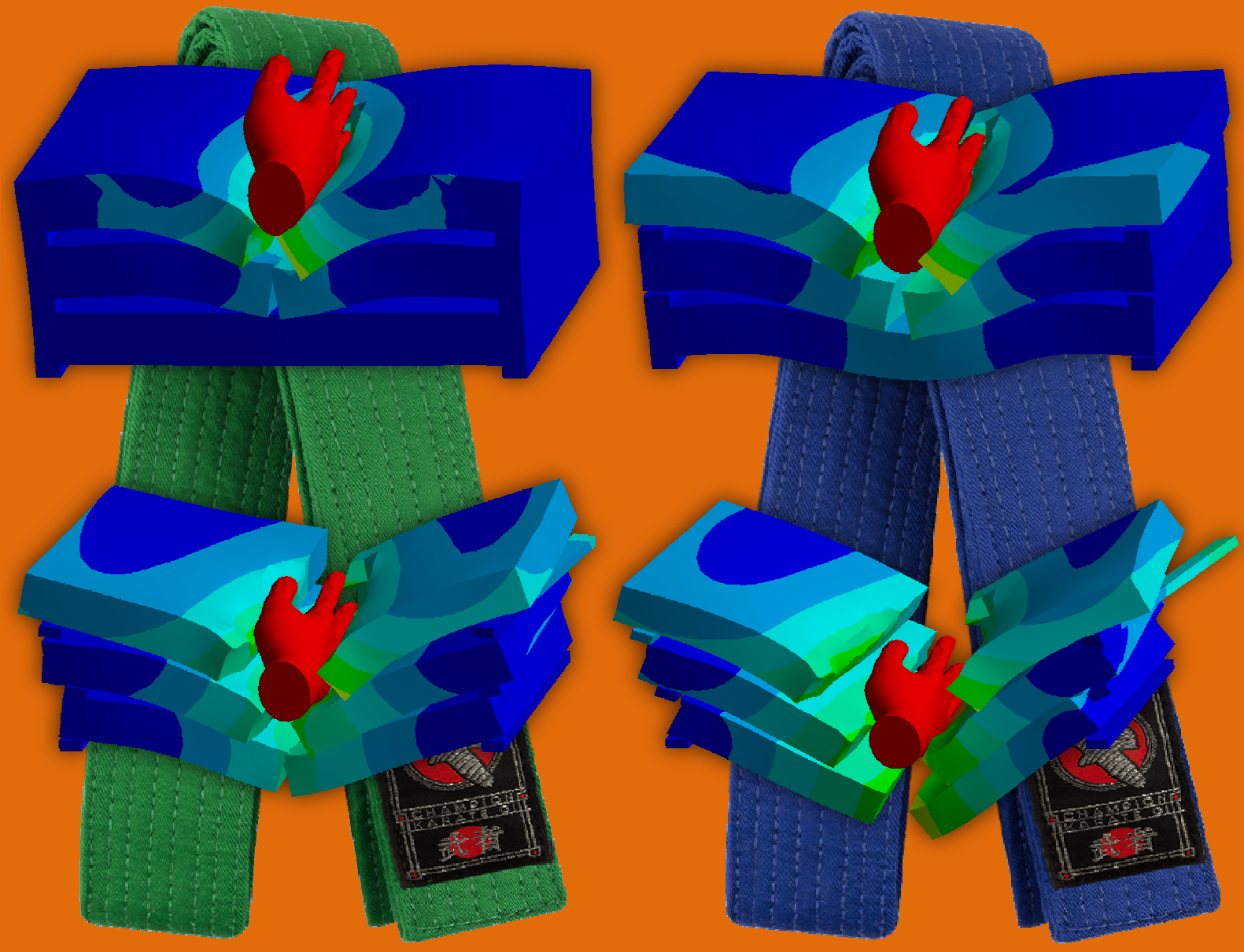
Become a Black Belt in ANSYS Workbench

by Claudiu Danila

Volume 2: GREEN and BLUE belt

-20 fast tutorials for advanced users-



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

Foreword

Hi all!

My name is Claudiu and I am in the FEA domain since the year 2000. It is my pleasure to share with you, in the form of short tutorials, 20 of the simulations that are present on the Solved FEA pages of *expertfea.com* and *GrabFEA.com*

In this 2nd volume, the rhythm is a little more alert, this meaning that you have done before, in ANSYS Workbench, at least the fixed cantilever beam FEA everyone does in the beginning :))

I hope you will find the spaced out, aerated perspective on the problems agreeable, because I didn't want the information to be crowded in any way and miss some important points!

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

With the risk of repeating myself, if you like this domain, start working NOW! Maybe some voices said that it is hard, maybe others said that there are no workplaces in this domain and try to discourage you - do not listen to anyone pulling you aside from this path and begin the work ASAP!

My luck was that 17 years ago, there was no one to discourage me, because this domain was in its beginnings. But I had to spend countless nights doing FEA, searching for software demos, for 3D models and so on - while my friends were partying in the clubs or becoming Quake and Half Life champions... In order to obtain something, you need to give something; in this case, you need to give time, effort and dedication. Not much, if you start TODAY; the important thing is to work constantly!

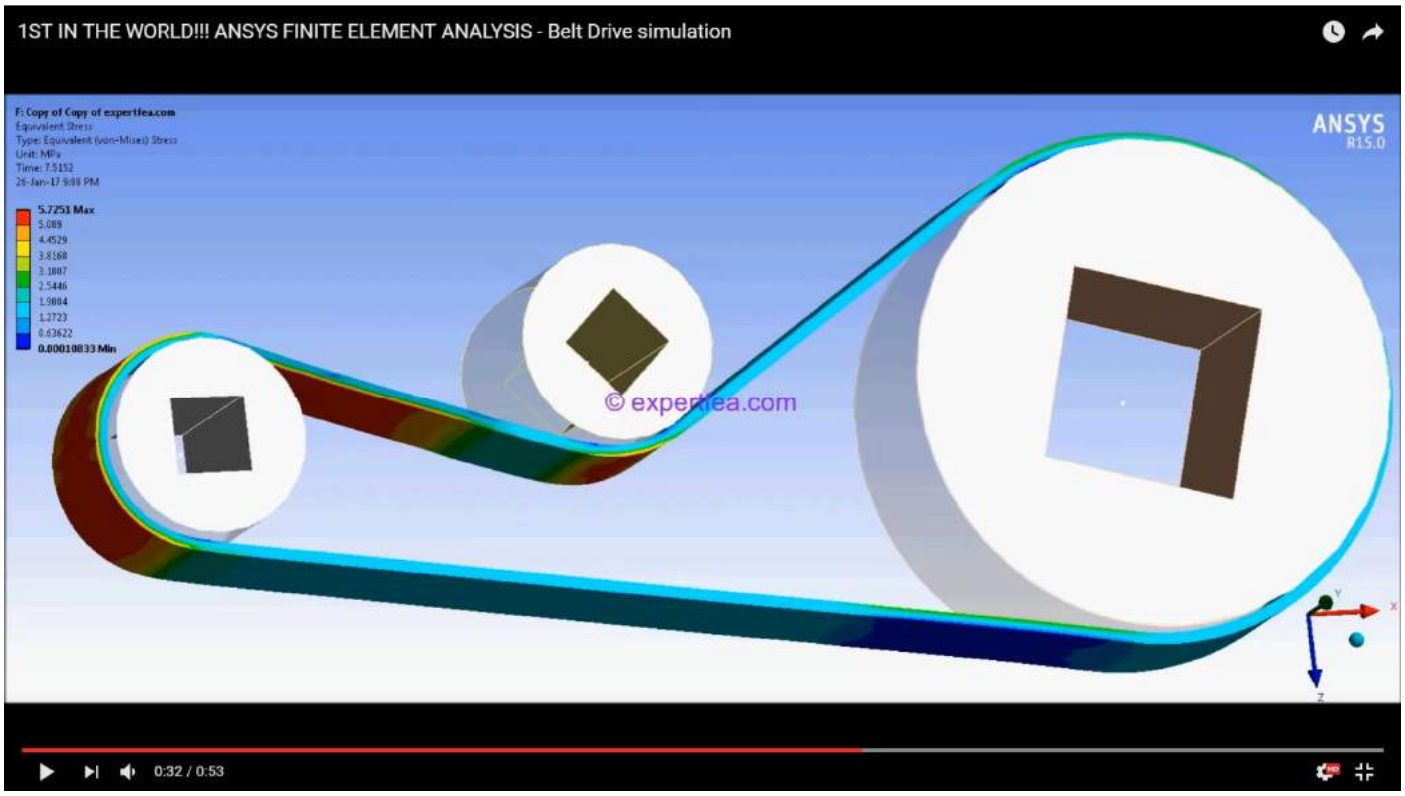
The water drops carve the hardest rock, not using their force, but by repetition - do the same, with small but constant effort and you'll carve your place in the FEA world, sooner than you think!

Also, prepare yourself for the 3rd volume, which will be the most advanced, but with the least details; you need to do things very fast and by yourself, because in the next volume no one will hold your hands, it's a promise!

Have faith in yourself and in your awesome future!

Claudiu, 26th of March 2017

CASE 21: ANSYS FINITE ELEMENT ANALYSIS - Transient Structural Belt Drive simulation



Engineering Data (Materials): All rollers are made of default Structural Steel. Duplicate a Structural Steel material and change the properties to these ones, to obtain a rubber. Apply this rubber to the belt.

Outline of Schematic A2: Engineering Data

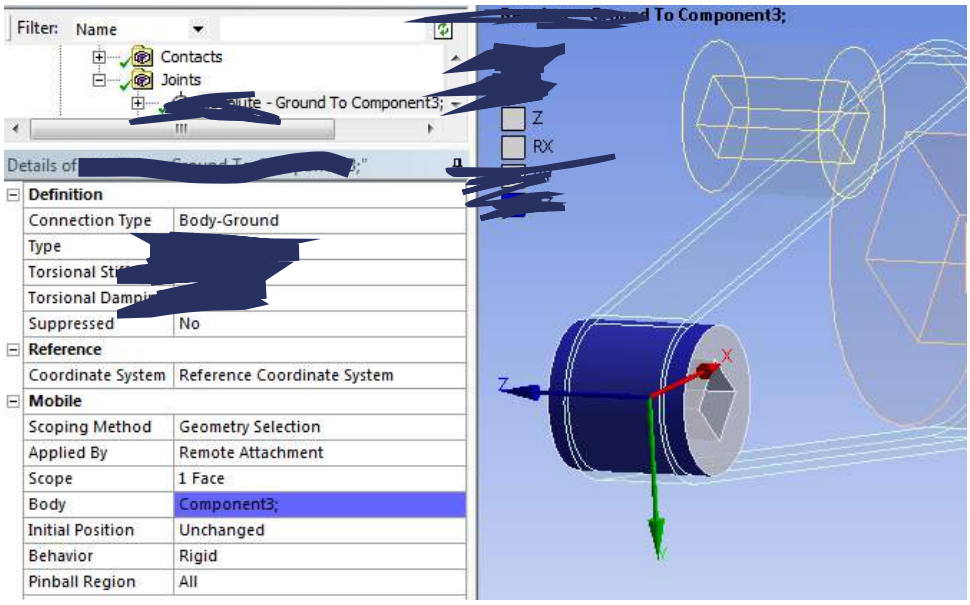
	A	B	D
1	Contents of Engineering Data		Description
2	Material		
3	Rubber 60 MPa		Fatigue Data at zero mean stress comes from 1998 BPV Code, Section 8, Div 2, Table 5-110.1
4	Structural Steel		Fatigue Data at zero mean stress comes from 1998 BPV Code, Section 8, Div 2, Table 5-110.1
*	Click here to add a new material		

Properties of Outline Row 3: Rubber 60 MPa

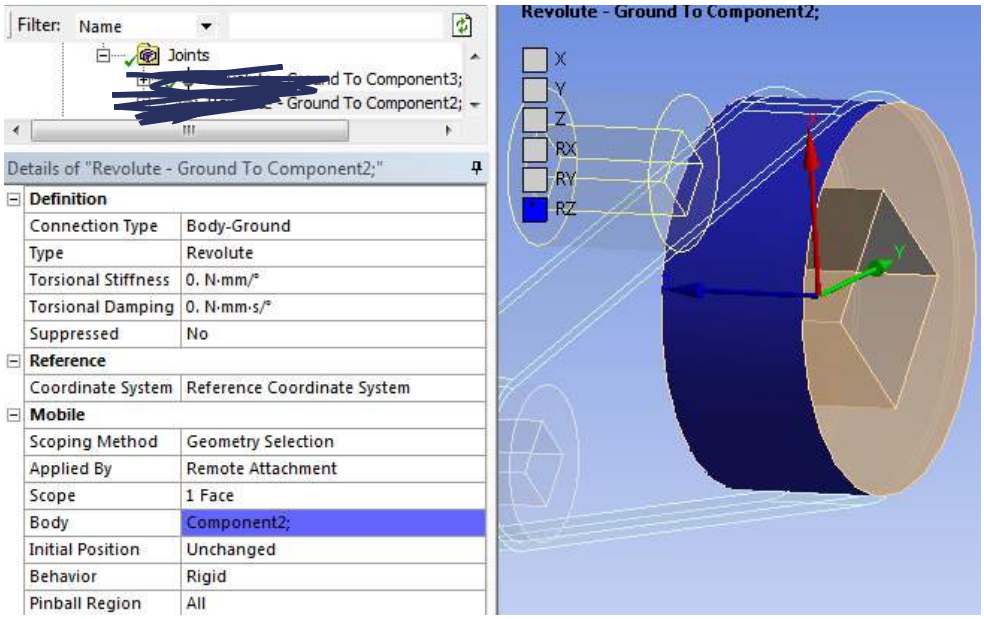
	A	B	C
1	Property	Value	Unit
2	Density	[redacted]	kg m^-3
3	Isotropic Elasticity		
4	Derive from	Young's Modulus and Poisson's Ratio	
5	Young's Modulus	[redacted]	Pa
6	Poisson's Ratio	[redacted]	

Geometry: 2017_jan_25_belt_wheels_v2.x_t

Create a Body-Ground Revolute Joint on the small roller, blue here.

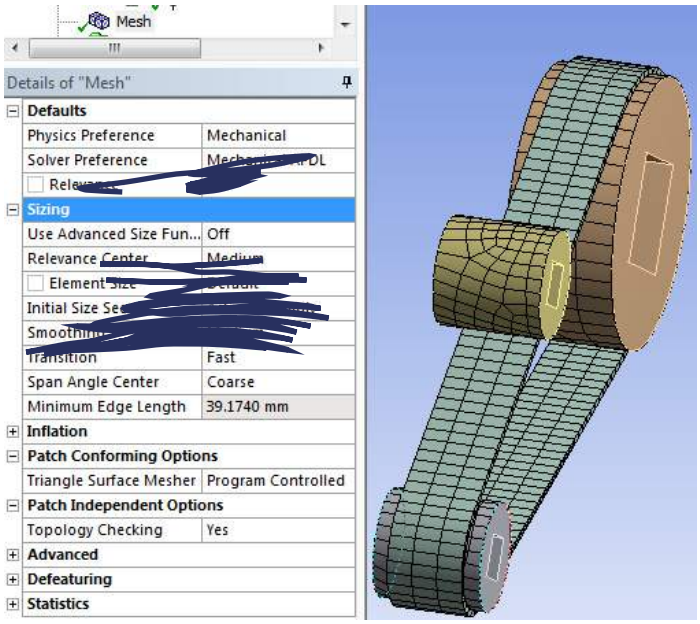


Create a [redacted] on the big roller, blue here.



Create a [redacted] on the tensioning roller, blue here.

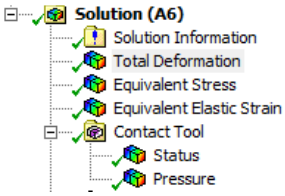
Mesh: The mesh has these details and it should look like here.



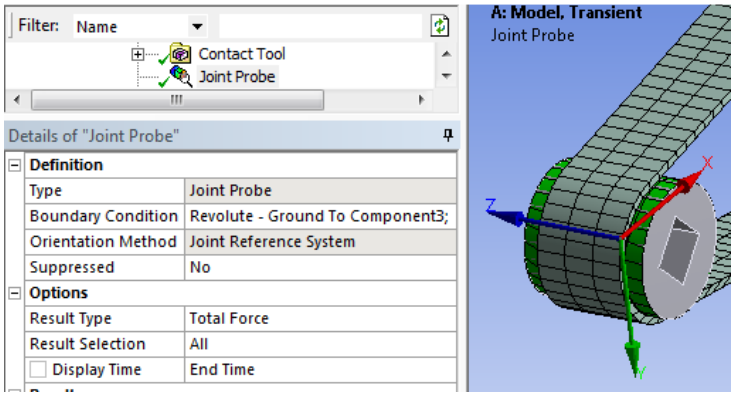
Tabular Data

	Steps	Time [s]	
1	1	0.	
2	1	1.	
3	2	2.	
4	3	3.	
5	4	4.	
6	5	5.	
7	6	6.	
8	7	7.	
9	8	8.	

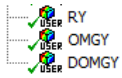
Solution: Insert these default items, for all parts.



Insert a Joint Probe for the small roller.



After the solving is done, click Solution, then Worksheet button to access these extra results for Angle, Angular Velocity and Angular Acceleration.



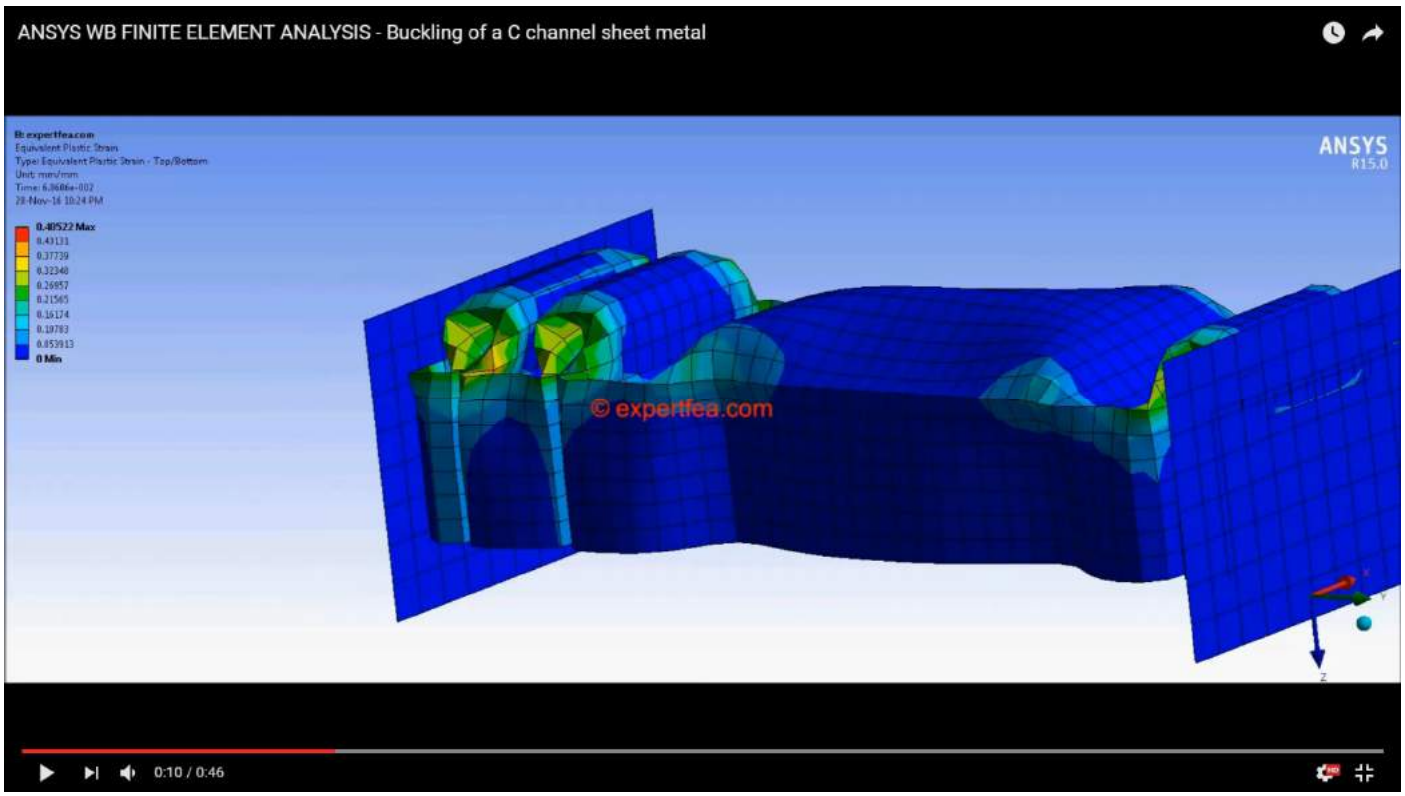
Further homework:

- increase the mesh Relevan[redacted]olve, draw the conclusions

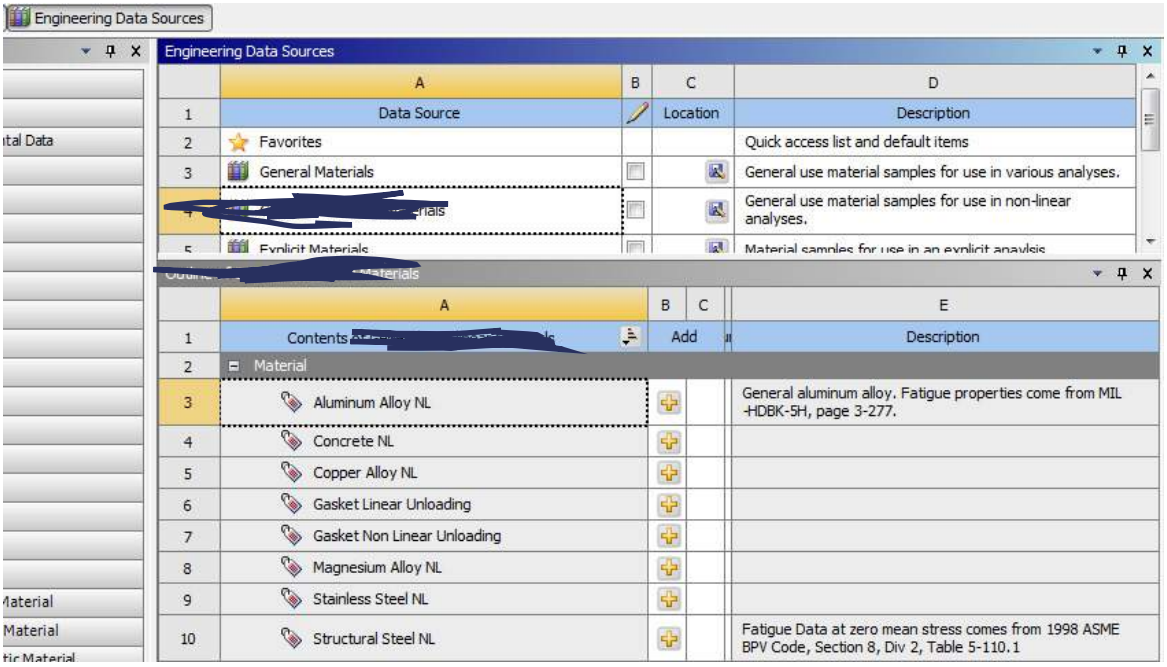
- apply Frictional co[redacted], solve, draw the conclusions

- chang[redacted]of the rubber to half its original value, solve, draw the conclusions

CASE 22: ANSYS WB FINITE ELEMENT ANALYSIS - Explicit Dynamics Buckling of a C channel sheet metal



Engineering Data: Add Structural Steel [illegible] from gene[illegible]ear Materials.



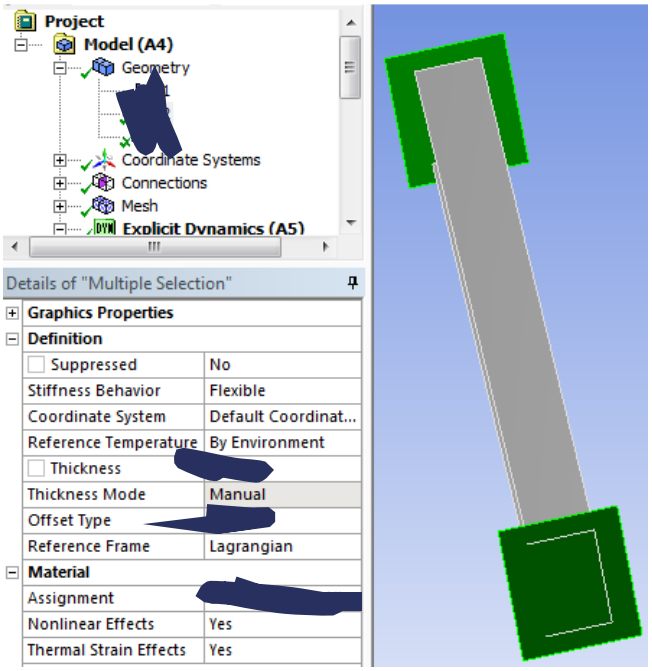
Geometry:2016_nov_28_C_channel_buckling.stp

Apply Structural [redacted] the part seen green here and make its Thickness = 1 mm.

[redacted]

[redacted]

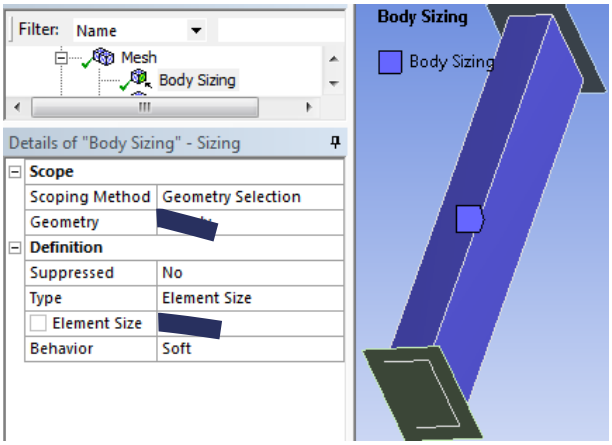
Apply Structural Steel NL to the parts seen green here and make their Thickness = 20 mm.



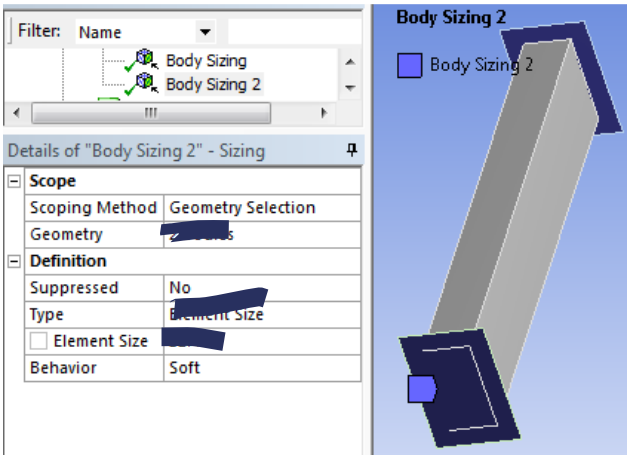
Connections: [redacted]

Mesh: Assign these details.

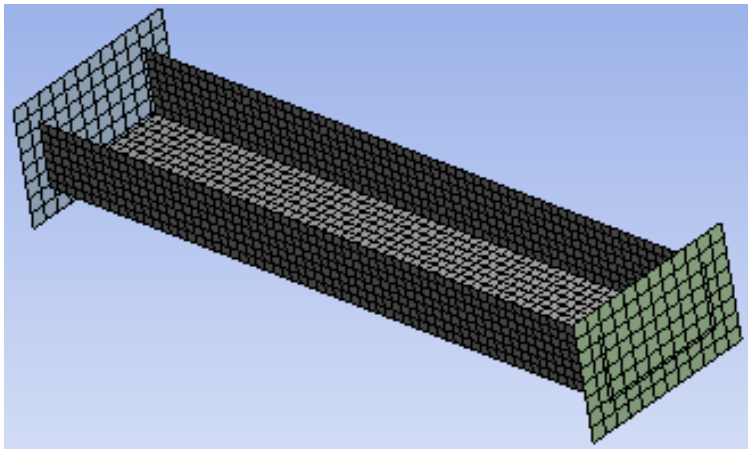
To the blue part assign this Mesh Sizing.



To the blue parts assign this Mesh Sizing.

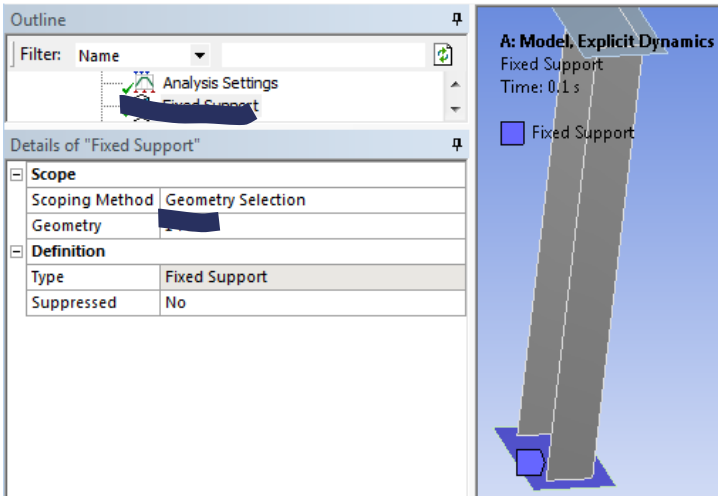


Correctly made, the mesh should look like here.

