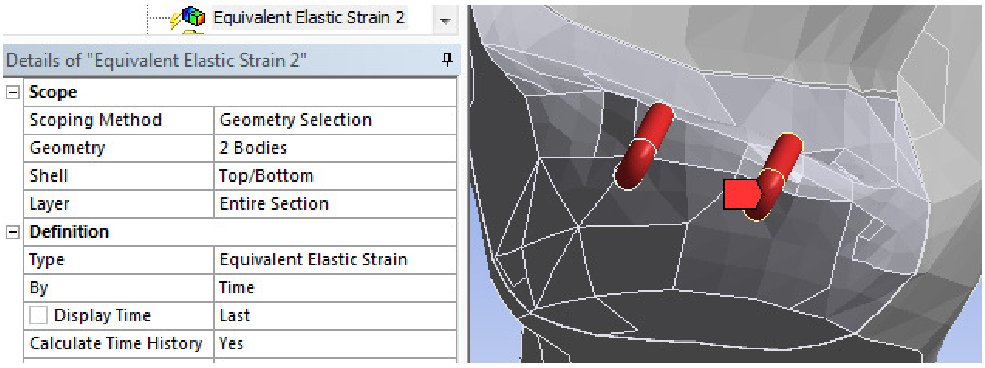


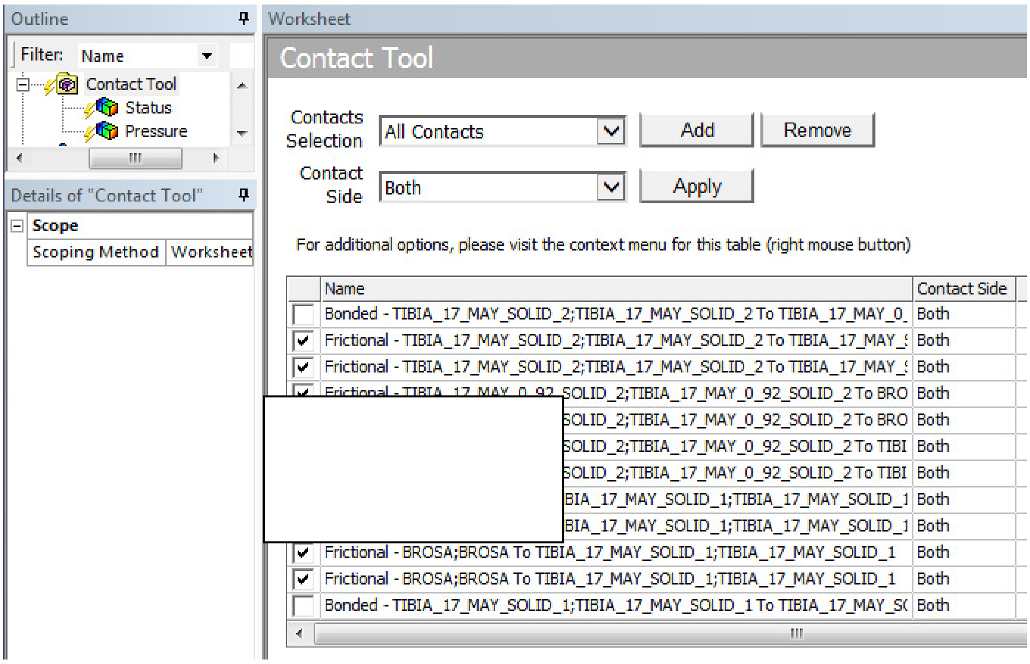
Equivalent Stress only in the 2 condyles (inner and outer), red here.

Equivalent Stress only in the 2 outer parts of the tibia, red here.

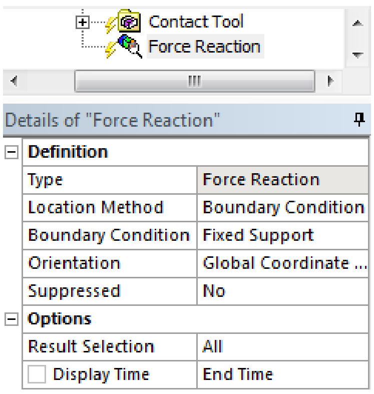


Equivalent Elastic Strain only in the 2 broaches, red here.





Insert a Force Reaction with Boundary Condition = Fixed Support. Solve.



Further homework:

- suppress the Virtual Topology, does the mesh run? If yes, solve, draw the conslusions

- double the force values, solve, draw the conclusions

- decrease the Friction Coeffieient to half, solve, draw the conclusions

- refine the mesh, solve, draw the conclusions

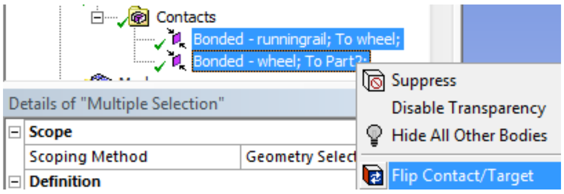
CASE 10. ANSYS Workbench Static Structural FEA simulation of a rotating train wheel on a railway (trial)

Geometry: 2013_10_28_wheel_rail2_gap.x_t

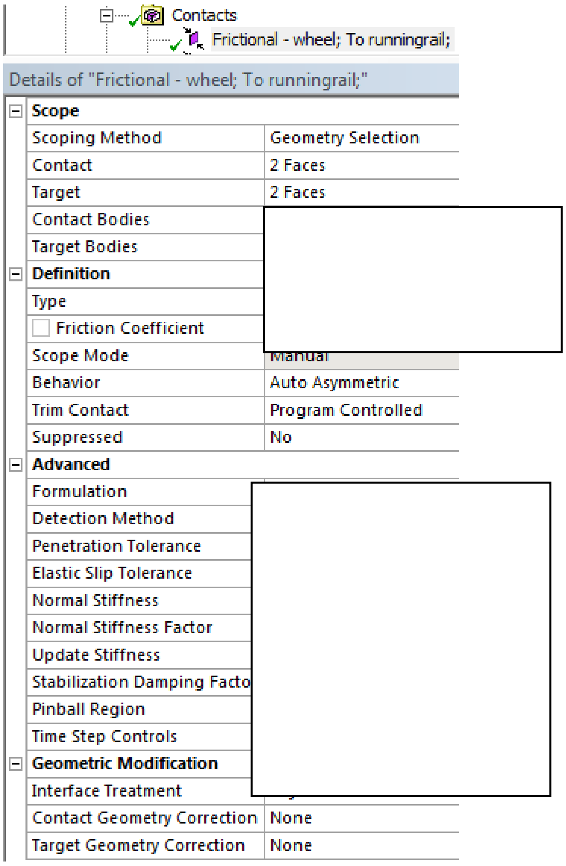
All bodies are from default Structural Steel.

Connections, Contacts: Right click Contacts, rename Based on Definition.

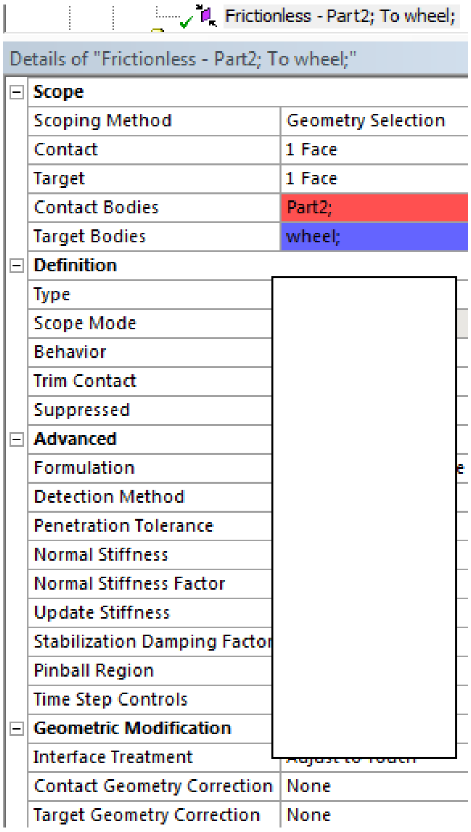
Select both contacts, right click and flip their sides as here.



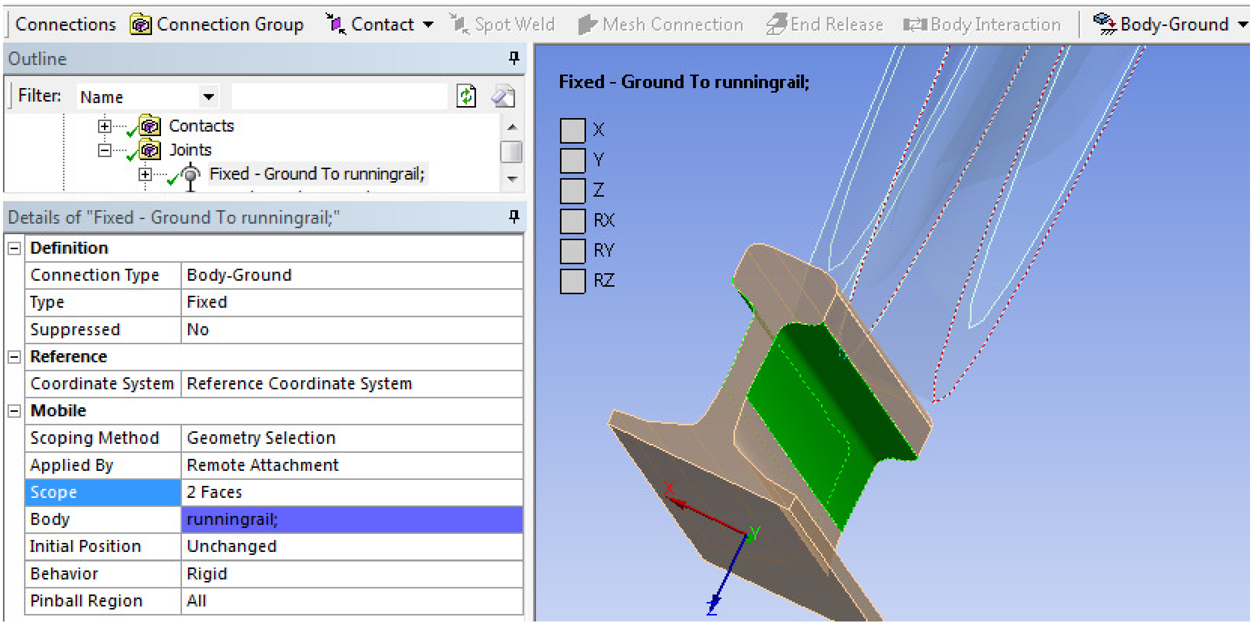
For the 1st contact insert these details.

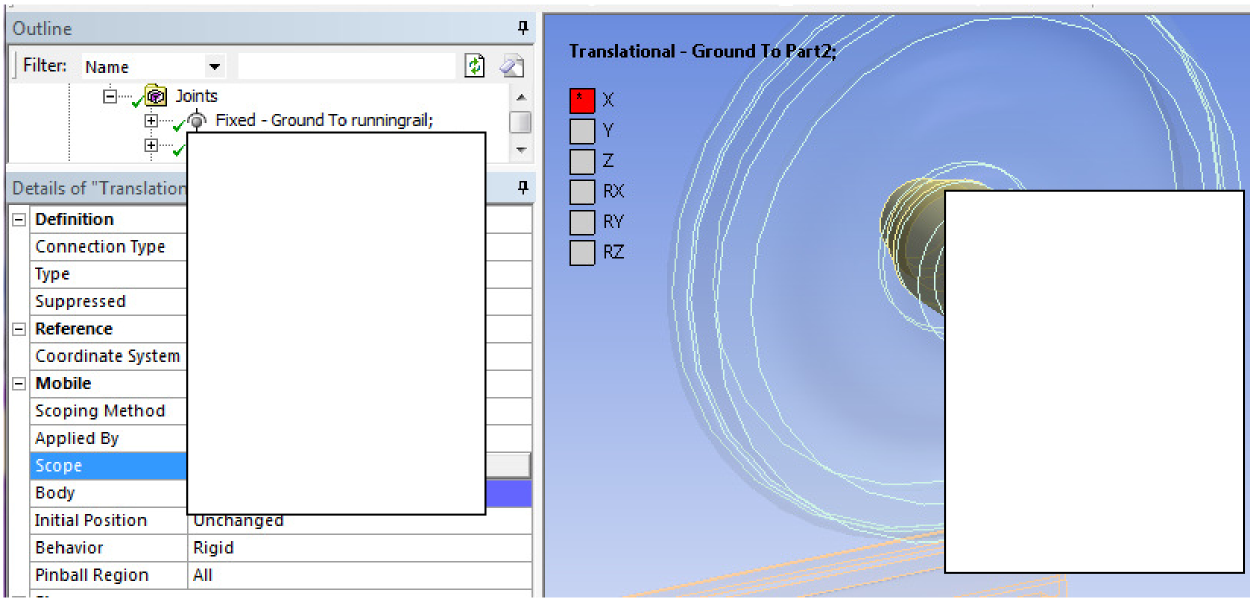


For the 2nd contact insert these details.

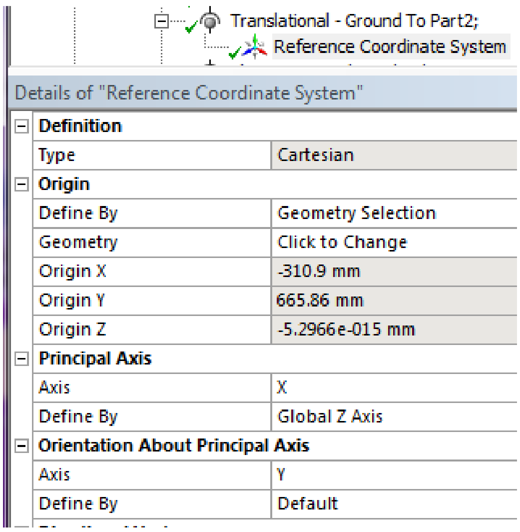


From the Connections toolbar insert Body-Ground, Fixed on the lateral faces of the rail, shown here in green.

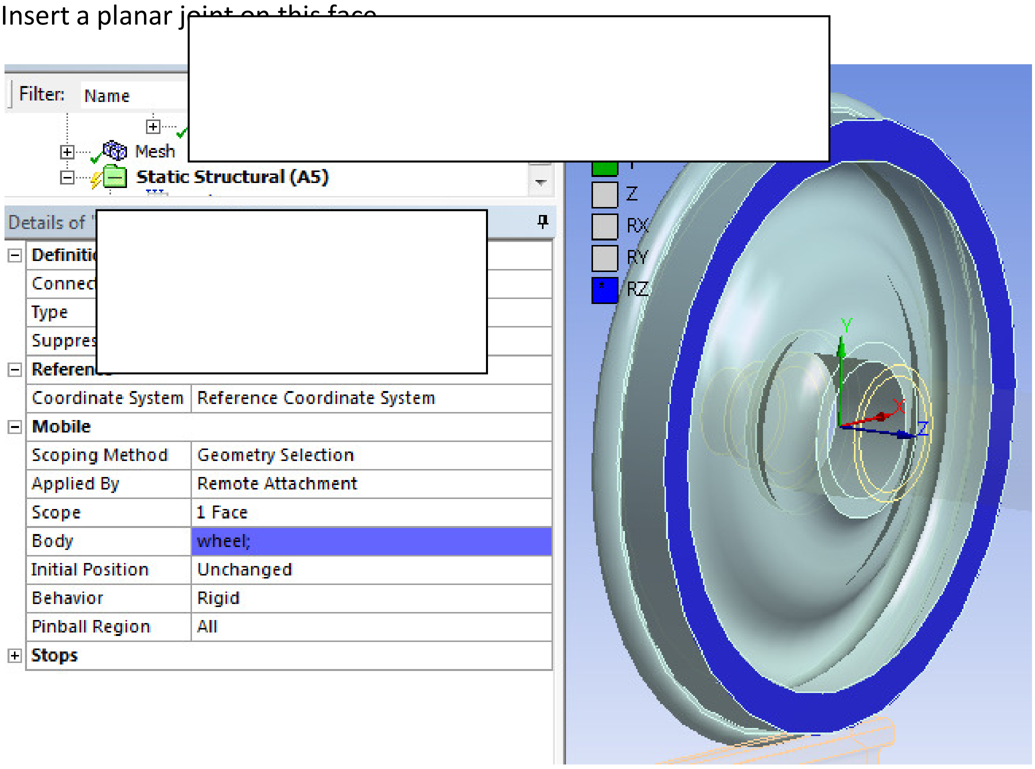




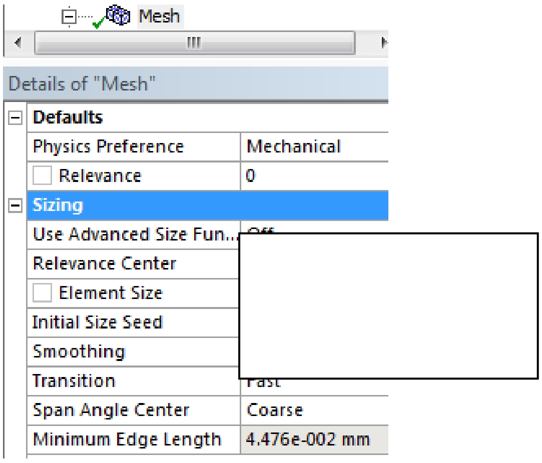
Expand it to reach Reference Coordinate System and assign these details.



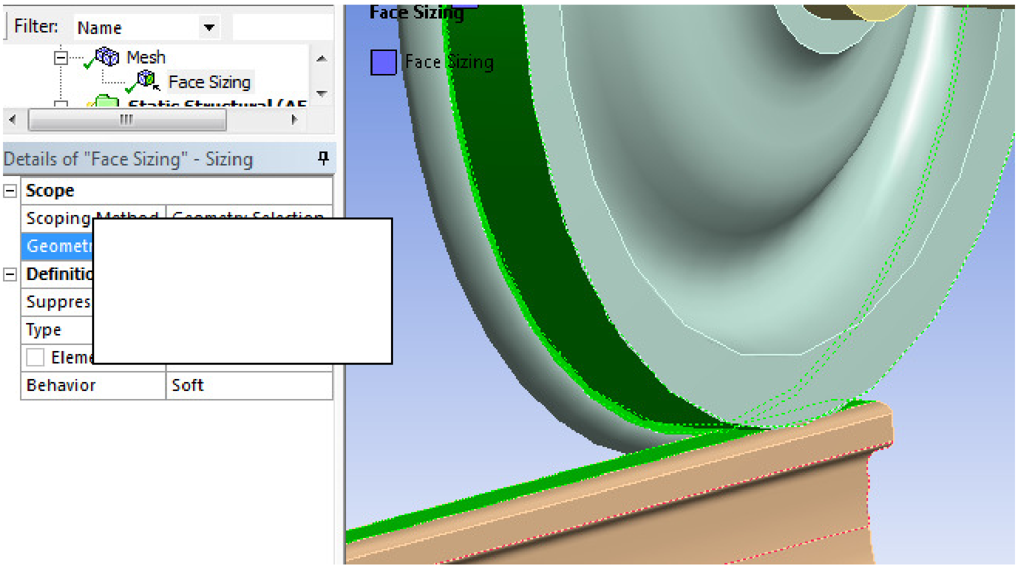
Insert a planar joint on this face.



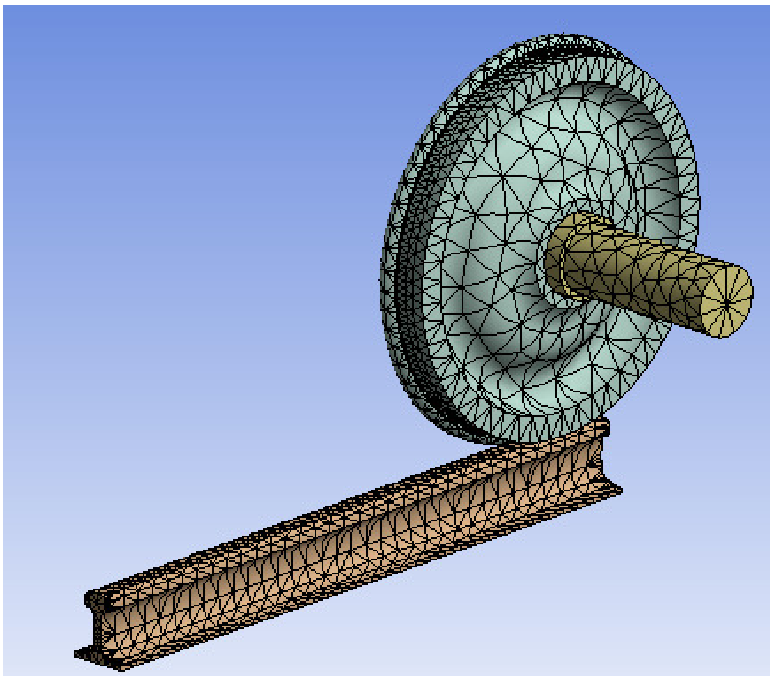
Mesh: Insert these details to the mesh.



Insert a Mesh Sizing of to these 3 faces, green here.

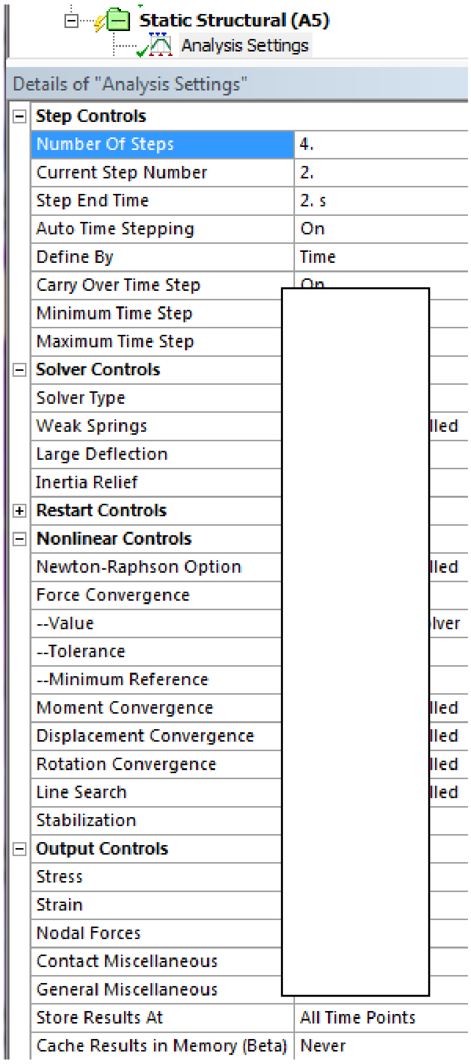


In the end, the mesh should look like here.



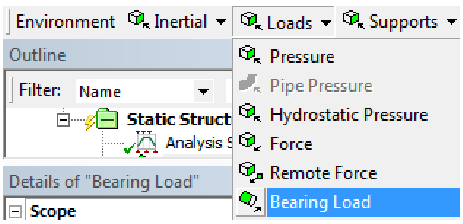
Static Structural, Analysis Settings: Make Number of Steps = 4.

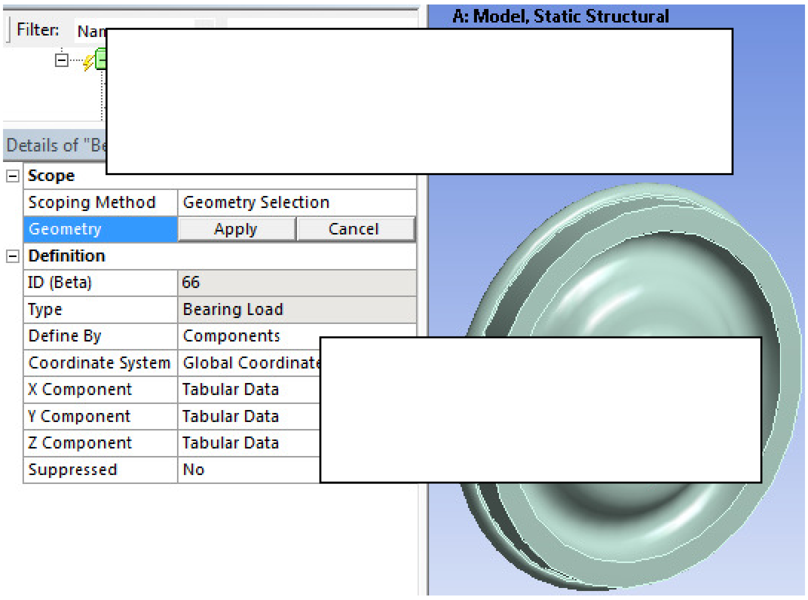
Assign these details to the 1st timestep. Output Controls, Yes 5 times.



Assign these details to the other timesteps.

