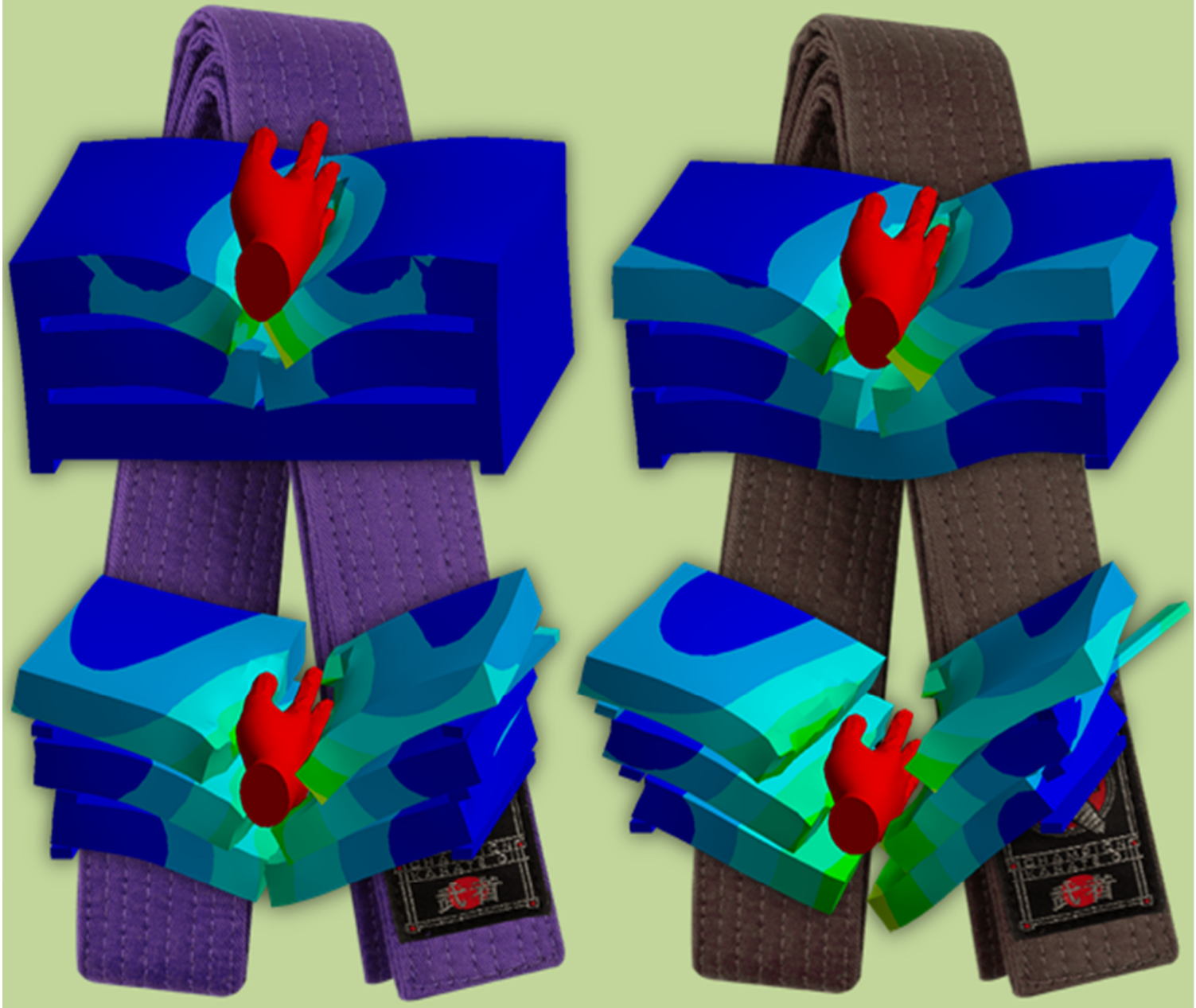
Become a Black Belt in ANSYS Workbench

by Claudiu Danila

Volume 3: purple and brown belt

- 20 fast tutorials for advanced users -



We dedicate this book to God's Ecstatic Beauty and Beatific Love:

Tripura Sundari

Foreword

Hi all!

I held your hand in the first 2 volumes, now it's not the case anymore; this means that you need to know by now how to create a contact, a joint and how to apply a displacement - prior to reading and applying this book - because I kept the explanations to a minimum. If you are a beginner and still want to apply what's in this book, you might want to check firstly the **Hands-on ANSYS Workbench – nothing else** volumes and visit the Webinars page from *expertfea.com*. Remember that there are also for sale the MECHDAT files related to this book, if somehow you can't apply the instructions we've shown in the book.

In this book I did my best to vary the domains, to avoid the book to become boring. As usually, to extend the amount of work you do in FEA, remember to follow also the Homework sections, because they are very valuable in further understanding the software and the domain.

Also, remember that, in order to master the FEA field, CAD modeling should be no problem for you. In the department where you'll work, your designer colleagues will send you geometries made in SolidWorks, Catia, ProEngineer, Inventor, Unigraphics etc., so do not make the mistake in spending too much time in becoming an expert in DesignModeler, but invest your time in Spaceclaim the most, since it's the best tool that exist for FE analysts.

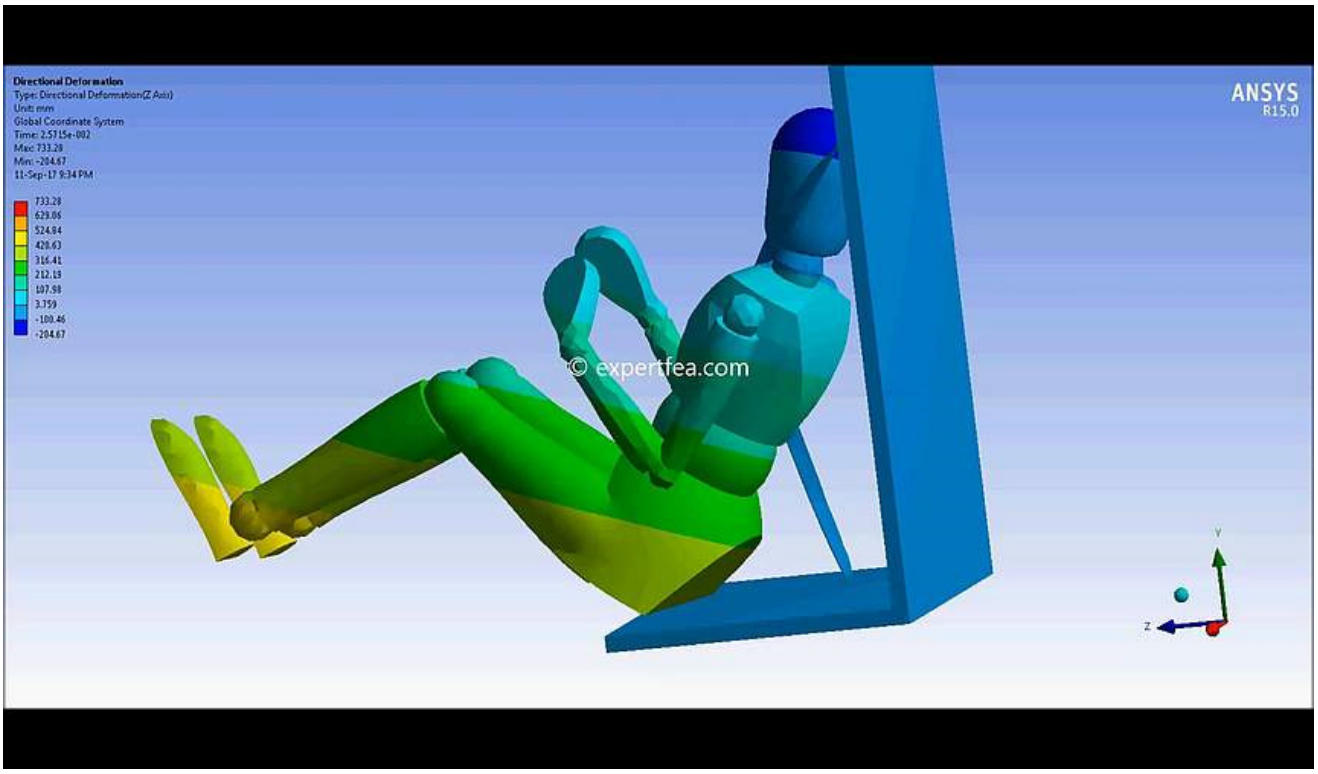
And something I wanted to tell my customers since some time ago: the scenarios presented on *expertfea.com* are of the highest difficulty (we are the 1st to present bearings, stampings, machining, train wheels on tracks etc. in ANSYS Workbench); each scenario is a result of weeks and months of practice, trial and error – this is why you'll conclude that there is a very big gap between your knowledge and what is presented in these books. I present a fast forward of my work, I compressed the FEA work very much and put it in writing. But the weeks I spent on these scenarios, you'll need to spend them too, for other cases, in your future as a stress engineer.

I can assure you now that there will be a 4th volume, a Black belt one, with the hardest cases and the least details. It will be published in the next year, I presume. It takes a lot of time to present high complexity scenarios and not to adhere to the usual trend of the FEA books that exist nowadays (which contain some chapters explaining F = k * X, a chapter with beams, one with trusses, a chapter with 2D FEA, one with 3D FEA, a chapter with vibrations, one with static, and the book is pretty much done).

Meanwhile, be sure to look forward for the book I will publish in the next months, with a title like: **Fluid – Structure Interaction using coupled Eulerian – Lagrangian Explicit Dynamics in ANSYS Workbench** – in which the complex interactions between fluids, solids and explosive materials will be presented in 20 practical FEA scenarios.

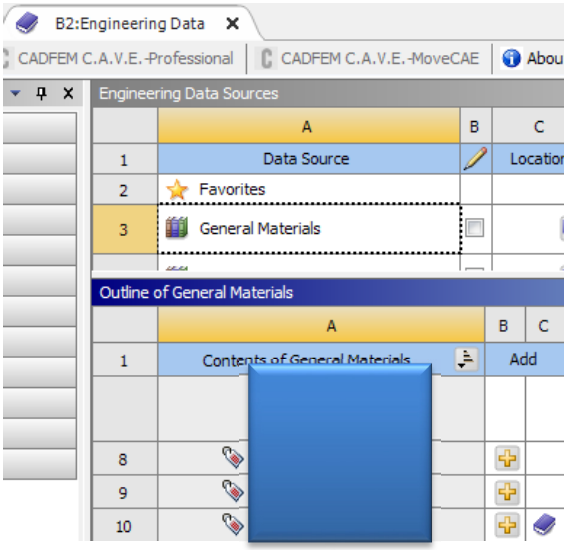
As always, I wish you all the best and believe in yourself and your future! **Claudiu, 14th of March 2018**

CASE 41: ANSYS WB FINITE ELEMENT ANALYSIS - Simulation of seat belt and dummy at 150g deceleration



Explicit Dynamics

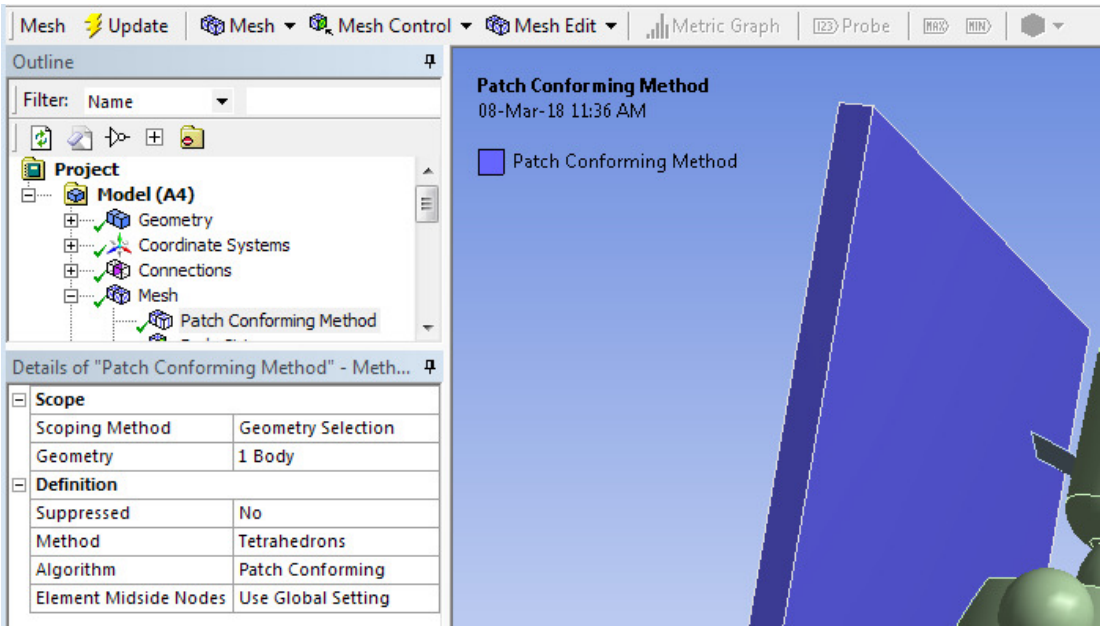
Engineering Data (Materials): Add Polyethylene from general materials library (click the yellow plus sign).



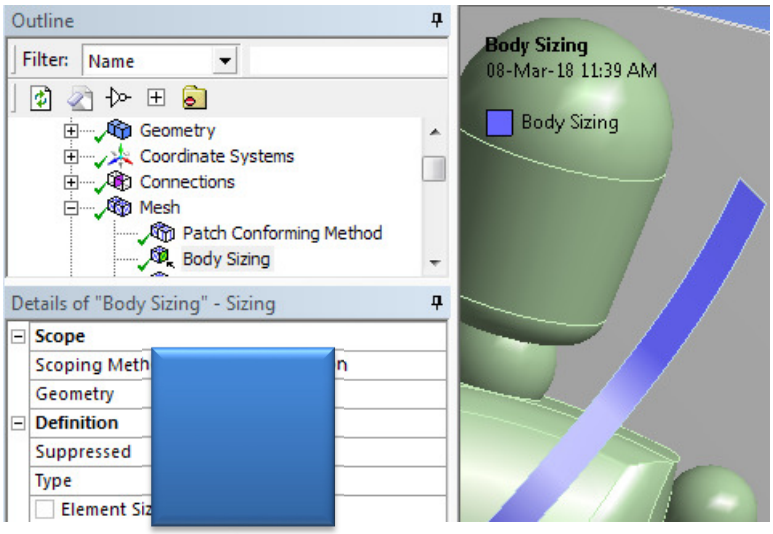
Geometry: 4sept2017v5_seatbelt_manikin.x_t

For the puspose of this FEA only, assign the shell part representing the seatbelt 2 mm thickness and leave it as Structural Steel.

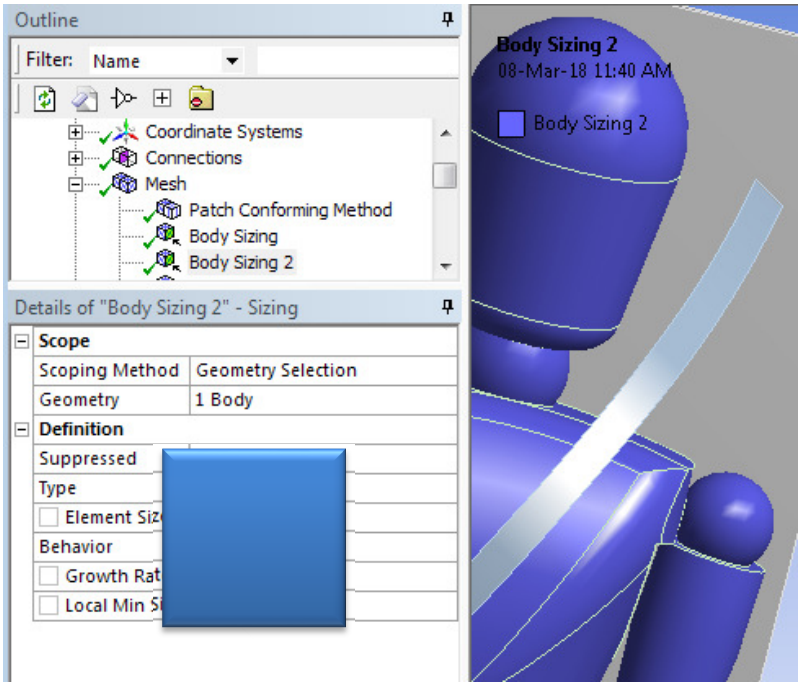
Assign Mesh Control, Method, Tetrahedrons to the seat, blue here.



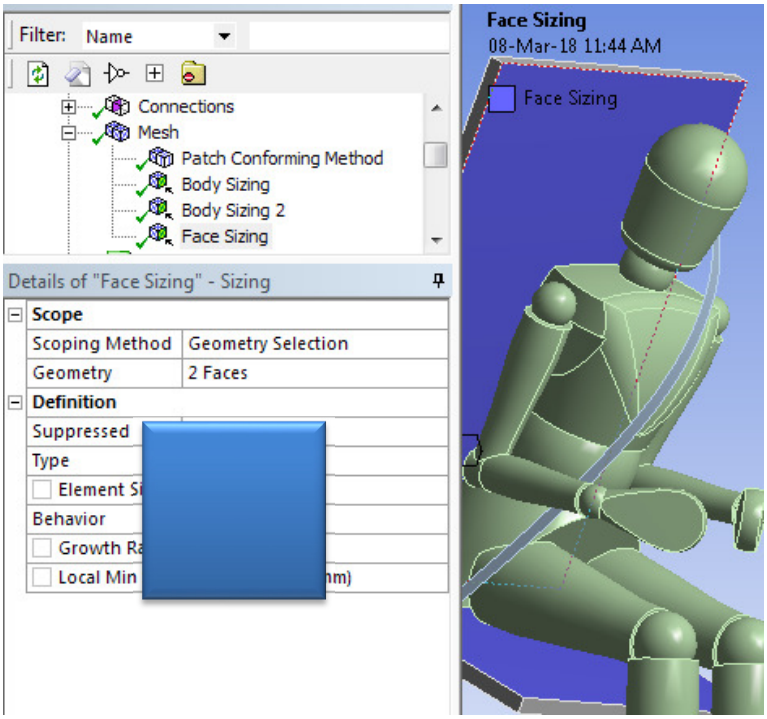
Insert this Mesh Control, Sizing to the seatbelt, blue here.



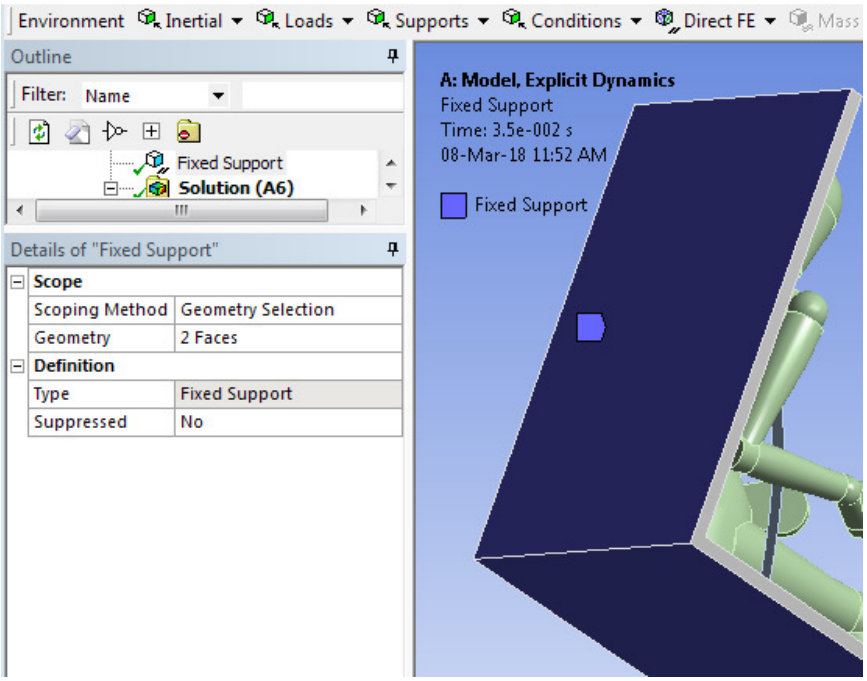
Insert this Mesh Control, Sizing to the dummy, blue here.