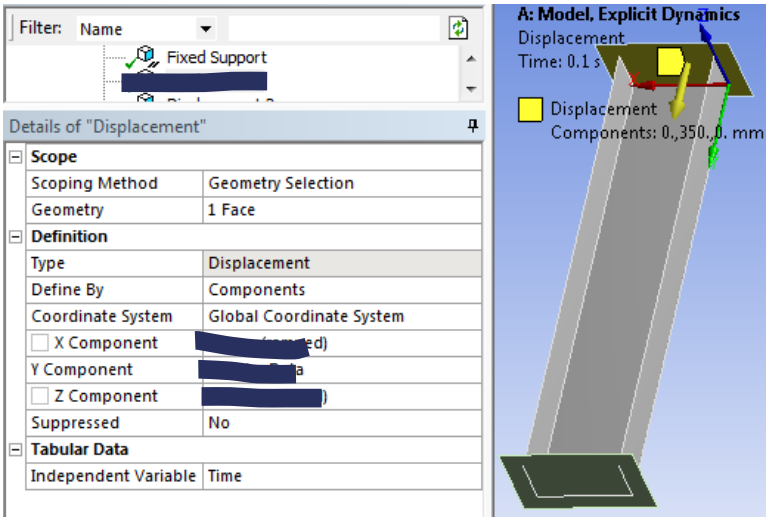


Analysis Settings: Insert these details.

Environment toolbar: Fix the inner face, blue here, from a supporting part.

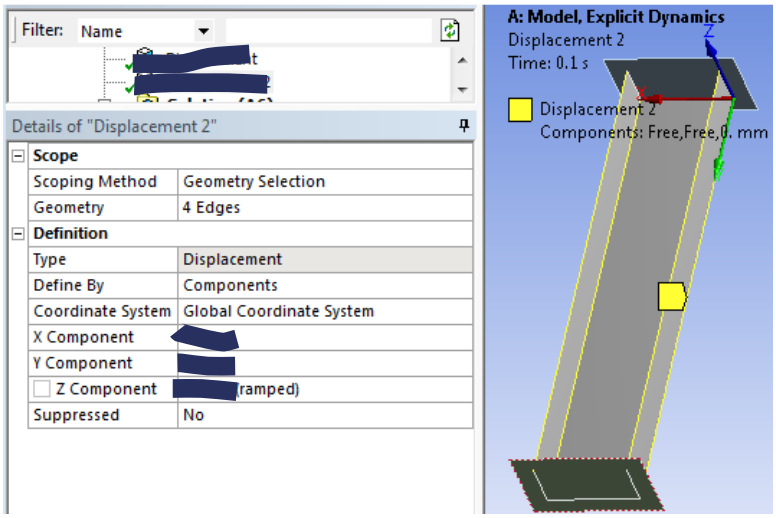


Apply a Displacement on the inner face, yellow here, on the opposite supporting part,.

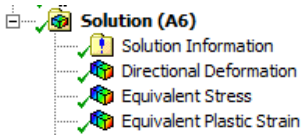


[redacted]

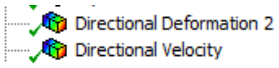
On these 4 yellow edges of the C section part, block the Z displacement, as seen here.



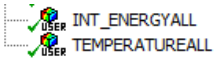
Solution: Insert these default items, on all bodies.



Create these items only for Y axis.



After the FEA was solved, click Solution, then Worksheet button and apply these default items, for all bodies.



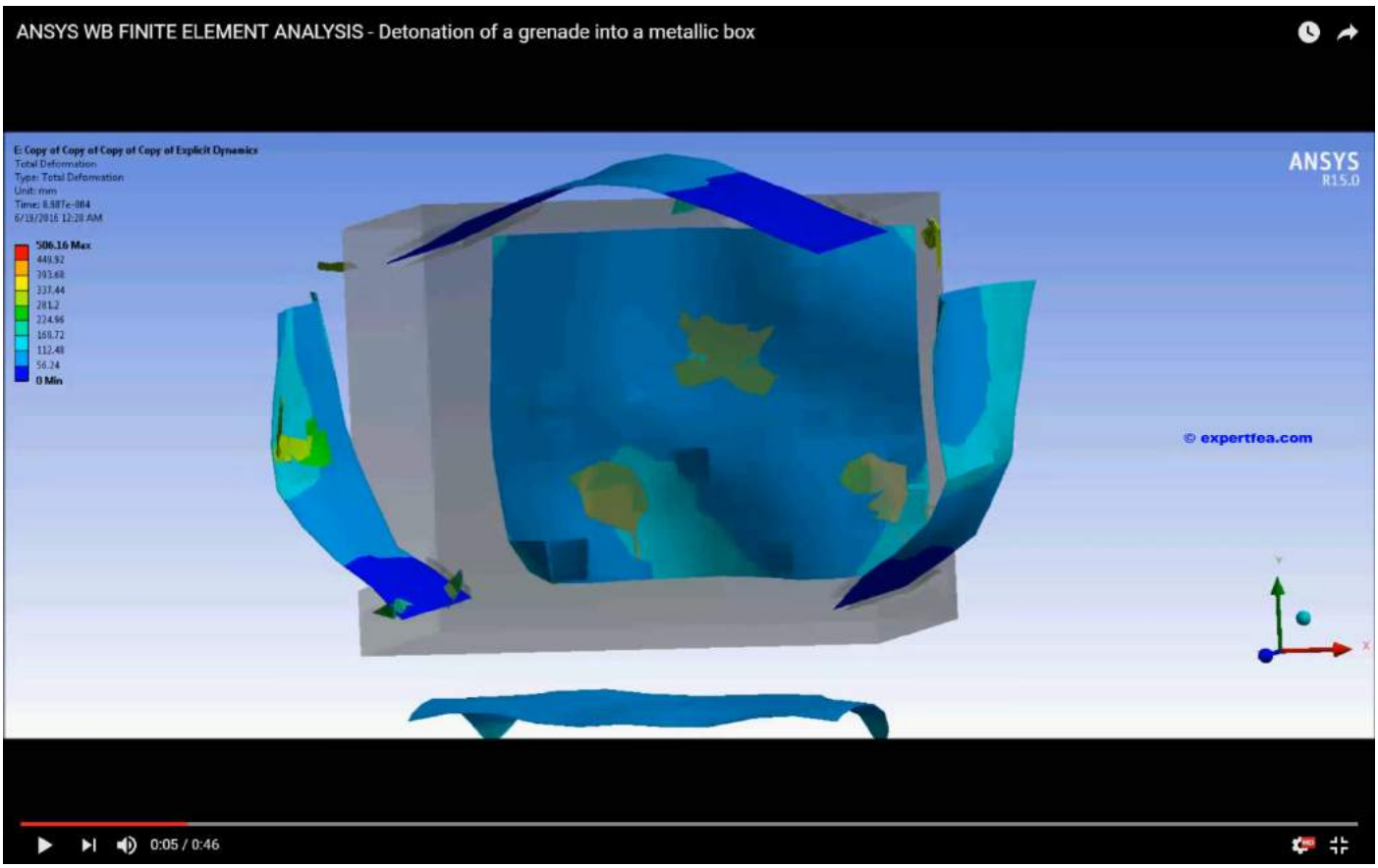
Further homework:

- decrease the mesh sizings [redacted], solve, draw the conclusions

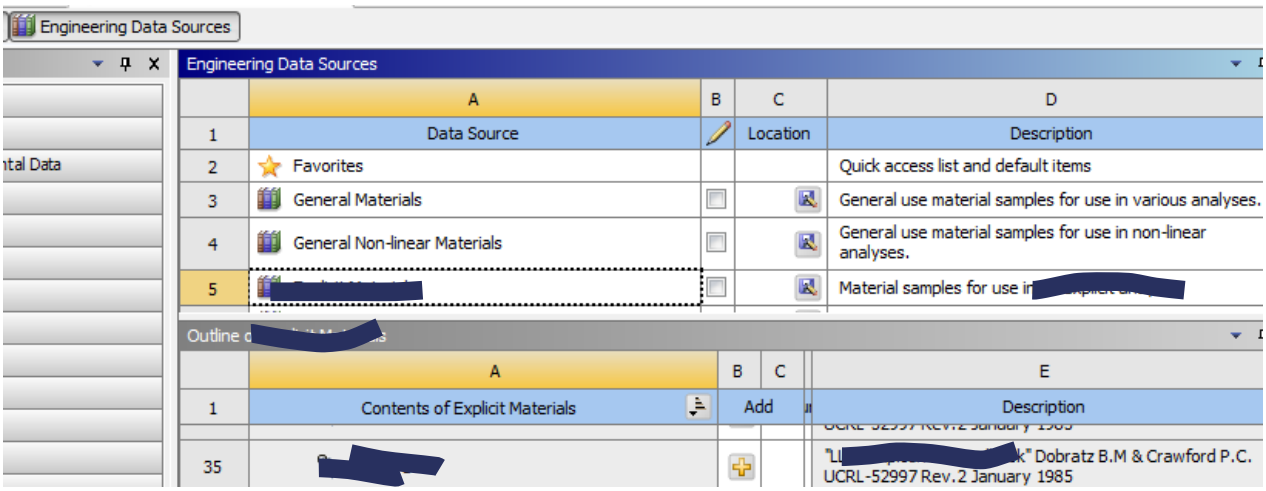
- apply [redacted] contacts between C section part and the supporting plates, solve, draw the conclusions

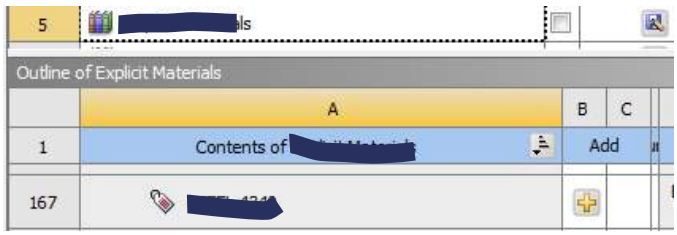
- delete [redacted] displacement of the C section edges and let them move freely, solve, draw the conclusions

CASE 23: ANSYS WB FINITE ELEMENT ANALYSIS - Explicit Dynamics Detonation of a grenade into a metallic box



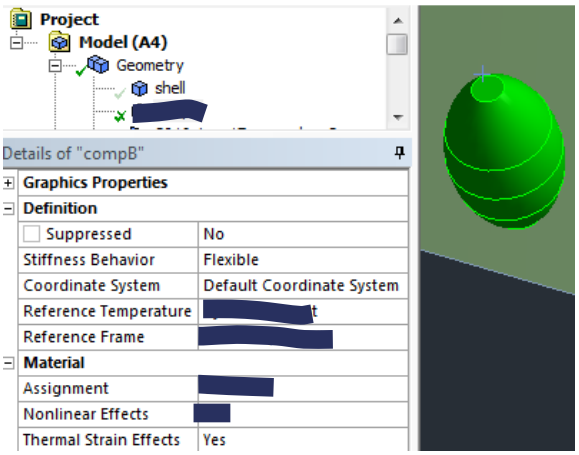
Engineering Data: From Explicit Materials, [redacted]



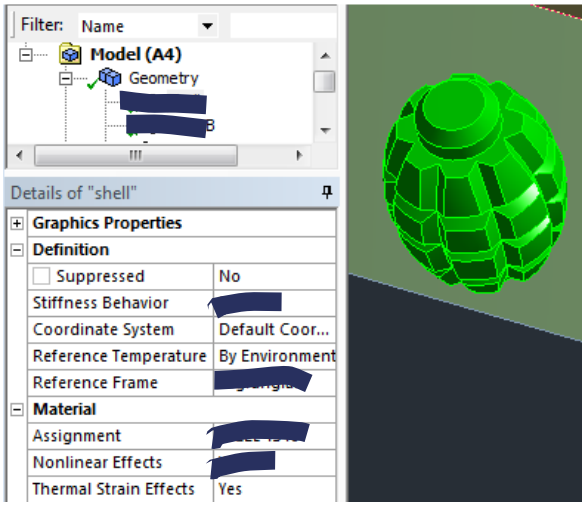


Geometry:2016_ju[redacted]

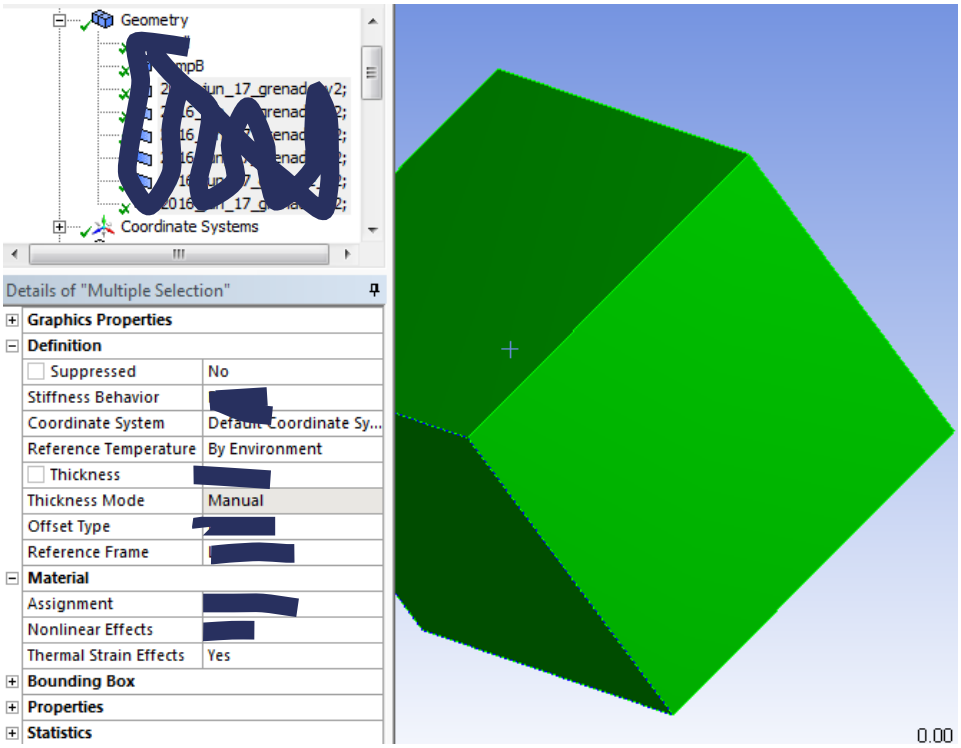
Hide the bodies to see the inner explosive part, green here and [redacted] Make Refer[redacted]me = [redacted], which applies to gaseous and liquid bodies. R[redacted] the Eulerian frame when doing future similar FEA!



To the grenade body, green here, assig[redacted].

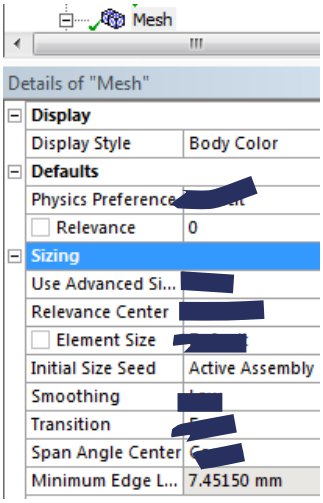


To the outer shell bodies, green here, appl[redacted]d Thickness = 1 mm.

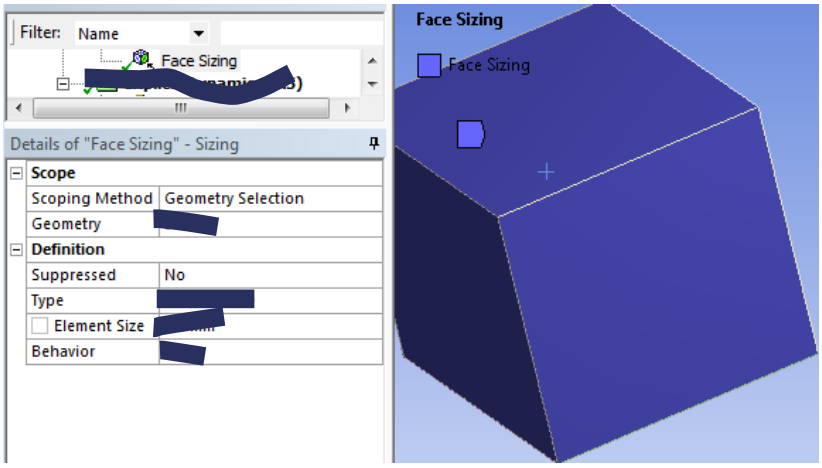


Connections: Suppress the Contacts.

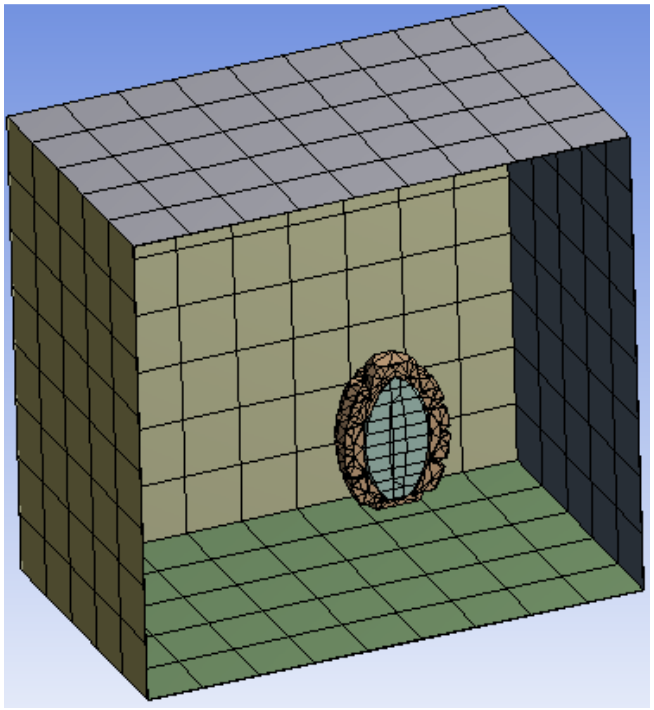
Mesh: Assign these details.



On all cube faces, blue here, apply this mesh sizing.

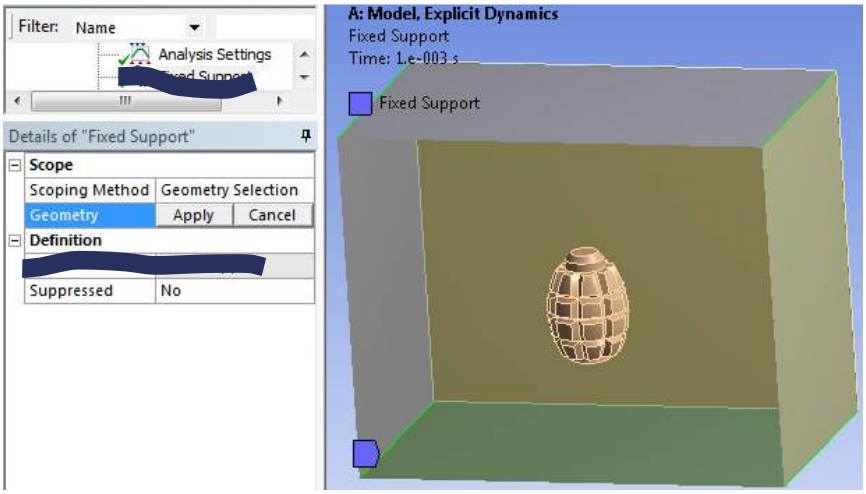


In section view, the mesh should look like here.



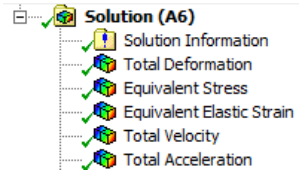
Analysis settings: Carefully apply these details. [redacted]ls dictates the meshing of the virtual body (e.g.: surrounding air); tune these values in connection with how powerful is your workstation.

Fix these [redacted]s, seen green here.

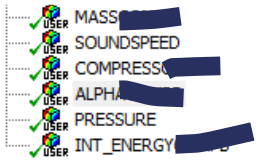


From the Loads toolbar, insert a [redacted]. Location = scope the inner explosive part as good as you can (hide the unimportant bodies), then change the coordinates to have the point in the middle of [redacted] as possible, as indicated here by the red sphere. [redacted] that the explosion begins immediately after the solving; applying any positive value will delay the explosion accordingly. The [redacted] load will be active in the corresponding toolbar only when an explosive material will be present in the FEA (implemented from [redacted] Data)!

Solution: Insert these default items, for all bodies.



After the FEA was solved, click Solution, then Worksheet button and apply these default items, for all bodies.



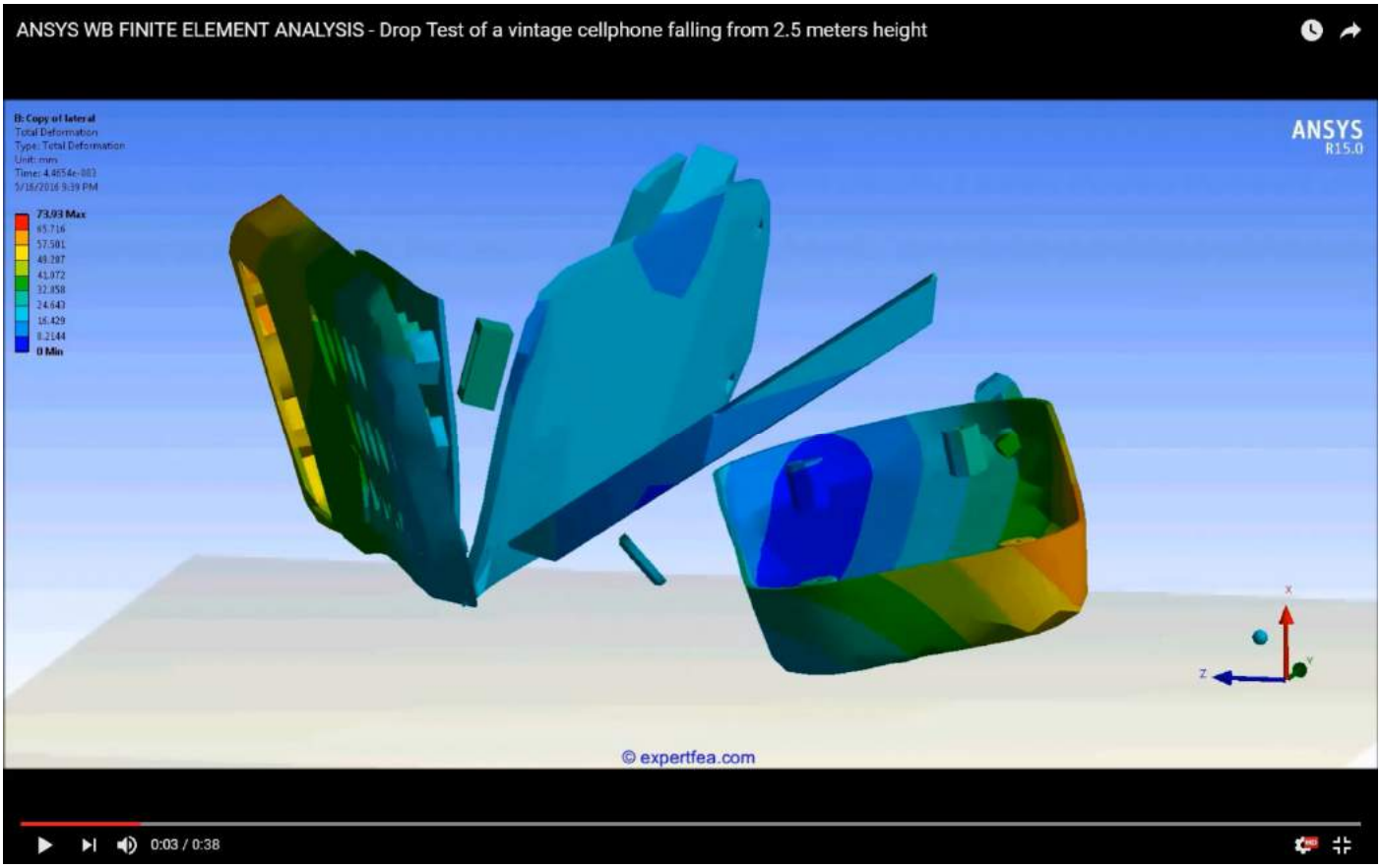
Further homework:

- decrease the mesh sizings in half, solve, draw the conclusions

- for Euler Domain Controls make [redacted] 2 instead of 1.2, solve, draw the conclusions

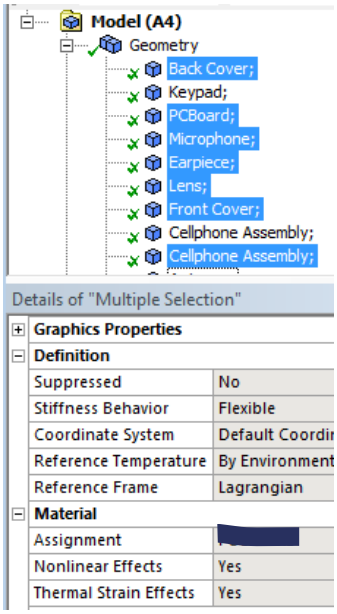
- change the [redacted] to another material, solve, draw the conclusions

CASE 24: ANSYS WB FINITE ELEMENT ANALYSIS - Explicit Dynamics Drop Test of a vintage cell phone falling from 2.5 meters height

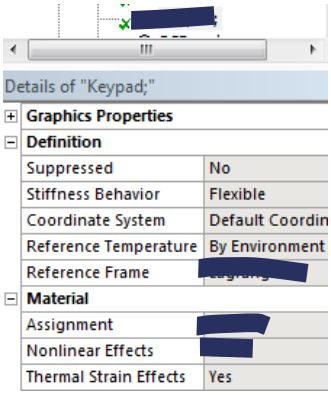


Geometry:2016_m[redacted]_v1.x_t

Assign[redacted]B material to the parts selected here with blue.

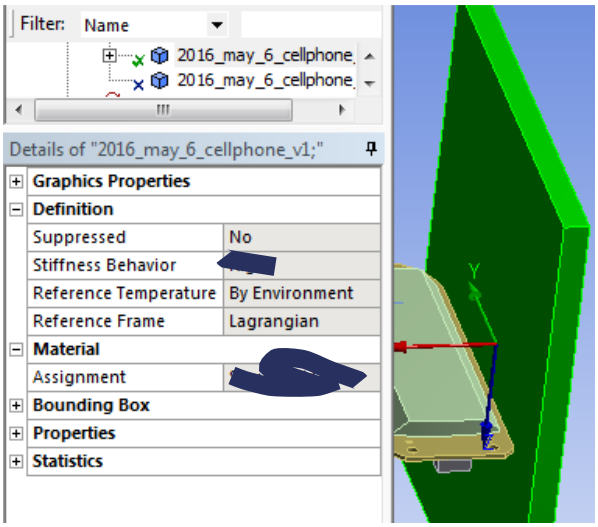


Make the Keypad [redacted]



Apply [redacted] to the battery and antenna, green here.

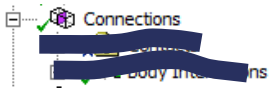
Make the floor, green here, as [redacted] Suppress the last body.



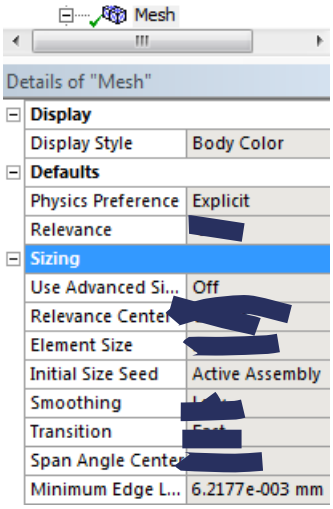
While on the Model branch, insert [redacted] with High Behavior and obtain similar count values.

