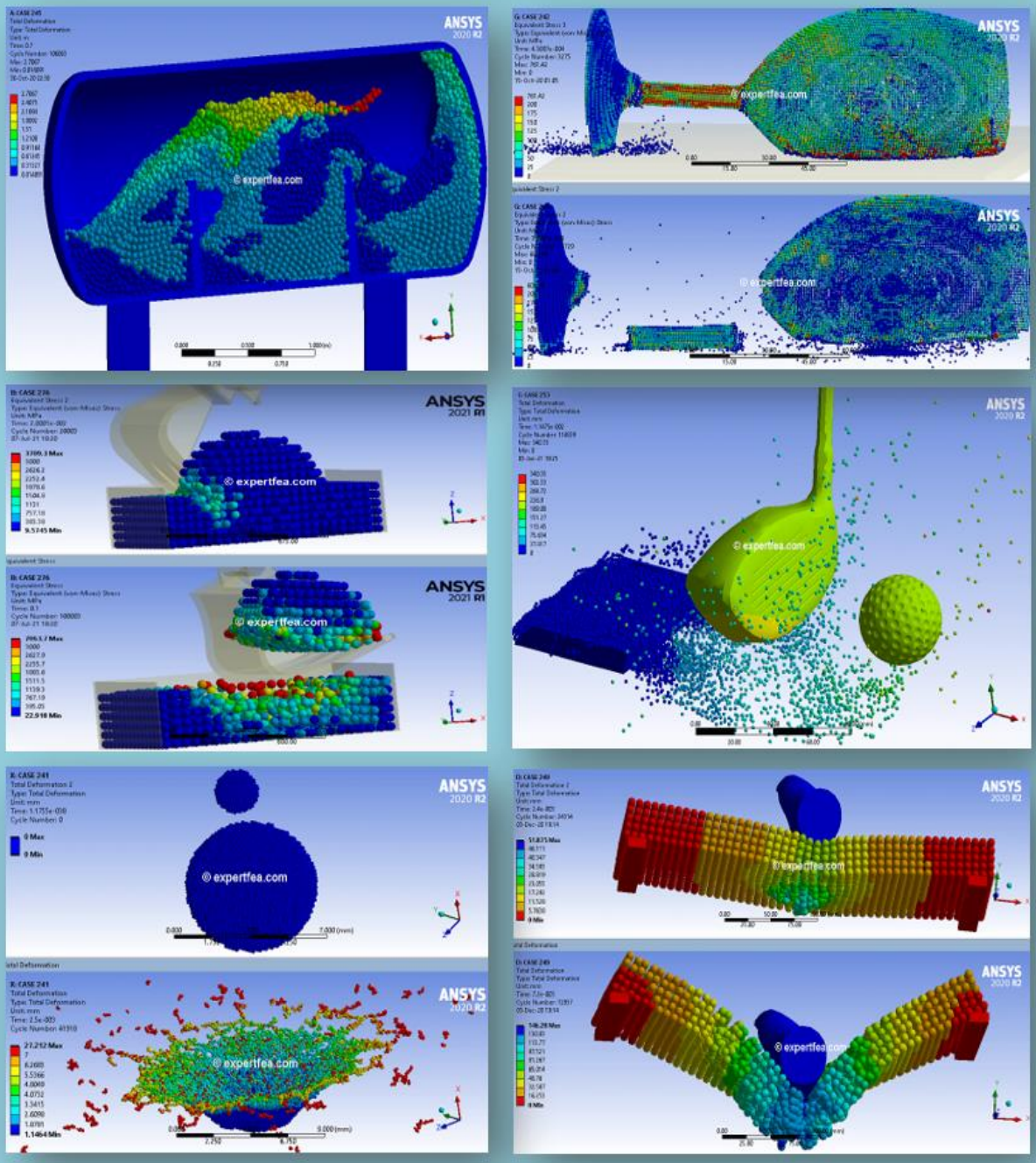
EASY USE OF SMOOTH PARTICLE HYDRODYNAMICS IN ANSYS WORKBENCH

-VOL. 1-

BY CLAUDIU DANILA

(QUICKLY SIMULATE CONCRETE, SAND, FLUIDS, GLASS AND SANDSTONES IN EXPLICIT DYNAMICS)




Om Namah Shivaya
ॐ नमः शिवाय

Foreword

Dear FEA and ANSYS lovers,

For some of us, ANSYS users - wanting to simulate high non-linearities, breakage and flow - the wait for SPH (Smooth Particle Hydrodynamics) in the default Explicit Dynamics module was over in summer 2020 with the release of v2020 R2! Since then, I worked intensively on it and after a few dozens of scenarios, and after my friends and customers asked for some tutorials using SPH, here they are, comprised in this volume.

Because there are more scenarios made but we wanted to respect July as the publishing month, more awesome scenarios will be comprised in the next volume, together with hyperelastics, soils and ceramics.

If you do not have the patience to wait for volume 2, dare to change yourself the materials in these simulations to what you need, then further on, work on making the FEA successful.

You will find here also some tips and advice on known issues and limitations, so this will spare you days of trials and errors for which you did not guess the causes.

As is the case for most tutorials books, they are presented in a simple manner and they are very easy to apply, but you also need to know that they are the result and pinnacle of days and weeks of work. When changing some parameters, you will probably need to invest some good hours to make the FEA solve till the end and avoid sudden stops.

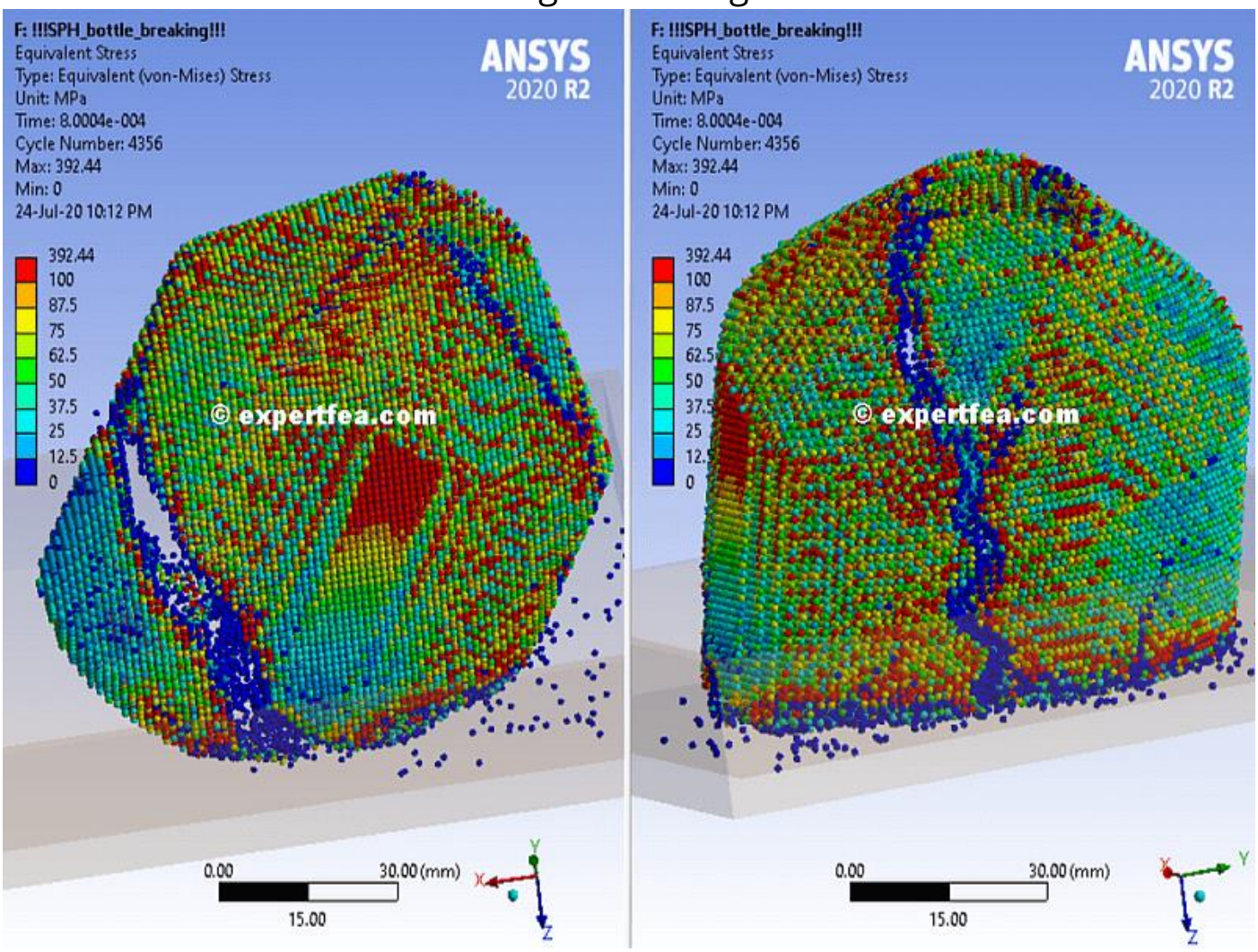
Each tutorial has a homework section in which you can vary some parameters as advised, but also feel free to change more and be creative in pushing the simulation's boundaries and limitations.

We express our happiness that we can now perform SPH simulations in the default Explicit Dynamics module, because previously was possible only in "elite" explicit FEA such as Autodyn, Abaqus, LS-DYNA and Radioss. It is nice to know that we can perform nowadays the most complex simulations in the software that we know best: ANSYS Workbench.

May God/ Allah/ Brahman bless you with all the best and remember to always believe in yourself and in your bright future! ❤️

Claudiu Danila, 8th of July 2021

SPH2 - 235. 1st in the World!!! ANSYS WB Explicit Dynamics SPH - Drop test and breaking of a thick glass bottle

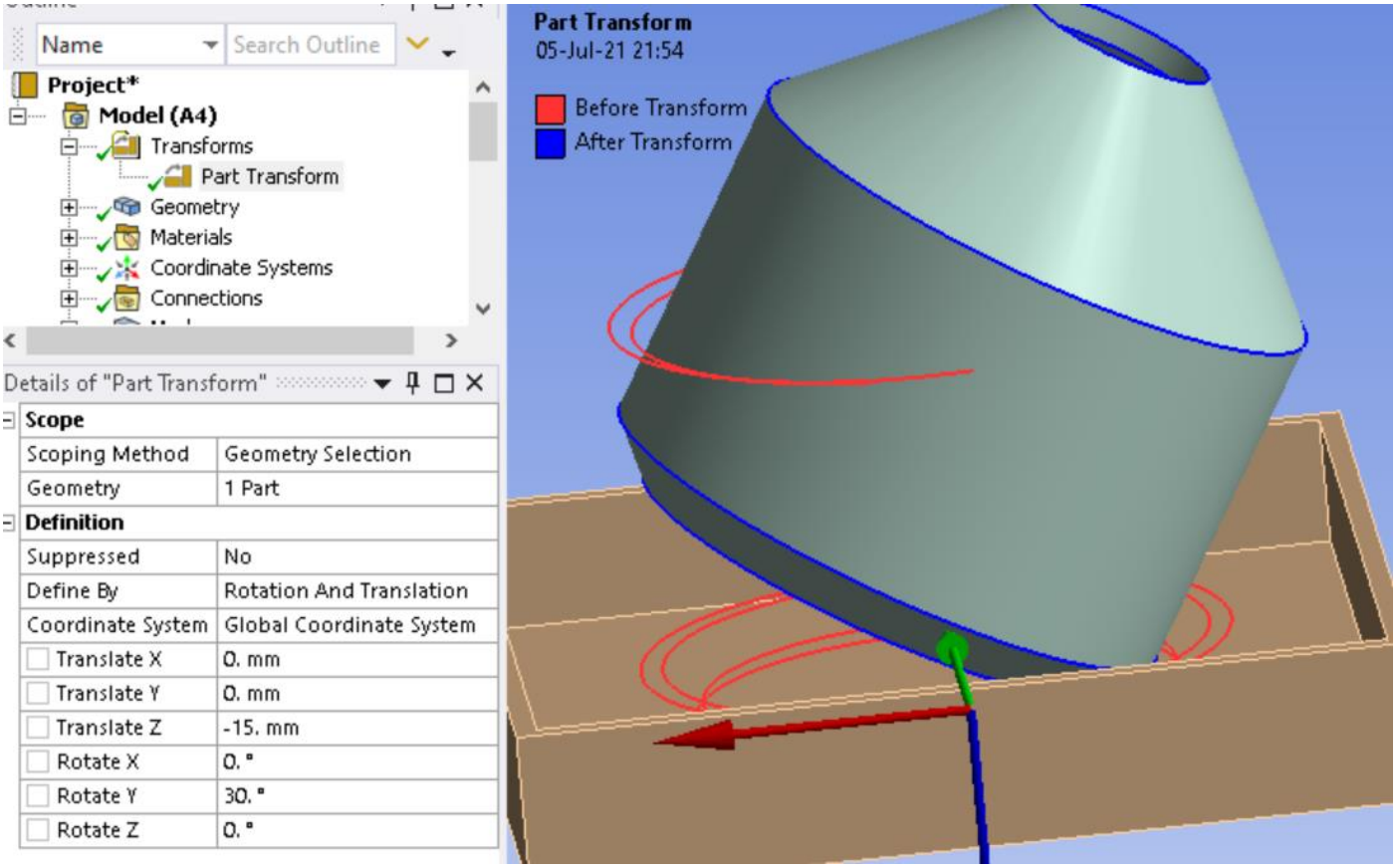


Drag and drop an Explicit Dynamics module from the Toolbox.

Import the geometry file named: *2020_jul_23_SPH_bottles_floor_v2.x_t*

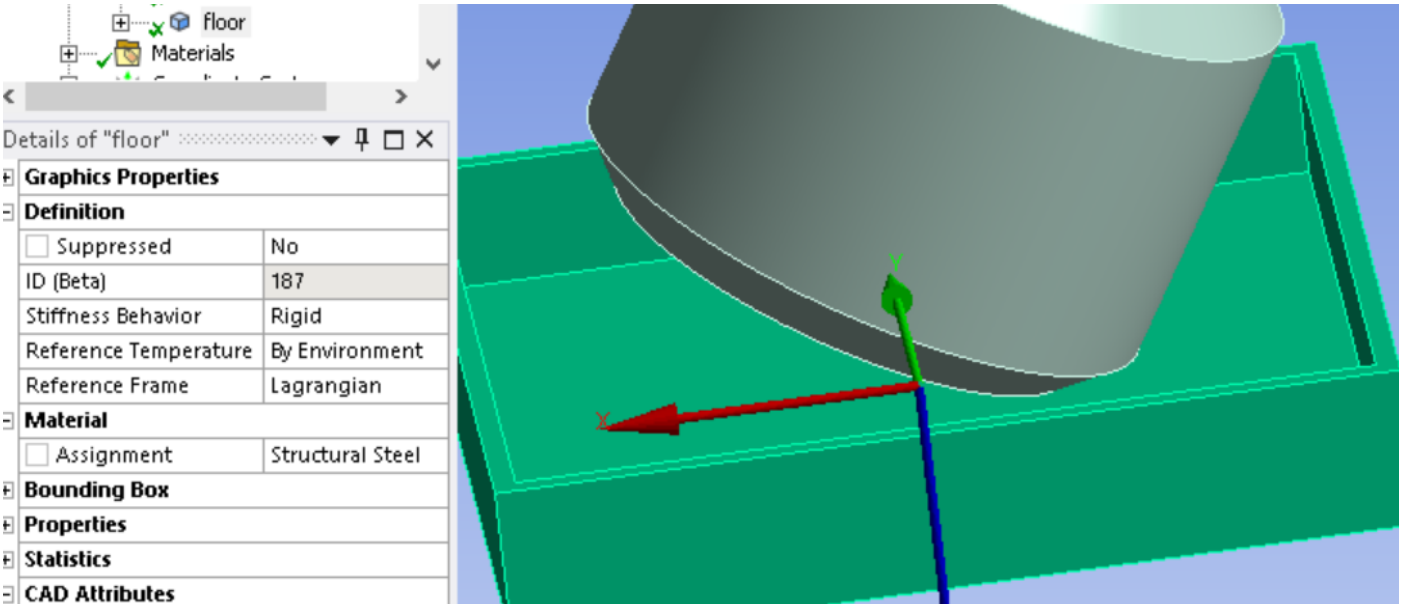
Double click Model to enter the FEA.

Transforms: Assign these details for the glass body, to make it hit the floor at an angle.

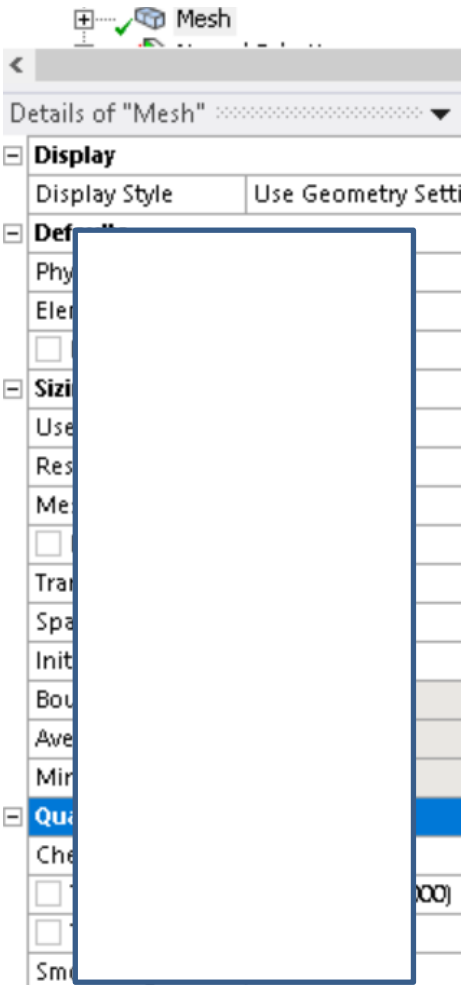


Geometry: Rename the bodies as seen here. Suppress the 1st part. Assign these details to the bottle part.

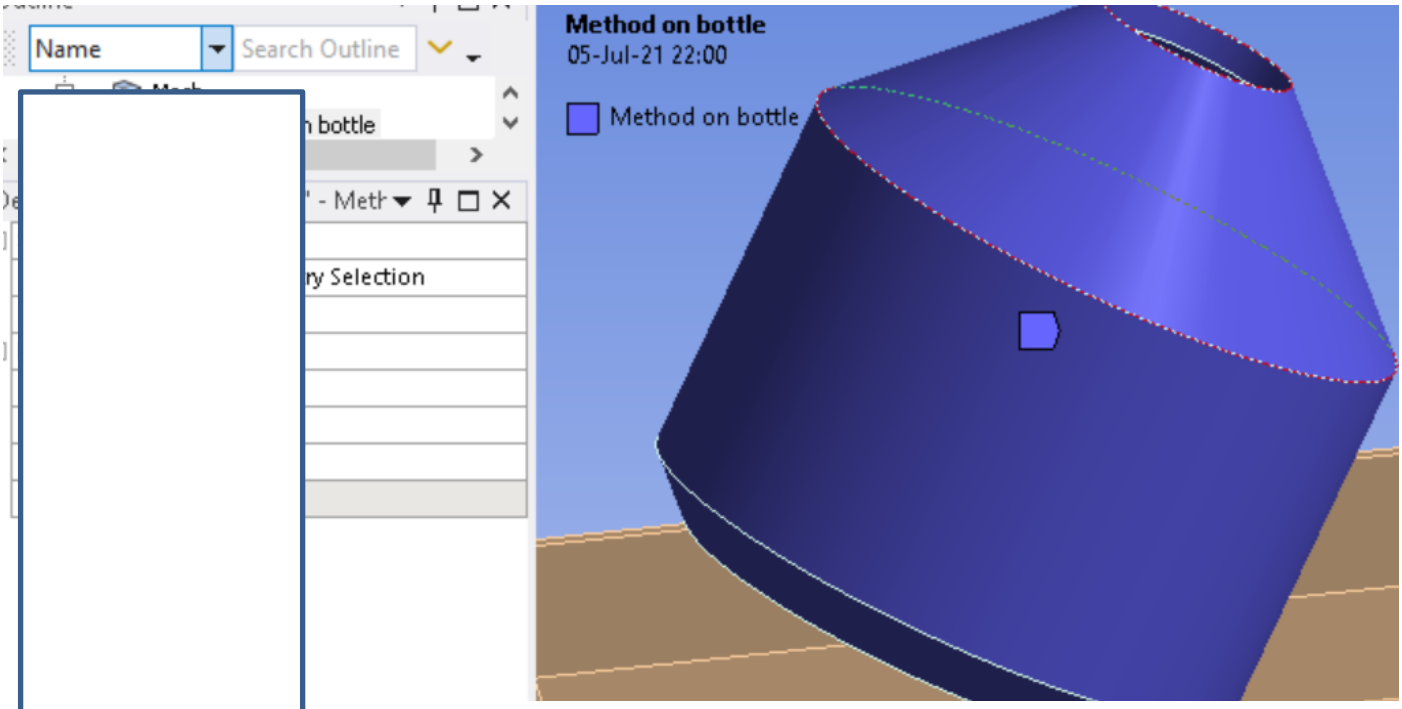
The floor part is Rigid, Steel.