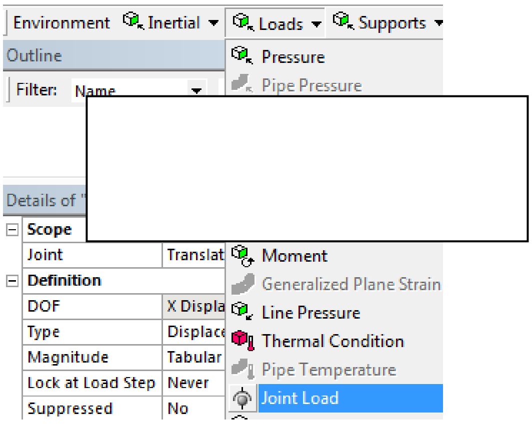
After you hide the other bodies, from the Environment toolbar, insert a Bearing Load inside the wheel, on the green face.

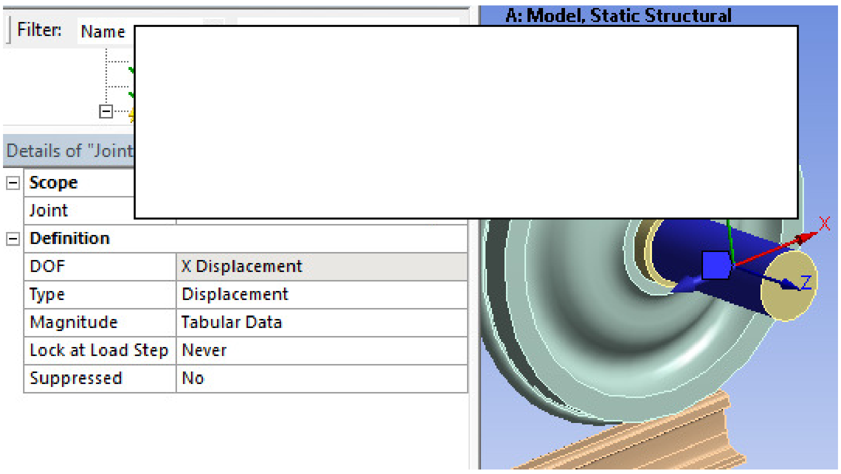




Apply these values:

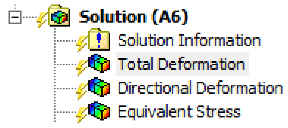
From the Environment toolbar, insert a Joint Load selecting the



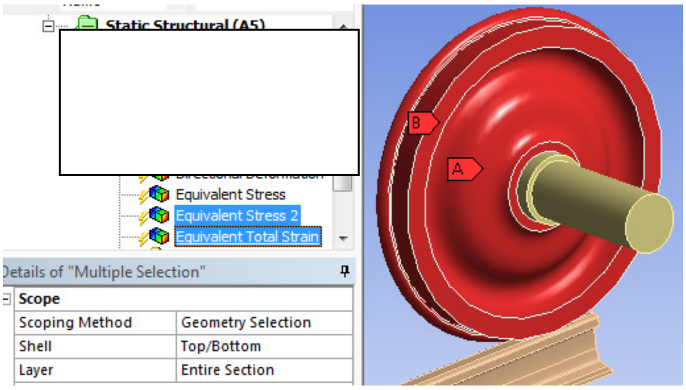


Apply these values, making sure that the direction of the blue arrow is towards the lenght of the rail, not opposite.

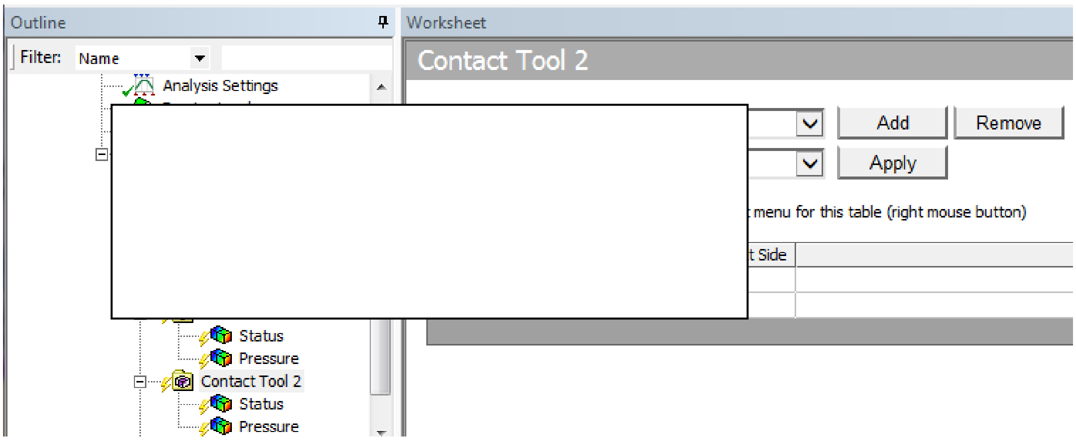
Solution: Insert these items as default, for all bodies.



Insert an Equivalent Stress and Total Strain only on the wheel, red here.

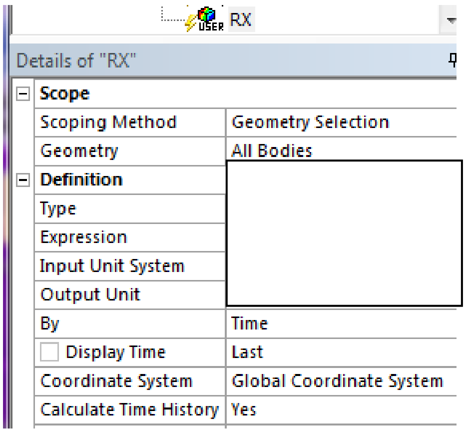


Insert 2 separate Contact Tools, one for each contact pair. Insert also a Pressure item in them.



Insert a Probe, Force Reaction as seen here.

For the angle about global X axis, Create a User Defined Result with these details. Solve.



<u>**Further homework:**</u>

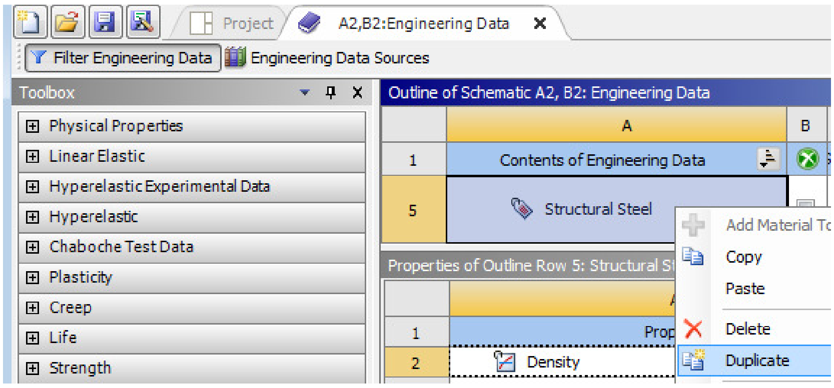
- change Frictionless to Frictional with 0.1 Friction Coefficient, solve, draw the conclusions

- double the Bearing Load, solve, draw the conclusions

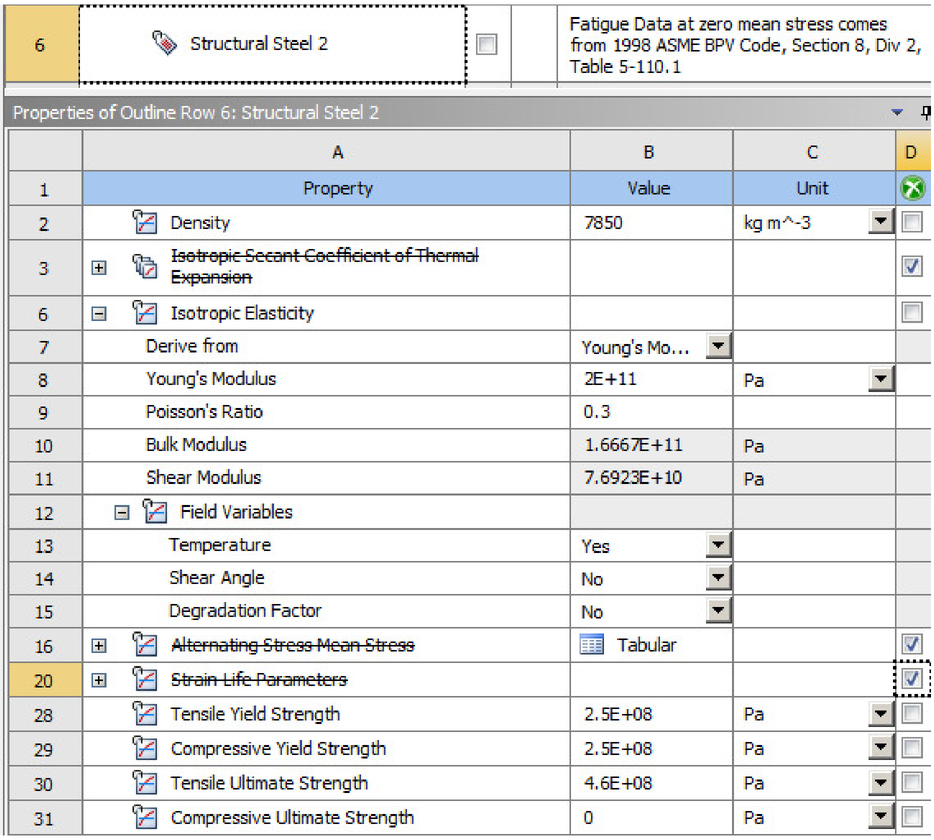
- refine the mesh of the entire assembly, solve, draw the conclusions

CASE 12. ANSYS Workbench Static Structural FEA of a plastic buckle clip

Engineering Data (Materials): Right click the default Structural Steel, Duplicate.

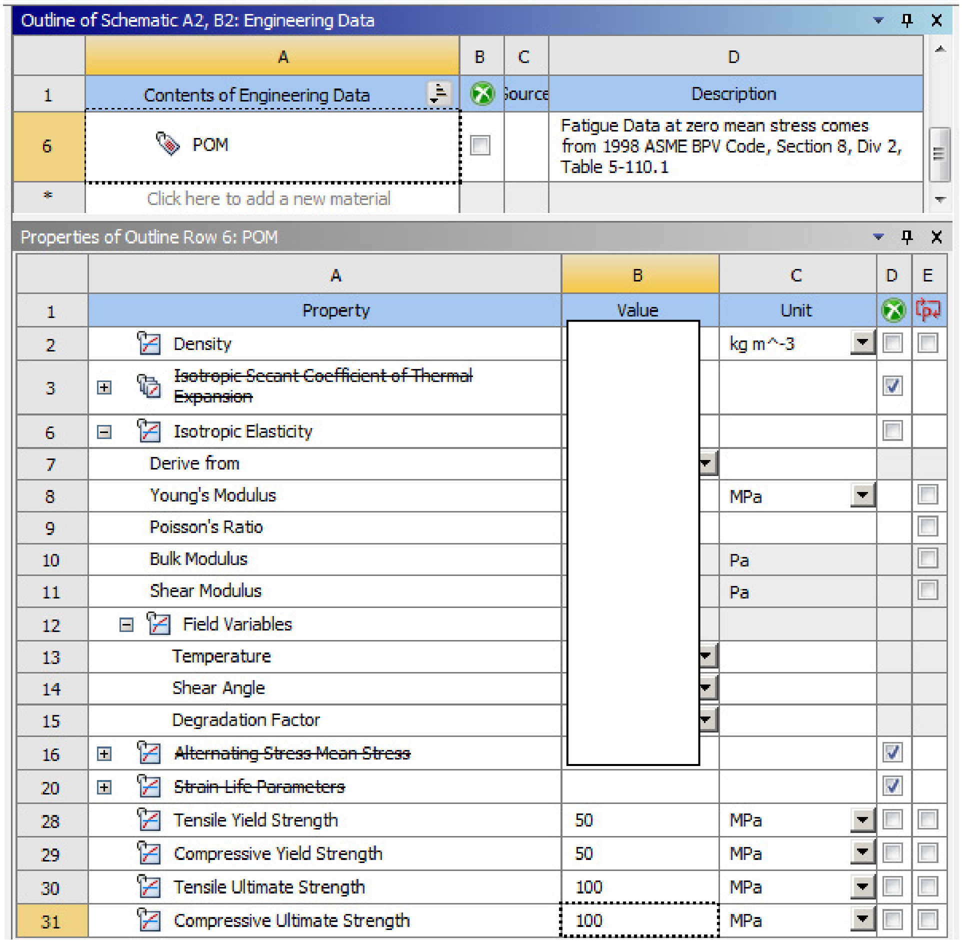


Suppress the fields shown here with strikethrough font by checking them in column D. Name the material as POM.



	A	B	C	D
1	Property	Value	Unit	
2	Density	7850	kg m^-3	☐
3	~~Isotropic Secant Coefficient of Thermal Expansion~~			☑
6	Isotropic Elasticity			☐
7	Derive from	Young's Mo...		
8	Young's Modulus	2E+11	Pa	
9	Poisson's Ratio	0.3		
10	Bulk Modulus	1.6667E+11	Pa	
11	Shear Modulus	7.6923E+10	Pa	
12	Field Variables			
13	Temperature	Yes		
14	Shear Angle	No		
15	Degradation Factor	No		
16	~~Alternating Stress Mean Stress~~	Tabular		☑
20	~~Strain-Life Parameters~~			☑
28	Tensile Yield Strength	2.5E+08	Pa	☐
29	Compressive Yield Strength	2.5E+08	Pa	☐
30	Tensile Ultimate Strength	4.6E+08	Pa	☐
31	Compressive Ultimate Strength	0	Pa	☐

Insert these details.

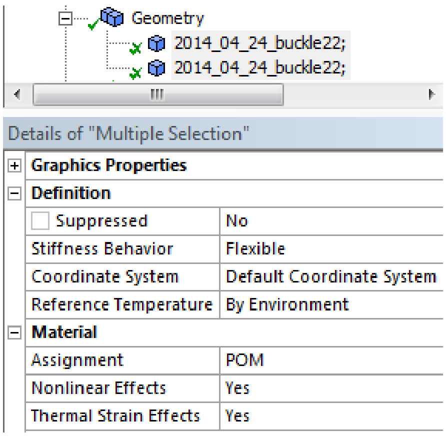

Outline of Schematic A2, B2: Engineering Data

	A	B	C	D
1	Contents of Engineering Data		Source	Description
6	POM			Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5-110.1
*	Click here to add a new material			

Properties of Outline Row 6: POM

	A	B	C	D	E
1	Property	Value	Unit		
2	Density		kg m^-3		
3	~~Isotropic Secant Coefficient of Thermal Expansion~~				
6	Isotropic Elasticity				
7	Derive from				
8	Young's Modulus		MPa		
9	Poisson's Ratio				
10	Bulk Modulus		Pa		
11	Shear Modulus		Pa		
12	Field Variables				
13	Temperature				
14	Shear Angle				
15	Degradation Factor				
16	~~Alternating Stress Mean Stress~~				
20	~~Strain-Life Parameters~~				
28	Tensile Yield Strength	50	MPa		
29	Compressive Yield Strength	50	MPa		
30	Tensile Ultimate Strength	100	MPa		
31	Compressive Ultimate Strength	100	MPa		

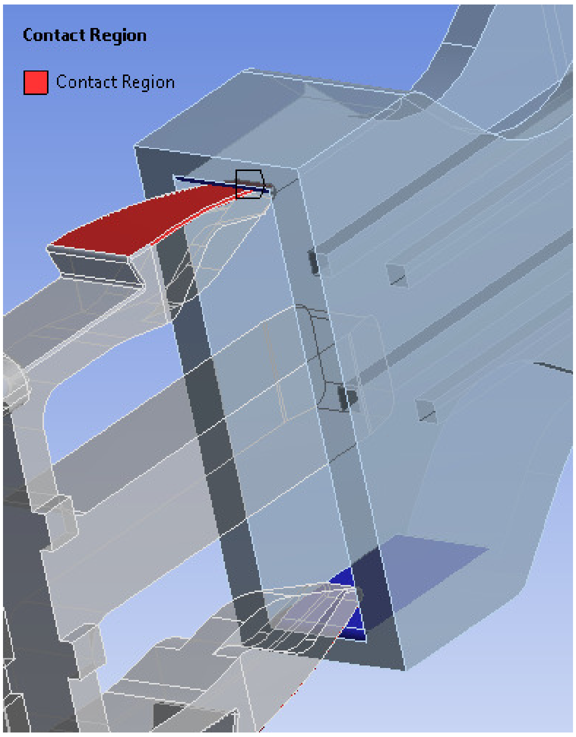
Assign the POM material to both parts.



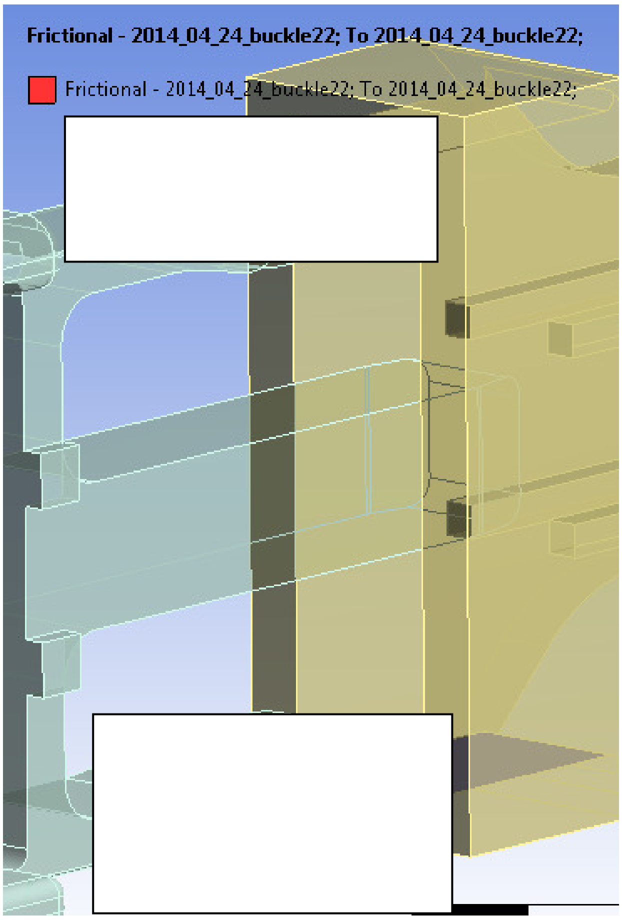
Details of "Multiple Selection"

Graphics Properties	
Definition	
Suppressed	No
Stiffness Behavior	Flexible
Coordinate System	Default Coordinate System
Reference Temperature	By Environment
Material	
Assignment	POM
Nonlinear Effects	Yes
Thermal Strain Effects	Yes

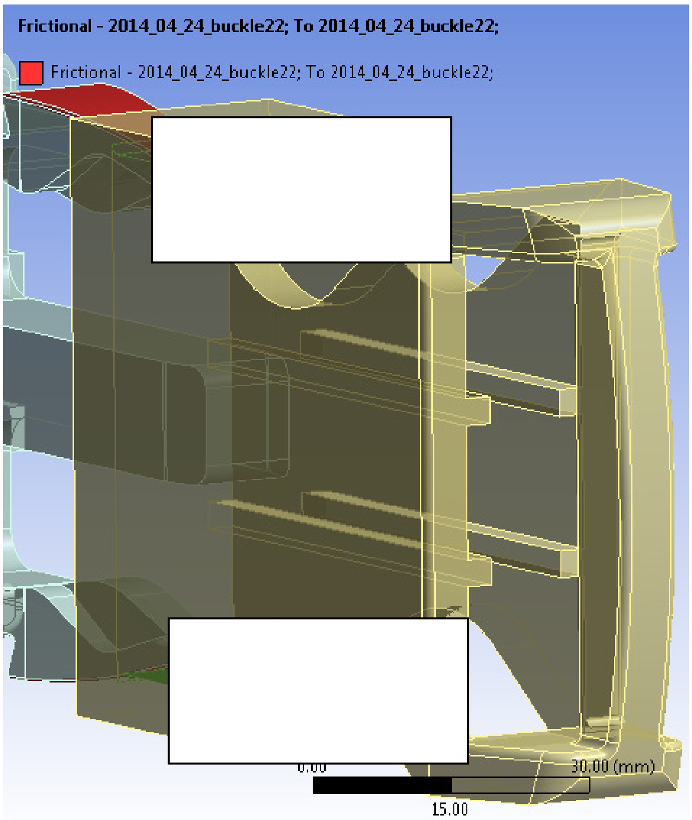
<u>Connections, Contacts:</u> The defualt contact looks like here. We'll select the contacting faces in a proper manner.



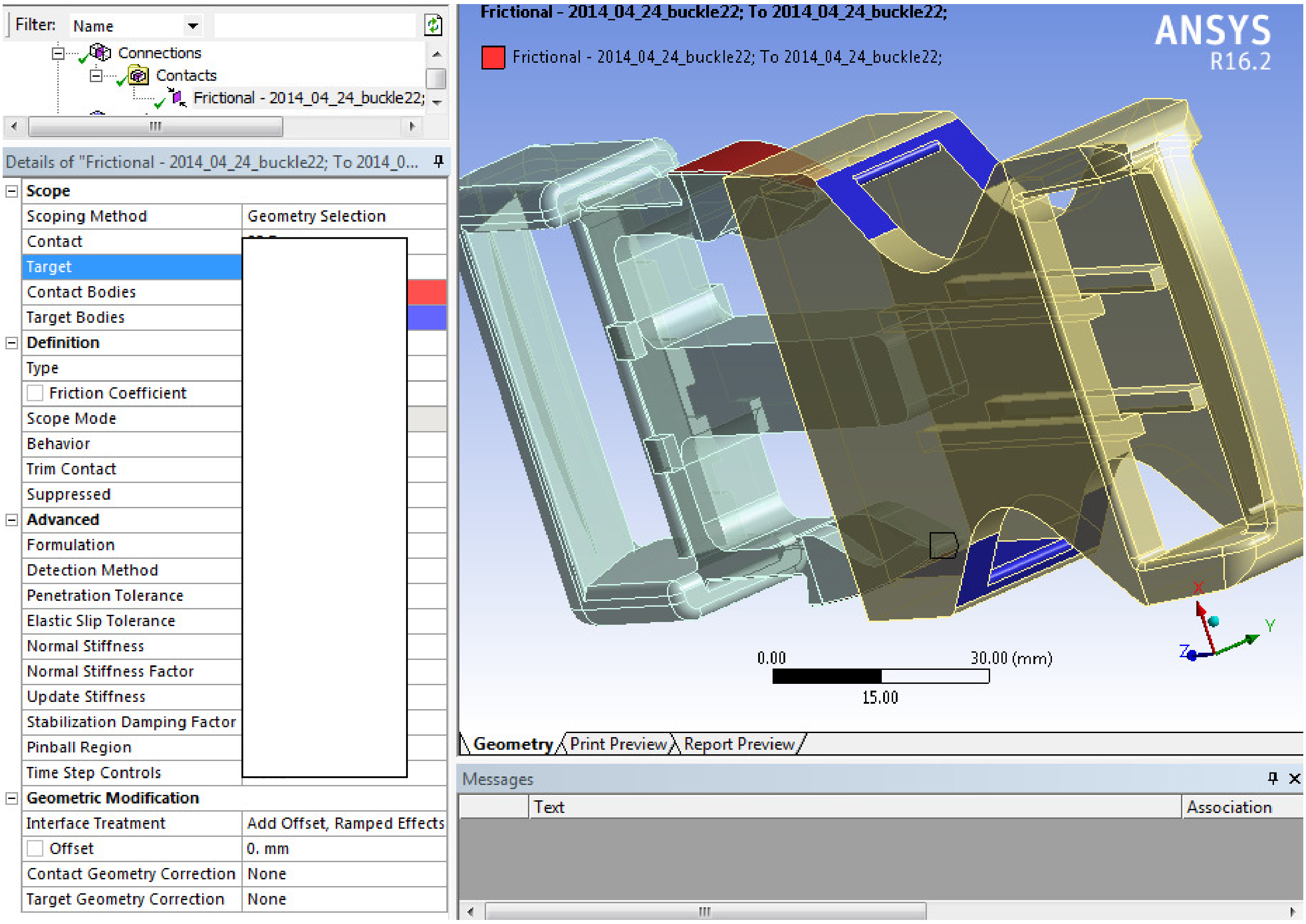
Click Scope, Contact, 6 Faces, then extend the selection to the faces shown green here. Apply.



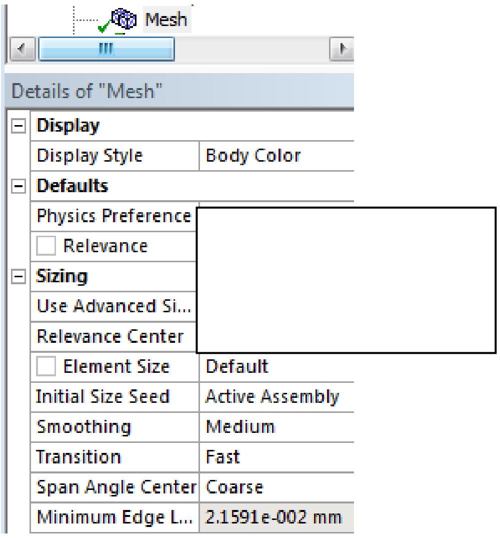
Click Scope, Contact, 4 Faces, then extend the selection to the faces shown green here. Apply.