[redacted]

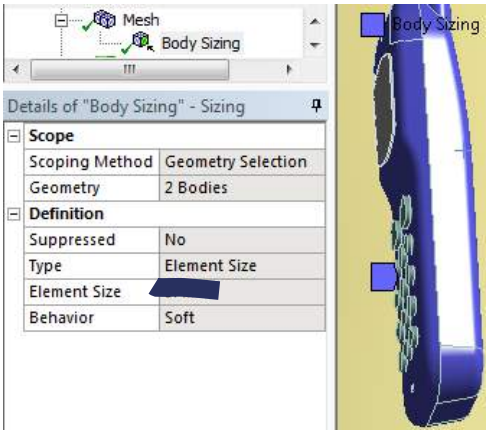
Connections: Suppress the [redacted]



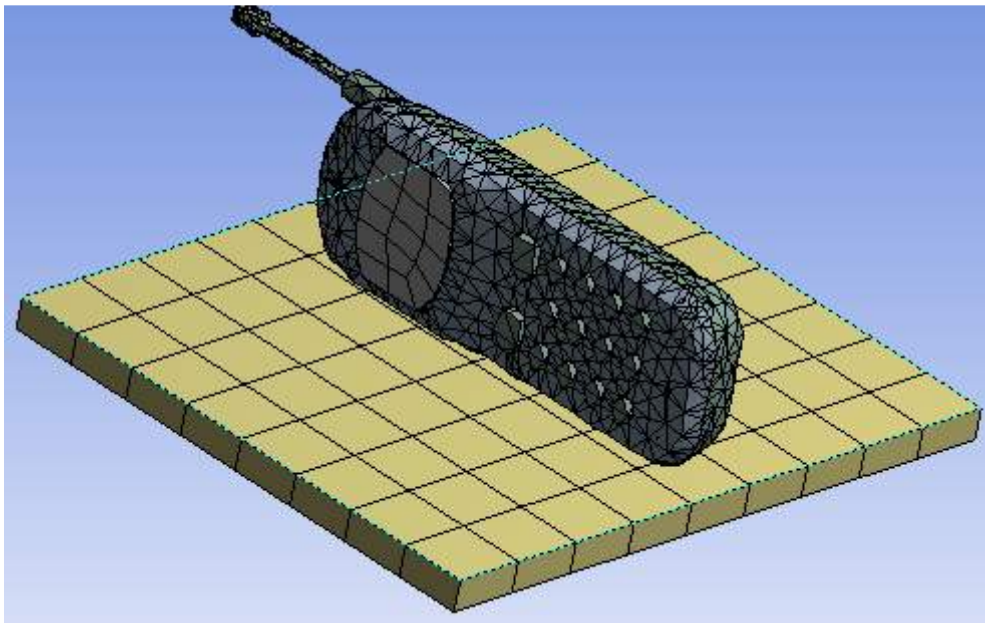
Mesh: Assign these details.



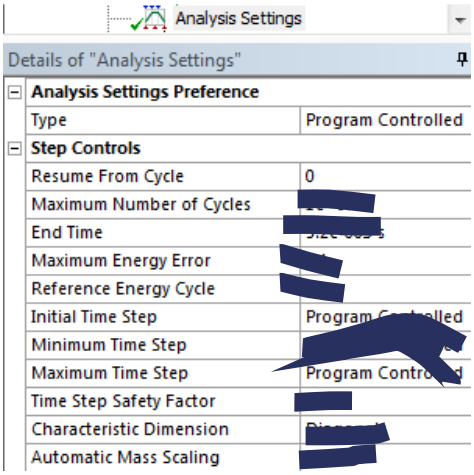
For these 2 housing parts, seen here in blue, apply this sizing.



Properly made, the mesh should look the same with this one.

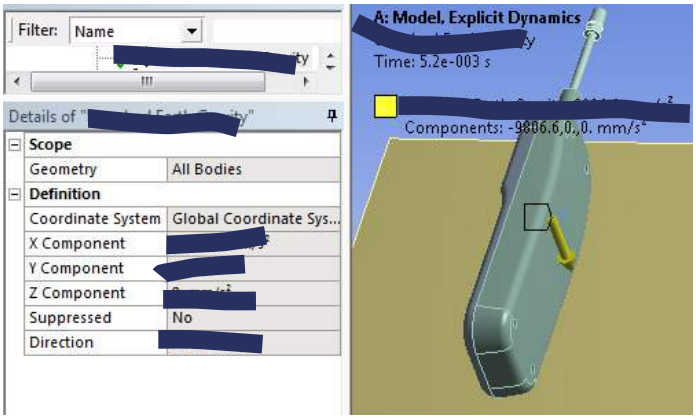


Analysis Settings: Insert these details.

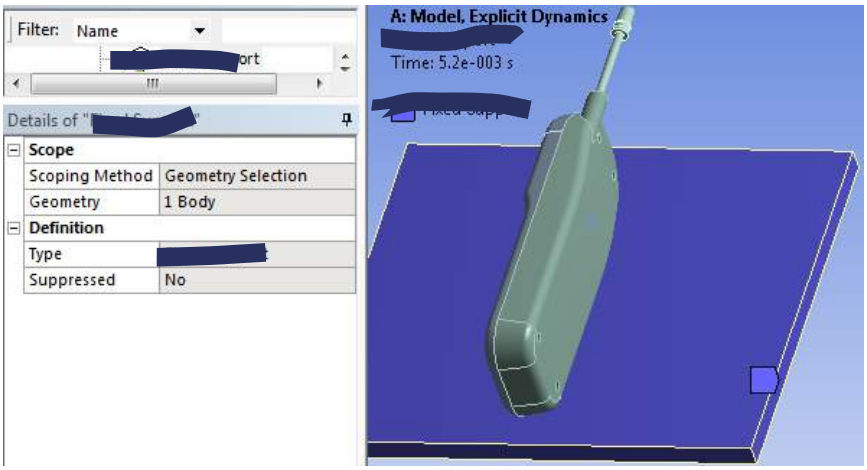


Initial Conditions: Apply this [redacted] to all parts, blue here, excepting the floor.

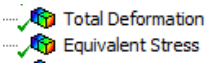
Environment toolbar: From the Inertial toolbar ass[redacted]een here.



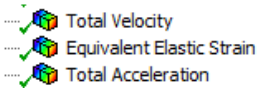
Fix the floor body, seen blue here.



Solution: Apply these default items only on the phone parts.



Insert these default items on all the bodies.



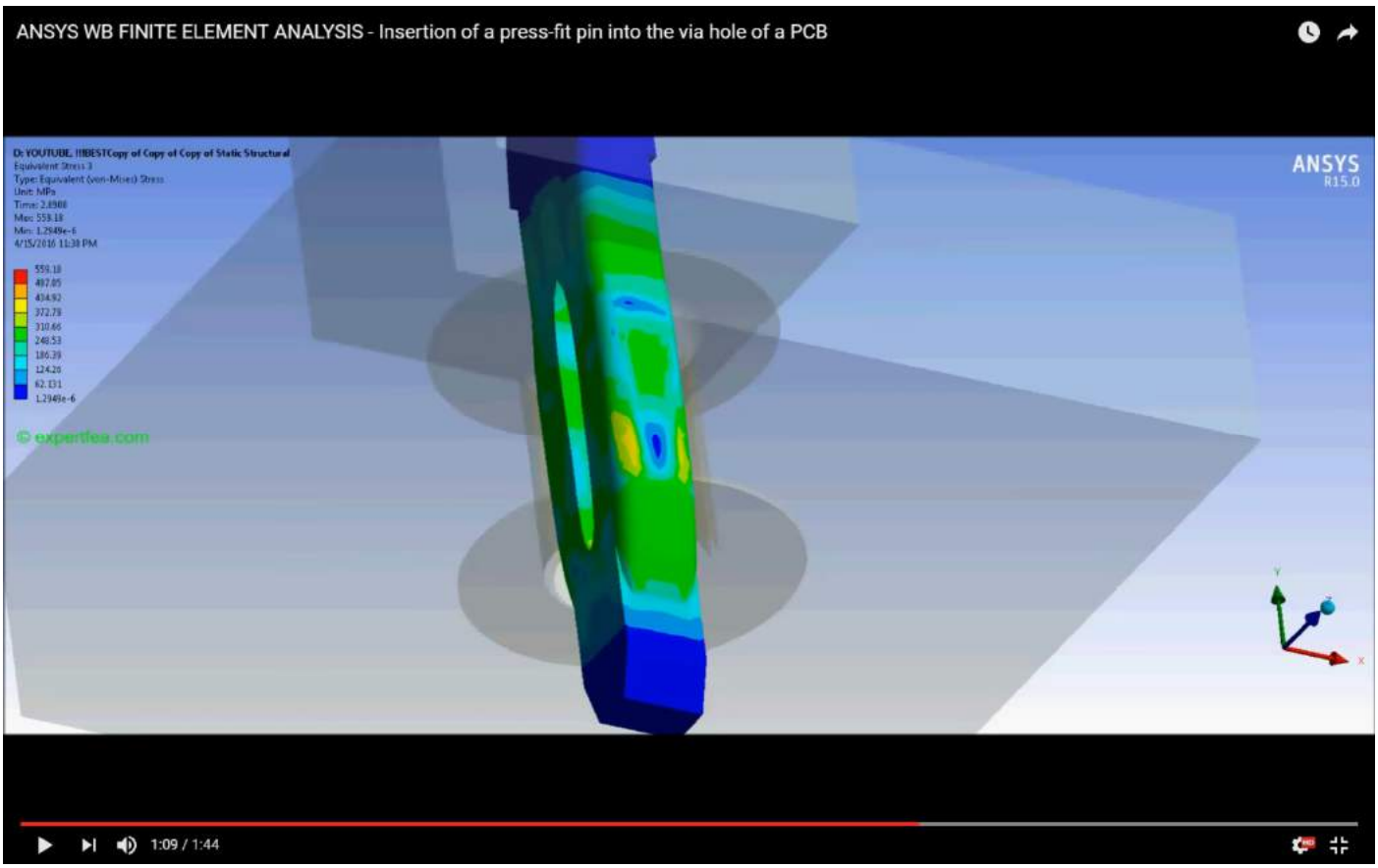
Further homework:

- delete the Virtual [redacted], solve, draw the conclusions

- decrease the falling [redacted], solve, draw the conclusions

- change the [redacted] to any other material, solve, draw the conclusions

CASE 25: ANSYS WB FINITE ELEMENT ANALYSIS - Static Structural Insertion of a press-fit pin into the via hole of a PCB



Engineering Data: Duplicate a [redacted] and create a Bronze with these details.

Outline of Schematic A2: Engineering Data

	A	B	D
1	Contents of Engineering Data		Description
2	Material		
3	Bronze		

Properties of Outline Row 3: Bronze

	A	B	C
1	Property	Value	Unit
2	Density	[redacted]	kg m^-3
3	Isotropic Elasticity		
4	Derive from	Young's Modu...	
5	Young's Modulus	[redacted]	Pa
6	Poisson's Ratio	[redacted]	
7	Bulk Modulus	1.0773E+11	Pa
8	Shear Modulus	3.859E+10	Pa
9	Field Variables		
10	Temperature	Yes	
11	Shear Angle	No	
12	Degradation Factor	No	
13	Bilinear Isotropic Hardening		
16	Tensile Ultimate Strength	[redacted]	Pa

Duplicate a [redacted] and create [redacted] material with these details.

Duplicate a Polyethylene and create a [redacted] material with these details.

Outline of Schematic A2: Engineering Data

	A	B	D
1	Contents of Engineering Data		Description
7			

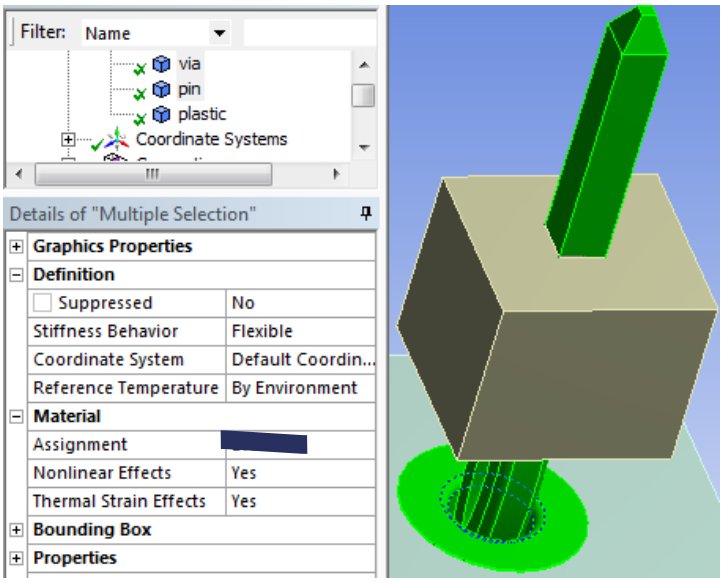
Properties of Outline Row 7

	A	B	C
1	Property	Value	Unit
2	Density		kg m^-3
3	Isotropic Secant Coefficient of Thermal Expansion		
4	Coefficient of Thermal Expansion		C^-1
5	Reference Temperature		C
6	Isotropic Elasticity		
7	Derive from	Young's Modu...	
8	Young's Modulus		Pa
9	Poisson's Ratio		
10	Bulk Modulus	1.55E+10	Pa
11	Shear Modulus	3.3214E+09	Pa
12	Field Variables		
13	Temperature		
14	Shear Angle		
15	Degradation Factor		
16	Tensile Yield Strength		Pa
17	Tensile Ultimate Strength		Pa

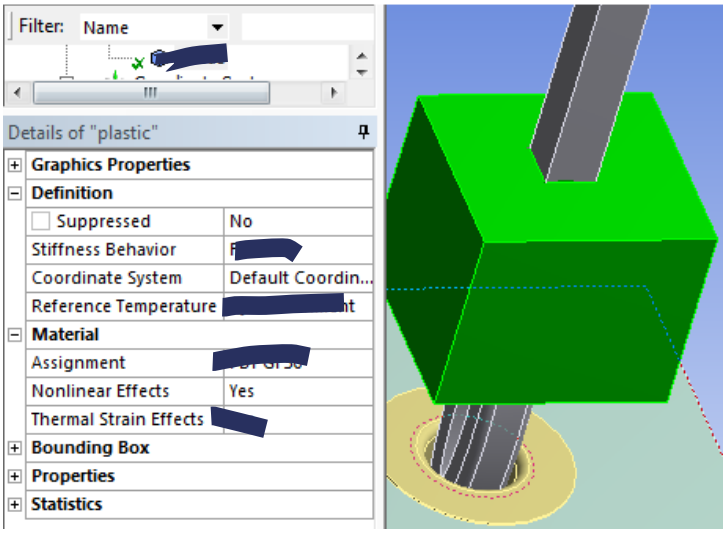
Geometry: Assign [redacted]aterial on the PCB part, green here.

[redacted]

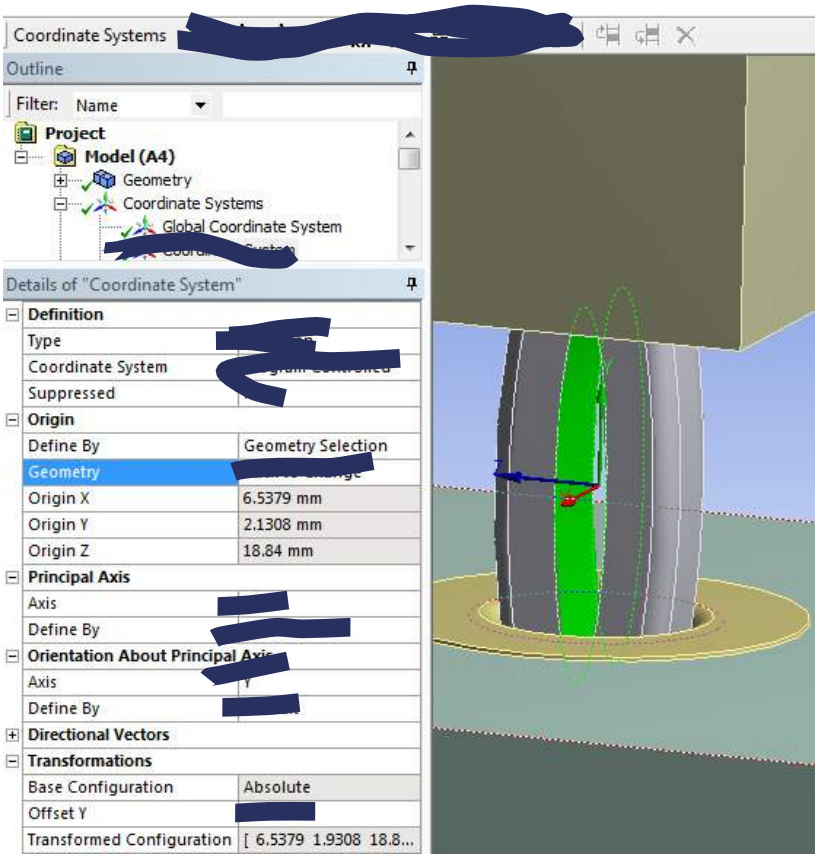
Assig[redacted]material on the via and pin, green here.



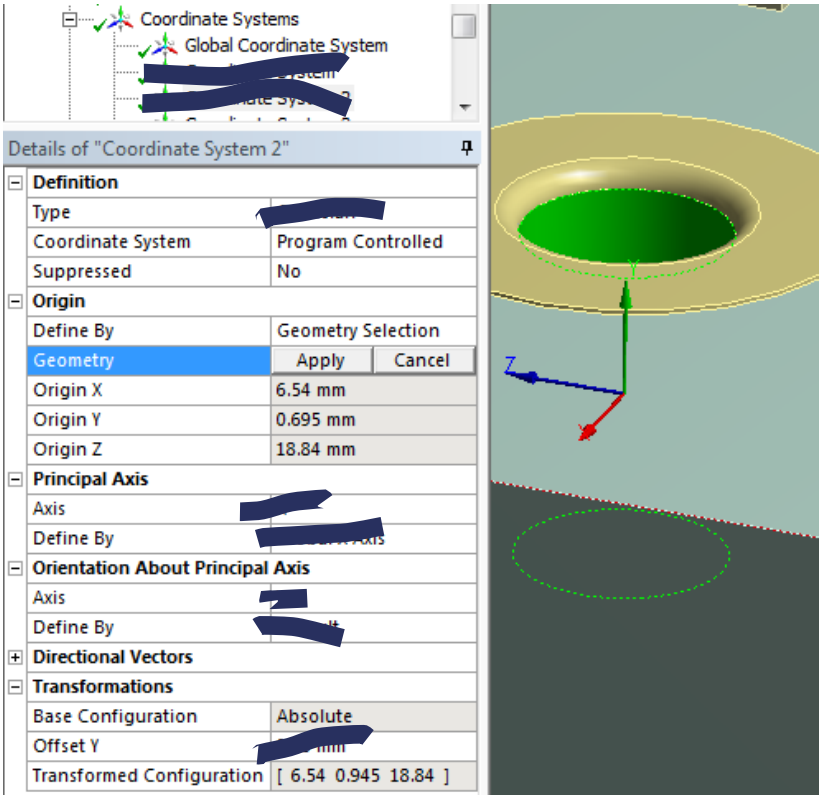
Assign PBT GF30 material on the via and pin, green here.



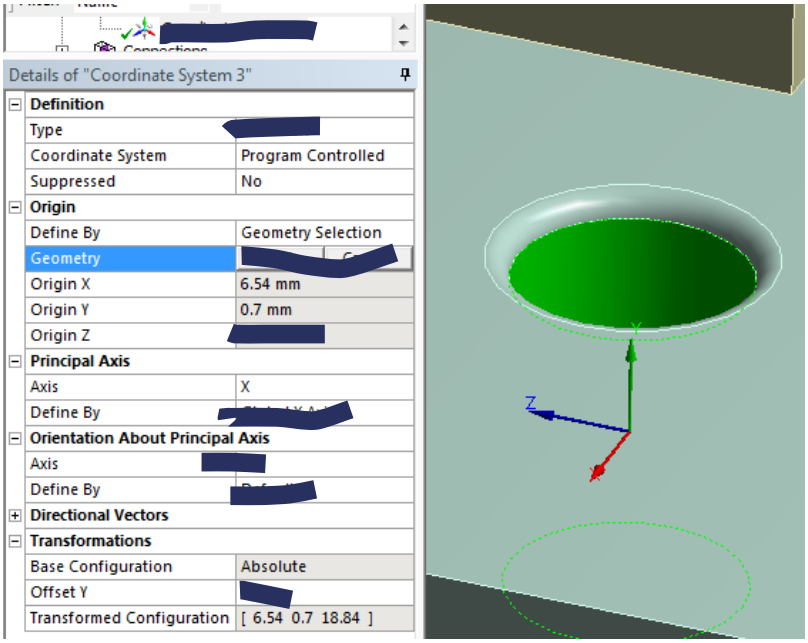
Coordinate Systems: Apply a [redacted] System on the inner green face of the pin, as seen here. [redacted] on the Y axis from the ⌂Y button.



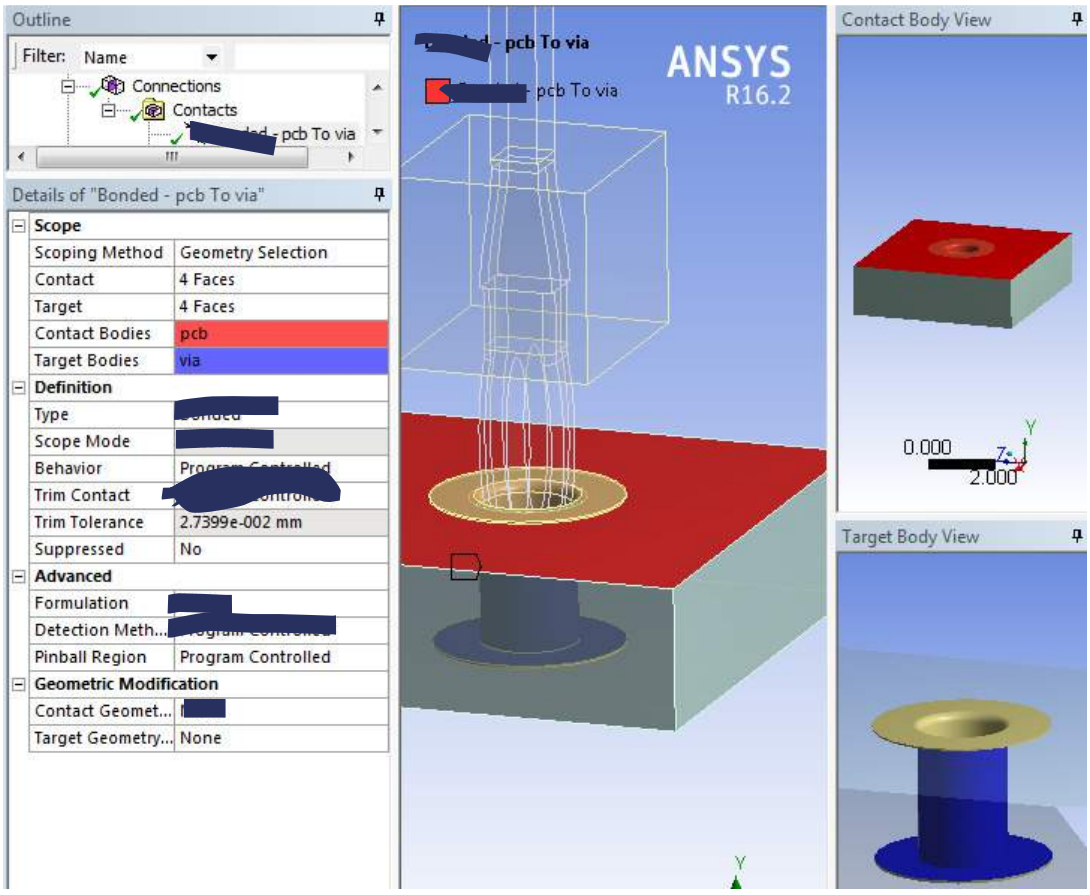
Apply a Coordinate System on the inner green face of the via, as seen here. Apply an offset on the Y axis from the ⌂Y button.



Apply a Coordinate System on the inner green face of the via hole in PCB, as seen here. No offset on the Y axis.

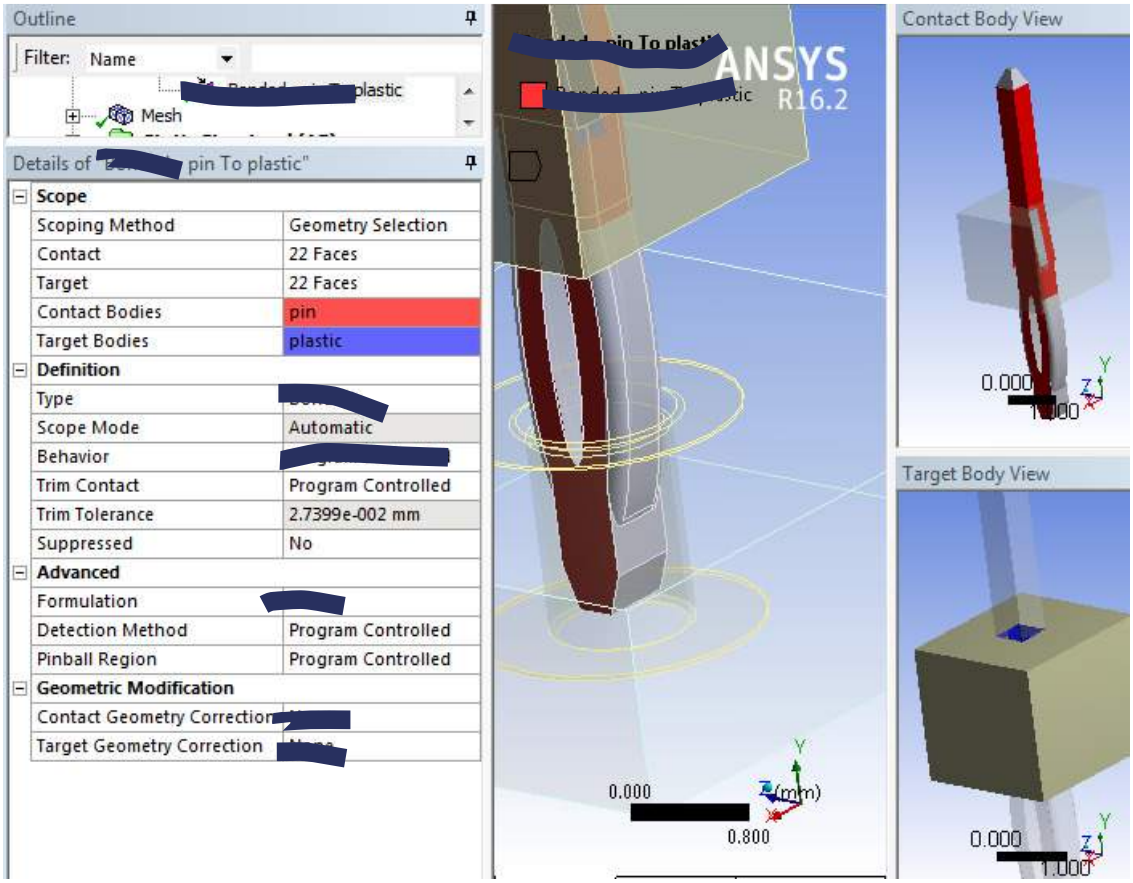


Connections, Contacts: Create a bonded contact between the via body and its hole in the PCB.

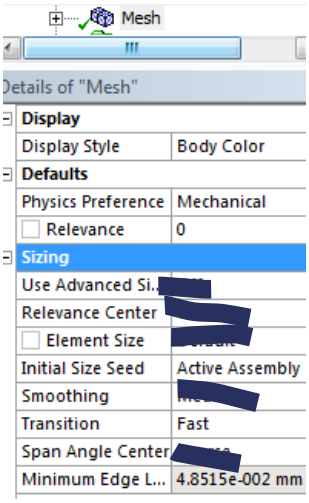


Create a Frictionless contact between pin and via, with these details. Carefully select these faces, differently colored in red and blue, for the Contact and Target side.

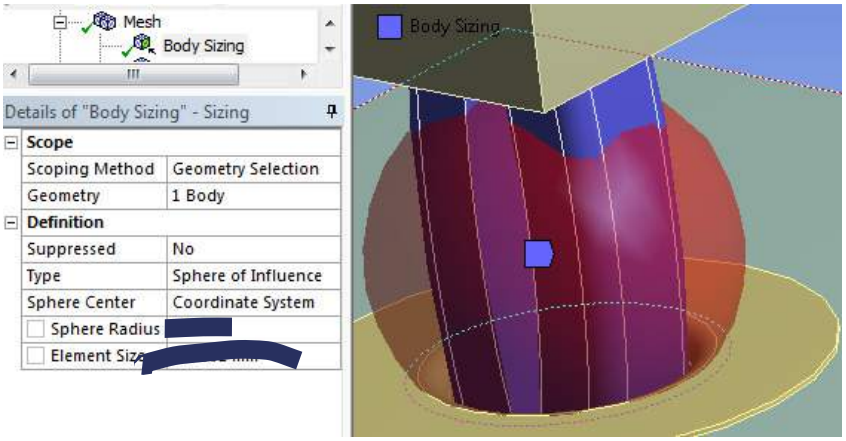
Create a bonded contact between the pin and the surrounding plastic.



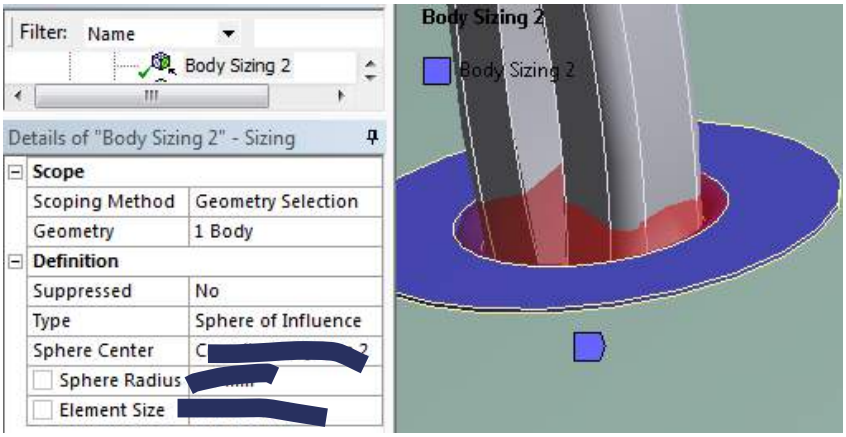
Mesh: Assign these details.