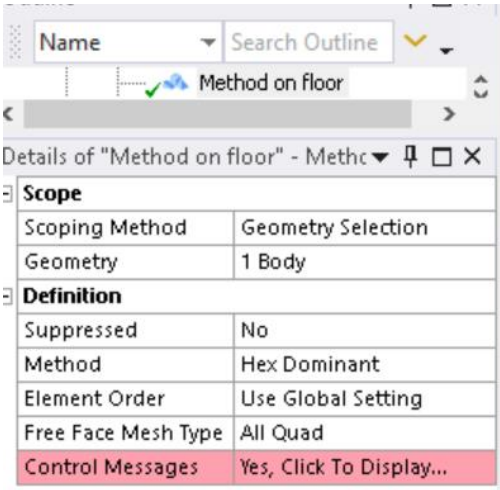
Mesh: Assign these details.

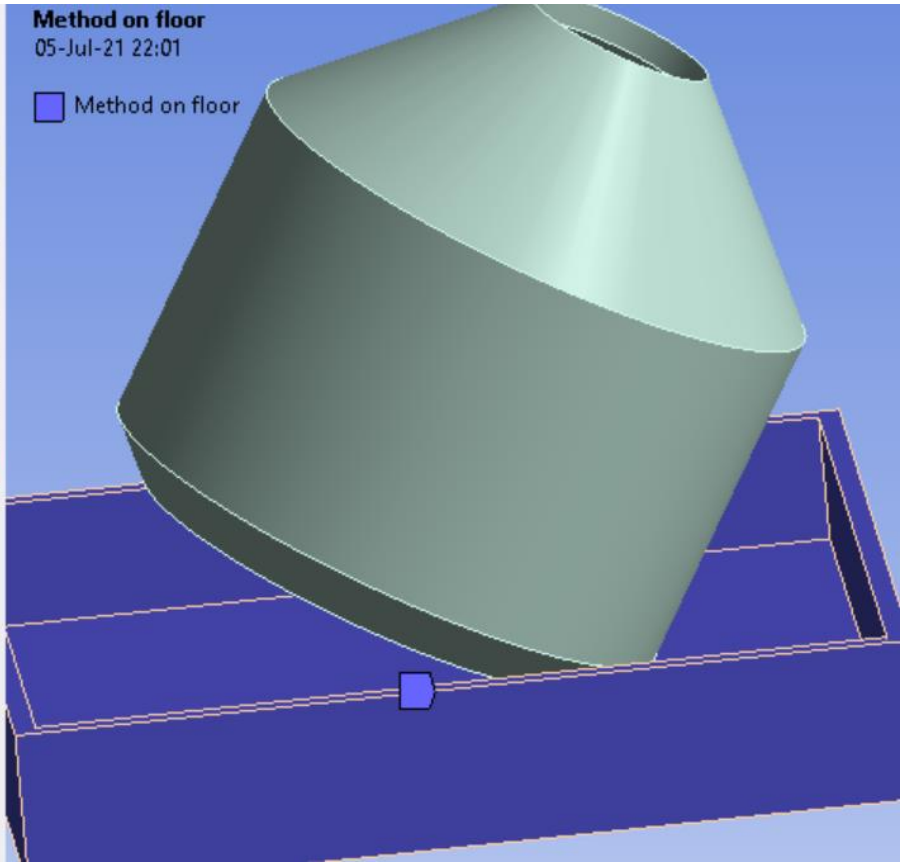


Assign this Method on bottle.



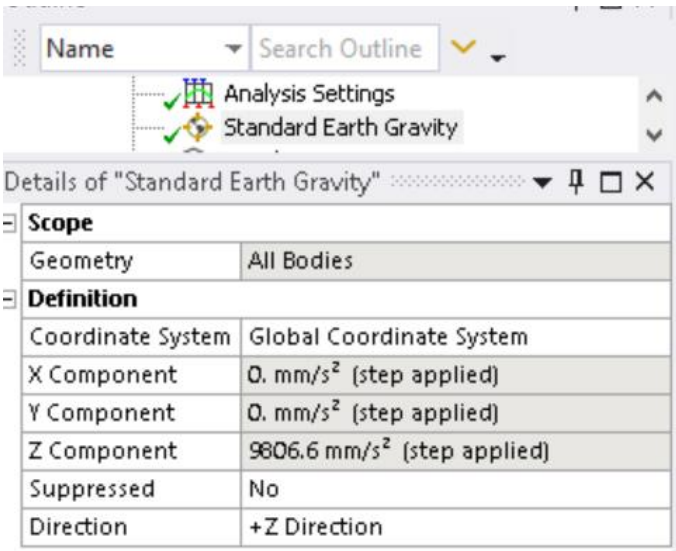
A on floor.

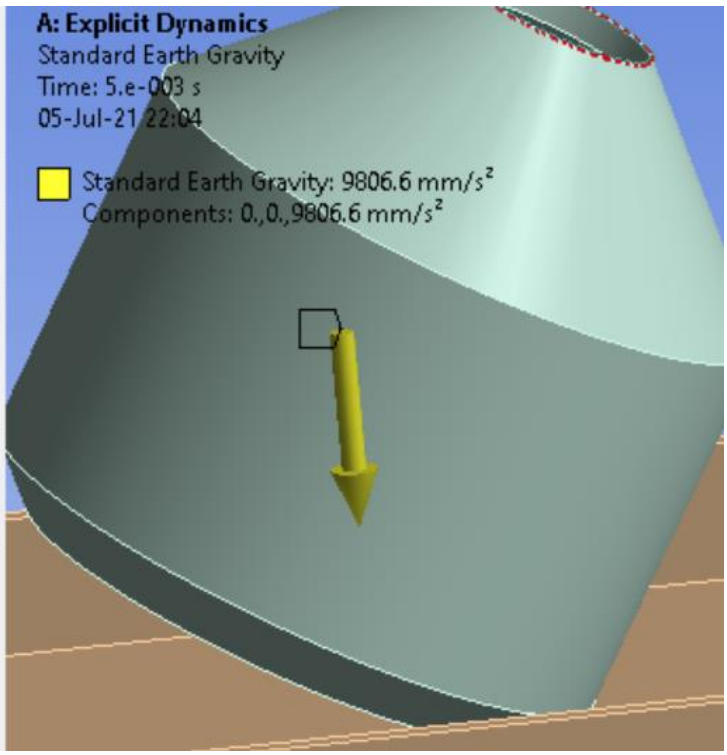




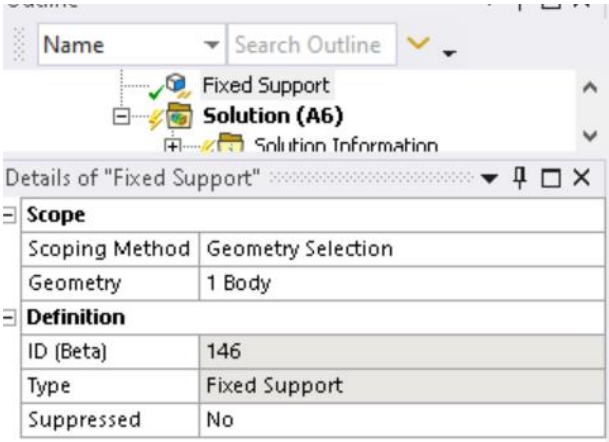
Assign this Sizing on floor.

<u>Standard Earth Gravity:</u> Insert it along the +Z axis.



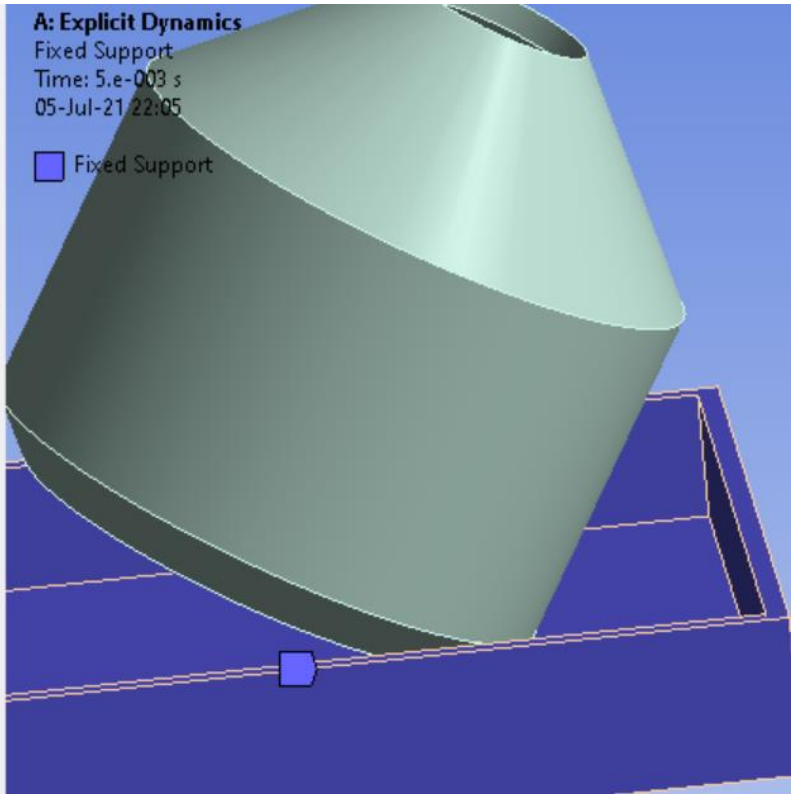


Fix the floor tray. Save, Solve.

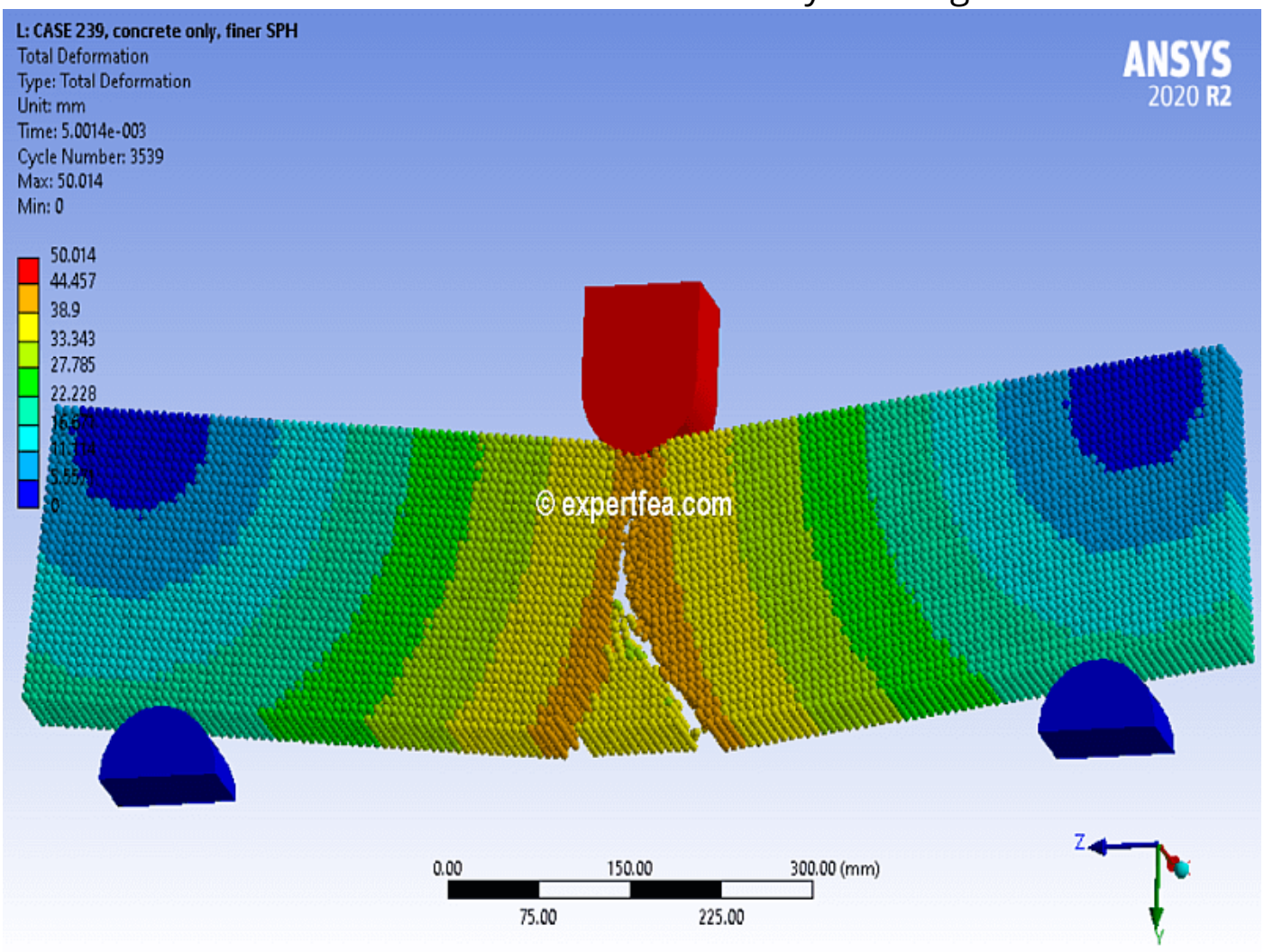


Details of "Fixed Support"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
ID (Beta)	146
Type	Fixed Support
Suppressed	No



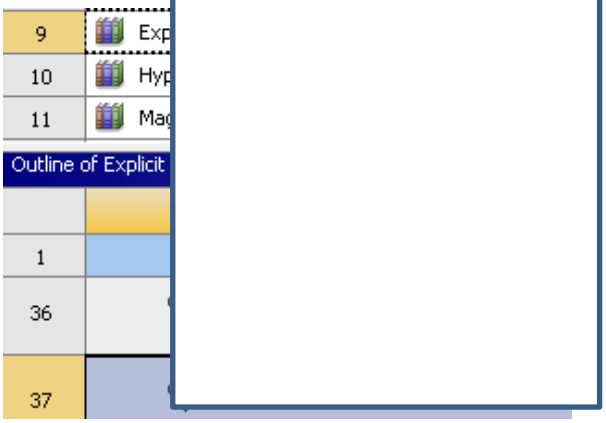
SPH4 - 239. 1st in the World!!! ANSYS Explicit Dynamics SPH - Fracturing an unreinforced concrete beam by bending



Drag and drop an Explicit Dynamics module from the Toolbox.

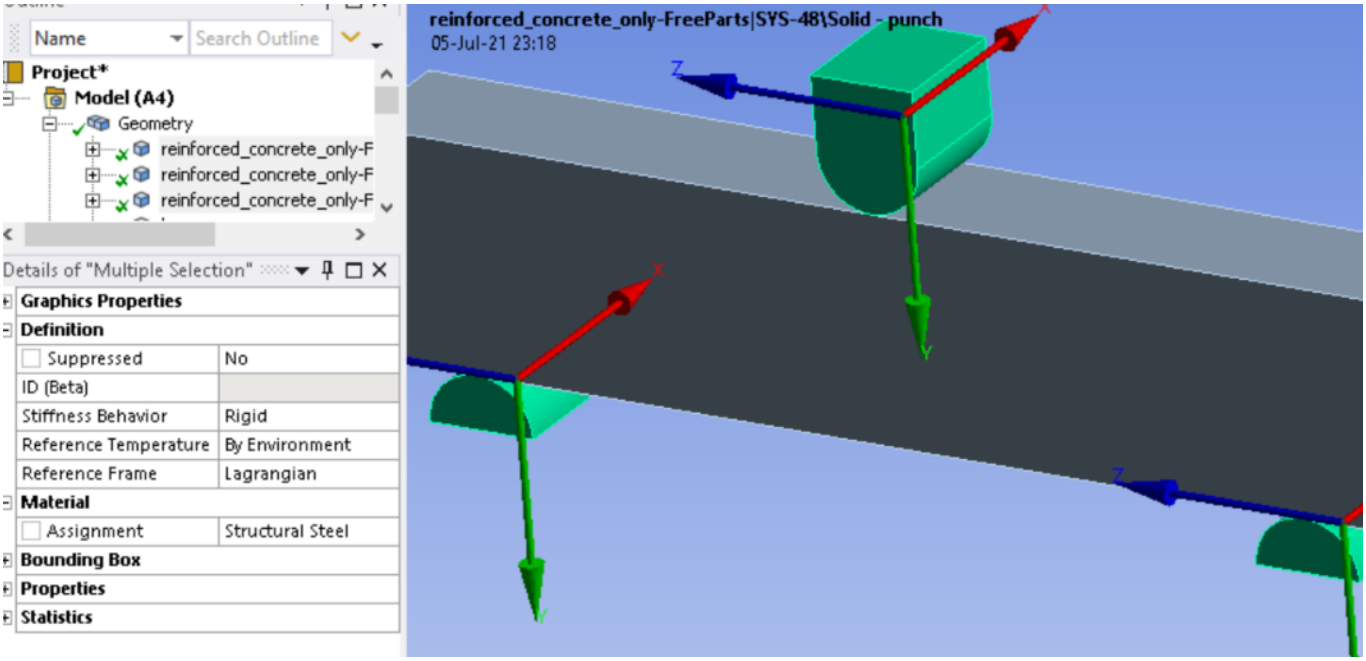
Import the *e_only.stp*

Engineering m the Explicit Materials.

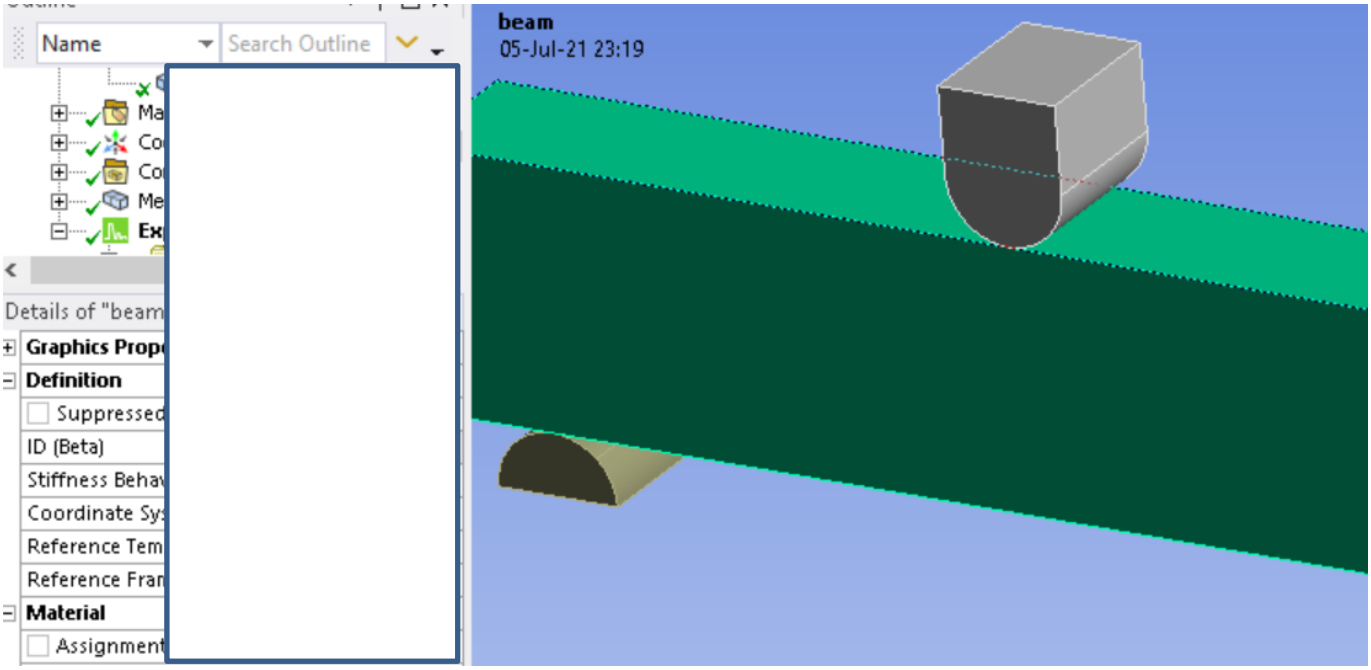


Double click Model to enter the FEA.

