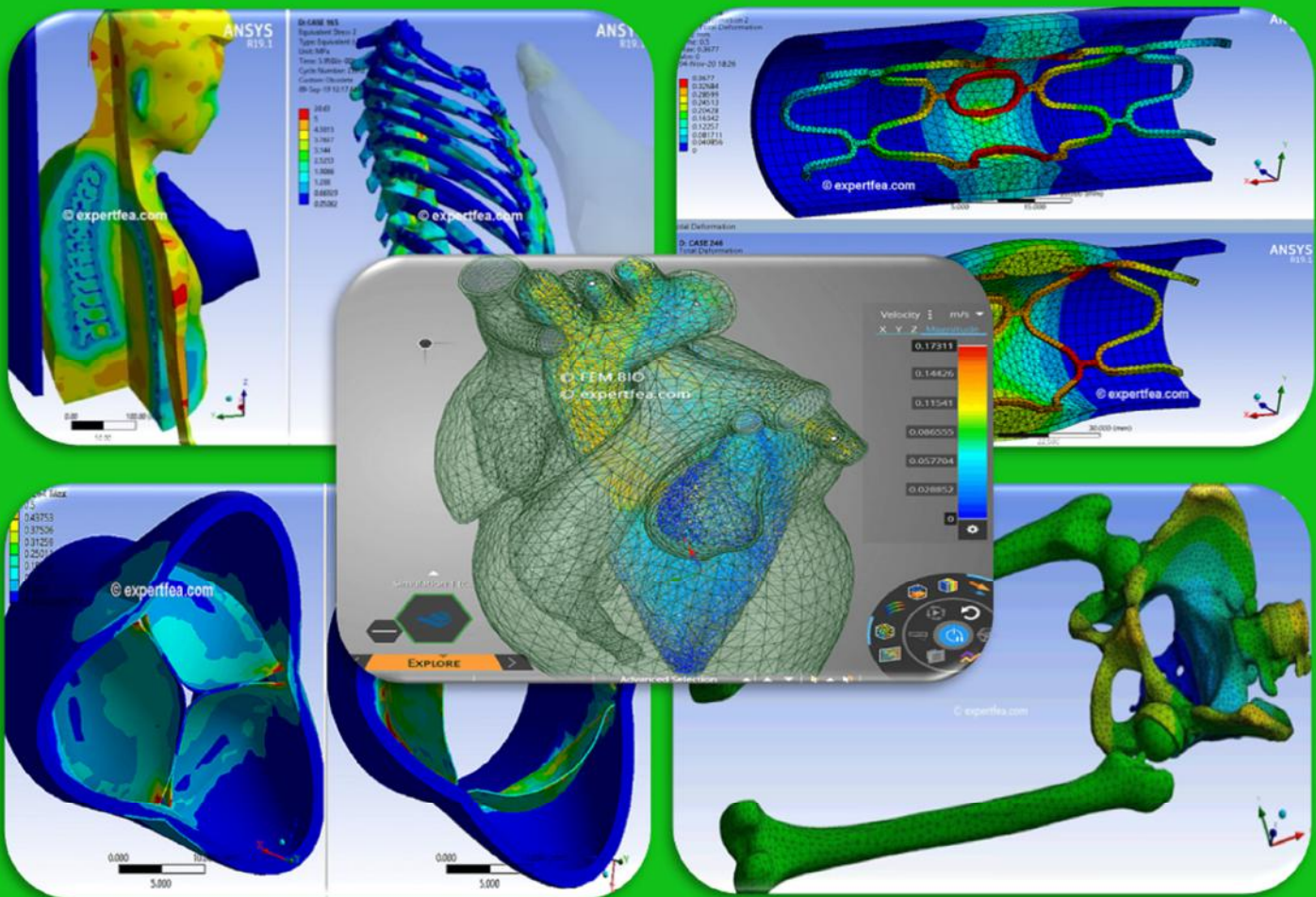


# BIOMECHANICS MODELING & SIMULATIONS MADE EASY

- vol. 1 -

(Intro to Biomechanics, Modeling & Simulations  
+2 Modeling Tutorials in ANSYS Spaceclaim  
+9 FEA Tutorials in ANSYS Workbench  
+ 1 CFD Tutorial in ANSYS Discovery)



**We dedicate this book to God, the Only Reality and the Supreme Mystery**



**Content:**

<b>Foreword .....</b>	<b>4</b>
<b>Introduction to Biomechanics, modeling and simulations .....</b>	<b>5</b>
<b>Assumptions and strategies before modeling and simulations .....</b>	<b>7</b>
<b>Typical simulation steps or workflow in ANSYS .....</b>	<b>9</b>
<b>On the usage of material properties (Engineering Data) .....</b>	<b>11</b>
<b>Obtaining proper biomechanical geometry .....</b>	<b>15</b>
<b>Types of contacts or interactions between parts .....</b>	<b>22</b>
<b>Types of finite elements and mesh .....</b>	<b>24</b>
<b>Types of boundary conditions .....</b>	<b>27</b>
<b>Solver and time settings .....</b>	<b>29</b>
<b>Solution items and post-processing .....</b>	<b>32</b>
<b>2 modeling tutorials with geometry conversions in Spaceclaim .....</b>	<b>34</b>
<b>5 FEA simulations made in Static Structural .....</b>	<b>80</b>
<b>4 FEA simulations made in Explicit Dynamics .....</b>	<b>227</b>
<b>1 CFD simulation made in ANSYS Discovery .....</b>	<b>327</b>

## Foreword

Hi all!

Welcome to the 1<sup>st</sup> book in the world with hands-on applied Biomechanics modeling and simulations, a book that the world needed in the last 10-20 years, without any exaggeration.

This book is meant to help the absolute beginners in simulating the behavior of simple then complex biomechanical assemblies, explaining in minute details how the simulation domain should be approached, how to obtain the proper geometry, and what ANSYS options are available to assign the proper boundary conditions, discretization and solver settings for reliable, thus useful results.

The volume starts with introductory elements in Biomechanics, modeling and simulations, then 2 modeling tutorials in Spaceclaim follow, after that 9 simulation tutorials are presented in order to gradually accustom the user with the needs of the full simulation procedure using ANSYS products.

Besides the PDF file, we also offer the 3D models on which the tutorials should be applied and successfully solved.

This is the 1<sup>st</sup> volume we made in Biomechanics modeling and simulations, so we presented some of the easier scenarios found on FEM.bio and expertfea.com – thus we opened the way for the more advanced simulations to appear in the next Biomechanics volumes. Be sure to also attempt doing the homework assignments, in order to acquire experience via trials and errors, followed by full successes in this awesome domain!

As always, we wish you all the best and believe in yourself and in your bright future! If you don't do it, then who else is going to do it for you? So, start here and now! 😊

**Claudiu, 28th of November 2022**



## Introduction to Biomechanics, modeling and simulations

Here is our definition of Biomechanics: “The science of how the living systems react when acted upon by certain mechanical factors. These processes can be modeled and analyzed with the help of simulation software”.

Britannica.com says that “Contemporary biomechanics is a multidisciplinary field that combines physical and engineering expertise with knowledge from the biological and medical sciences.”

So, in this book we attempt to unite the efforts and knowledge of the engineers with the ones of medics or biologists. Nowadays, given the advance of science, biologists can analyze and obtain various data regarding the behavior or the state of certain tissues, organs or systems, via microscopy processes, or via radiology, CT and RMN. Engineer and physicists can analyse and simulate in a virtual environment (with the help of computer programs) the same data and functions, but without the need of cells, tissues, or organs prelevated or tested on live patients.

Due to the fact that the costs of physical tests are very high when conducting research activities which need to be applied for certain categories and ages of people, a great need arises to find out the behavior and the physiology of certain biological parts using virtual life-like representations, or 3D models obtained either from conversion of CT or RMN slices into 3D parts or 3D models built by talented graphicicians who use 3D software like Maya, Blender, 3D Studio Max to achieve the same accuracy of the organs' shapes and sizes.

In both cases mentioned above, the resulting 3D models will further be imported to simulation software in order to be (virtually) subjected to certain forces, velocities, temperatures, voltages etc. A great advantage is that in a virtual environment, the simulator can “push the envelope” by exaggerating certain loads or restraints, in order to find out the limits of the respective organs or prosthetics or protective gear – without causing the harm that would have appeared with a live subject, *in vivo*.

Even though the suffix “mechanics” concerns only with forces, pressure, moments and similar restraints, we can certainly extend the scope of simulations also to the thermal domain, to the electromagnetical domain or even to the optics domain. After all, any action, be it mechanical, thermal, electrical or optical, it will cause a visible result on the human body, sooner or later, given that the human systems signal, react and adapt to all disturbing factors, in order to establish a state of equilibrium or a state of normal function, which is generally defined as *health*.

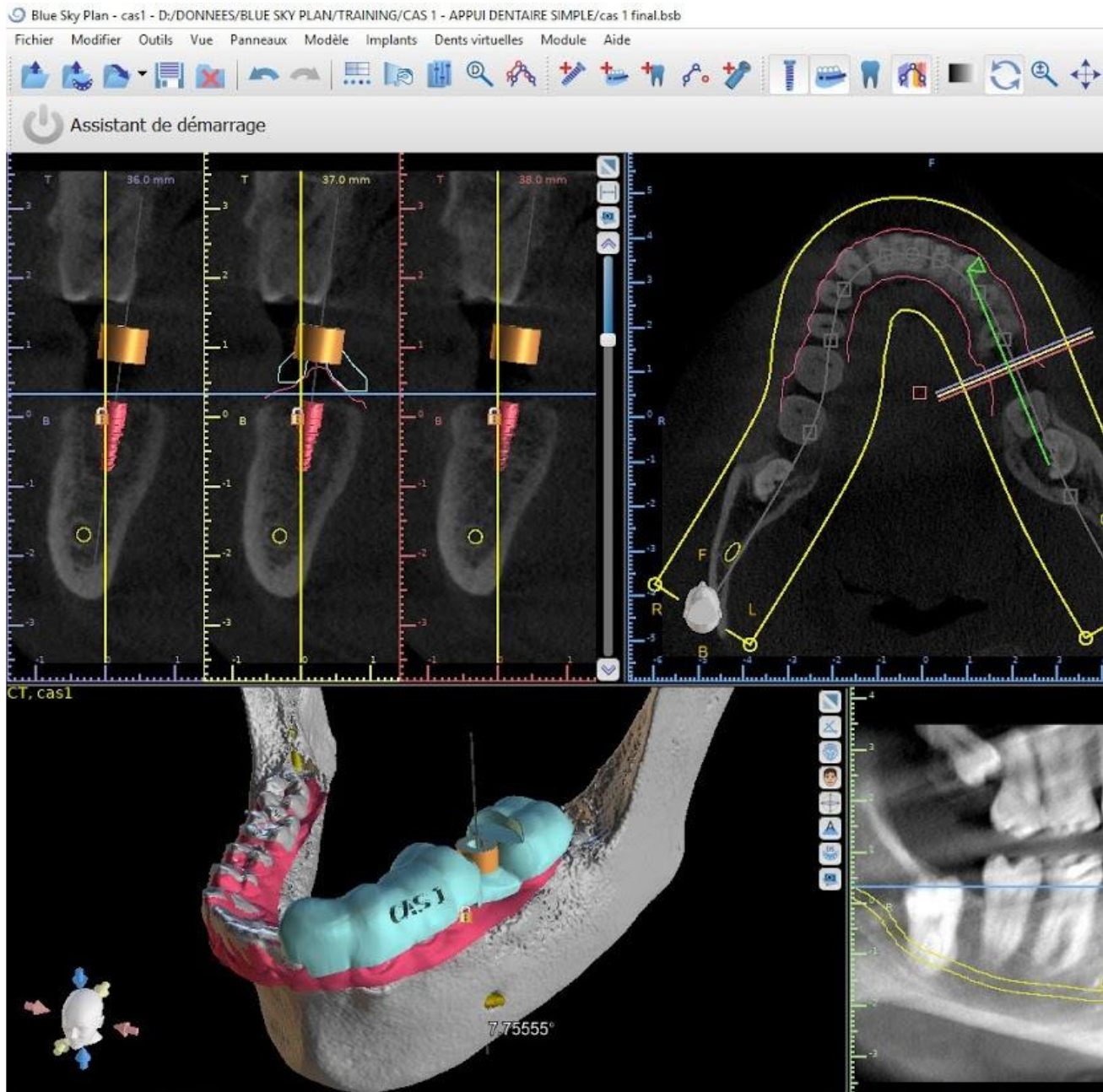
*An useful advice is to often access this webpage which contains the anatomy of the human body, with an easy to use, intuitive interface*

<https://www.zygotebody.com/#nav=1.66,150.81,16.87,0,0,0,0&sel=p;;h;;s;c:0;o:0&layers=0,1,9415>

Because in this volume we will treat only the structural domain of simulations, we will discern 3 main types of simulations regarding the involved bodies or parts:

- 1) when the human organs are simulated alone, without any additional external factors (for example when the human body performs certain movements, such as eversion or flexion, or when the heart pumps blood into the circulatory system or when the bolus is transported via the digestive system etc.)
- 2) when external devices help regain or optimize the functioning of certain organs (as in the case of prosthetics or orthotics, implants, stents, hearing aids, canes etc.); here the interaction becomes

We show here a screenshot from the Blue Sky Plan software, based on the existing x-rays.



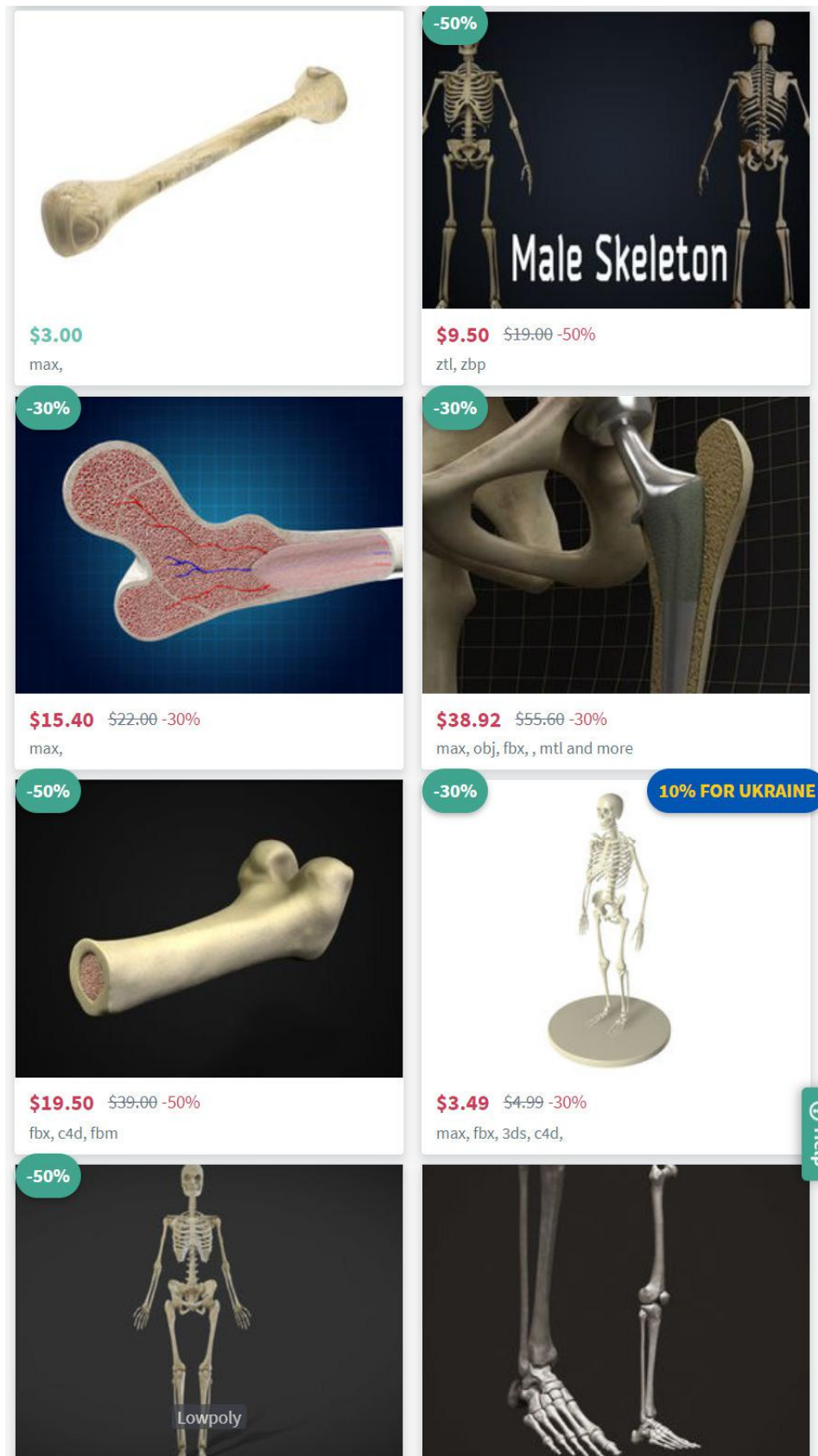
### **Artistic modeling**

This is the method that we often use and highly recommend, giving the best balance between the best shape fidelity and accuracy and time for pre-processing before going to simulation. Such 3D models can be found for free in a limited number on GrabCAD.com, or paid models in an unlimited number on websites like CGTrader.com, Turbosquid.com and many others.

For example, these are the results when we search for “femur” on GrabCAD.com, with around 3 pages, but with lots of other models, which are not all femurs, but they have in common that they might be related to the human skeleton. All these are available for free download after an account has been created.

simulations, the reality, in most of the times, proves to be different. After all, not much can be required from free things.