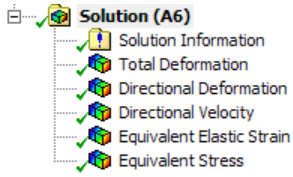
Insert this Mesh Control, Sizing to the inner faces of the seat, blue here.



Analysis Settings: Insert these details. Remember to always use Mass Scaling when dealing with Explicit Dynamics!



Solution: Insert these default items, for all parts.



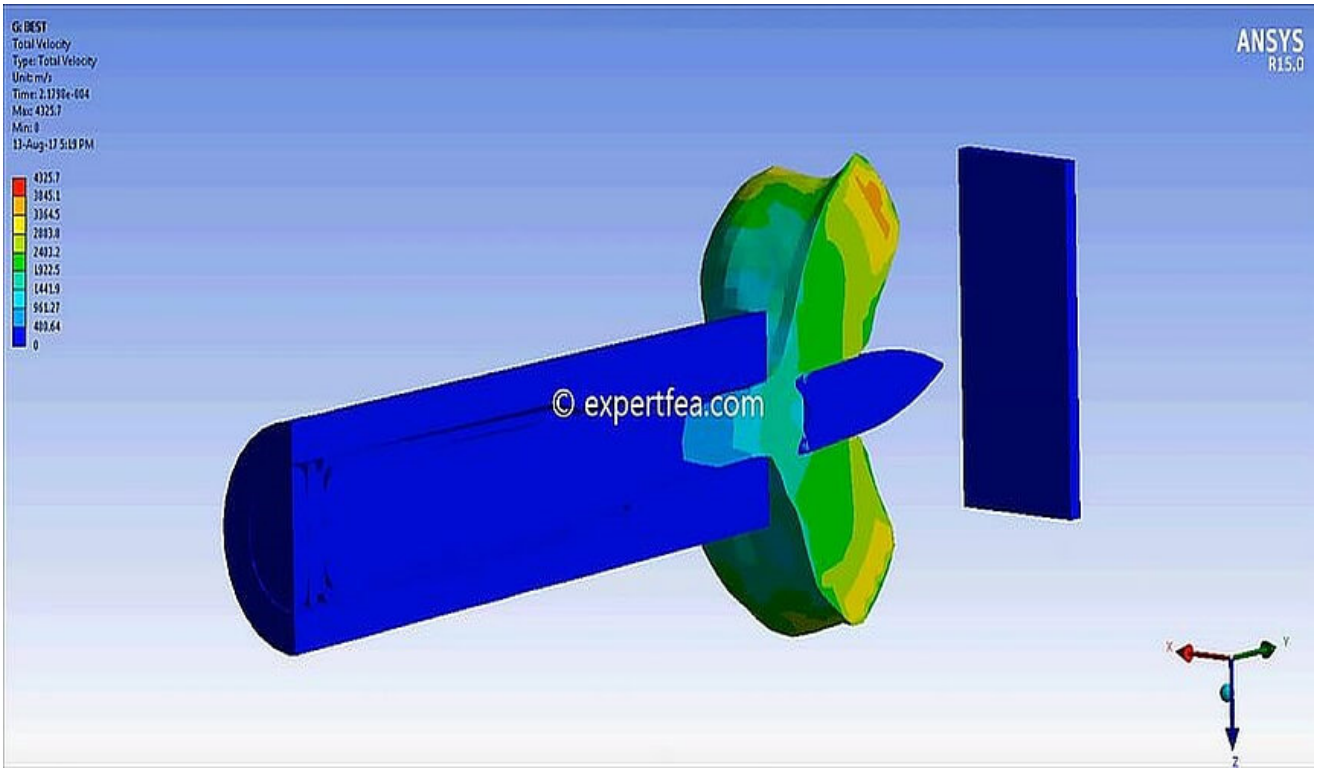
Further homework:

- replace Structural Steel with Structural Steel NL from the Non-Linear Materials library, solve and draw the conclusions

- increase Acceleration to 2000 m/s, solve and draw the conclusions

- make Mass Scaling = Off, solve and draw the conclusions

CASE 44: ANSYS WB FINITE ELEMENT ANALYSIS - Firing of a rifle bullet into a plate



Explicit Dynamics

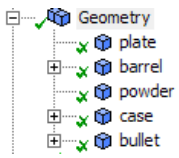
Engineering Data (Materials): Add Copper Alloy from General Materials library and POLYCARB, LEAD and TNT from Explicit Materials, as the ones seen here.

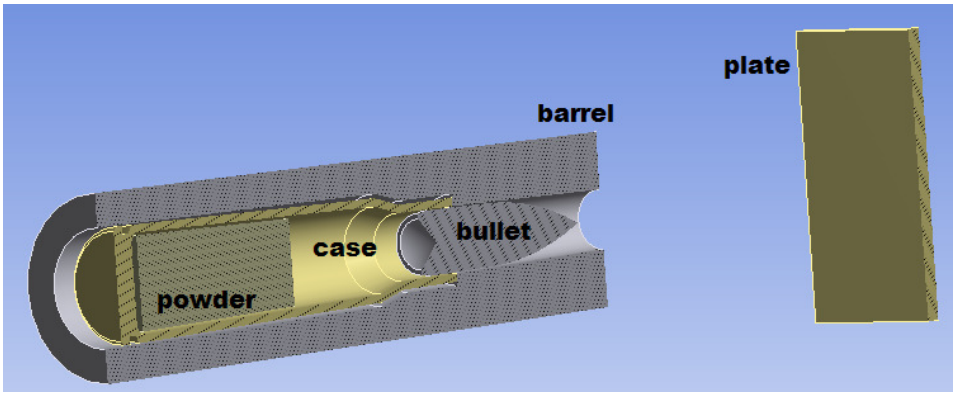
Outline of Schematic A2: Engineering Data

	A	B	C	D	E
1	Contents of Engineering Data			Source	Description
2	Material				
3	[redacted]by				
4	[redacted]				"Equation of State and Strength Properties of Selected Materials". Steinberg D.J. LLNL. Feb 1991
5	[redacted]				S.M. Walley et al. "Strain rate sensitivity of polymers.." DYMAT Journal-vol.1-3-sept 1994
6	[redacted]Steel				Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5-110.1
7	[redacted]				JWL Equations of State Coeffs. for High Explosives Lee Finger & Collins. UCID-16189. January 1763
*	[redacted] a new material				

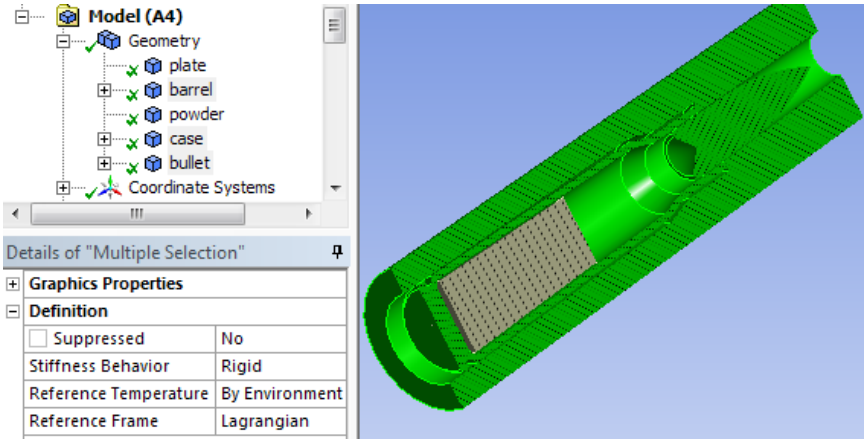
Geometry: 2017_aug_04_bullet_plate_v3.x_t

Assign these names to the parts.

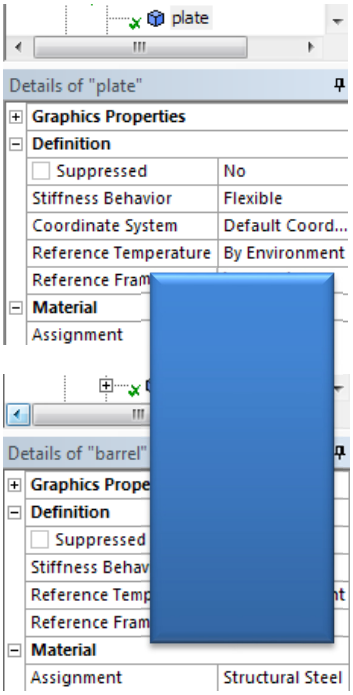




Make barrel, case and bullet, green here, as Rigid.



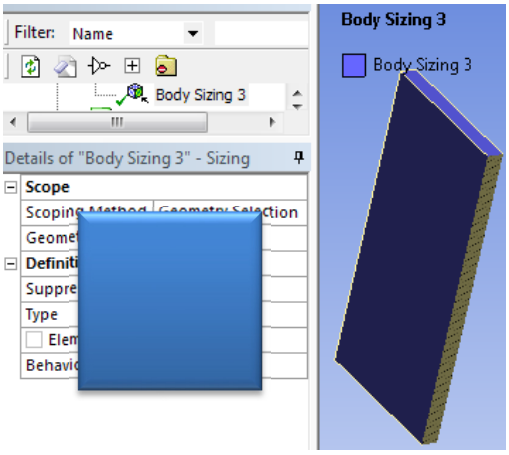
Assign the following materials.

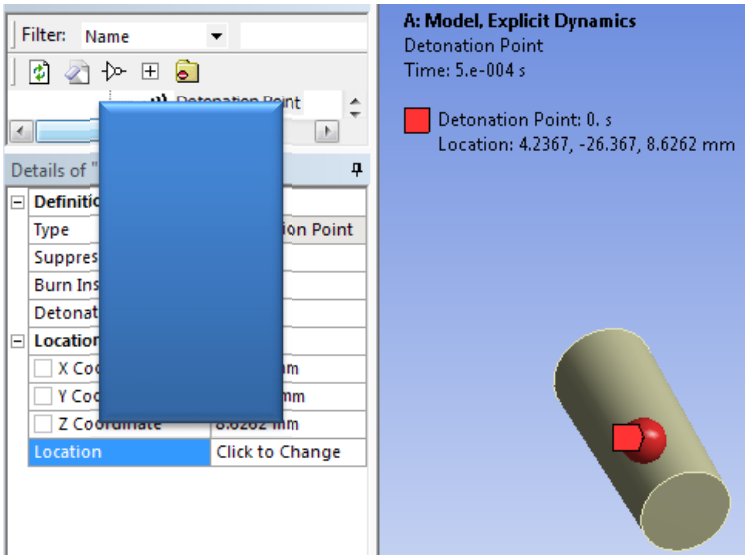


Reference Frame = Eulerian (Virtual)!

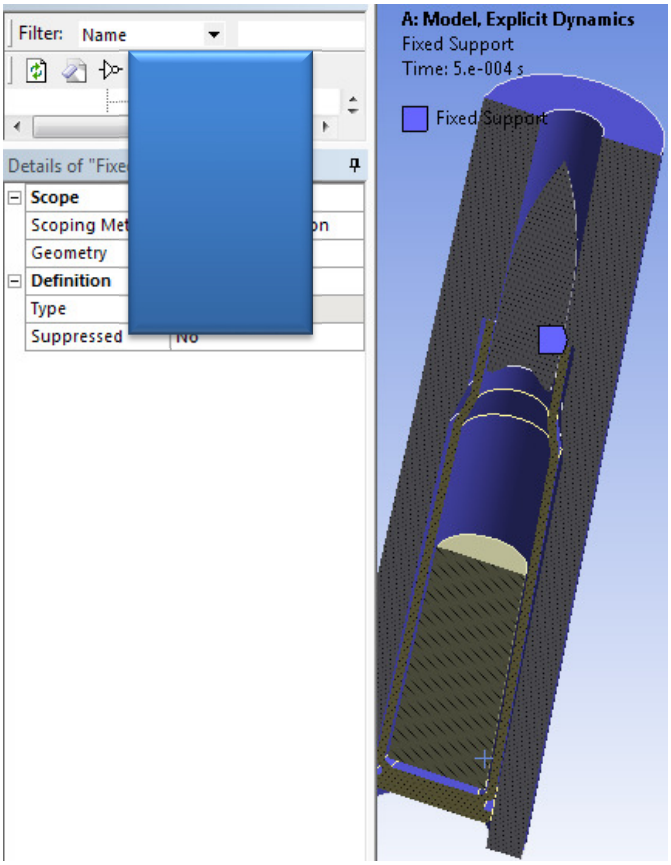
Insert this Hex Method on the barrel and bullet, blue here.

Make the plate mesh as here.

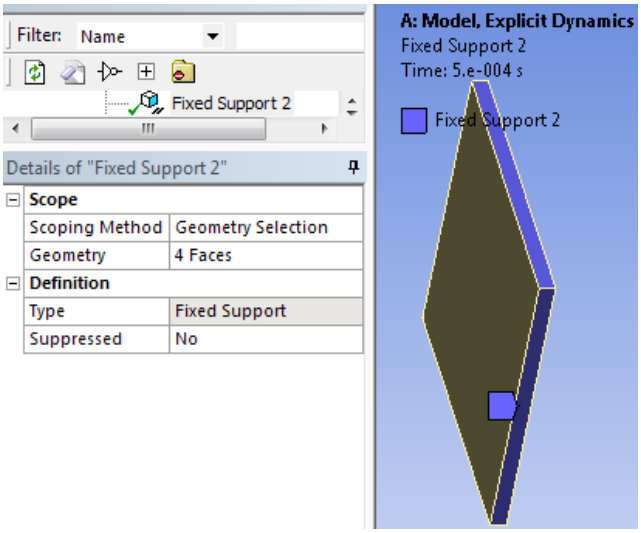




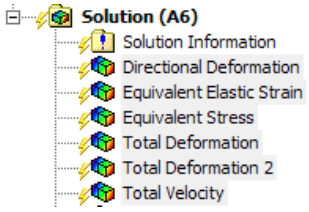
From the Supports toolbar, fix the barrel and the case bodies, blue here.



Do the same for the 4 lateral blue faces of the plate. All 3 bodies couldn't be included in a single Fixed Support item, since they have different Stiffness Behaviors.



Solution: Insert these default items, for all parts.



Solve the FEA. After it is finished, select the Solution branch, and from the Worksheet button, create these User Defined [redacted] names in the Expression column; a sample here.

Expression
EPPLVECTORS
EPPLEQV_RST
MATERIAL
STATUS
EROSION
TYPE
STRAIN_1
EFF_STN
EFF_PL_STNALL
THICKNESS
VISC_PRES
INT_ENERGYALL
TEMPERATUREALL
DAMAGEALL
TIMESTEP

- INT_ENERGYALL
- MATERIAL
- STATUS
- VISC_PRES
- INT_ENERGYPOWDER
- MASSPOWDER
- COMPRESSPOWDER
- TEMPERATUREPOWDER
- ALPHAPOWDER
- SOUNDSPEED
- TIMESTEP
- MASS_SCALE
- PRESSURE

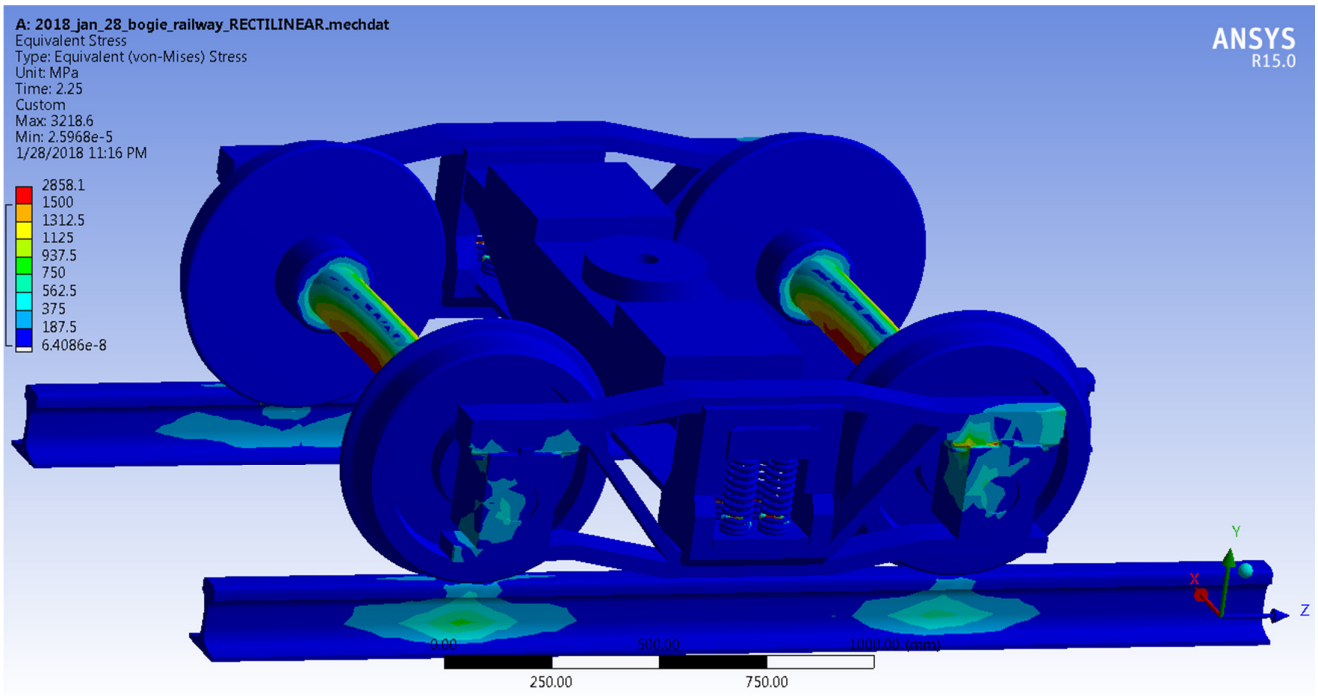
Further homework:

- replace POLYCARB with Polyethylene from the General Materials library, solve and draw the conclusions

- suppress the Hex Dominant method, solve and draw the conclusions

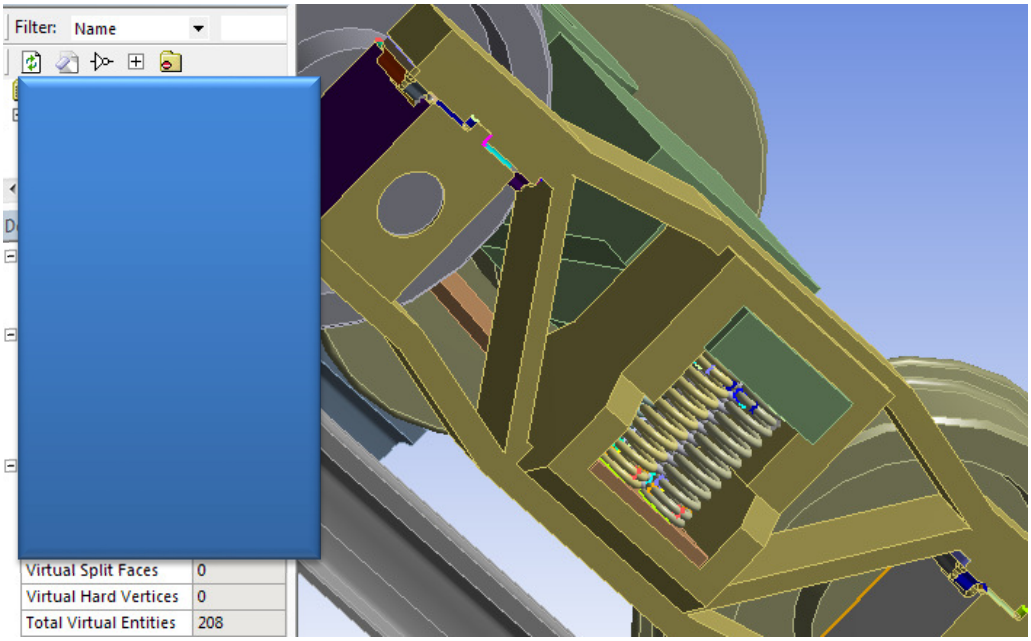
- in Analysis Settings, make | End Time | 1.e-003 s |, **solve and draw the conclusions**

CASE 45: ANSYS WB FINITE ELEMENT ANALYSIS - Train bogie travel with rotation of wheels on tracks



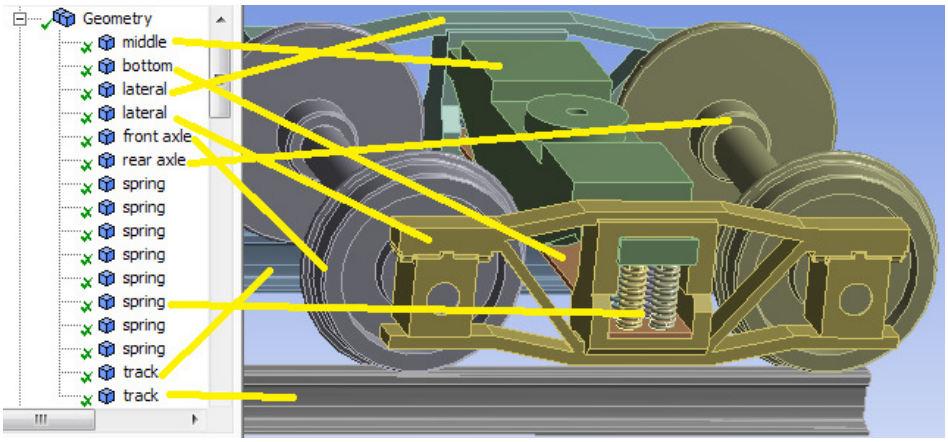
Transient Structural

Model: Insert a Virtual Topology.