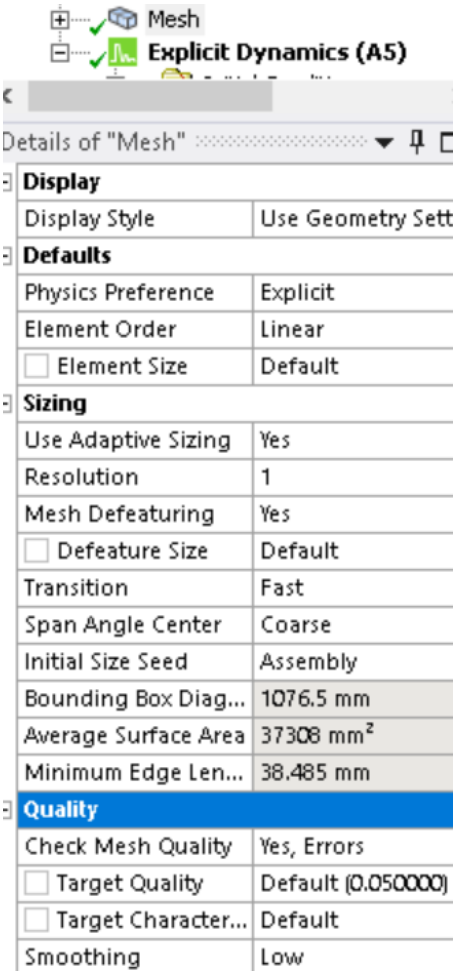
Geometry: Make the green parts as Rigid, Steel.



Assign these details to the beam part.

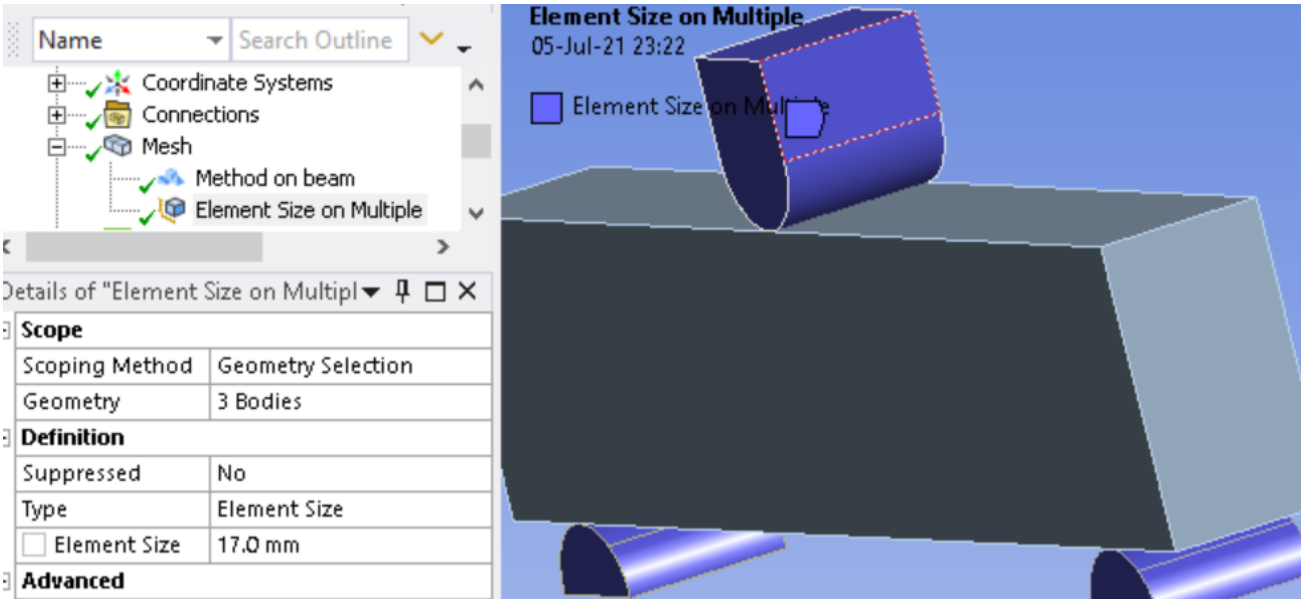


<u>Mesh:</u> It should have these details.



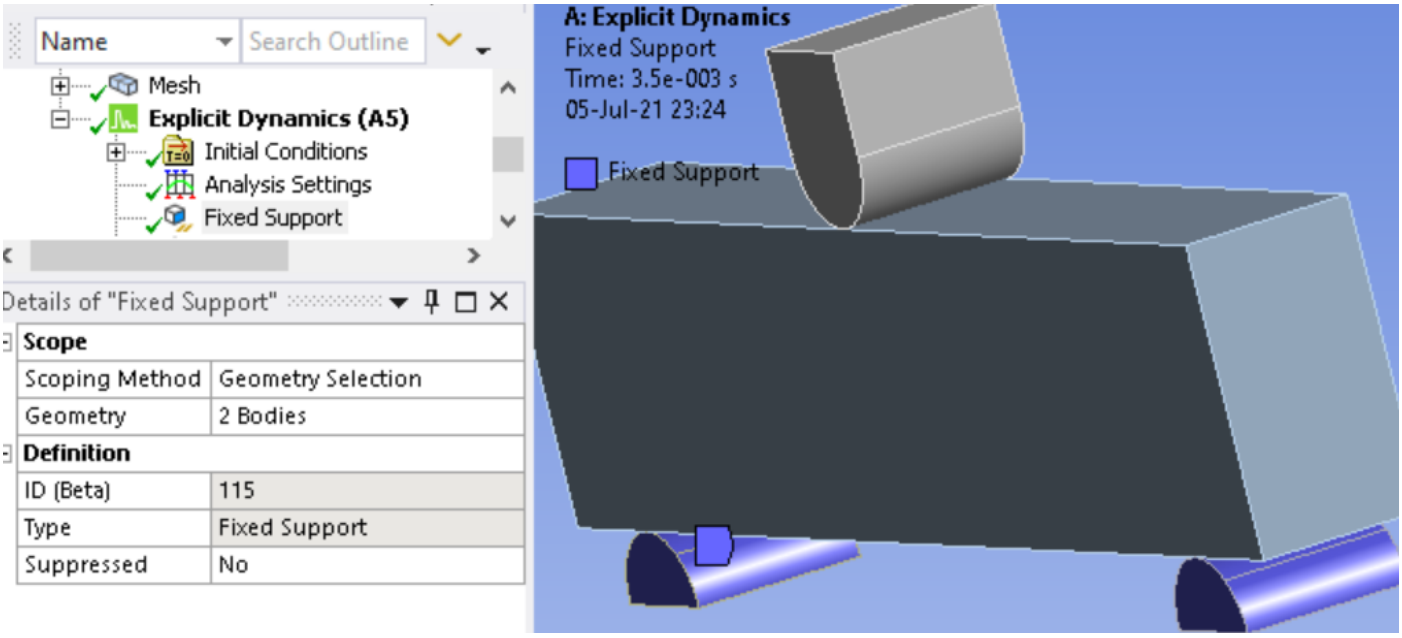
Assign this Method on beam.

Create this Sizing for the rigid parts, blue here.

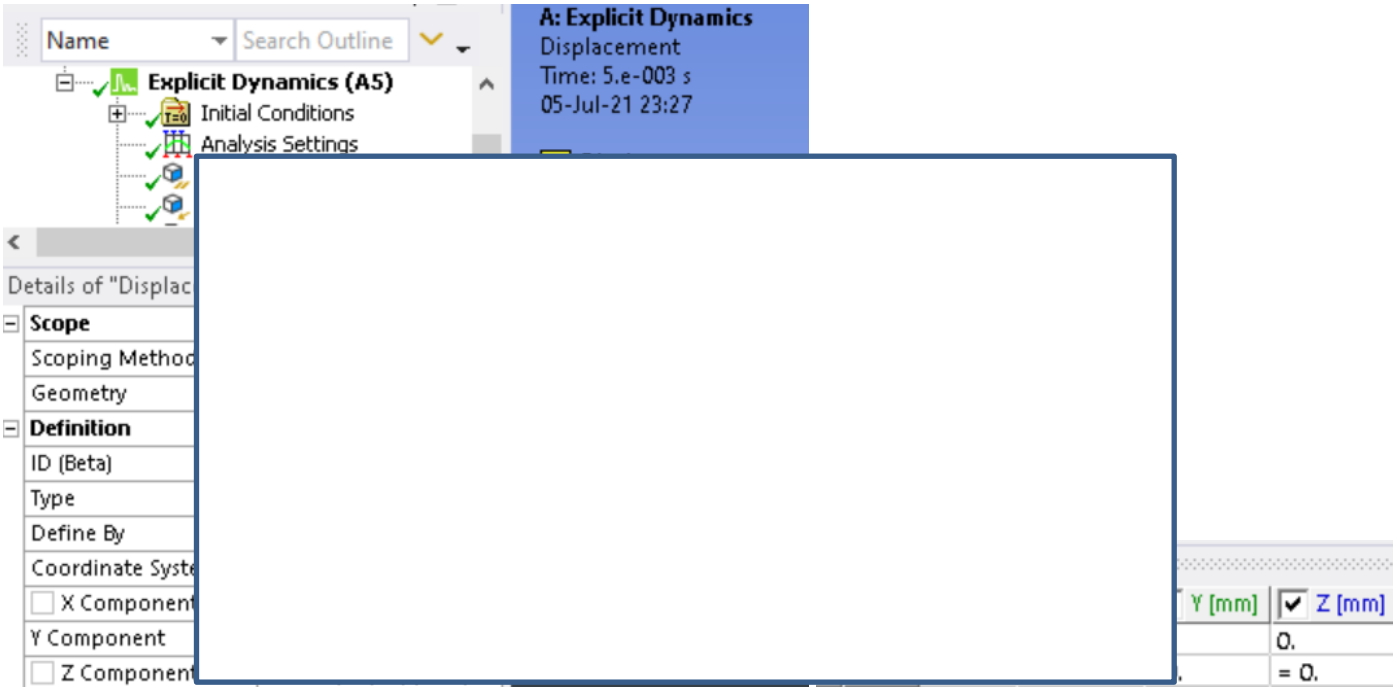


Analysis Settings: Insert these details.

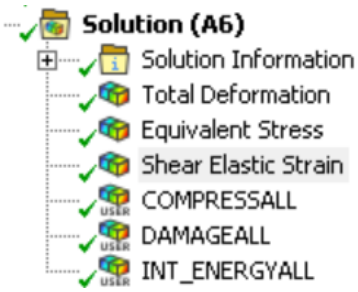
Fix the 2 blue supports on the bottom.



Apply this Displacement on the center part to crack the beam. Save, Solve.

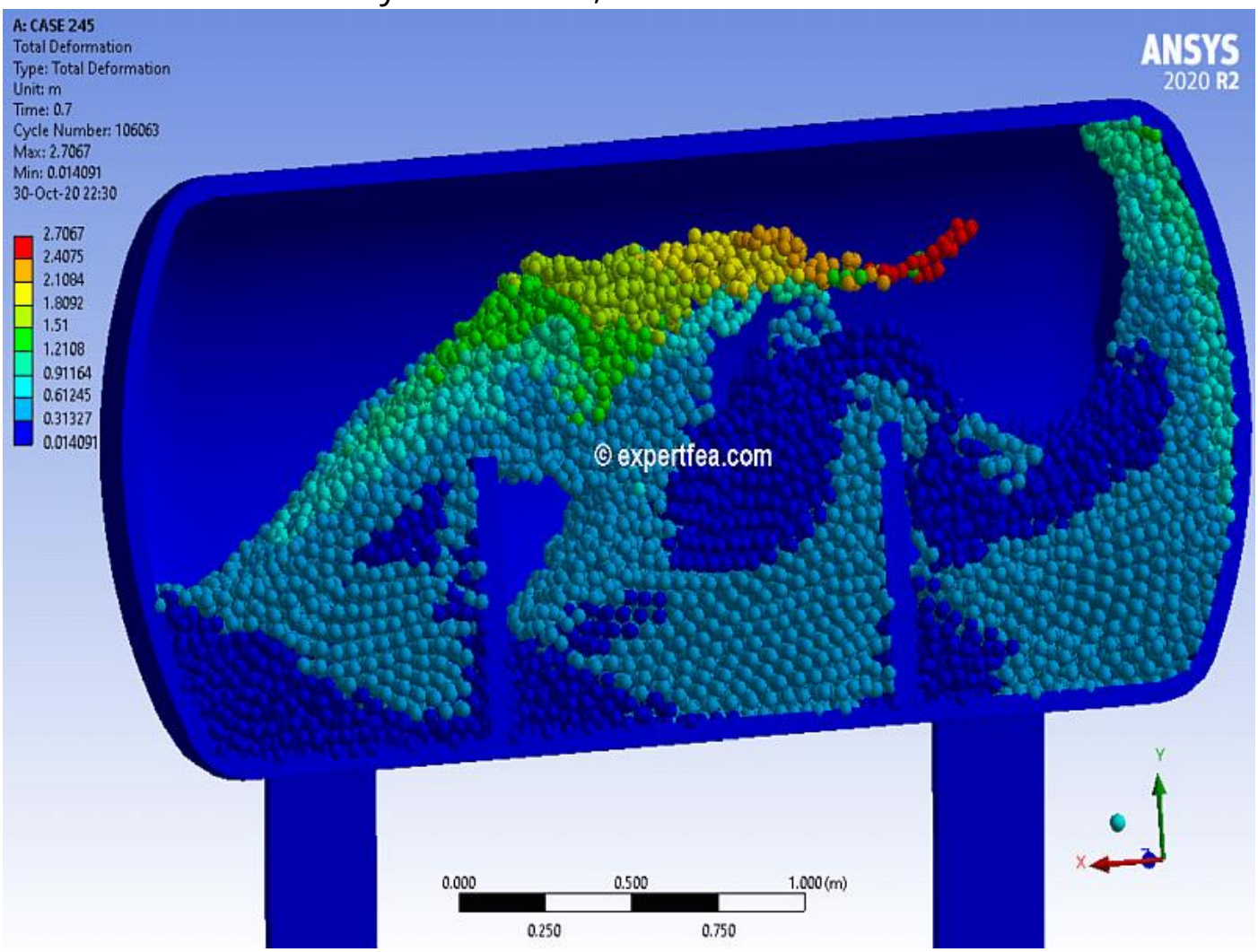


Solution: After the solving has finished, insert these items form the respective toolbar and from the Worksheet button. Also check our animated results from YouTube, if needed.



Congratulations, you have finished this FEA using Smooth Particle Hydrodynamics!

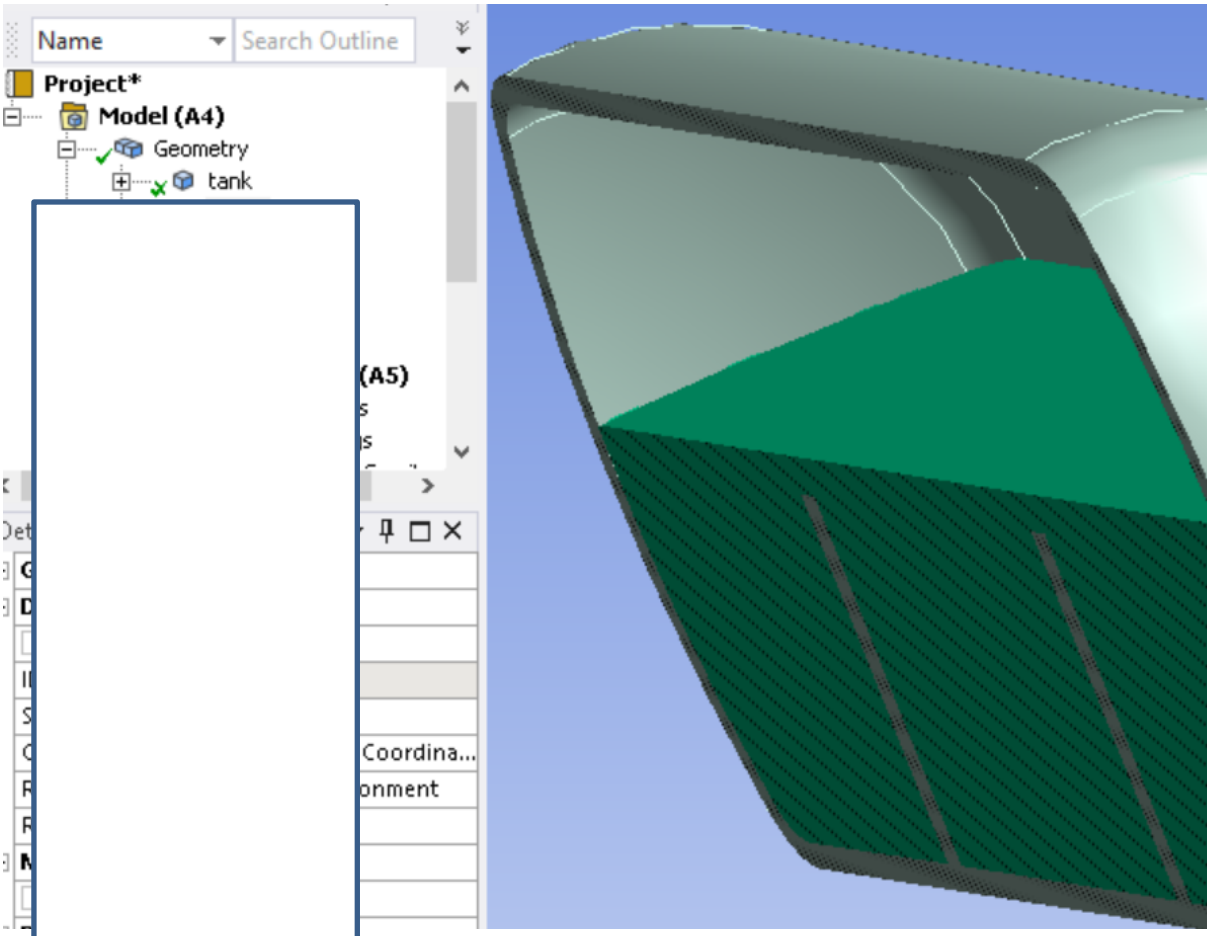
SPH8 - 245. Water tank sloshing due to seismic load - ANSYS Explicit Dynamics SPH, 1st in the World!!!



Drag and drop an Explicit Dynamics module from the Toolbox.