Below is the result after searching “femur” on CGTrader.com, which, if properly filtered, according to our needs and available finances, can lead to very satisfactory results.



The models are available as K (or KEY) files for LS-DYNA, and they contain the meshed model, without geometry. This might be a problem for people looking to alter the geometry, since it does not exist. There are experts who successfully converted the meshes into 3D models suitable for solid or shell meshing, but we know that a high level of experience is required to do so.

Types of contacts or interactions between parts

Everything in nature is connected and it has interactions at a very small or at a very large scale, even though not all human minds and actual instruments can or can't conceive it, yet. These interactions occur in the human body at the separation between its parts because they are either very well glued or bonded together (like the muscles are connected to the bones), or they are sliding (like the cartilages move inside joints) or they are far away (like the hand is in relation to the leg or the head).

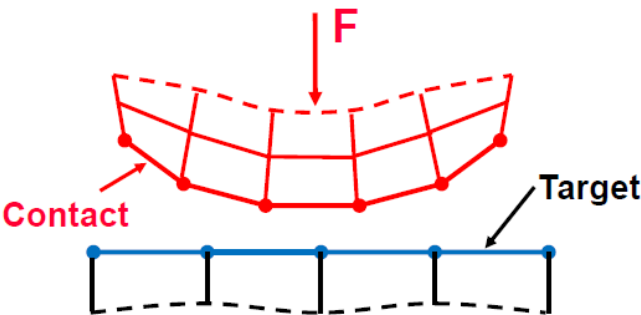
We show here a short categorization taken from Araújo, Kaique & Barros, Rodrigo & Neto, Joel. (2021). *Structural analysis of pile cap as onshore wind turbine foundation. Revista IBRACON de Estruturas e Materiais*. 14. 10.1590/s1983-41952021000500004.

Contact types	Behavior	
	Gap	Sliding
Bonded	No	No
No Separation	No	Yes, $\mu = 0$
Rough	Yes	No, $\mu = \infty$
Frictionless	Yes	Yes, $\mu = 0$
Frictional	Yes	Yes, $\mu > 0$

$\mu$ : coefficient of friction (dimensionless scalar value)

A contact has 2 sides, one (the source, or Contact in ANSYS) that moves towards the other (the Target in ANSYS). The separate boundaries find each other by automatically constructing gap elements between the outer nodes, which are spring-like bodies with a defined length (which give the gap or the distance between the bodies) and a stiffness k (if the force in that spring element is  $F = -k \cdot x$  then the stiffness is  $k = -F / x$ ).

Here is a schematic representation of a basic contact.




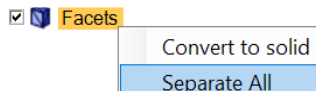


## Spaceclaim tutorial 1: Healing and conversion of a STL shell body into a solid

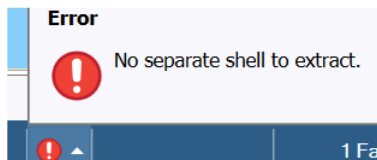
Having obtained a shell geometry as STL, we are going to verify, then heal it if problems are found, then convert it in fully solid body, used for meshing inside the ANSYS environment.

Open Spaceclaim. Drag and drop *Spaceclaim Tooth 1.STL* into the Spaceclaim window.

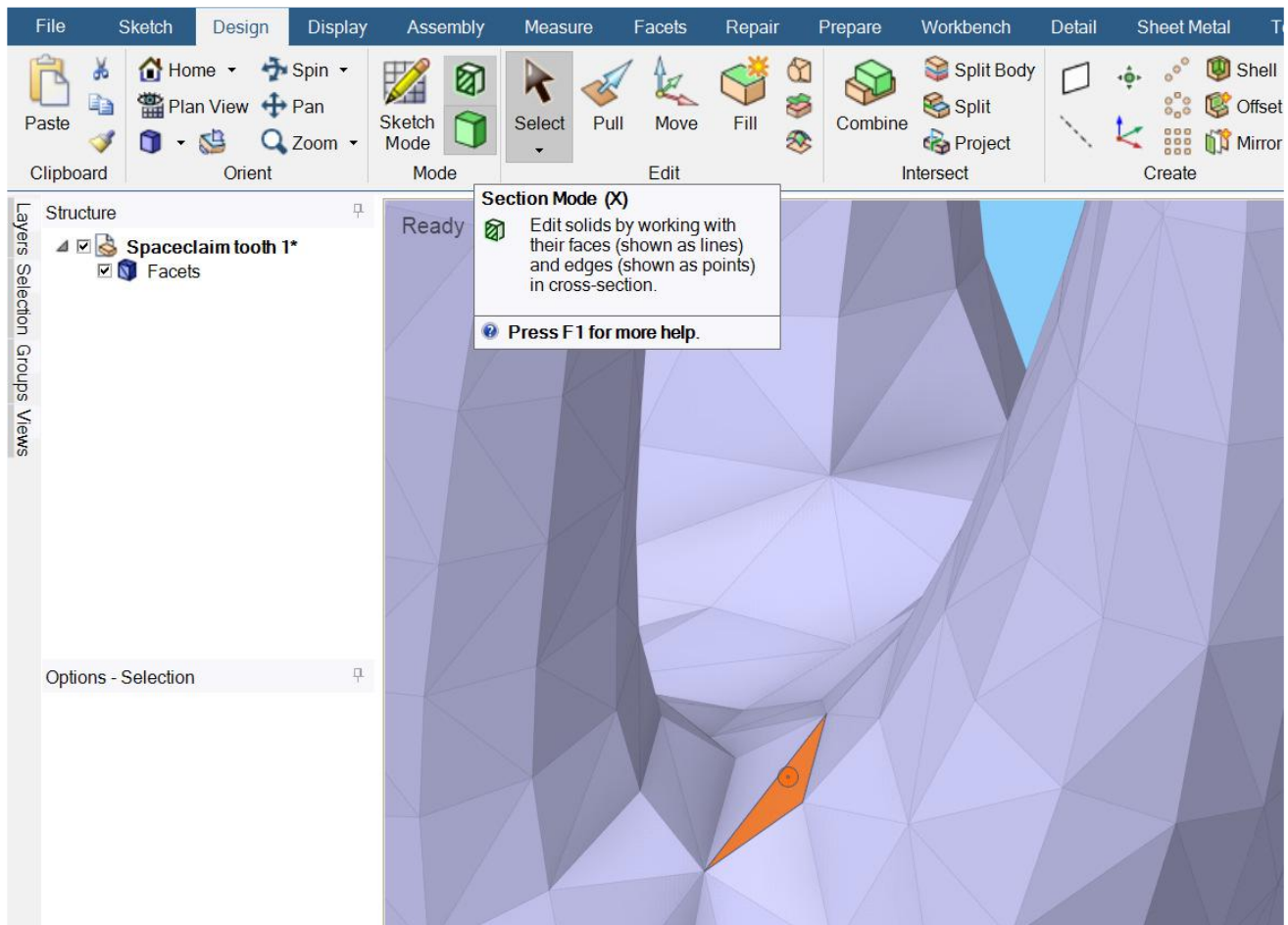
In the  beginning we'll check if the file contains 1 or more bodies, this making the difference between a single part and an assembly, and also because some stray bits and pieces might be interpreted as parts of an assembly, but if they are too small and neglectable, they surely need to be deleted. Right click Facets, Separate All (don't jump to converting to solids at this moment, have patience ;).



On the bottom bar, this message will confirm that no other parts are present, which is what we needed.

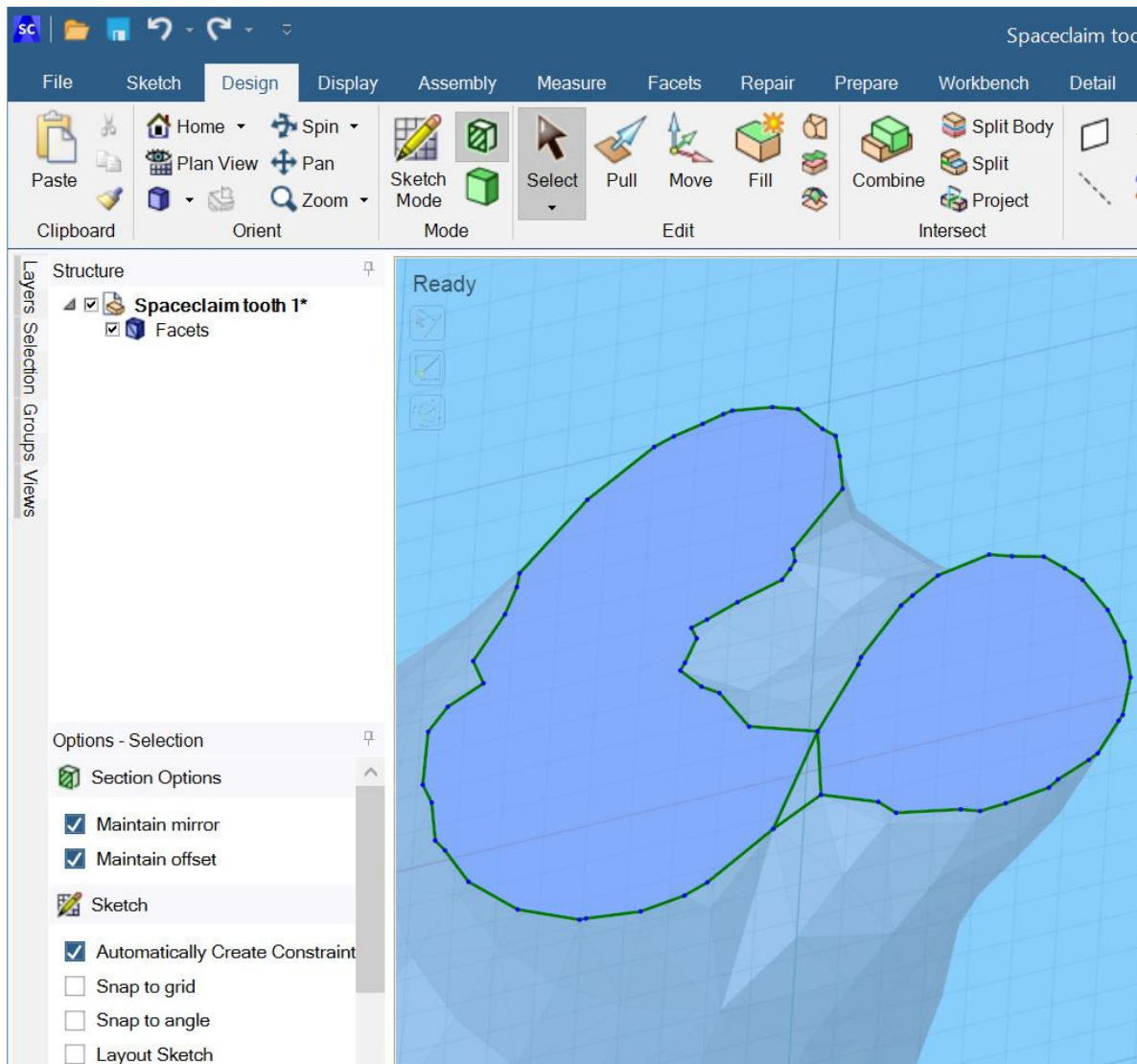


To see that the STL geometry is a shell part (empty inside), let us create a section view. Select a triangle (or facet) from the body which will be the reference for the section plane. Go to Design menu, then press the Section Mode button.



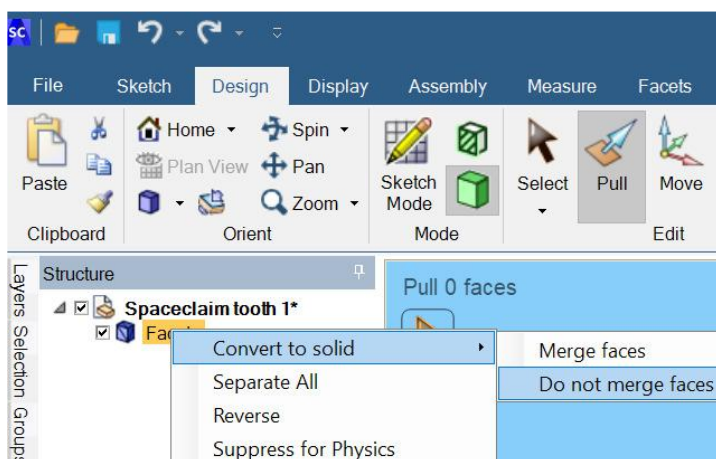


Because there is no hatched region, this means that the geometry is empty.

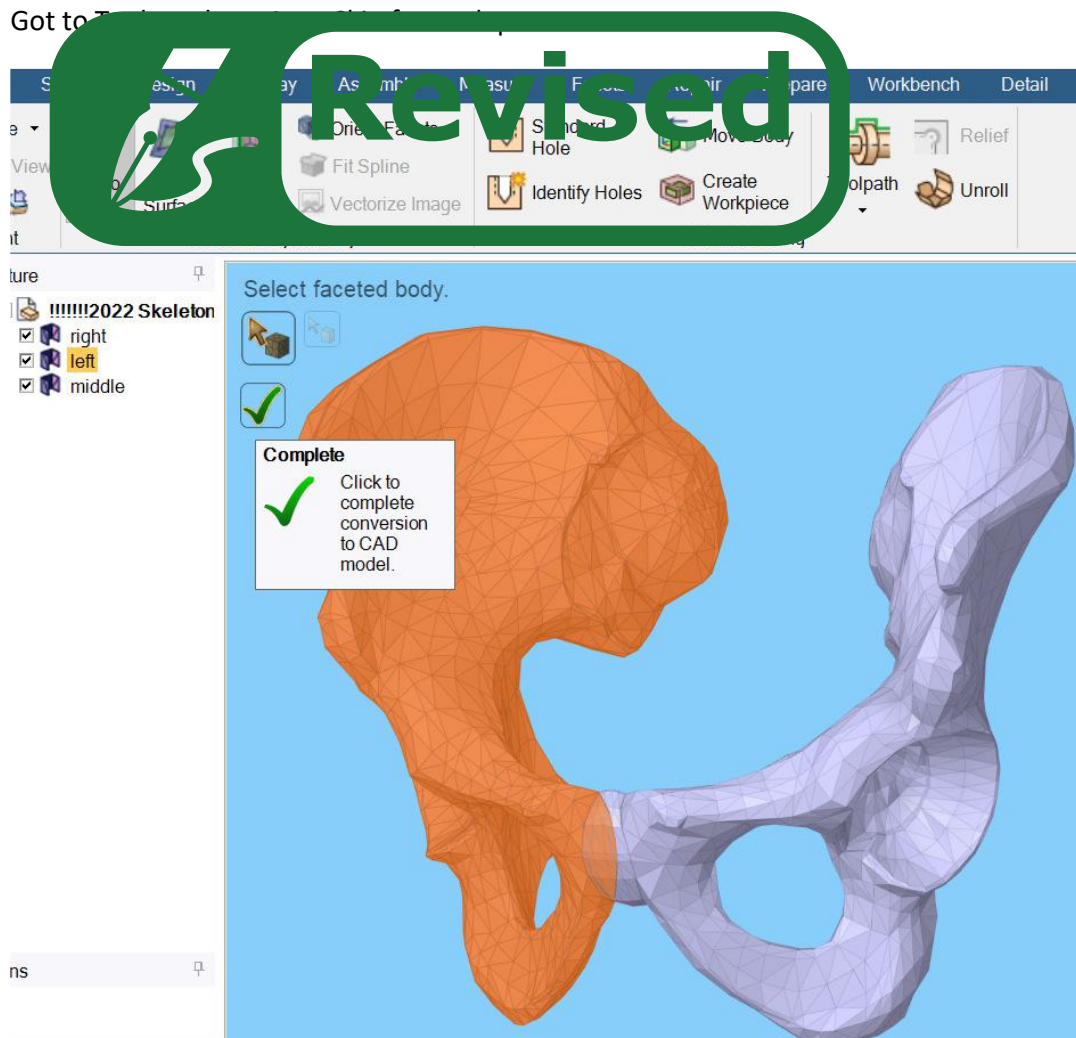
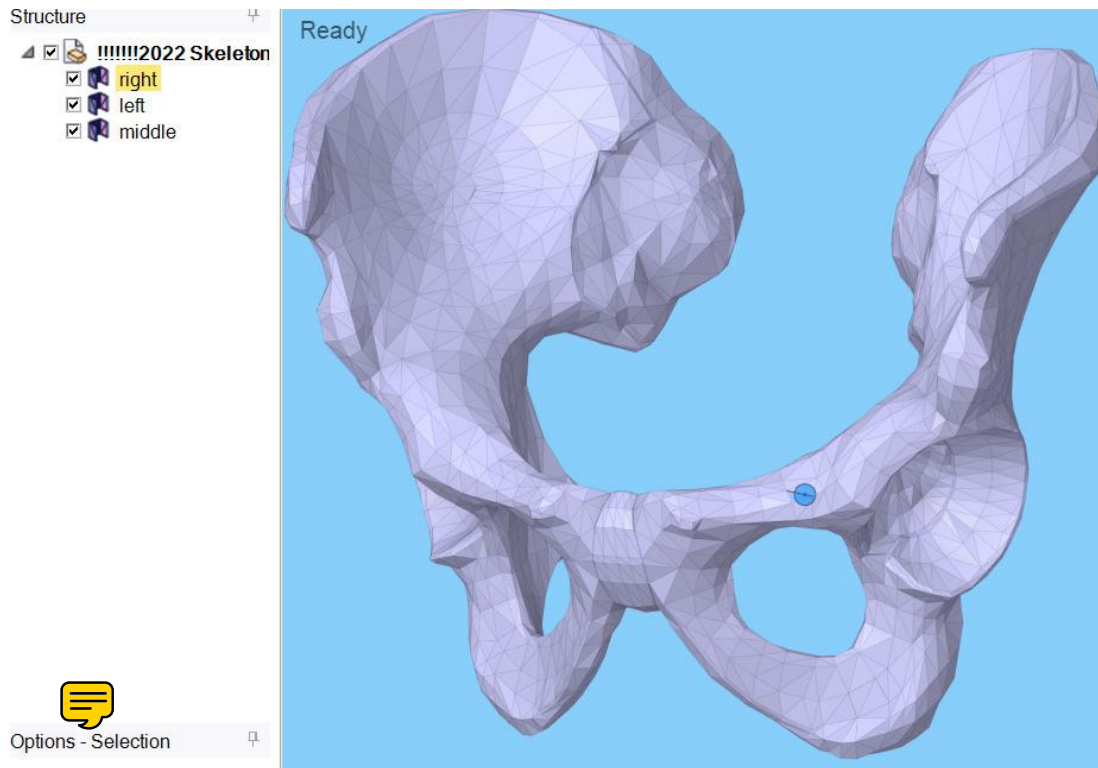


The easiest way to obtain a solid part is to convert it from the very beginning, with the disadvantage that the facets will still remain and will further dictate the meshing.