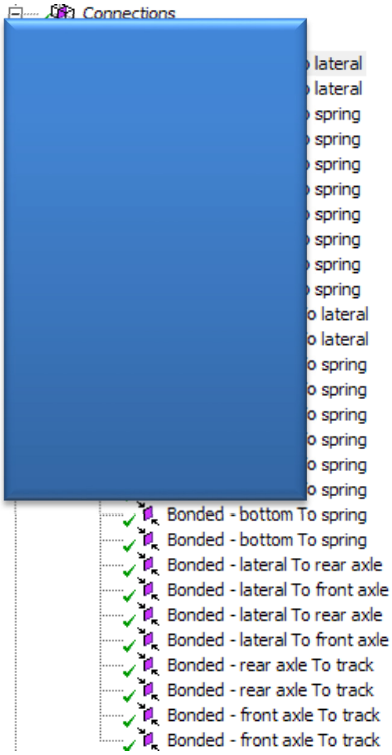


Geometry: 2018_jan_10_bogie_railway_v5.x_t

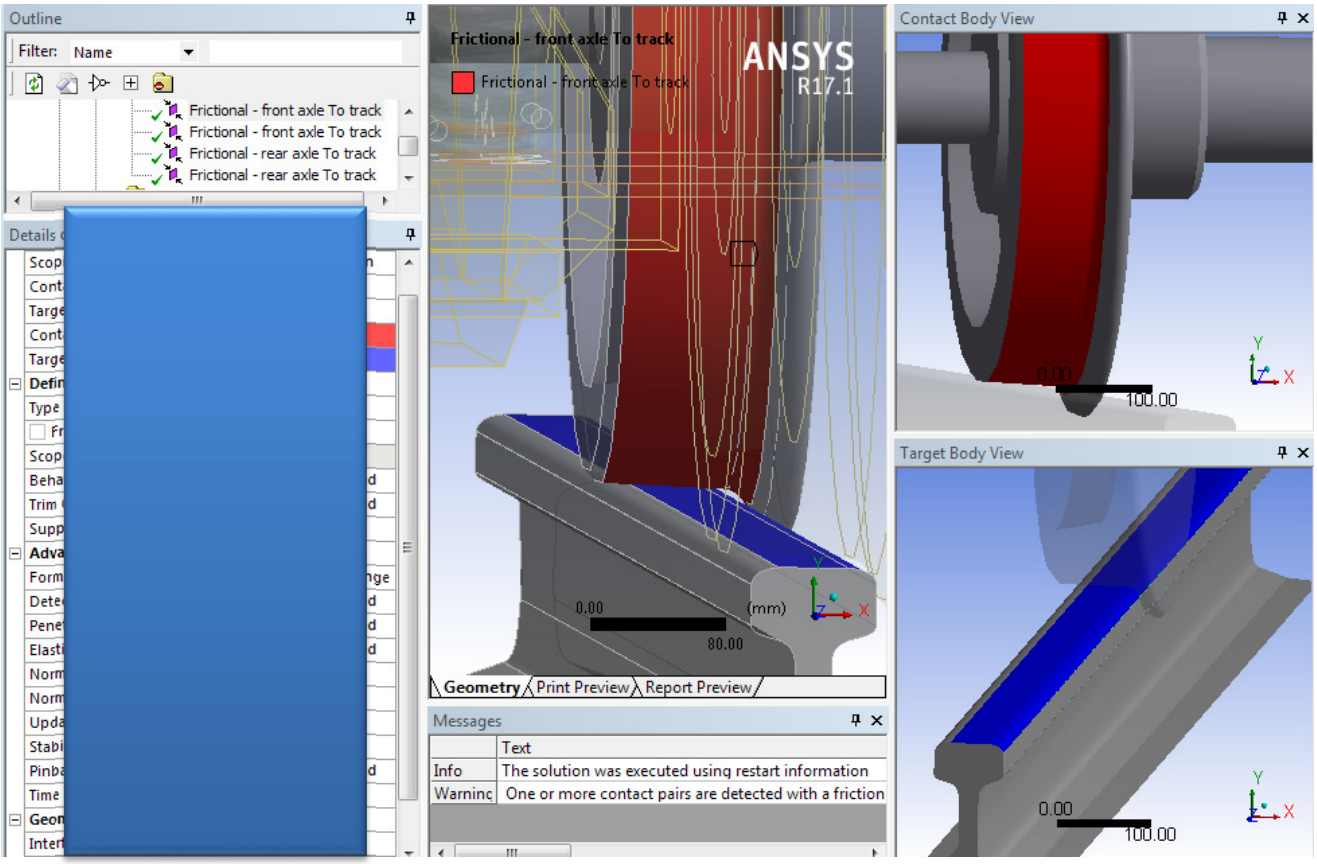
Assign these names to the parts.



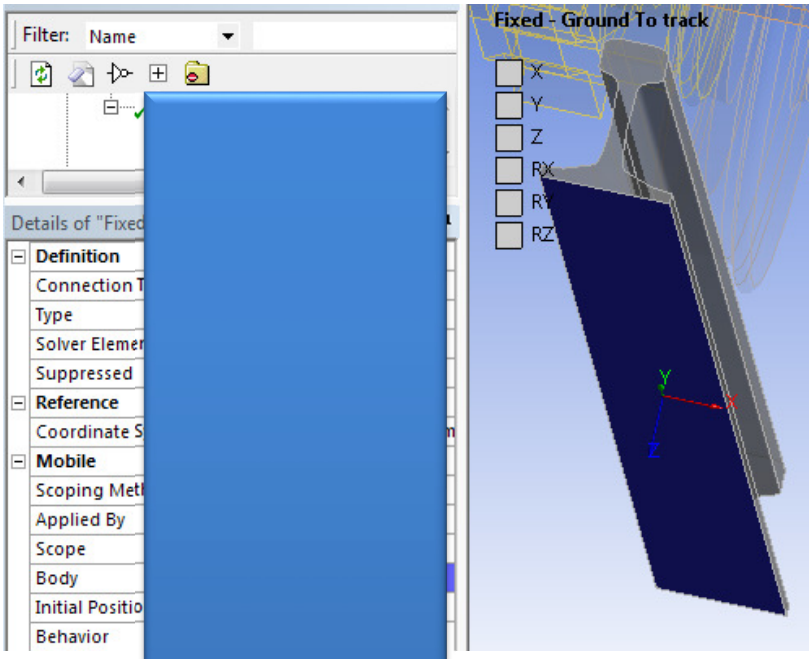
Contacts: Right click, Rename Based on Definition. They should look like here.



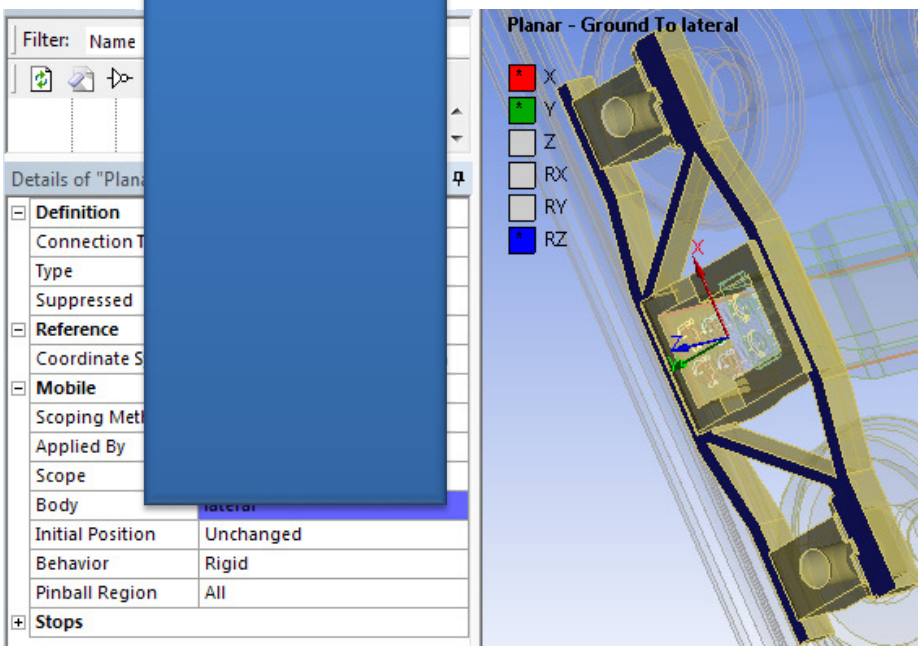
Make Formulation for all contacts as MPC. Delete middle To lateral contacts.



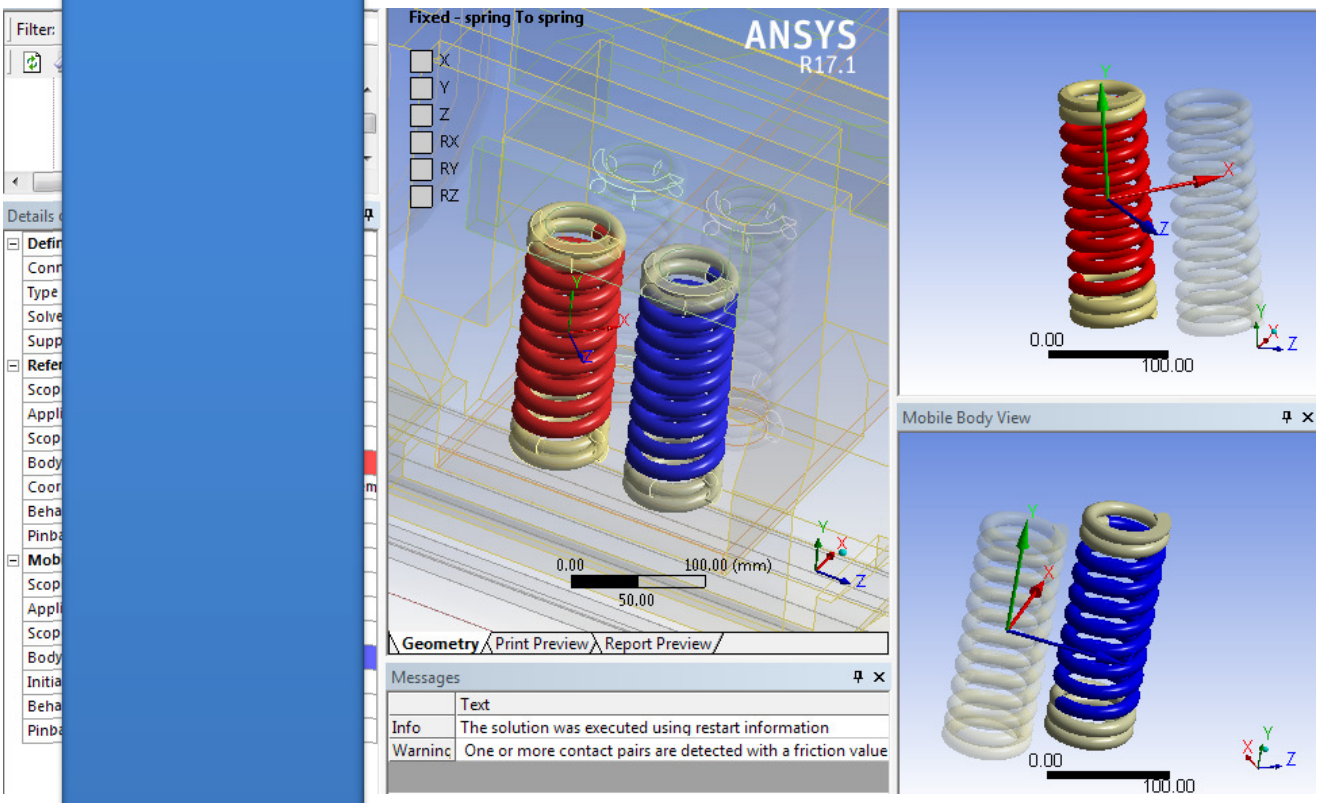
Joints: Body-Ground. Add this Planar Joint by selecting all faces of the middle. Observe the X axis parallel to the movement direction.



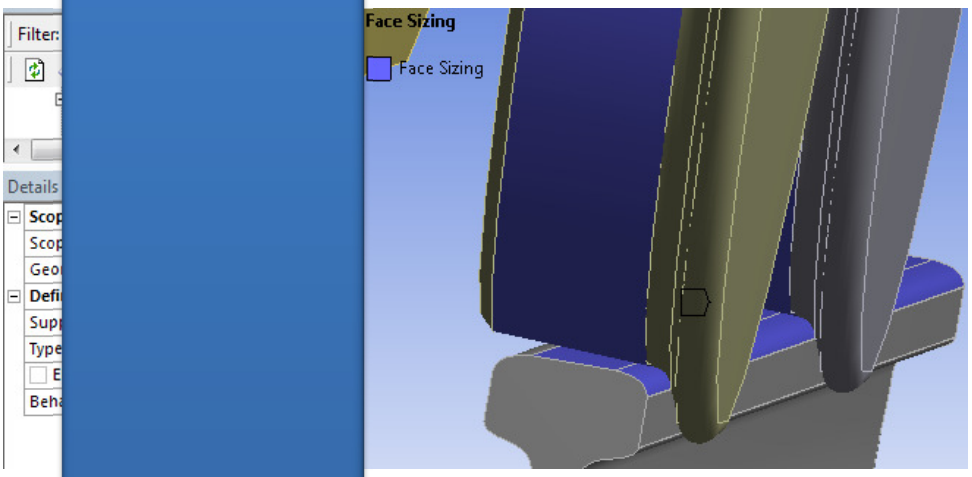
Assign 2 Plan ts, selecting the outer blue faces.



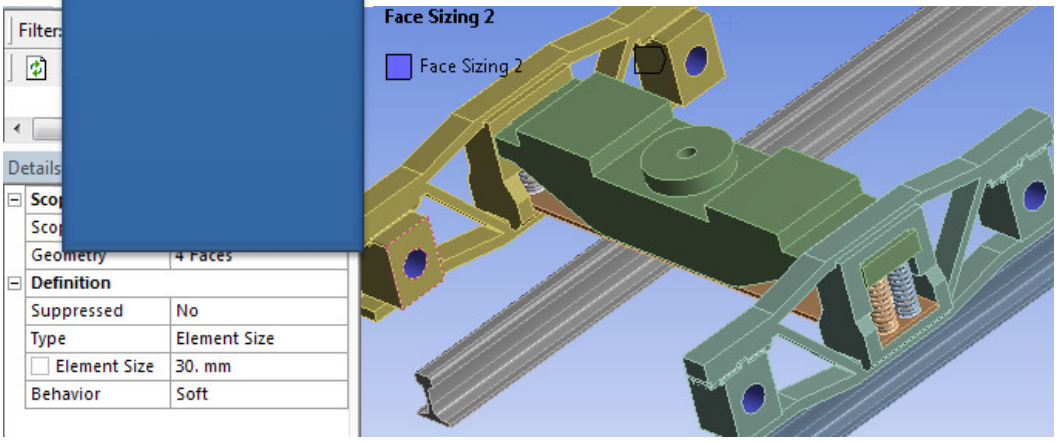
Body-Body. Add 2 Fixed Joints between each lateral-middle pair. Select also the opposite faces (invisible here).



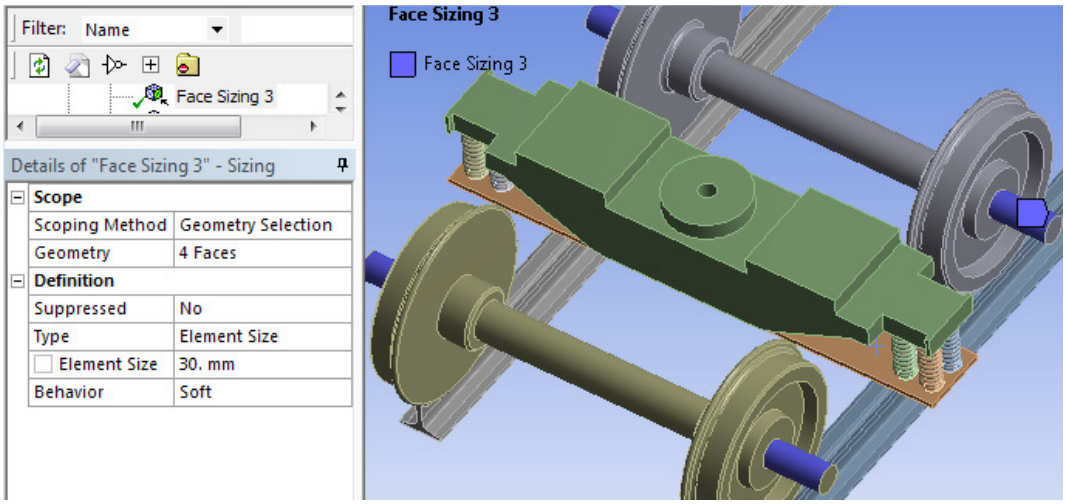
Mesh aces seen in blue here, also for the other side of the track.



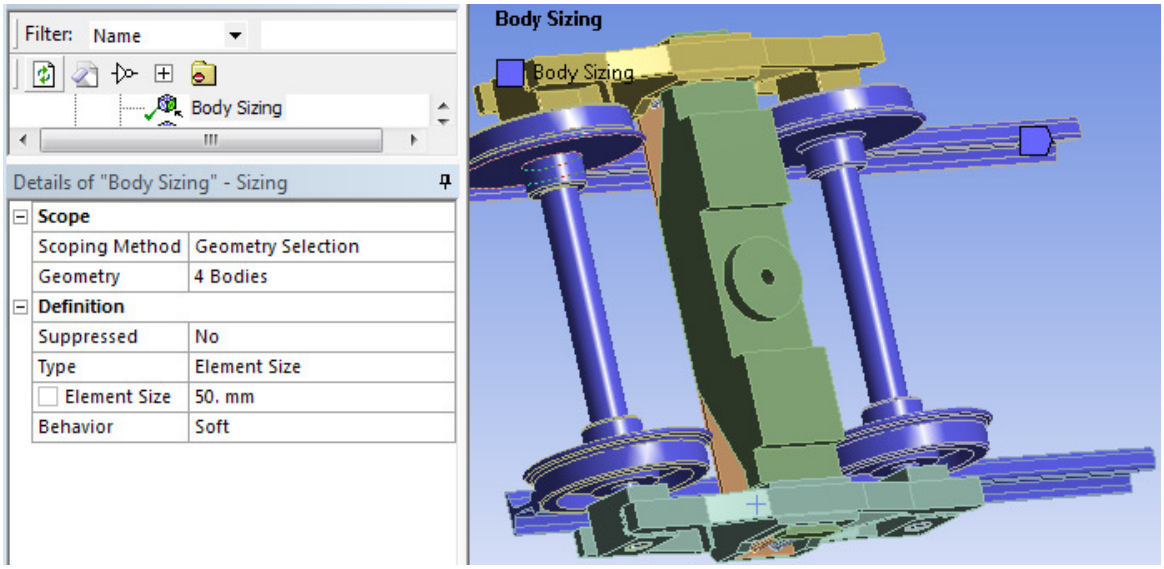
Add t e lateral parts, blue here.