Import the geometry file named: *2017_nov_15_tank_sloshing_v3.x_t*

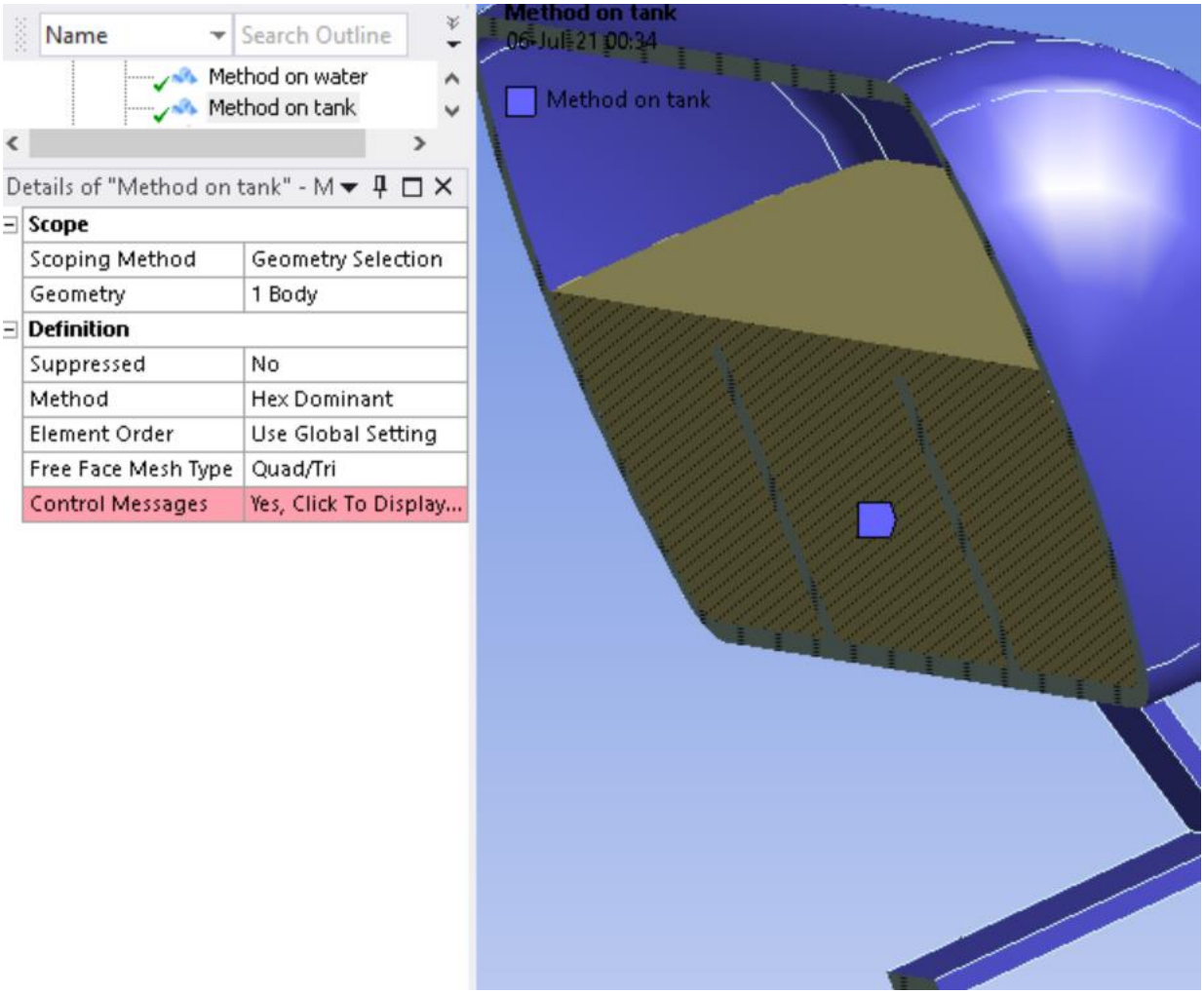
Double click Model to enter the FEA.



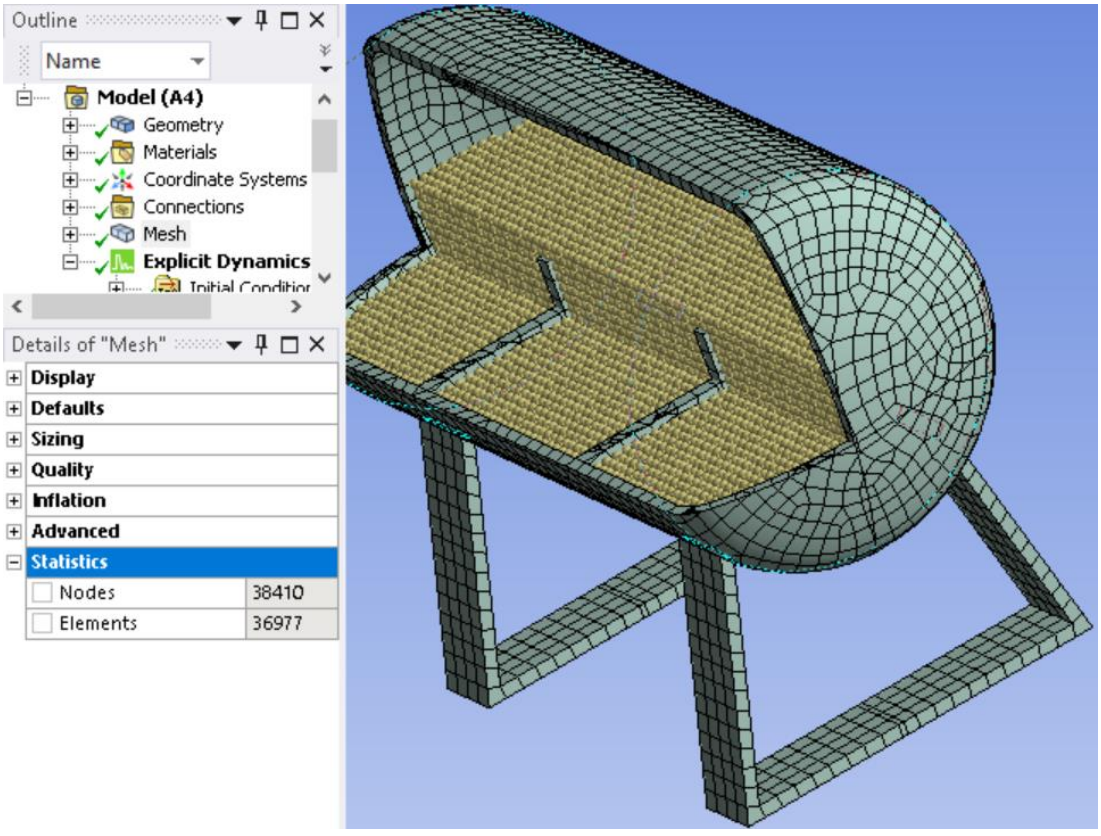
M ils.

Cr vater.

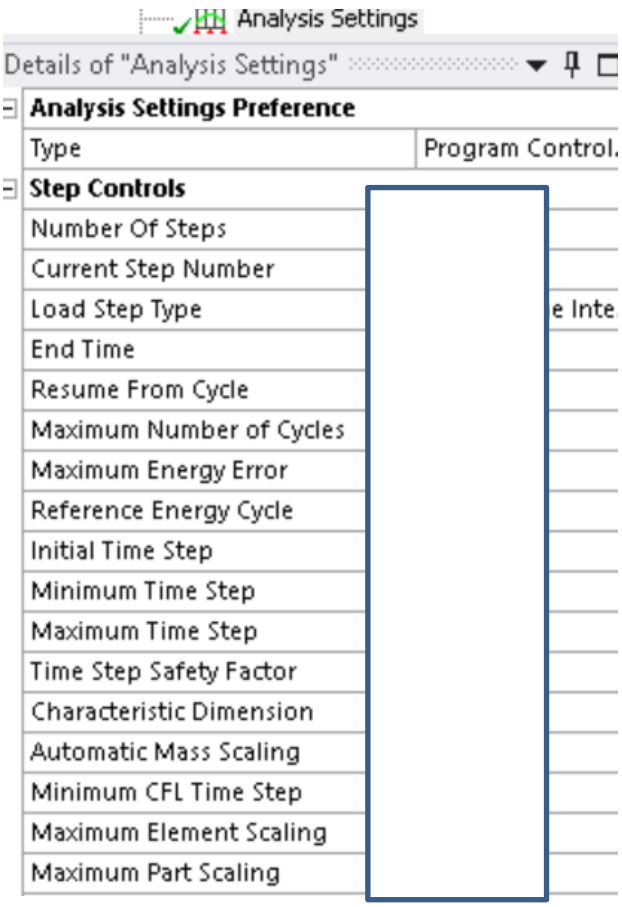
Ar k.

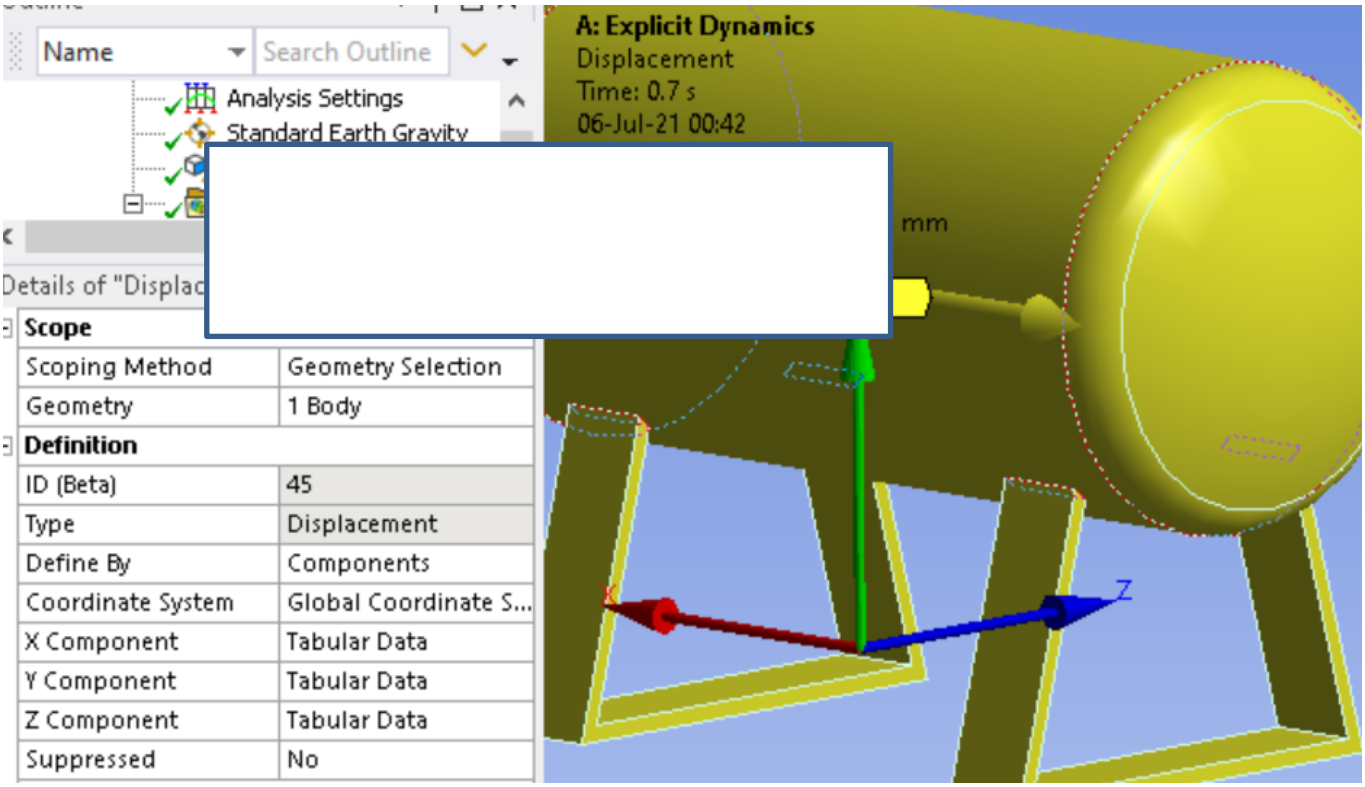


Properly made, the mesh should look like here.



Analysis Settings: Assign these details.



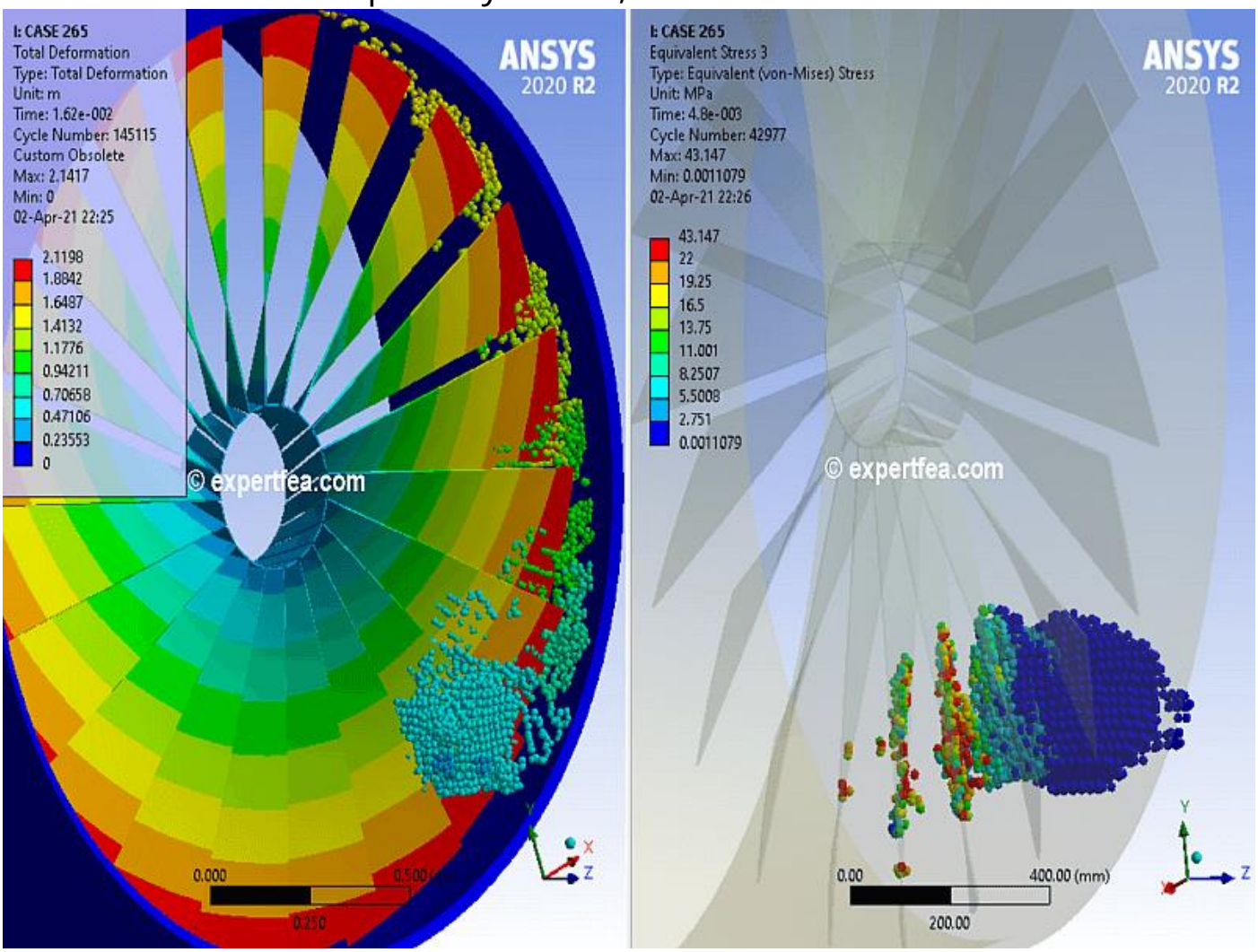


Use these values. If you cannot force the divisions of time, you might want to assign 6 steps in Analysis Settings, instead of 1. Save, Solve.

Tabular Data

	Steps	Time [s]	X [mm]	Y [mm]	Z [mm]
1	1	0.			
2	1	0.1			
3	1	0.2			
4	1	0.3			
5	1	0.4			
6	1	0.7			

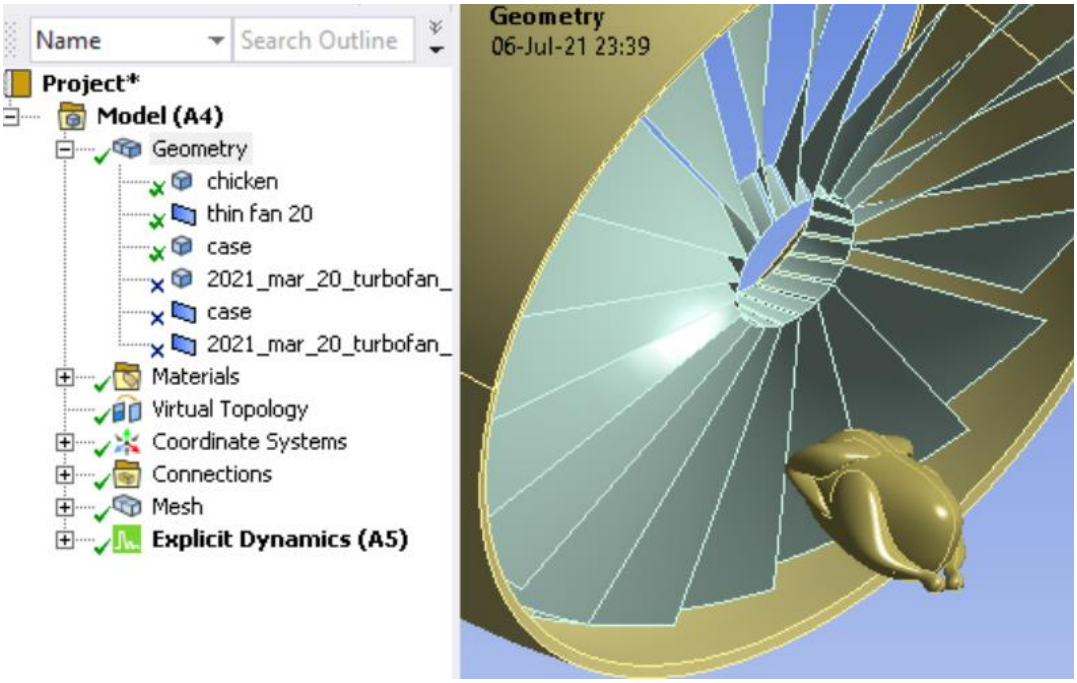
SPH13 - 265. Frozen bird strike jet engine test using SPH - ANSYS WB Explicit Dynamics, 1st in the World!!!



Drag and drop an Explicit Dynamics module from the Toolbox.

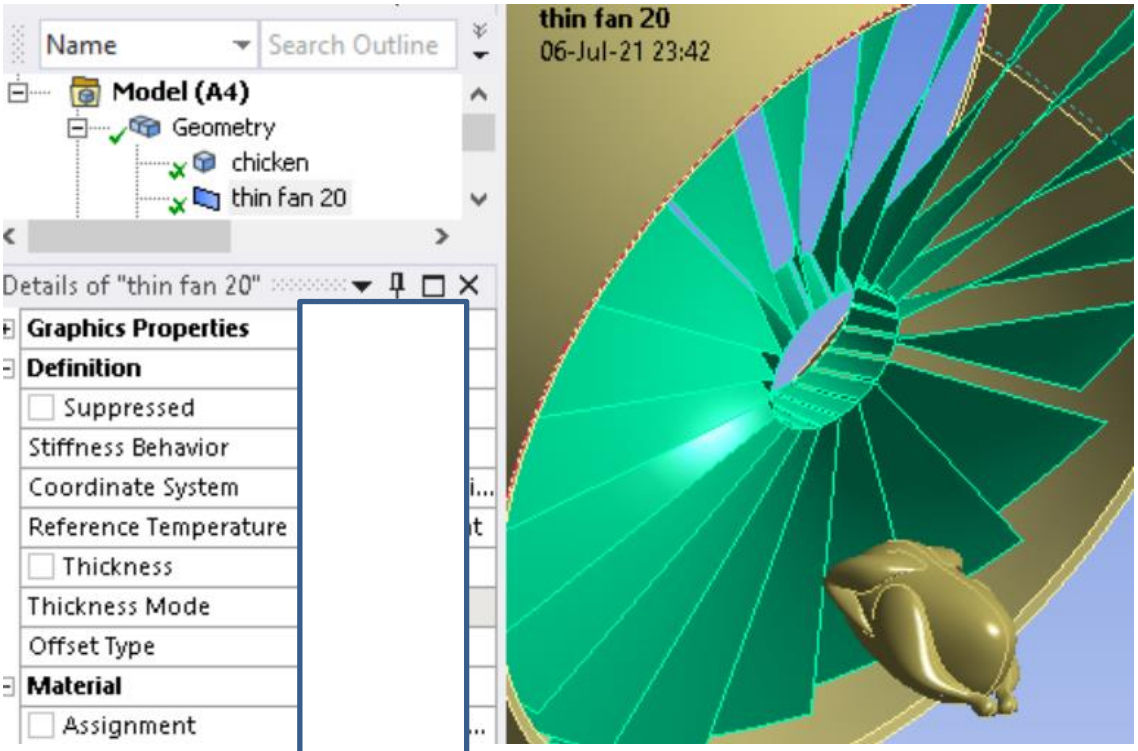
Import the geometry file named:

Enter the FEA by double clicking the Model cell.