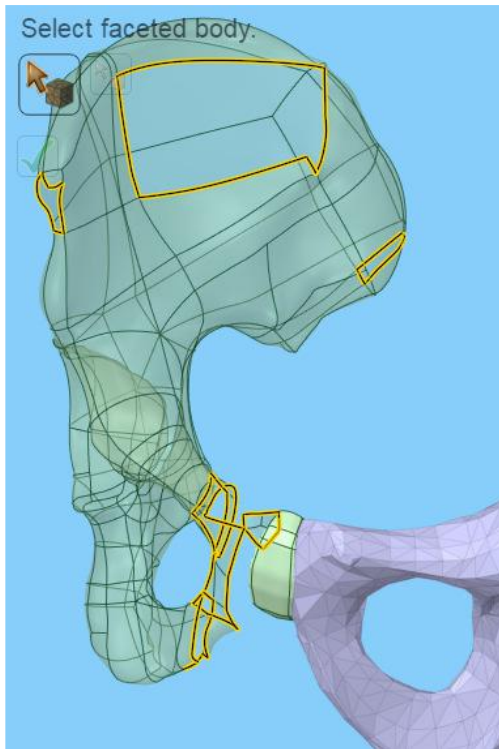
Right click the imported STL named Facets in the tree, Convert to solid, Do not merge faces.



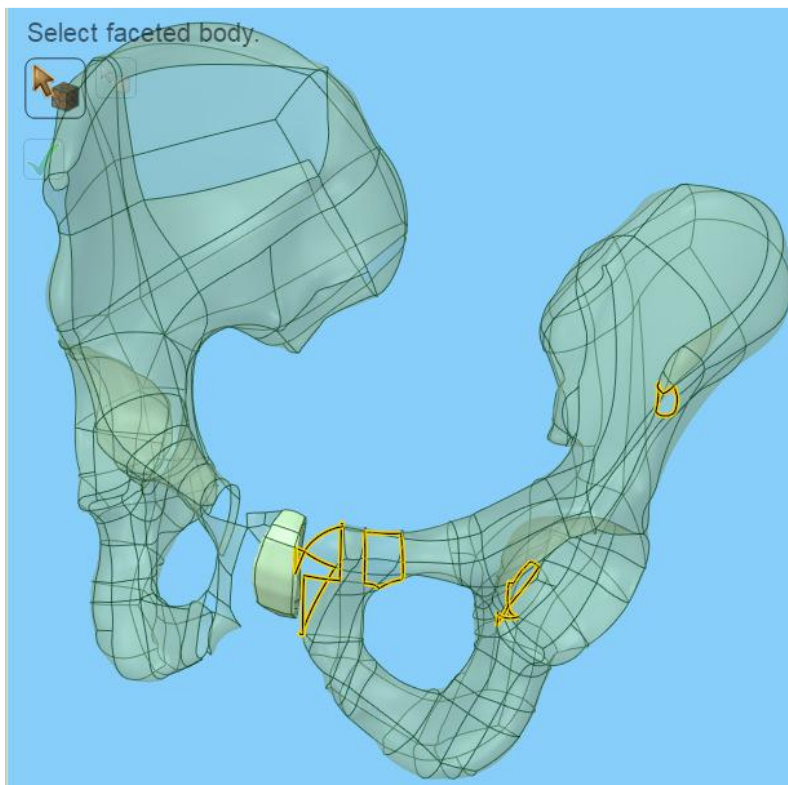
Right click Show All will unhide the important bodies.



Go and Auto Skin the left part and you will get this unacceptable geometry.

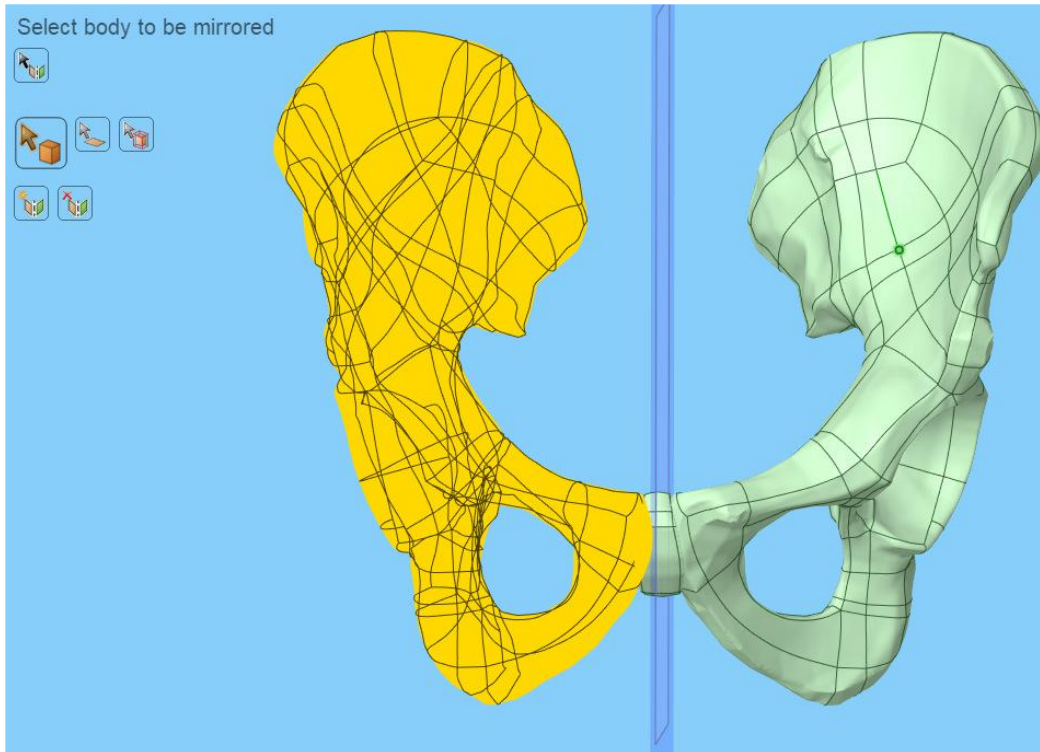


Go and Auto Skin the right part and you will get this unacceptable geometry, but better looking than the left part, with less errors.

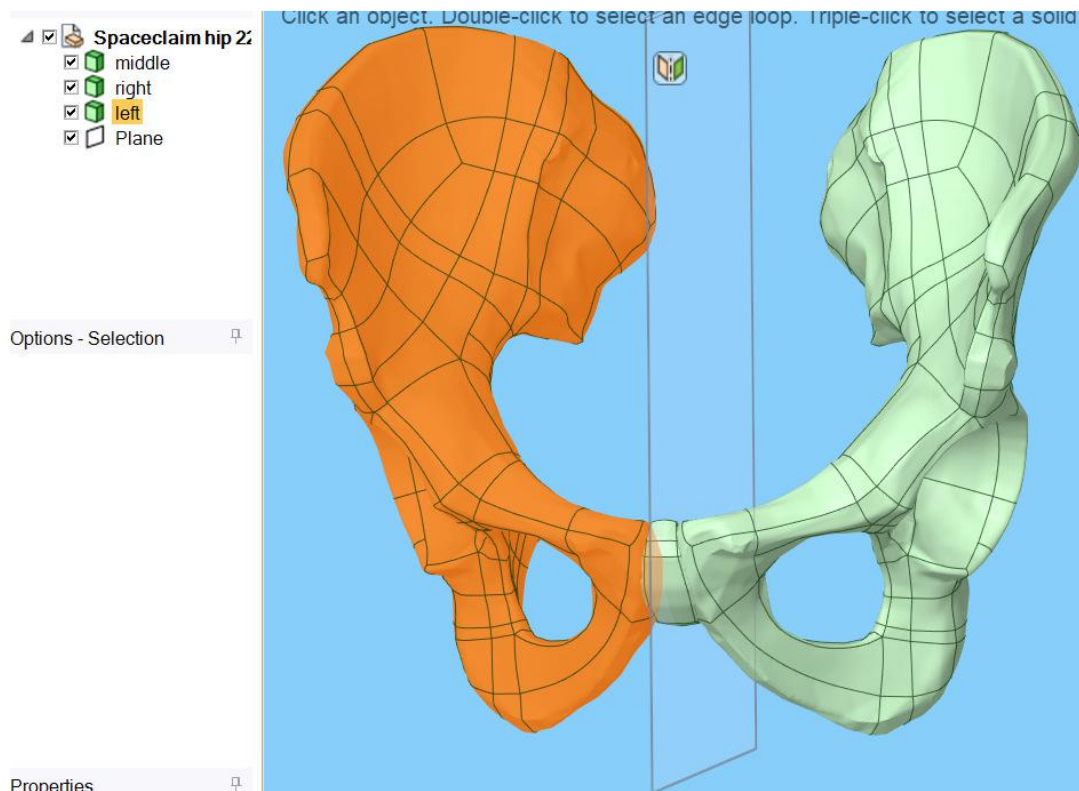




Go to Design, Mirror and with the plane selected as reference, the mirrored hip bone will appear on the left side.

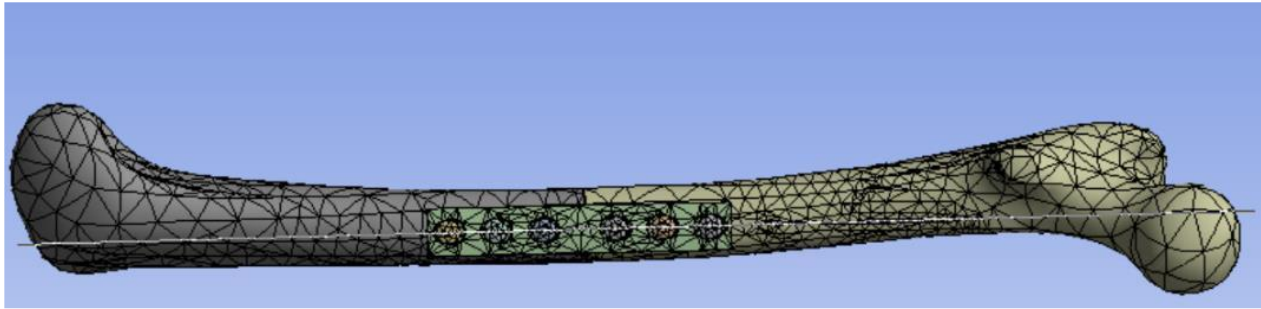


Rename the parts to look as here.

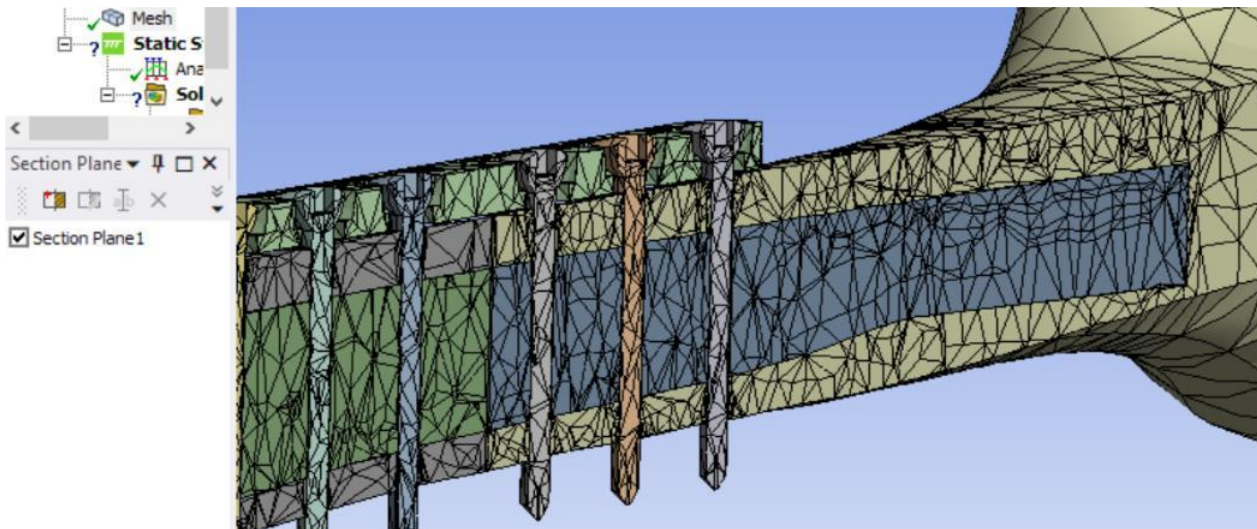


Before assuming that everything is correctly made - because we know that the plane was positioned by hand, without highest accuracy – ALWAYS check for any collisions in the assy!

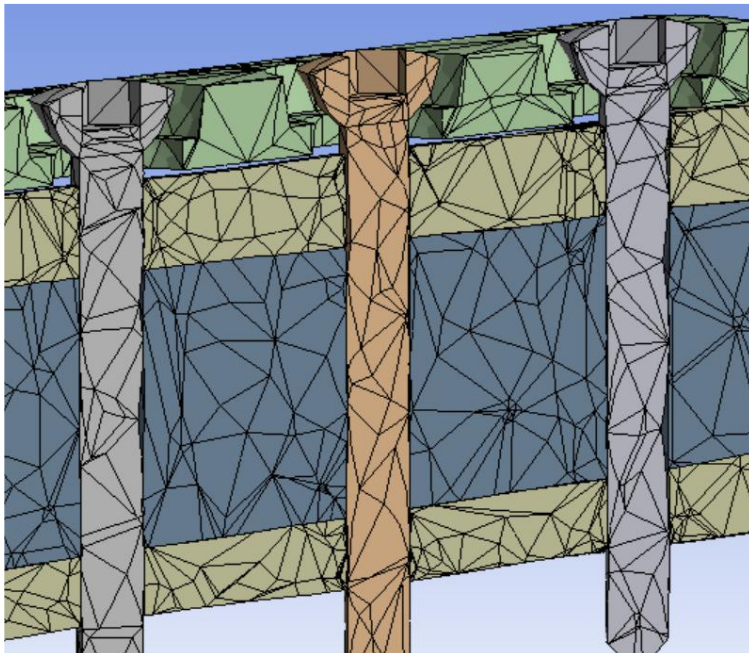
Draw a straight line that will encompass the region that you want to see in section view, something like what is shown here along the bone length.



This is the resulting view.

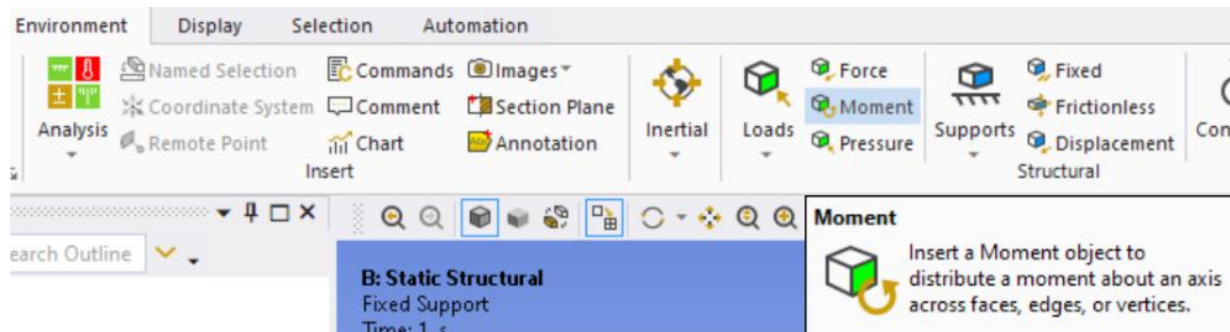


When we rotate with middle mouse then zoom in via scrolling the mouse wheel, we can see a decent mesh, which we recommend that you refine in the later stages, when you need more reliable results.