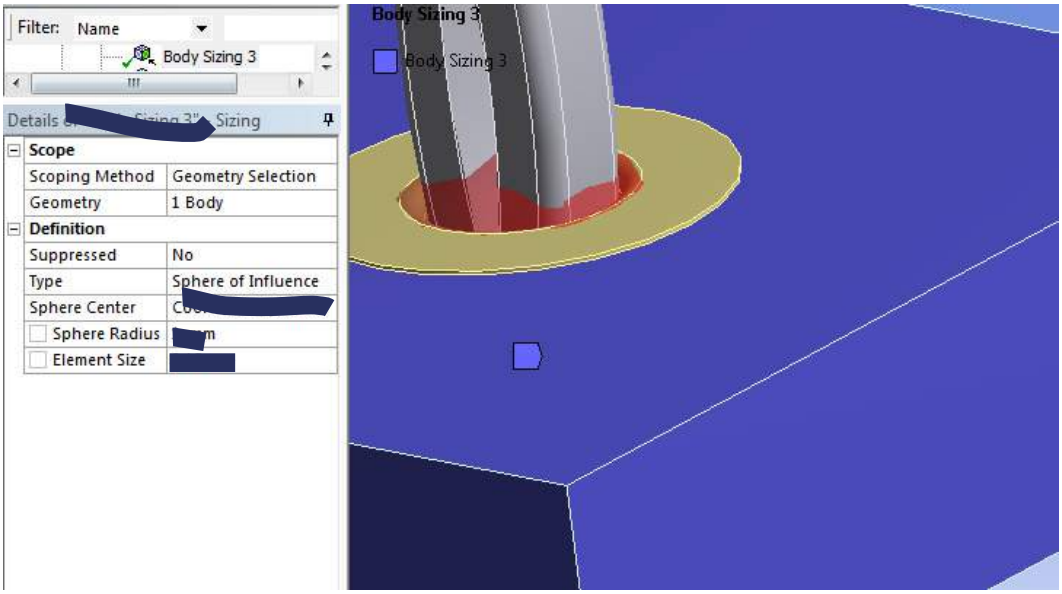
Insert this mesh sizing.



Apply this mesh sizing.

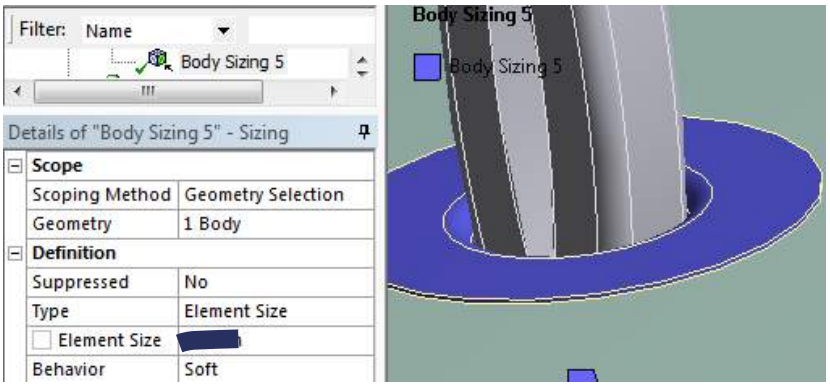


Create this mesh sizing.

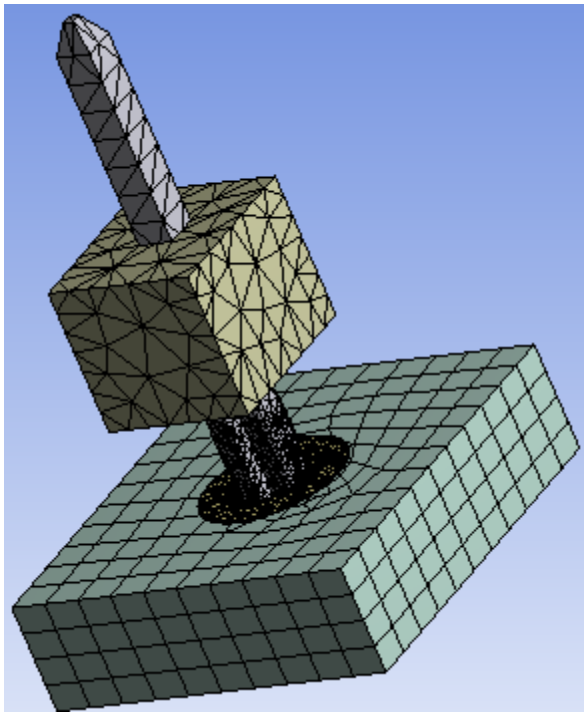


Assign this sizing on the PCB part.

Make the mesh sizing of the via part as seen here.



Correctly made, the mesh should look similar to this one.



Analysis Settings: Assign these details to the 1st step.

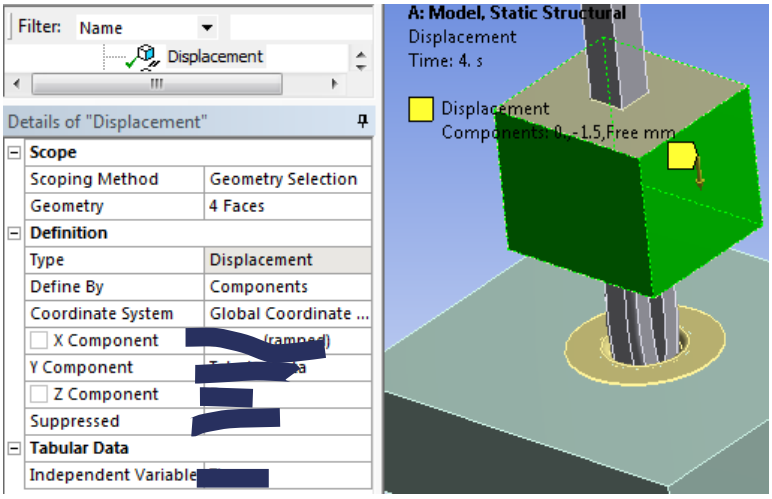
Analysis Settings

Details of "Analysis Settings"

Step Controls	
Number Of Steps	[redacted]
Current Step Number	[redacted]
Step End Time	[redacted]
Auto Time Stepping	[redacted]
Define By	Time
Initial Time Step	[redacted]02 s
Minimum Time Step	[redacted]-002 s
Maximum Time Step	[redacted]
Solver Controls	
Solver Type	Program Controlled
Weak Springs	[redacted]
Solver Pivot Checking	Program Controlled
Large Deflection	[redacted]
Inertia Relief	Off
Restart Controls	
Nonlinear Controls	
Newton-Raphson O...	Program Controlled
Force Convergence	Program Controlled
Moment Convergence	[redacted]
Displacement Conve...	Program Controlled
Rotation Convergen...	[redacted]
Line Search	Program Controlled
Stabilization	[redacted]
--Method	[redacted]
--Energy Dissipation...	[redacted]
--Activation For First...	[redacted]
--Stabilization Force...	[redacted]

Select the remaining steps and insert these details (Carry Over Timestep = On).

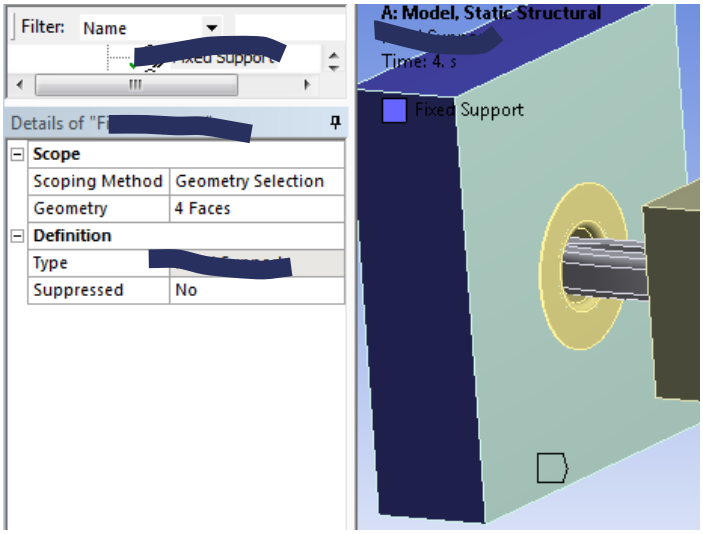
Environment toolbar: Create this displacement on these green faces of the plastic surrounding the pin.



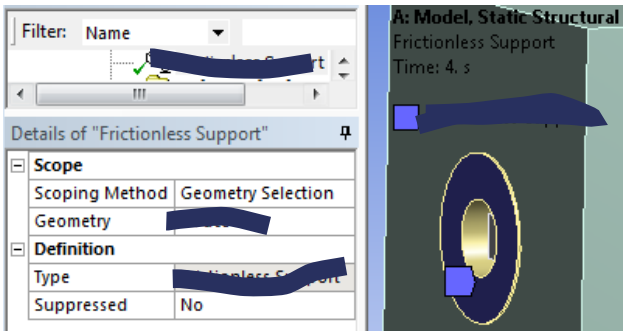
Tabular Data

	Steps	Time [s]	X [mm]	Y [mm]
1	1	0.	0	0
2	1	1.		
3	2	2.		
4	3	3.		
5	4	4.		-1.5

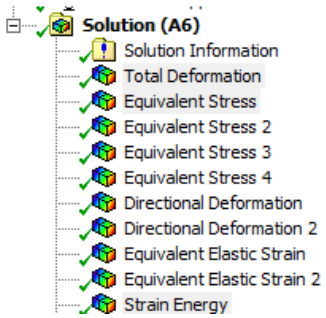
Fix these blue faces on the PCB part.



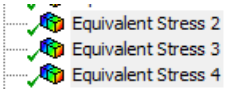
Apply a support on this blue face on the bottom of the via.



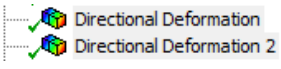
Solution: Insert Total Deformation, Equivalent Stress and Strain Energy as default, for all bodies.



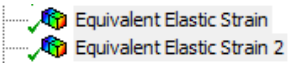
Apply Equivalent Stress separately for via, pin and PCB.



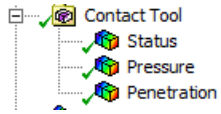
Create these items for via and PCB on X axis.



Apply these items for pin and via.



Assign this Contact Tool items only for the [redacted] contact.



Insert this probe for the Displacement condition.

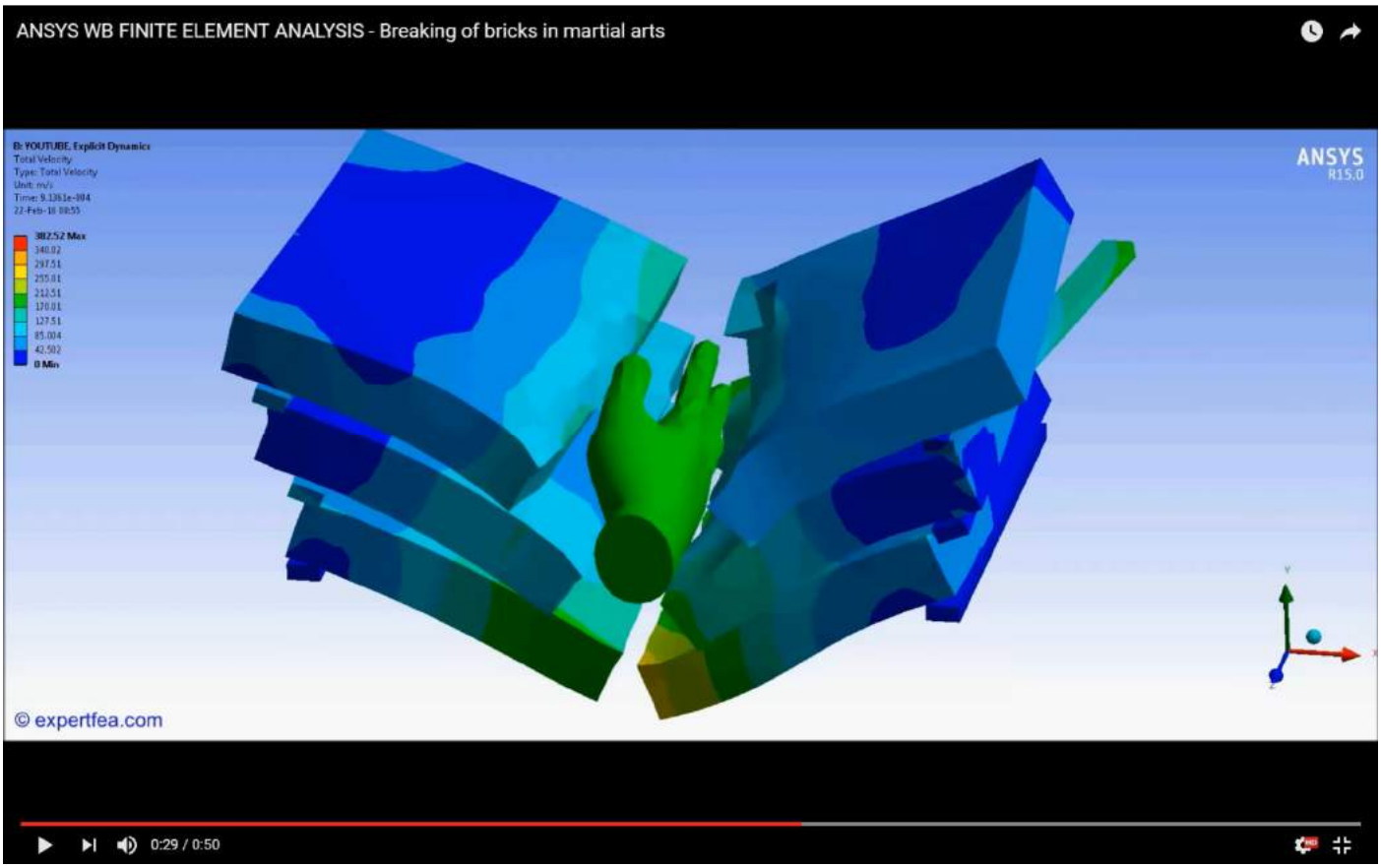
Further homework:

- change Frictionless contact to [redacted], solve, draw the conclusions

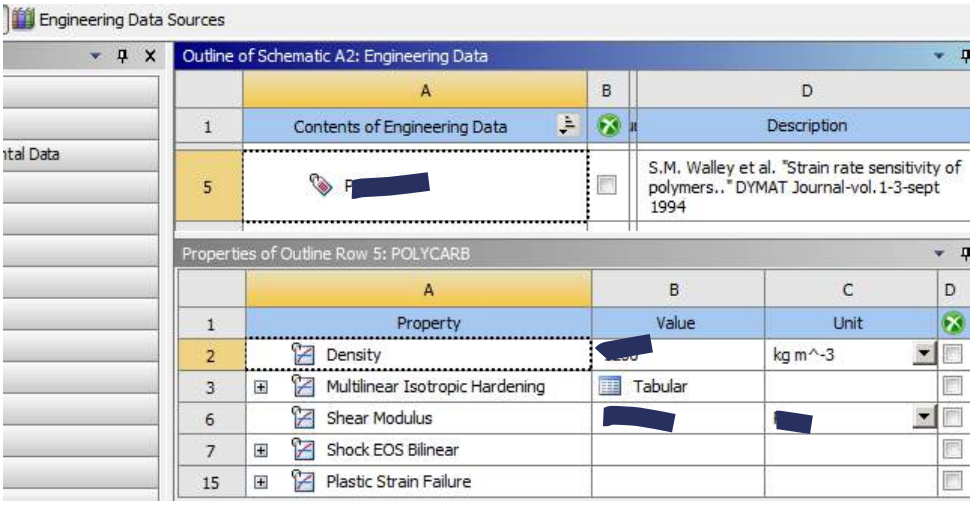
- make Minimum Ti[redacted], solve, draw the conclusions

- change the Bronze material to Copper [redacted], solve, draw the conclusions

CASE 26: ANSYS WB FINITE ELEMENT ANALYSIS - Explicit Dynamics Breaking of bricks in martial arts



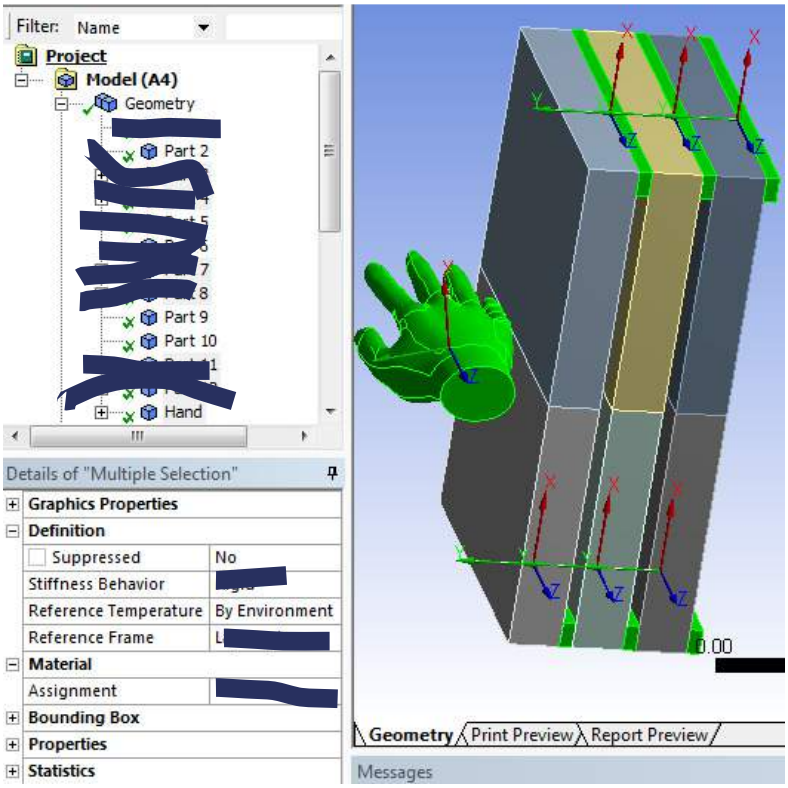
Engineering Data: From [redacted] materials, insert [redacted].



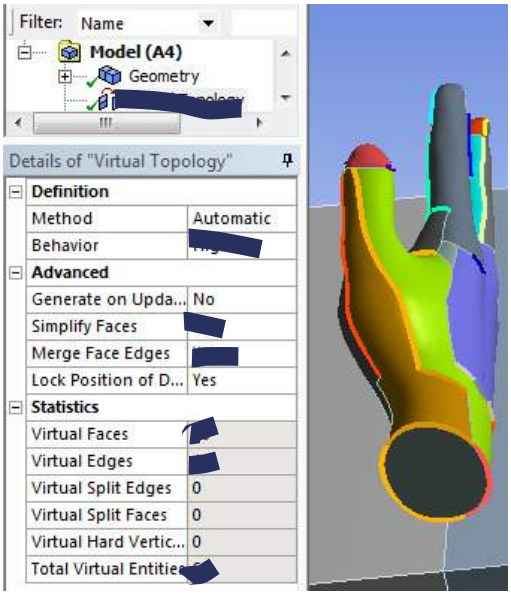
Geometry:2016_feb_14_[redacted]

Make all bricks of [redacted] and leave them with Stiffness Behavior [redacted]

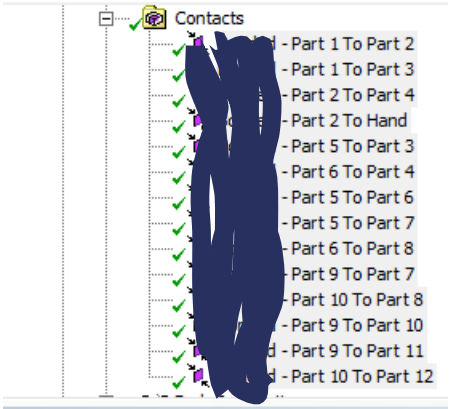
Make all the other parts as [redacted], including the hand. Their material is unimportant, because of the infinite stiffness.



Create a Virtual Topology similar to the one seen on this hand.



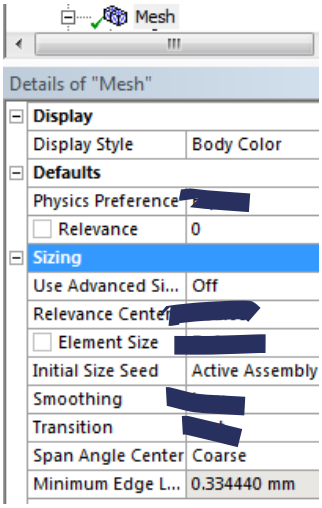
Connections: Assign these details to all the contacts.



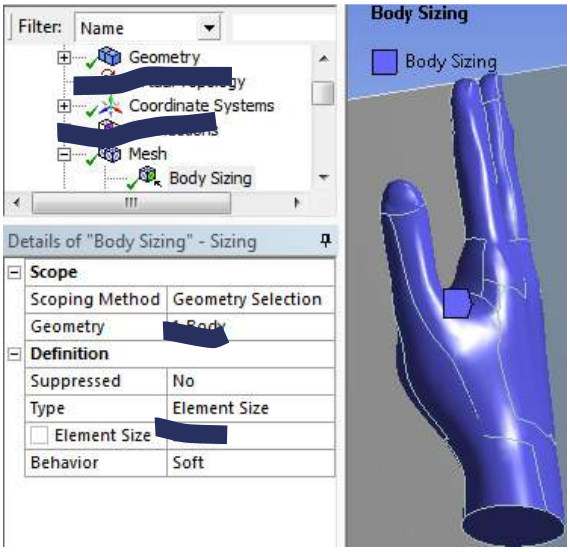
Details of "Multiple Selection"

Scope	
Scoping Method	Geometry Selection
Definition	
Type	
Scope Mode	Automatic
Behavior	Program Controlled
Trim Contact	
Trim Tolerance	1.3546 mm
Maximum Offset	1.e-004 mm
Breakable	
Normal Stress Limit	
Normal Stress Exponent	
Shear Stress Limit	
Shear Stress Exponent	
Suppressed	

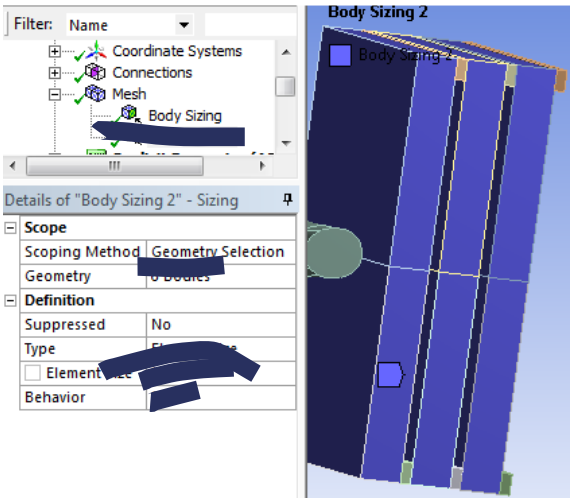
Mesh: Insert these details.



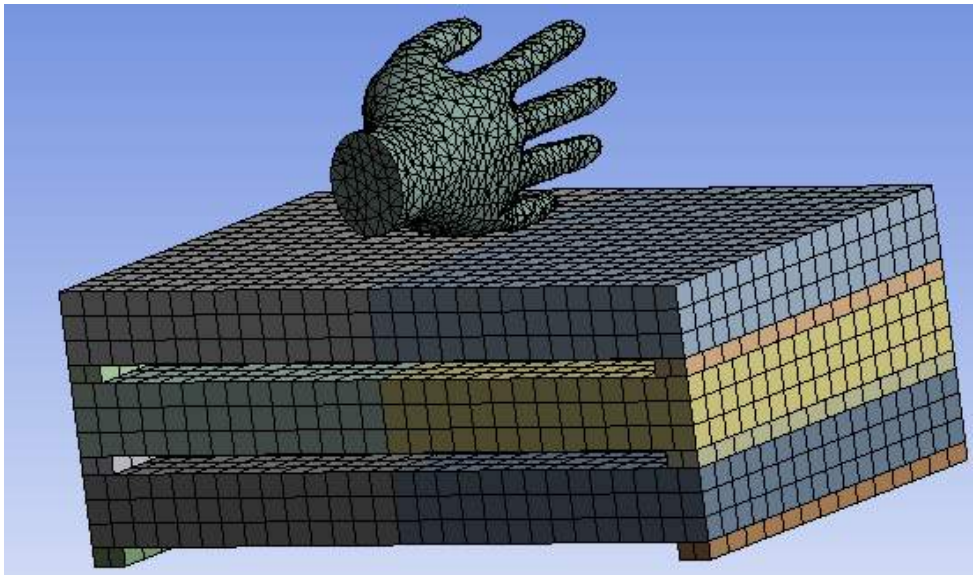
Assign this sizing on the hand, blue here.



Create this sizing for the bricks, blue here.



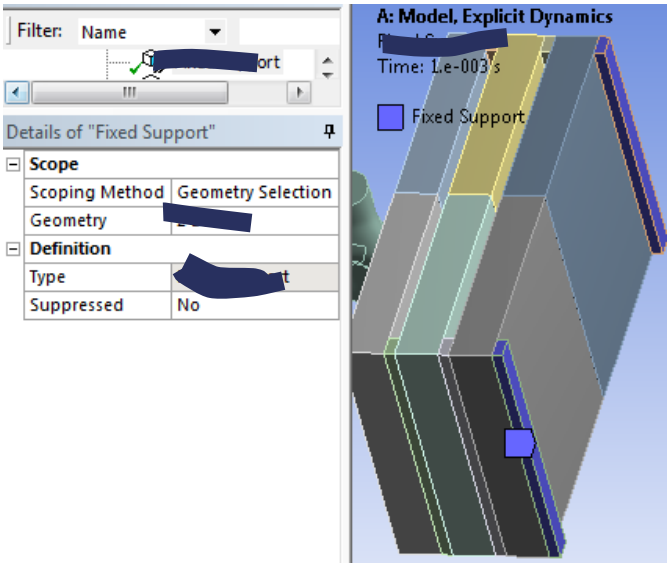
Correctly made, the mesh should look like here.



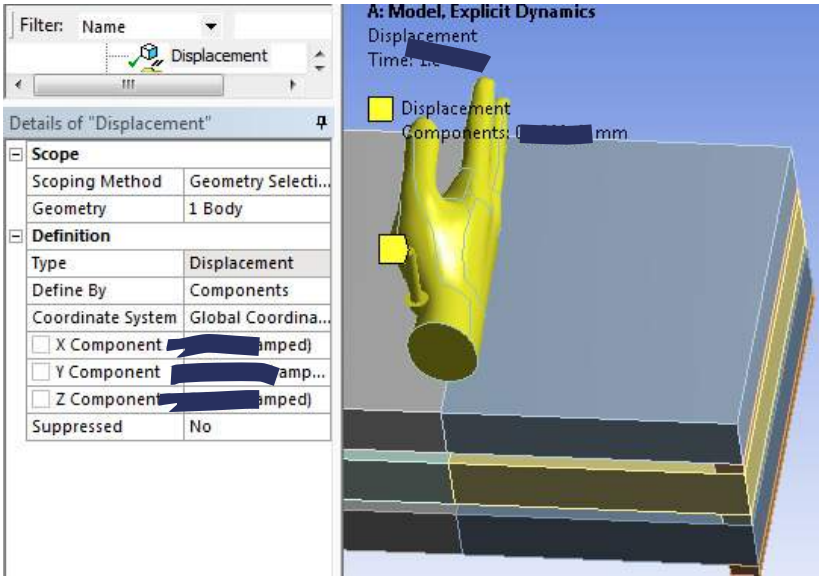
Analysis Settings: Insert these details.

[redacted]

[redacted] the 2 supporting parts on the bottom, blue here.

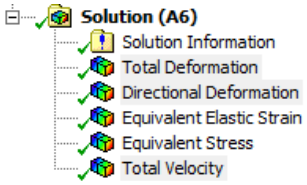


Create this Displacement on the hand.

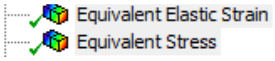


[redacted]

Solution: Create the grey items for all bodies. Y is the Directional Deformation axis.



Assign these items only for the bricks.