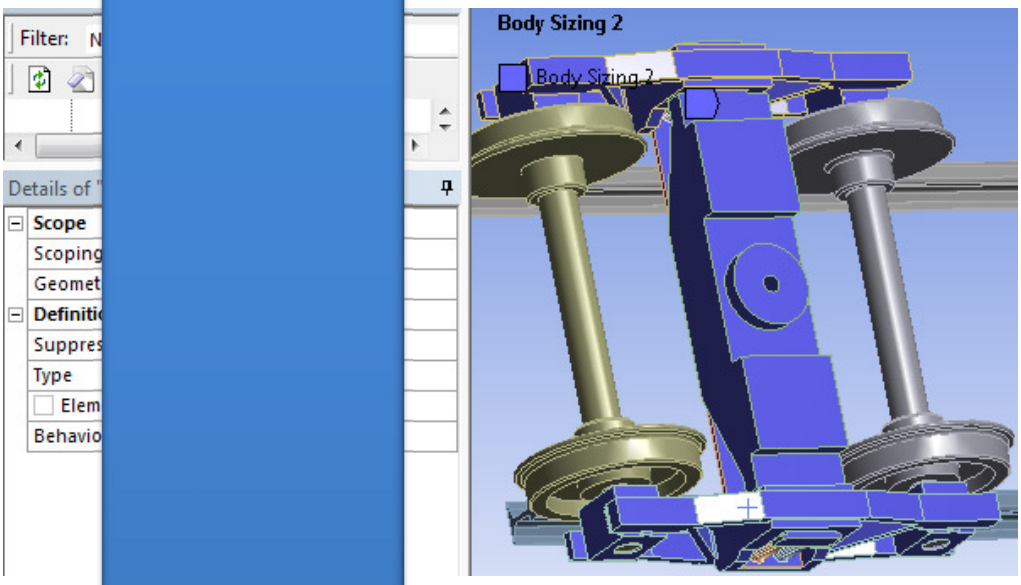
Create this sizing for the end shafts, blue here.

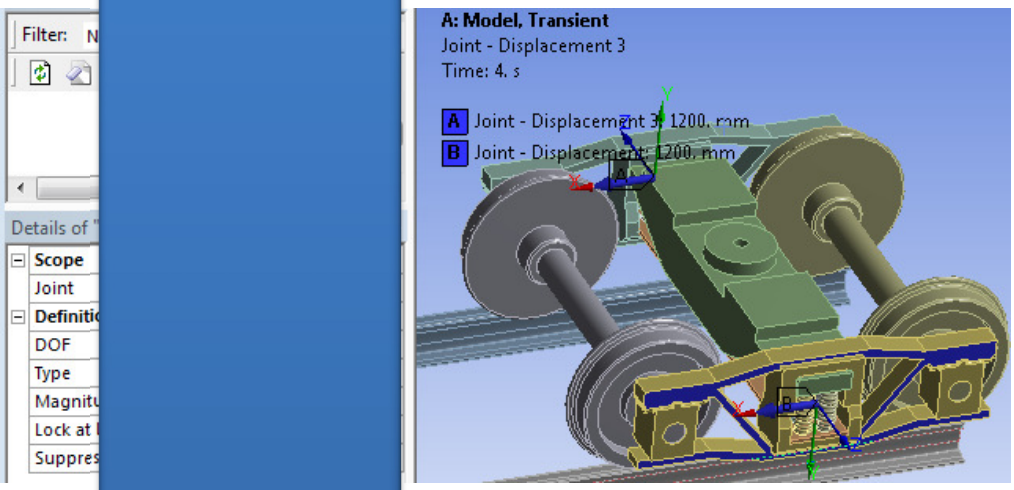


Create this sizing for the bodies seen here in blue.

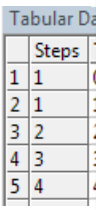


And assign this sizing for the other bodies, seen here in blue.

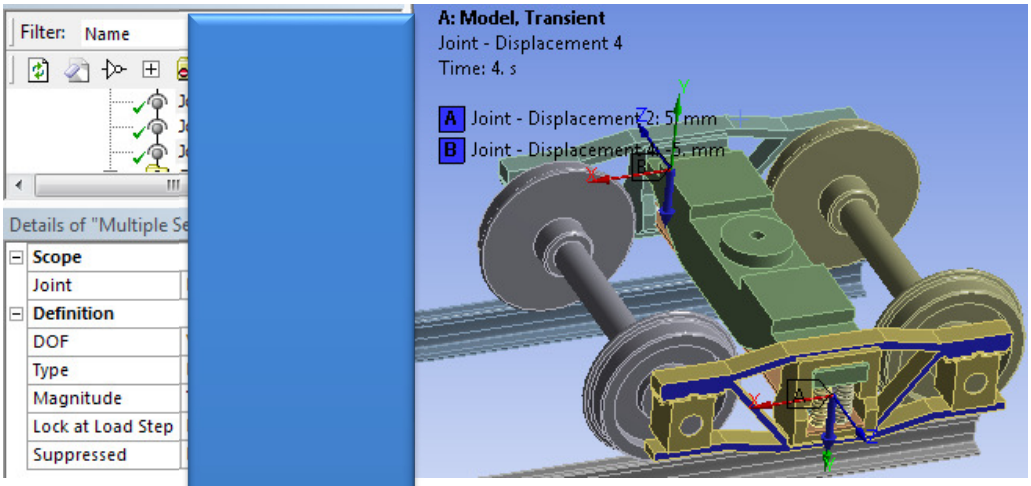




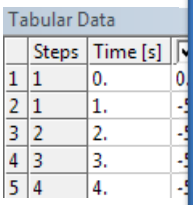
Insert th .



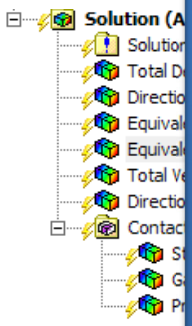
From Loa 2 items on Y axis, for the same joints.



To create friction[illegible]e wheels, press them down with these displacements.



Solution: Insert t[illegible]for all parts. Feel free to select-unselect them as you wish, to scope only for certain parts.



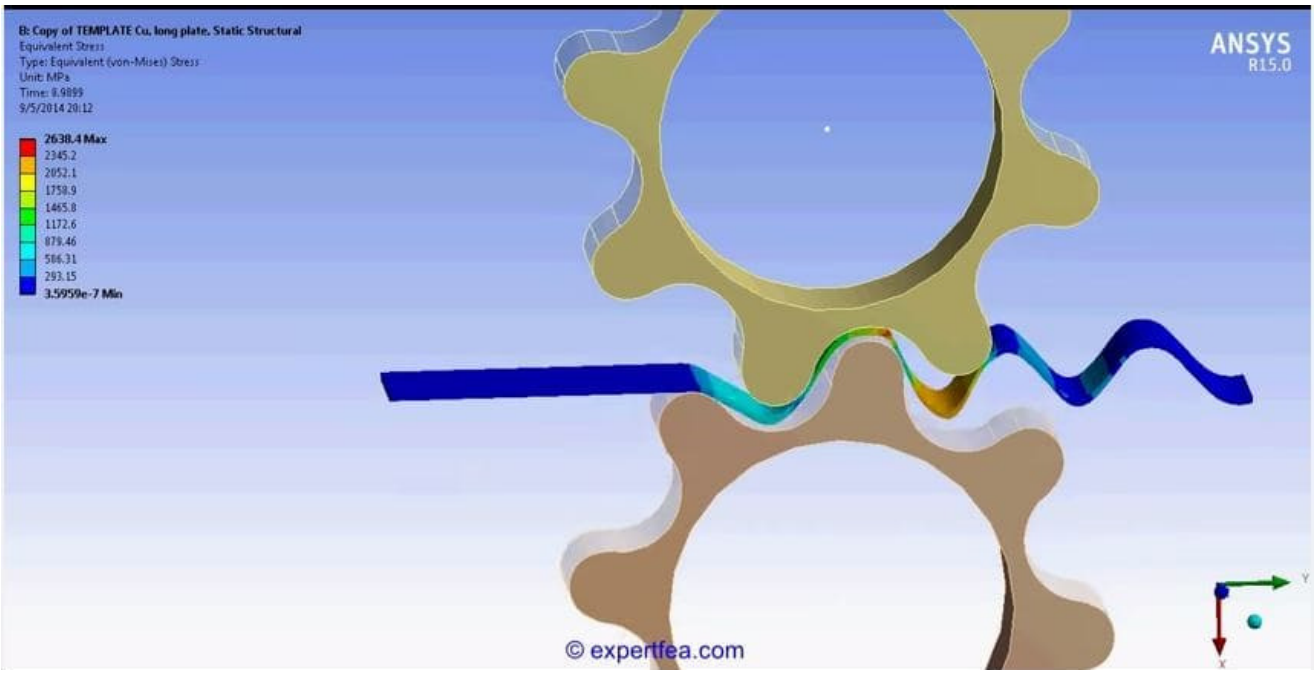
Further homewo[illegible]

- delete the Virt[illegible]and draw the conclusions

- lower to 0.2 th[illegible]oefficient of all 4 Frictional - front axle To track **, solve and draw the conclusions**

- in Analysis Settings, make Stabilization = Off, solve and draw the conclusions

CASE 48: ANSYS Workbench Static Structural FEA of a copper plate corrugation between sprockets

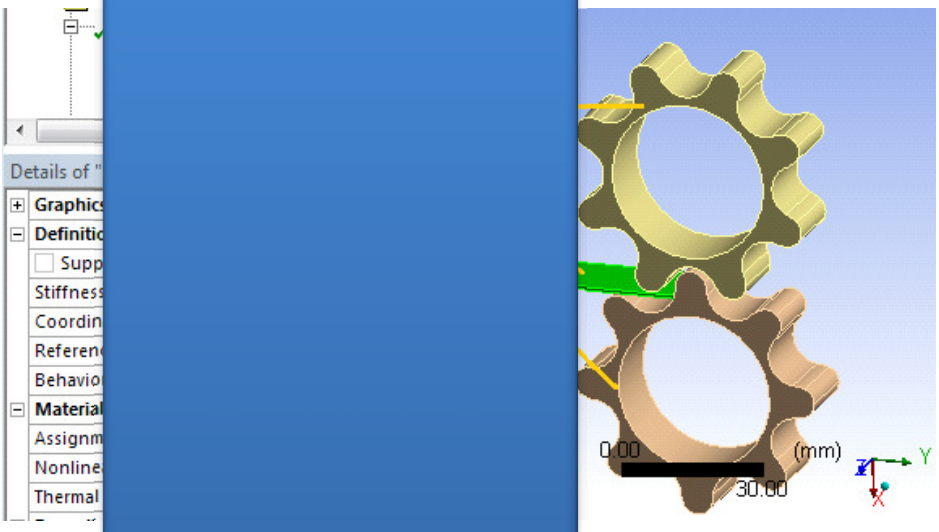


Static Structural

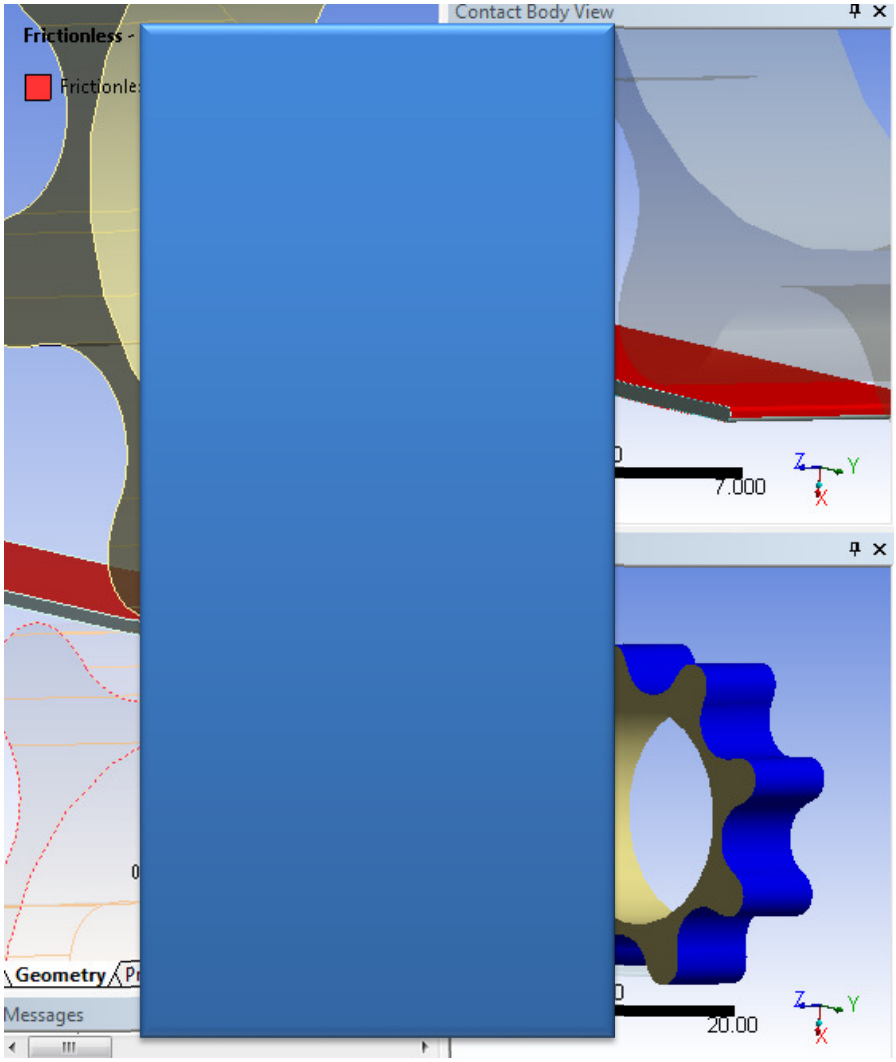
Engineer[redacted]y, insert Copper Alloy NL.

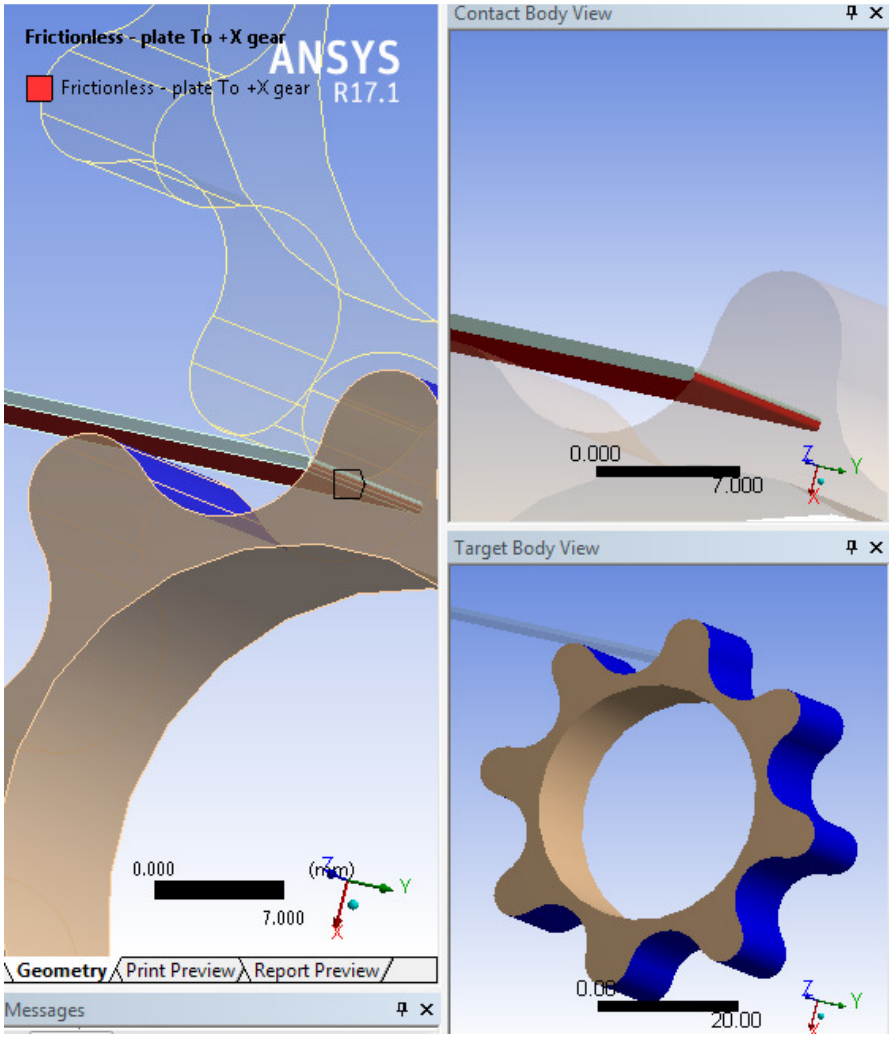
Geometr[redacted]t

Rename [redacted] material to the plate, green here. Make the gears Rigid.

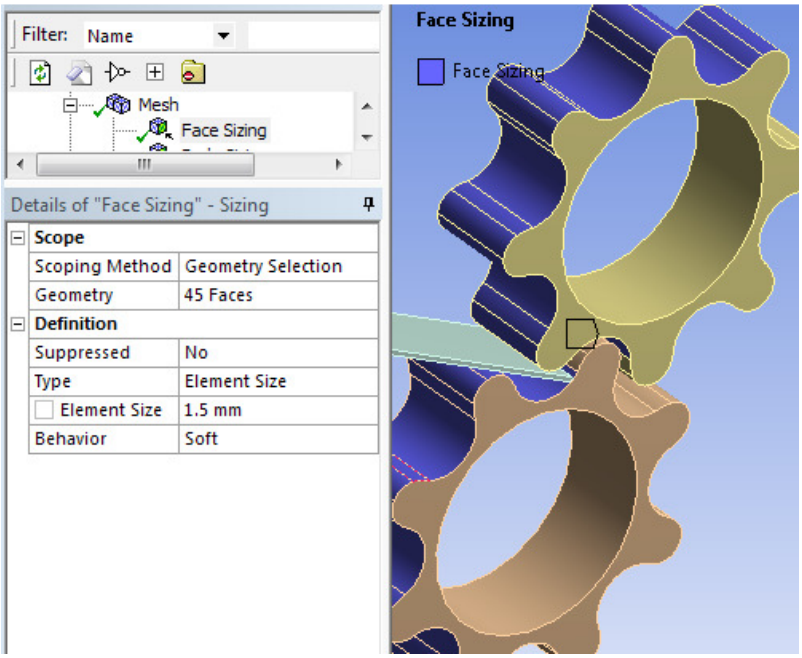


Coordina[redacted]oordinate System button. Apply the Y Offset = -15 mm.

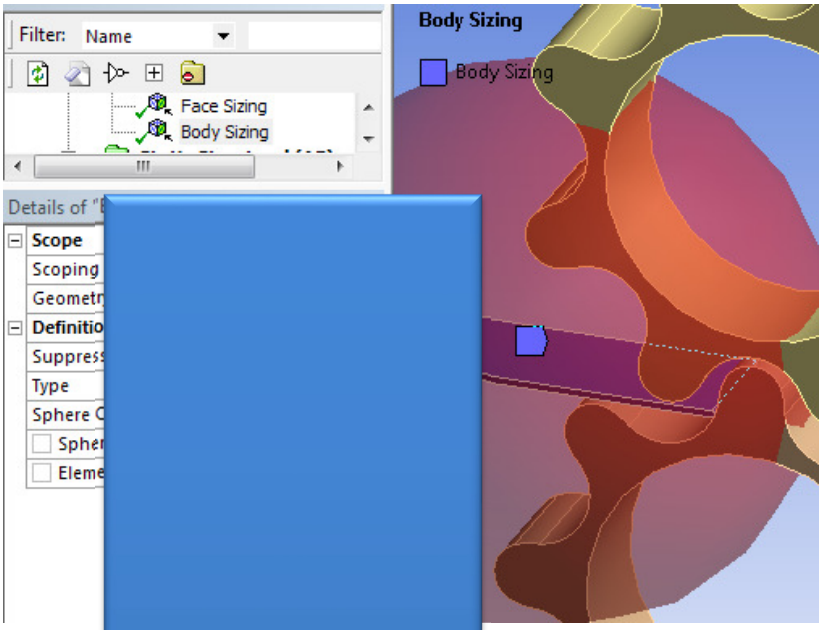

Contact Body View
Frictionless -
7.000
20.00
Geometry
Messages



Mesh: Define this Face Sizing for the blue faces, seen here.