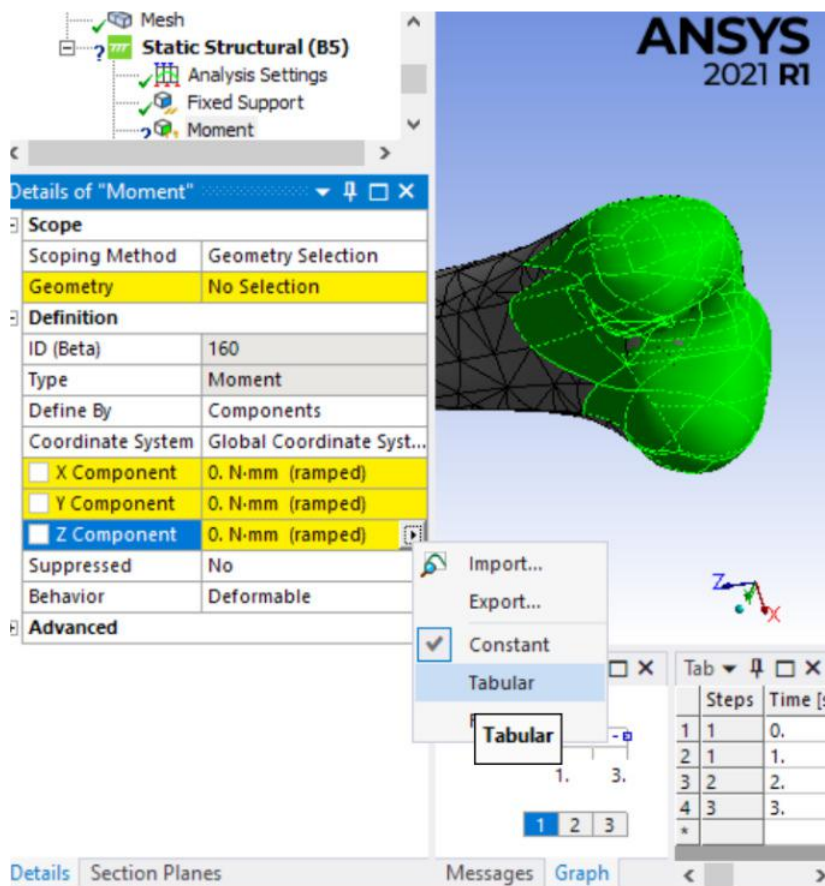




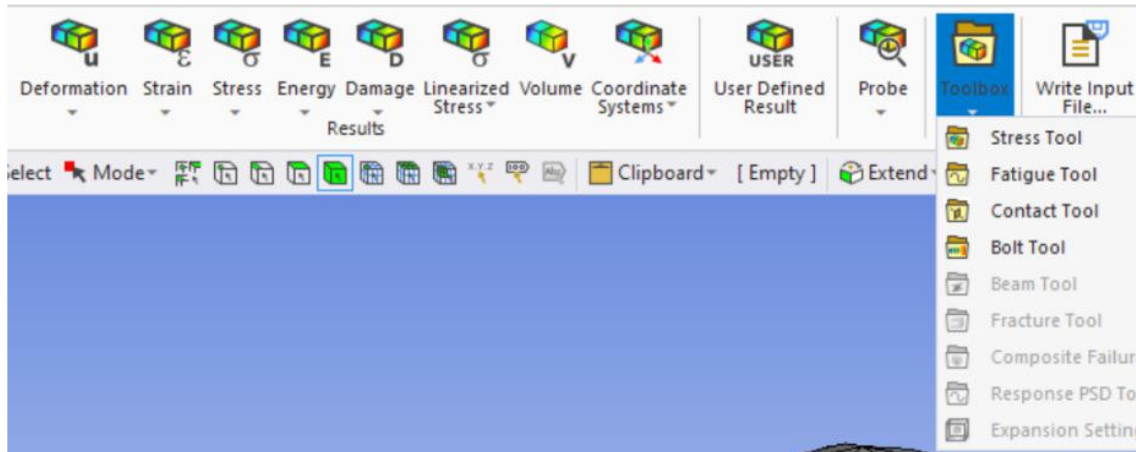
From the same Environment tab or from right clicking in the main window, insert a Moment.



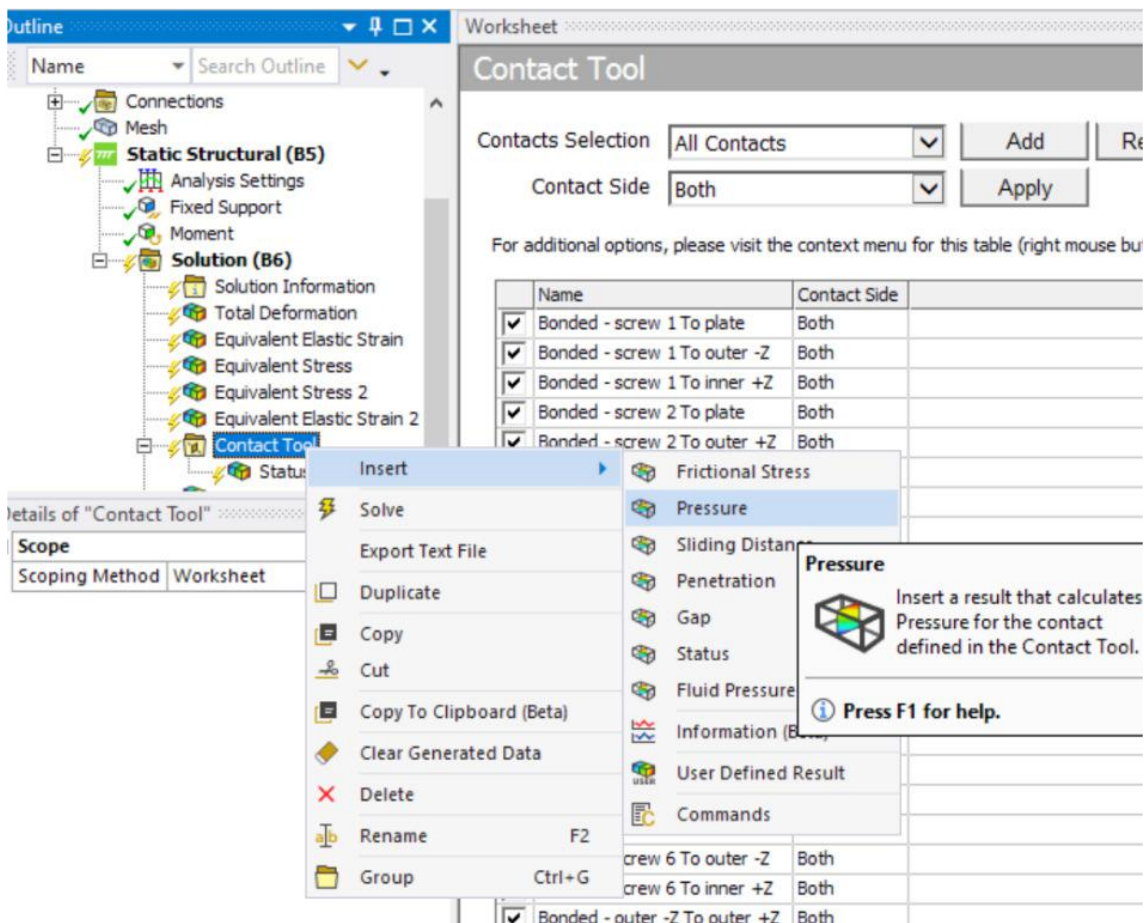
Select the opposite side, where the knee joint exists, Apply. Change Define By from Vector to Components. for the Z Component change from 0 or Constant to Tabular, since we have not 1 but 3 time steps, so 3 rows are needed in the Steps-Time-X-Y-Z table.



To evaluate what occurs in the contacts, from the Solution tab, click the Toolbox drop down menu then select Contact Tool.

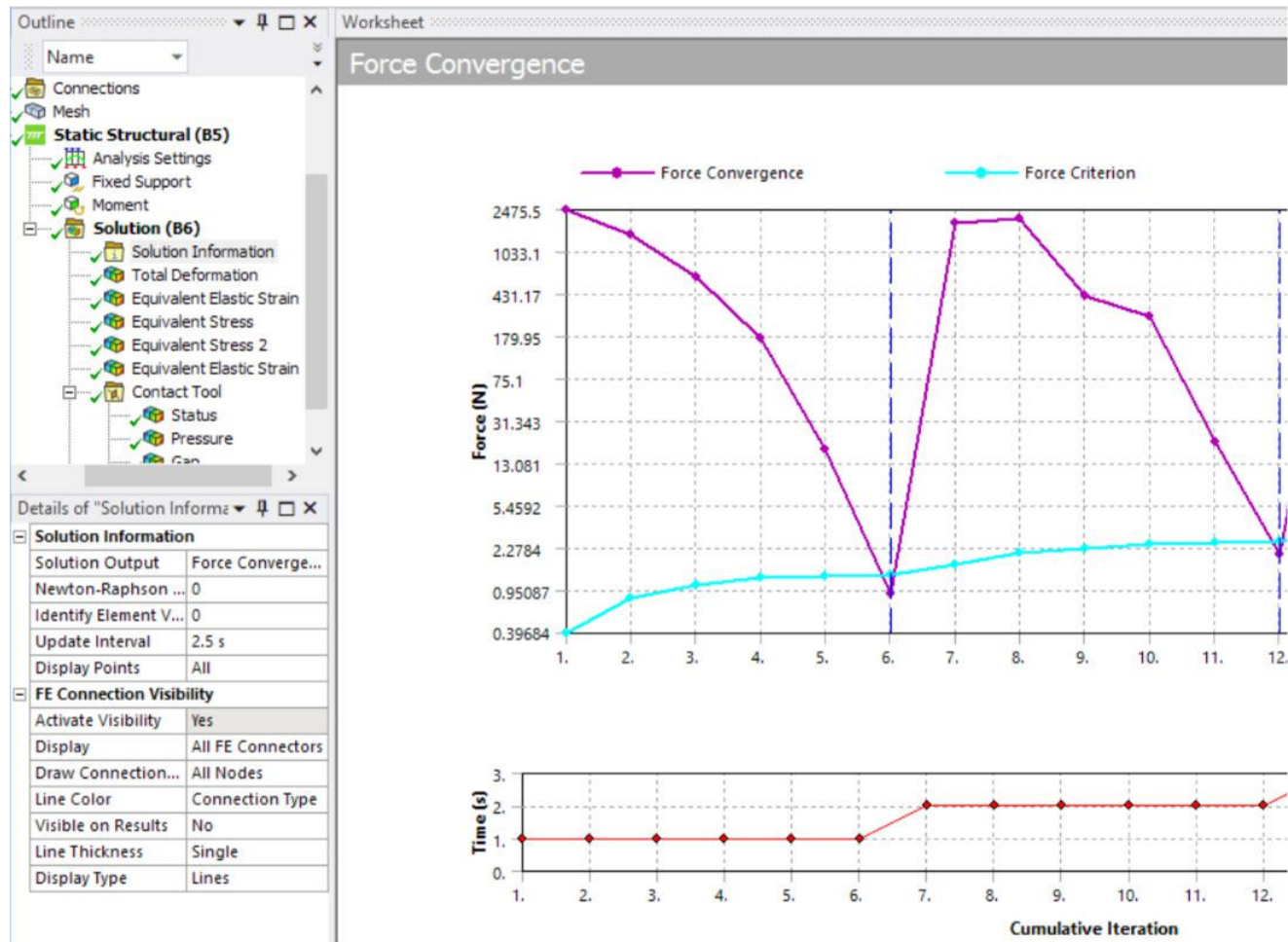


Click Contact Tool from the model tree in the left, the contacts list appears, right click and insert a Pressure and a Gap item.

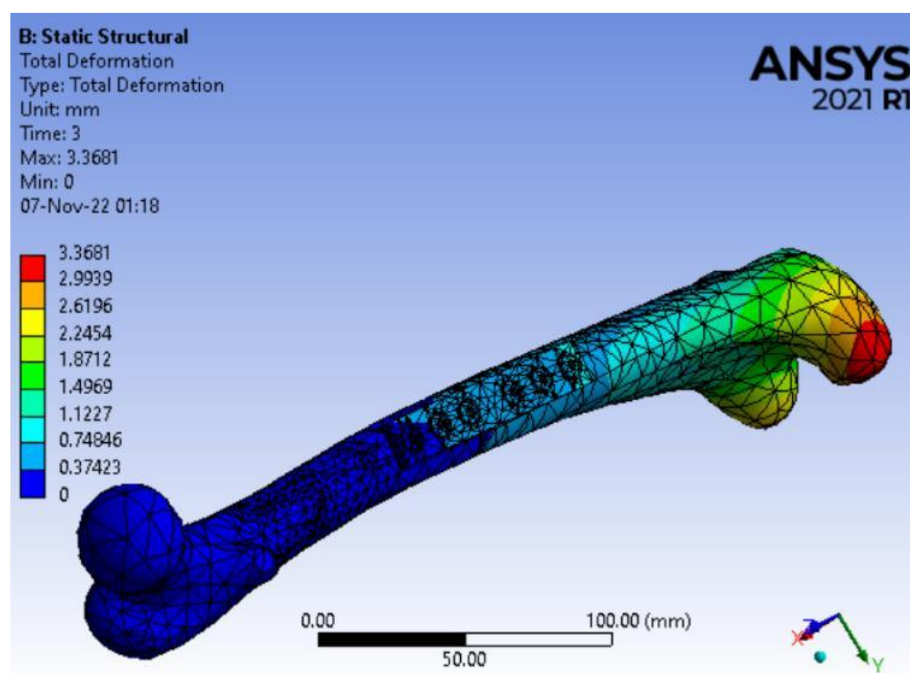


Save. Before solving, make sure that you assign (almost) all physical cores to the Solve tab, near the Solve button.

During or after solving, change Solution Output from Solver Output to Force Convergence to see such useful convergence plots.

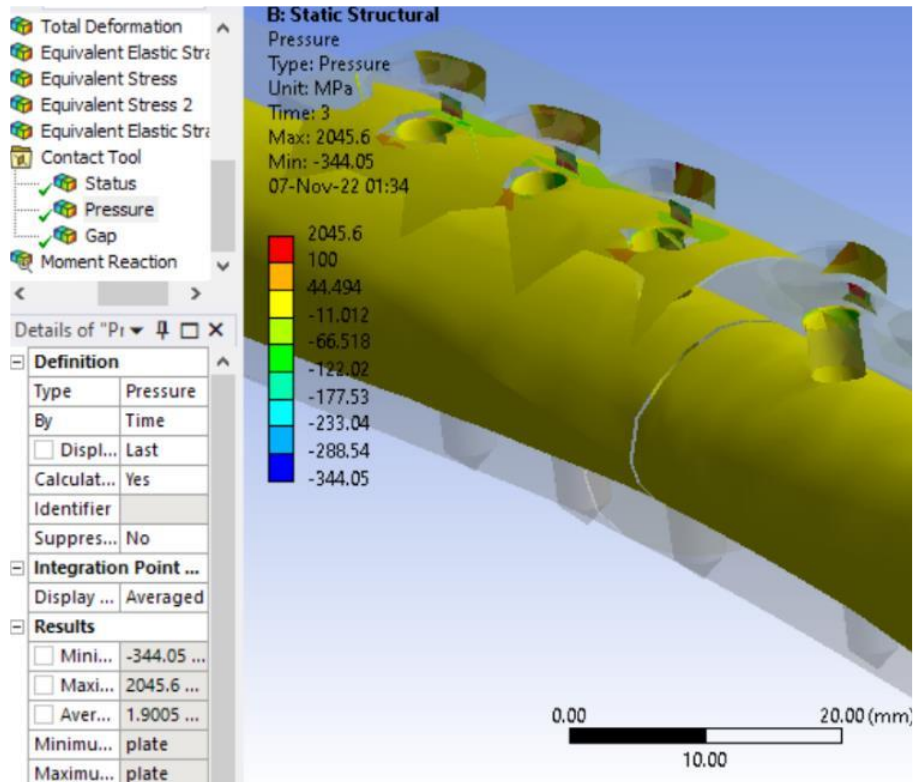


Here is the Total Deformation plot in which we make sure that we do not have exaggerated values (e.g.: a few meters or kilometers), for a bone of 0.4 meters in length, under twisting load.