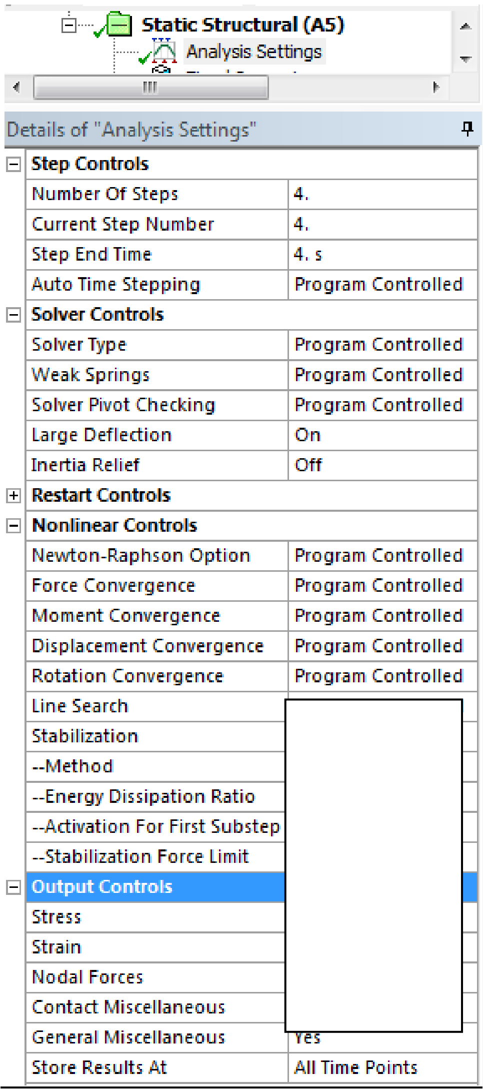
This should be the final result. Assign the details shown here.

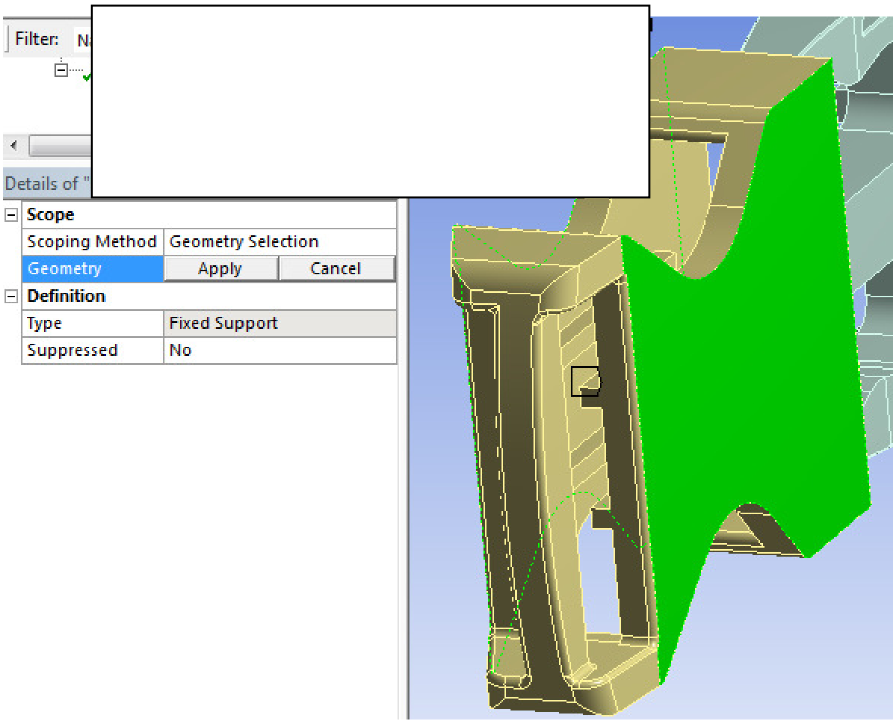


Mesh: Insert these details.

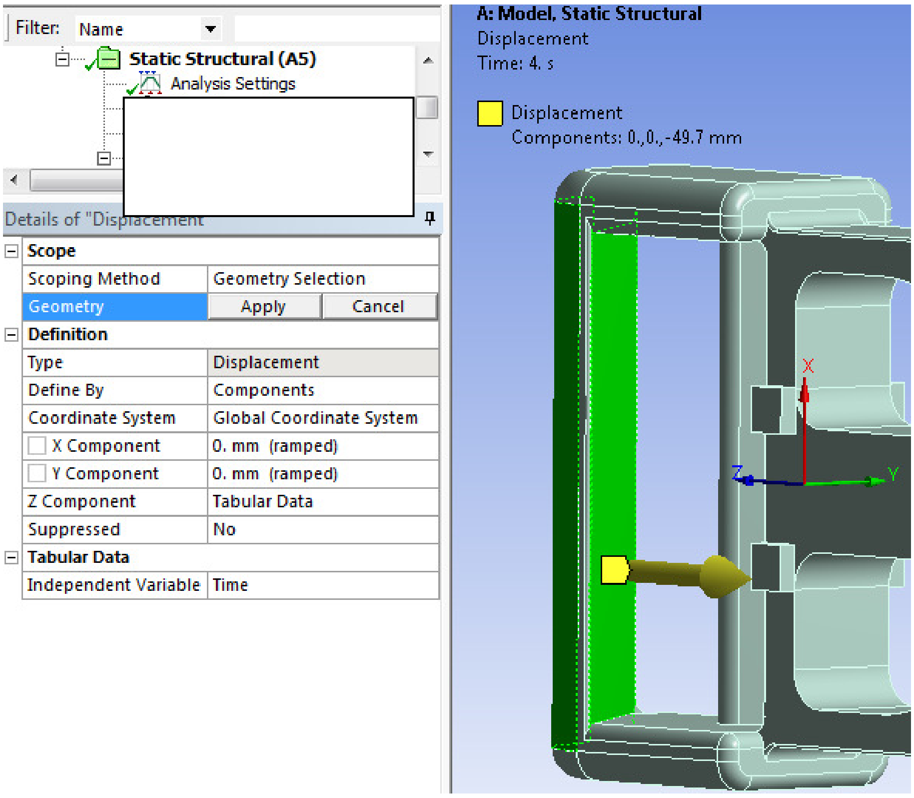


<u>Static Structural, Analysis Settings:</u> Insert these details. Ouput Controls, Yes 5 times.

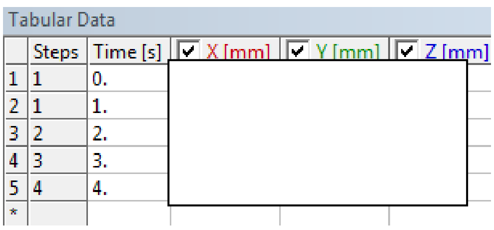




Apply a displacement on axis towards the mating part, on the faces green here.

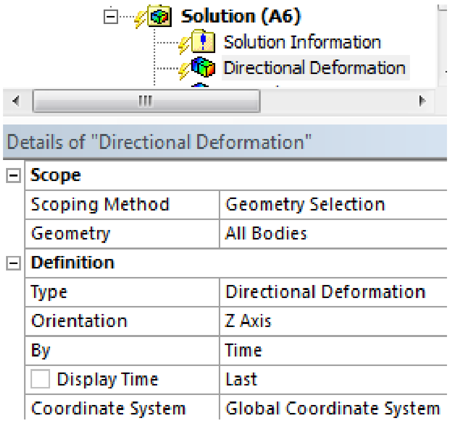


Apply these values.

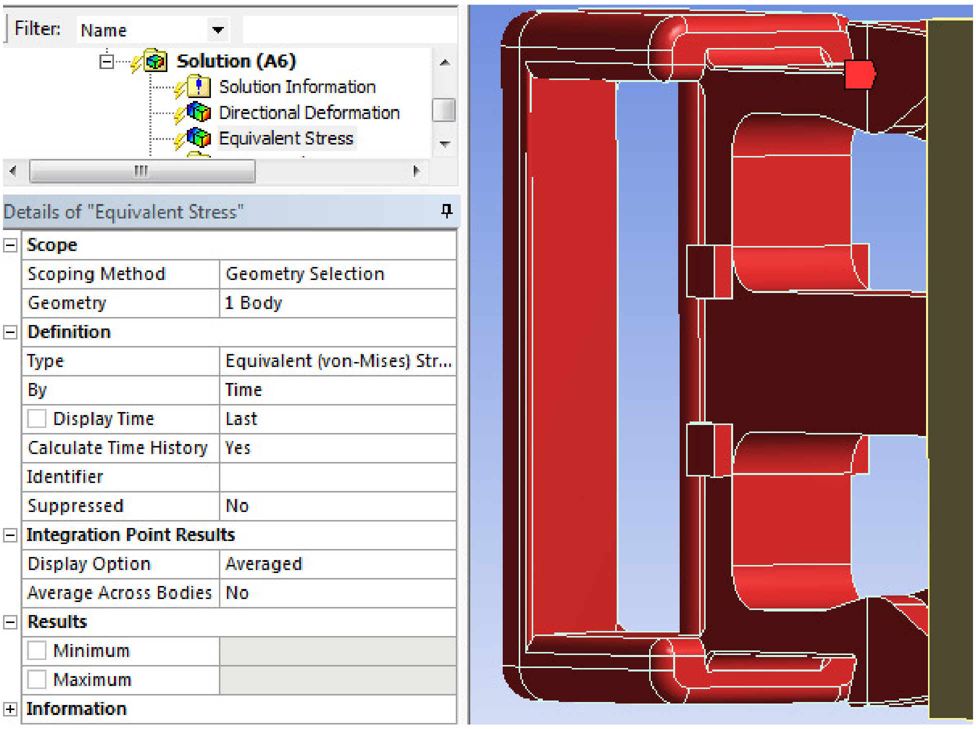

Tabular Data

	Steps	Time [s]	X [mm]	Y [mm]	Z [mm]
1	1	0.			
2	1	1.			
3	2	2.			
4	3	3.			
5	4	4.			
*					

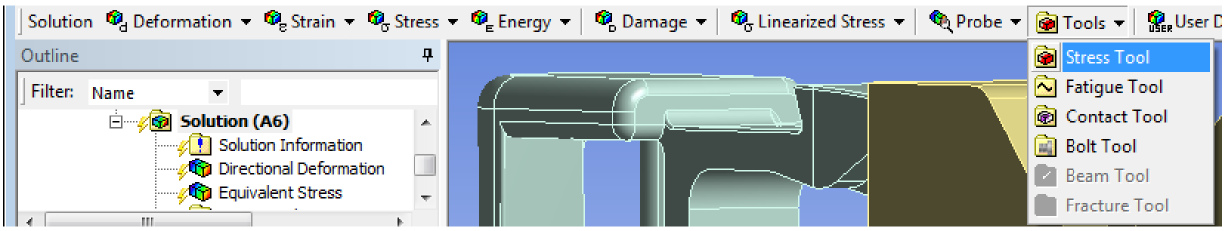
<u>Solution:</u> Insert a Directional Deformation on axis.



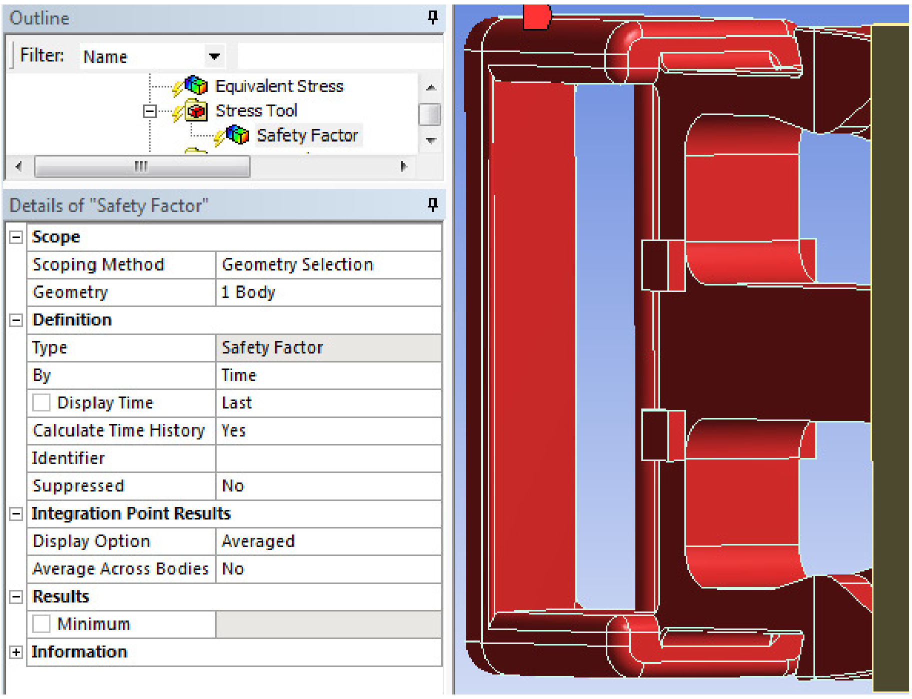
Create an Equivalent Stress only on the body shown red here.



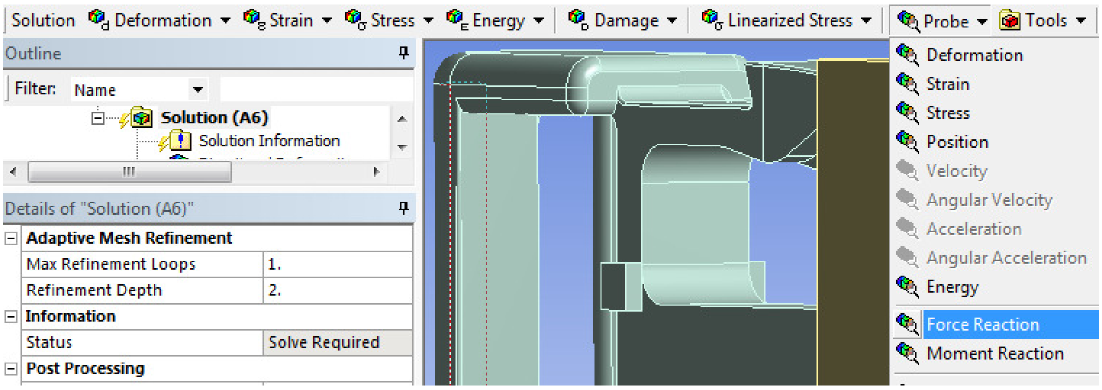
From the Solution toolbar, go to Tools and click Stress Tool.



Scope only the red part, as seen here.



Click Probe, Force Reaction.



Choose these details.

Create another Rorce Reaction, with these details. Solve.

Further homework:

- decrease the Friction Coefficient value to half, solve, draw the conclusions

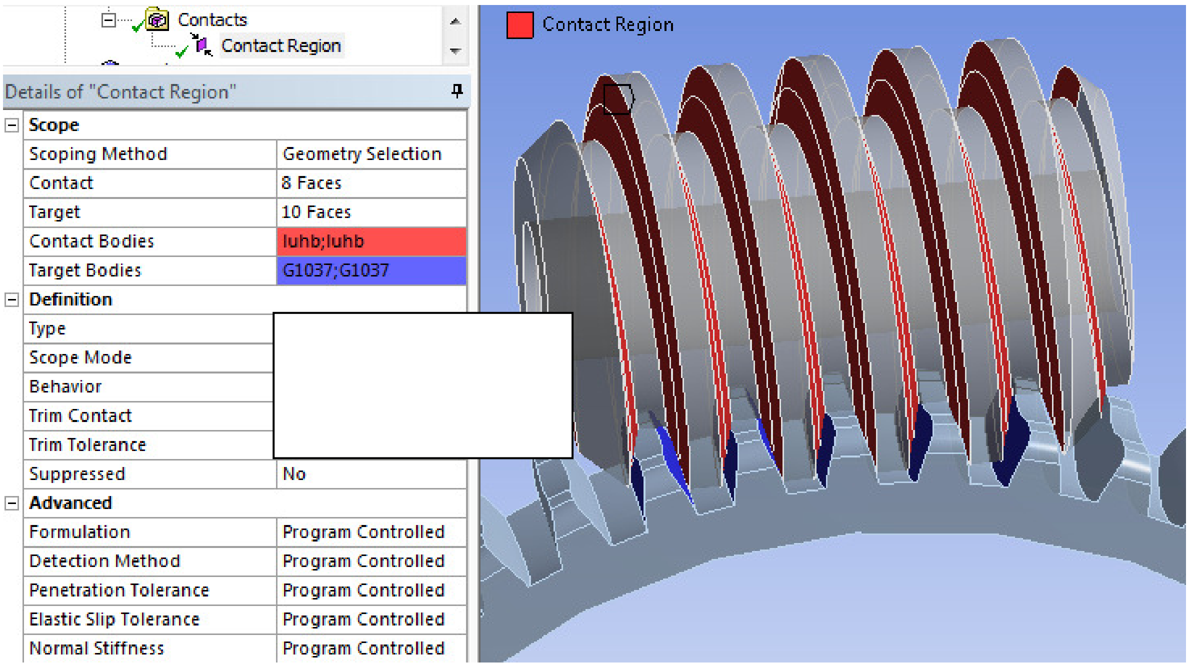
- double the Young's Modulus value, solve, draw the conclusions

- turn Stabilization to Off, does the FEA finish? Solve, draw the conclusions

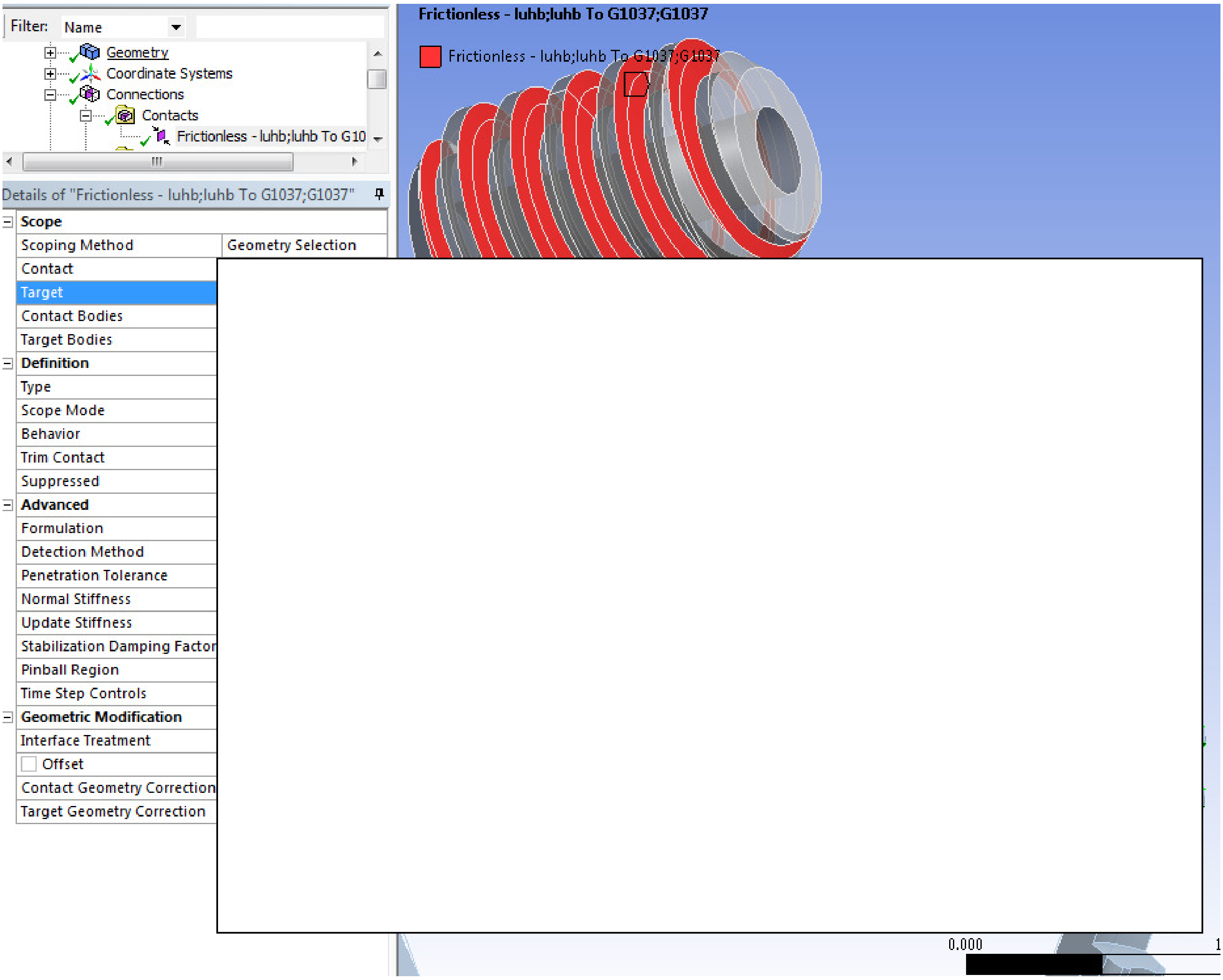
CASE 13. FIRST IN THE WORLD!!! ANSYS Workbench Transient Structural FEA of a worm drive (screw and wheel)

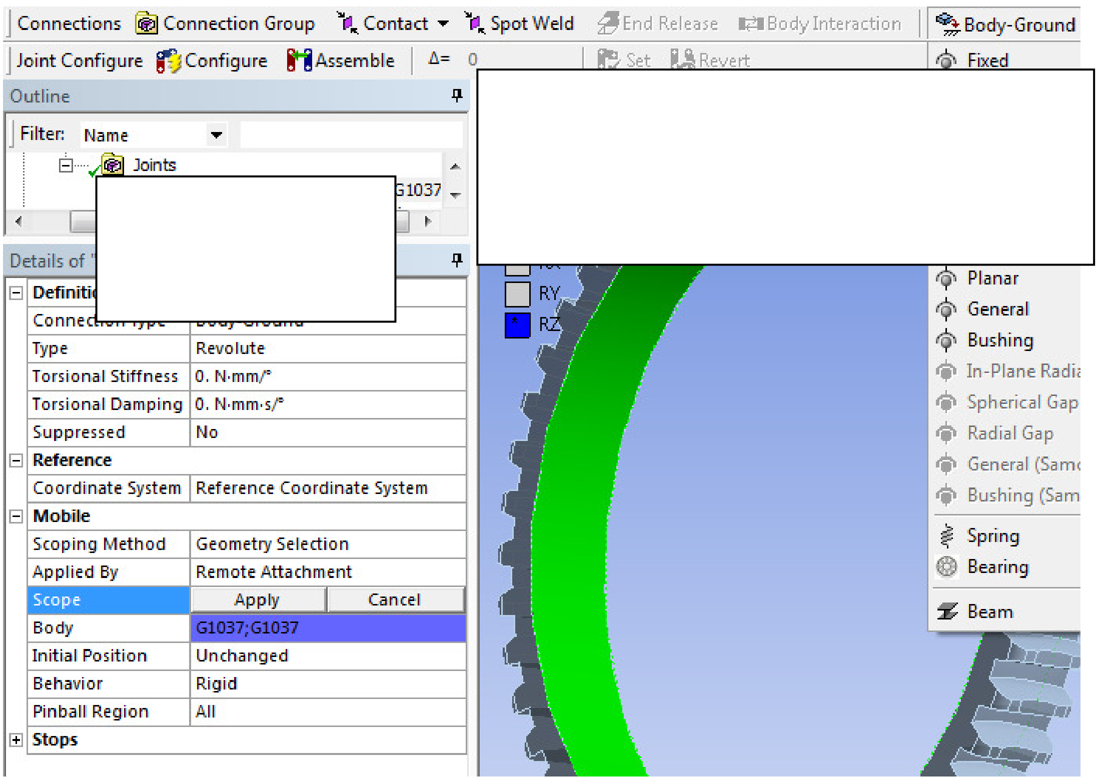
Geometry: 2015_jul_24_worm_gear_FEA1.x_t

Connections, Contacts: Let us expand the selection for Scope, Target. Click on 10 Faces.