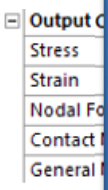


Create this Body Sizing for the plate, blue here. For Sphere Center, use the Coordinate System you created before.

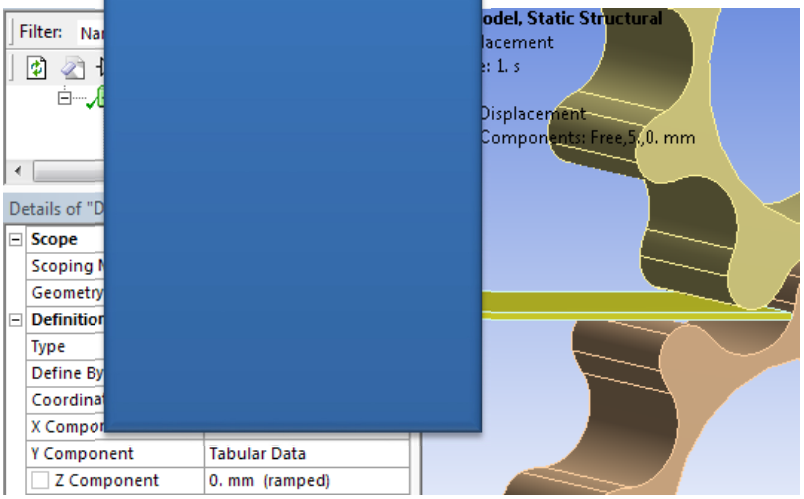


Analysis S l 10 steps; you can select them from the Graph tab.

Select Ye



Environm faces of the plate part, yellow here.

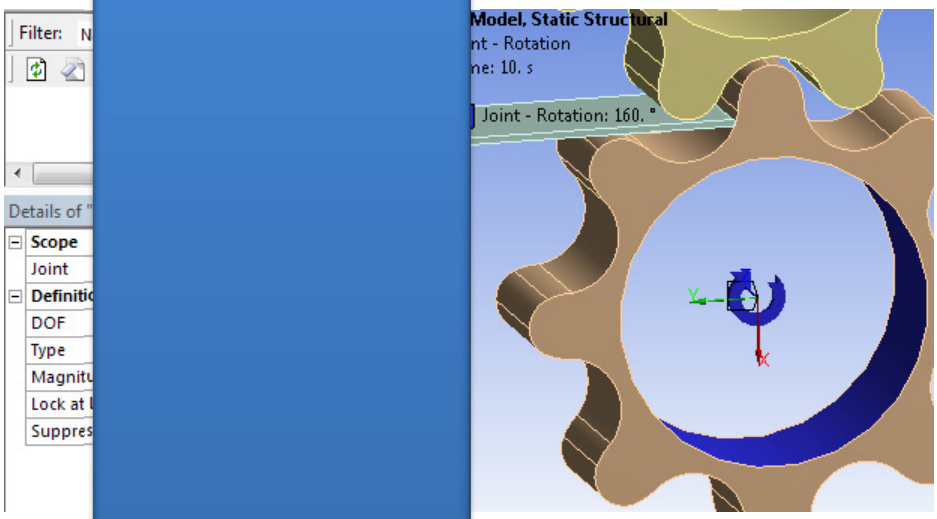


Insert these values on the Graph tab.

Tabular Data

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

As Joint

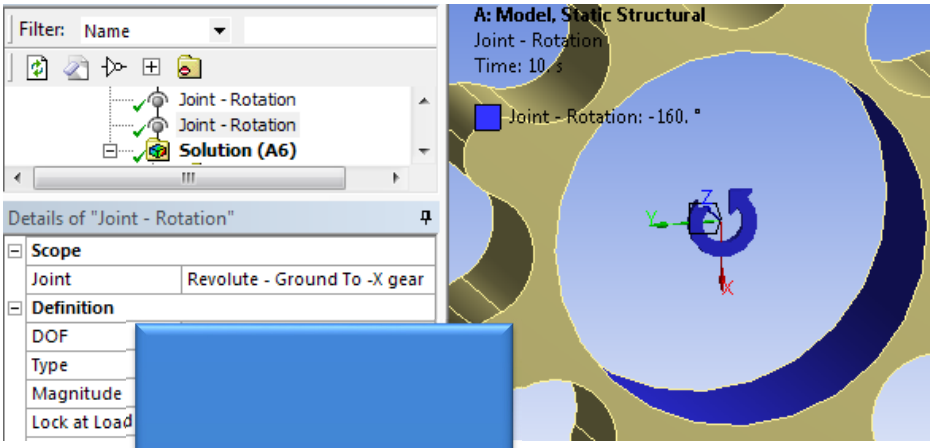


Use thes

Tabular Da

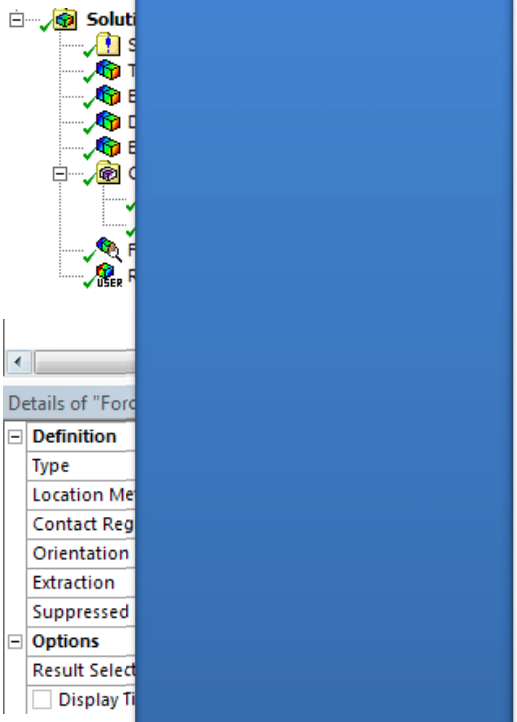
	Steps		
1	1		
2	1		
3	2		
4	3		
5	4		
6	5		
7	6		
8	7		
9	8		
10	9	9.	= 143.
11	10	10.	160.

Create another Joint Load for the other gear. Watch the rotation direction so that the gears mesh one with each other.



Use these v

Solution: Ins rts. The RZ item is available from Worksheet, in the Expression column, after the so



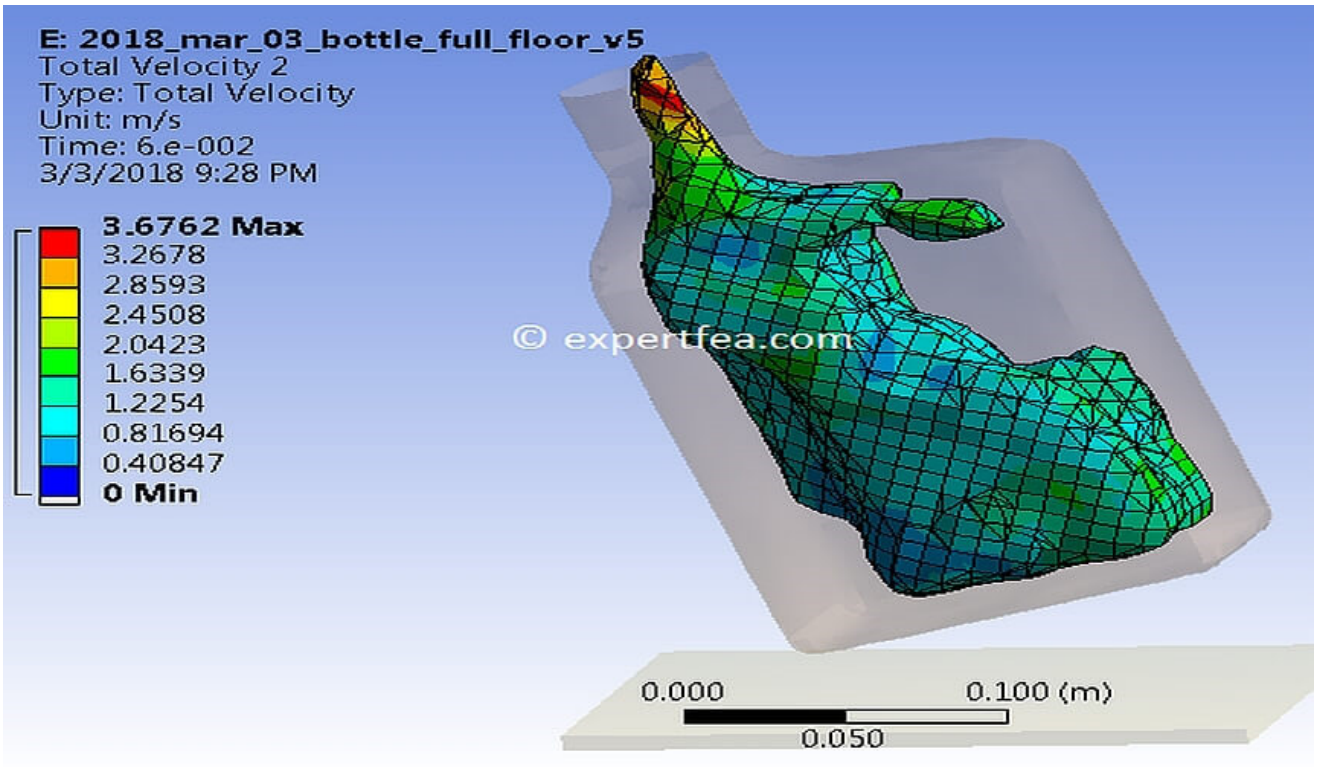
Further hon

- change th um Alloy NL, solve and draw the conclusions

- make the g e, solve and draw the conclusions

- turn off th e conclusions

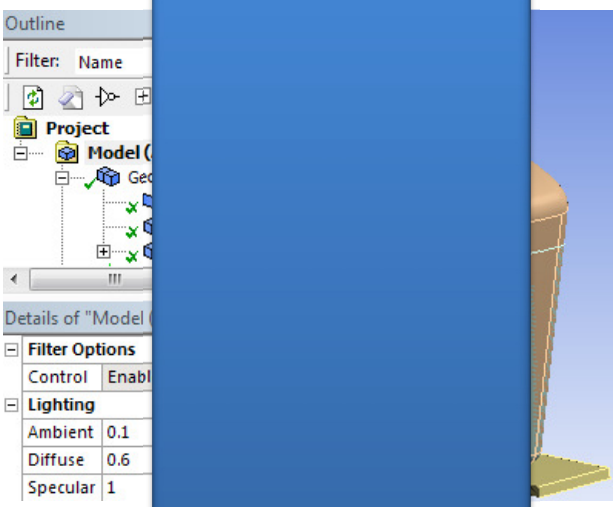
CASE 51: ANSYS WB EXPLICIT DYNAMICS WITH FLUIDS - Drop test of a recipient with water from 2 meters height



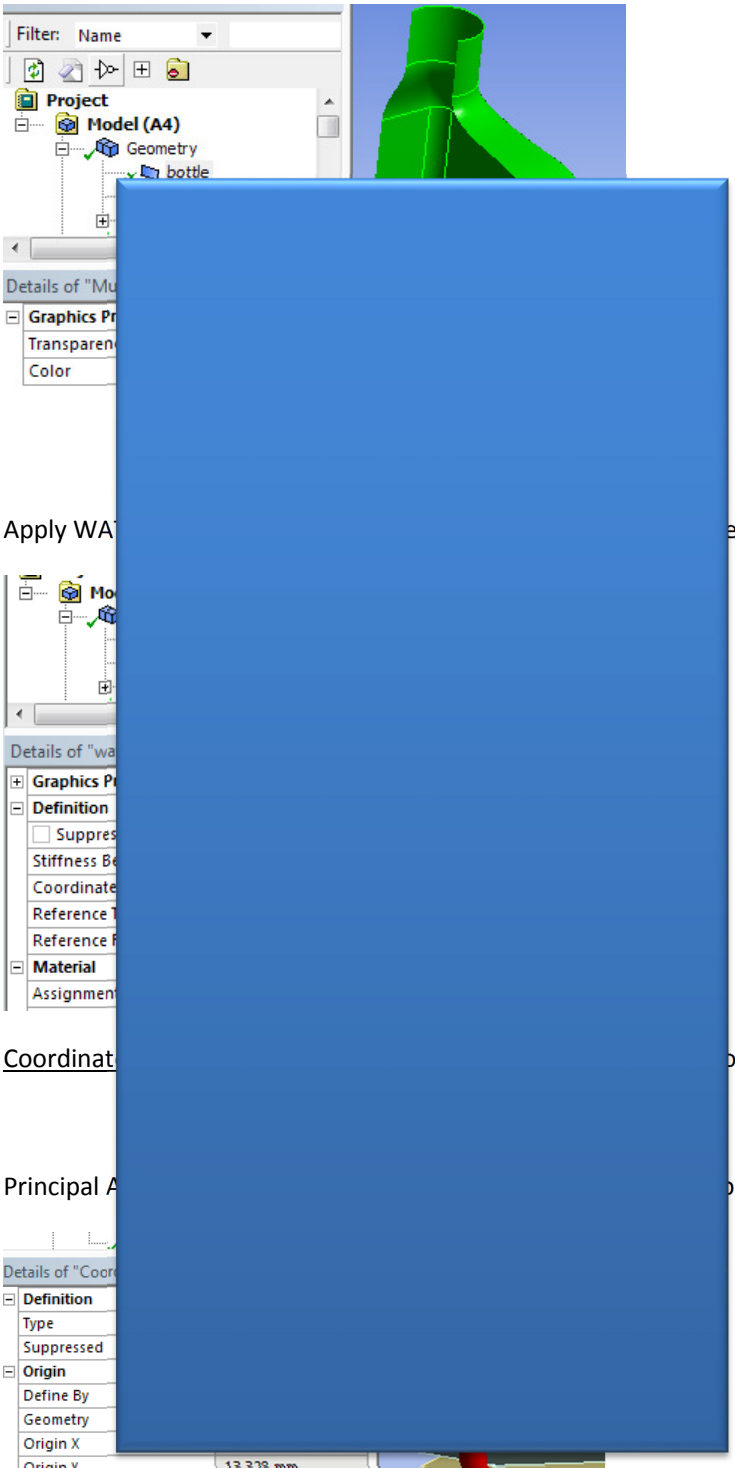
Explicit Dynar

Geometry: 20 x_t

Rename the p



Bottle and flo om Polyethylene. Bottle's thickness is 0.6 mm. Make the floor Rigid.

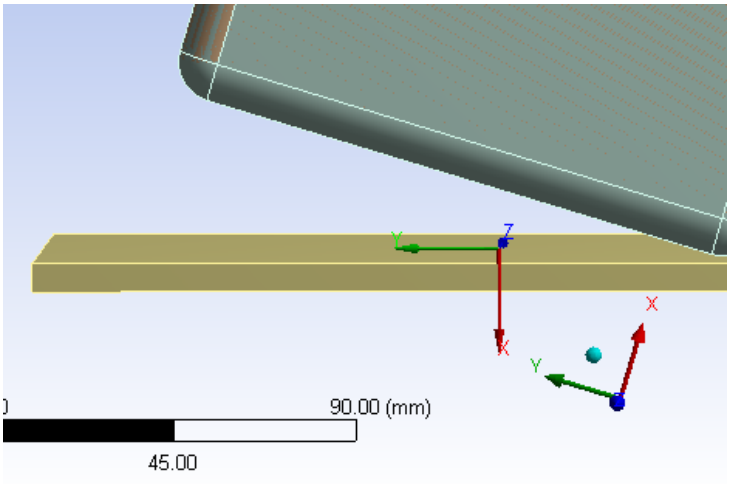


Apply WA eference Frame = Eulerian (Virtual).

Coordinat oordinate System button.

Principal A ly.

In relation to the Global Coordinate System, the new Coordinate System (sticking to the floor) should look like here.

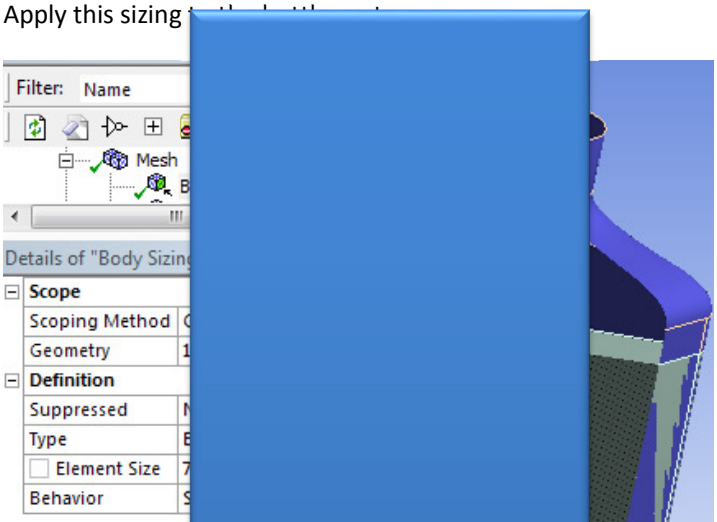


Contacts: Delete them all.

Mesh: Make

Smoothing	Low
Transition	Fast

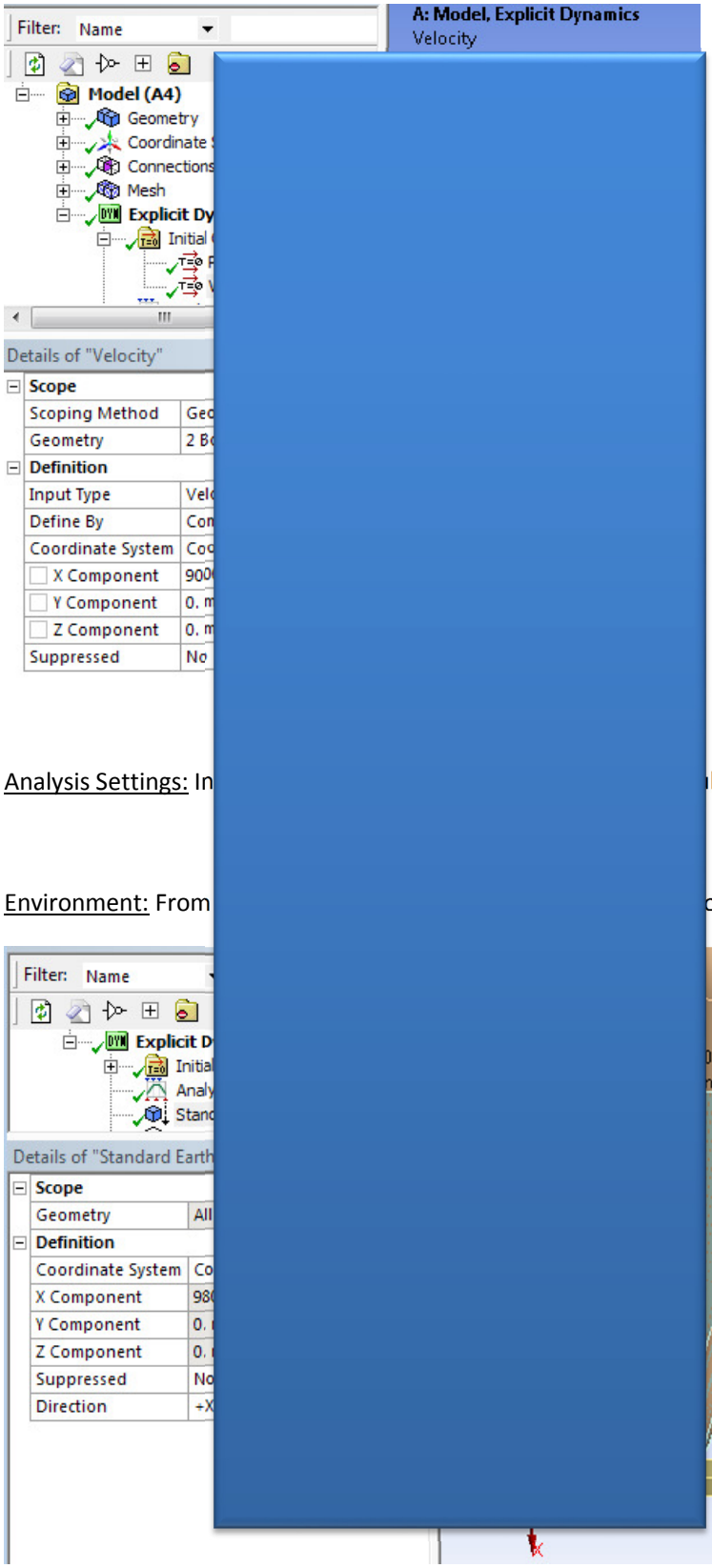
Apply this sizing



Define this sizing

Initial Conditions

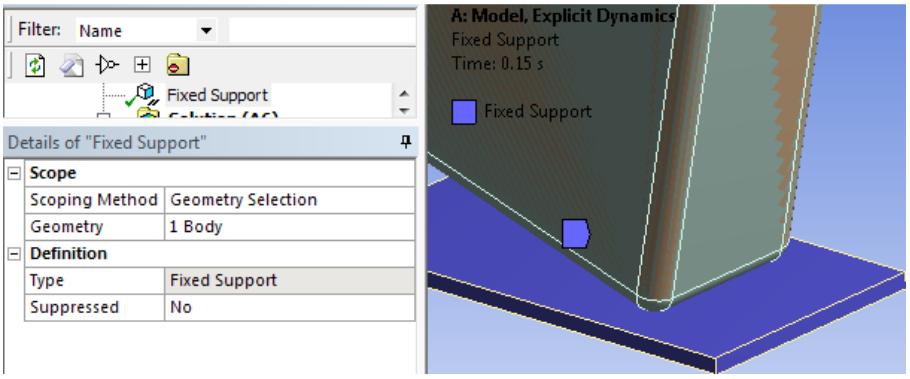
nd water, blue here.