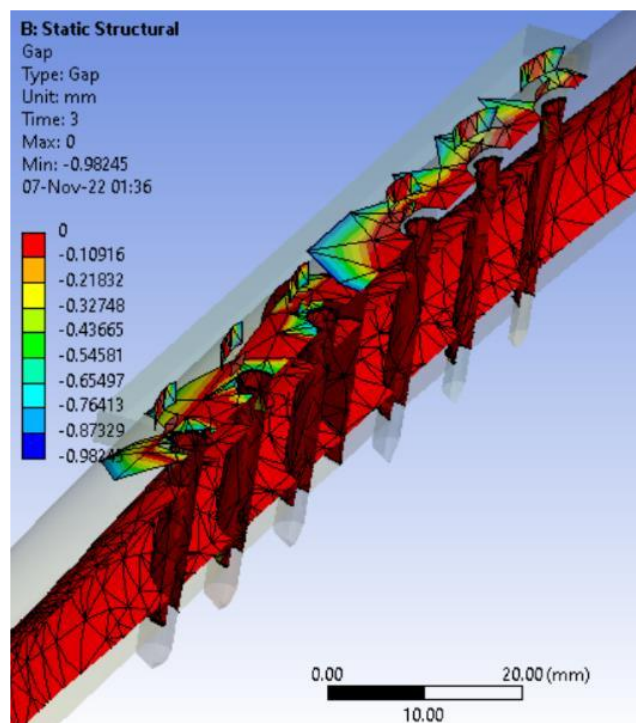


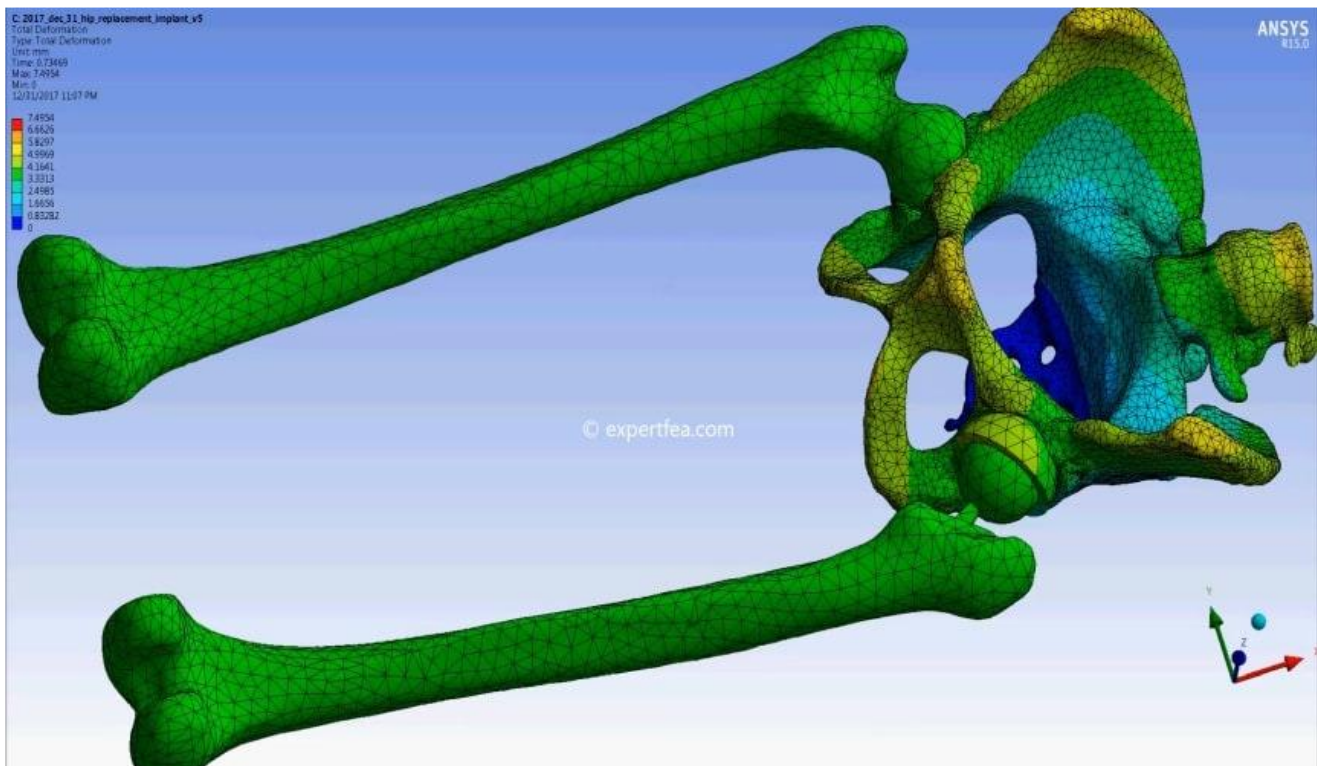


Here is the Pressure in contacts, with a maximum on plate, where it is constrained by the screws heads. We filtered the values with 100 MPa for a clearer view of the colors.



Here is the Gap plot in contacts, with the highest and lowest values on plate.



**CASE 140: ANSYS WB FINITE ELEMENT ANALYSIS - Total hip replacement implant simulation**

Once in Workbench, double click or drag and drop a Static Structural scenario from the simulations list in the left.

Engineering Data (Materials): Go to General Materials and Add the Titanium Alloy by clicking the yellow + sign from column B.

4	General Materials	
---	-------------------	--

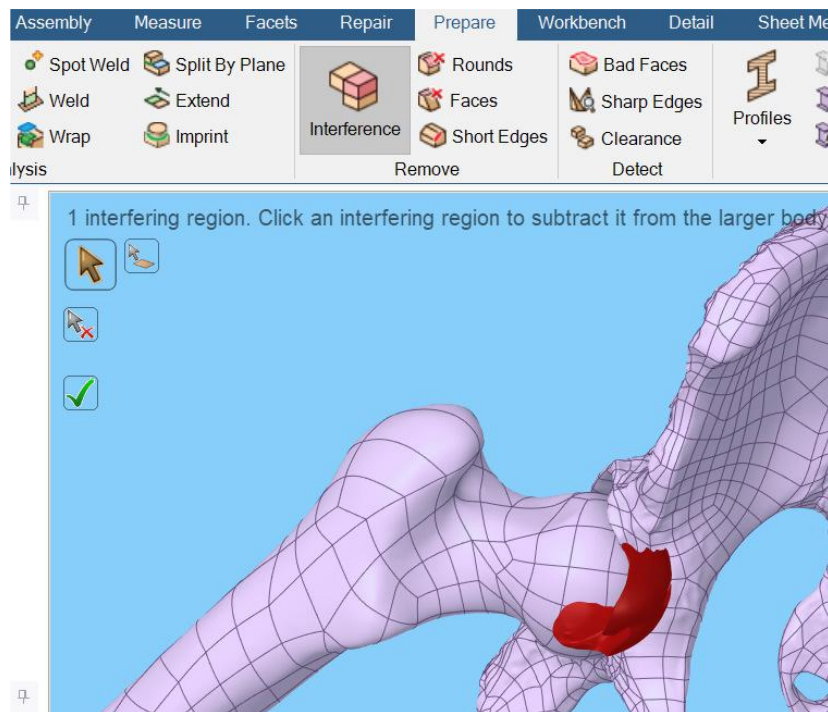
Outline of General Materials		
	A	B C
1	Contents of General Materials	Add
13	Structural Steel	+
14	Titanium Alloy	+

To easily create the materials for the bone parts, right click the Titanium Alloy, Duplicate.

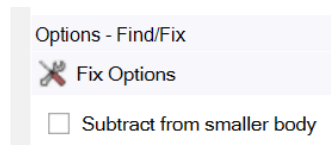
Outline of Schematic B2: Engineering Data		
	A	B C
1	Contents of Engineering Data	
2	Material	
3	Cortical bone	
4	Titanium Alloy	
5	Trabecular bone	



Now repeat the Interference command and you receive such a warning.

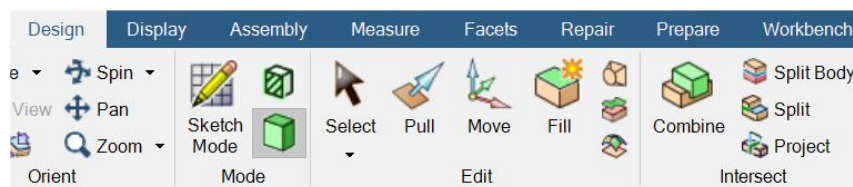


By default, if we press the green check button, it will subtract the smaller body from the larger one, leaving a carve-out or dent in it, with the shape of the red body shown here. Or you can subtract the larger body from the smaller one, leaving a dent on it with the same shape.

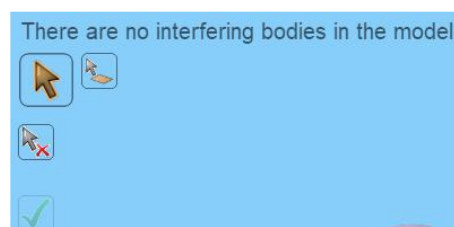


But at this moment we do not know what the software perceives as being large or small: is it because of the difference in volume or because of the total area? You can Undo the command and check the Subtract from smaller body option, if you needed to carve out that one.

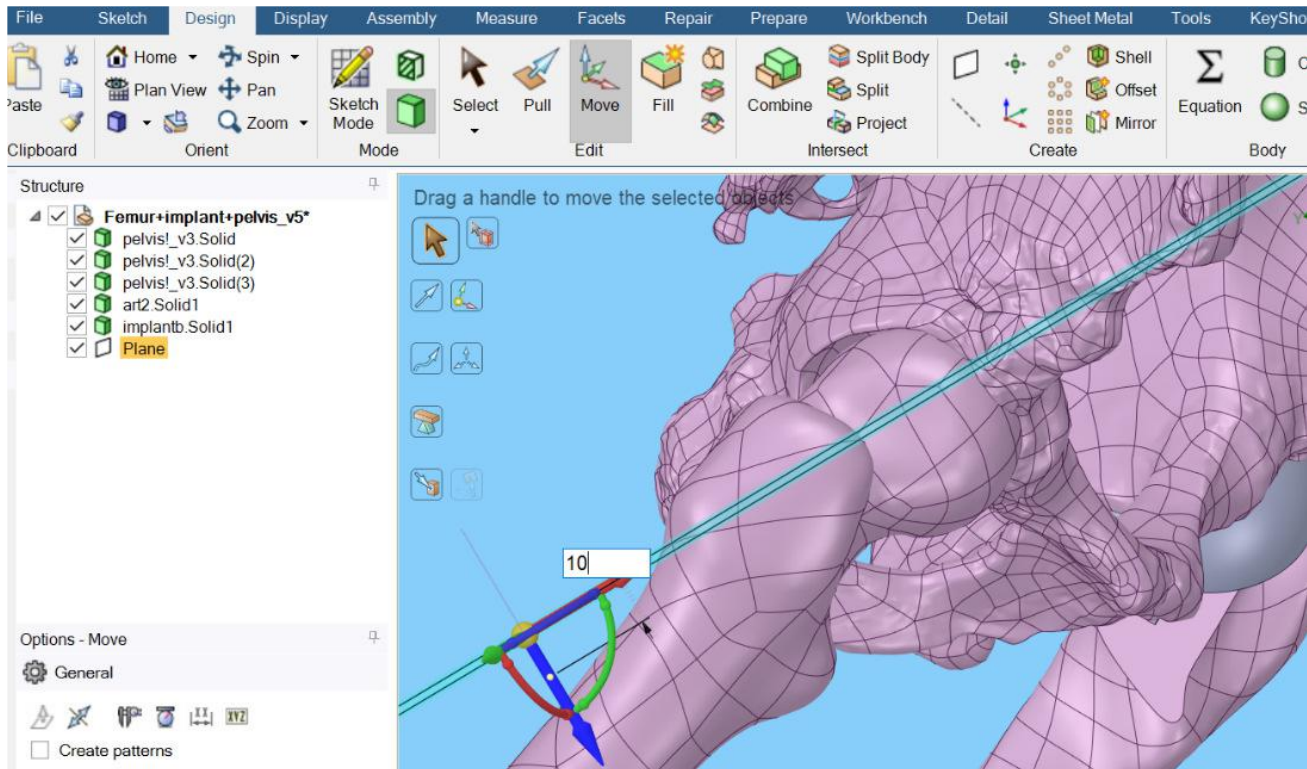
People sometimes use the Combine command from the Design, Intersect tabs, keeping in mind that extra options appear, which can be more useful.



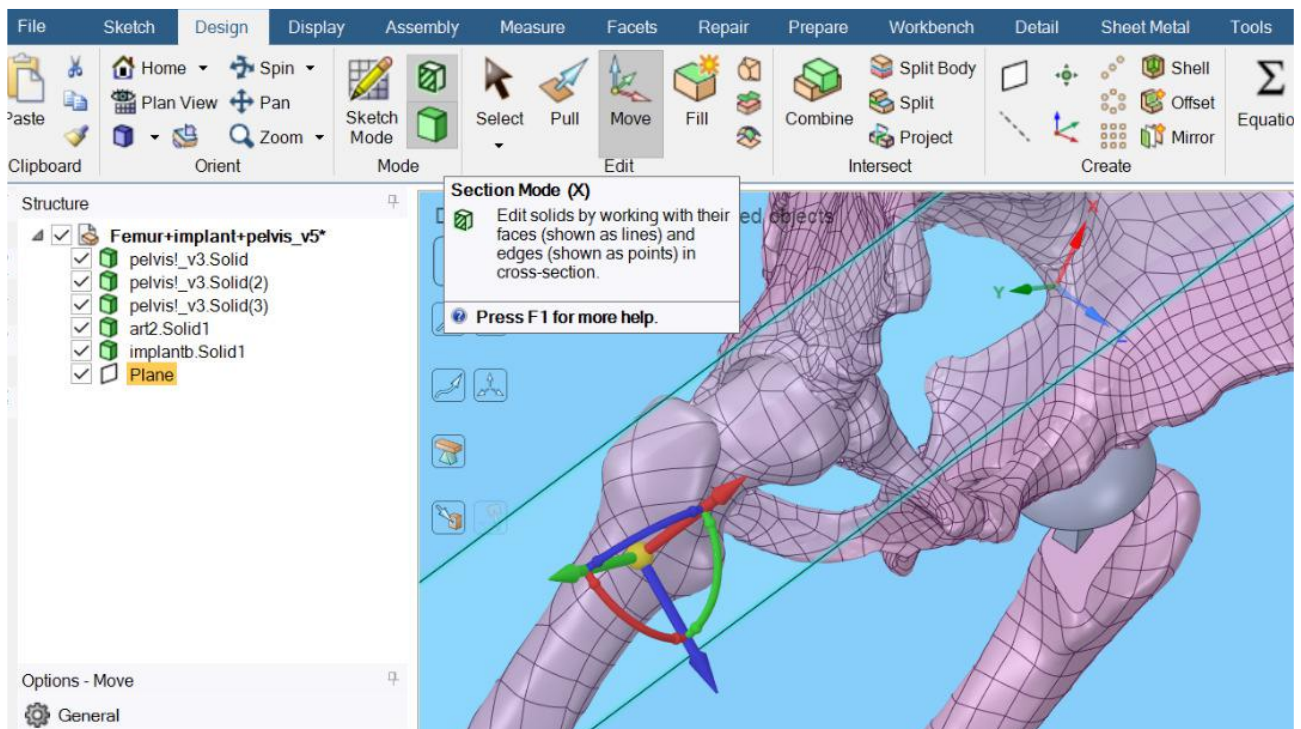
Click the Complete green check and you get this happy message.



Selecting the plane will highlight it in blue; go to Design tab, Move, then drag the blue Z axis 10 mm towards the back, Enter.

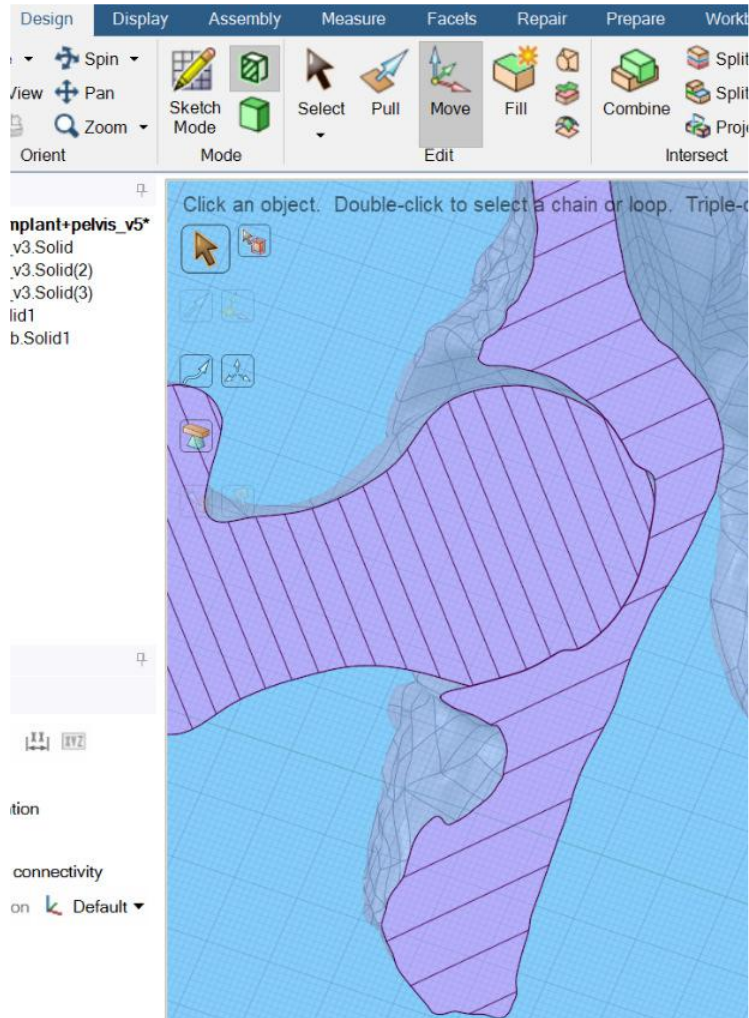


With the plane selected, click Design tab, Section Mode button.