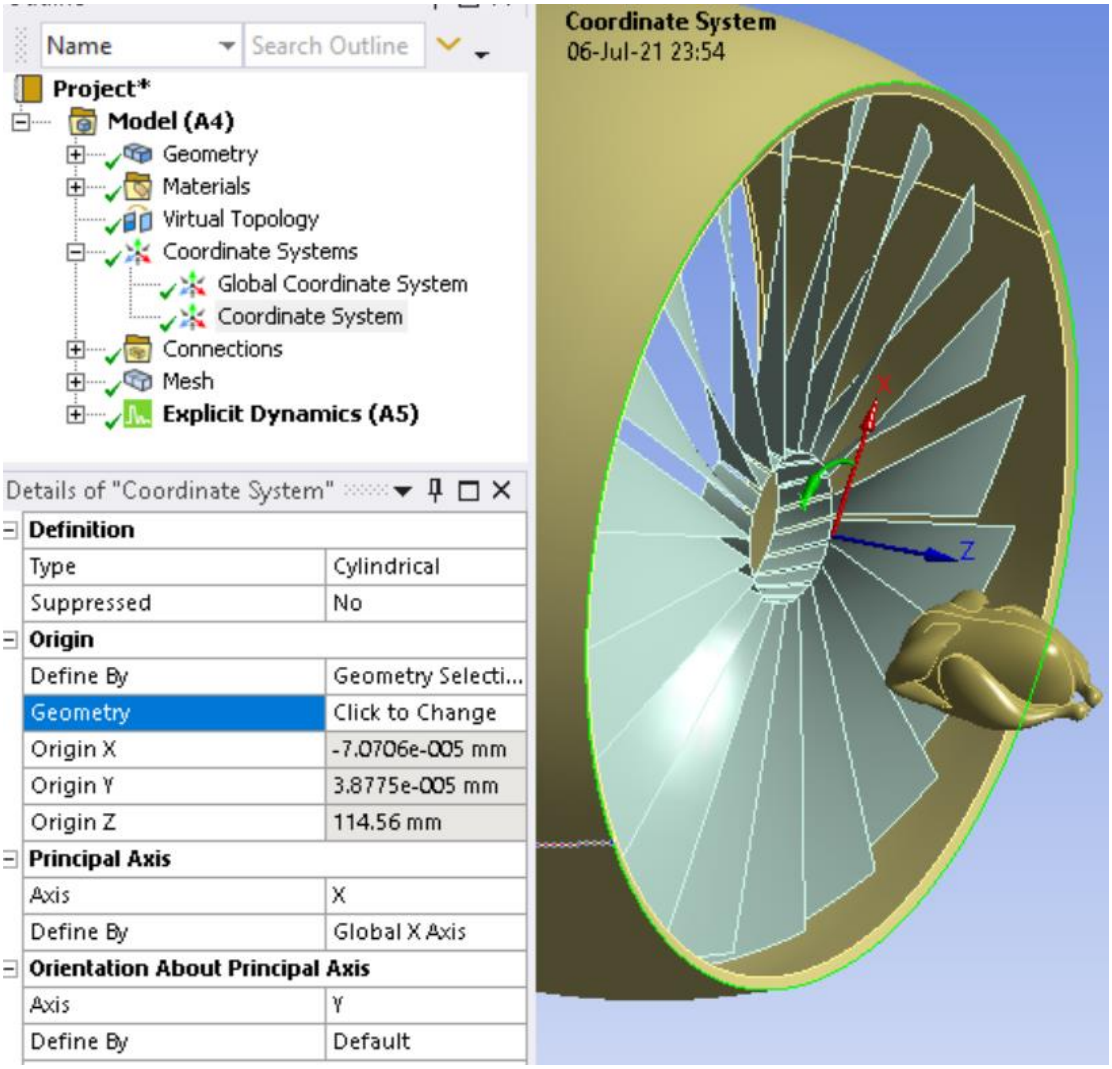
The green chicken should have these details.

This green turbo fan should have these details.



The green case shou e same material as the fan part.

Coordinate Systems elect only edges, select this green circular edge of the fan.

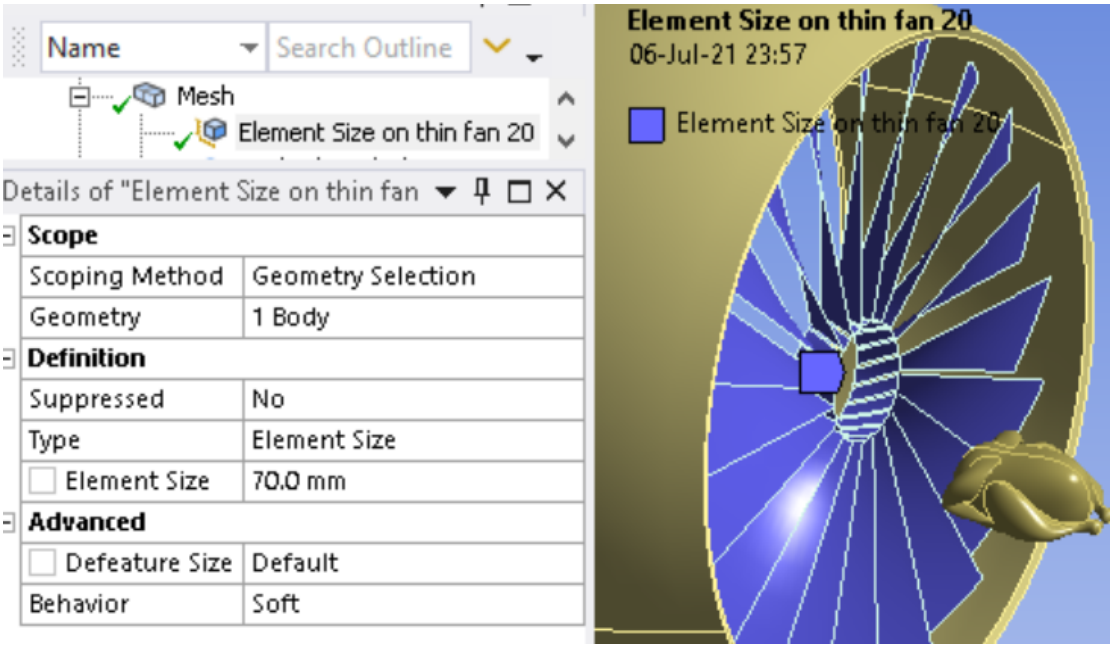


Details of "Coordinate System"

Definition	
Type	Cylindrical
Suppressed	No
Origin	
Define By	Geometry Selecti...
Geometry	Click to Change
Origin X	-7.0706e-005 mm
Origin Y	3.8775e-005 mm
Origin Z	114.56 mm
Principal Axis	
Axis	X
Define By	Global X Axis
Orientation About Principal Axis	
Axis	Y
Define By	Default

Mesh: Assign these details.

Ctrl+B to select only bodies, create this Sizing for the fan.

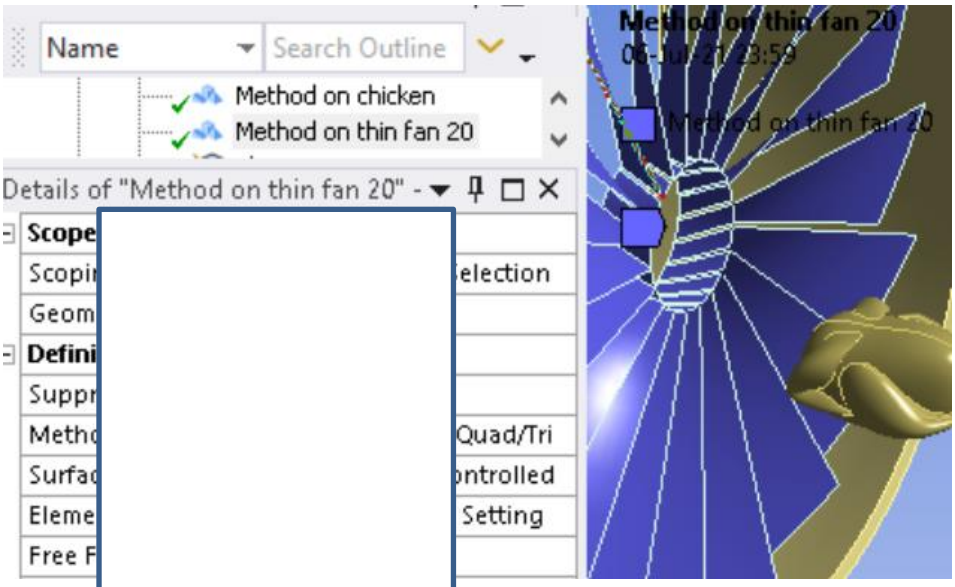


Details of "Element Size on thin fan

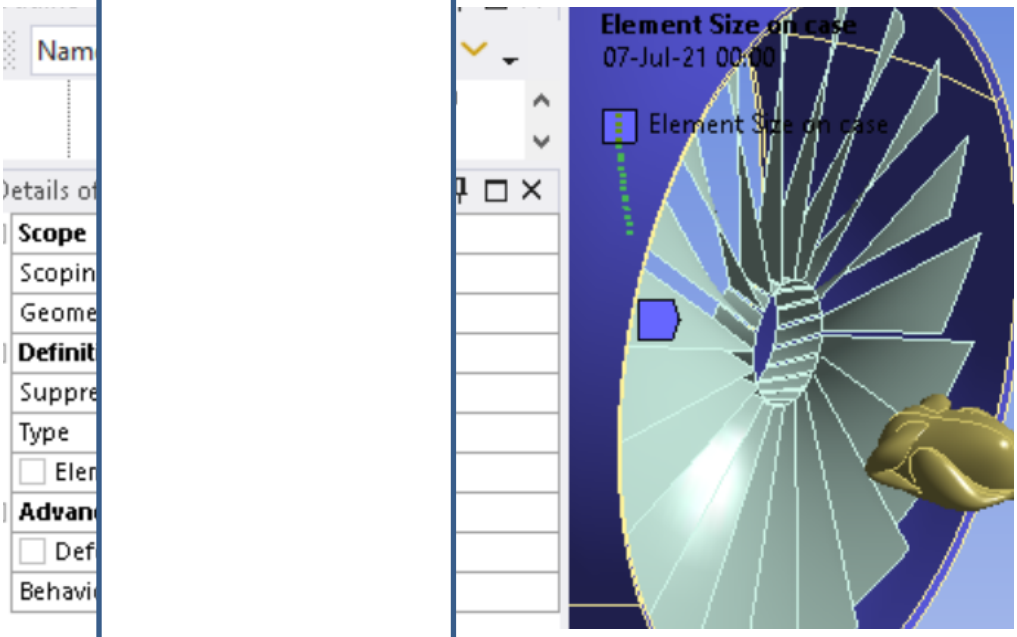
Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Type	Element Size
Element Size	70.0 mm
Advanced	
Defeature Size	Default
Behavior	Soft

Insert this Method for the SPH bird.

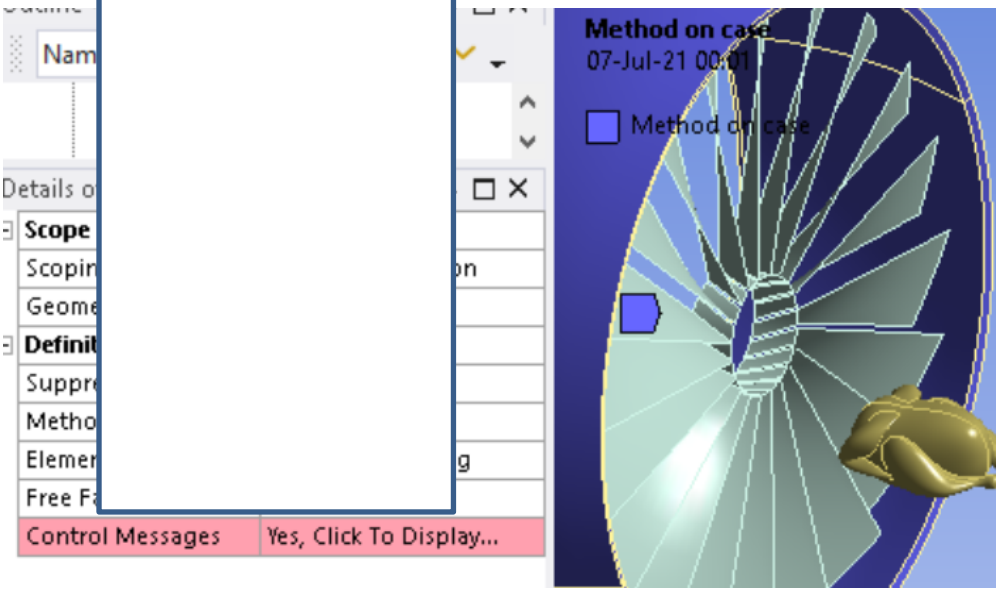
Create this Method for the fan.



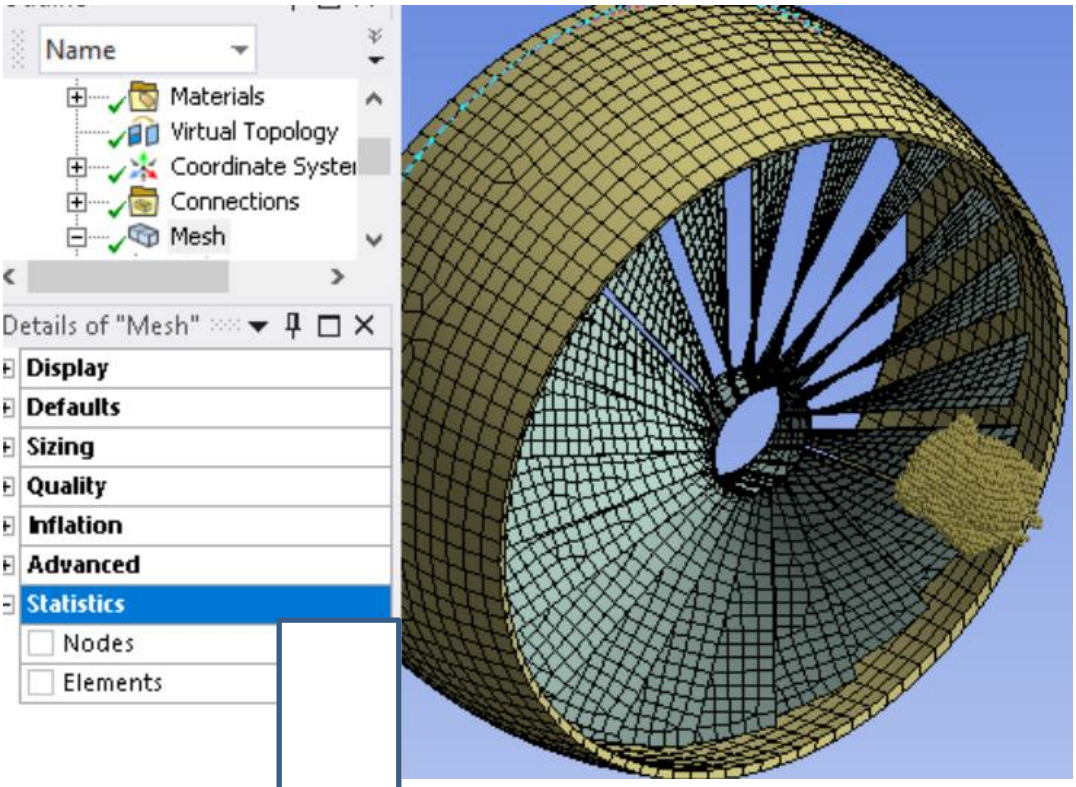
The ca ng.



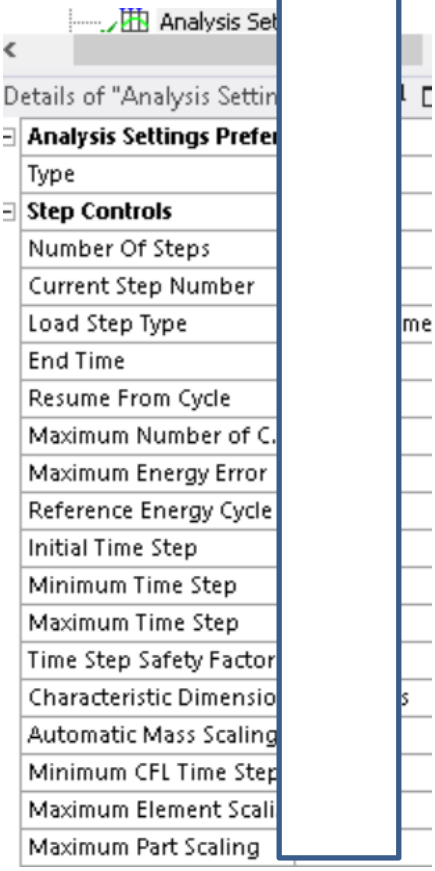
Assign enerate Mesh.



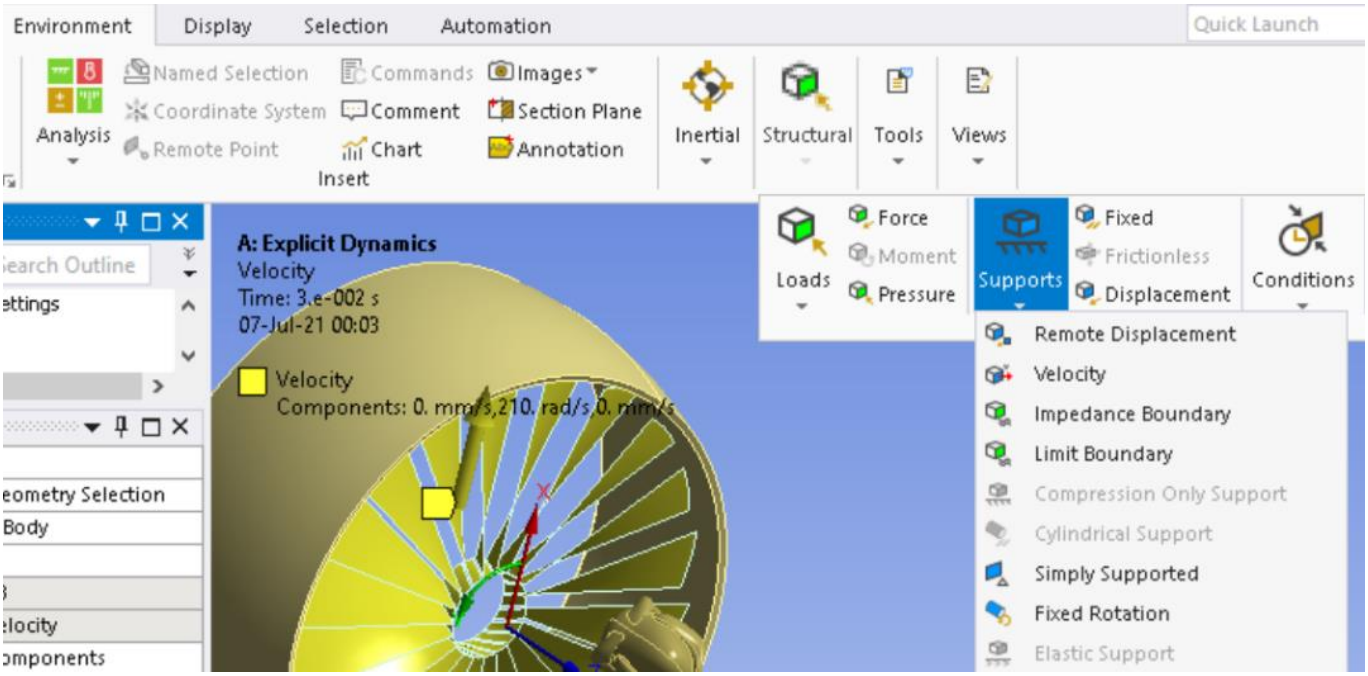
Properly made, the mesh should look like here.