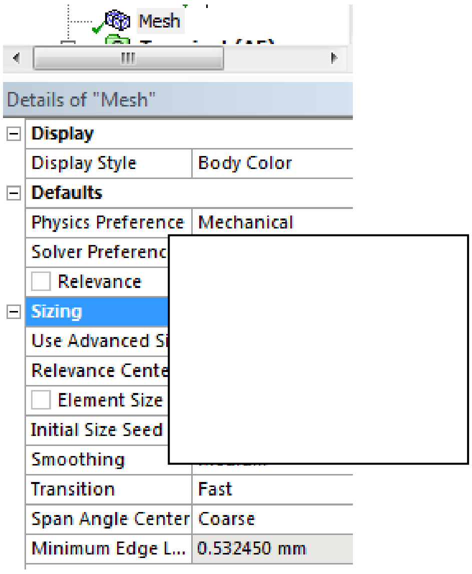


With Ctrl pressed, click to add more faces, for a portion of around quarter of the circle. Type = Frictionless.

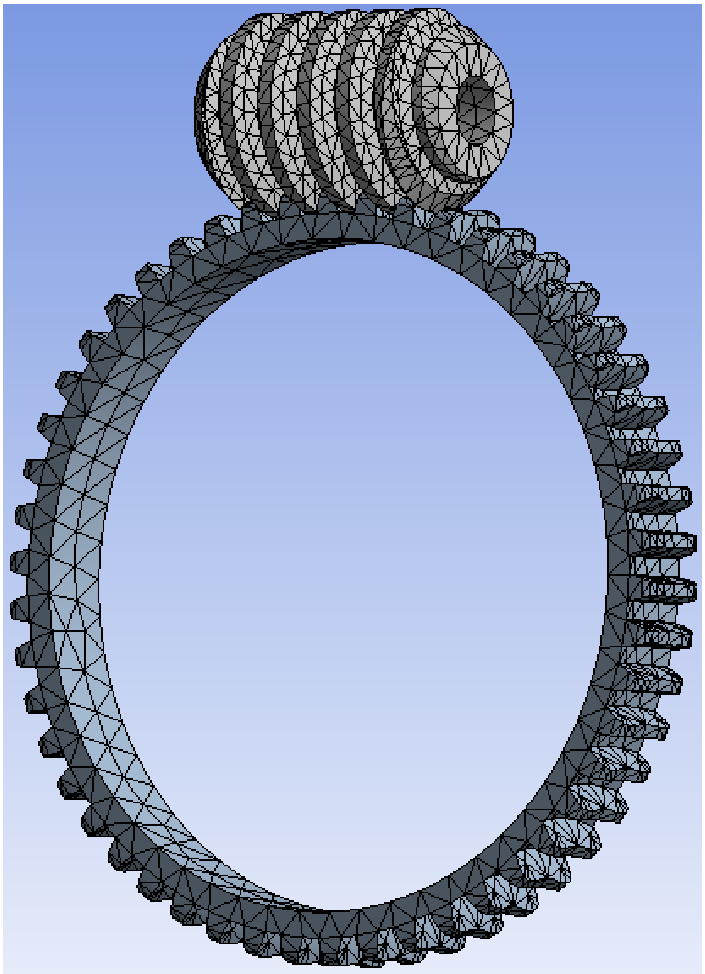

Filter: Name
Geometry
Coordinate Systems
Connections
Contacts
Frictionless - luhb;luhb To G10
Details of "Frictionless - luhb;luhb To G1037;G1037"
Scope
Scoping Method
Geometry Selection
Contact
Target
Contact Bodies
Target Bodies
Definition
Type
Scope Mode
Behavior
Trim Contact
Suppressed
Advanced
Formulation
Detection Method
Penetration Tolerance
Normal Stiffness
Update Stiffness
Stabilization Damping Factor
Pinball Region
Time Step Controls
Geometric Modification
Interface Treatment
Offset
Contact Geometry Correction
Target Geometry Correction
Frictionless - luhb;luhb To G1037;G1037
Frictionless - luhb;luhb To G1037;G1037
0.000


Connections
Connection Group
Contact
Spot Weld
End Release
Body Interaction
Body-Ground
Joint Configure
Configure
Assemble
Set
Revert
Fixed
Outline
Filter: Name
Joints
Details of
Definition
Type
Revolute
Torsional Stiffness
0. N·mm/°
Torsional Damping
0. N·mm·s/°
Suppressed
No
Reference
Coordinate System
Reference Coordinate System
Mobile
Scoping Method
Geometry Selection
Applied By
Remote Attachment
Scope
Apply
Cancel
Body
G1037;G1037
Initial Position
Unchanged
Behavior
Rigid
Pinball Region
All
Stops
RY
RZ
Planar
General
Bushing
Spherical Gap
Radial Gap
Spring
Bearing
Beam

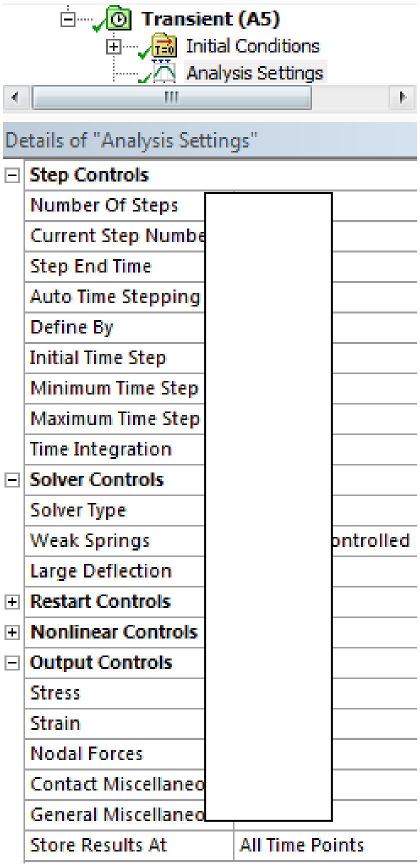
Mesh: Insert these details.



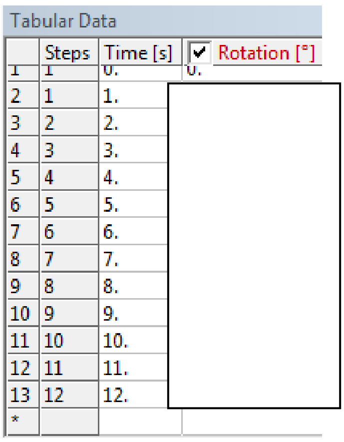
The mesh should look like here.



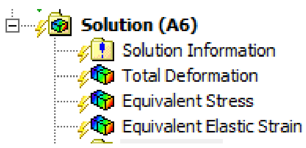
Transient, Analysis Settings: Assign these details for all steps, the 1st one is shown here. For the other steps, make



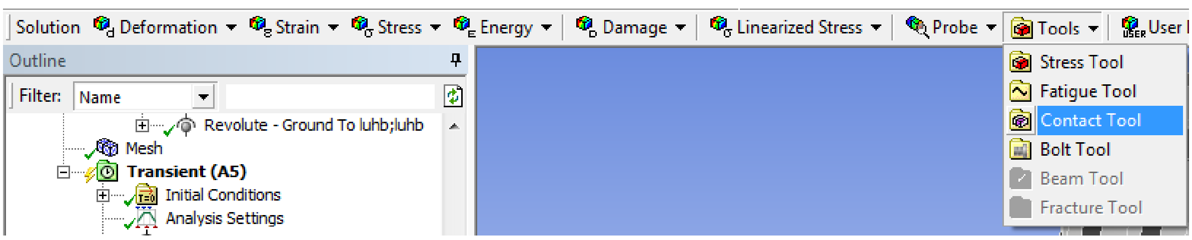
Enter these values.



<u>Solution:</u> Insert these items as default.

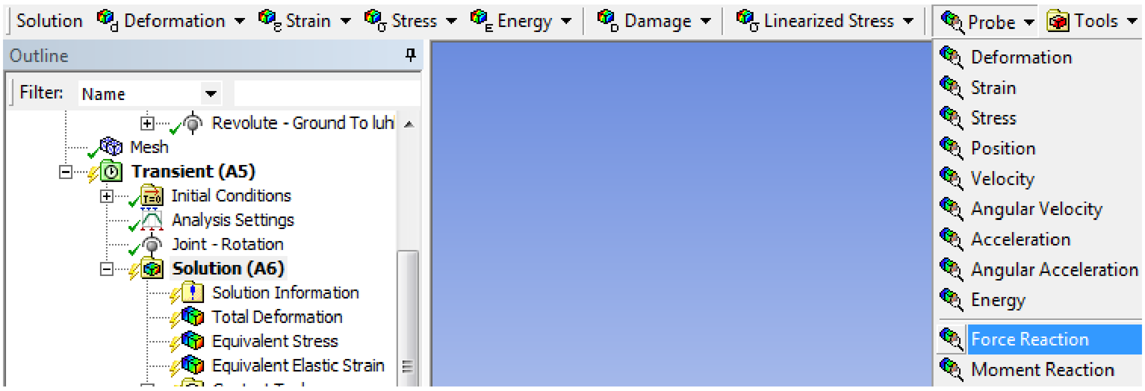


Apply a contact Tool.



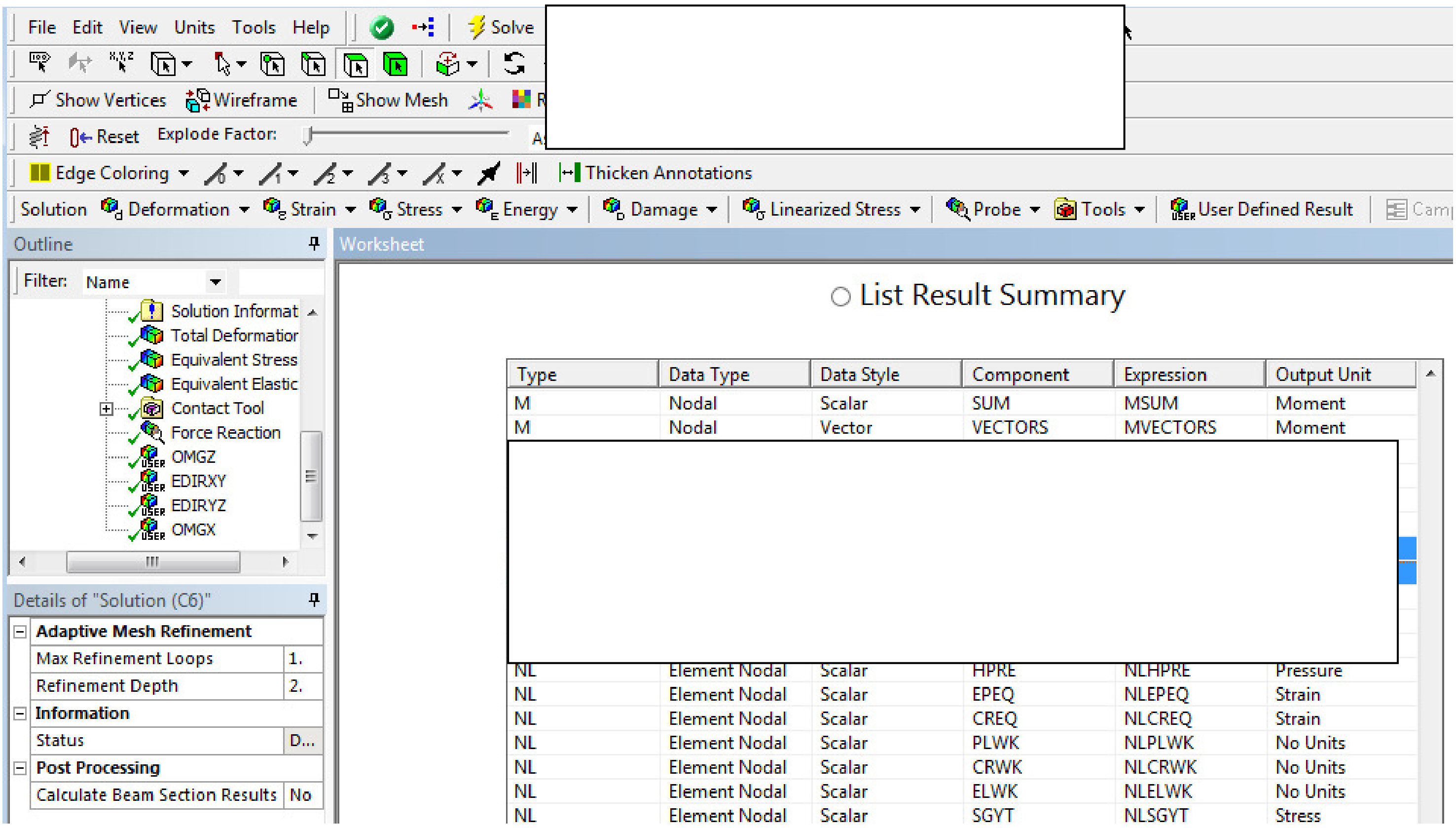
Right click on it and assign a Pressure item.

Insert a Force Reaction.



Apply these details on it. Solve.

Right click any Solution item, Evaluate All Results.



Further homework:

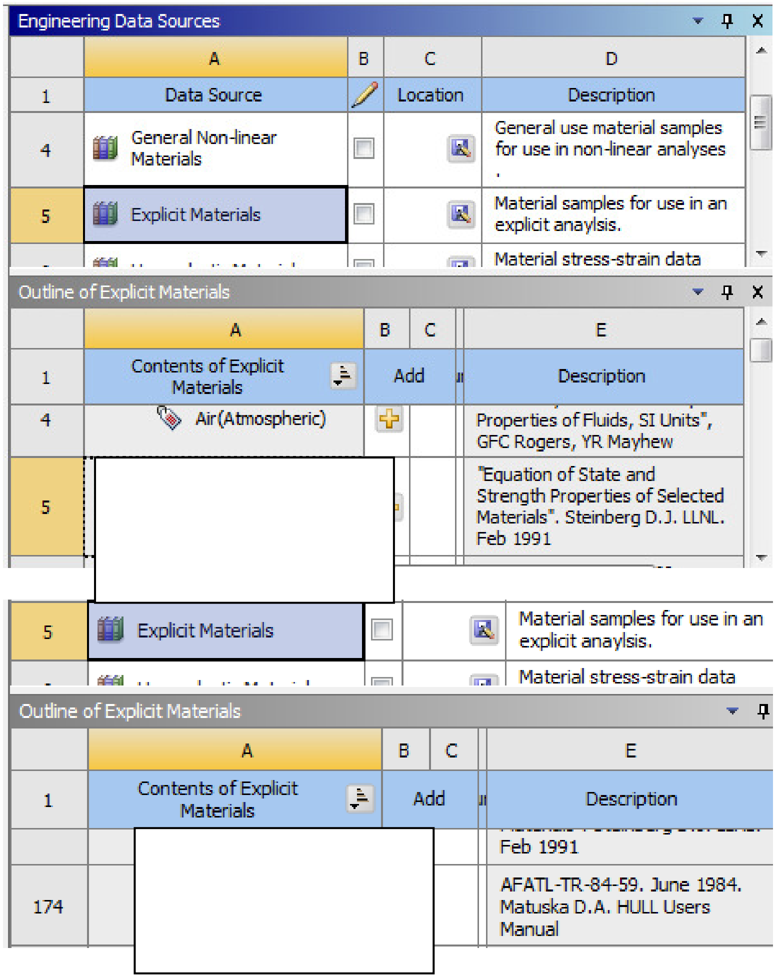
- change Frictionless to Frictional with Friction Coefficient = 0.1, solve, draw the conclusions

- change mesh Relevance to 35, solve, draw the conclusions

- make the Behavior of joints as Flexible, solve, draw the conclusions

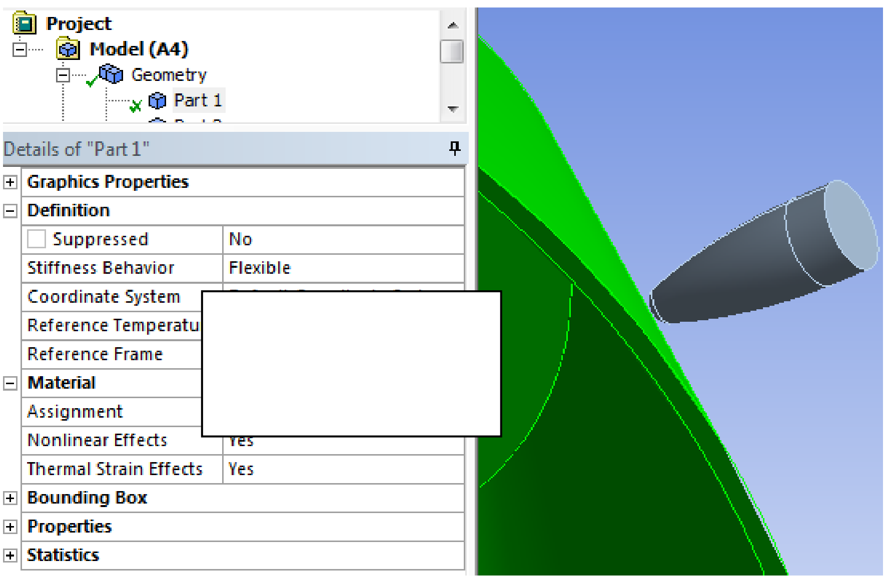
CASE 16. ANSYS Workbench Explicit Dynamics FEA of bullet penetrating through a bucket (optimized)

Engineering Data (Materials): Go to Explicit Materials and Add these materials by pressing the yellow plus in column B.

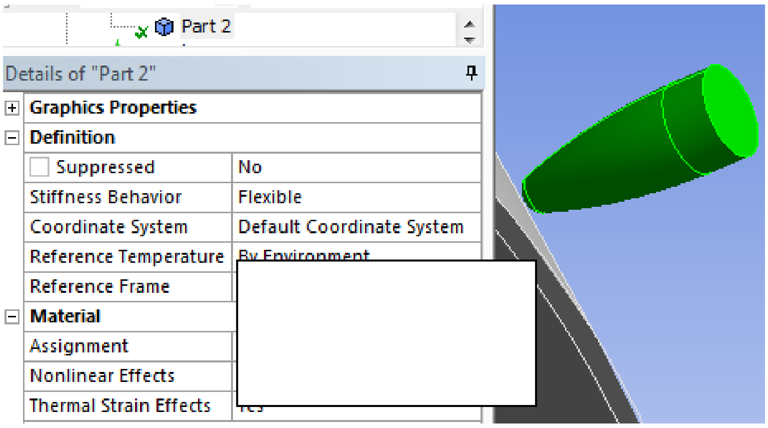


Geometry: bullet_bucket1.igs

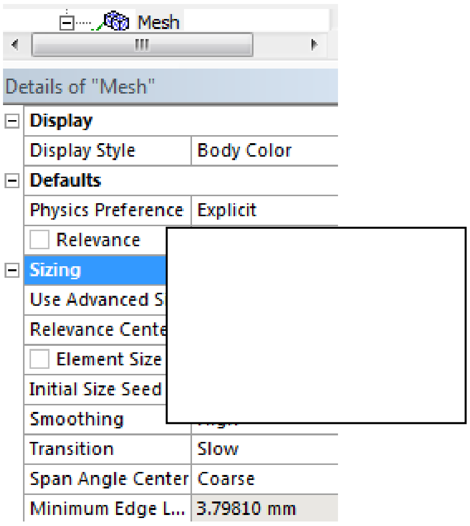
Assign material to the green bucket seen here.



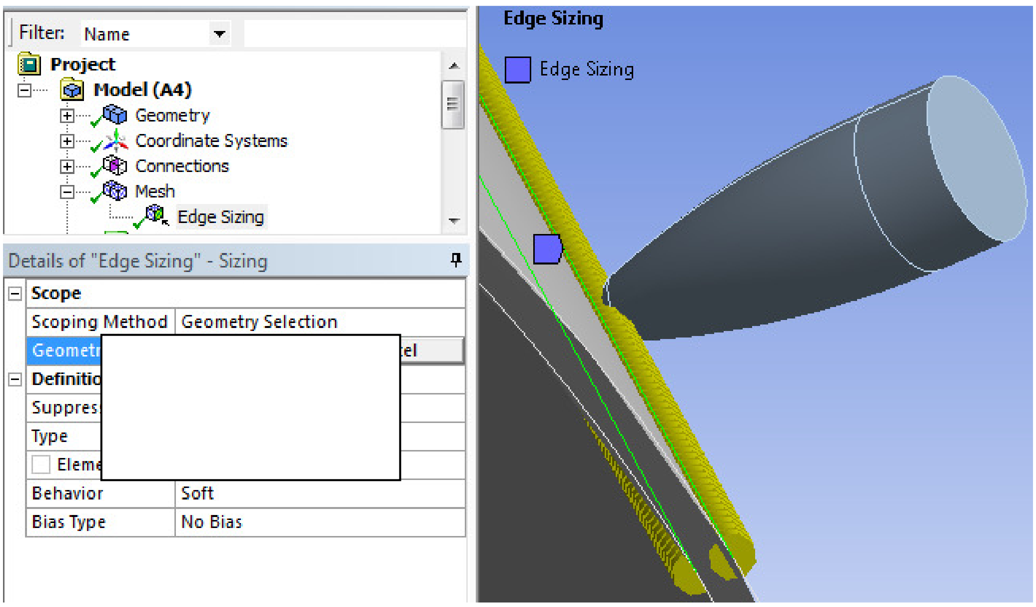
Apply material to the bullet, green here.



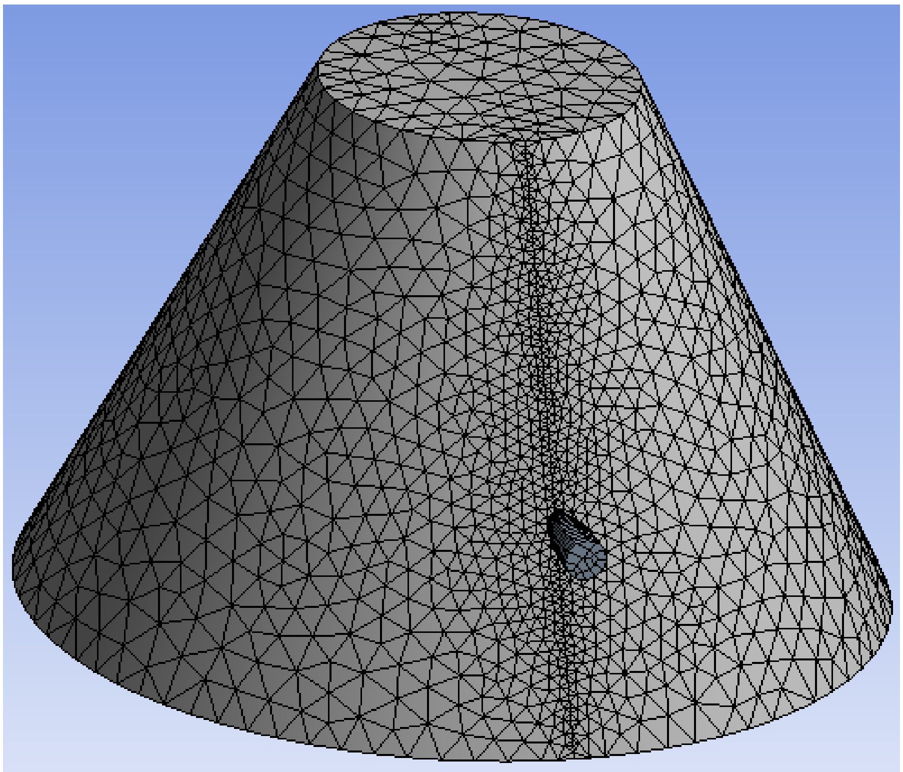
Mesh: Assign these details.



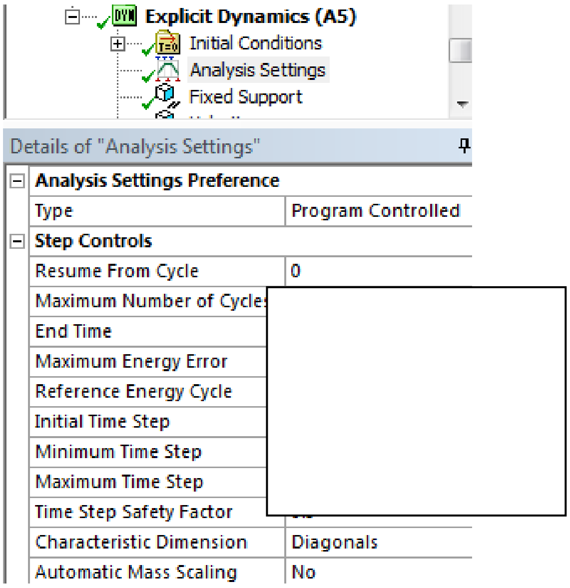
Apply a mesh sizing of on these 2 edges and the other ones from the opposite side of the bucket.



The mesh should look like here.