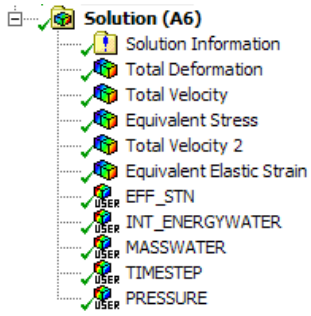
Analysis Settings: In lerian domain.

Environment: From ordinate System we created before.

Fix the floor, blue here.



Solution: Insert these default items, for all parts. The items in CAPS LOCK are available from Worksheet, after the solving is finished, from Expression column.



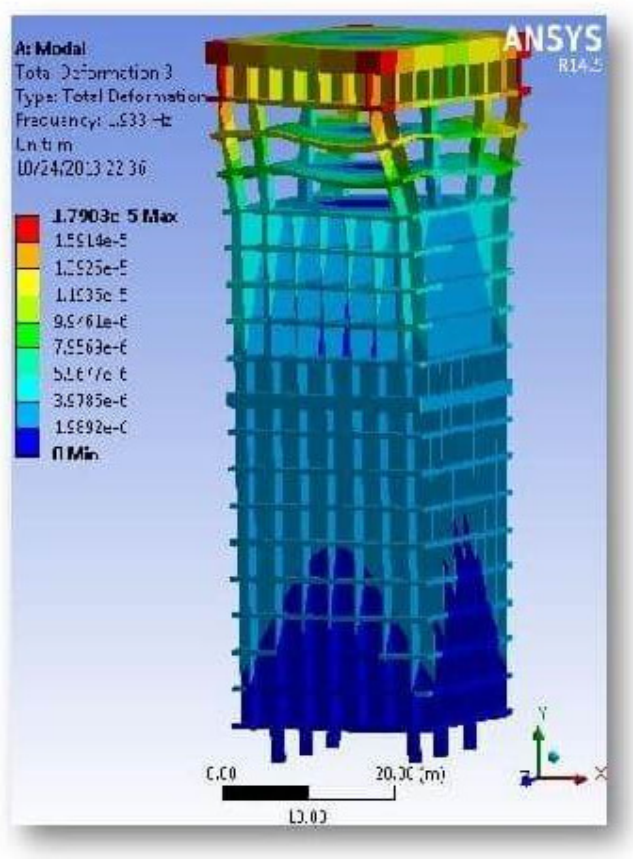
Further homework:

- make bottle's thickness 0.8 mm, solve and draw the conclusions

- in Euler Domain Controls, make this change, solve and draw the conclusions

- in Connections branch, modify as seen here, solve and draw the conclusions

CASE 54: ANSYS Workbench Seismic Random Vibration analysis using PSD on a real size skyscraper



Modal

Engineering Data: Insert Concrete from General Materials library.

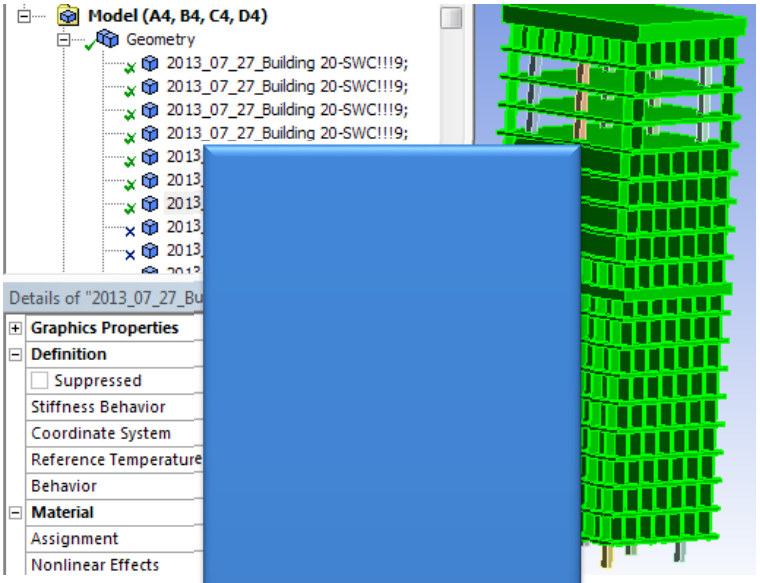
3	General Materials		General use material samples for use in various analyses.

Outline of General Materials

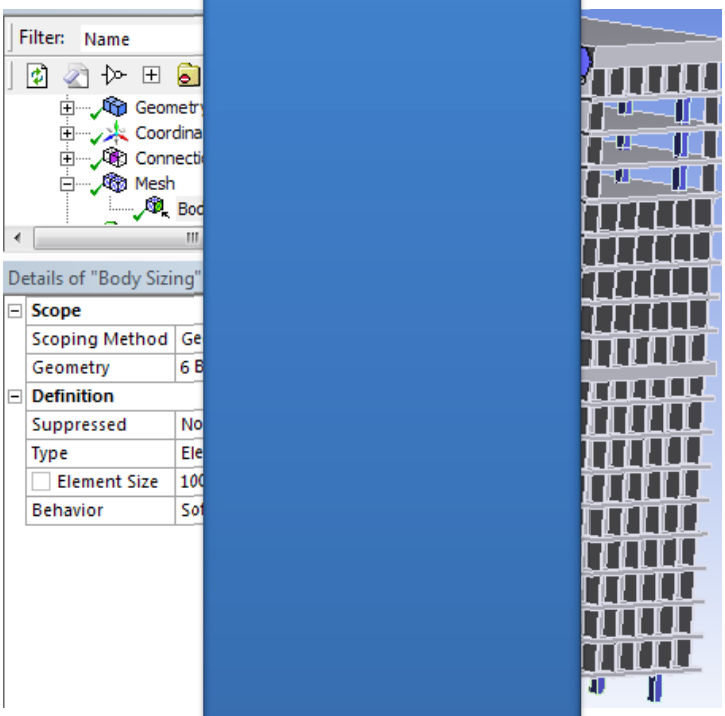
	A	B	C	D	E
1	Contents of General Materials	Add		Source	Description
2	Material				
3	Air				General properties for air.
4	Aluminum Alloy				General aluminum alloy. Fatigue properties come from MIL-HDBK-5H, page 3 -277.
5	Concrete				

Geometry: 2013_07_27_Building 20-SWC!!!9.x_t

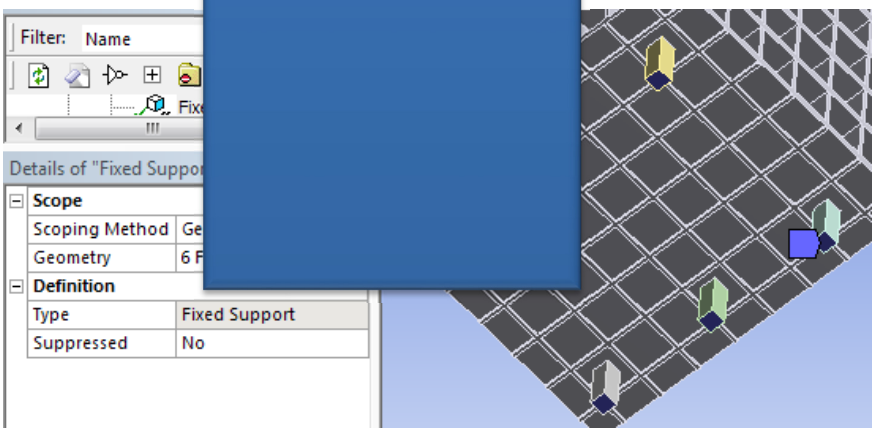
Except the first 7 parts, suppress the others. Apply Concrete material to the green part seen here.



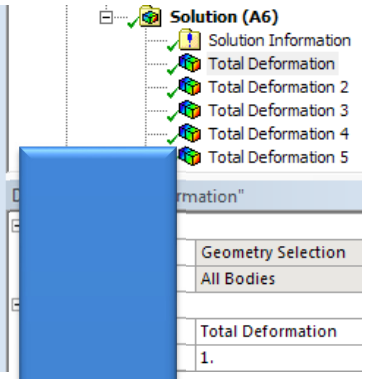
Mesh: Apply this 1 embers. The building has real dimensions.



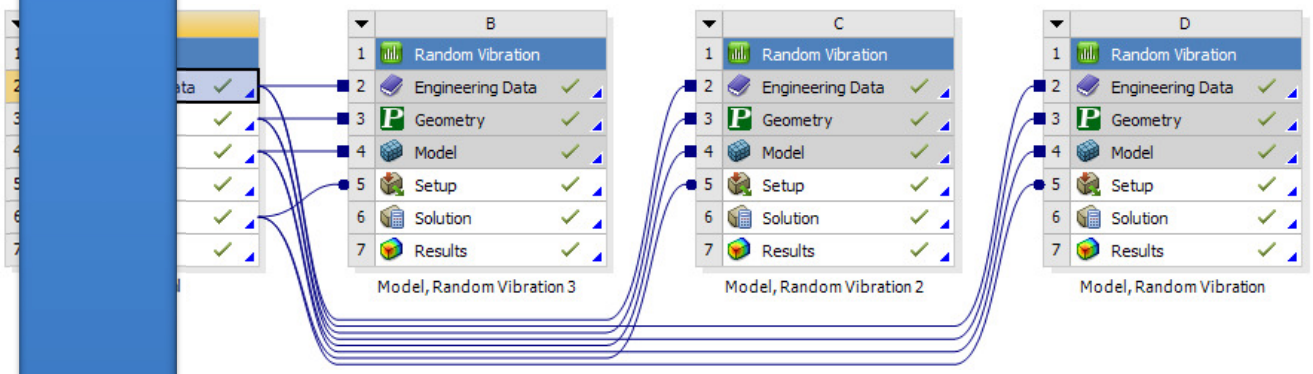
Environment: Fix t mns, seen blue here.



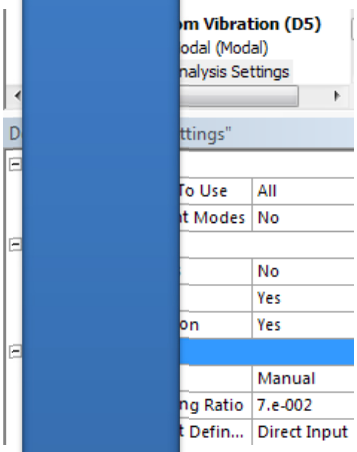
Solution: Insert Total deformation items and change their mode number.



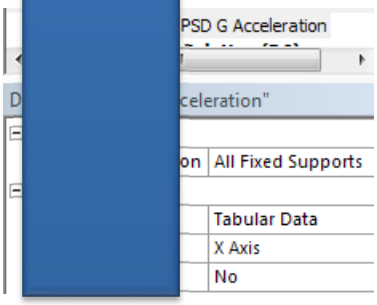
D ibration onto the Solution cell of the Modal scenario. For each axis, they should look like here.



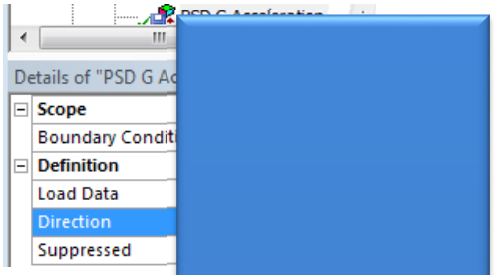
A s: Apply these details for all 3 scenarios.



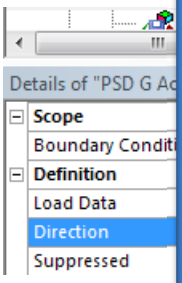
E efine this PSD G Acceleration for each axis.



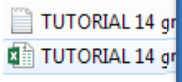
For the next scenario.



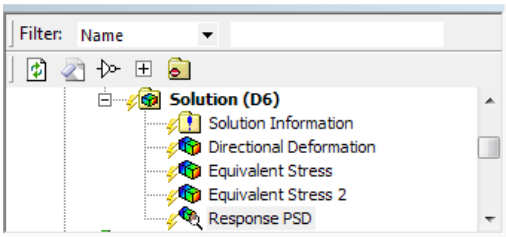
For the last case



Insert these det want to avoid tiring your fingers, use the files we provided for easier input.

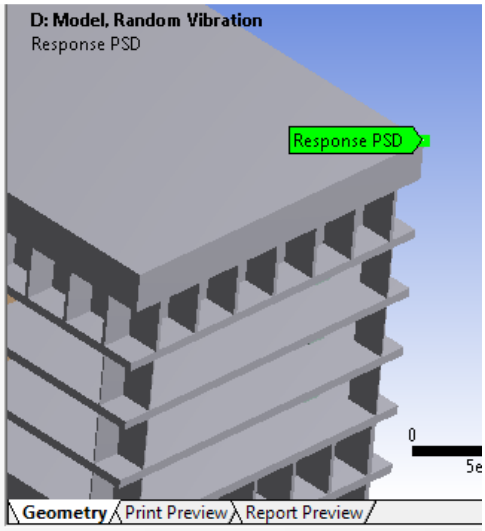


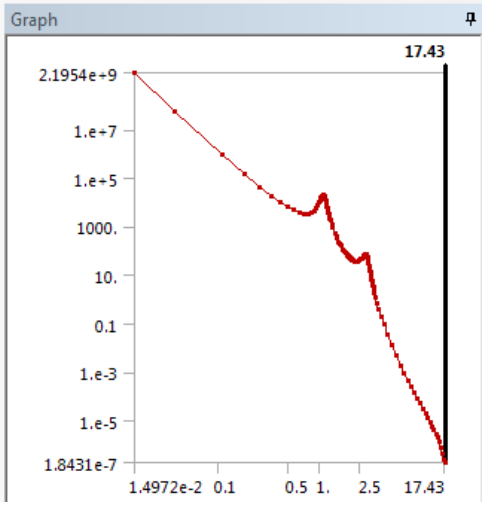
Solution: Apply axis/ scenario. For Response PSD, select a corner of the top of the building. Chang r each case.



Details of "Response PSD"

Definition	
Type	Response PSD
Location Method	Geometry Selection
Geometry	1 Vertex
Orientation	Solution Coordinate System
Reference	Absolute (including base motion)
Suppressed	No
Options	
Result Type	Displacement
Result Selection	X Axis
Selected Frequency Range	Full
Results	
RMS Value	5289.6 mm
RMS Percentage	100. %
Expected Frequency	0. Hz

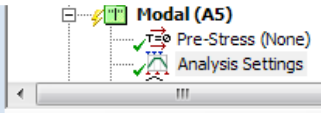




Further homework:

- make the Body Sizing = 250 mm, solve and draw the conclusions

- apply these changes to the frequency range, solve and draw the conclusions

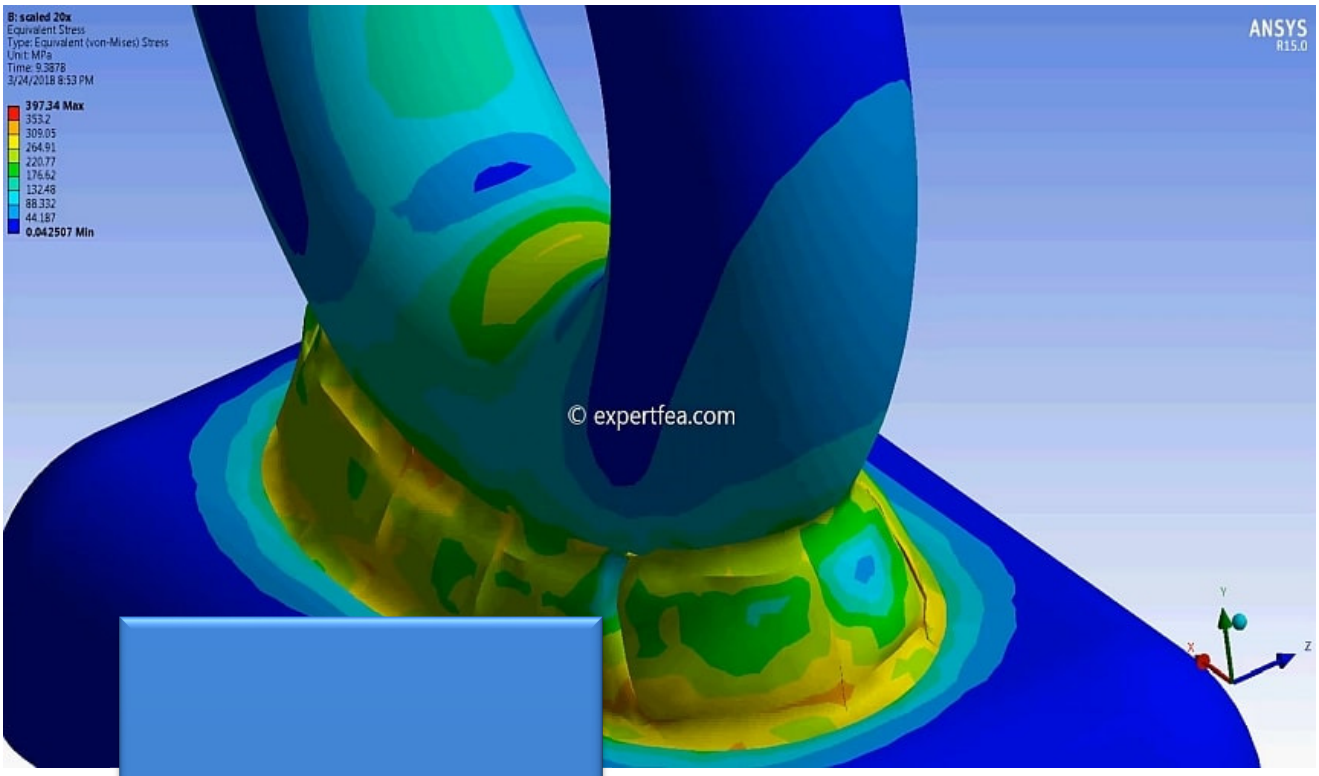


Details of "Analysis Settings"

Options	
Max Modes to Find	25
Limit Search to Range	Yes
Range Minimum	0. Hz
Range Maximum	25. Hz

- for Response PSD, select the top face of the building, green here, Evaluate All Results and draw the conclusions