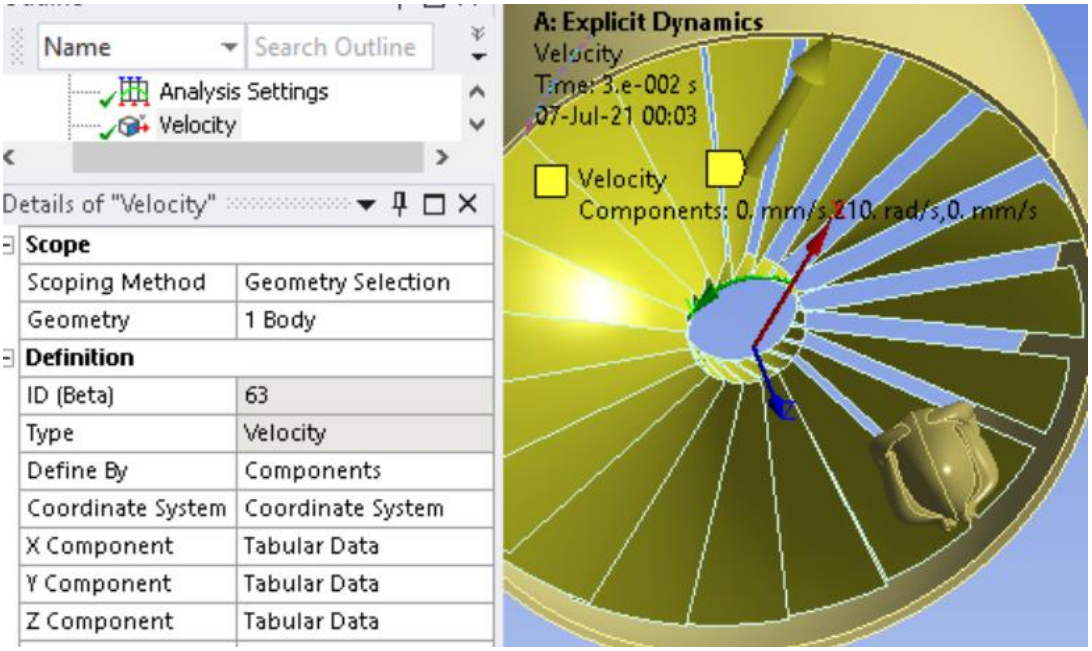
Analysis Settings: A ese details.



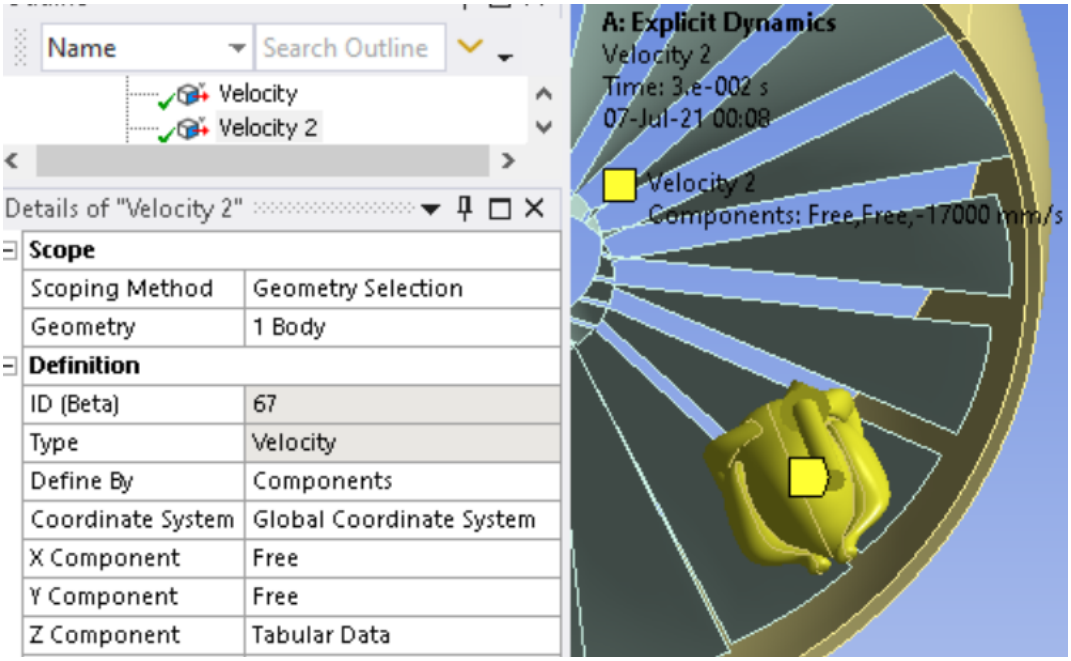
To create a Velocity, go to this pull-down menu.

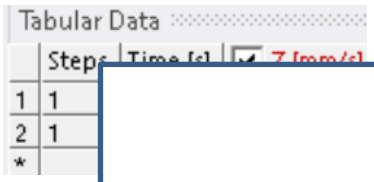


Select the yellow fan and apply the Velocity on it. Choosing the Coordinate System that we previously defined as Cylindrical, we will ensure a rotational movement, as seen here.

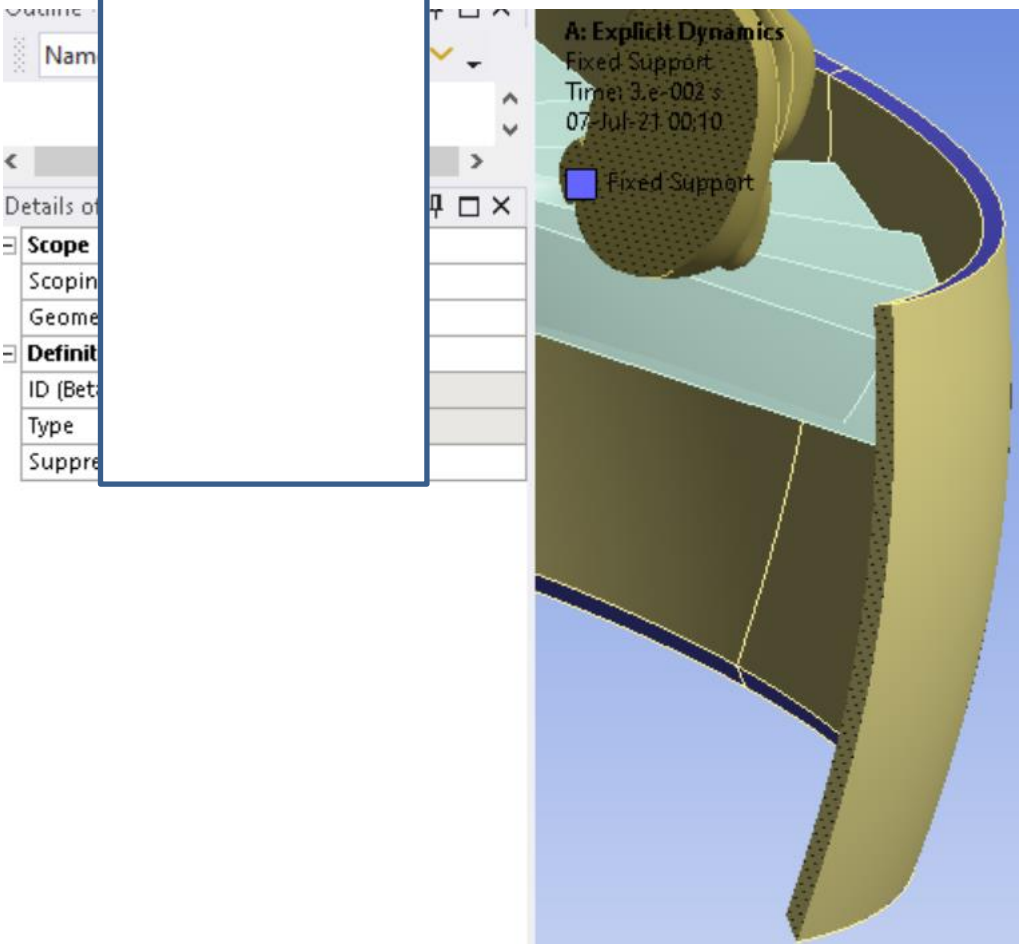


Create a Velocity relative to the Global CS for the yellow bird.



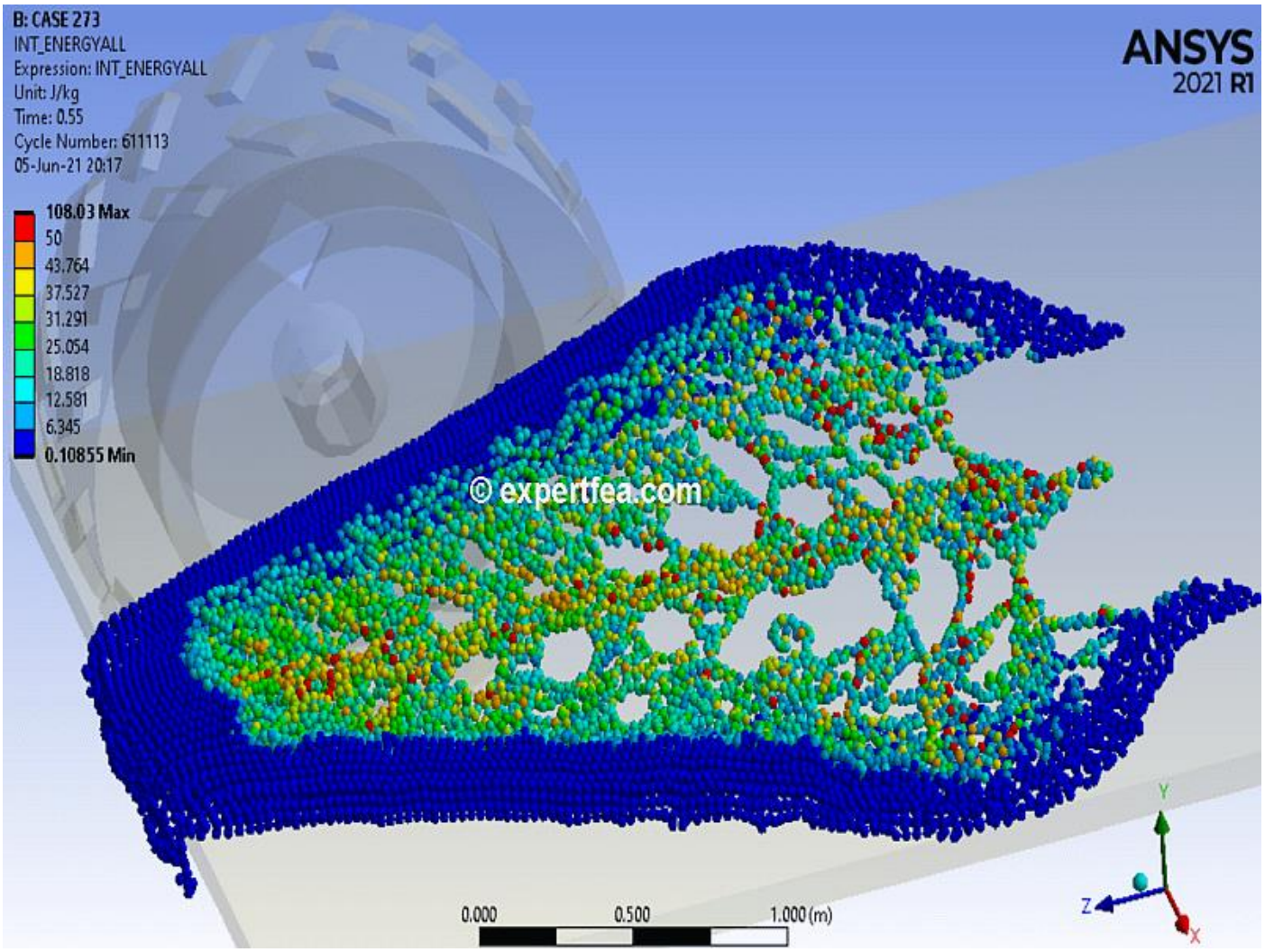


Fix the lateral of the case.



Solution: After the solving has finished, insert these items form the respective toolbar and from the Worksheet button. Also check our animated results from YouTube, if needed.

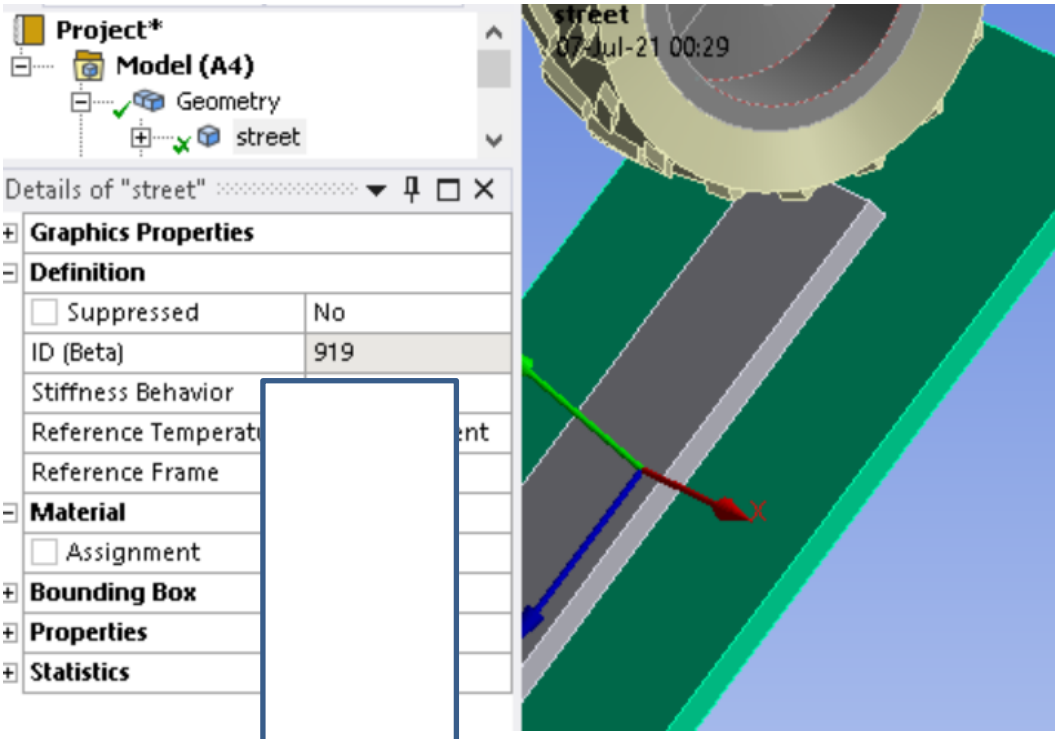
SPH14 - 273. Aquaplaning/ hydroplaning simulation with SPH - ANSYS Explicit Dynamics, 1st in the World!!!



Drag and drop an Explicit Dynamics module from the Toolbox.

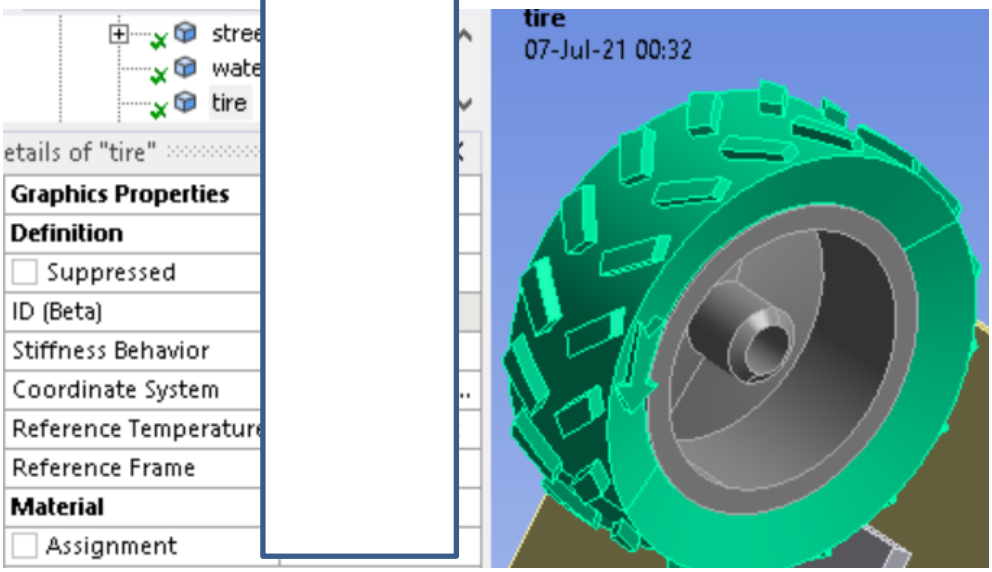
Import the geometry file named: *2021_jun_05_aquaplanning_v6.stp*

Assign these details to the green street.

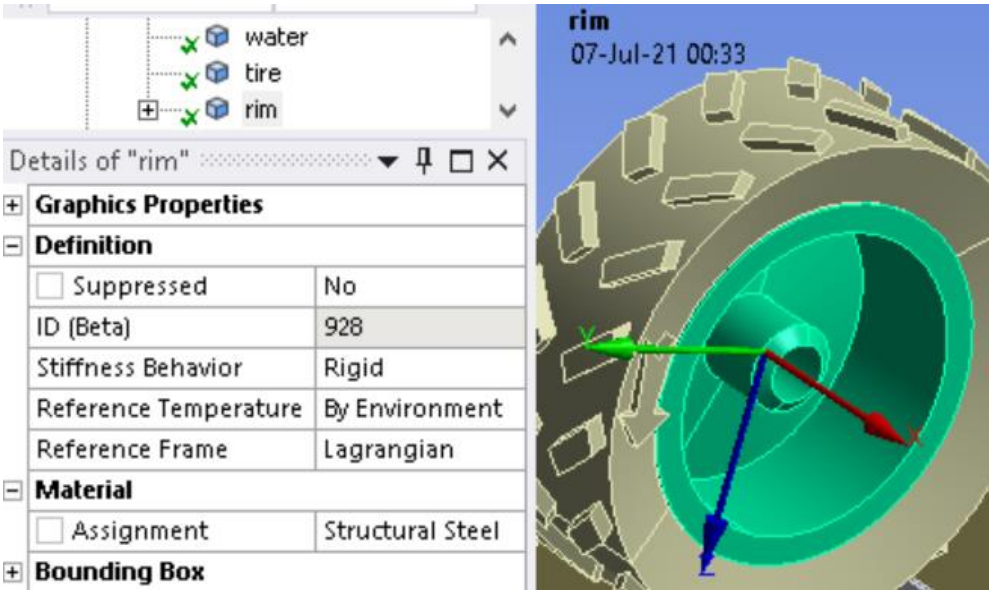


The green water s hese details.

This green tire sh se details.