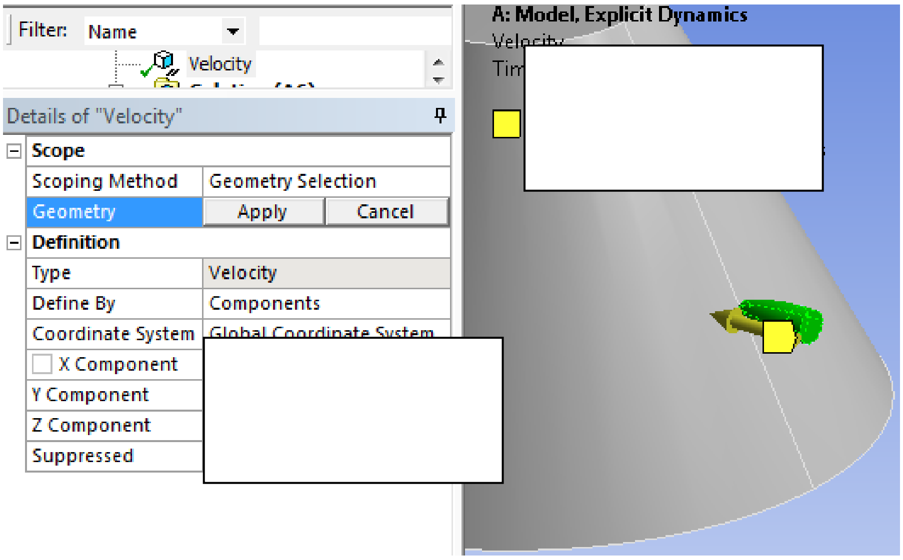


Explicit Dynamics, Analysis Settings: Insert these details with End Time =

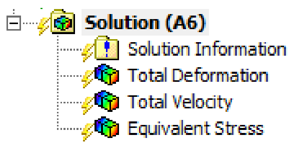


We'll apply the boundary conditions from the Environment toolbar.

Insert a Velocity on the bullet, shown green here, oriented towards the bucket



Solution: Create these items as default. Solve.



Further homework:

- change the bucket material to Aluminum Alloy NL, then Copper Alloy NL, solve, draw the conclusions

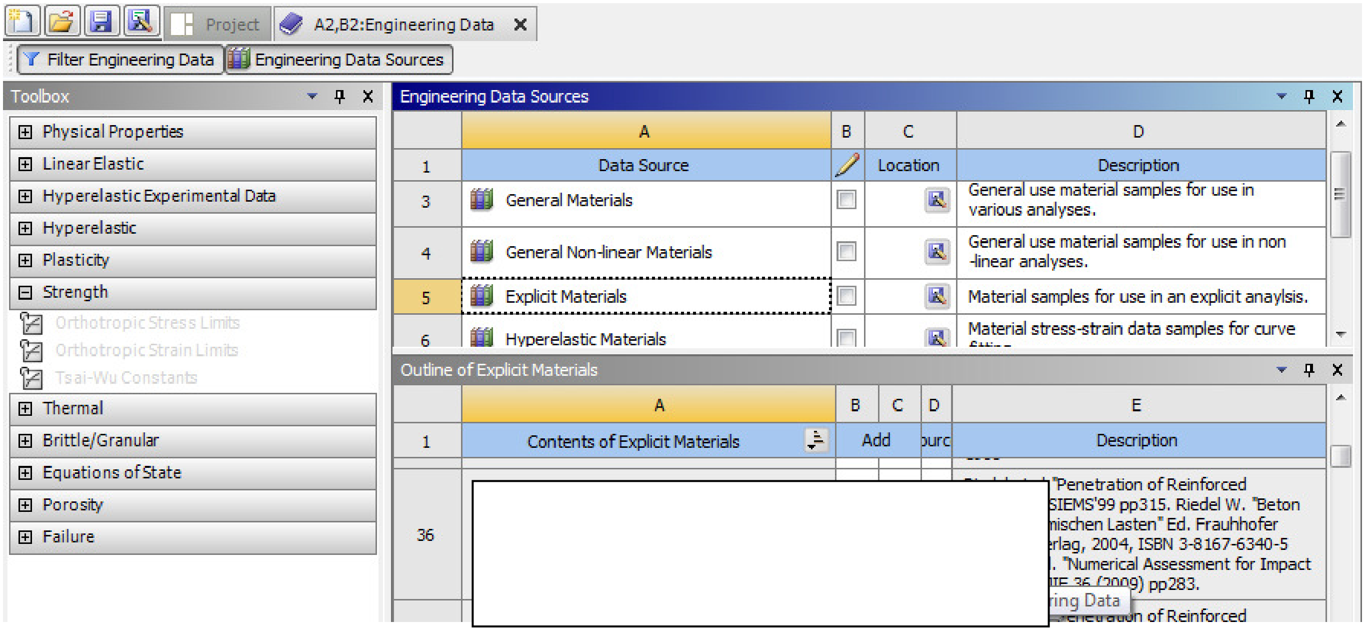
- change mesh size to 6 mm instead of 3 mm, solve, draw the conclusions

- change mesh Relevance to 77, solve, draw the conclusions

- turn Stabilization = Off, solve, draw the conclusions

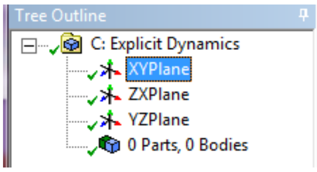
CASE 19. ANSYS Workbench Explicit6 Dynamics of testing of a reinforced concrete structure

Engineering Data (Materials): From Explicit Materials library, add to our scenario by pressing the yellow + sign int he column B.

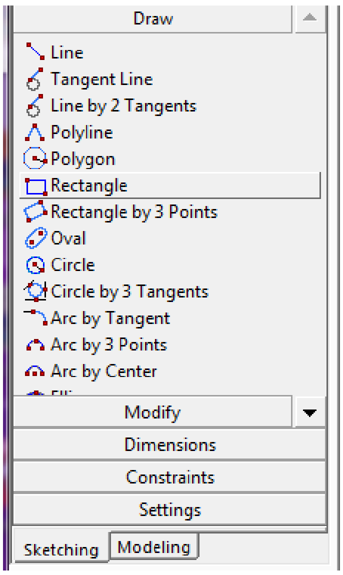


Geometry: Double click to create the geometry in Design Modeler.

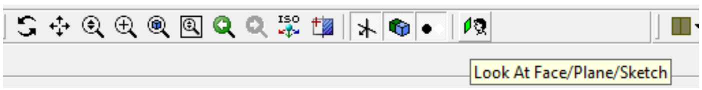
Click XYPlane from the tree.



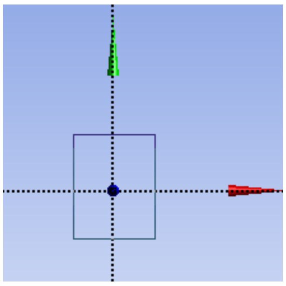
Go to Sketching, click Rectangle.



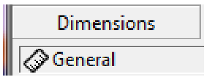
Click the Look At Face/ Plane/ Sketch button.



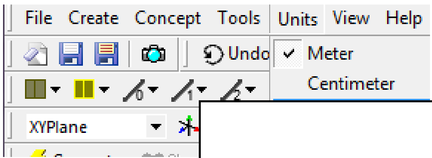
Drag a rectangle like this.



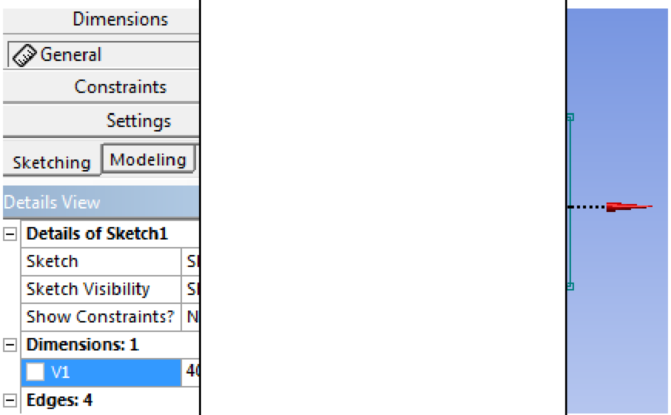
Go to Dimensions, click General.



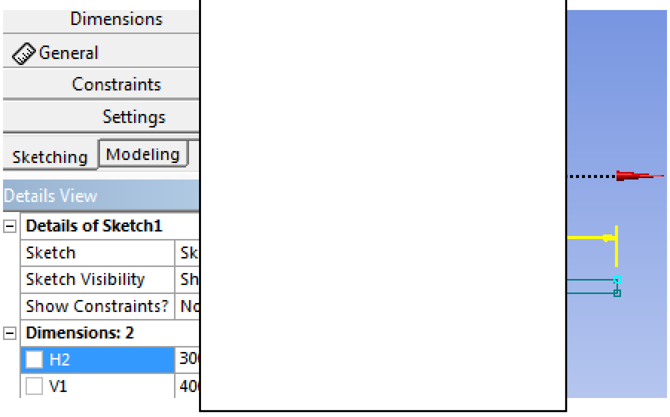
Modify Units to Millimeter.



Dimension the ve nter.

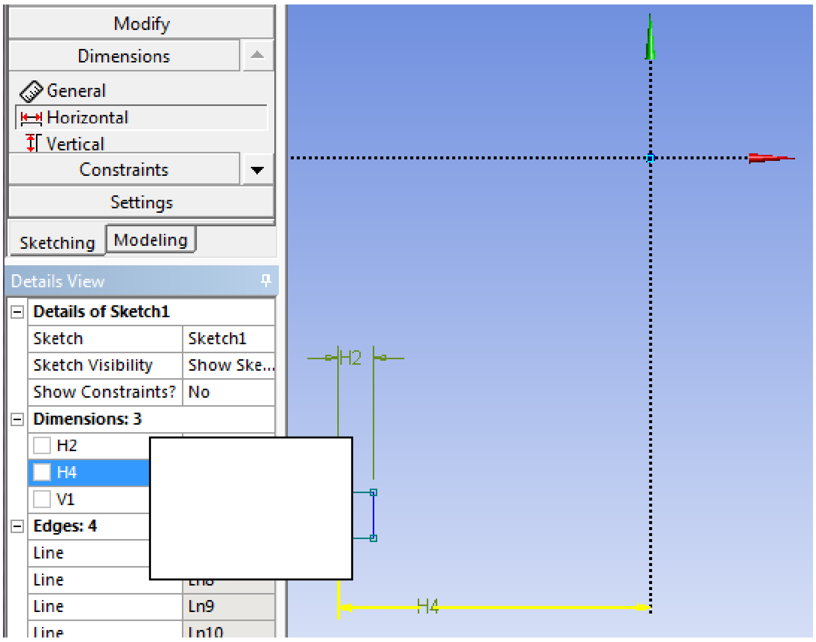


Dimension the to , Enter.

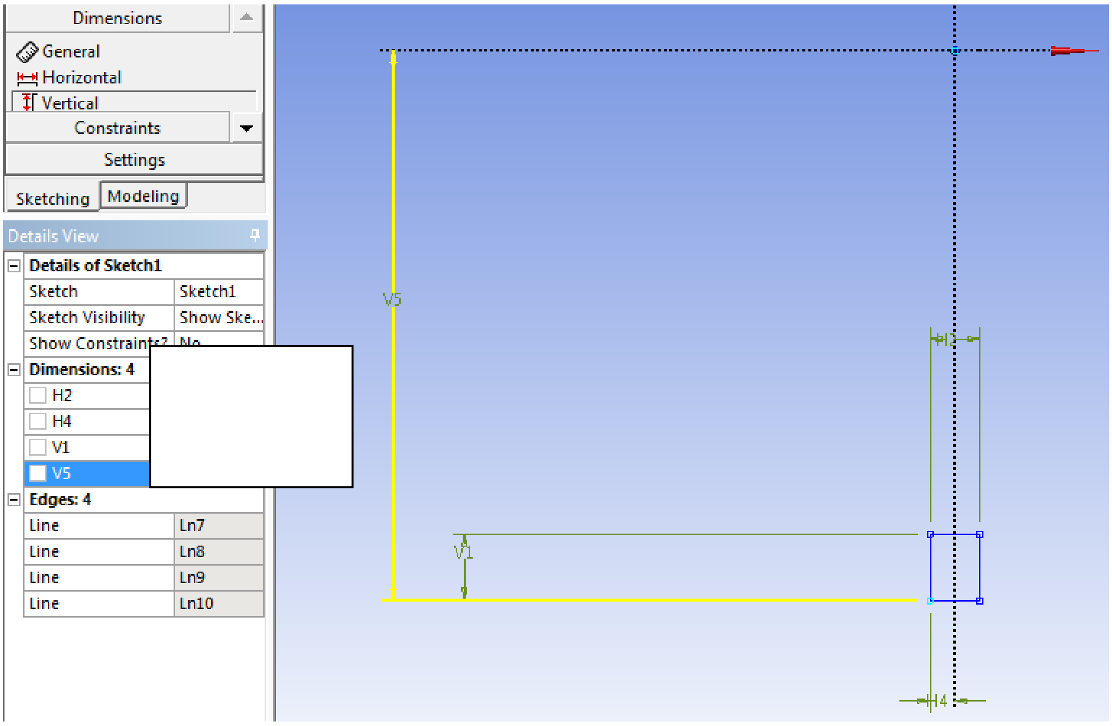


Press F7 or click the Zoom To Fit button. This is the result. Let's center the sketch.

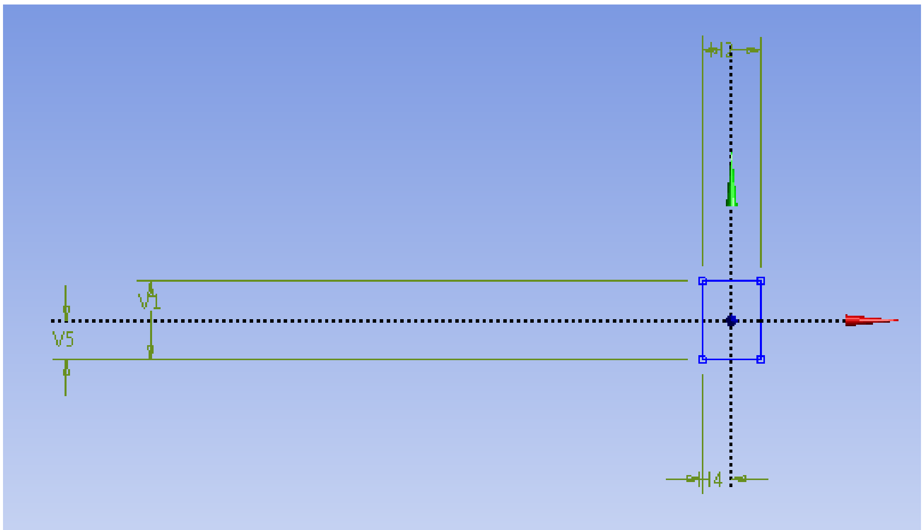
Dimensions, Horizontal, select the left line, then the green Y axis, , Enter. (The numbering can differ because we tested other dimensions, for you it can continue with H3 and so on).



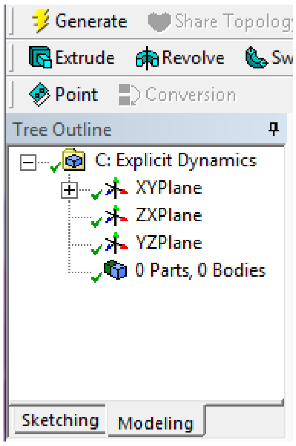
This is the result.



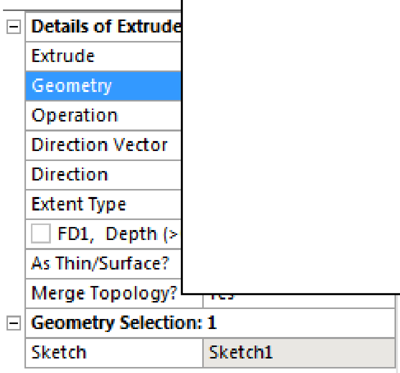
This is the result. (The dimensions can be randomly put or aligned, don't be be bothered by this).



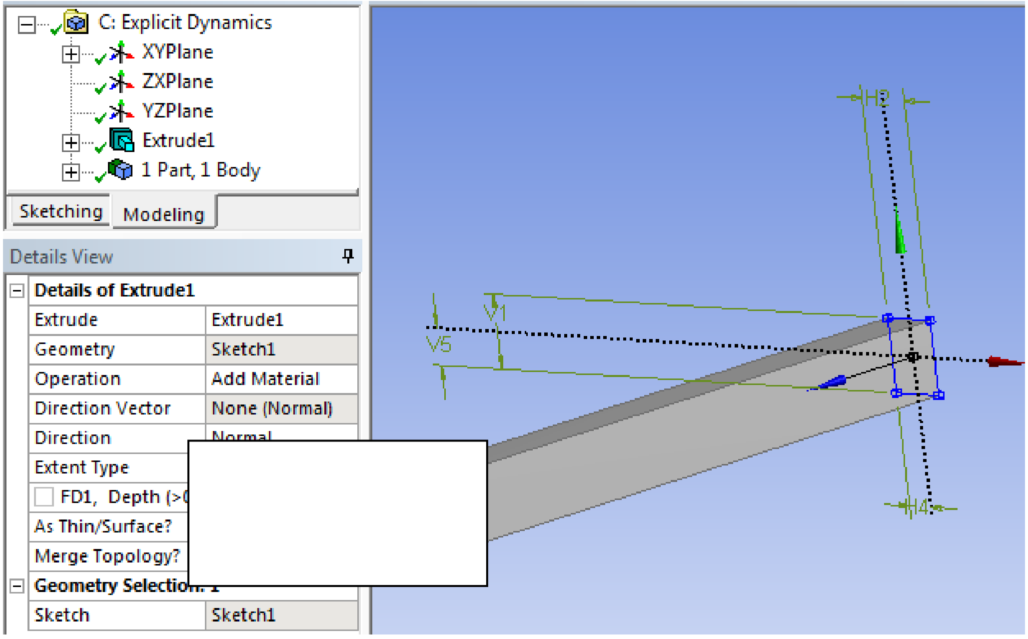
Go to the Modeling tab, click Generate, then Extrude.



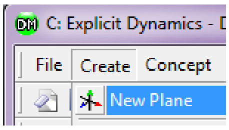
Geometry, App



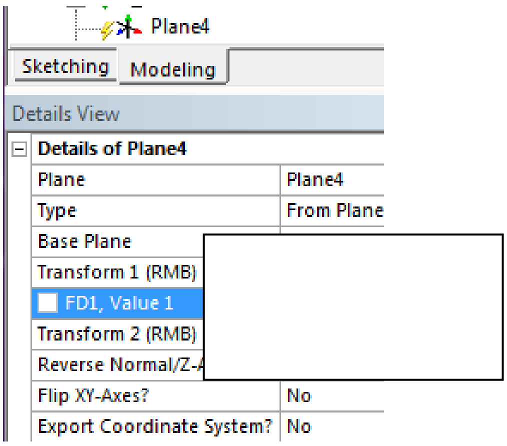
Righ click Extrude1, Generate. Rotate the geometry to see the result.



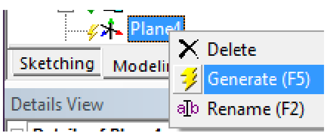
Create, New Plane.



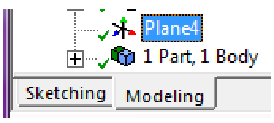
Apply these details.



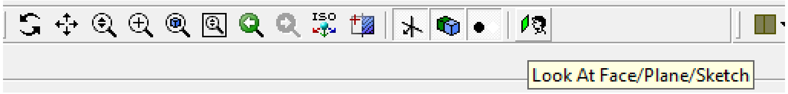
Right click the plane, Generate.



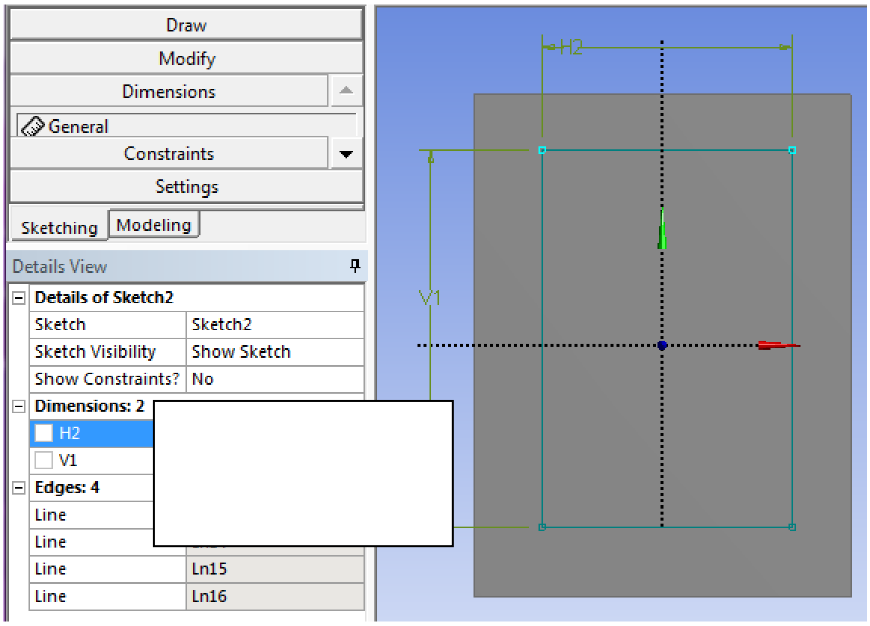
Select Plane4 (your numbering can be different), go to the Skecthing tab.



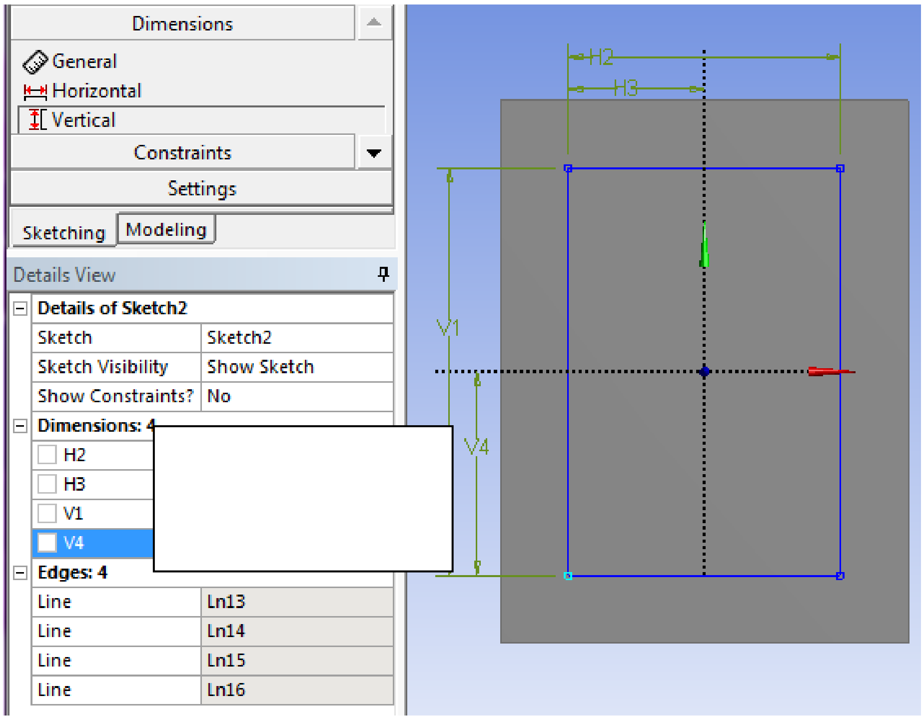
Click the Look At Face/ Plane/ Sketch button.



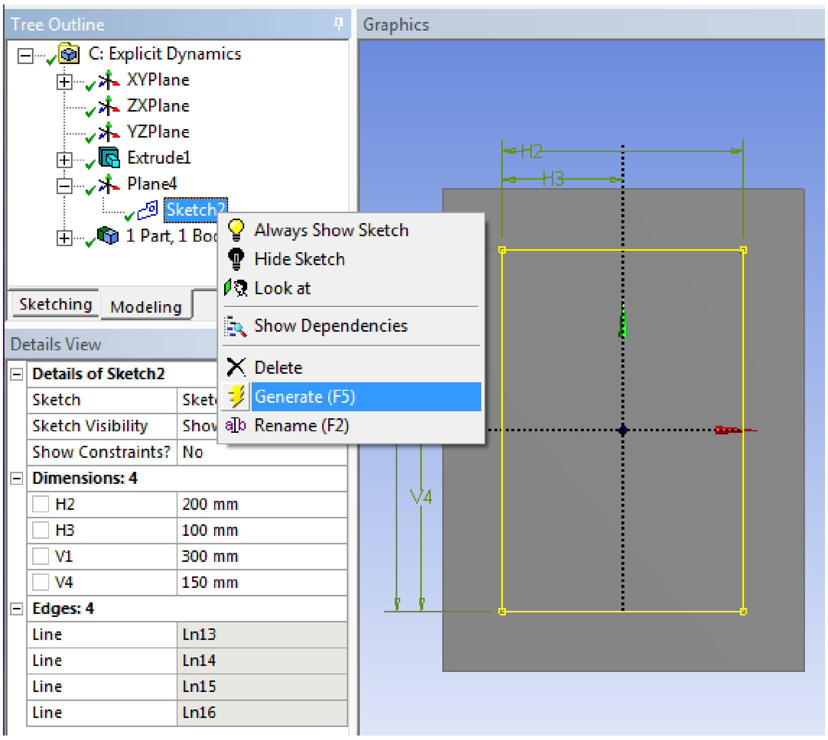
Dimensions, General, insert them on the left and top lines, as seen here.