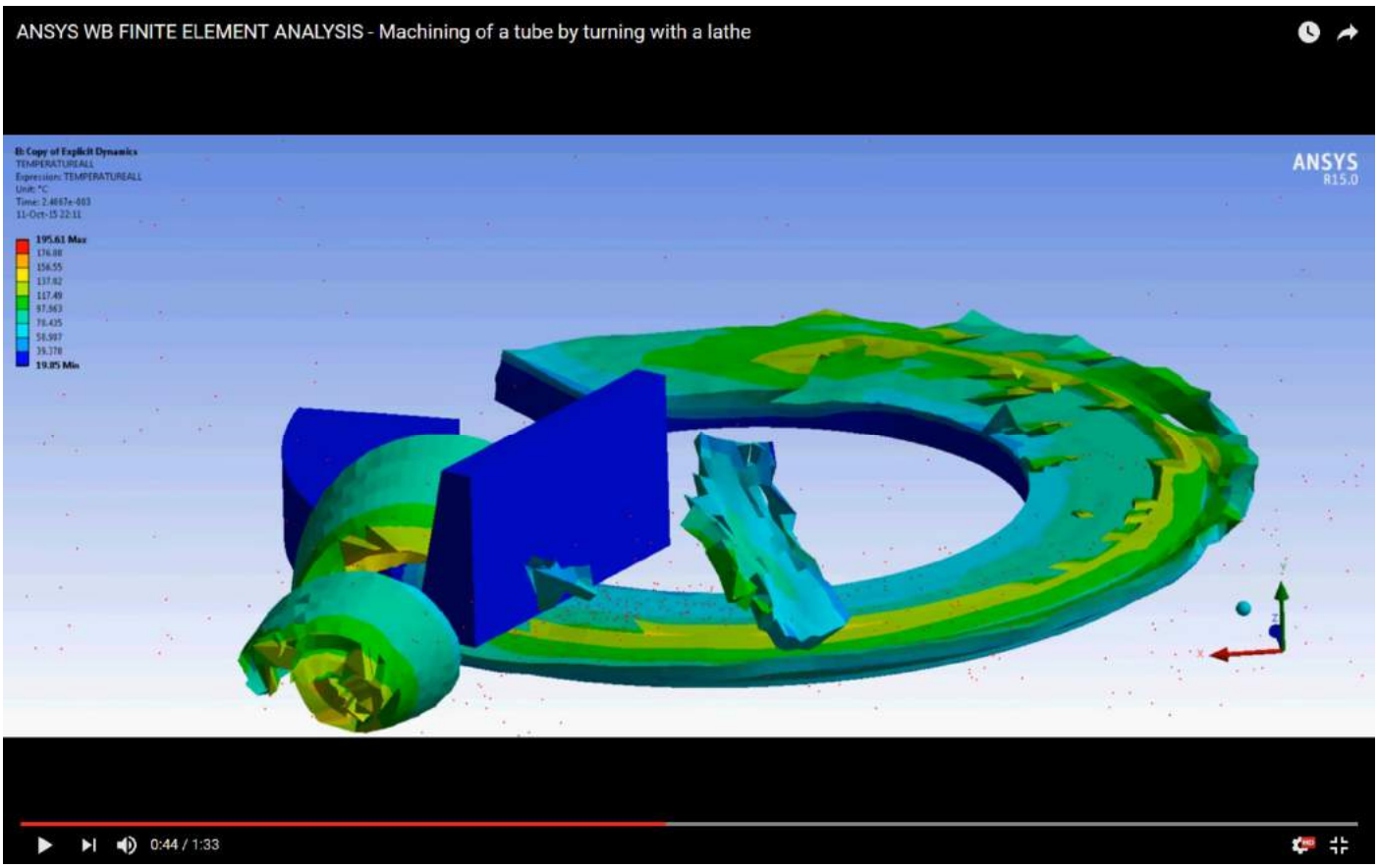
Further homework:

- in Connections, change Frictionless [redacted] to Frictional μ = [redacted], solve, draw the conclusions

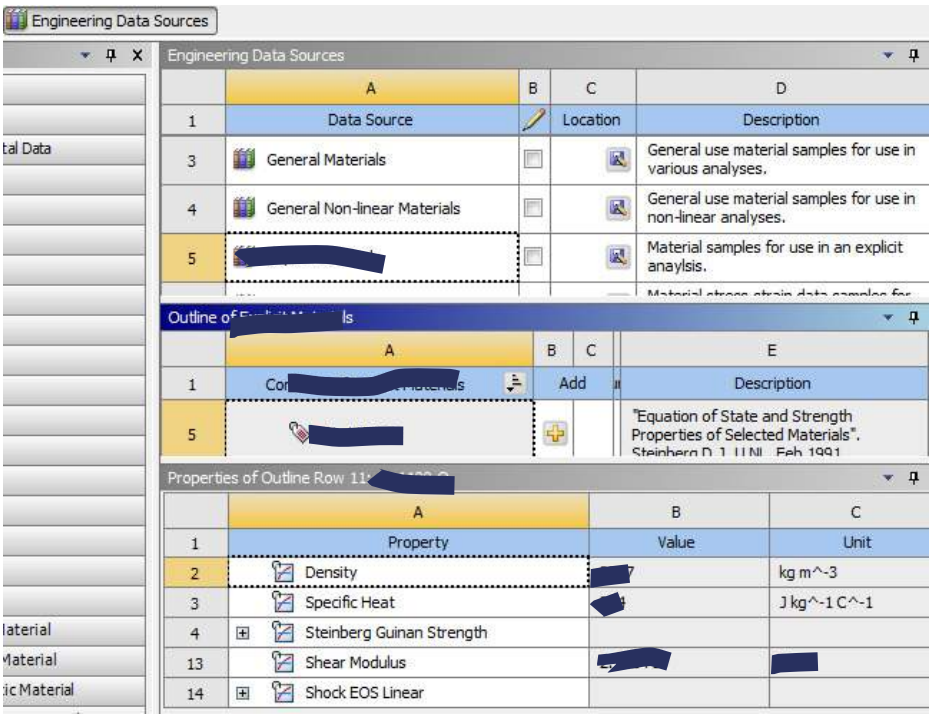
- make Maximum [redacted], solve, draw the conclusions

- in the contact details, decrease the stress limits to half their values, solve, draw the conclusions

CASE 28: ANSYS WB FINITE ELEMENT ANALYSIS - Explicit Dynamics Machining of a tube by turning with a lathe



Engineering Data: Assig[redacted]om the [redacted] Materials library.

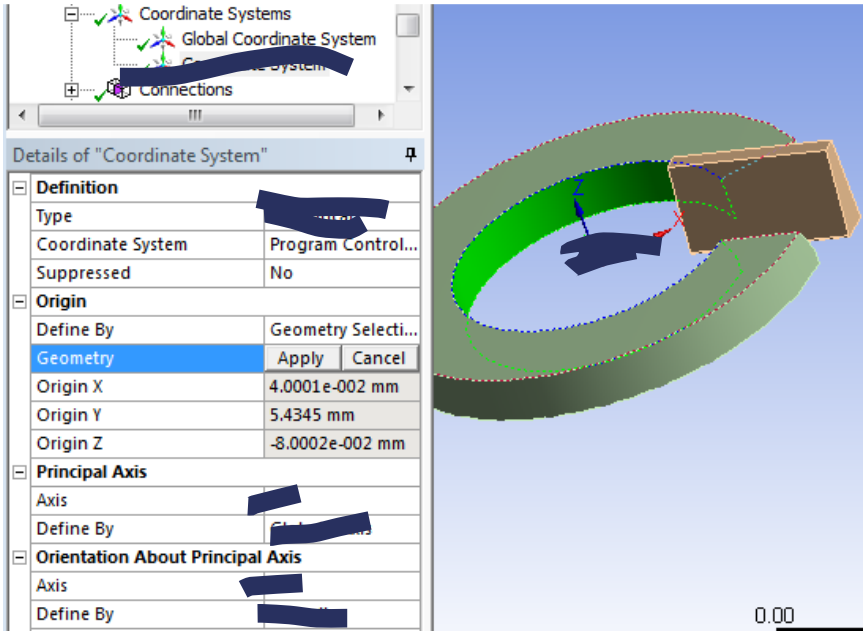


Geometry: 2015_oct_[redacted]

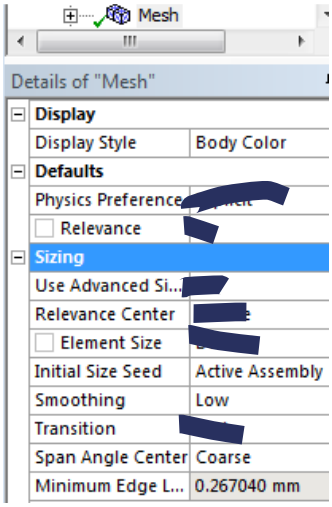
Assign the [redacted] material to the workpiece, green here. Make the tool's Stiffness [redacted].

[redacted]

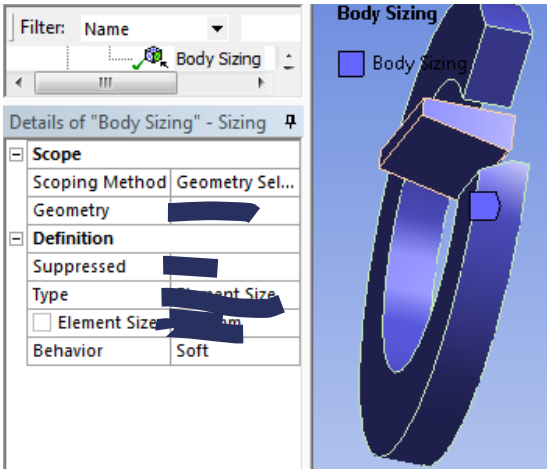
Coordinate Systems: Click the inside of the workpiece, green here, and assign these details ([redacted]). Make sure that the circular axis (green here) is the one that makes the tool rotate.



Mesh: Assign these details.

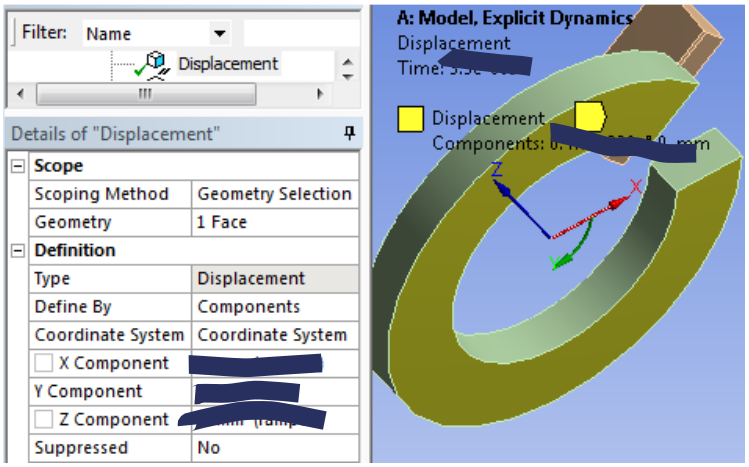


Insert this body sizing on both parts.



Analysis Settings: Insert these details.

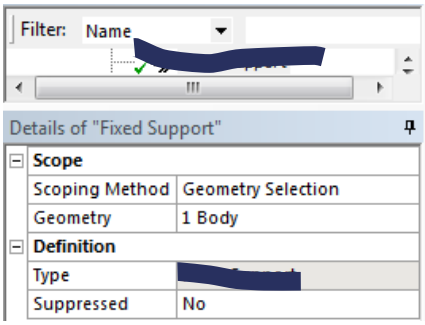
Environment toolbar: Create this displacement on the bottom face of the workpiece, yellow here.

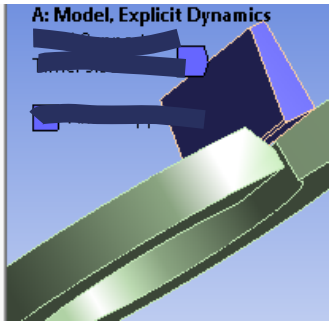


Tabular Data

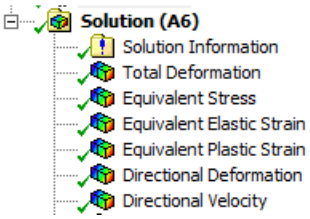
	Steps	Time [s]	X [mm]	Y [°]	Z [mm]
1	1	0.			
2	1				

Fix the tool, blue here.

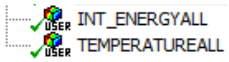




Solution: Create these items, as default, for all bodies.



After the FEA is solved, click Solution, then the worksheet button and select these items as default, for all parts.



Further homework:

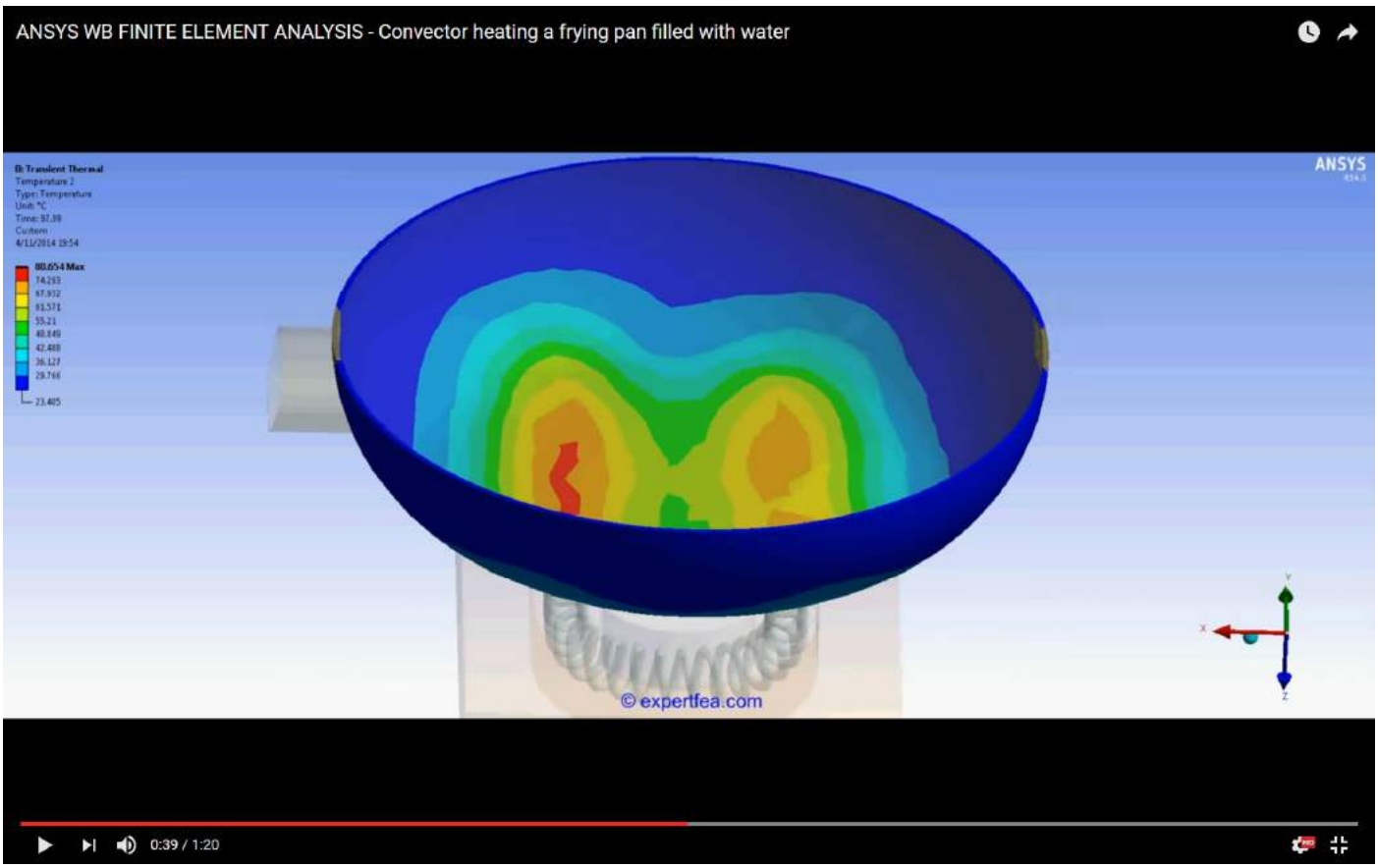
- in Connections, change F[redacted] to Frictio[redacted] 0.01, solve, draw the conclusions

- change the material of the sheet-metal part [redacted] solve, draw the conclusions

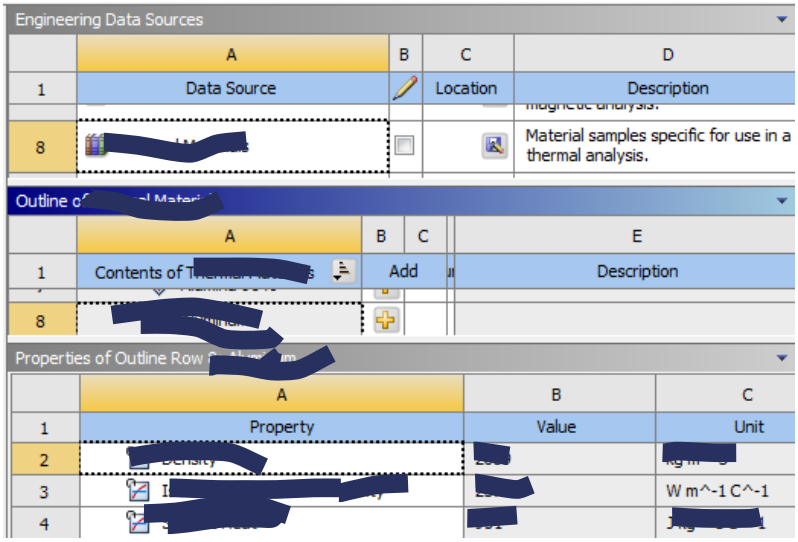
- change Stiffness Behavior of the tool part t[redacted] solve, draw the conclusions

- increase the mesh sizing to double their values, solve, draw the conclusions

CASE 30: ANSYS WB FINITE ELEMENT ANALYSIS - Transient Thermal Convector heating a frying pan filled with water



Engineering Data: From th[redacted]library, insert these items.



[redacted]

[redacted]

Duplicate one of the above materials and create this one.

Outline of Schematic A2: Engineering Data

	A	B	D
1	Contents of Engineering Data		Description
4	[redacted]		

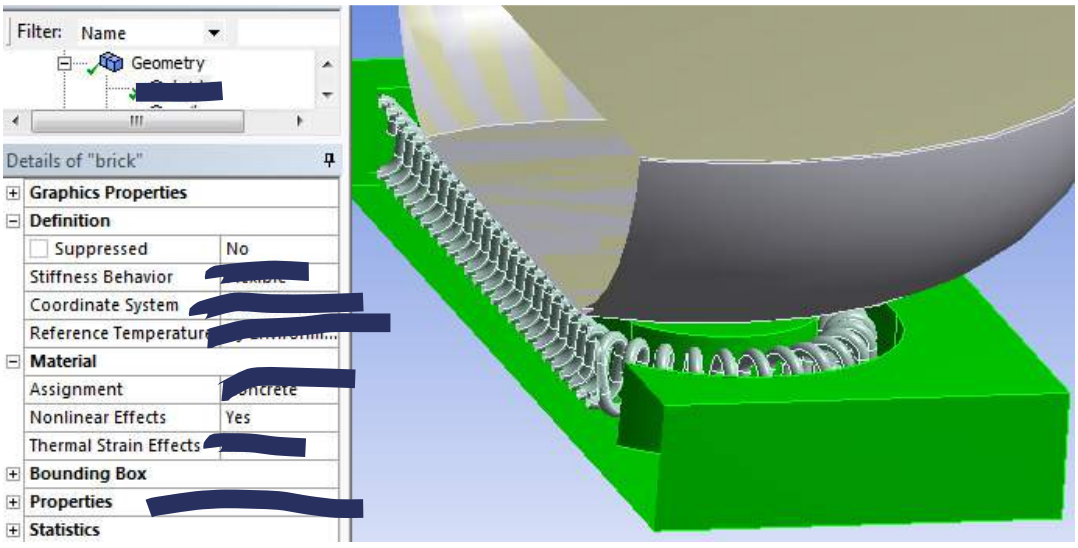
Properties of Outline Row 4: [redacted]

	A	B	C
1	Property	Value	Unit
2	Density	[redacted]	kg m^-3
3	Isotropic Thermal Conductivity	[redacted]	W m^-1 C^-1
4	Specific Heat	[redacted]	J kg^-1 C^-1

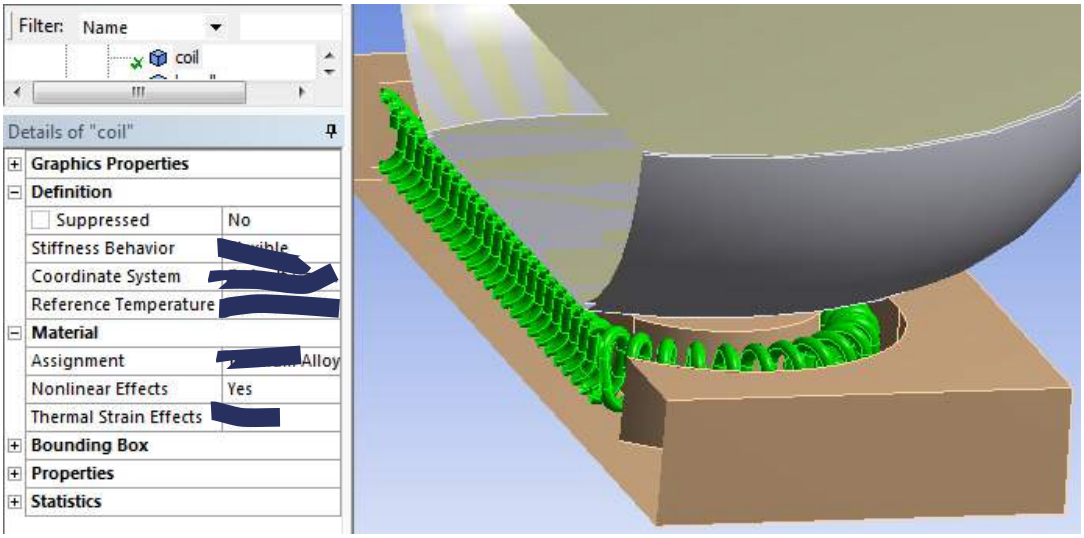
Geometry:201[redacted]

The materials names seen underneath might differ from the ones in th[redacted] assigned above. This is a neglect able matter.

For the brick body, green h[redacted]

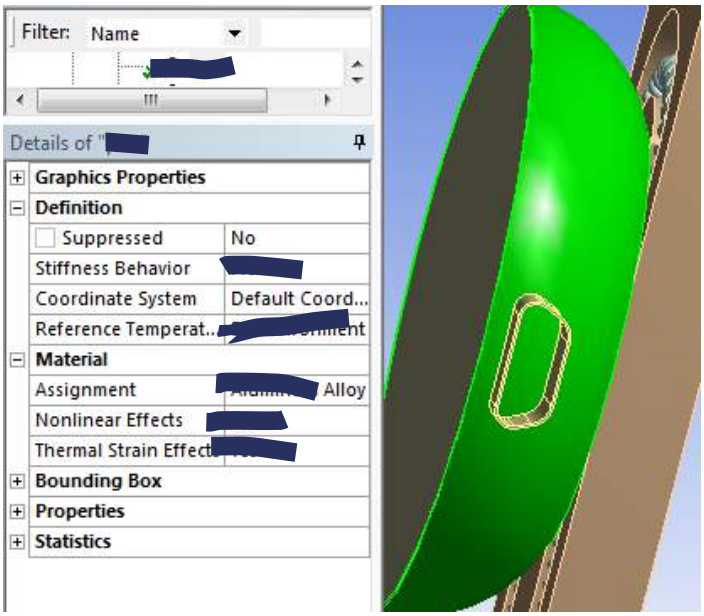


For the coiled body, green here, assign [redacted] Alloy.

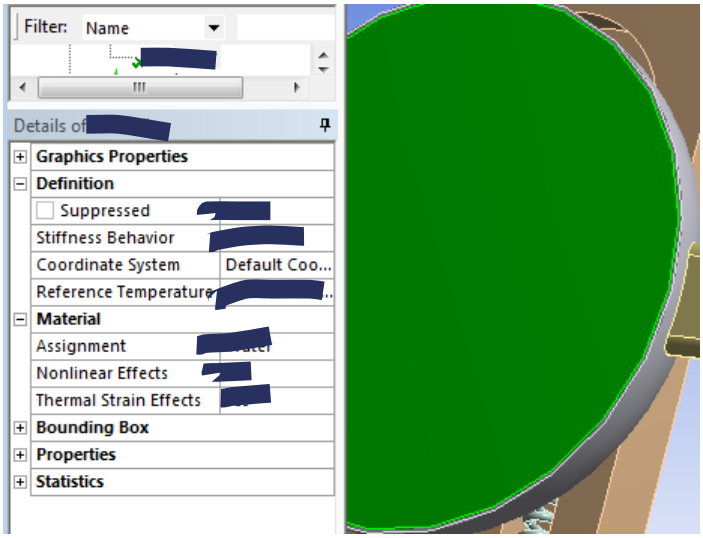


Leave the handle body, green here, as d[redacted]

For the frying pan, green here, assign [redacted] Alloy.

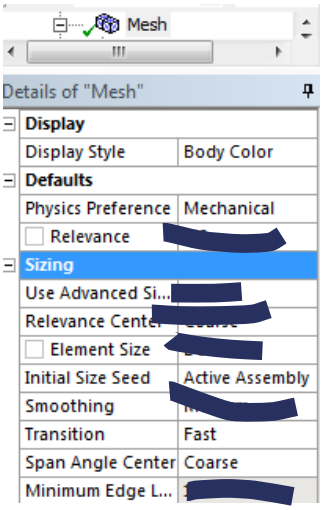


For the water body, green here, assign [redacted].

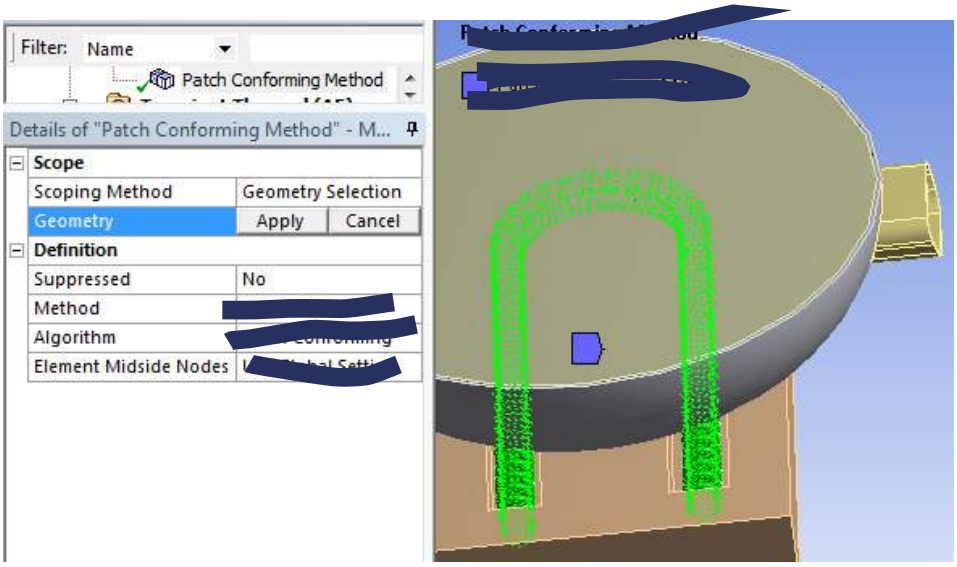


Connections: Leave all contacts [redacted]

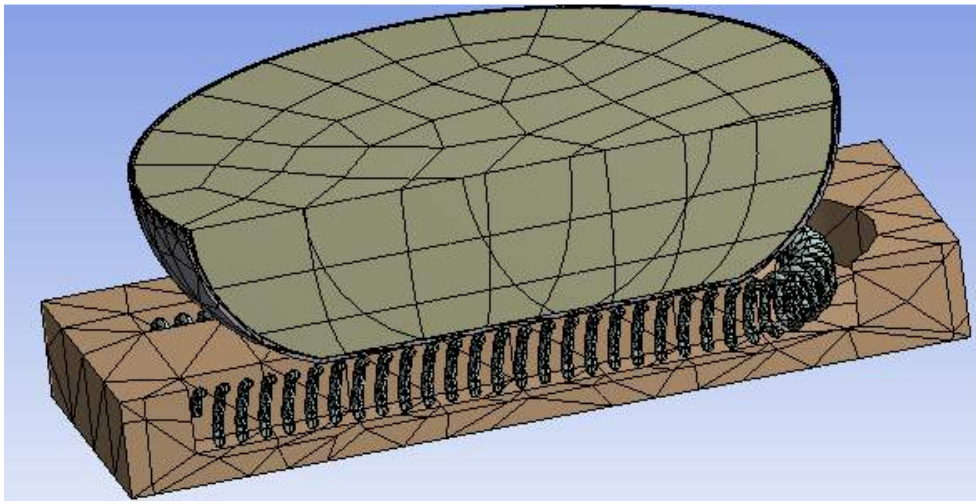
Mesh: Assign these details.



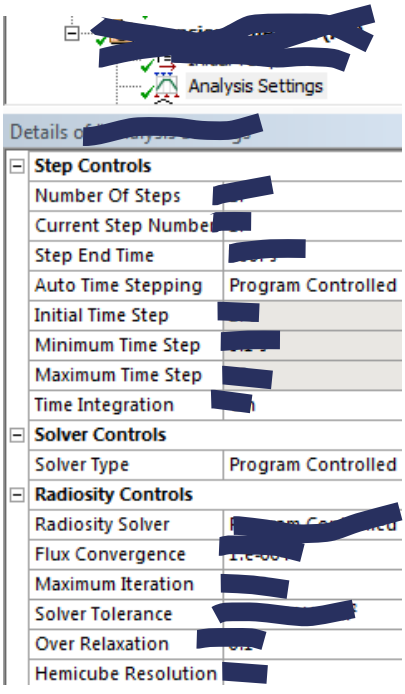
Create [redacted] method for the coil, green here.