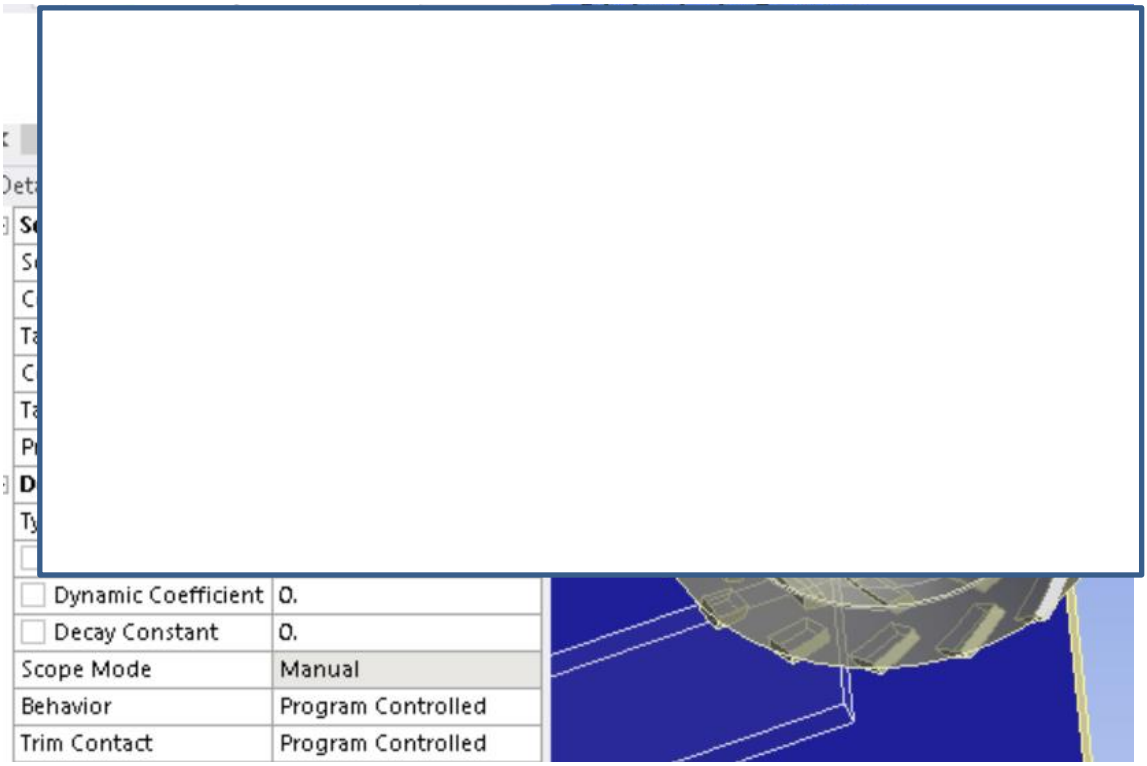
Insert these details on the green rim.



Coordinate Systems: Create one inside the rim, as seen green here.

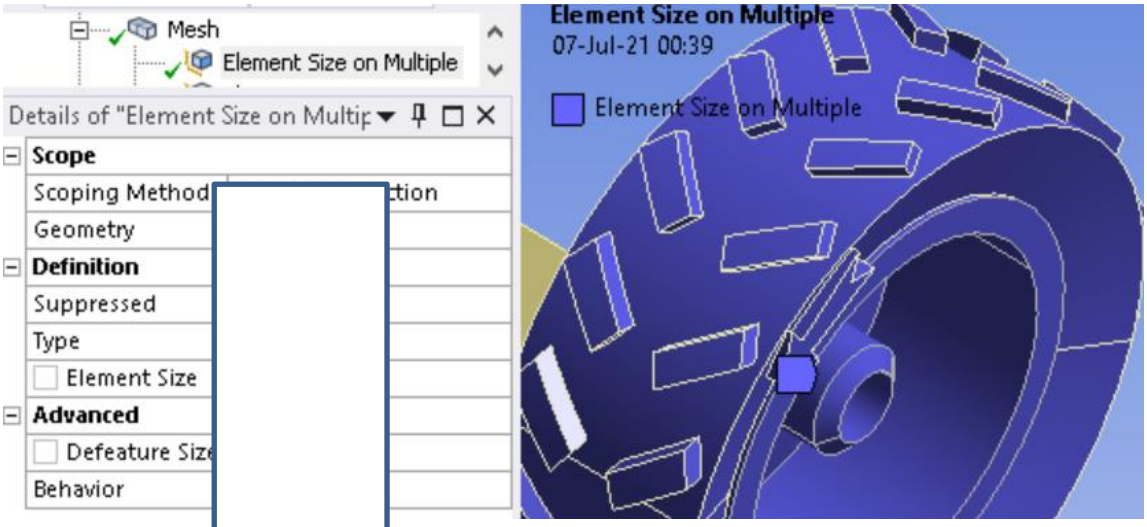
Connections: Delete the existing contacts.

Create this contact between the tire and the street.



Mesh: It should have these details.

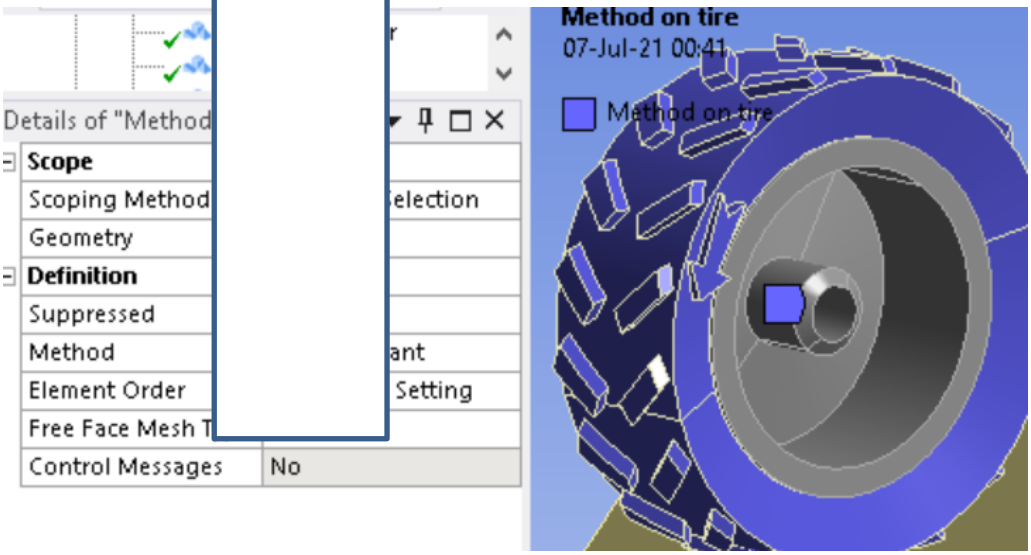
Assign this Sizing for tire and rim.



Create this Siz t.

Assign this Me SPH water.

The tire shoul Method, too.

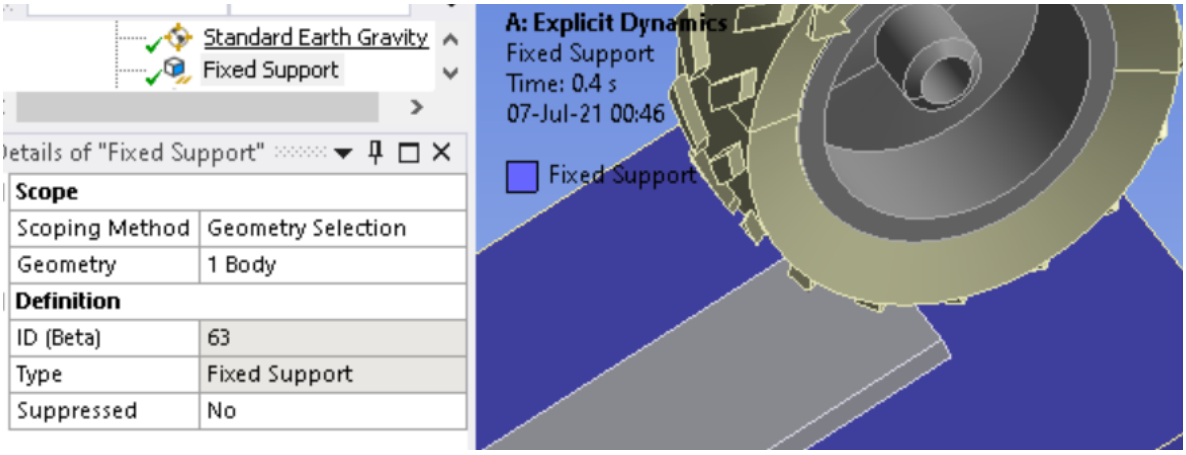


Create this Method for rim.

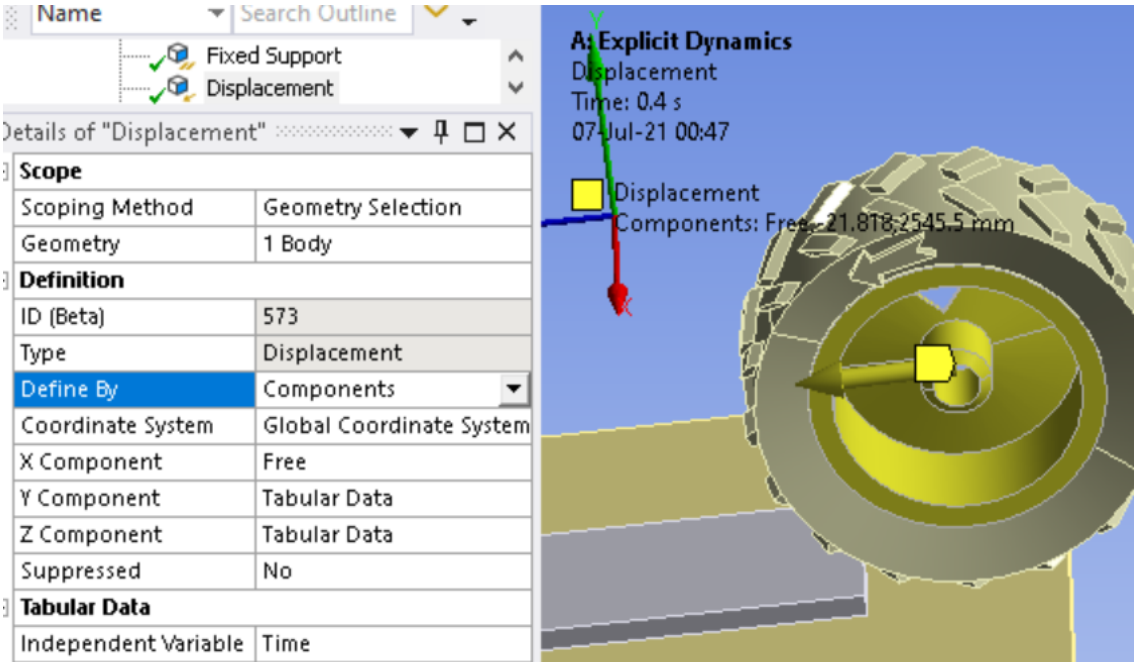
Analysis Settings: It should have these details.

Apply such a gravity.

Fix the street.

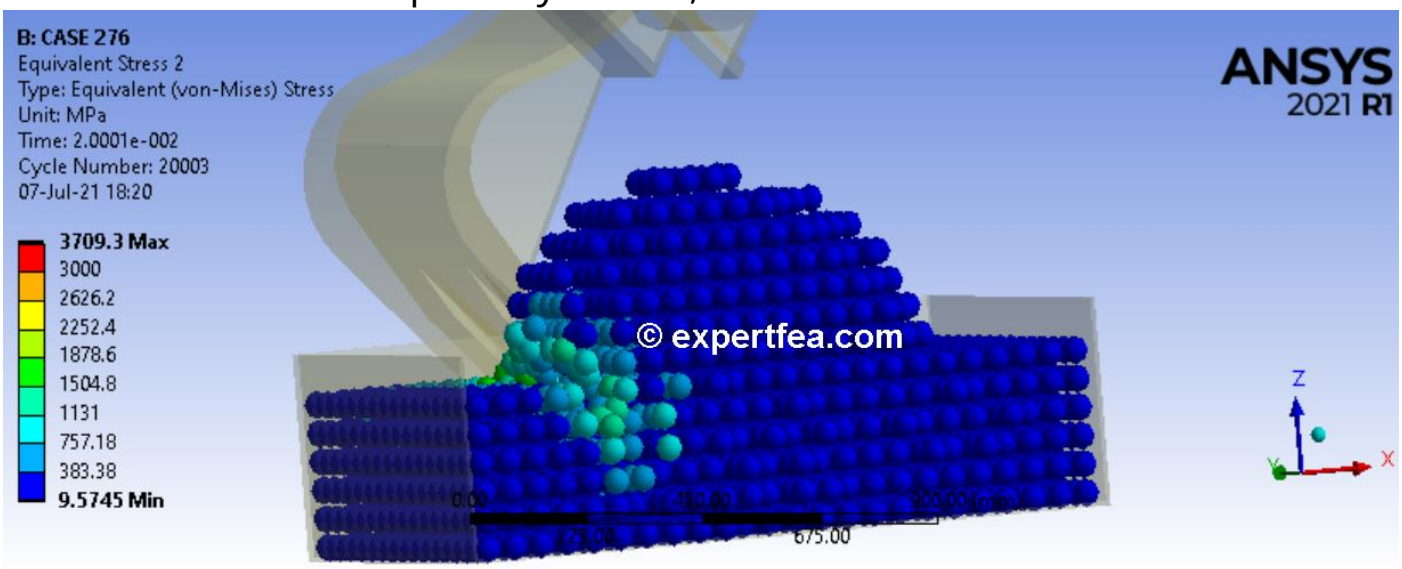


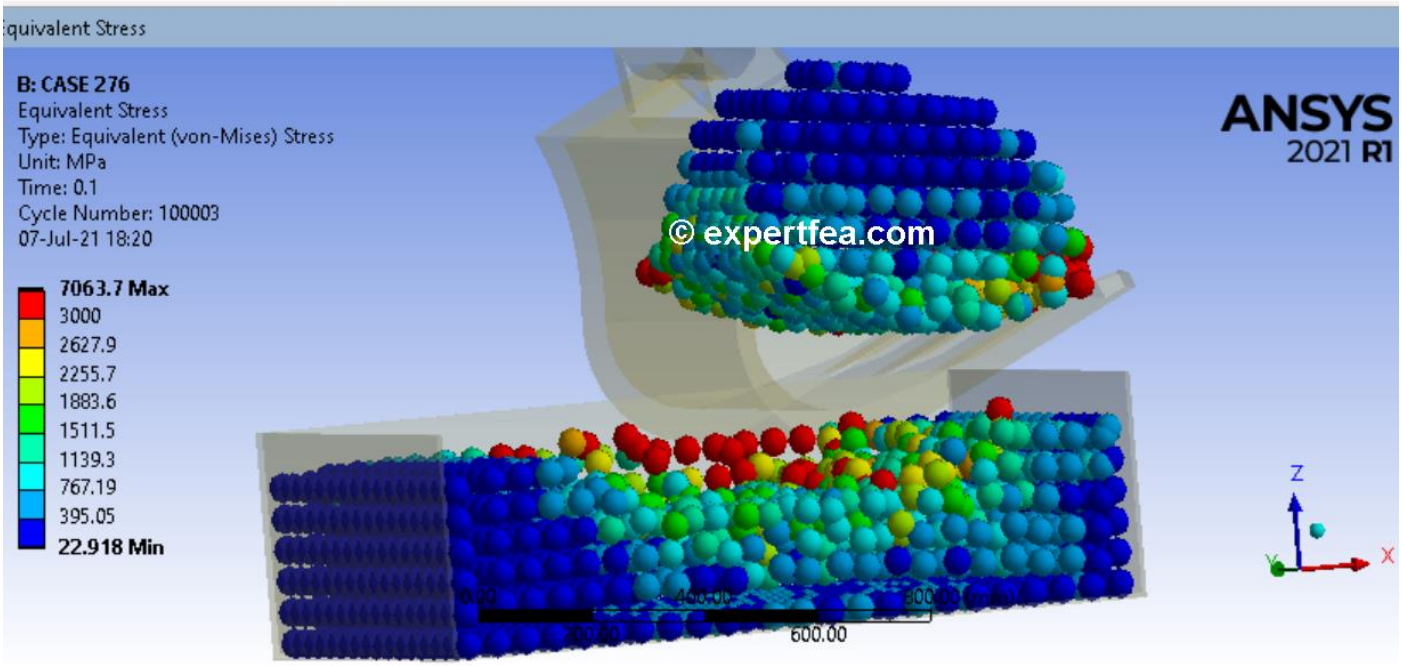
Apply this Displacement on the rim.



To ensure that the tire does not fall off the rim, insert this Displacement for the outer large face of the rim, yellow here.

SPH15 - 276. Excavation simulation of sandstones using SPH - ANSYS Explicit Dynamics, 1st in the World!!!



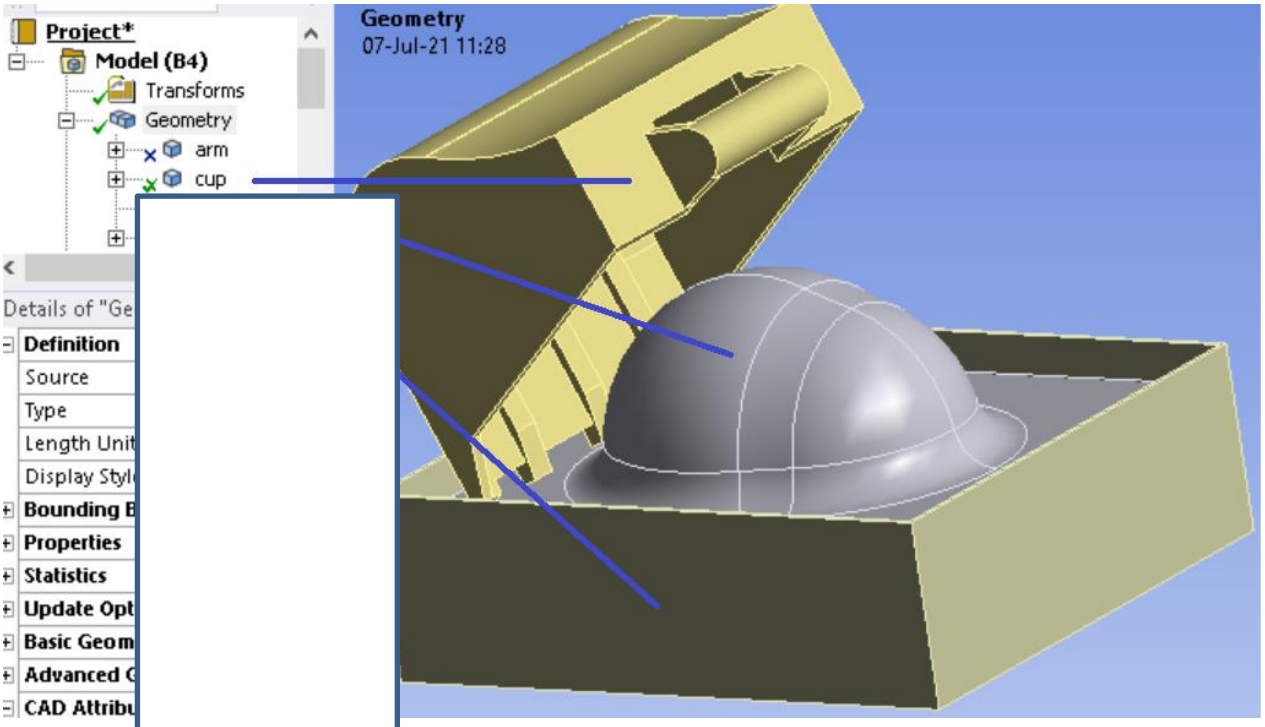


Drag and drop an Explicit Dynamics module from the Toolbox.