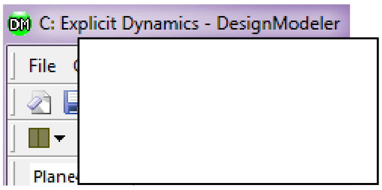


Dimensions, Horizontal =, as seen here.

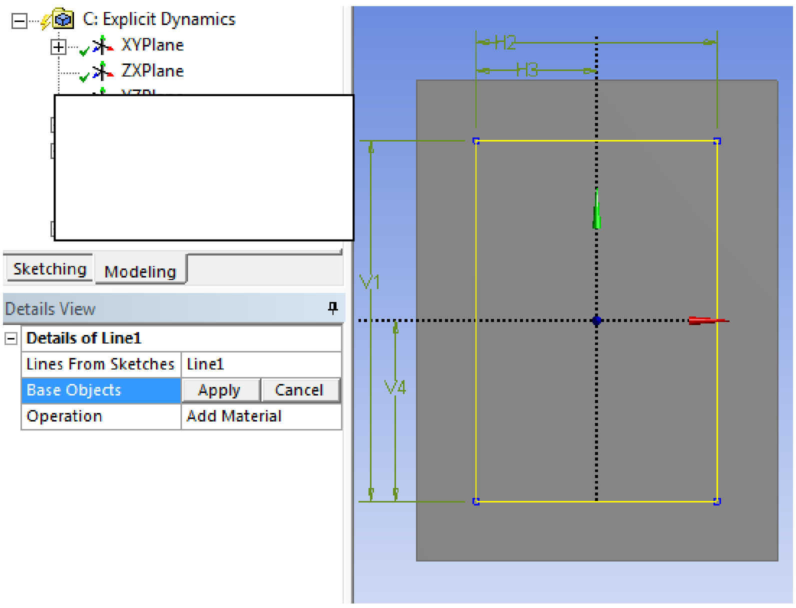


Go to the Modeling tab, right click Sketch2, Generate.



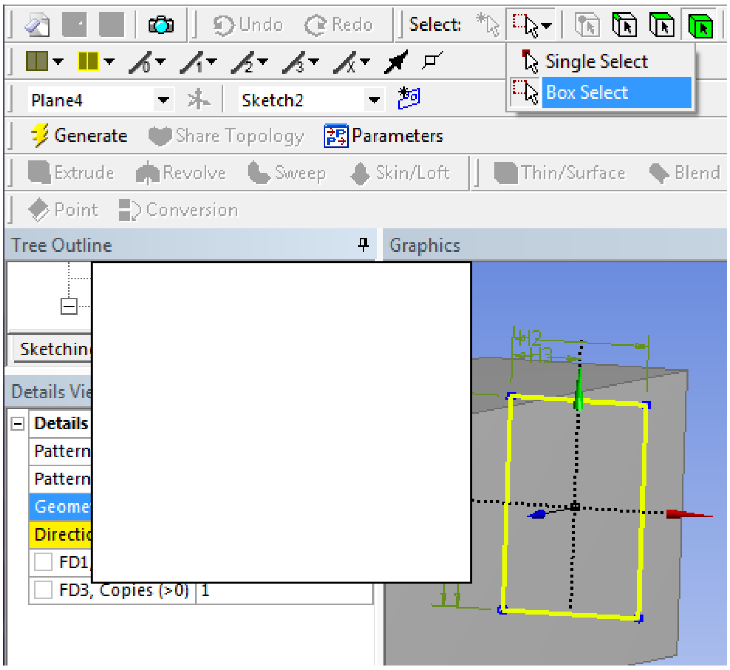


Select these 4 yellow lines, Apply.

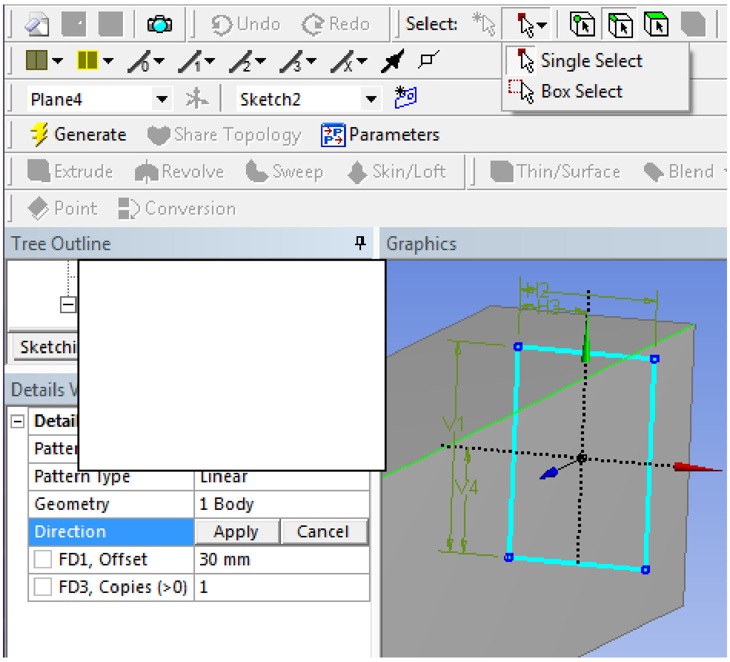


Right click Line1, Generate.

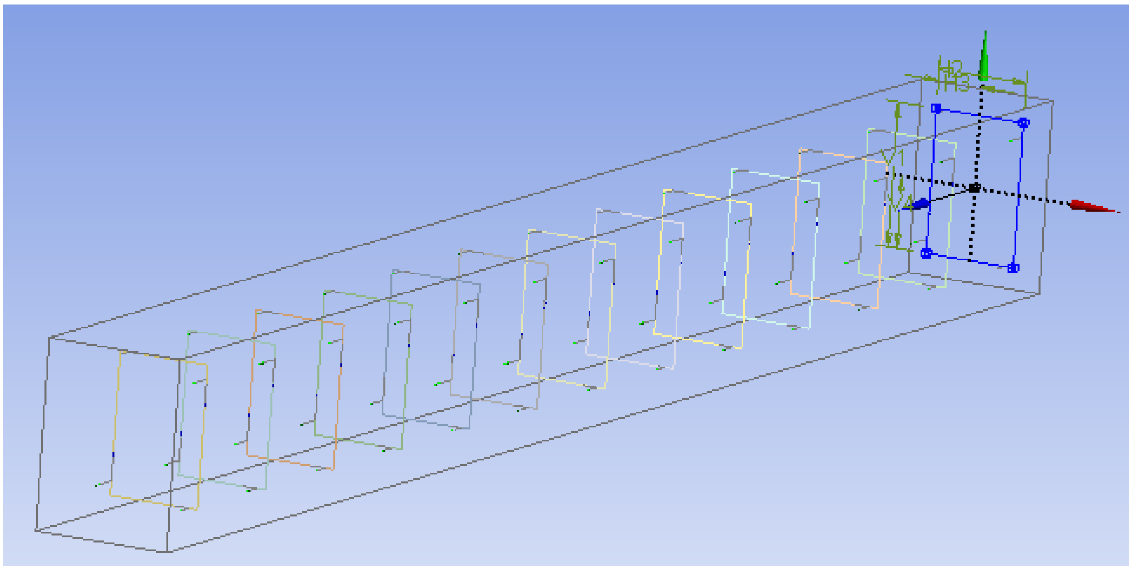
Select with Box Select all the yellow lines, Apply.



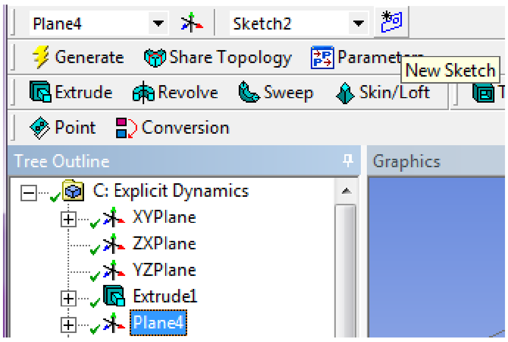
Revert to Single Select, Direction, select a long adge of the column (green here), Apply.



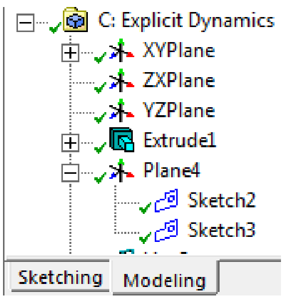
When changing View to Wireframe, we can see this. Observe that each line has its own coordinate system.



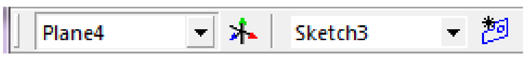
Select Plane4 (your numbering can be different), click the New Sketch button.



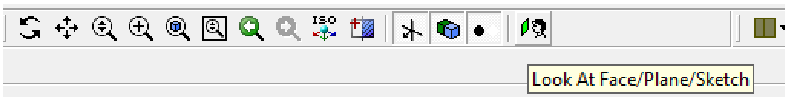
A Sketch3 appears, go to the Skecthing tab.



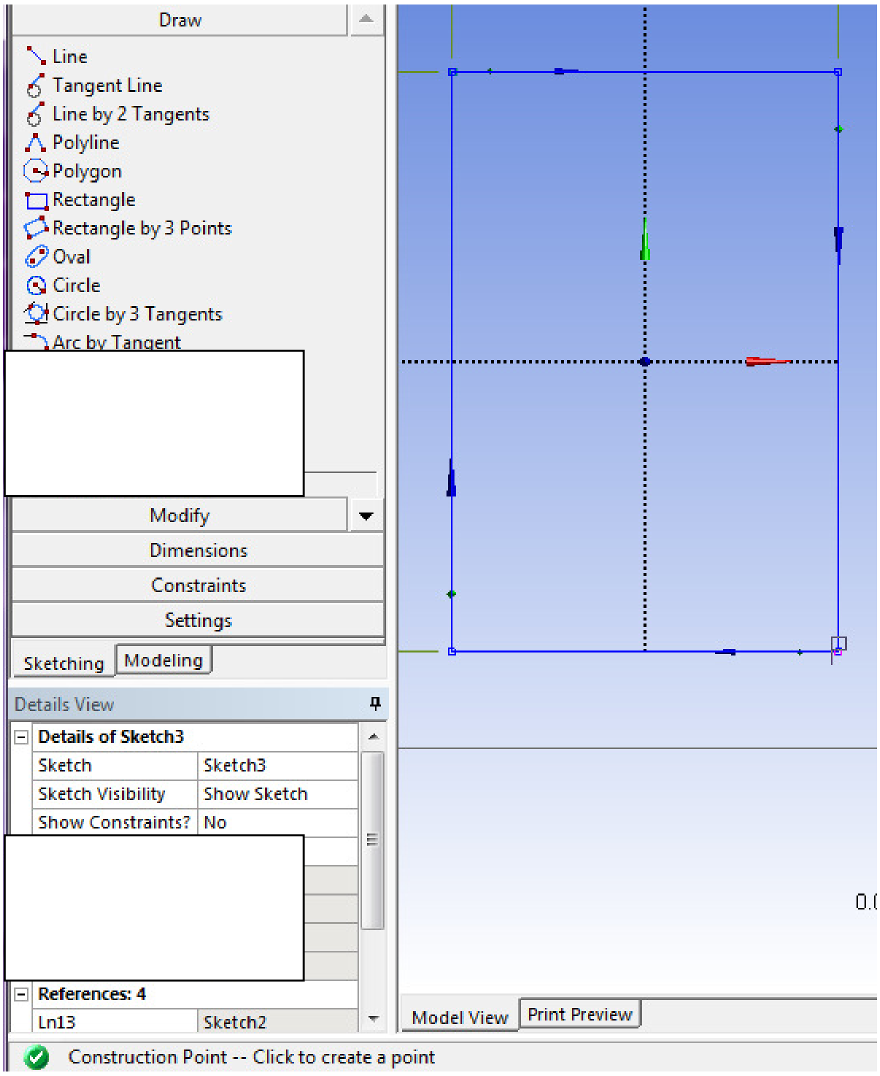
This helps to see where we are currently at.



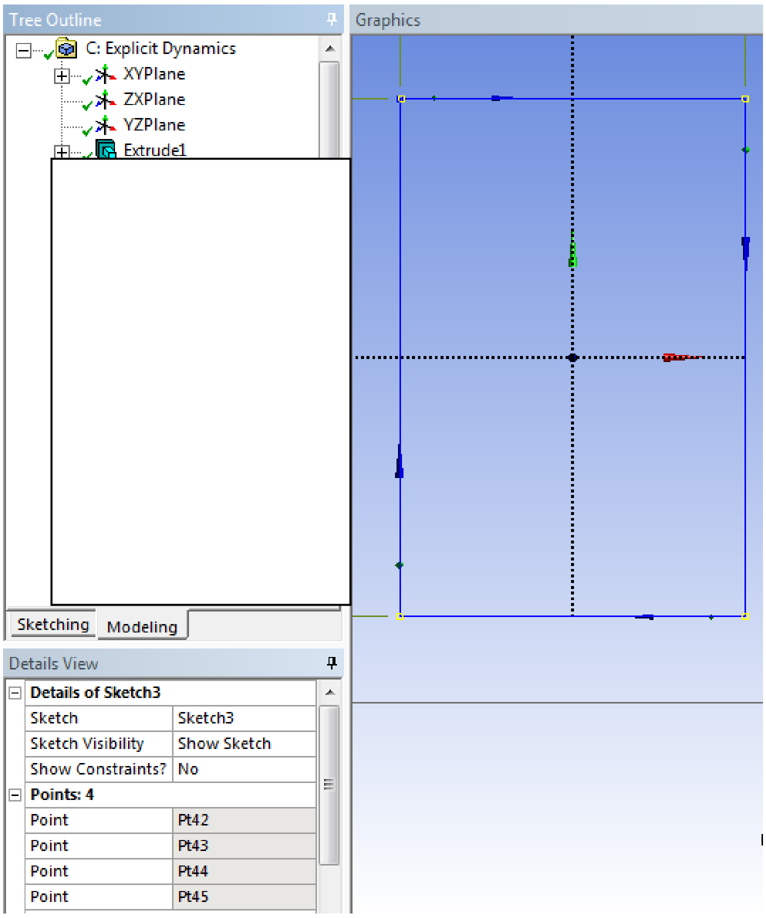
Click the Look At Face/ Plane/ Sketch button.



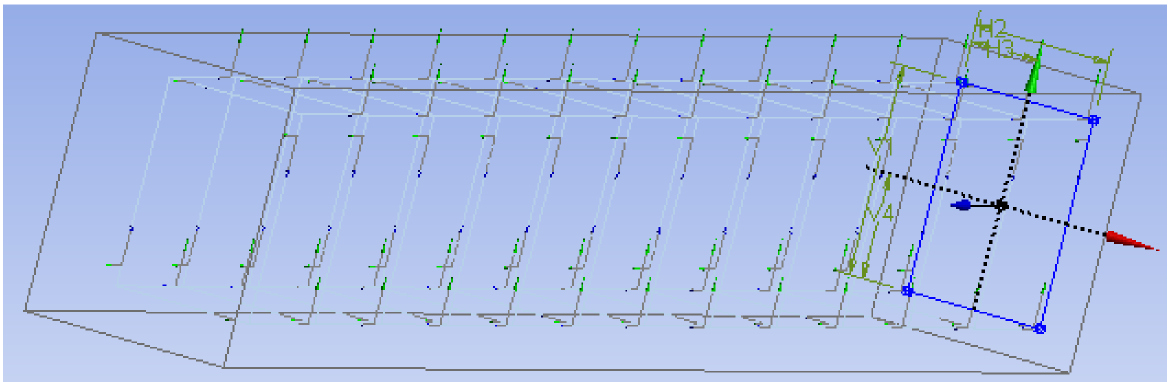
Apply 4 on each corner of the Sketch3.



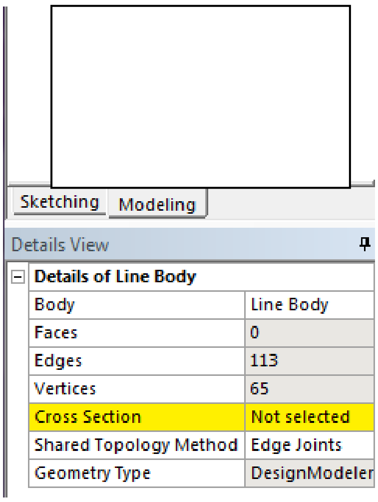
Got to the Modeling tab. There is no need to Extrude a sketch if it is shown with green check symbol. Press the Extrude button. Observe

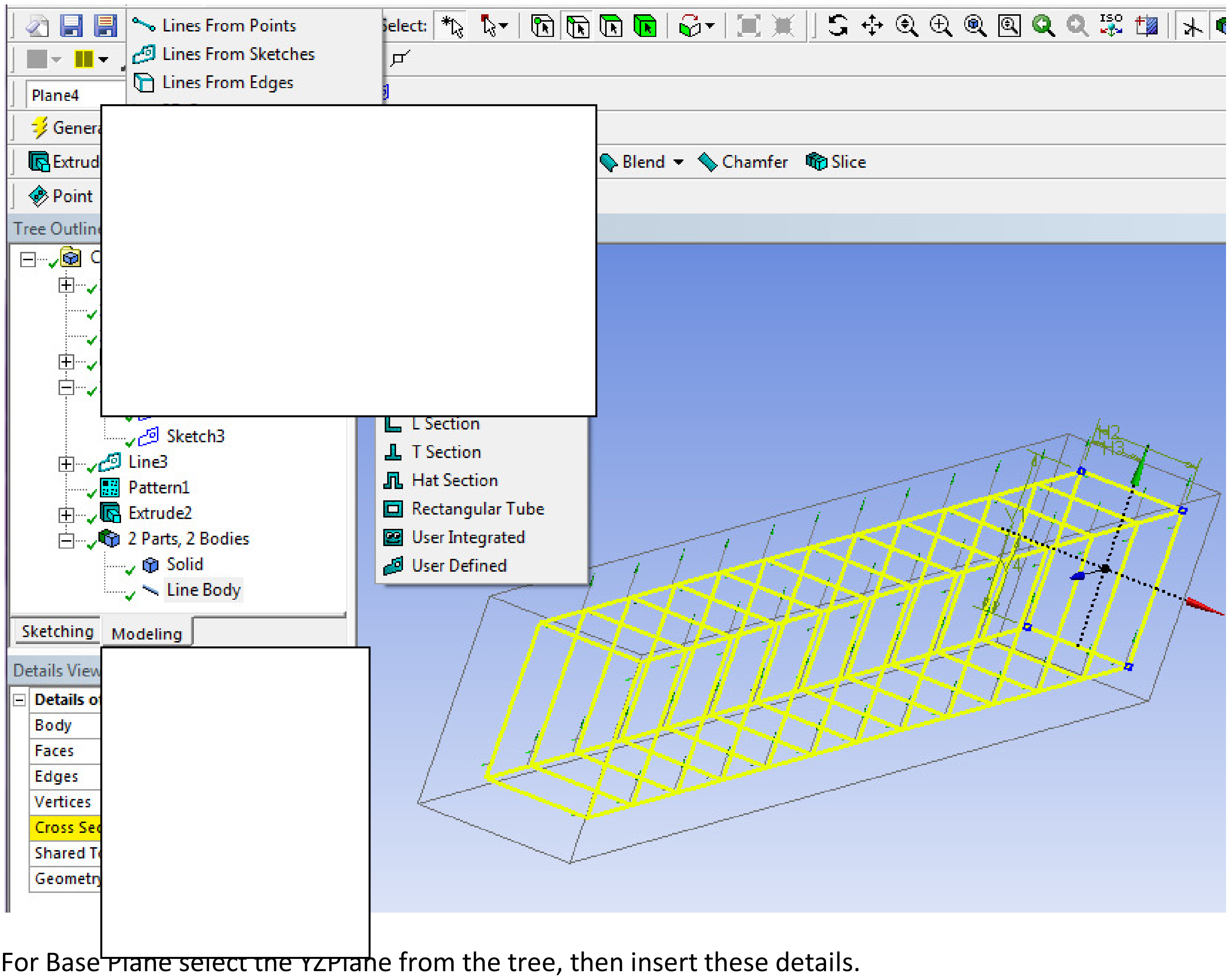


If still in wireframe view, these are the

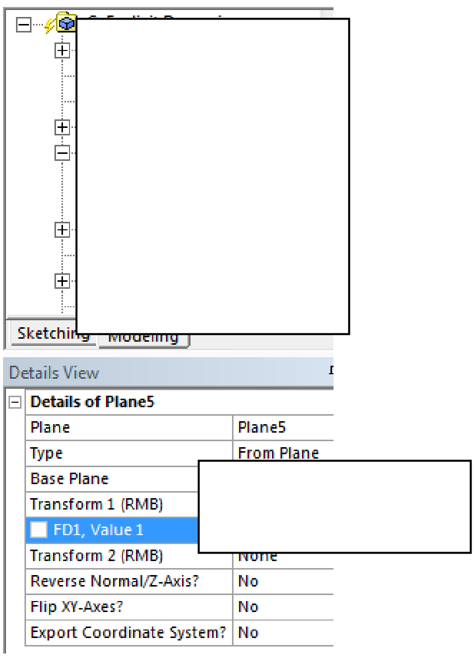


When clicking the Line Body, we observe that it needs a Cross Section.

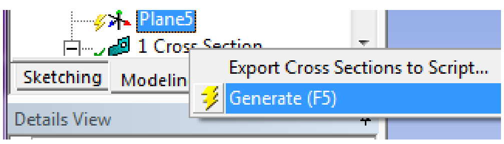




For Base Plane select the YZPlane from the tree, then insert these details.

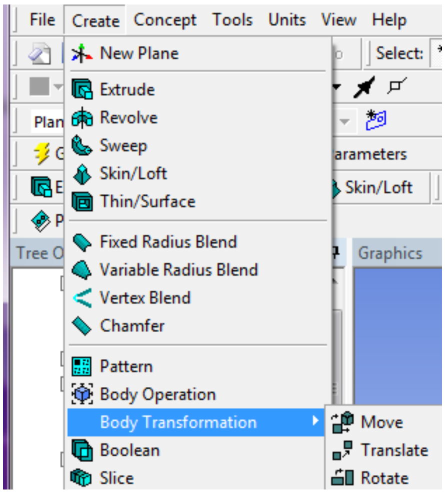


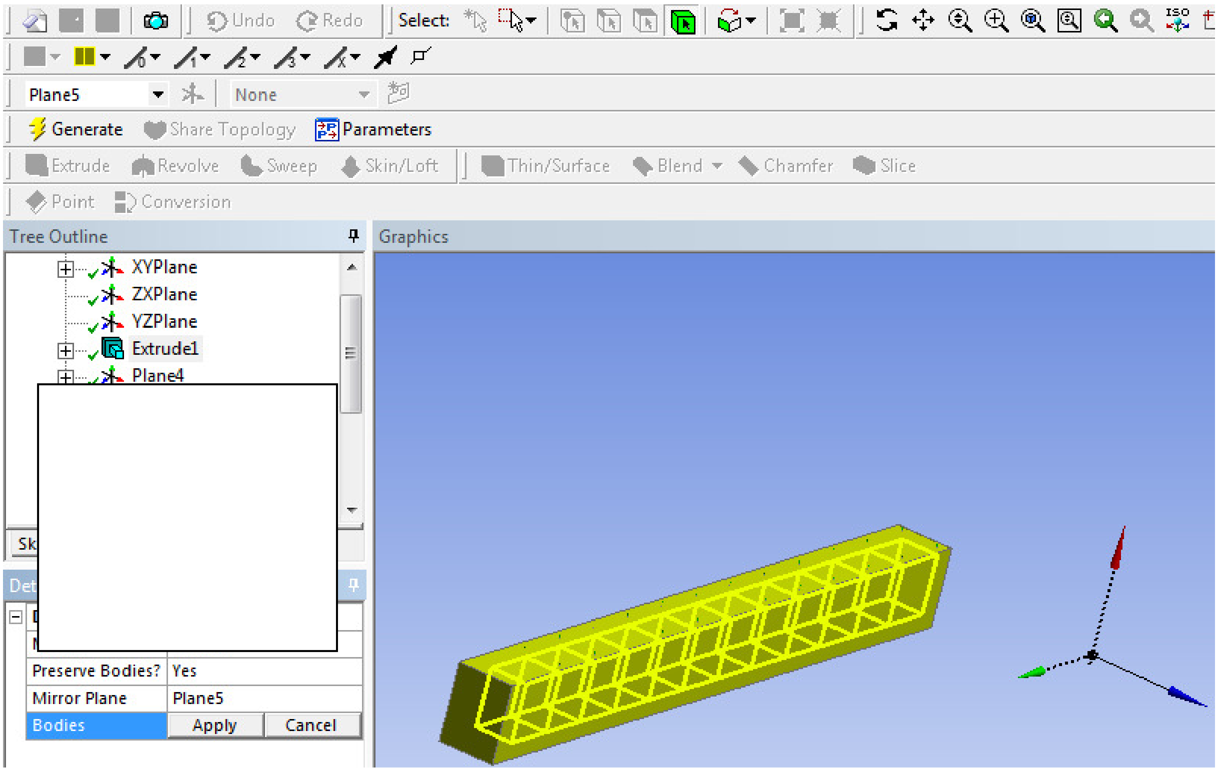
Right click Plane5, Generate.