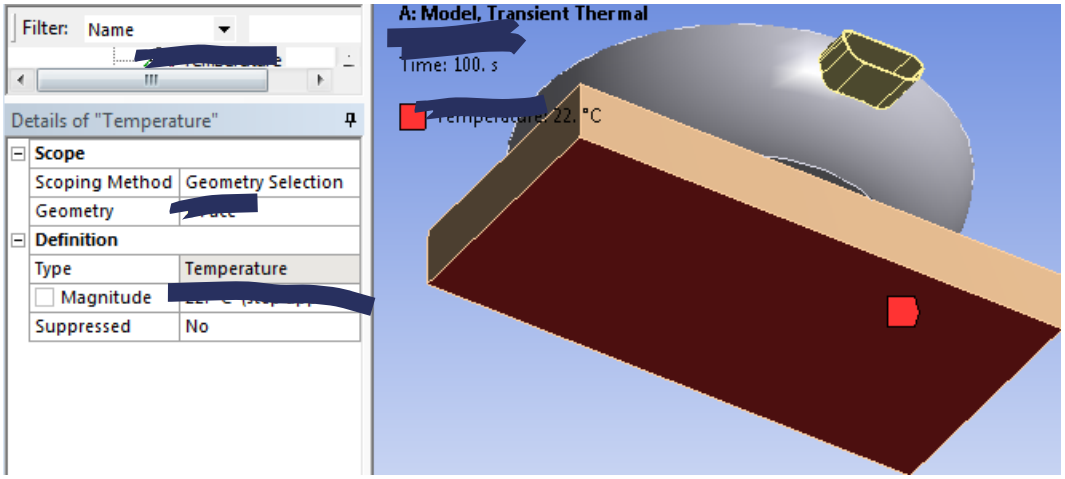
The mesh should look like here.



Analysis Settings: Apply these details.



Environment toolbar: Insert this default [redacted] red here.

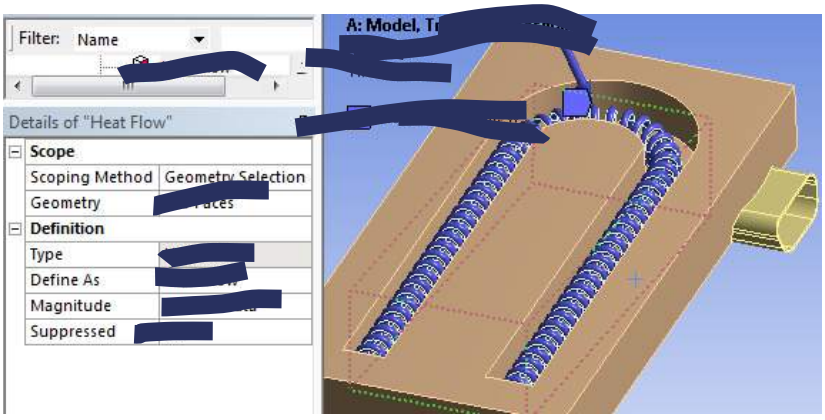


[redacted]

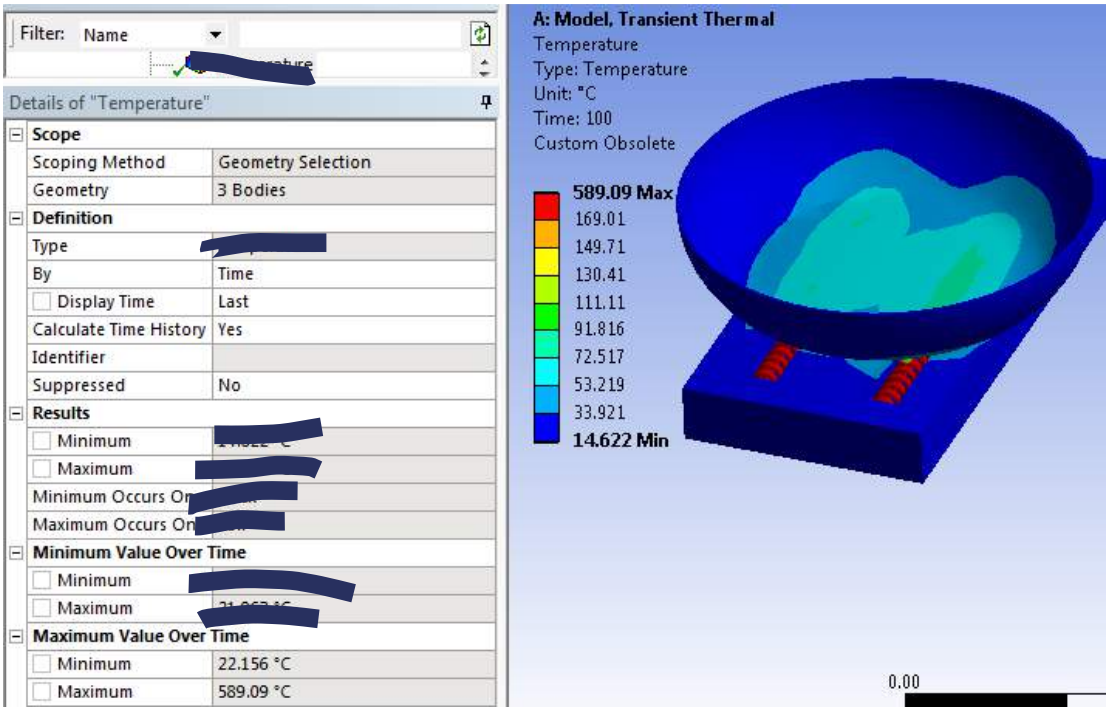
Apply a convection load on all faces of the assembly, as seen here in yellow.

[redacted]

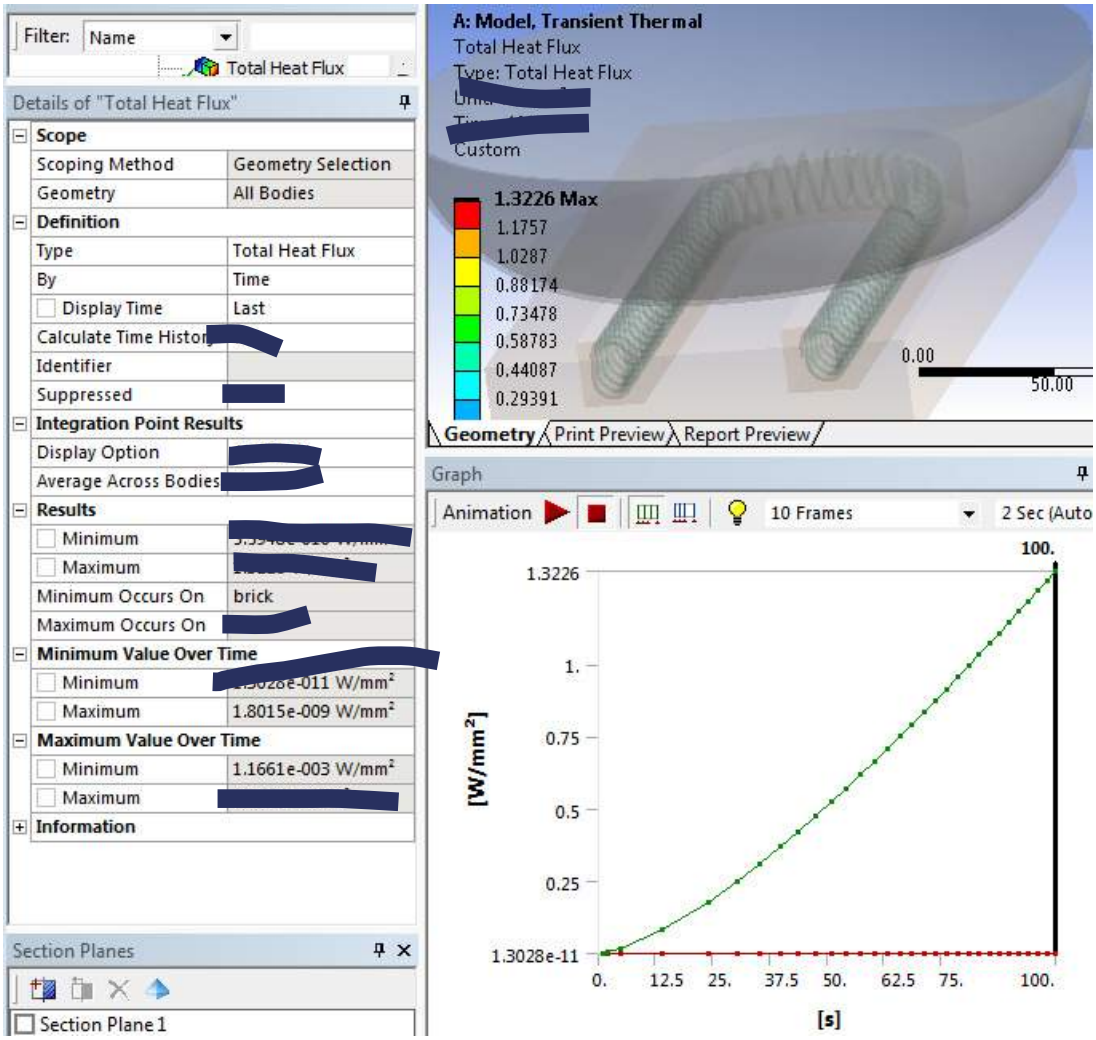
Create this heat flow load on the coil only, blue here.



Solution: Create a temperature item on these 3 bodies.

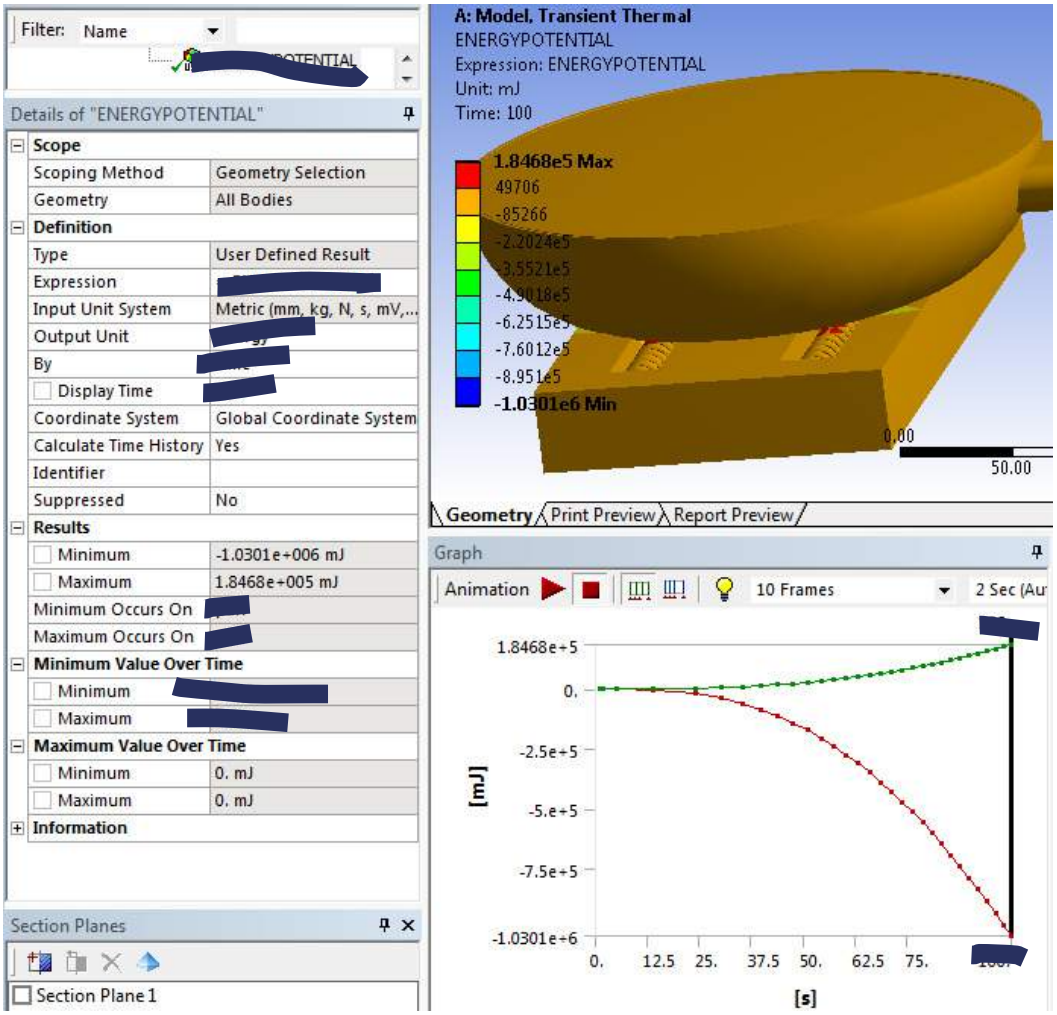


Create a total heat flux item on all bodies.



Scope for the temperature items separately, on each part.

After the FEA is solved, click Solution, t[redacted]scope for this item.



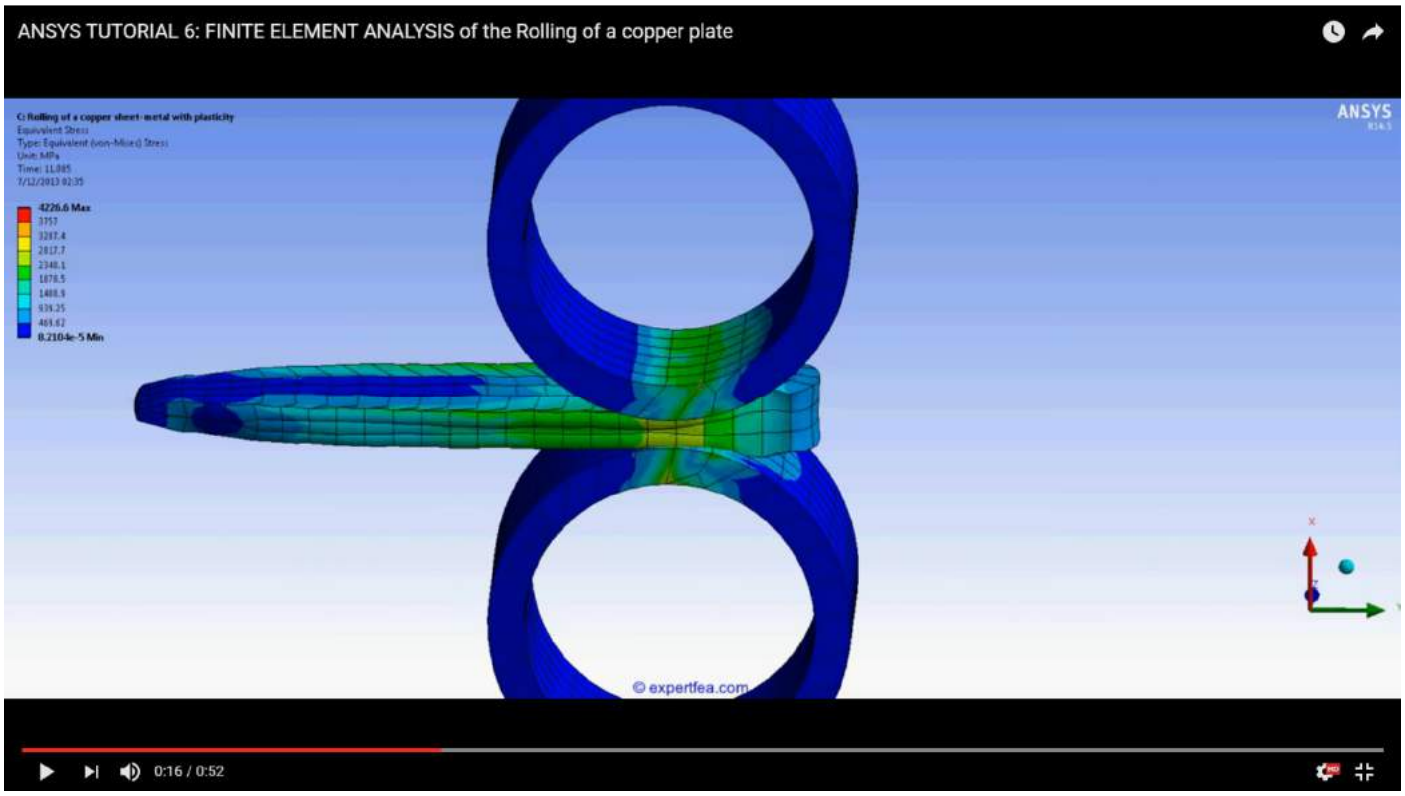
Further homework:

- in Connections, chang[redacted]nal $\mu = 0.05$, solve, draw the conclusions

- change Stiffness Behavior of the shell pa[redacted]ve, draw the conclusions

- increase the mesh[redacted]des, solve, draw the conclusions

CASE 31: ANSYS WB FINITE ELEMENT ANALYSIS - Static Structural Rolling of a copper plate

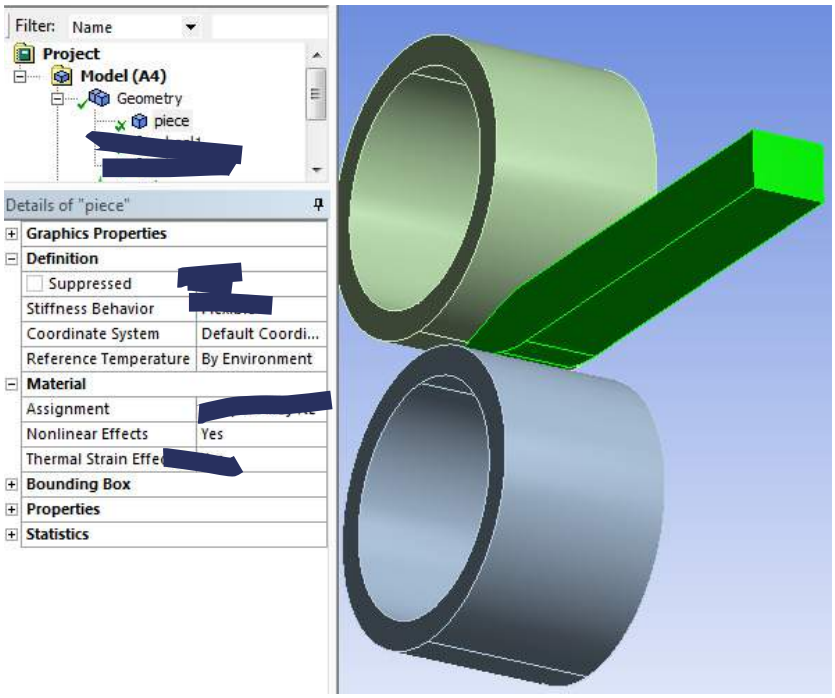


Engineering Data: Add C[redacted] Materials library.

[redacted]

Geometry: 2013_07_21_rolling.x_t

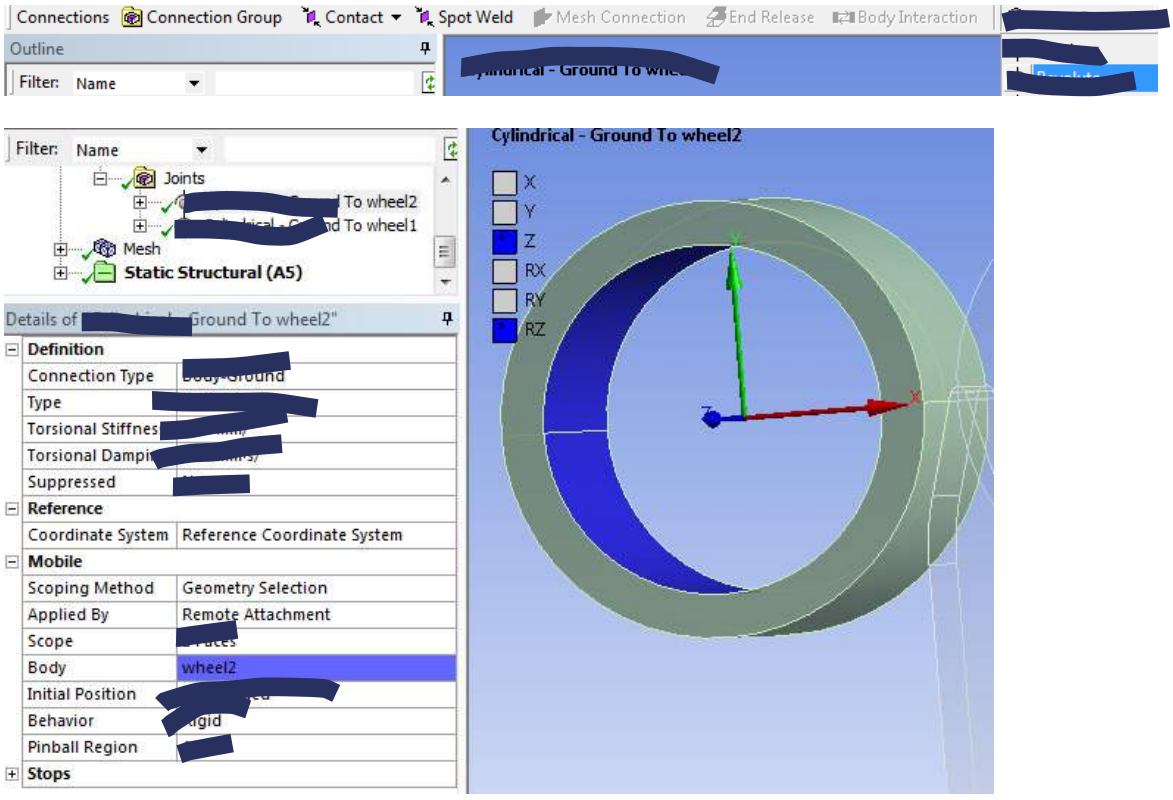
Assign C[redacted] NL to the green workpiece.

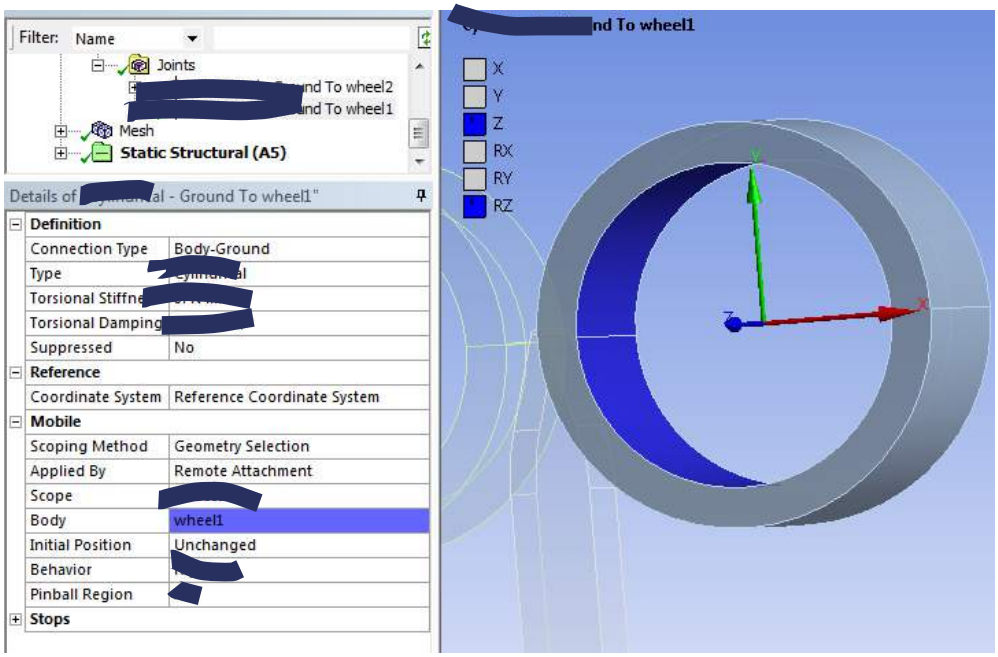


Connections, Contacts: Create such a contact between the workpiece and the roller.

Make a similar contact for the other roller.

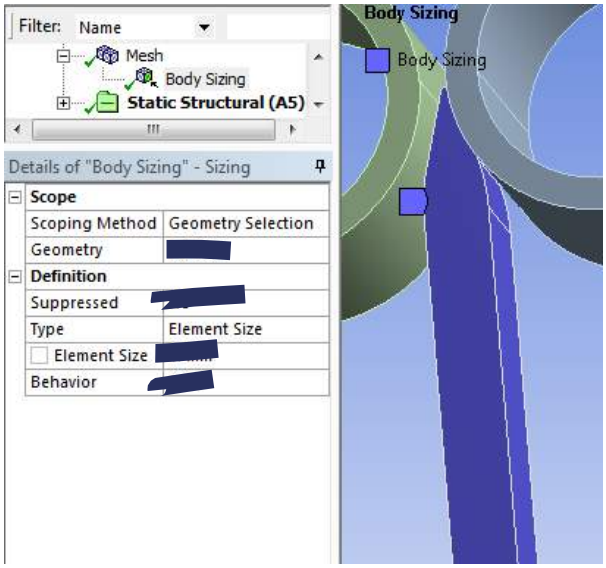
From the Connections toolbar, create a Body-Ground, [redacted] each roller.





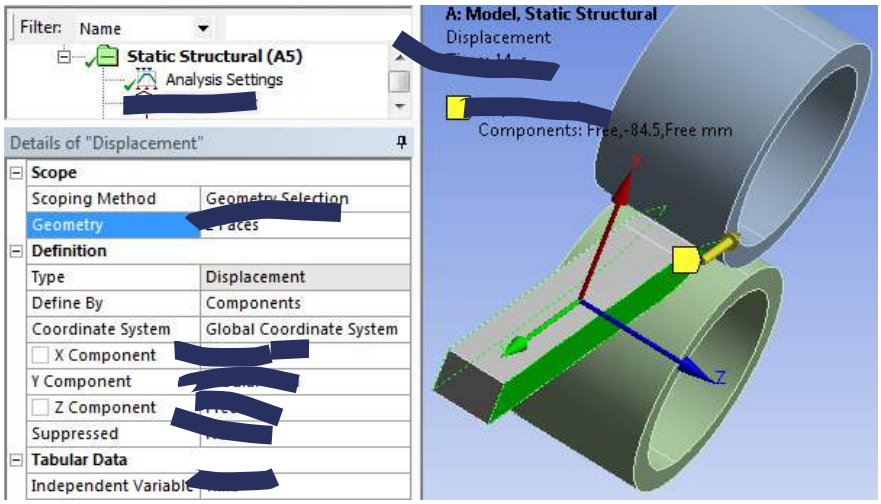
Mesh: Use these default details.

From the Mesh toolbar, apply a sizing of 4 mm on the blue piece.



Analysis Settings: Apply these details.

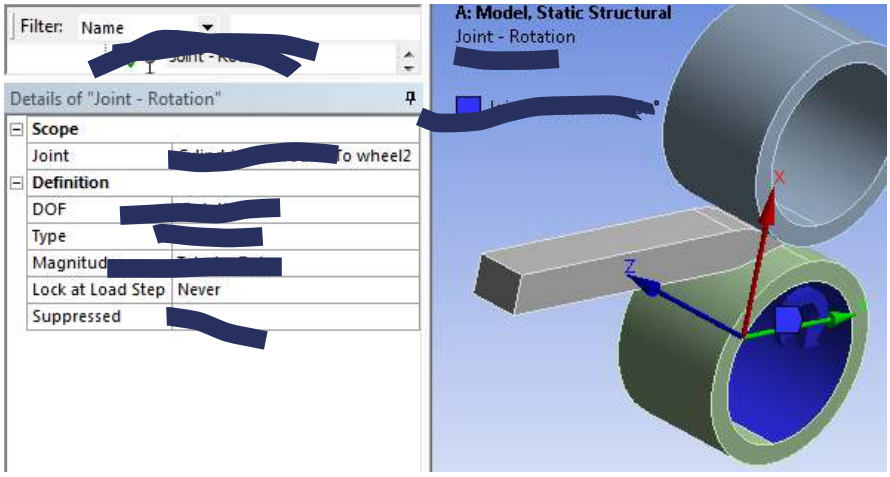
Environment toolbar: From the respective toolbar, go to Displacement and apply this item on both lateral faces of the workpiece.



Tabular Data

	Steps	Time [s]	Y [mm]
1	1	0.	
2	1	1.	
3	2	2.	
4	3	3.	
5	4	4.	
6	5	5.	
7	6	6.	
8	7	7.	
9	8	8.	
10	9	9.	
11	10	10.	
12	11	11.	
13	12	12.	
14	13	13.	
15	14	14.	

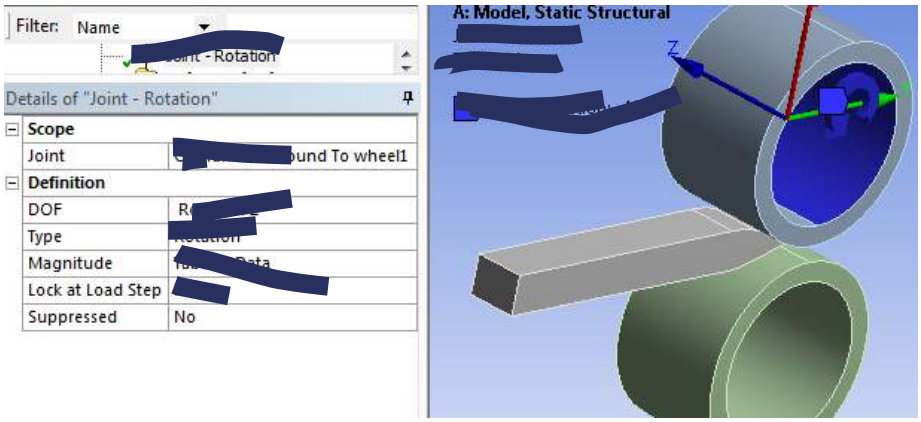
From the Environment toolbar, Loads, apply a Joint Load as this one.