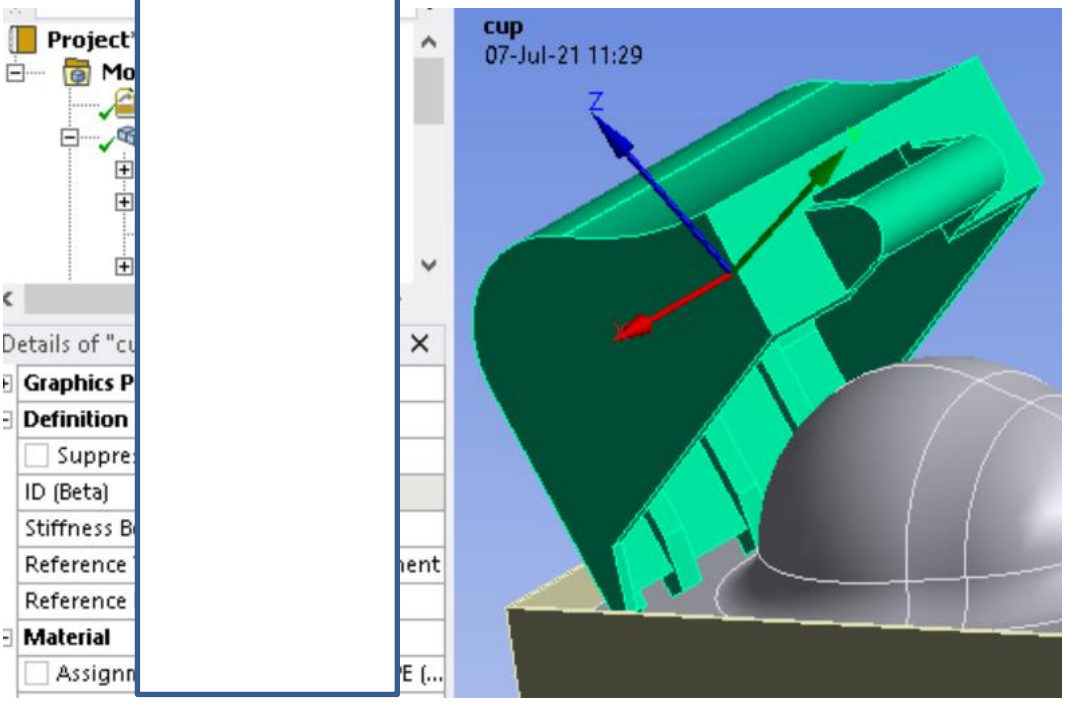
Import the geometry file named: *2021_jun_09_excavator_rocks_SPH_v2.stp*

Enter the FEA by double clicking the Model cell.

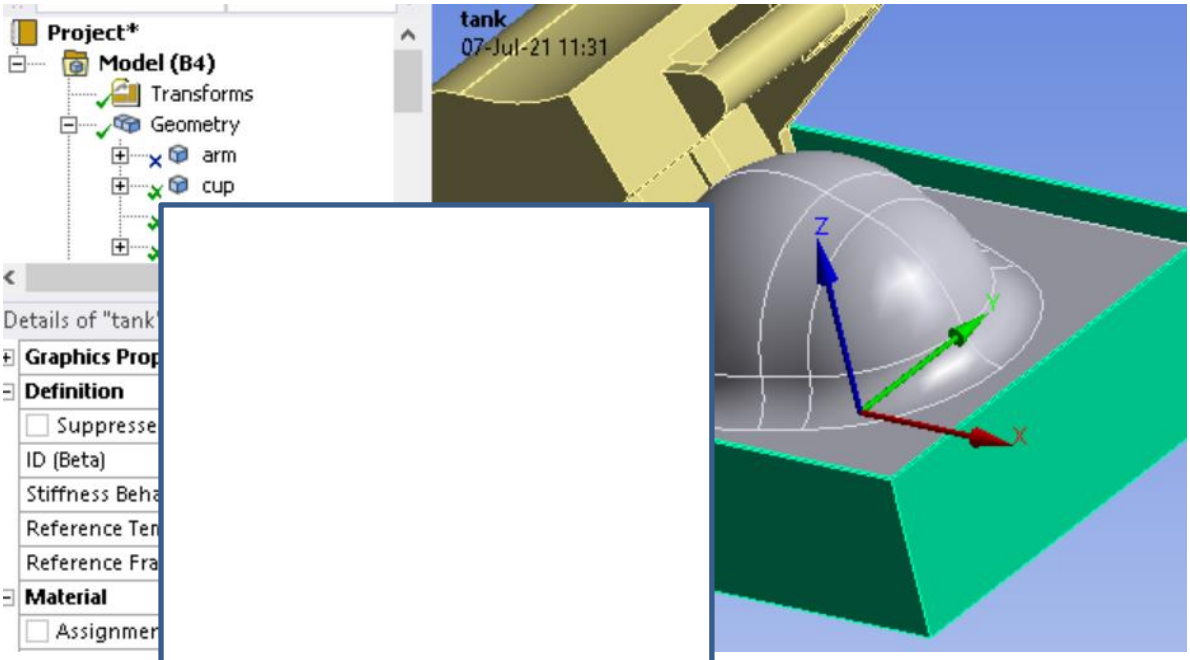
Geometry: Ensure this naming scheme. Suppress the arm part, not needed for this FEA.



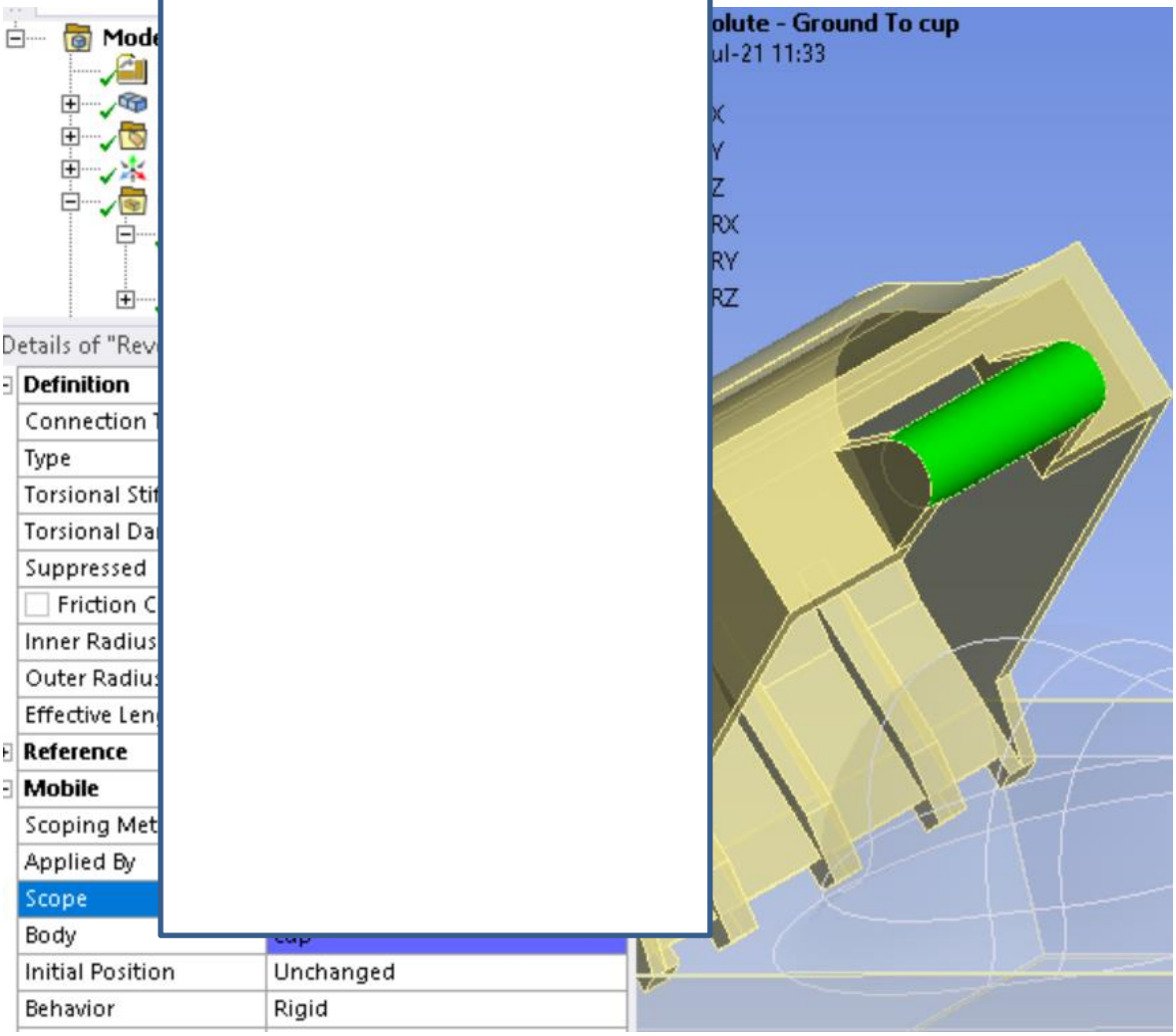
Assign th o.



Insert these details for tank.



Connectio cup, then go to Body-Ground, Revolute.



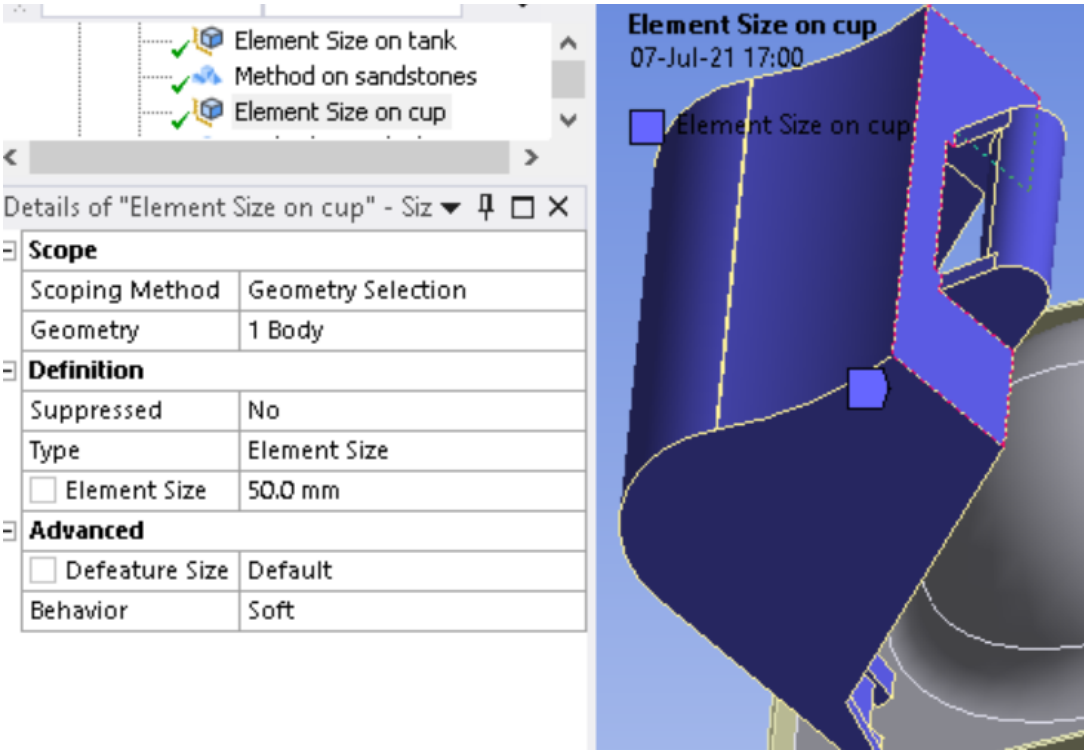
Press Ctrl+B to select only bodies, click the sandstones and the tank, blue here, then click the Body Interaction button and assign these details.

Mesh: Ensure these details.

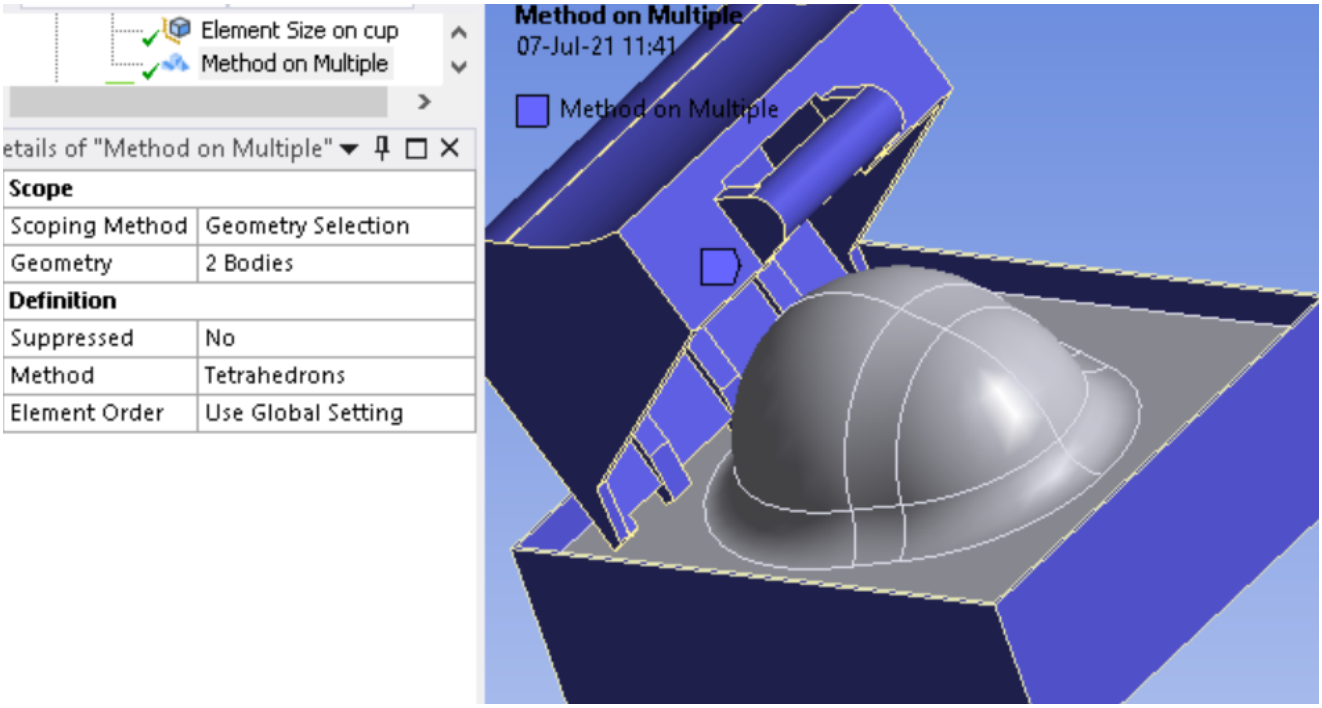
Apply this Sizing on the tank.

Create this Method for the SPH sandstones.

Assign this method for the cup.



Insert this Method for the cup and tank.



Analysis Settings: Ensure these details.

Analysis Settings

Details of "Analysis Settings"

Analysis Settings Preference	
Type	Custom
Step Controls	
Number Of Steps	1
Current Step Number	1
Load Step Type	e Int..
End Time	
Resume From Cycle	
Maximum Number o	
Maximum Energy Er	
Reference Energy Cy	
Initial Time Step	
Minimum Time Step	
Maximum Time Step	
Time Step Safety Fa	
Characteristic Dimer	
Automatic Mass Sca	
Minimum CFL Time	
Maximum Element S	
Maximum Part Scalir	

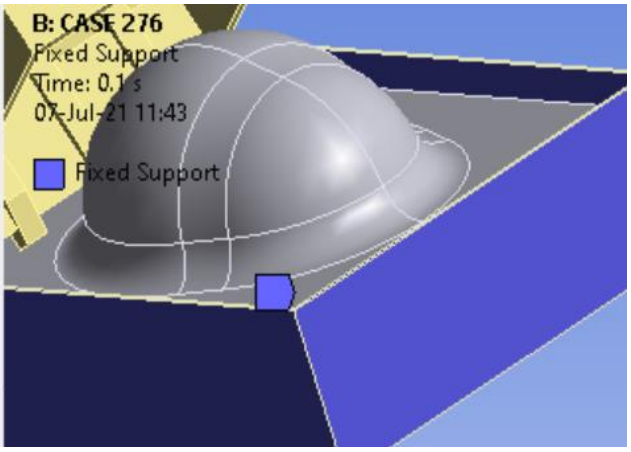
Prescribe the gra

Fix the tank.

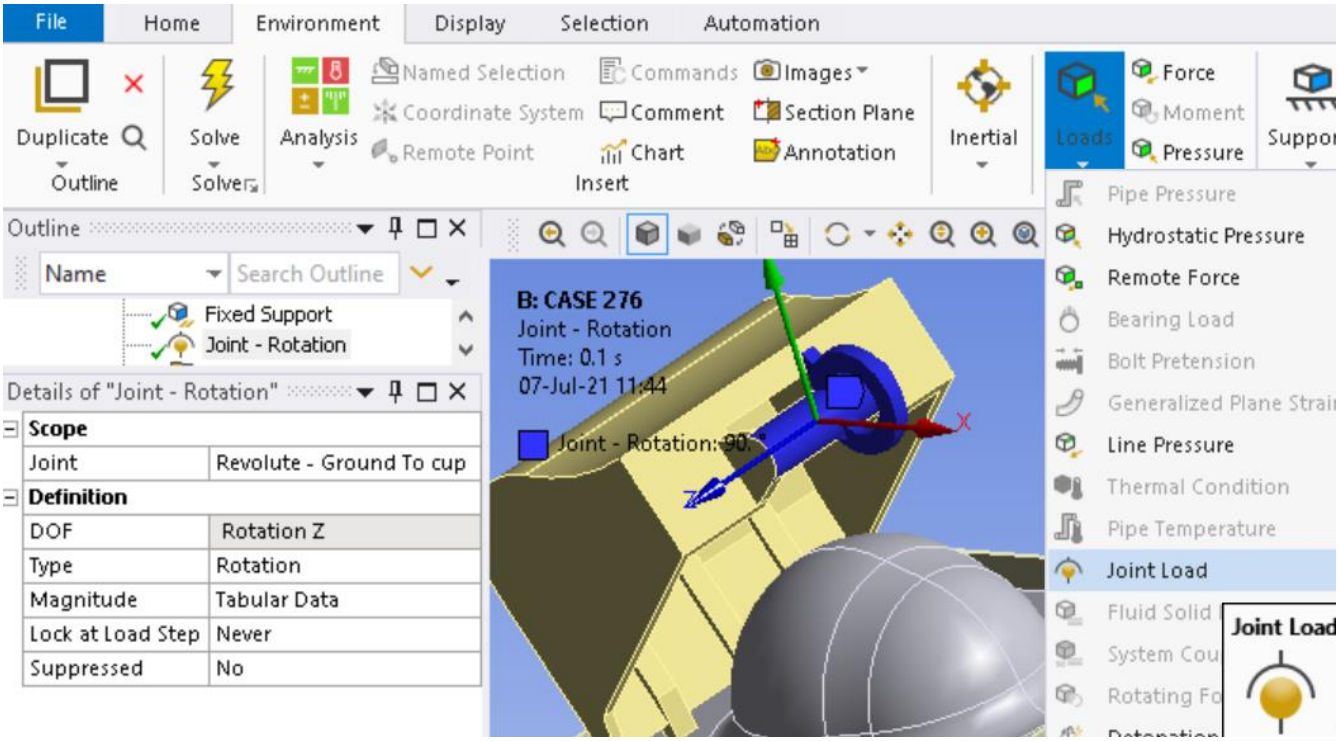
Star
Fixe

Details of "Fixed Suppor

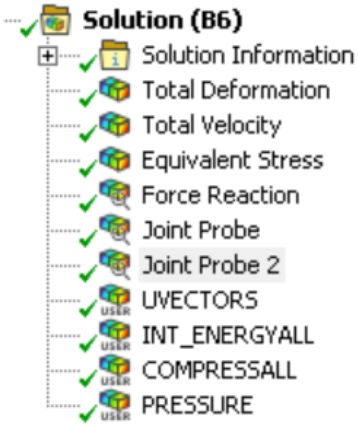
Scope	
Scoping Method	Geo
Geometry	1 Bo
Definition	
ID (Beta)	94
Type	Fixe
Suppressed	No



From Joint, Loads . Save, Solve.



Solution: After the solving has finished, insert these items form the respective toolbar and from the Worksheet button. Also check our animated results from YouTube, if needed.



Congratulations, you are now fully initiated in using Smooth Particle Hydrodynamics inside the default Explicit Dynamics module! Good luck with your newly gained expertise, practice and perfect your craft even more, and look forward for our next volumes on this topic!

Claudiu Danila, 2021