This is the result.

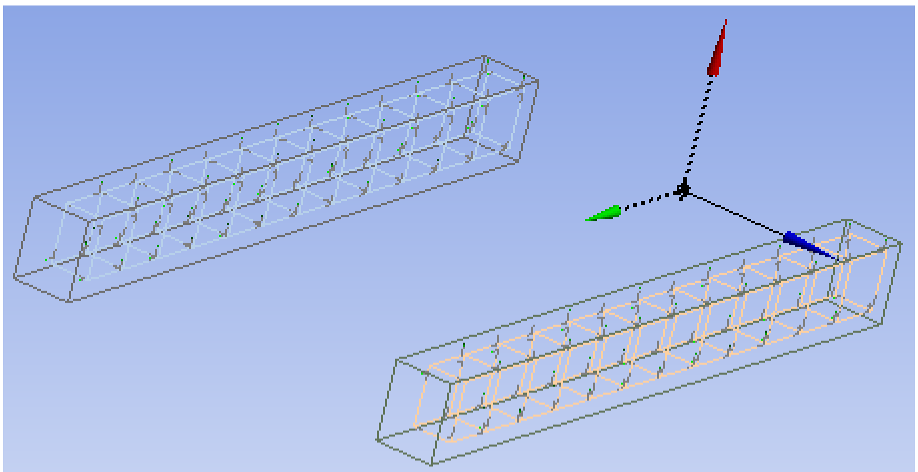
Go to Create, Body Transformation,





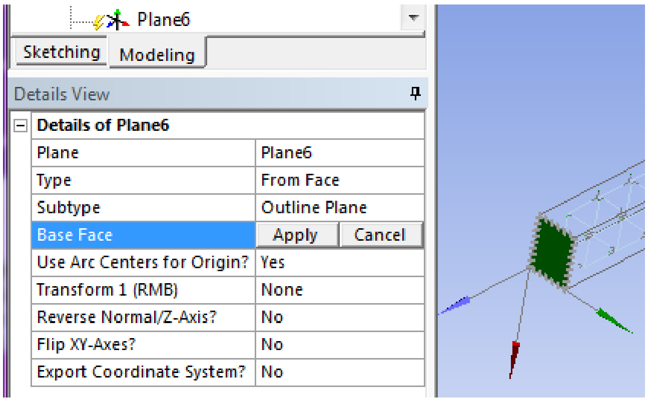
Select Mirror1, right click, Generate.

This is the result.



Create a New Plane.

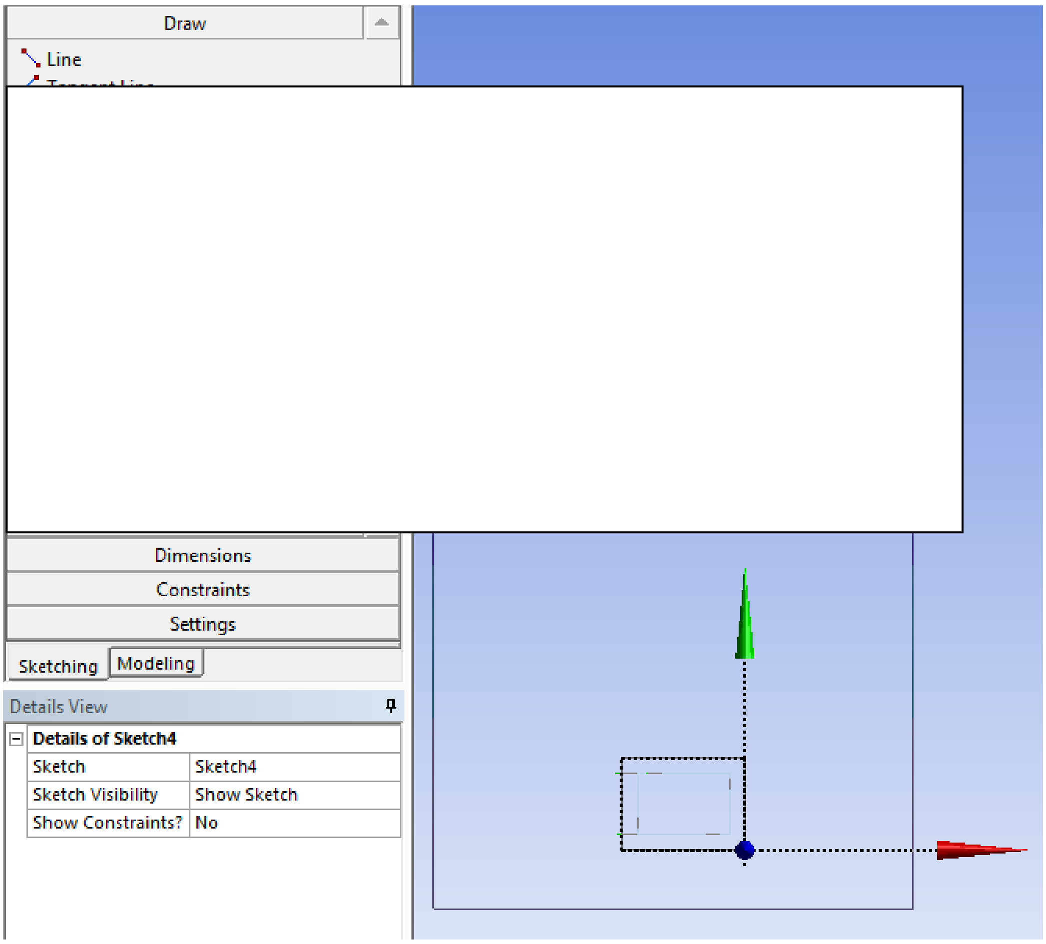
Insert these details and select one of the faces (green here), opposite of XYPlane base, Apply.

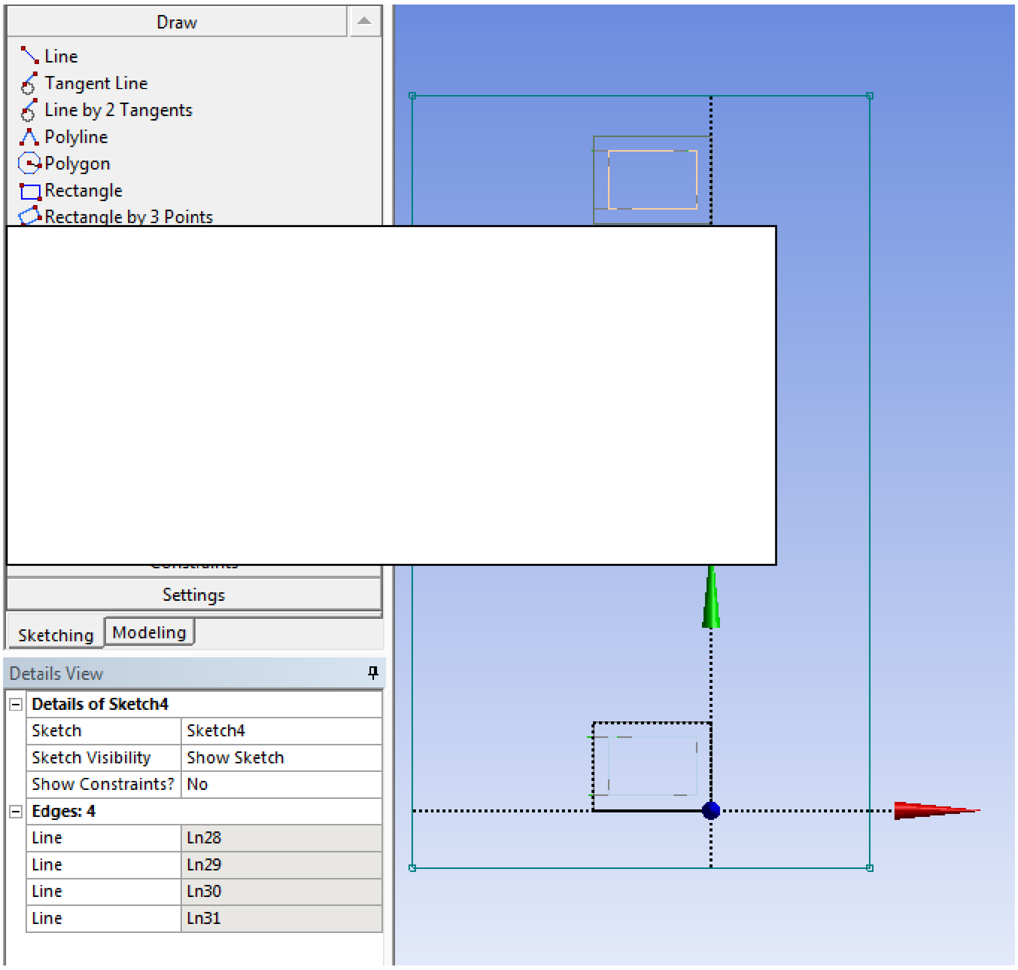


Select Plane6, right click, Generate.

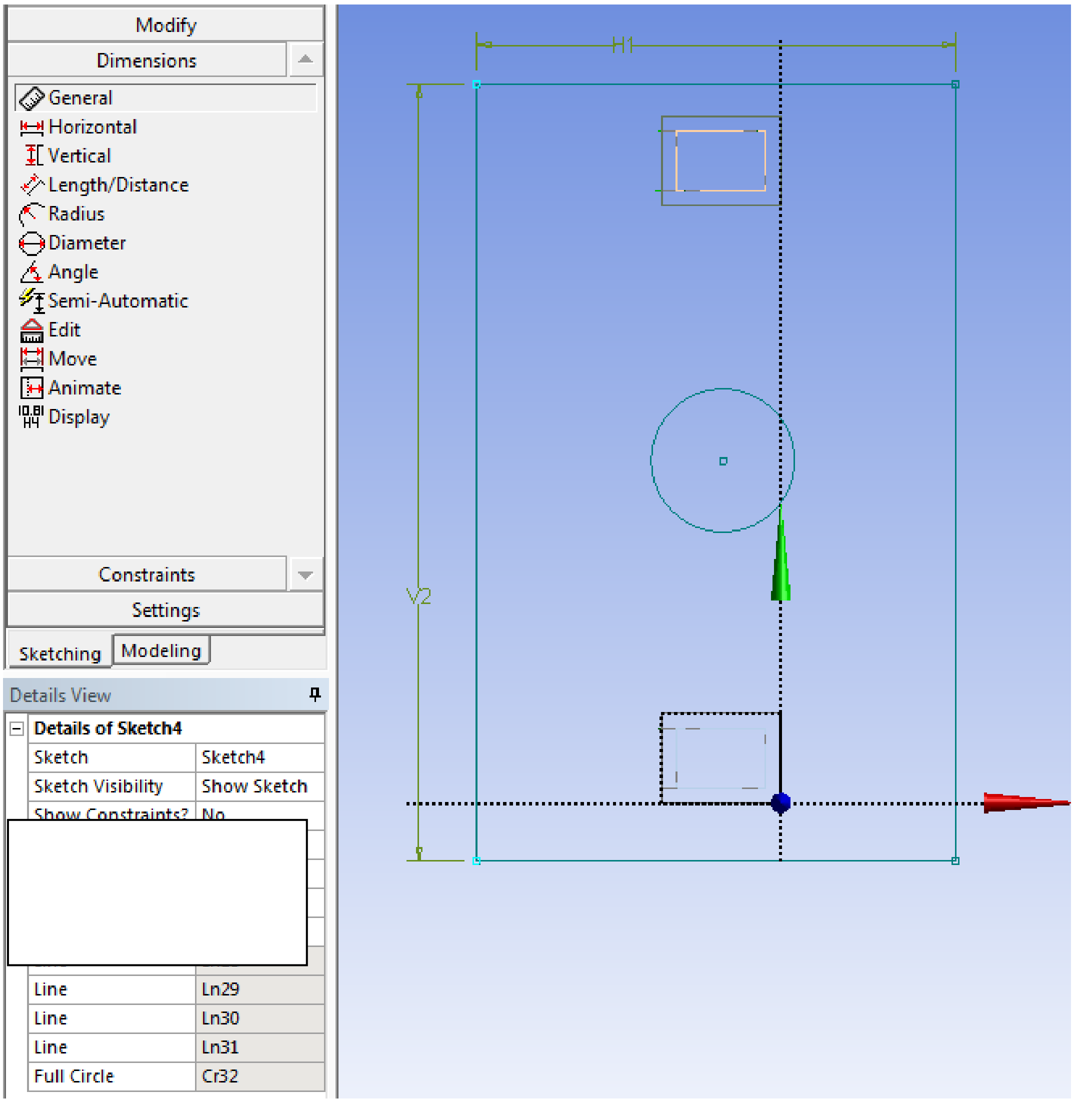
Select Plane6, New Sketch.

Go to the Sketching tab, click the Look At Face/ Plane/ Sketch button.

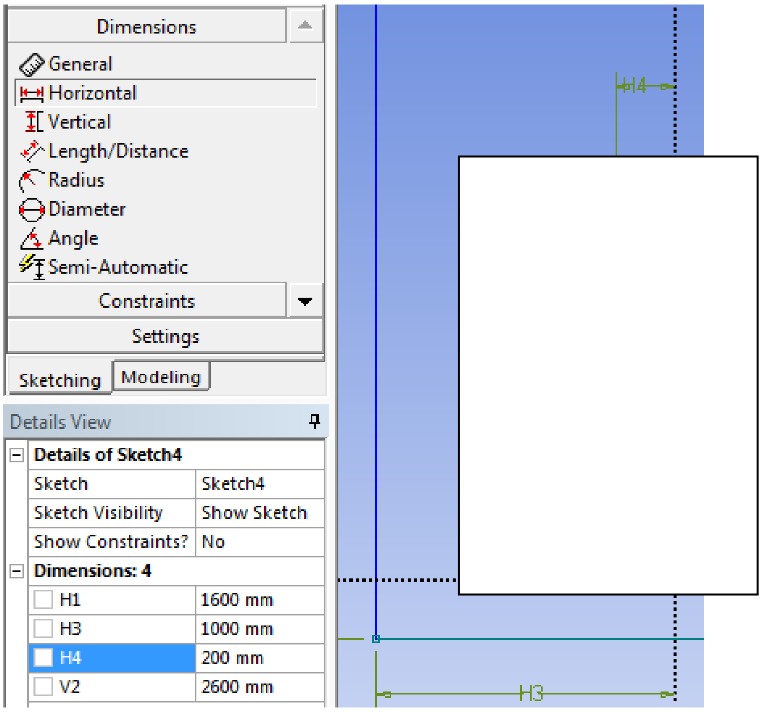




Go to Dimensions, General and assign these values for the top and left line.

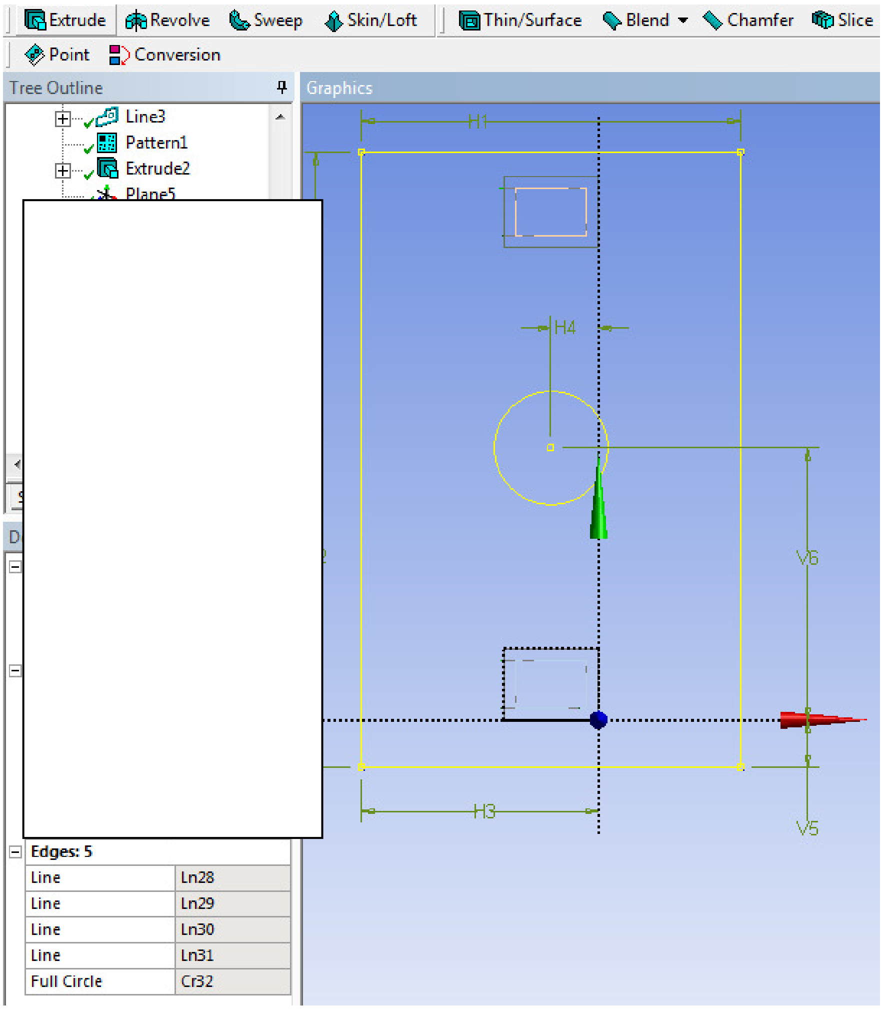


Dimension the left line about the Y (green) axis of Plane6 with 1000 mm and the circle about the same axis with 20 mm, as seen here.

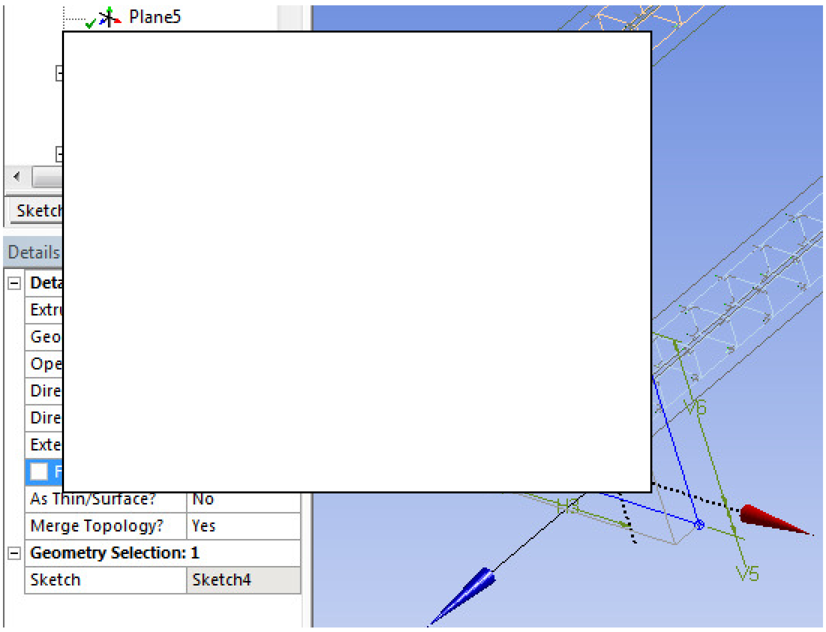


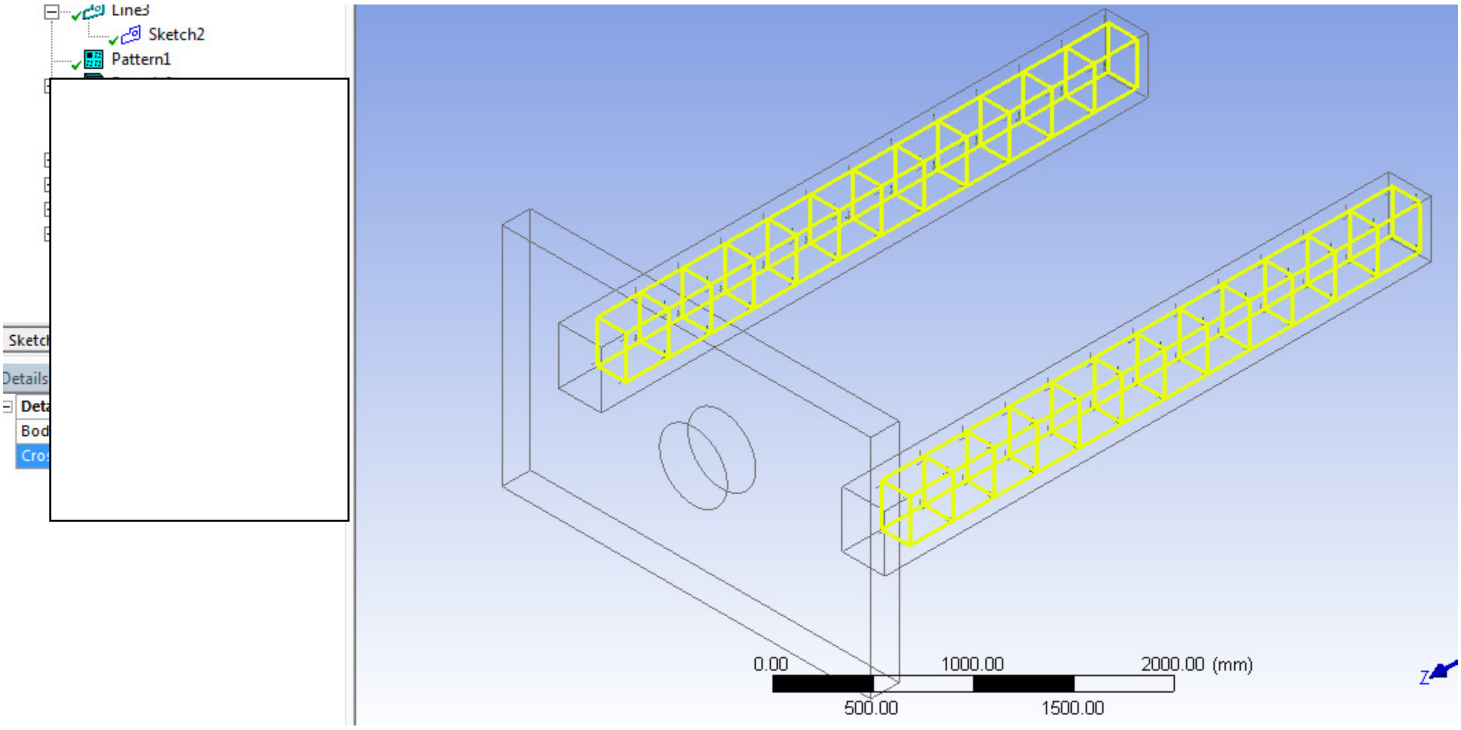
Dimension the bottom line about the X (red) axis of Plane6 with 2000 mm and the circle about the same axis with 1150 mm, as seen here.

Go to the Modeling tab, then click Extrude.

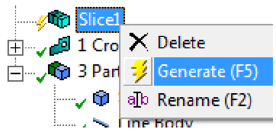


Assign these details, right click Extrude3, Generate.