Tabular Data

	Steps	Time [s]	Rotation [°]
1	1	0.	
2	1	1.	
3	2	2.	
4	3	3.	
5	4	4.	
6	5	5.	
7	6	6.	
8	7	7.	
9	8	8.	
10	9	9.	
11	10	10.	
12	11	11.	
13	12	12.	
14	13	13.	
15	14	14.	

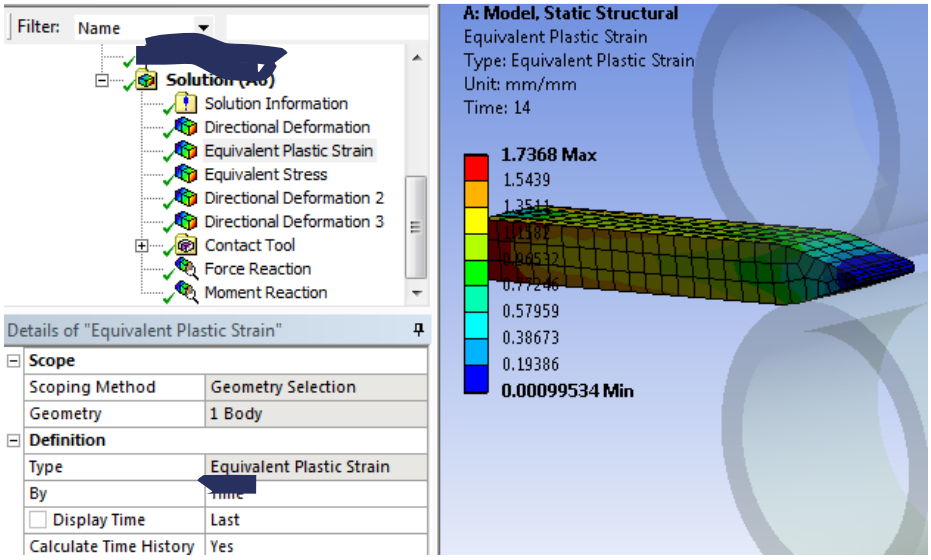
Apply a similar Joint Load for the other roller.



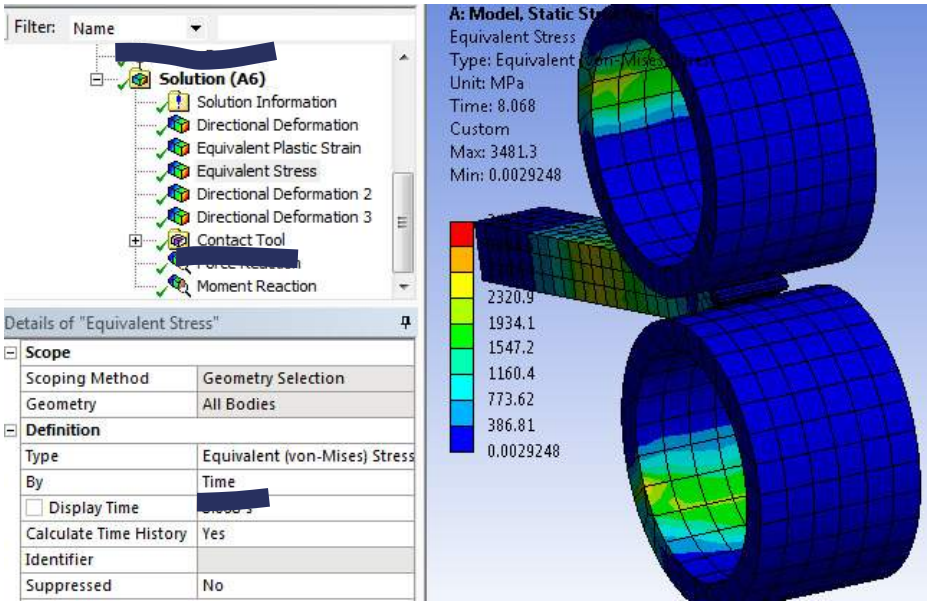
Solution: From the Solution toolbar insert the following items.

A Directional Deformation for the faces on the tip of the workpiece.

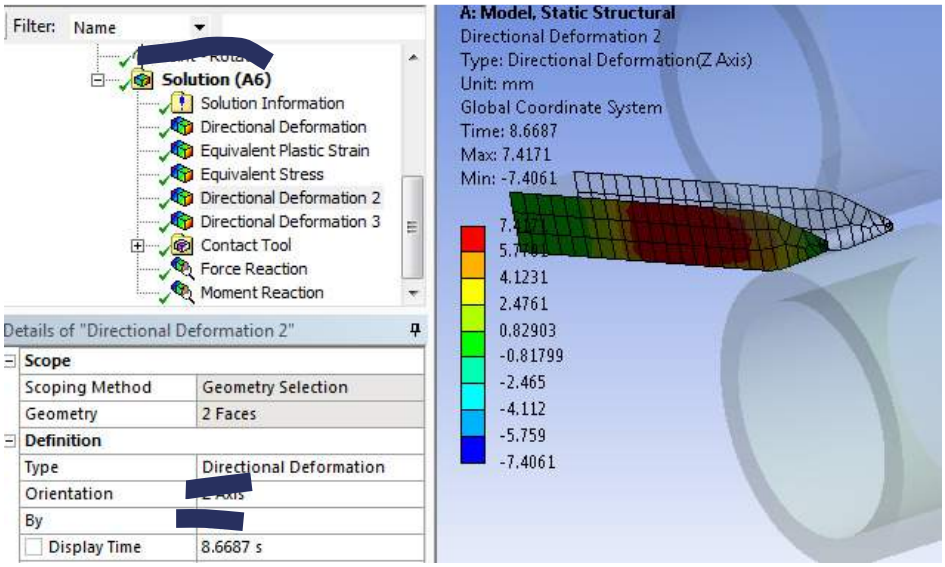
Apply a Plastic Strain only for the workpiece.



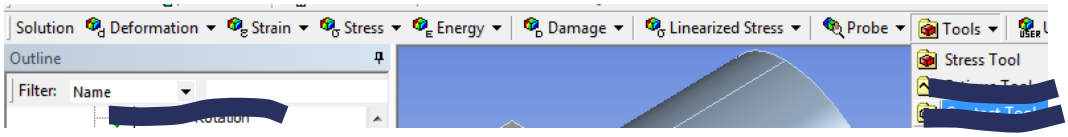
Assign a default Equivalent Plastic Strain for all bodies.



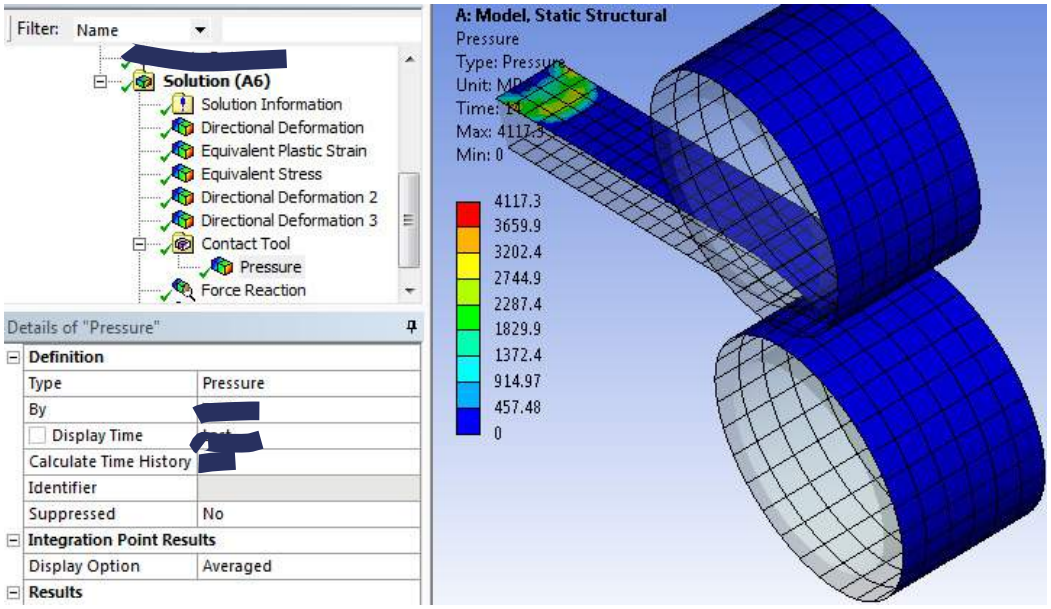
Create a Directional Deformation for the lateral faces of the workpiece.



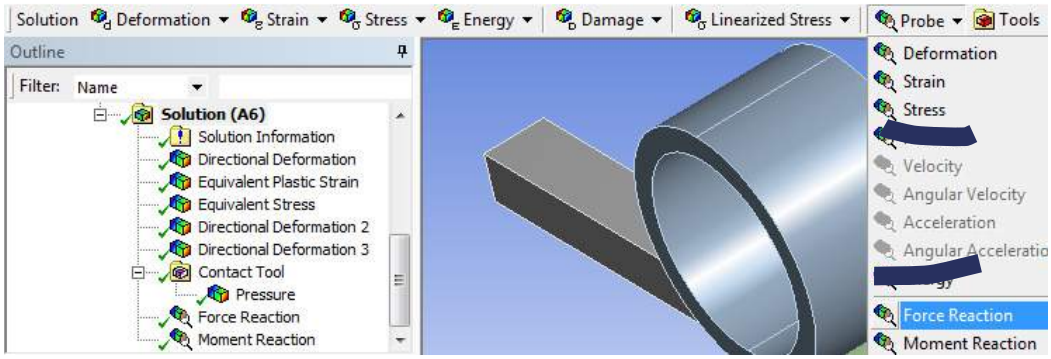
From Tools, Contact Tool, create this item.

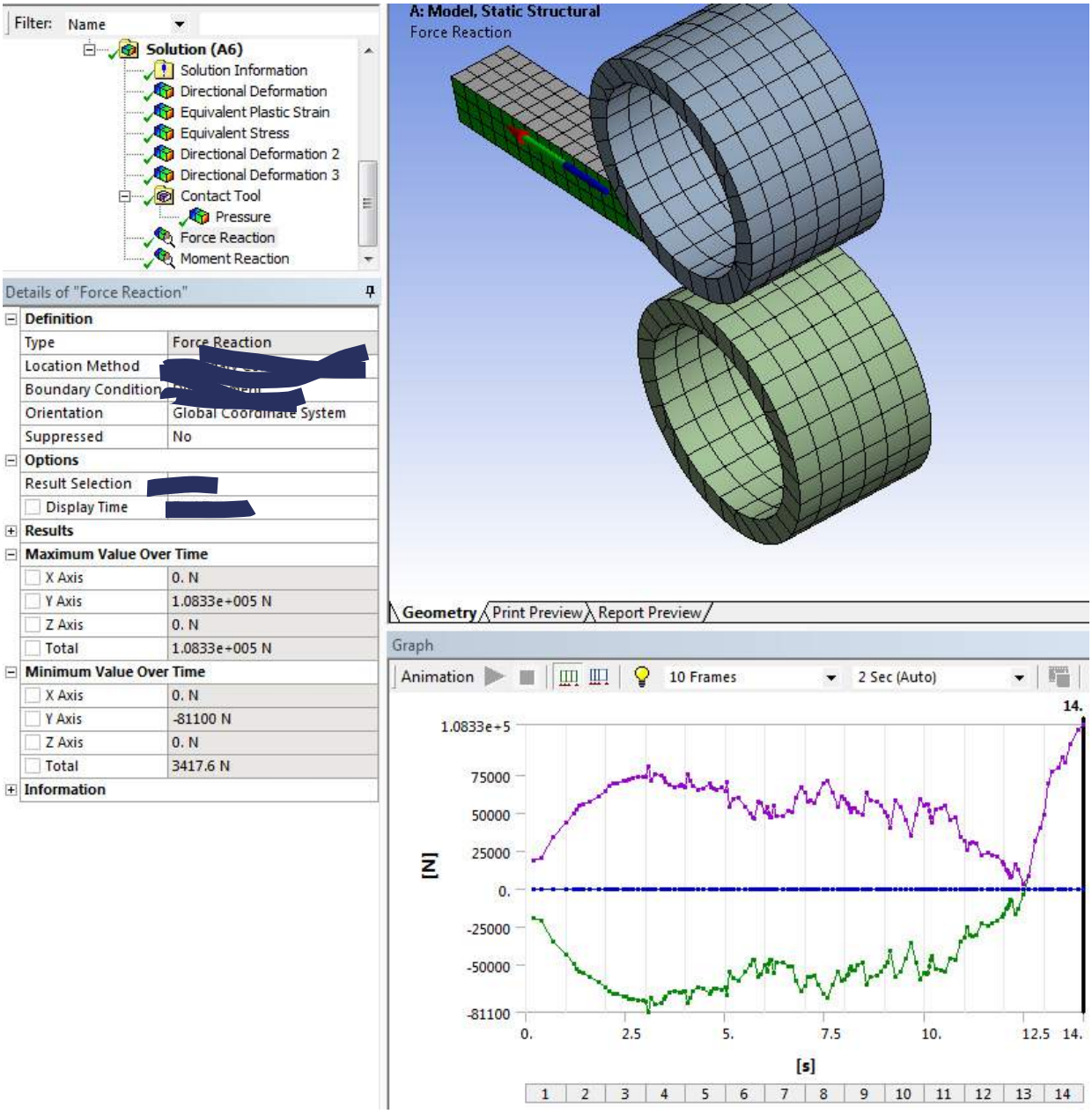


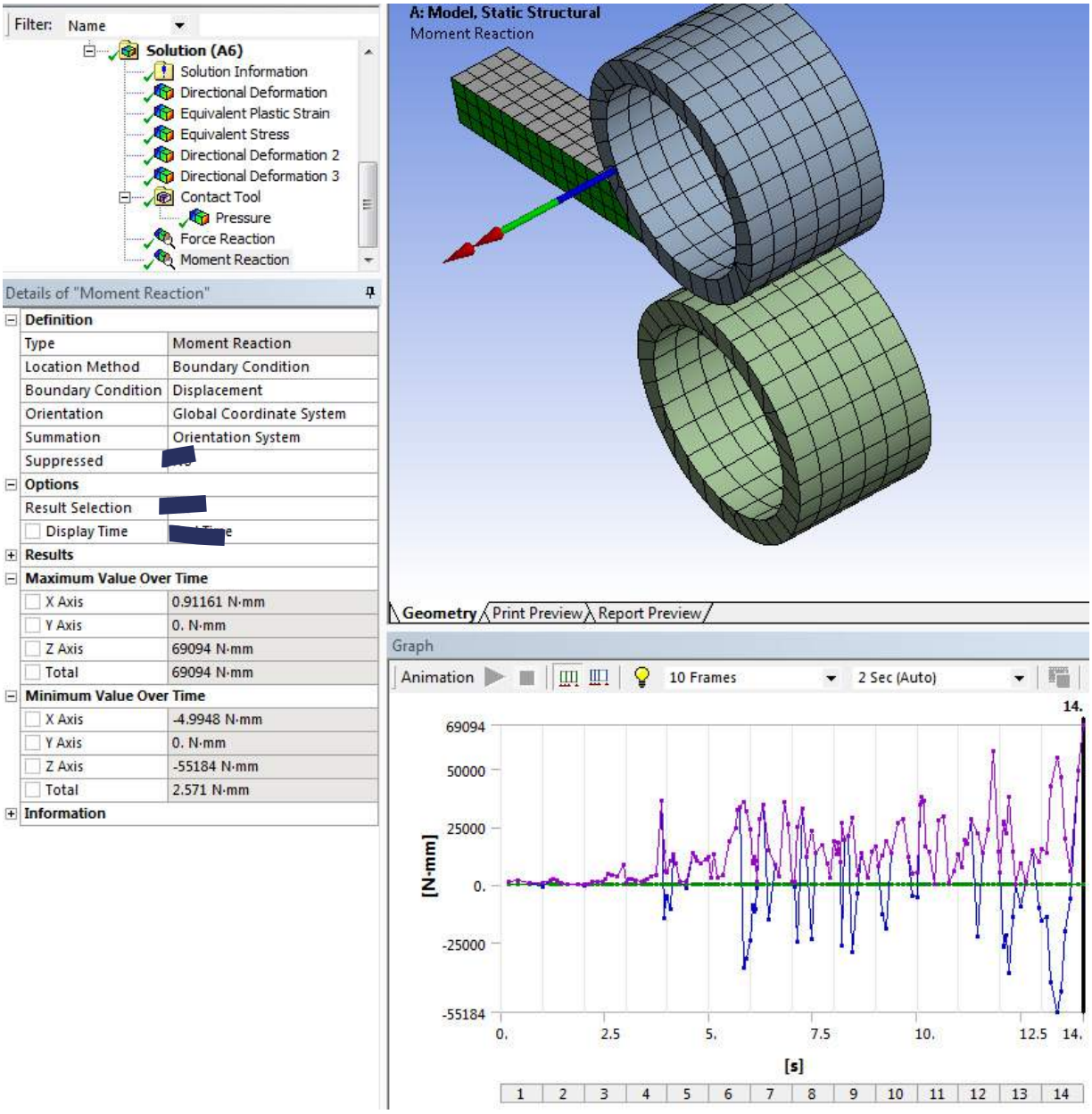
Right click, Insert, Pressure.



From Probe, Force Reaction and Moment Reaction, insert these items.




Filter: Name
Solution (A6)
Solution Information
Directional Deformation
Equivalent Plastic Strain
Equivalent Stress
Directional Deformation 2
Directional Deformation 3
Contact Tool
Pressure
Force Reaction
Moment Reaction
Details of "Force Reaction"
Definition
Type Force Reaction
Location Method
Boundary Condition
Orientation Global Coordinate System
Suppressed No
Options
Result Selection
Display Time
Results
Maximum Value Over Time
X Axis 0. N
Y Axis 1.0833e+005 N
Z Axis 0. N
Total 1.0833e+005 N
Minimum Value Over Time
X Axis 0. N
Y Axis -81100 N
Z Axis 0. N
Total 3417.6 N
Information
A: Model, Static Structural
Force Reaction
Geometry
Print Preview
Report Preview
Graph
Animation
10 Frames
2 Sec (Auto)
14.
1.0833e+5
75000
50000
25000
0.
-25000
-50000
-81100
[N]
0.
2.5
5.
7.5
10.
12.5
14.
[s]
1 2 3 4 5 6 7 8 9 10 11 12 13 14



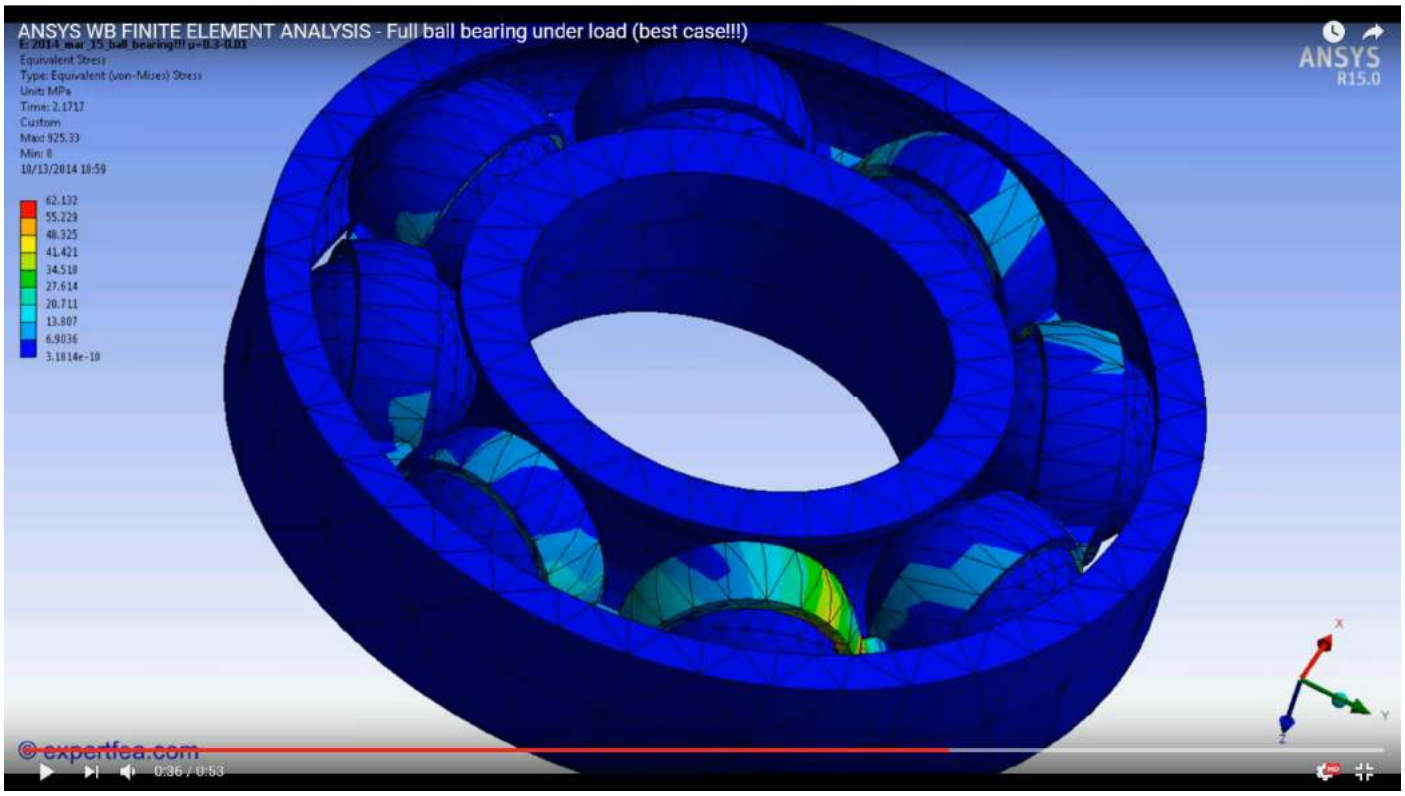
Further homework:

- in Connections, change Frictional Contact [redacted] to 0.1, solve, draw the conclusions

- replace Copper [redacted] solve, draw the conclusions

- decrease the mesh [redacted] solve, draw the conclusions

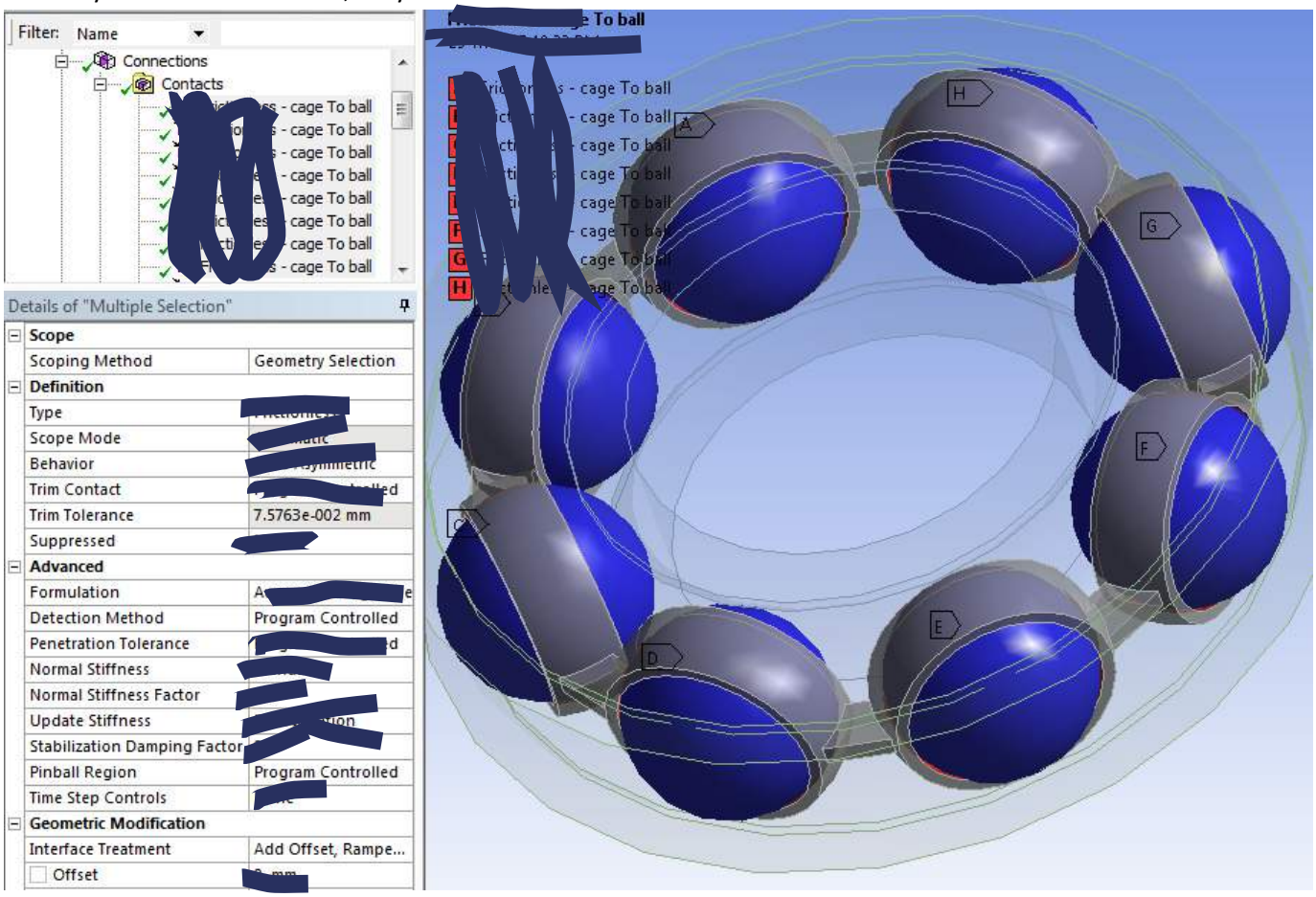
CASE 33: ANSYS WB FINITE ELEMENT ANALYSIS - Static Structural Full ball bearing under load



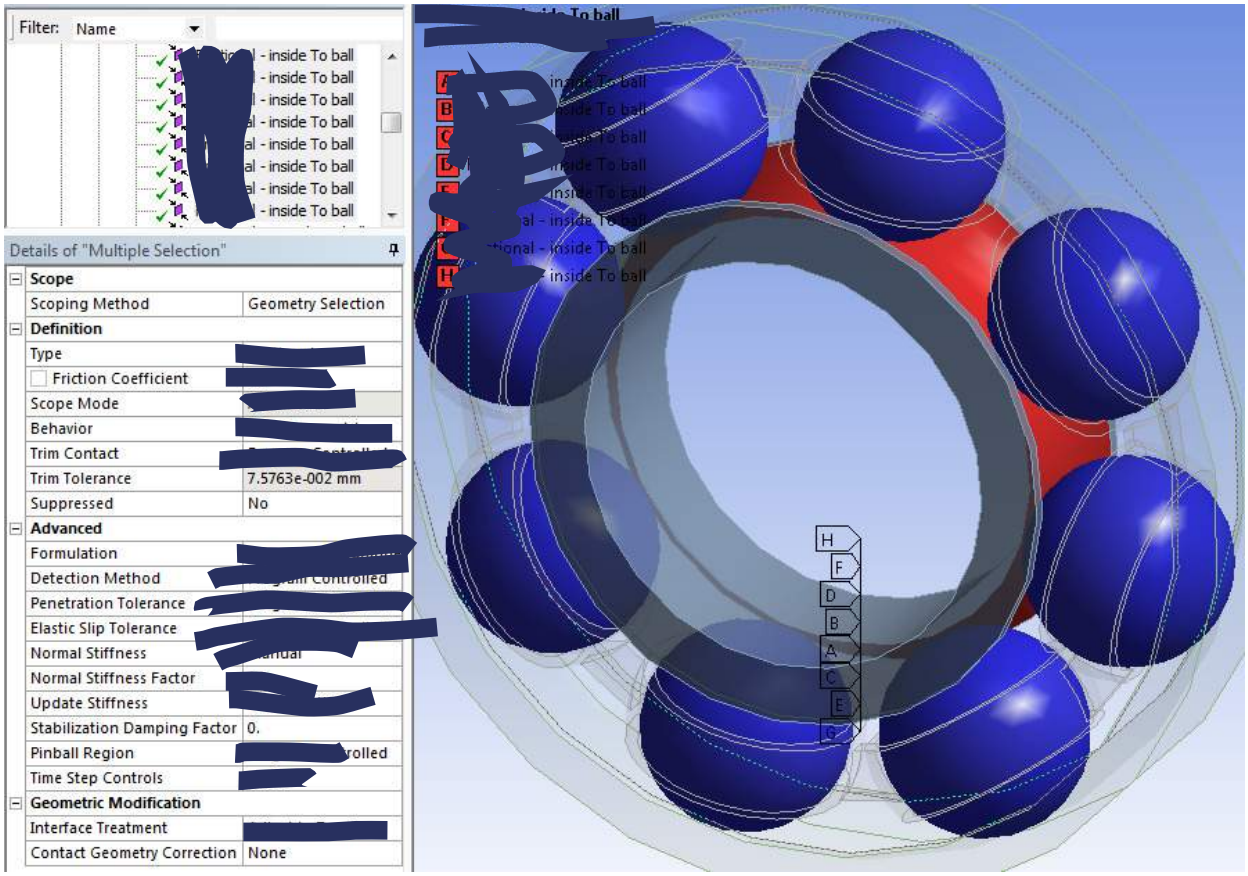
Engineering Data: N[redacted]

Geometry: 20[redacted]

Correctly done for all the balls, they should look like here.