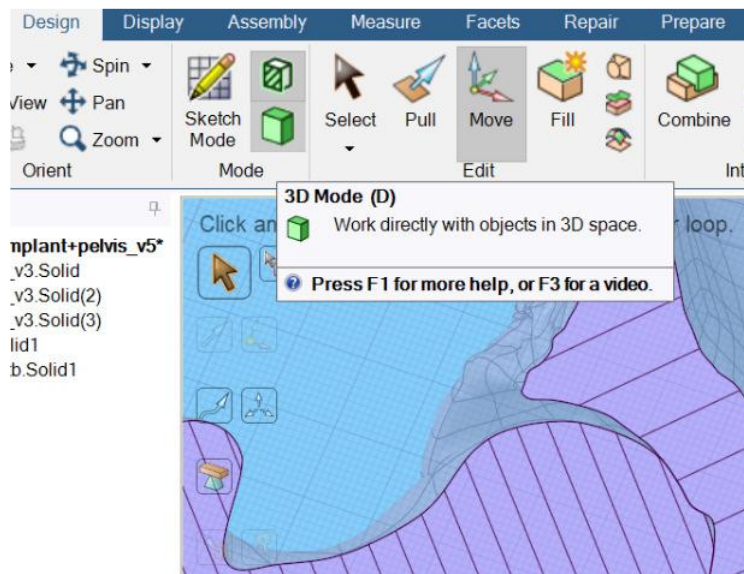




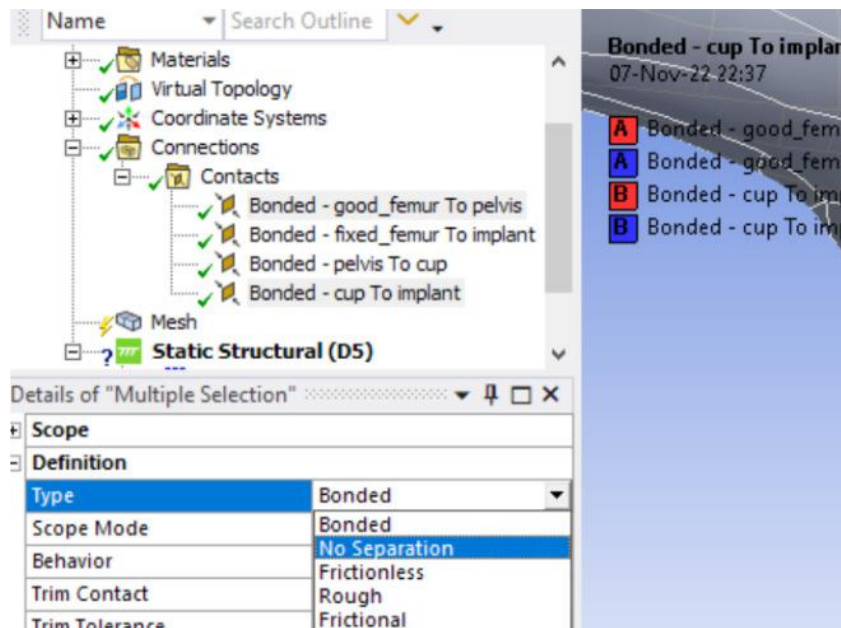
From this view we can say that the femoral head is intact because it maintained its spherical shape, while the pelvis bone seems thinned-out.



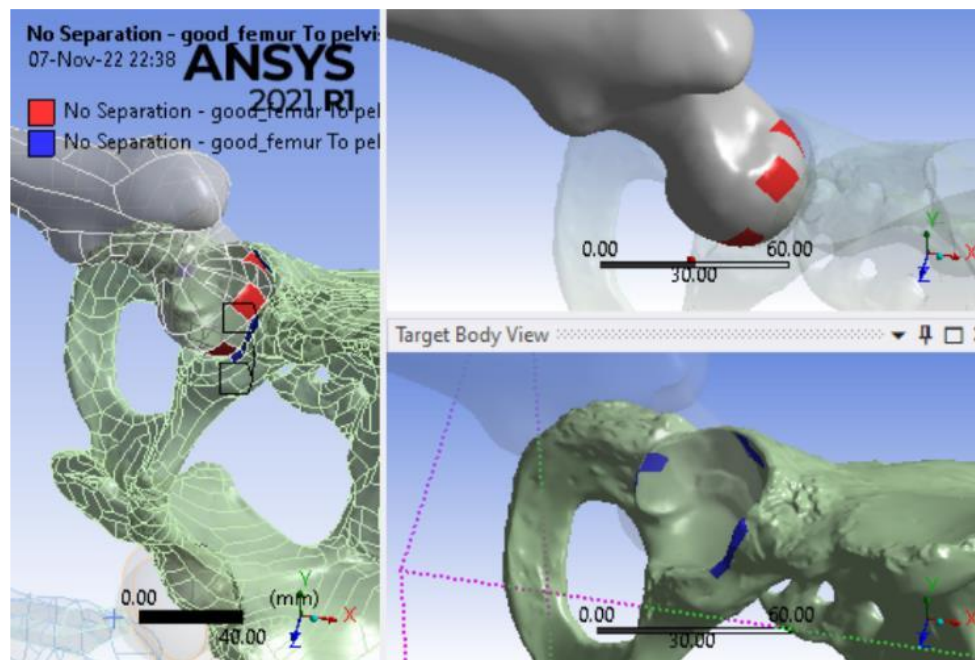
If we want to be 100% sure, we can compare their volumes, before and after the Interference or Combine operation was performed. Click the 3D Mode cube button to exit Section Mode.



Make the 1<sup>st</sup> and last Definition, Type as No Separation.

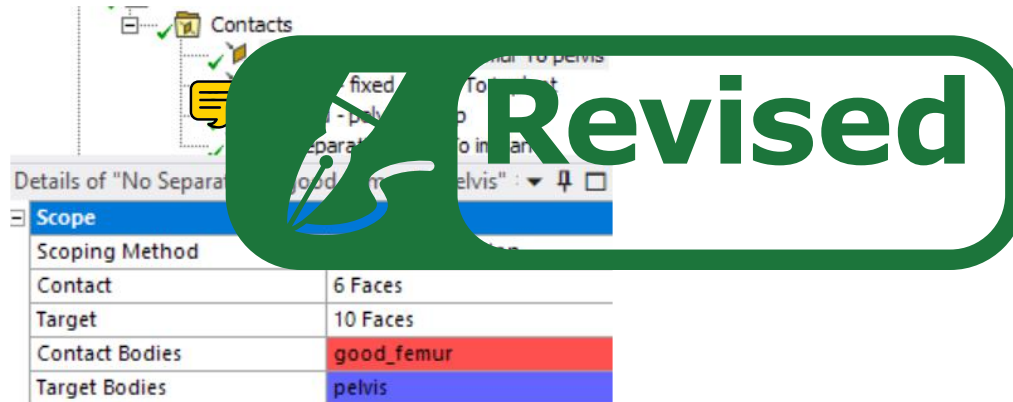


When we click the 1<sup>st</sup> contact to evaluate it, we can see that the parts barely touch each other.

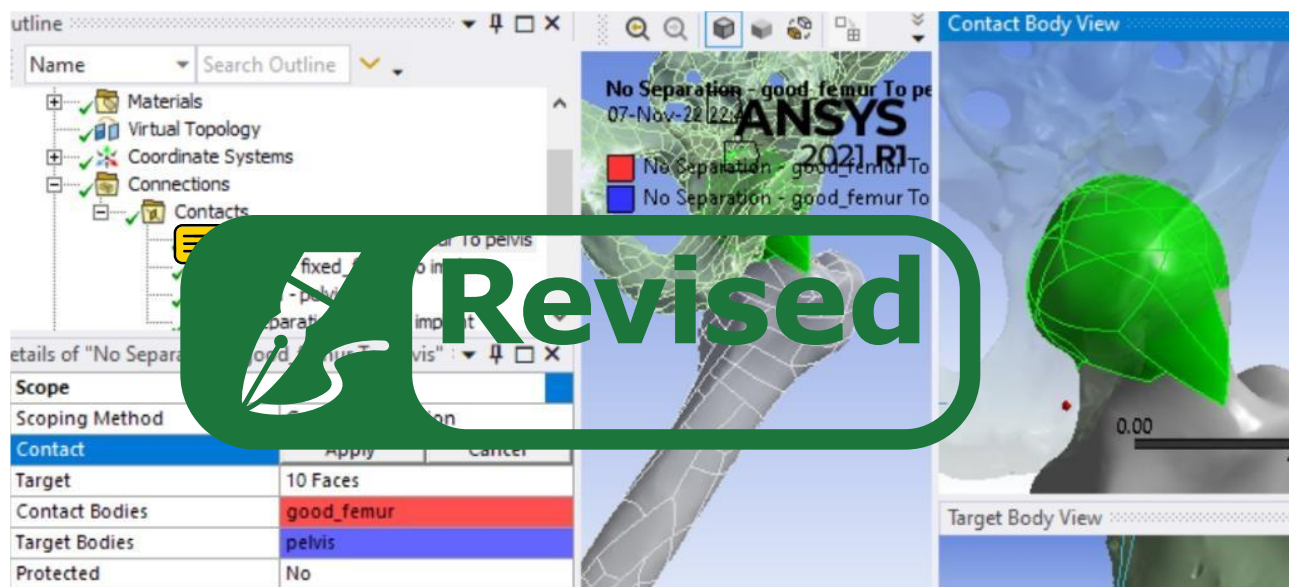


We need to propagate the selection to encompass more faces.

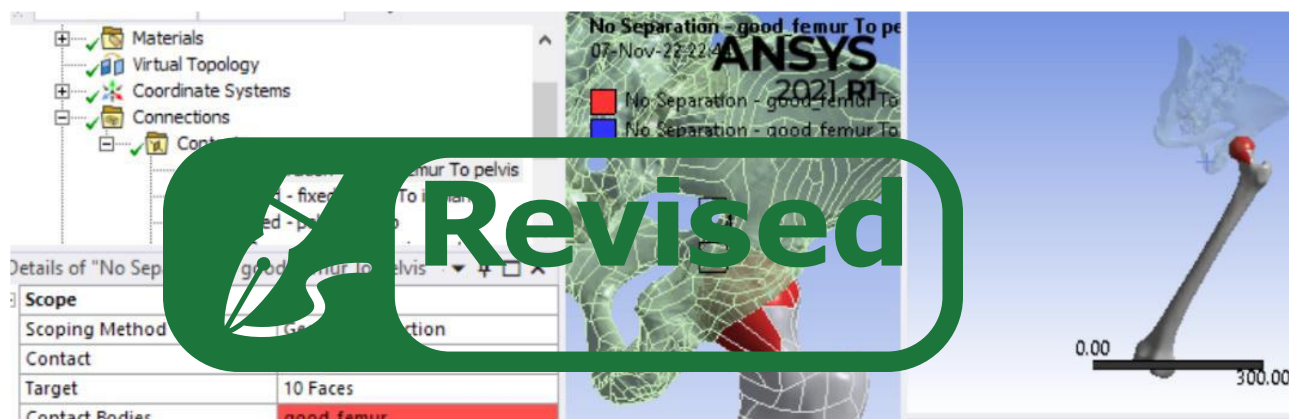
In FEA, any contact has 2 sides: a source and a target; in ANSYS, the source is called Contact, and the Target remained. Click Contact, 6 Faces, which is the source side colored in red.



We will select on the Contact Body View window additional faces with Ctrl pressed, Apply.

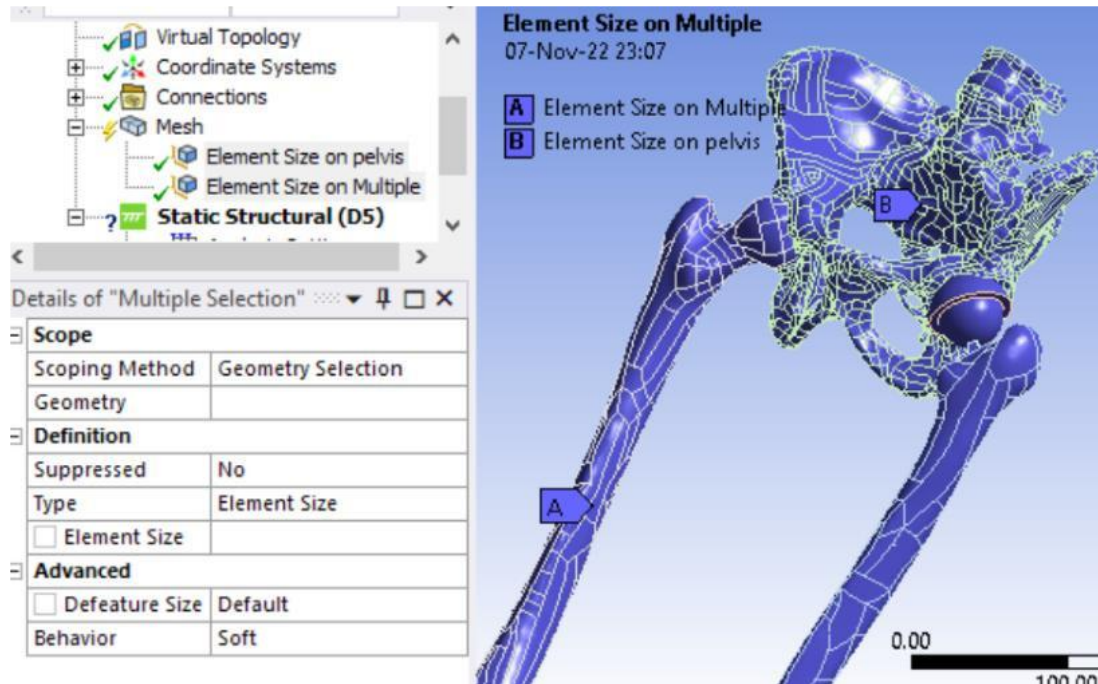


Instead of the previous 6 faces, we now have 21 (your number can be different, precision is not needed here, we just want to get more of the head in contact with the pelvis).

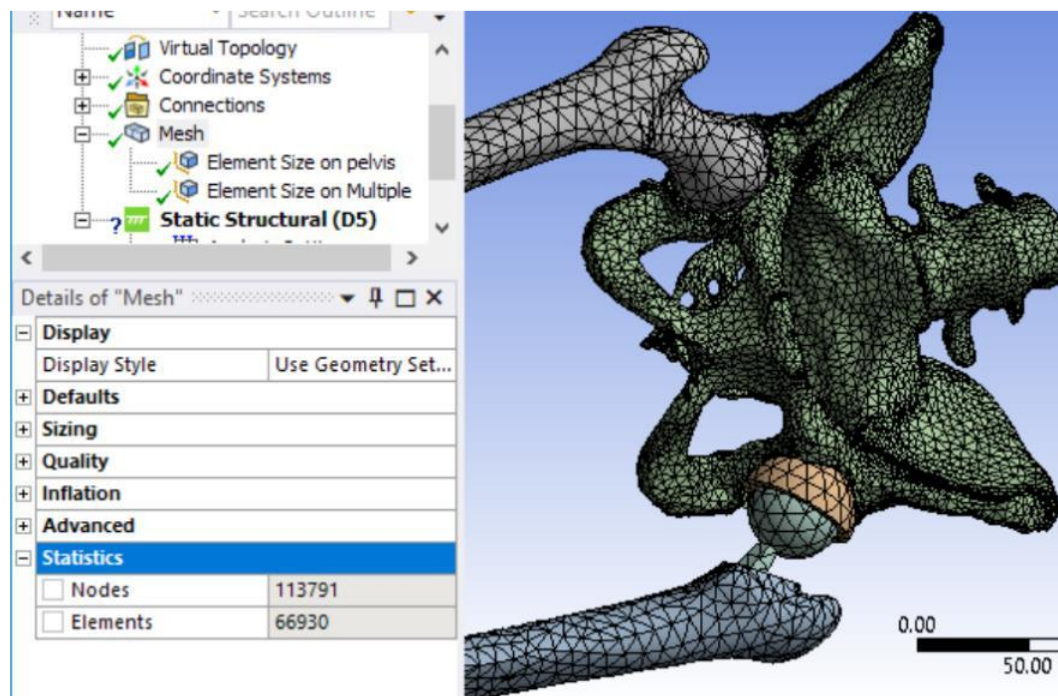




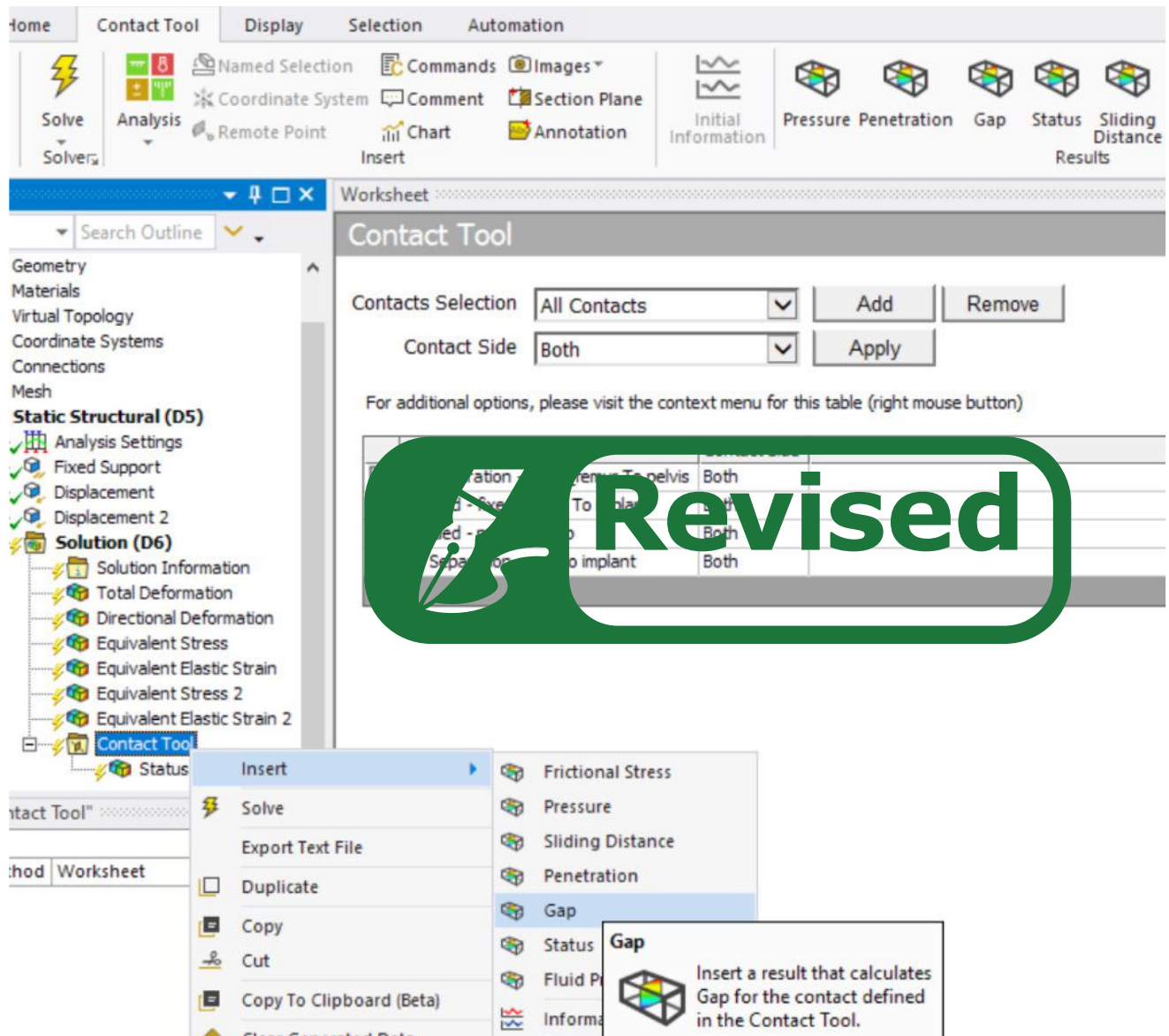
Right click each Body Sizing from the tree, Rename Based on Definition. Selected both, this should be the result.