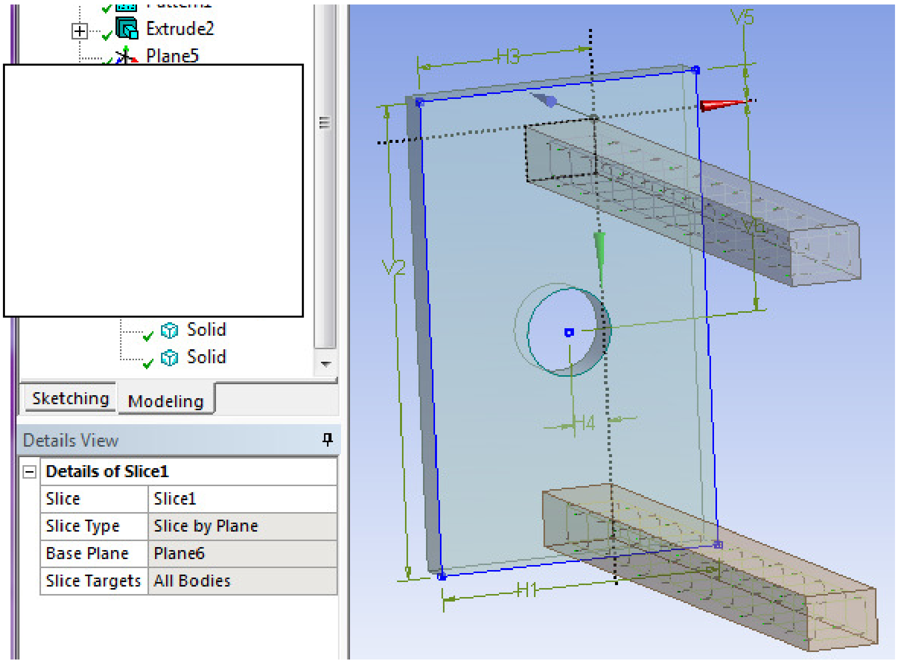




Select base Plane = Plane6, Apply. Right click Slice1, Generate.

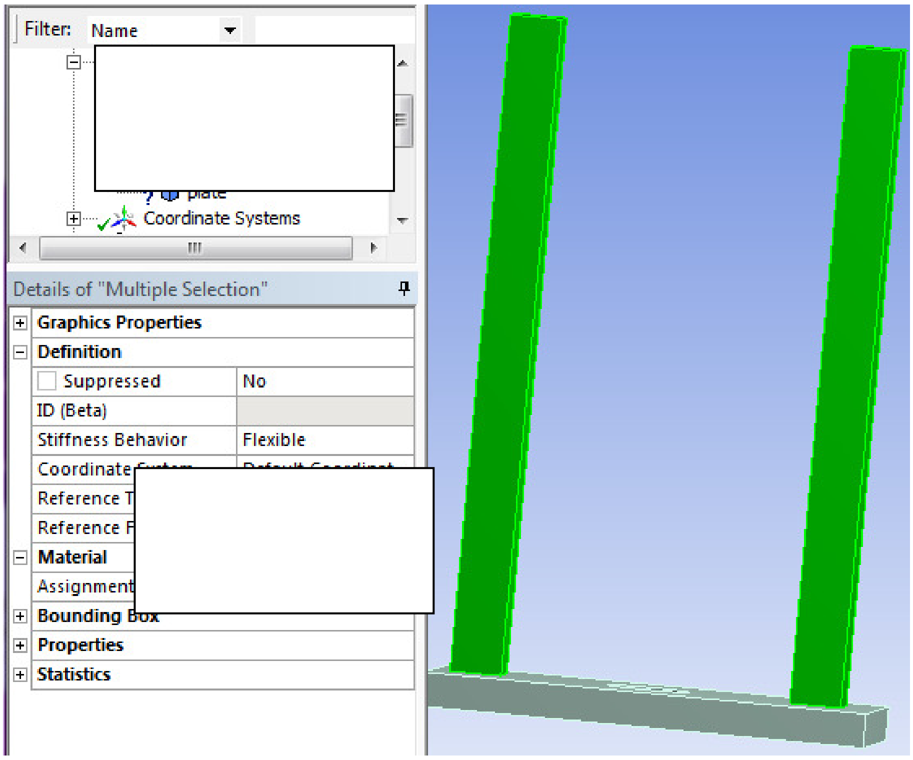


If everything went well, this should be the resulting geometry, observe the additional bodies; now the split are separated from the top plate with hole in it. In ANSYS Workbench, this is the only way that we can use the concrete reinforcing in Explicit Dynamics. File, Close DesignModeler.

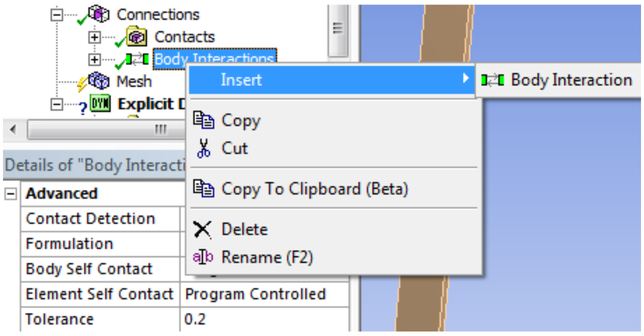


COMPRESSION

Geometry: Rename the bodies for easier handling.

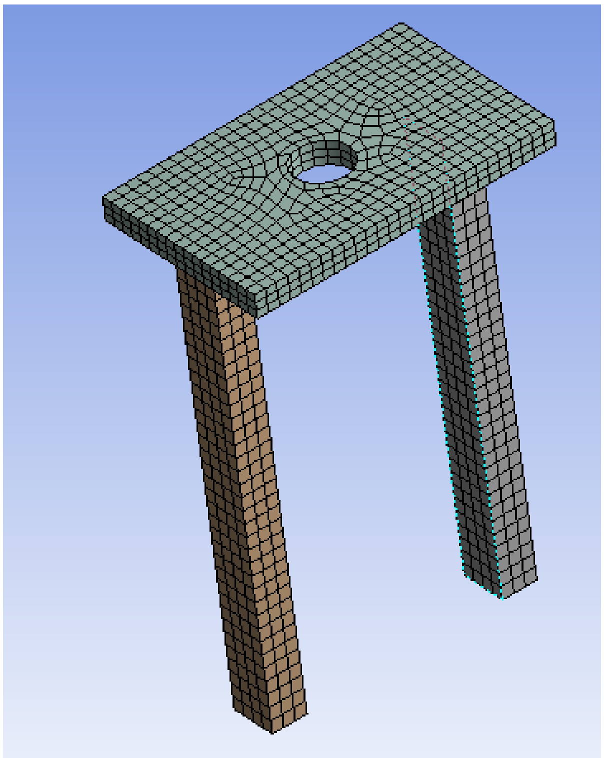


Connections: Right click Body Interaction, Insert, Body Interaction.

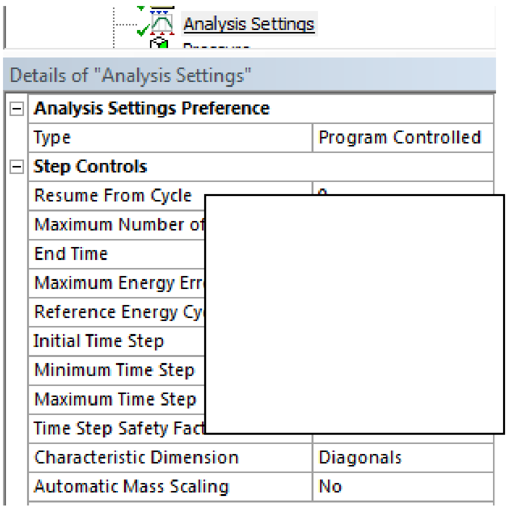


Mesh: It shoud have these details.

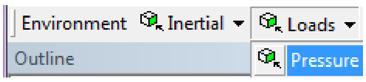
Once generated, the Mesh should look like here.



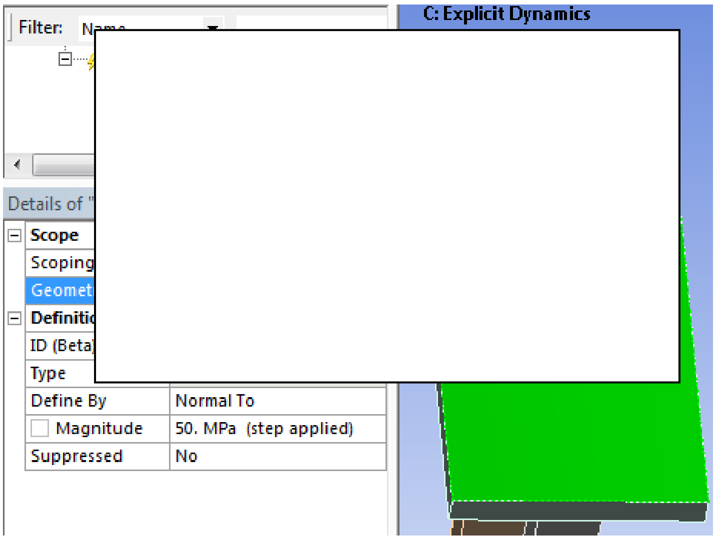
Explicit Dynamics, Analysis Settings: Assign these details. End Time



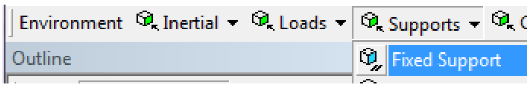
From the Environment toolbar insert a Pressure item.



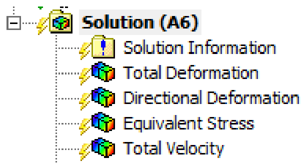
On the green face,



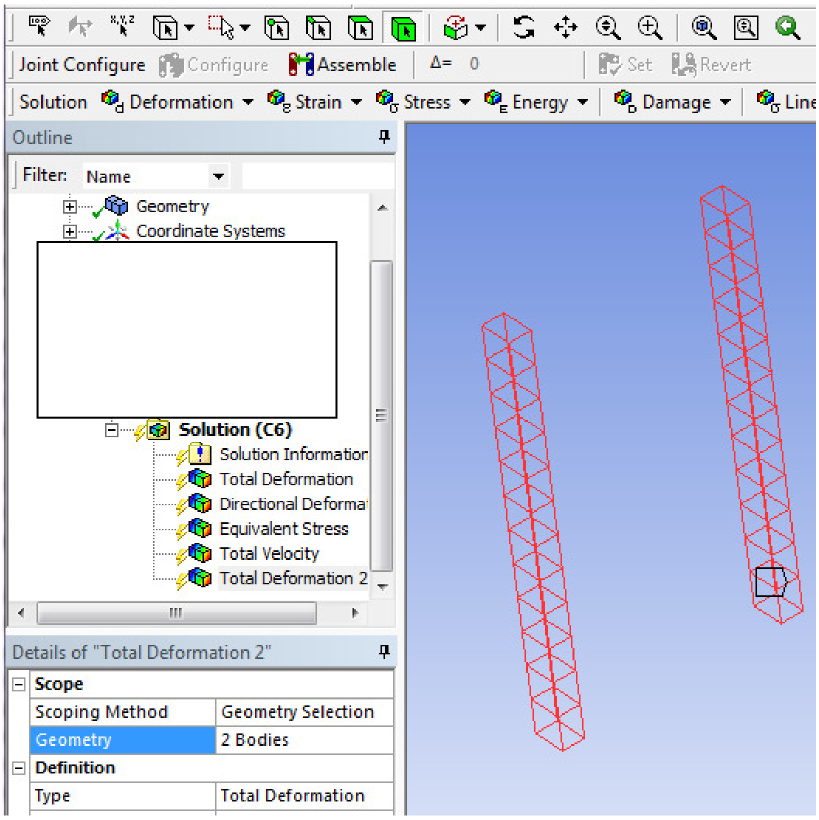
From the Environment toolbar insert a Fixed Support item.



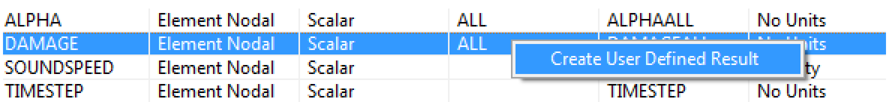
Solution: Insert these default items.



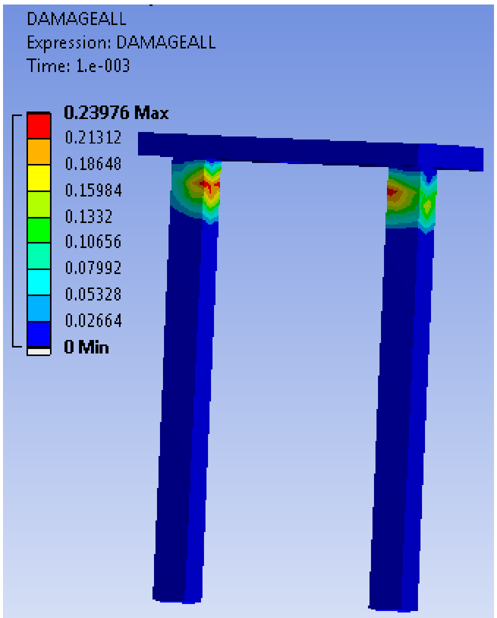
Hide with F9 the other bodies. Apply a Total Deformation only on the Line Bodies (red here), using the Box Selection.



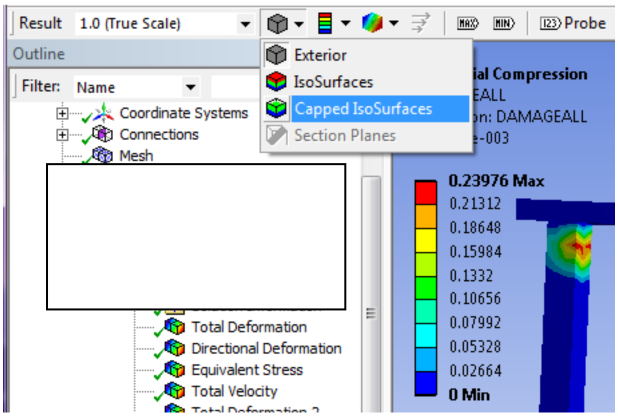
Insert an Equivalent Stress only on the Line Bodies (red here). Solve.



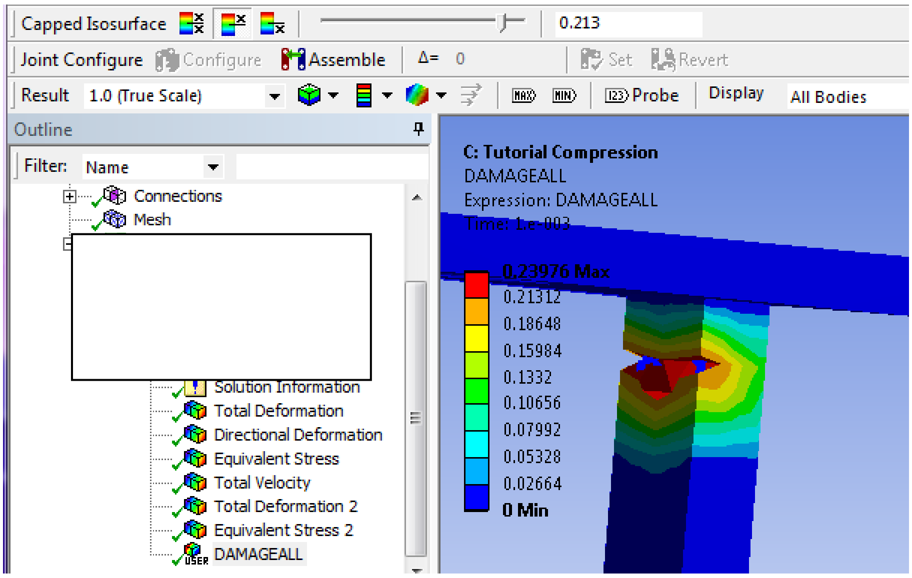
Right click ansy solution item Evaluate All Results. This is the default plot.



From the Result toolbar, choose Capped Isosurfaces.

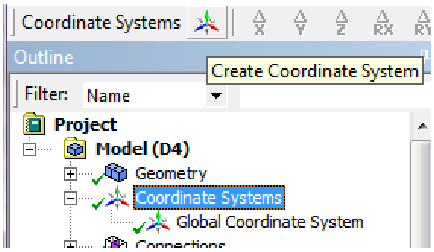


Zooming in, vor the default threshold of 0.213 it will show shome damaged elements.

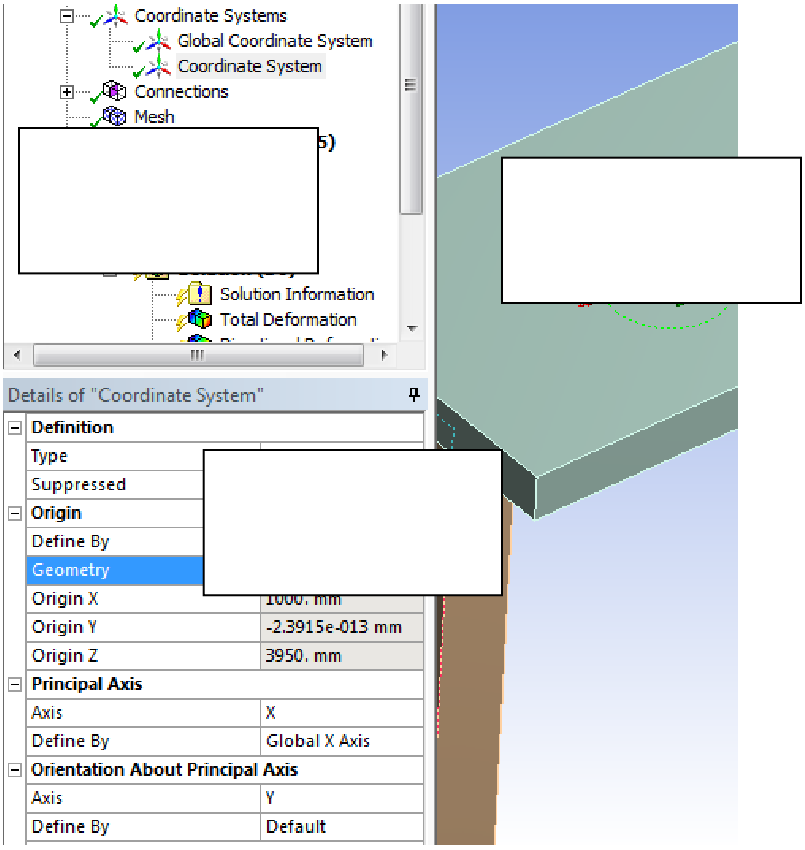


TWISTING: Go to Project Schematic and Duplicate the scenario.

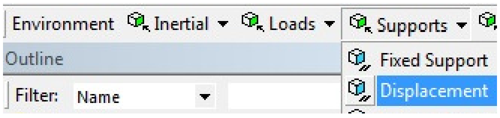
Coordinate System: Create one from its toolbar.



Apply it on the hole (green here), Type = Cylindrical.

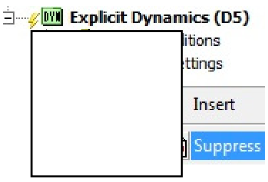


Explicit Dynamics: Assign A Displacement.

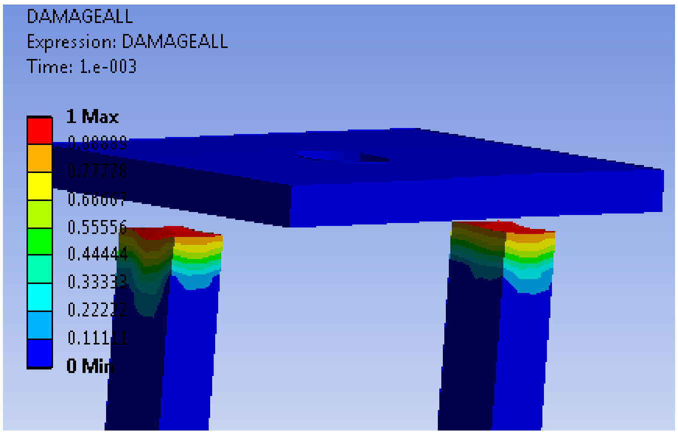


Select the hole, green here, Apply,

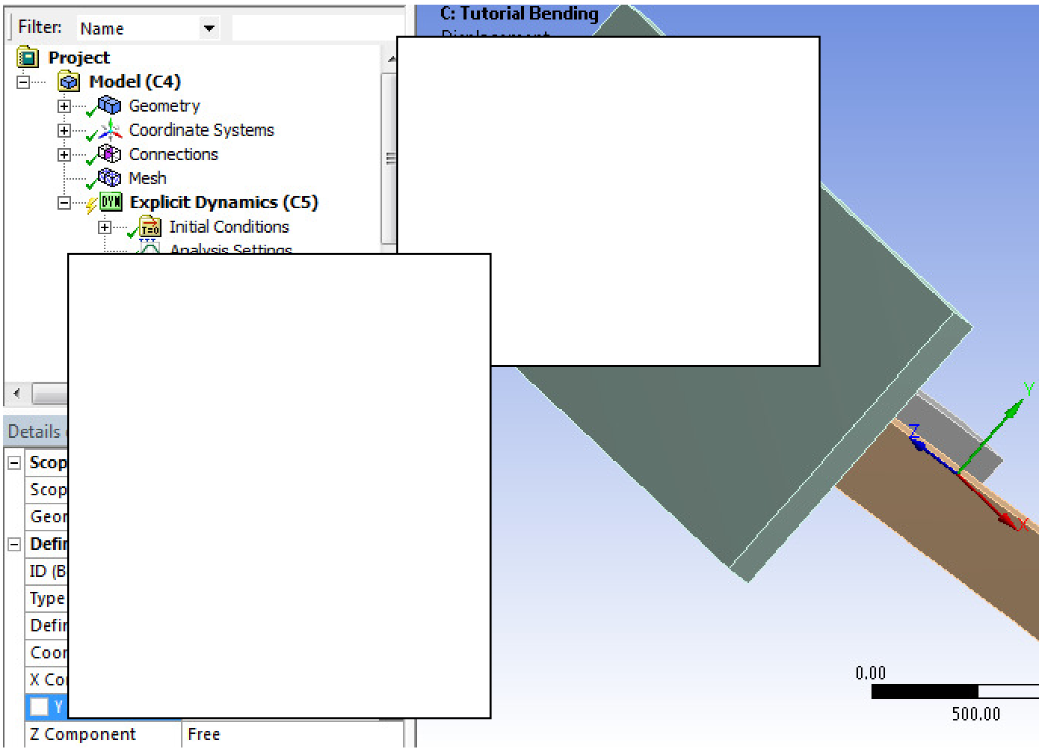
Suppress the existing, Solve.



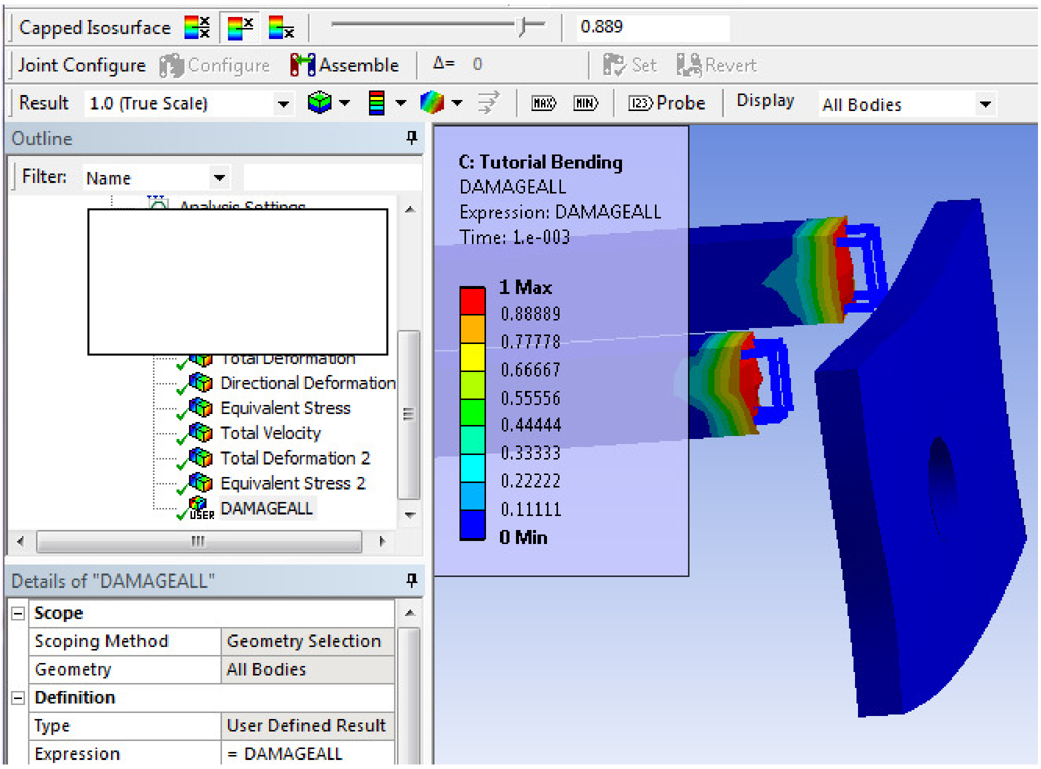
Observe that the DAMAGEALL item shows that the concrete will fail where it joins the plate. The rebars are a little lower. You can see them be decreasing the threshold of the Capped Isosurfaces item, as before. Again, twist with more degrees to see more damage.



BENDING: Go to Project Schematic and Duplicate the scenario.
Modify the Displacement to look like here. Observe the Coordinate System. Solve.



In this case, because of the damage, the plate slides in the direction of bending.



Further homework:

- change the

- double all the values of the loads and displacements, solve, draw the conclusions