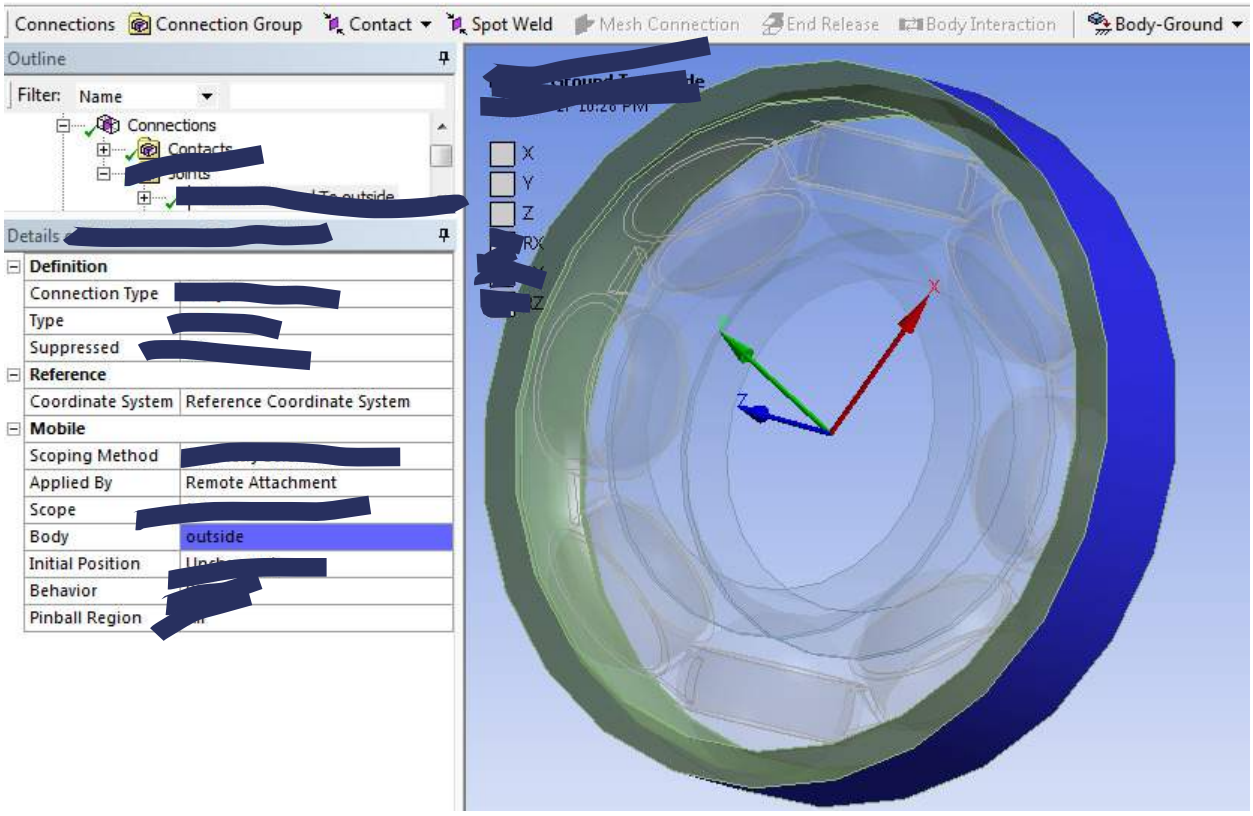


Correctly done for all the balls, they should look like here.

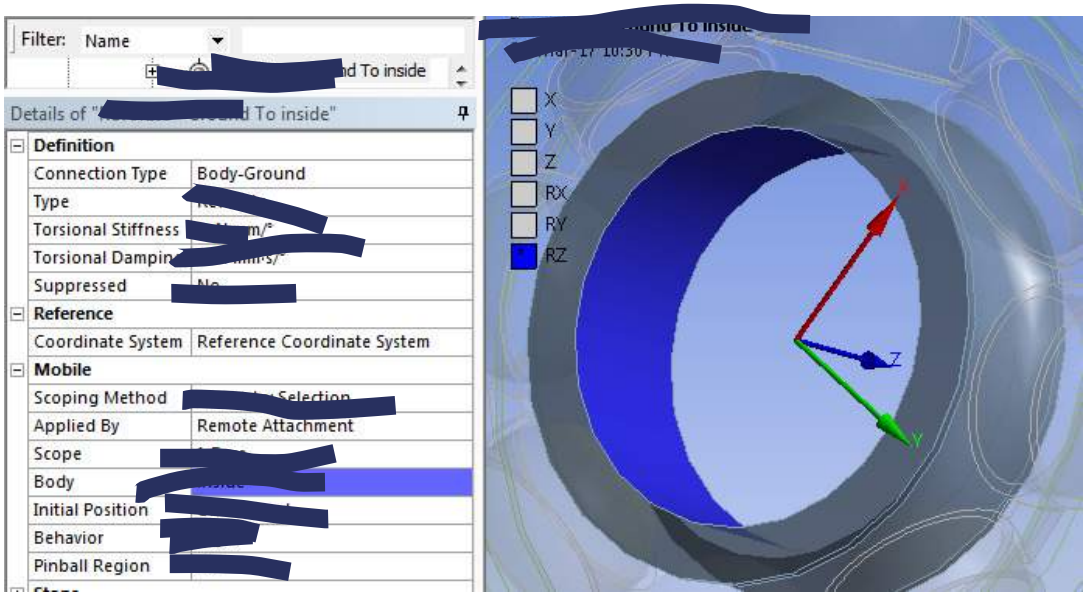


Correctly done for all the balls, they should look like here.

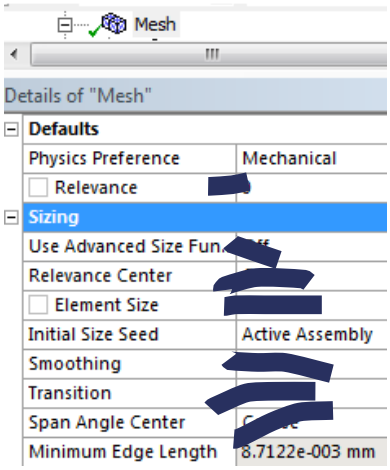
From the Connections toolbar [redacted] on the outer ring, blue here.



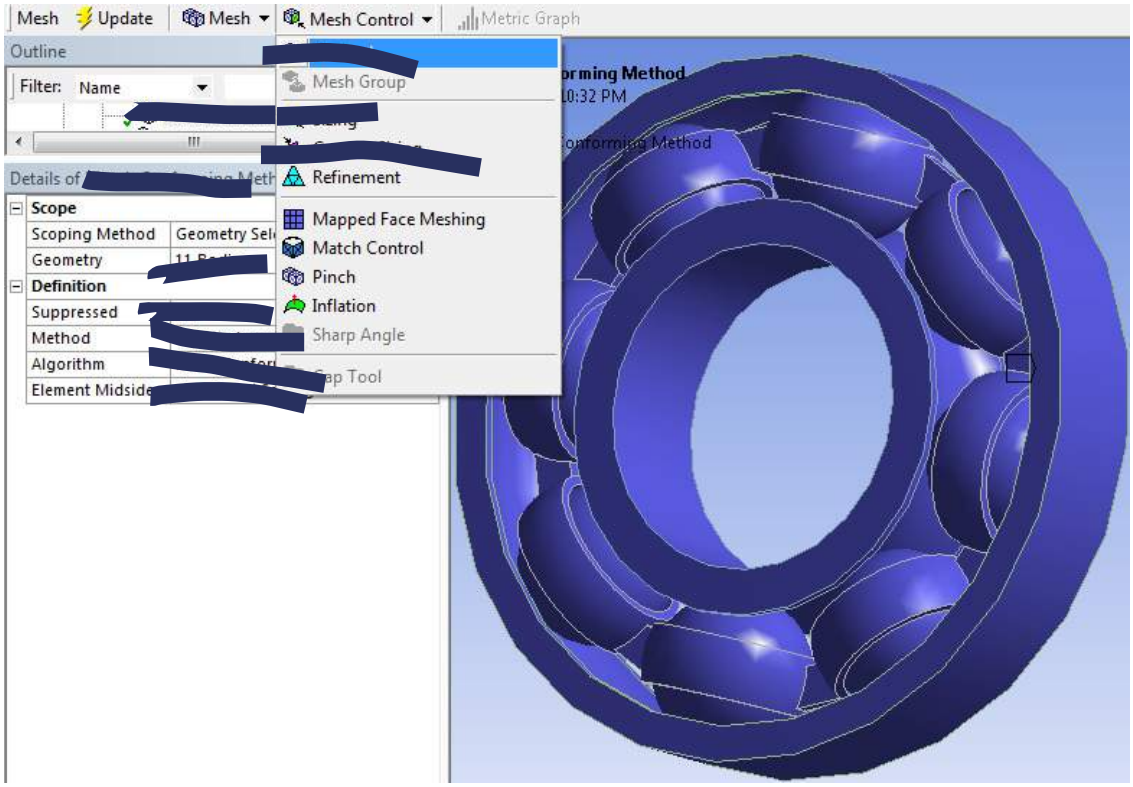
In a similar manner, create t[redacted]he inner ring, blue here.



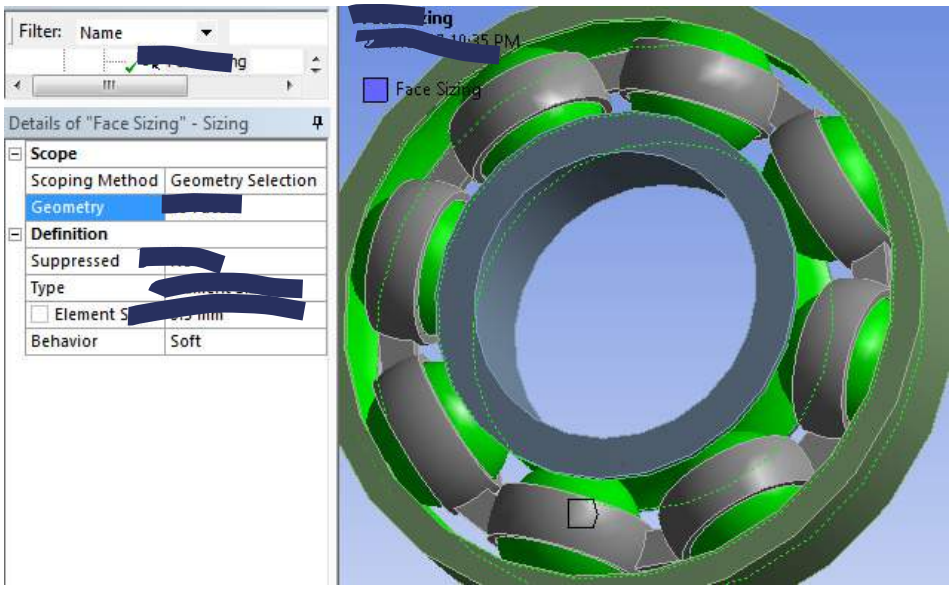
Mesh: Assign these details.



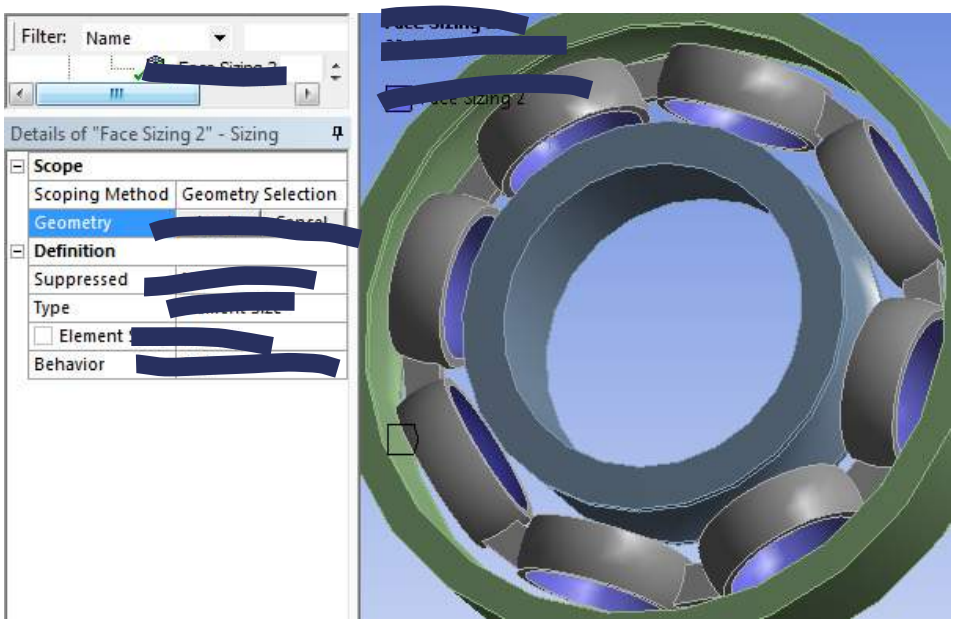
From the Mesh toolbar, Mesh Control, apply this Meth[redacted]



From the same toolbar and button create a Sizing on all [redacted] faces, except the [redacted] spherical faces of the cages, [redacted] are held on.



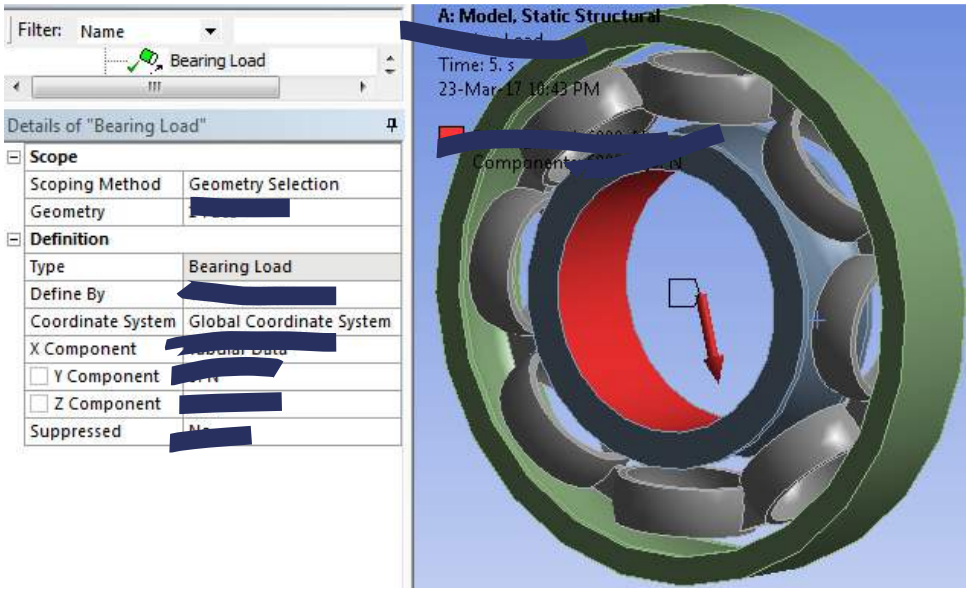
Insert a similar sizing as this one on the aforementioned internal spherical faces of the cage, seen blue here.



Analysis Settings: Apply these details, as seen here for the 1st timestep.

Select the other steps from the Graph tab and make for each of them Carry Over Timestep = On, keeping the same details as above.

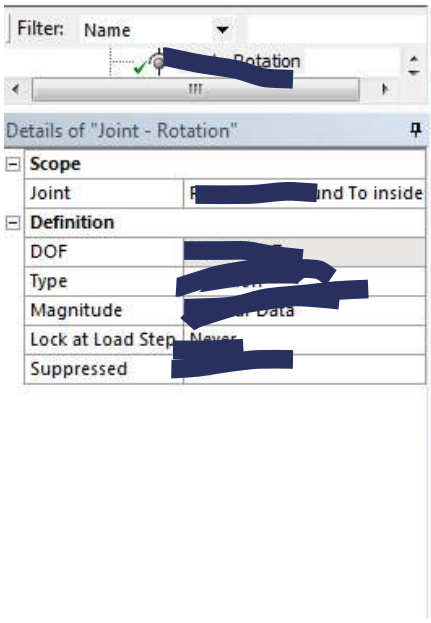
Environment toolbar: From the respective toolba[redacted] button, create a [redacted]ring Load on the inner ring, seen red here.

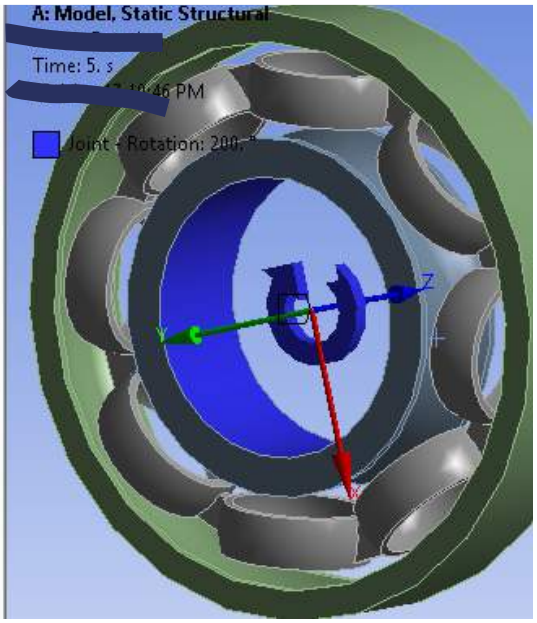


Tabular Data

	Steps	Time [s]	X [N]	Y [N]	Z [N]
1	1	0.	0.		= 0.
2	1	1.			0.
3	2	2.			= 0.
4	3	3.			
5	4	4.	= 6000.	= 0.	
6	5	5.	= 6000.	= 0.	

From the same location, apply a Joint Load similar to this one.

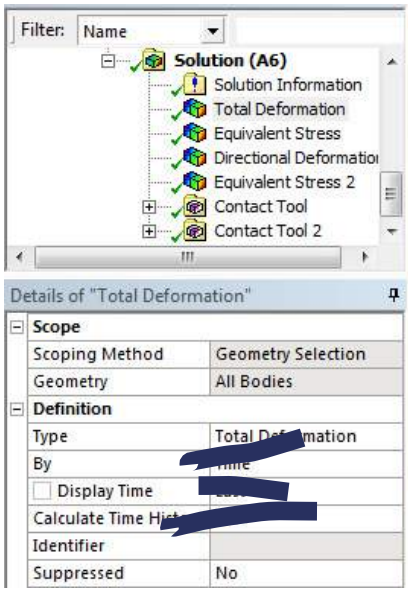


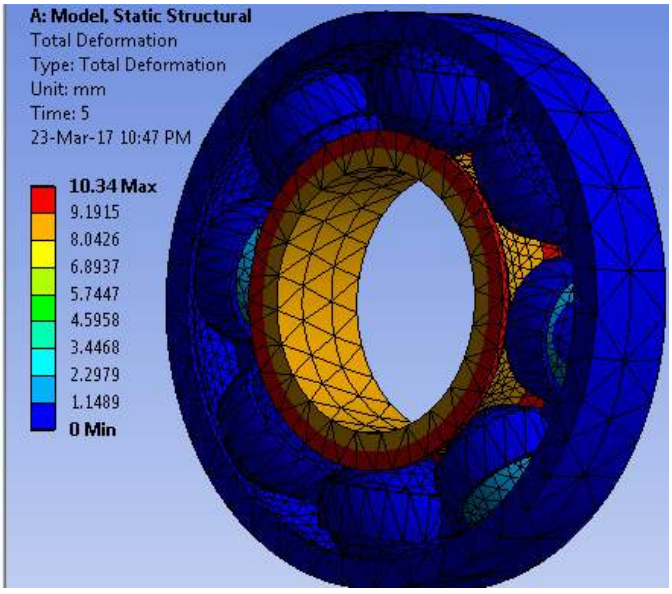


Tabular Data

	Steps	Time [s]	Rotation [°]
1	1	0.	
2	1	1.	
3	2	2.	
4	3	3.	
5	4	4.	
6	5	5.	

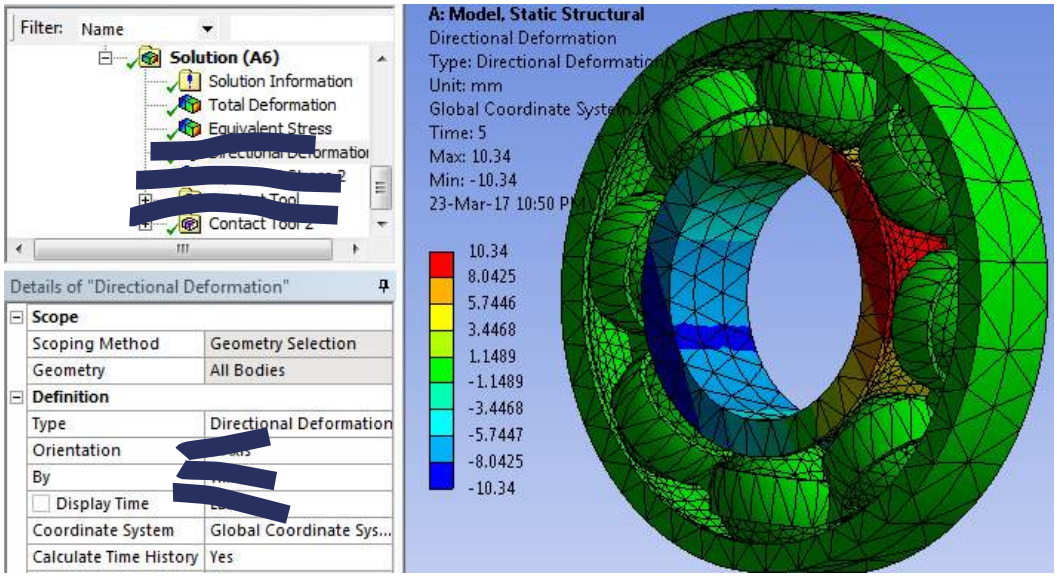
Solution: Apply a default Total Deformation on all bodies.



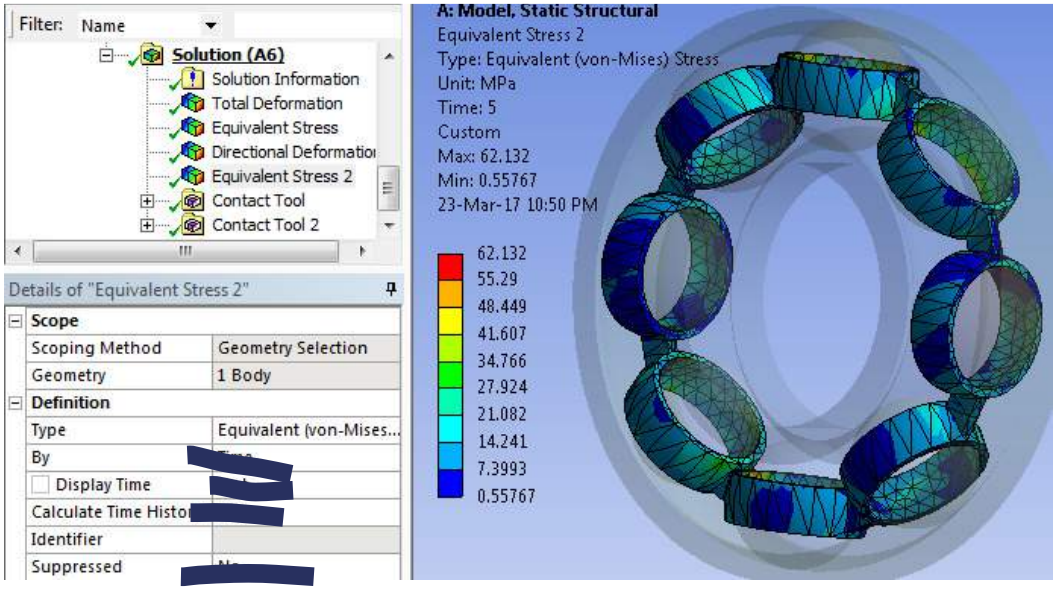


Create a default Equivalent Stress on all bodies.

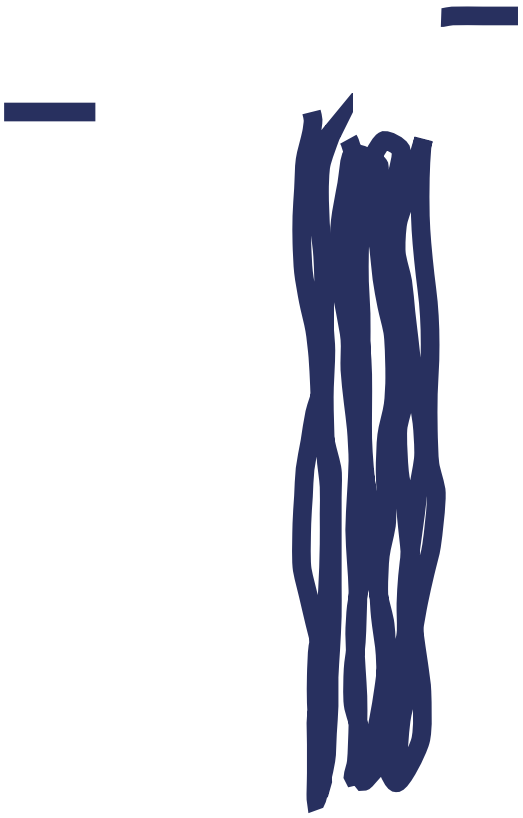
Make a default Directional Deformation on Y axis for all bodies.



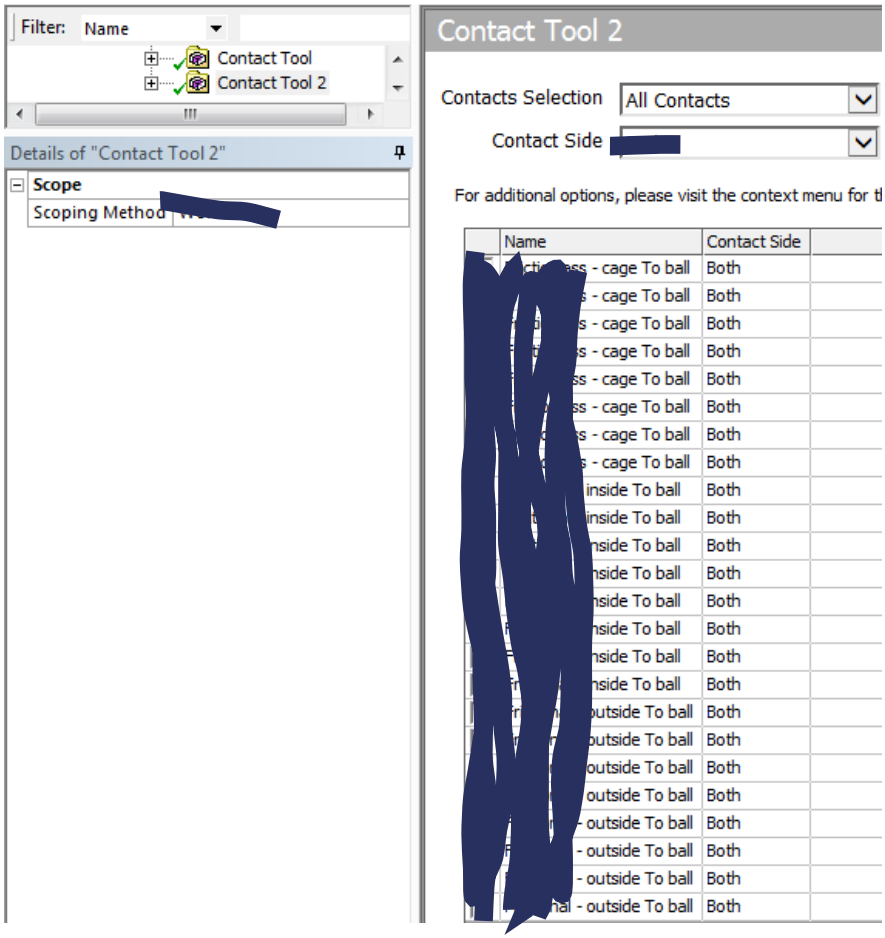
Insert an Equivalent Stress only for the cage.



Define a Contact tool only for these contacts. Right click, Insert, Pressure.



Define another Contact tool only for these contacts. Right click, Insert, Pressure.



Further homework:

- in Connections, change Frictionless contacts t[redacted], solve, draw the conclusions

- change S[redacted]avior of the inner and outer ring[redacted], draw the conclusions (if question marks appear on the existing contacts, right click on the[redacted])

- increase the mesh s[redacted], solve, draw the conclusions