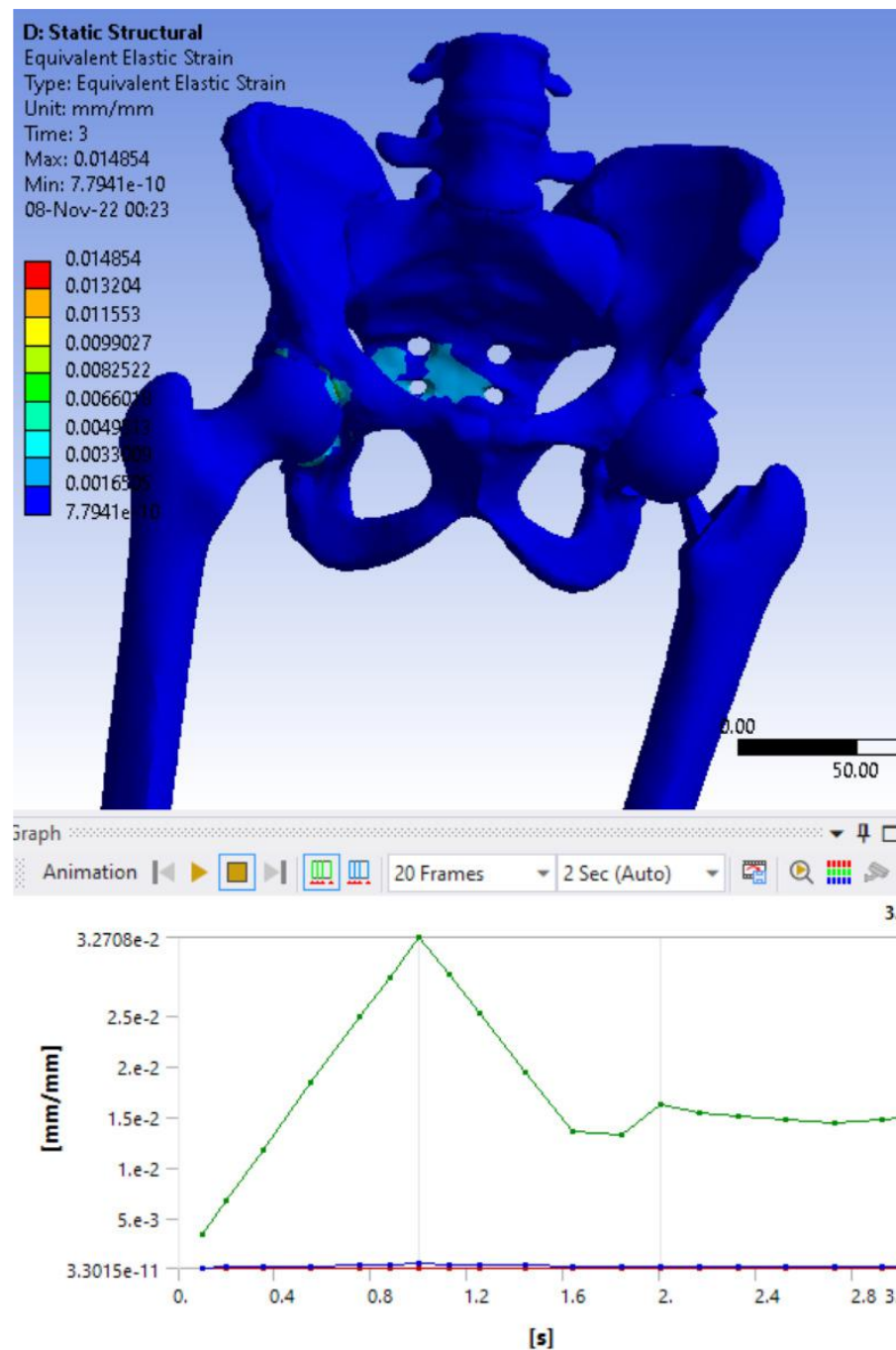
Right click, Generate Mesh and it looks like here; you can refine the mesh as you need. If Virtual Topology is not used, then the mesh count can increase, and meshing errors can appear.



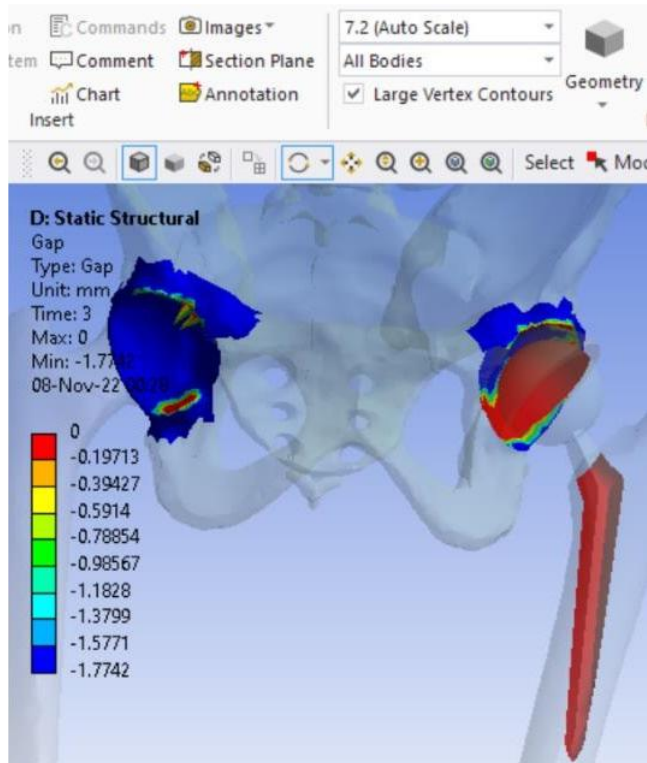
Right click on the Contact Tool which appeared in the tree, Insert, Pressure and Gap.



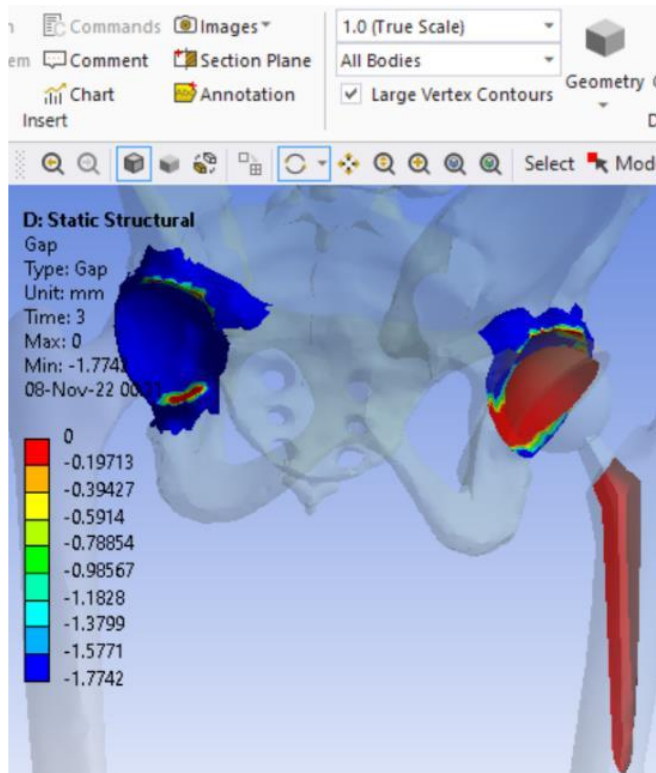
This is the Equivalent Train plot with a maximum of  $3.3\text{e-}2$  mm/mm.



Here is the Gap plot in contacts. The small peaks appearing on the left hip socket are due to the default Auto Scale which is 7.2 instead of the real one = 1.

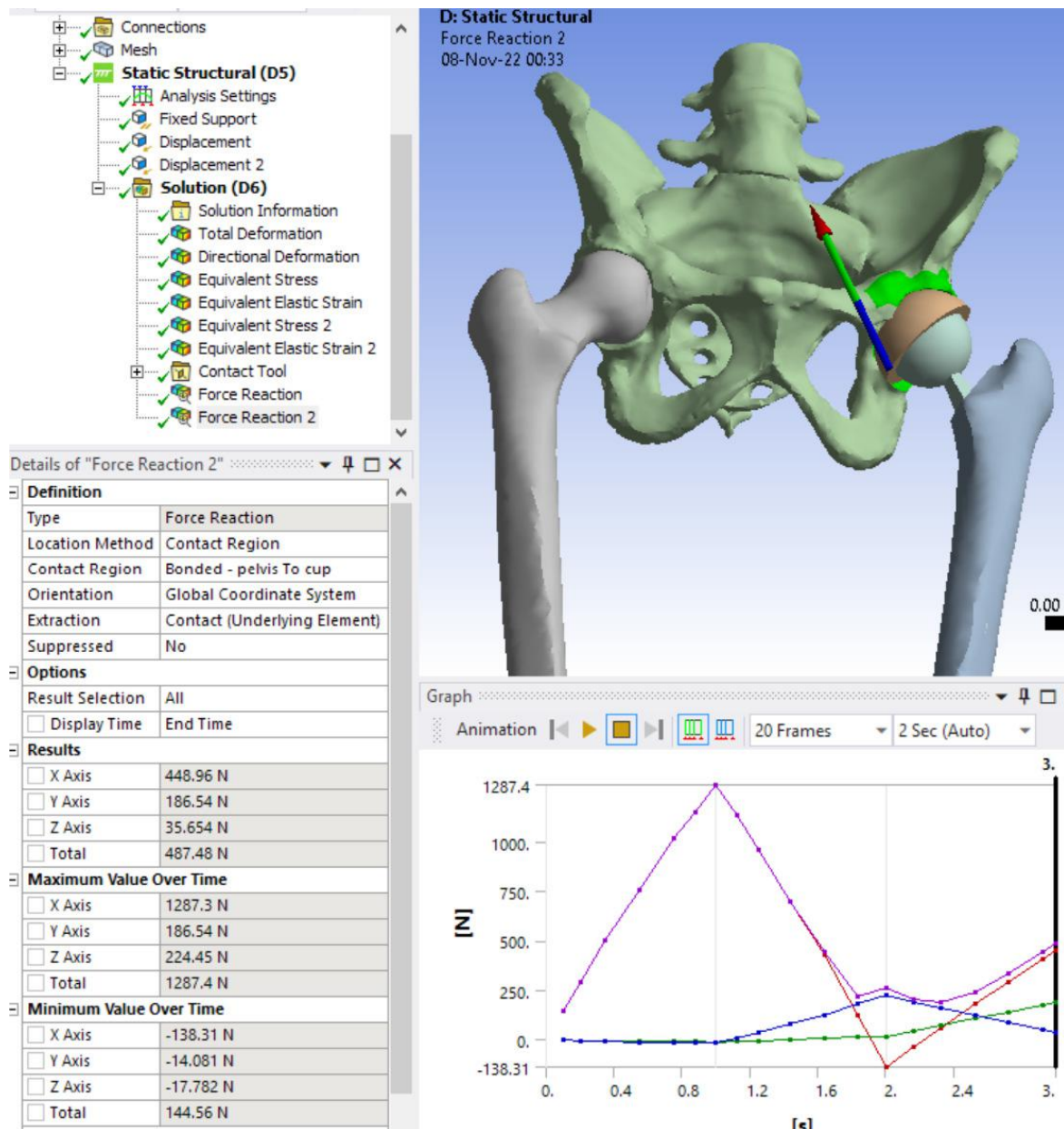


After we changed to True Scale = 1, the small peaks disappeared.





This is the Force Reaction plot with Details for the respective Bonded contact.



### Further homework:

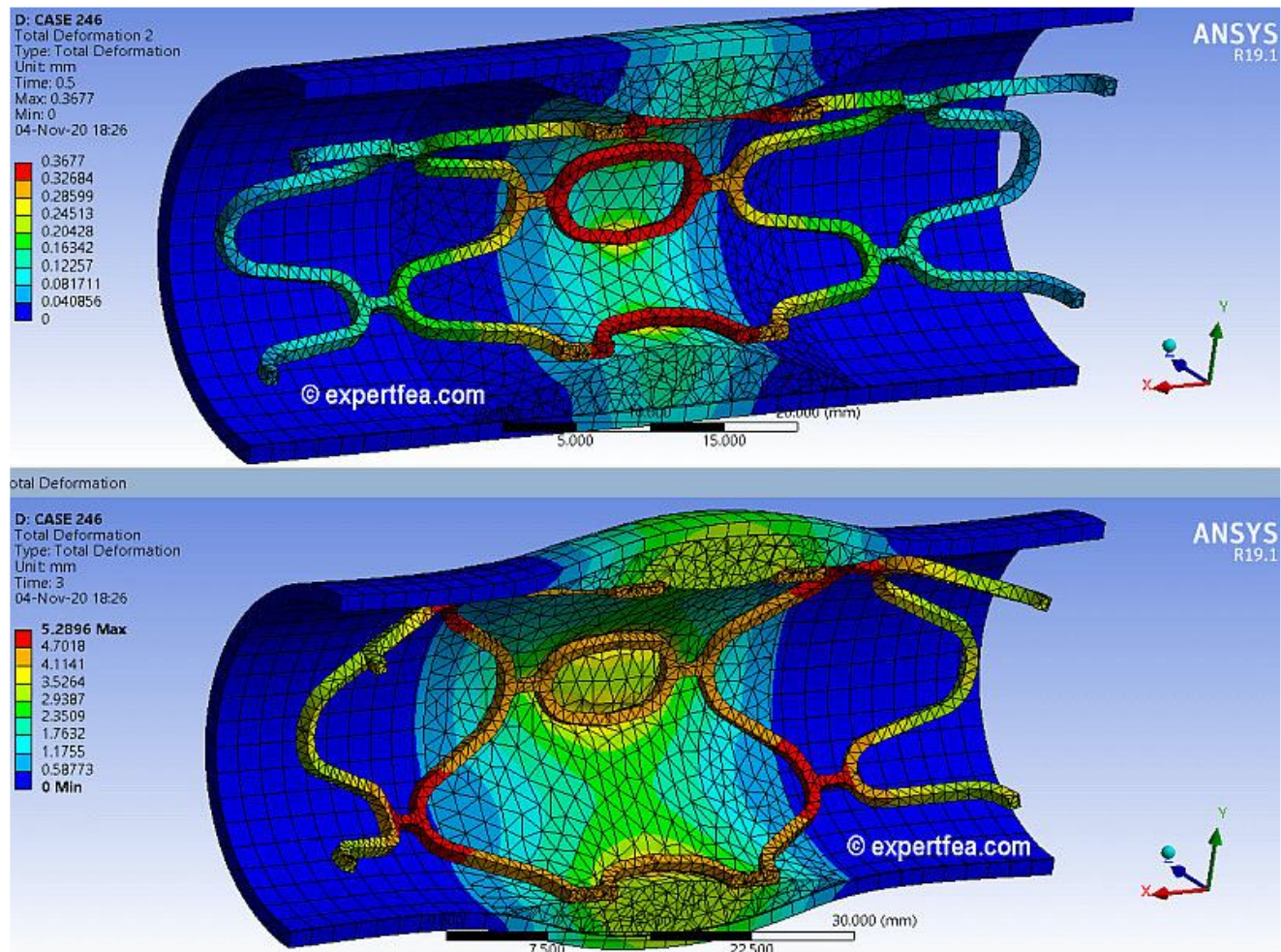
- to allow for plasticity (or remanent deformations) to occur at higher loads, replace Titanium Alloy with Titanium Alloy NL from Engineering Data, General Non-Linear Materials library; solve and draw the conclusions

- go to Mesh, make Element Sizing for the part  e 8 mm, solve and draw the conclusions

- make all contacts as Type = Frictional with  $\mu = 0.19$ , solve and draw the conclusions, see the Pressure in Contacts (if the FEA does not solve, end, start changing contacts to Bonded type)