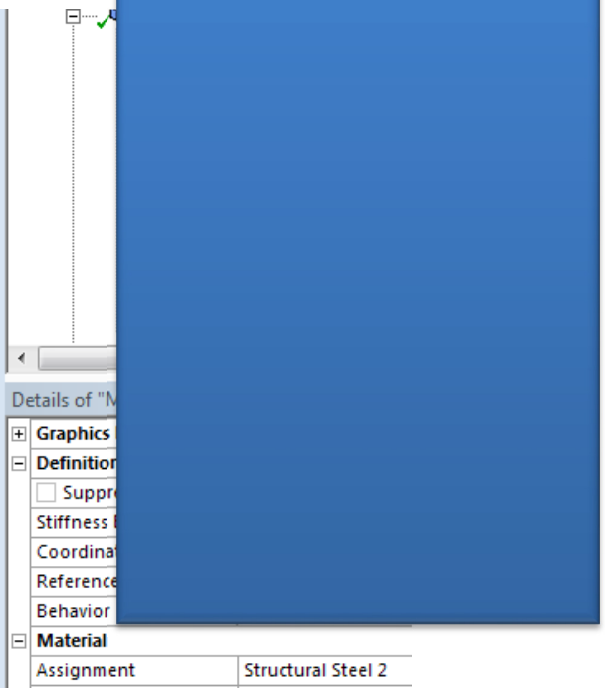
CASE 55: ANSYS WB FINITE ELEMENT ANALYSIS - Structural simulation of welding steel parts



Static Stru

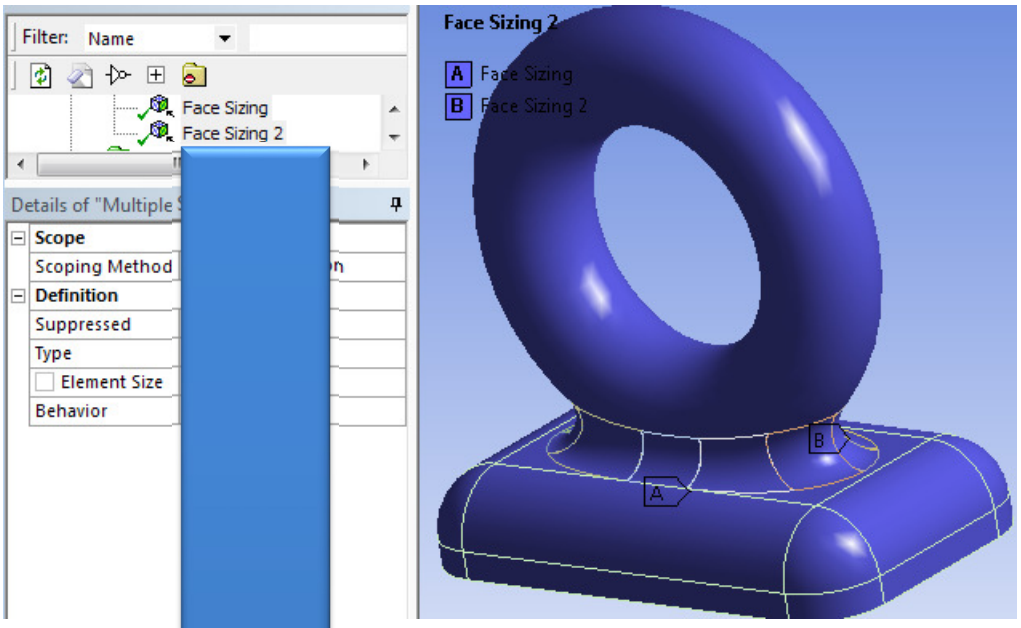
Geometry:

Assign Stru



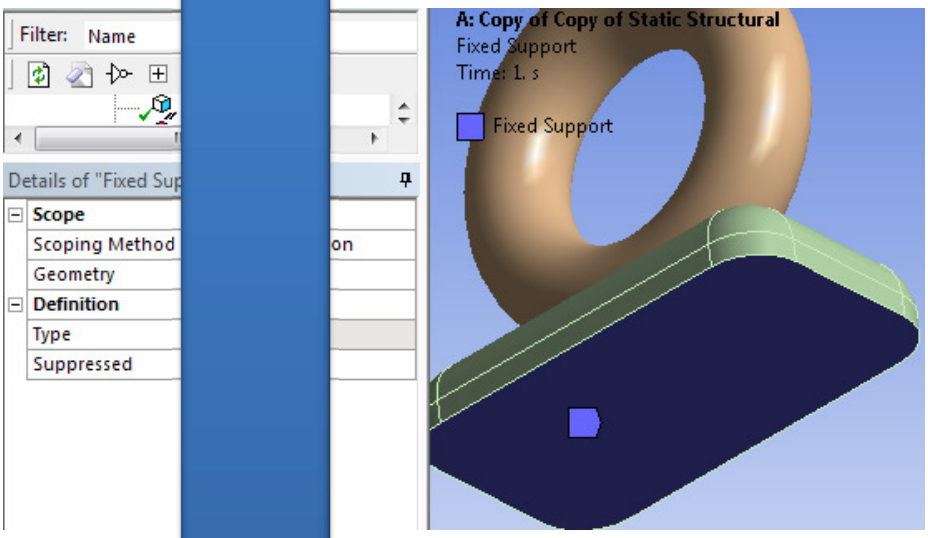
Mesh: Apply a Hex Dominant method to all bodies.

Insert this sizing to all faces, blue here.

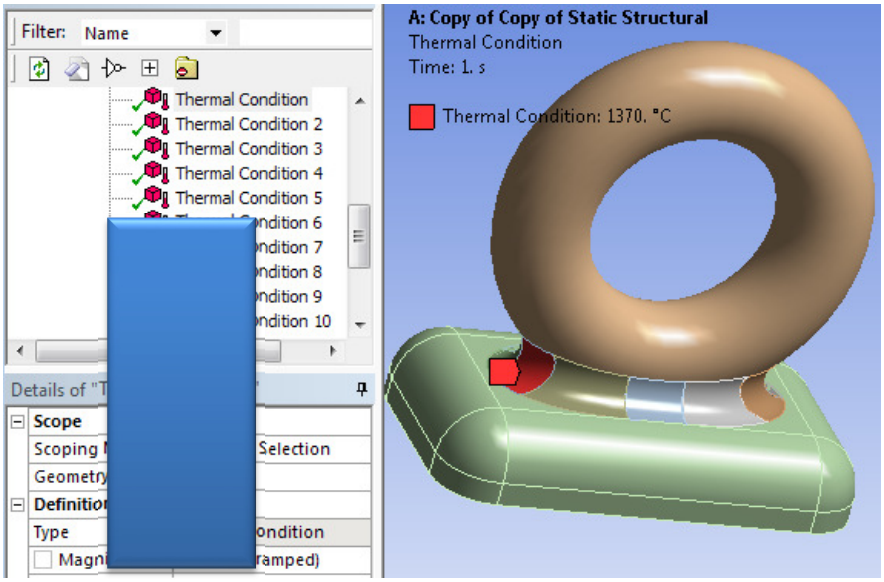


Analysis Setting e details for all steps.

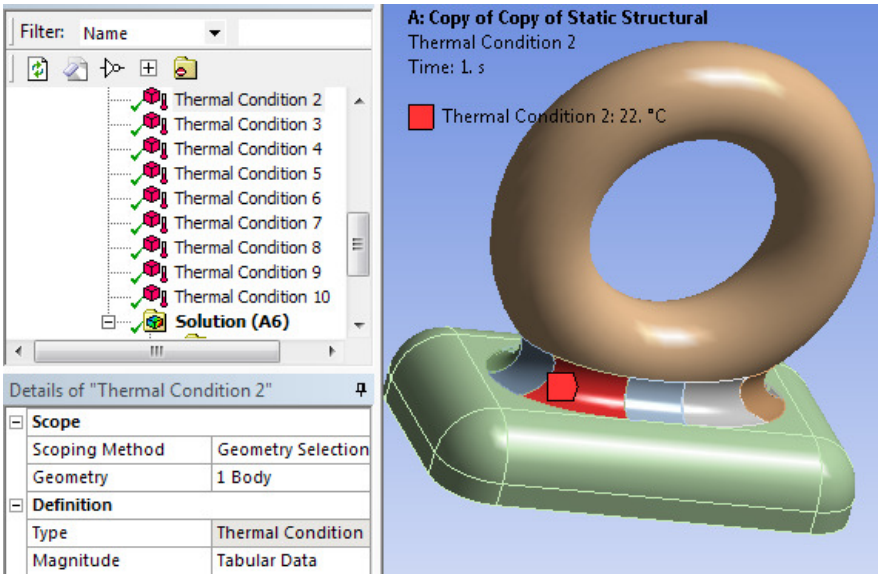
Environment: Fi face, blue here.



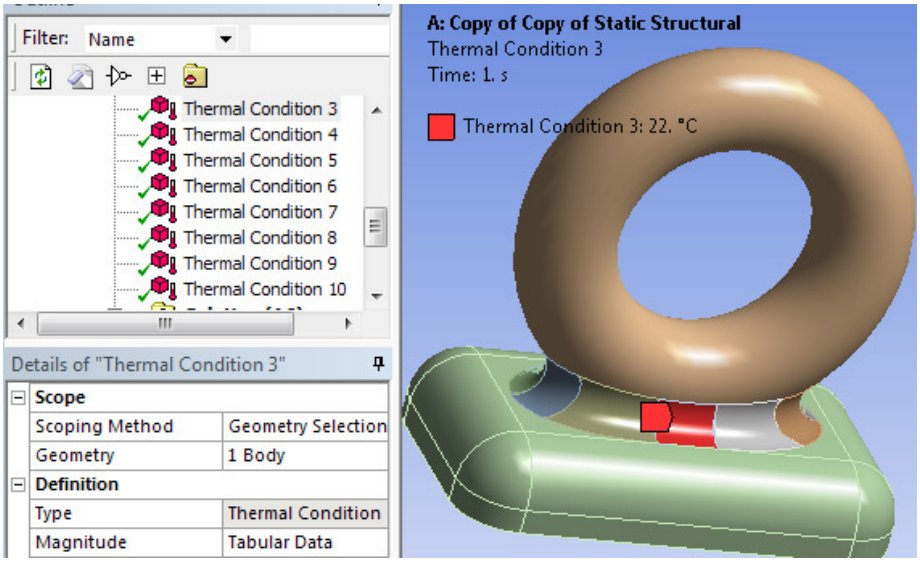
1st Thermal Con

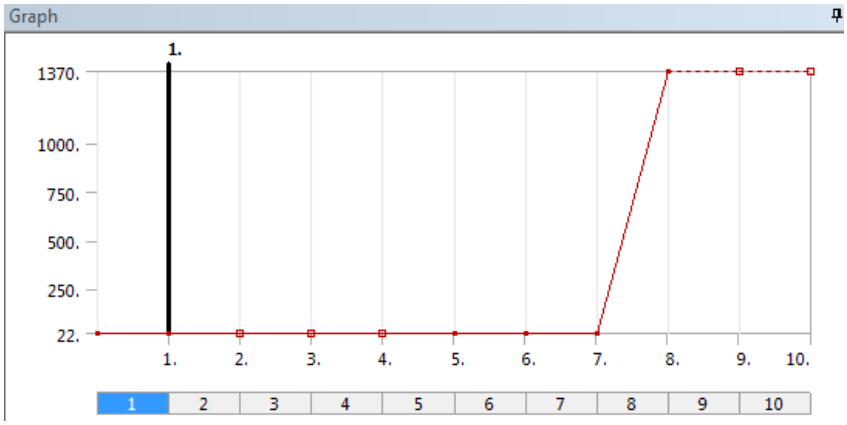


2nd Thermal Condition.

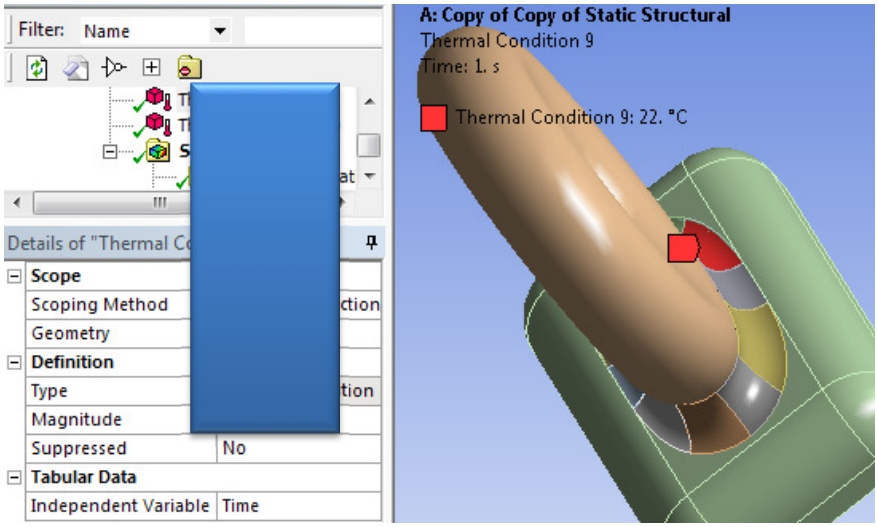


3rd Thermal Condition.

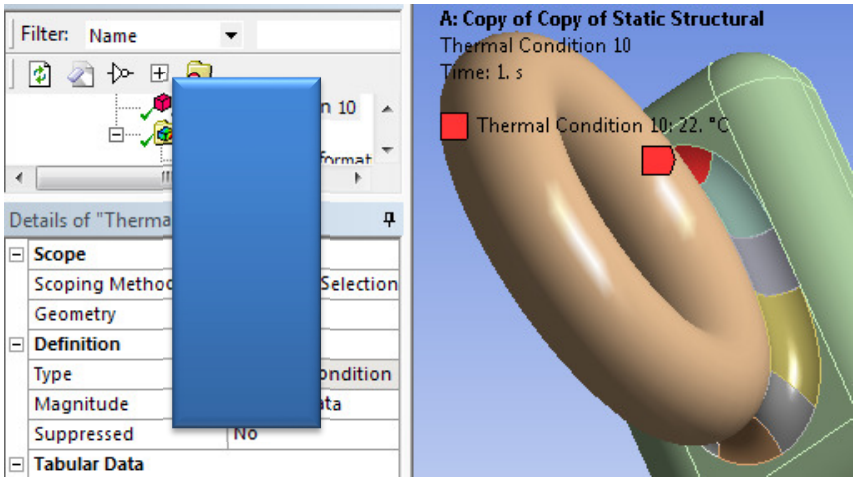




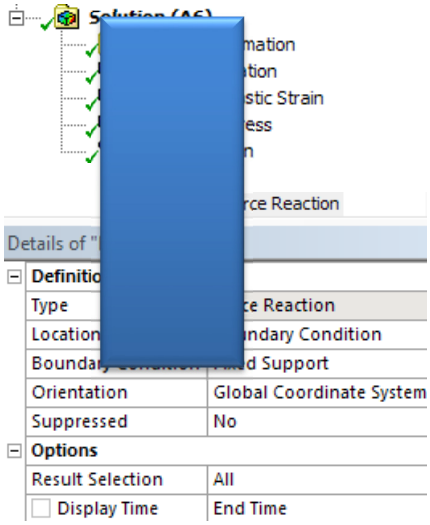
9th Thermal Condition.

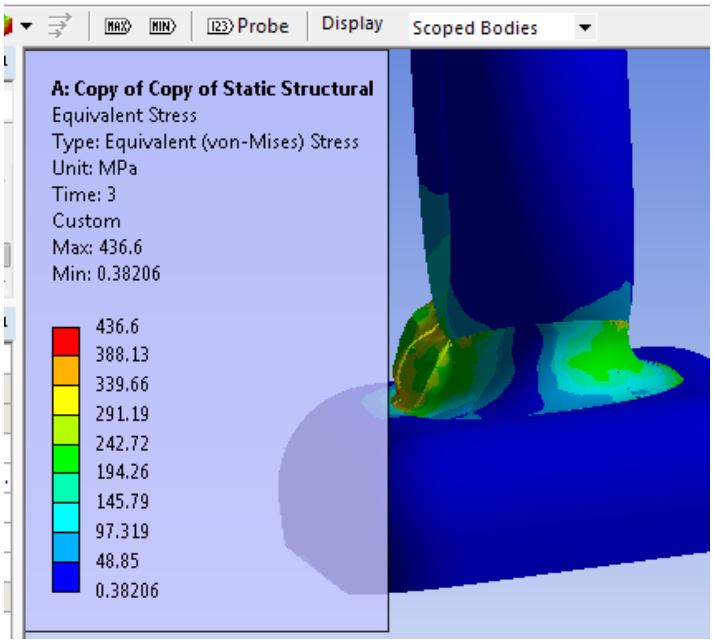


10th Thermal Condition.



Solution: Insert these default items, for all parts. You might want to increase the scale if you would like to show the dilation/ swelling of the weld seam.





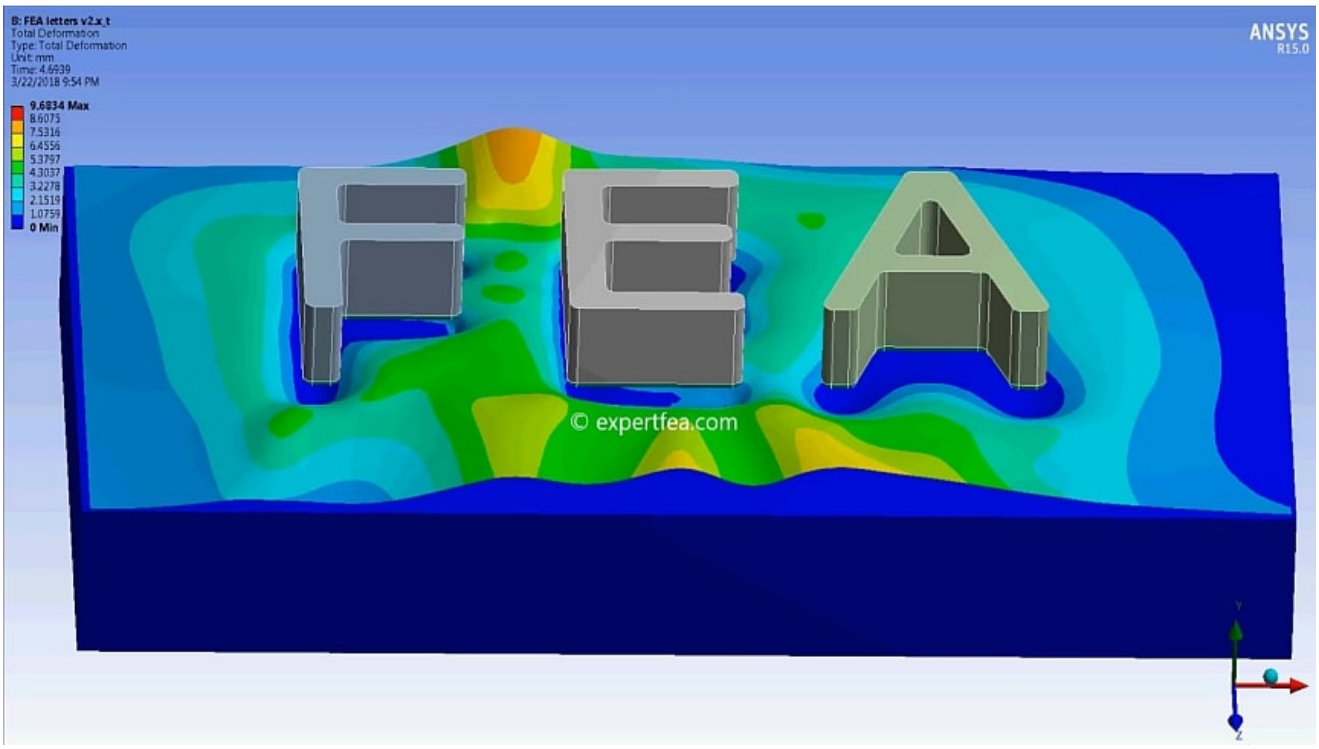
Further homework:

- change the 1370 deg C load into 3000 deg C load, solve and draw the conclusions

- change the default Bonded contacts to Frictionless, solve and draw the conclusions

- replace the Fixed Support with a Frictionless Support, solve and draw the conclusions

CASE 58: ANSYS WB FINITE ELEMENT ANALYSIS - Embossing/ Stamping of an Aluminum sheet-metal in a rubber die

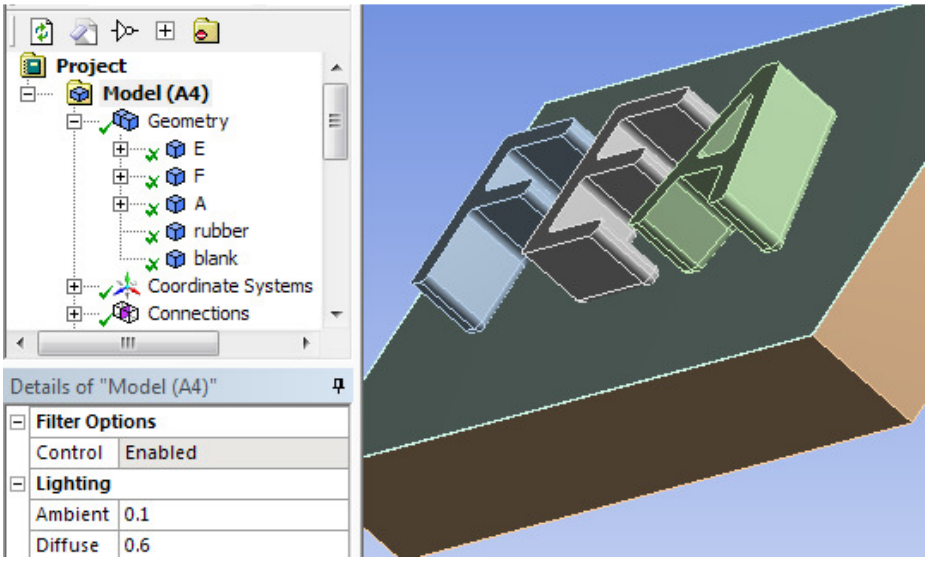


Static Structural

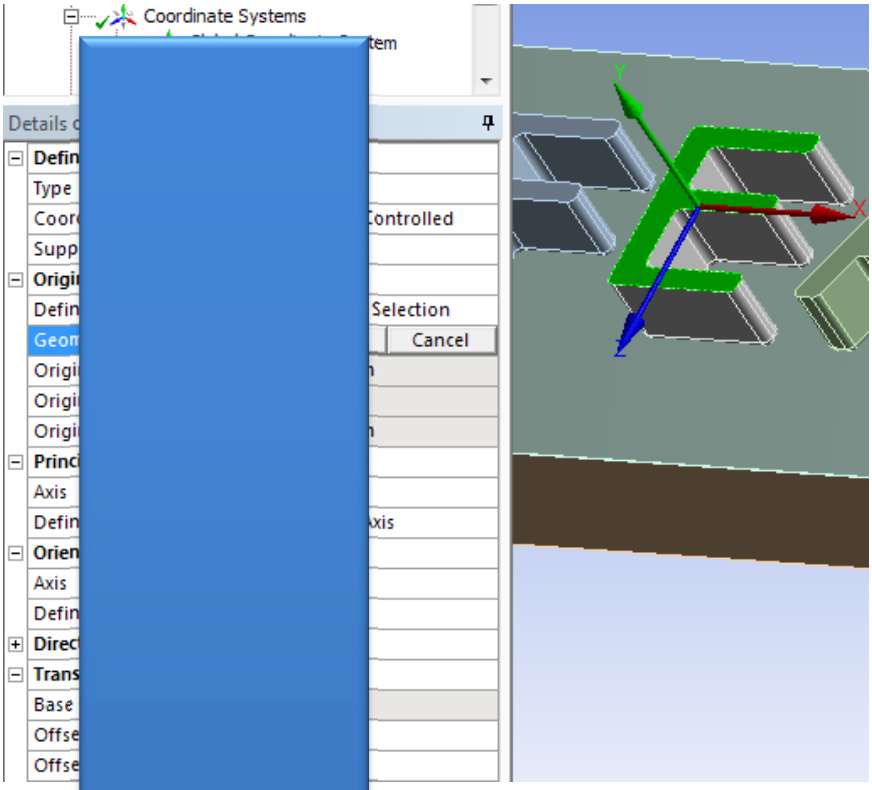
Engineering Data: Insert Aluminum Alloy NL from Non-Linear Materials library. Duplicate the default Structural Steel and assign it these details. Delete the other non-important characteristics, as seen here.

Geometry: FEA letters v2.x_t

Assign these names and materials; make the letters rigid.

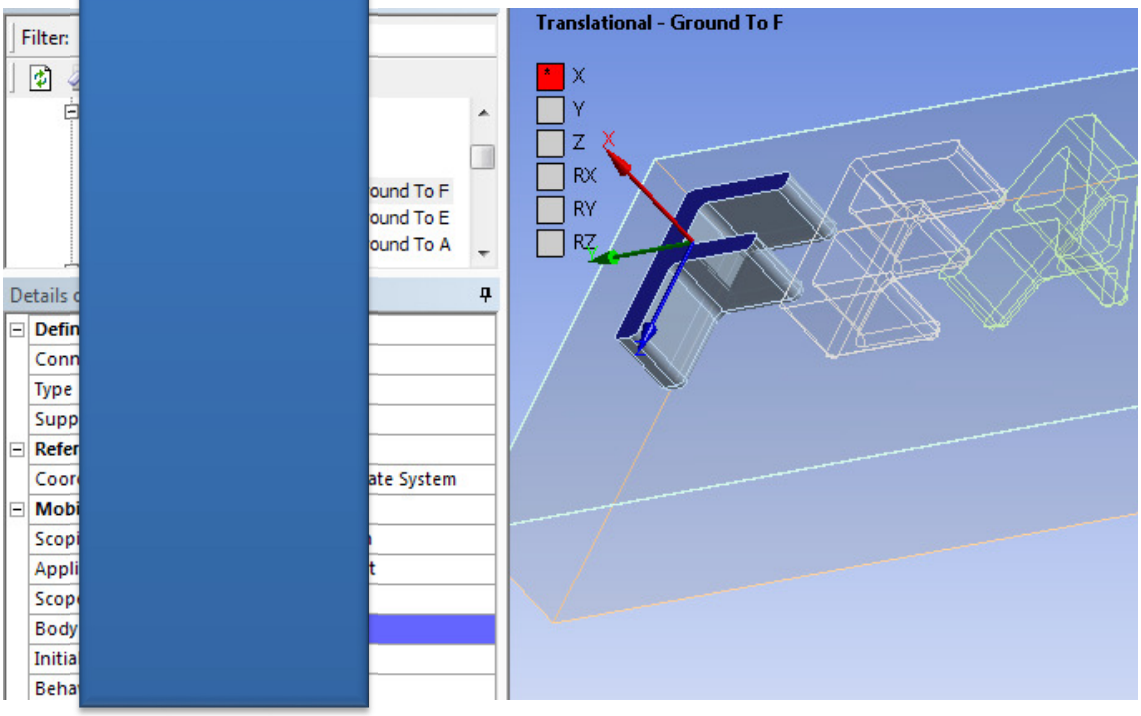


Coordinate Systems: Click on this green face of the letter E, then the Coordinate System button and set these offsets from the toolbar.



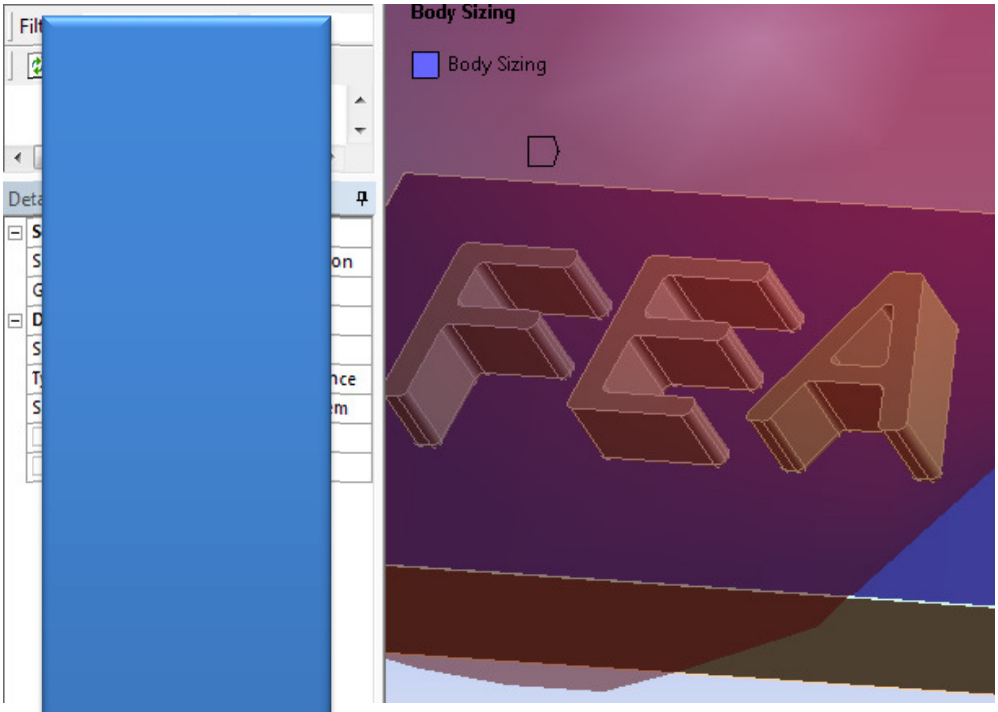
Conne Based on Definition. Suppress the unnecessary contacts. Assign these details to all the Fri

Joints: l Joints for each letter part.



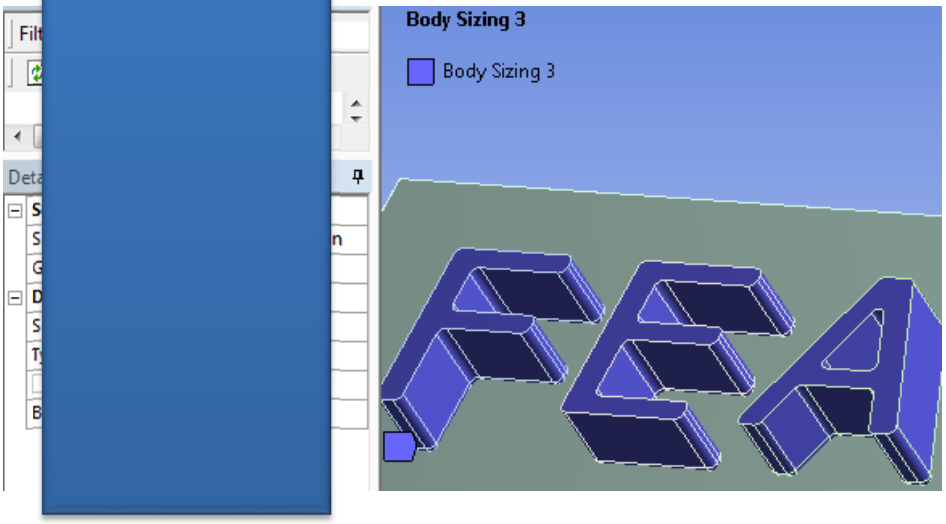
Mesh:

Insert this Body Sizing for the Blank part.

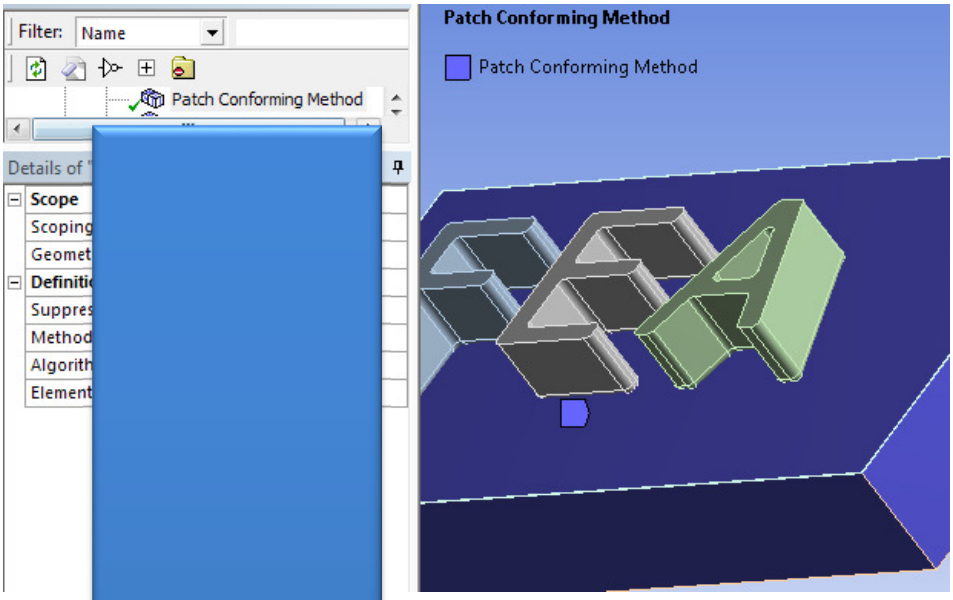


Def e Rubber die.

Add etters.



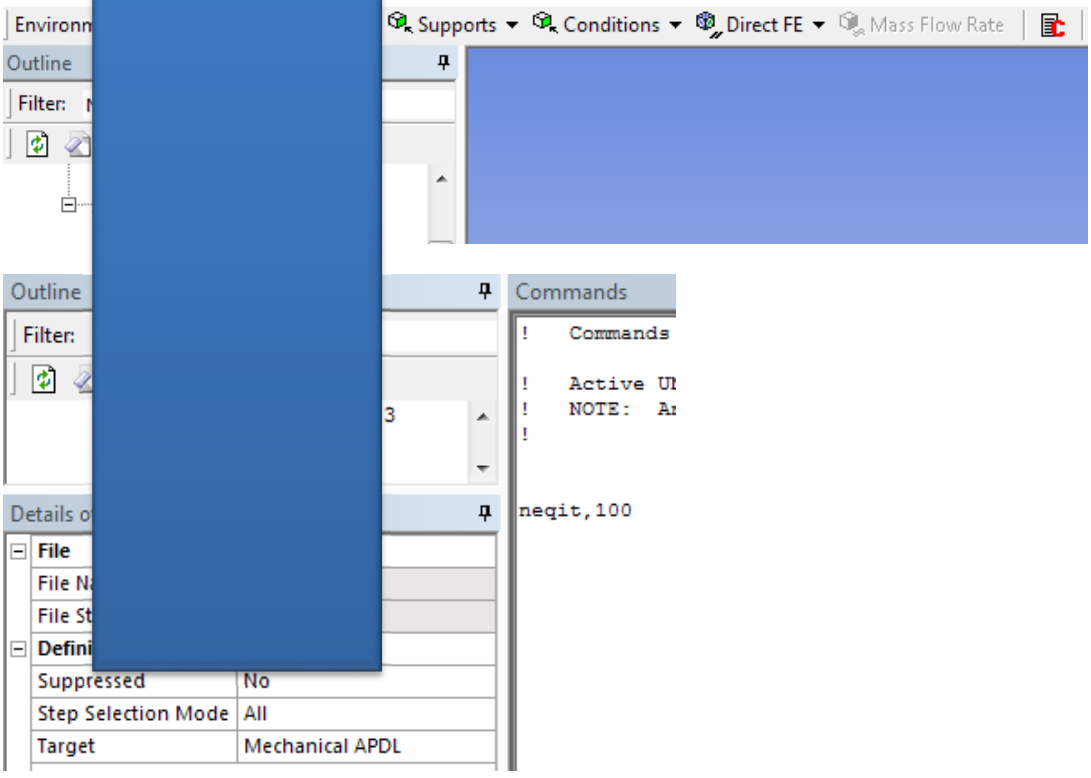
Mesh the Blank and the Rubber parts, blue here, with Tet elements.



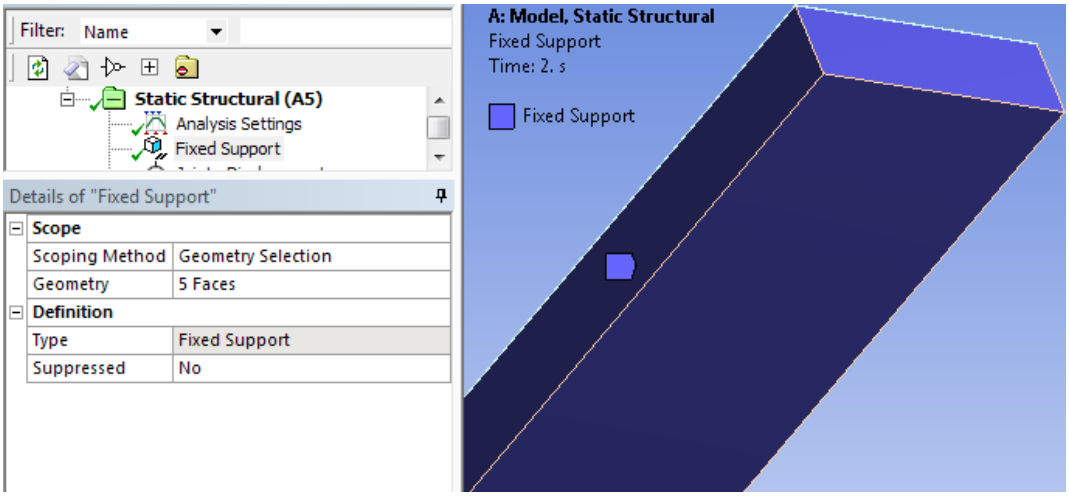
Analysis ails for the 1st step.

Apply th ing steps.

Press th far right on the Environment toolbar and insert **neqit,100** as seen here.

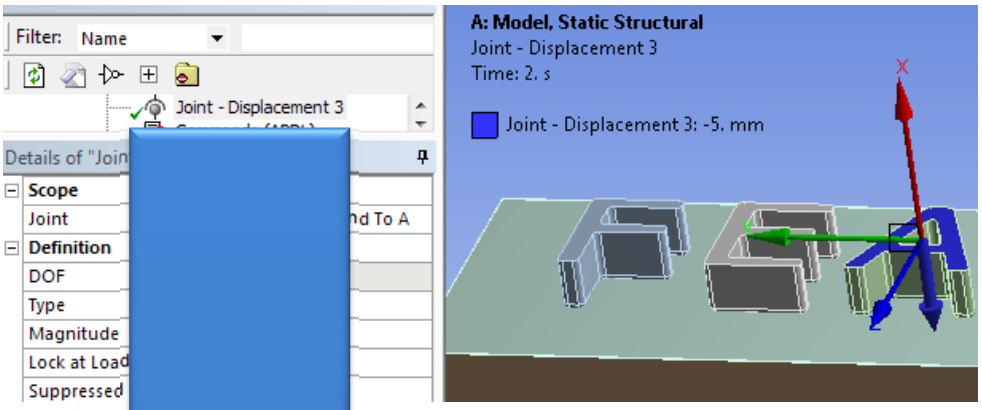


Environment: Fix the outer faces of the Rubber die, seen blue here.



Set this Joint Load for the letter F.

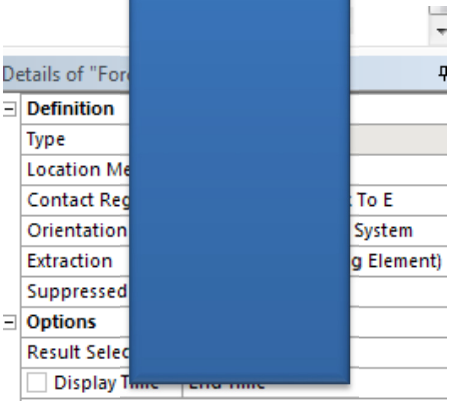
Add this Joint Load for the letter A.



All 3 joints s e values in the Graph tab.

Solution: In ems, for all parts.

Use these c Reaction item.



Further homework:

- change the material for the Blank part to be Copper Alloy NL, solve and draw the conclusions

- change the default Frictionless contact to Frictional = 0.1, solve and draw the conclusions

- for Joint Loads, assign these values, solve and draw the conclusions

THANK YOU, GOD! TO BE CONTINUED WITH VOLUME 4!