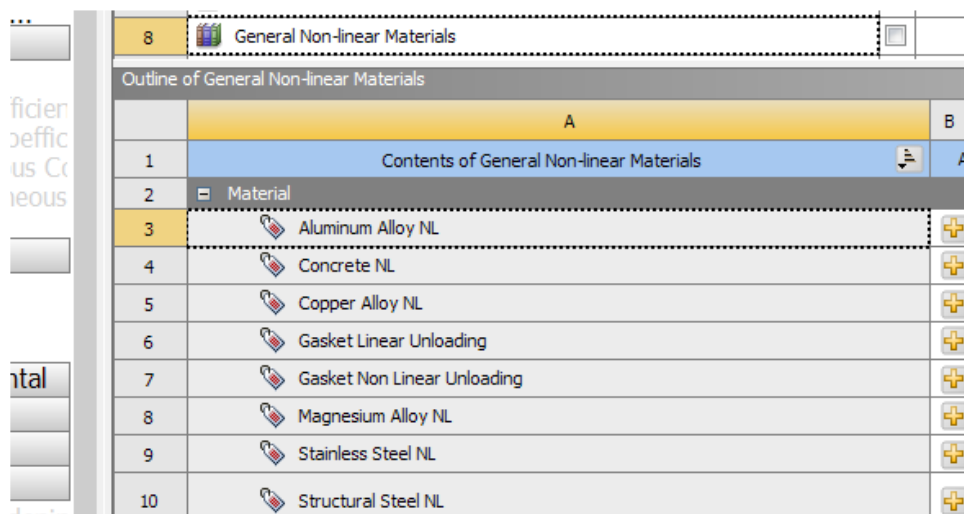
- change the Displacement values from 5 mm to 10 mm in their tables; solve and draw the conclusions



**CASE 246: Structural simulation of a steel stent expansion - ANSYS WB Static Structural**

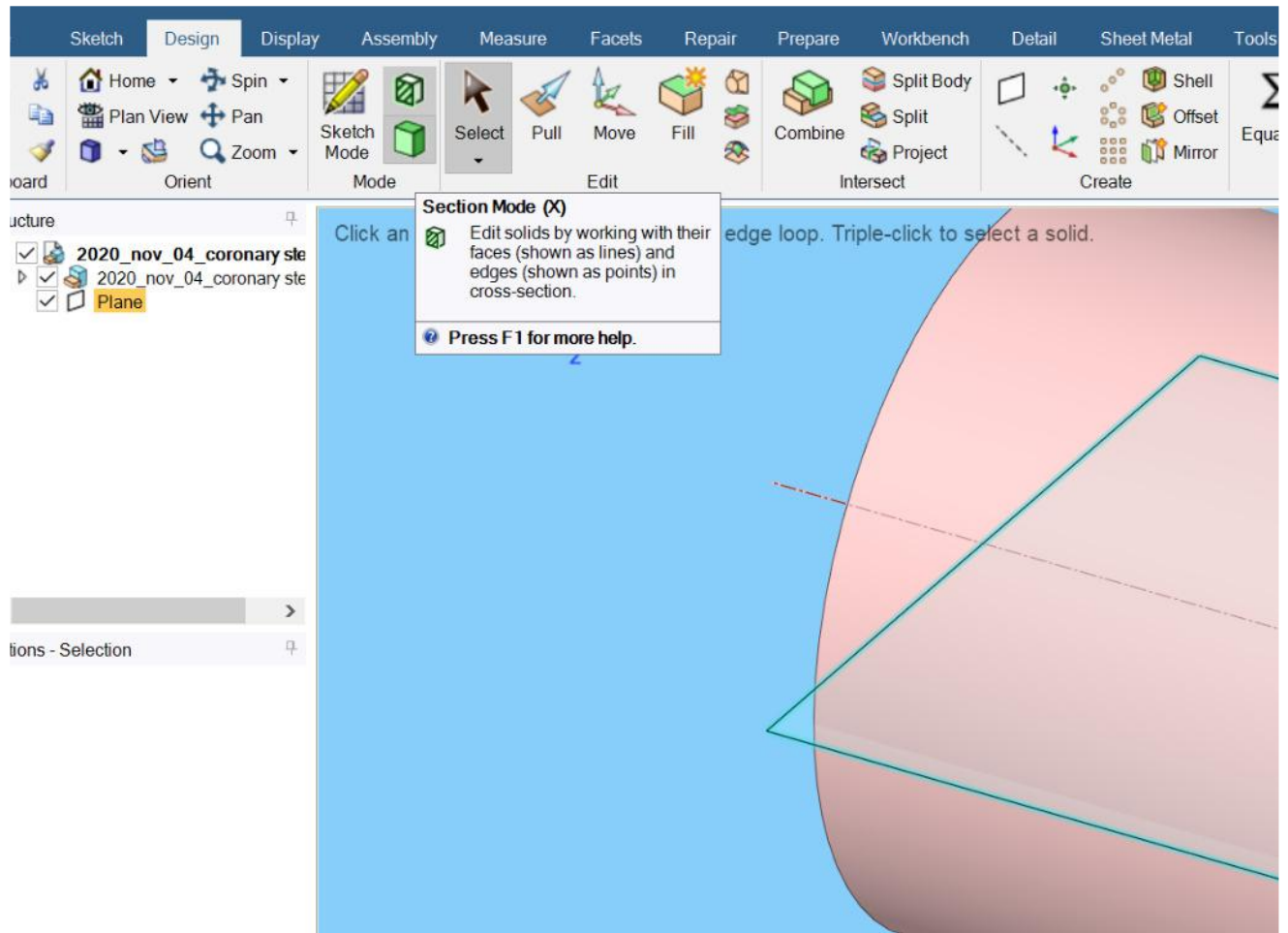
Once in Workbench, double click or drag and drop a Static Structural scenario from the simulations list in the left.

**Engineering Data (Materials):** In order to avoid unrealistically high stresses, we need to allow plasticity in the stent material, so go to General Non-Linear Materials and Add the Structural Steel NL material by clicking the yellow + sign from column B.

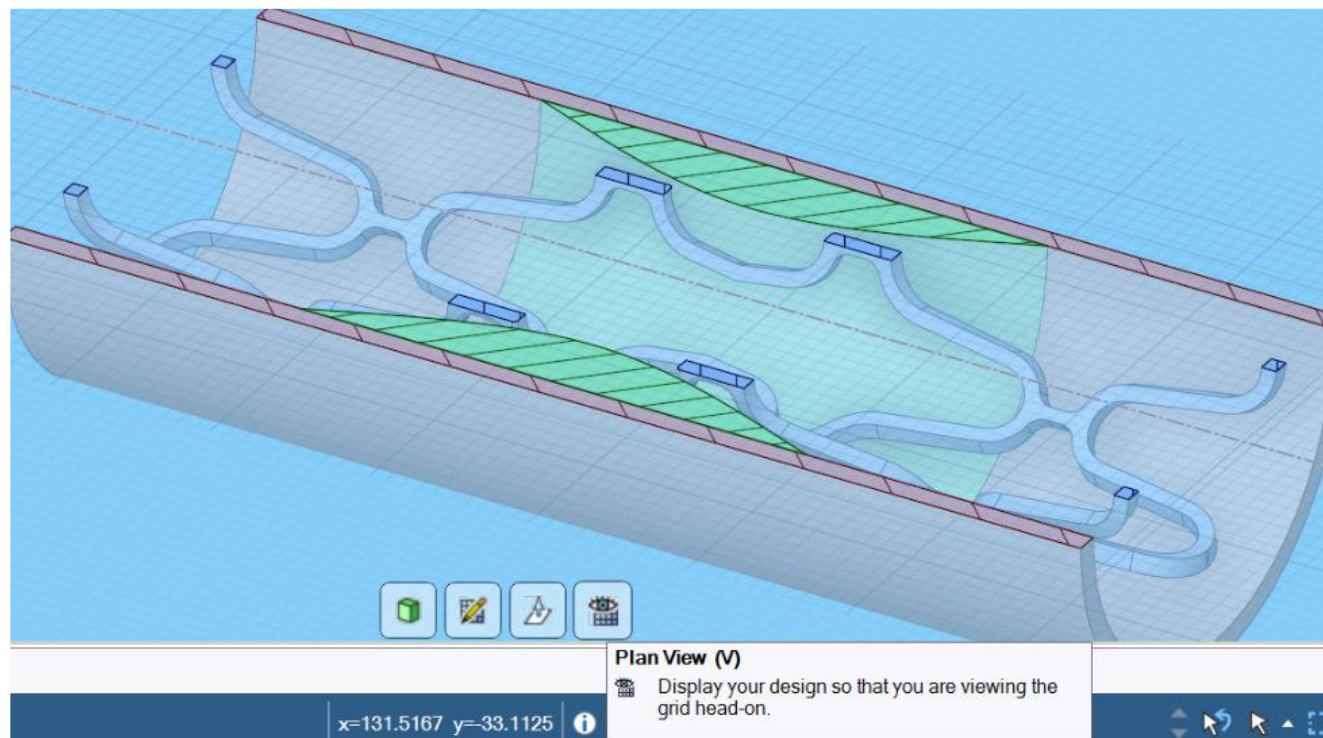




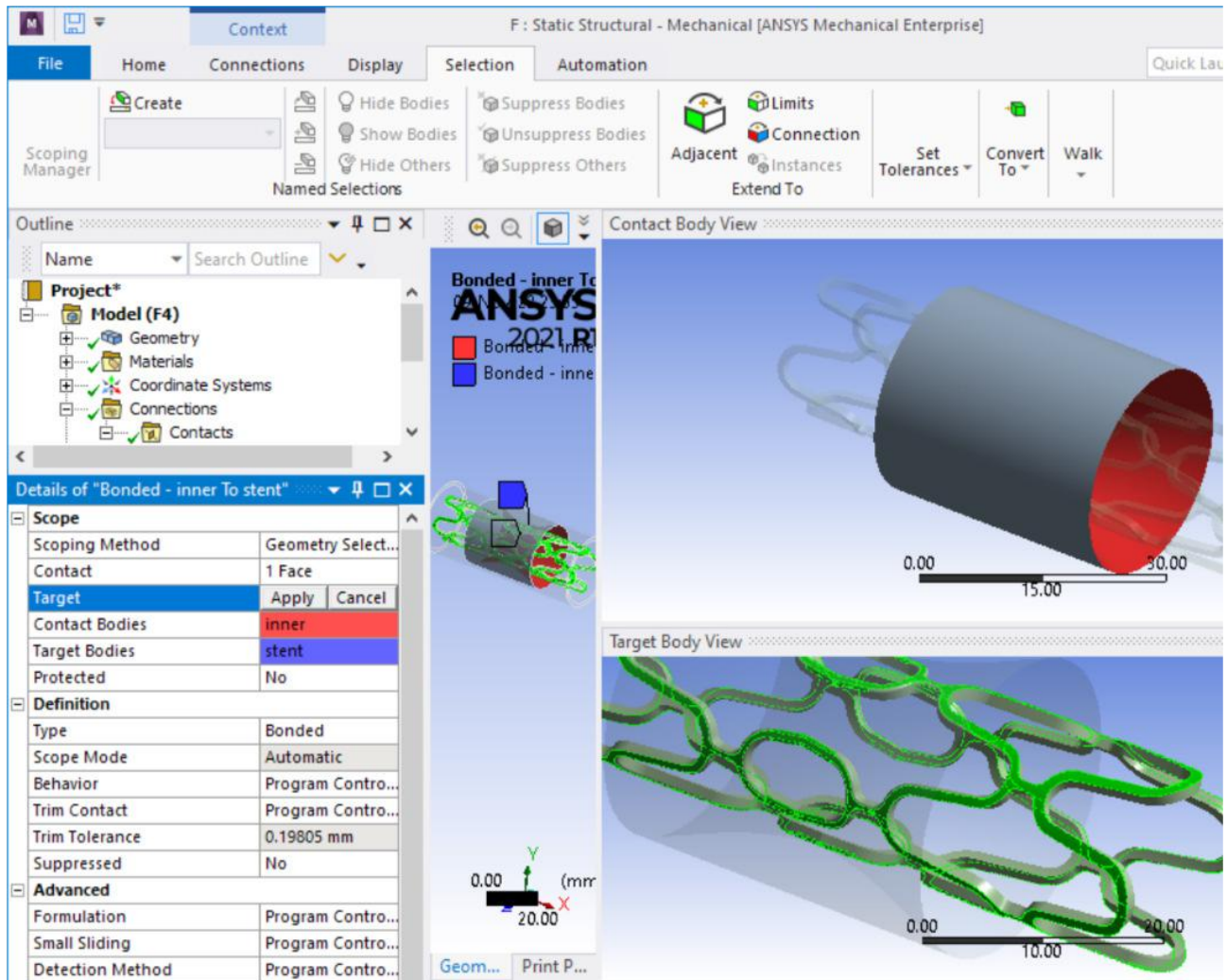
Hit Esc to exit the command, then go to the tree in the left, select the Plane, then click the button for the Section Mode. Shift + middle mouse to pan. Rotate the scroll of the mouse for zoom.



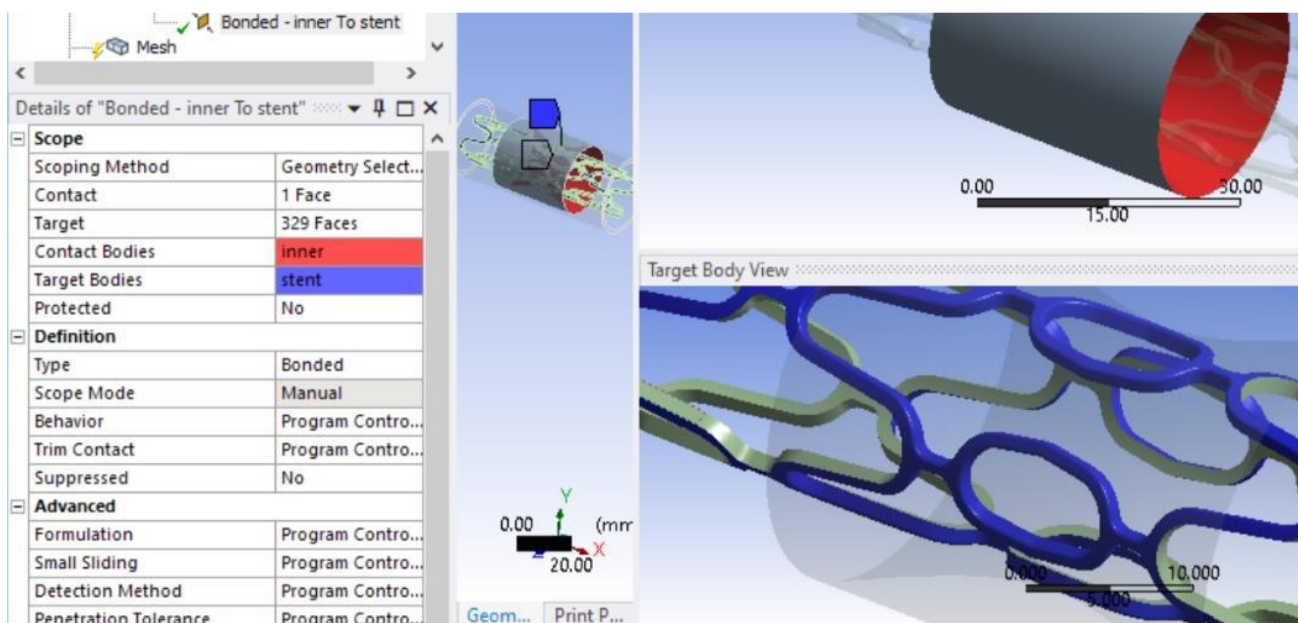
To have the view normal to your face, click the button from the left, the one with an eye over a grid.



Then go to the Selection tab on the top and click Adjacent 1 - 2 times to gather more faces than we had before, Apply.

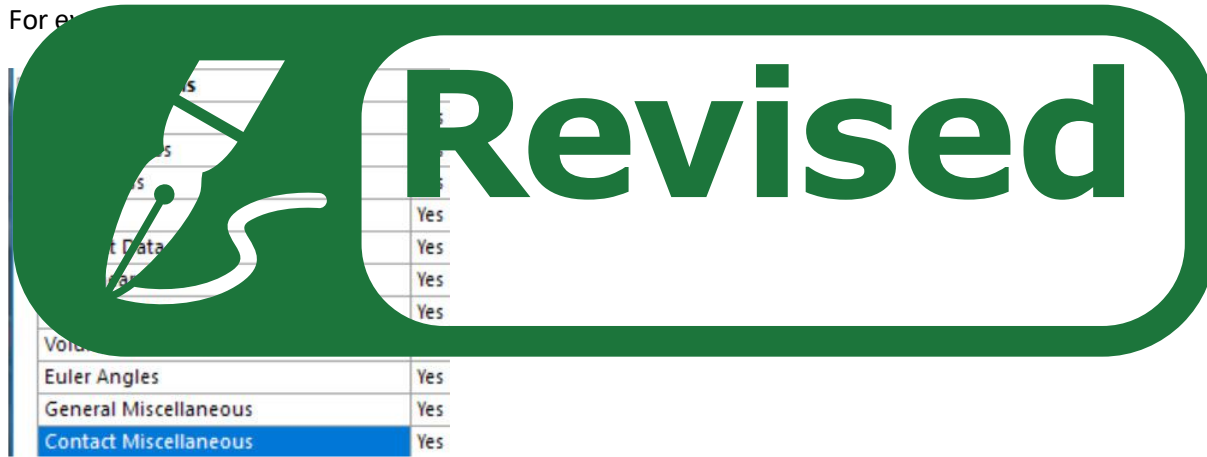


This is the resulting selection in blue after having clicked 2 times on Adjacent.

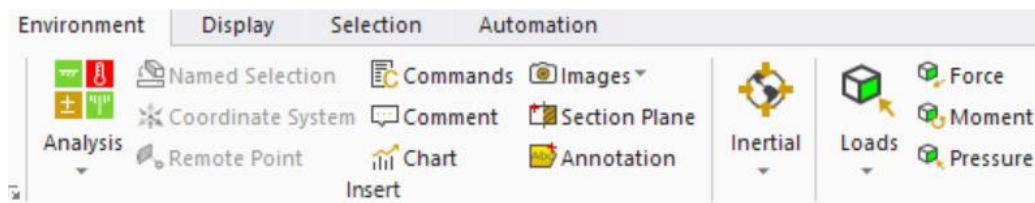




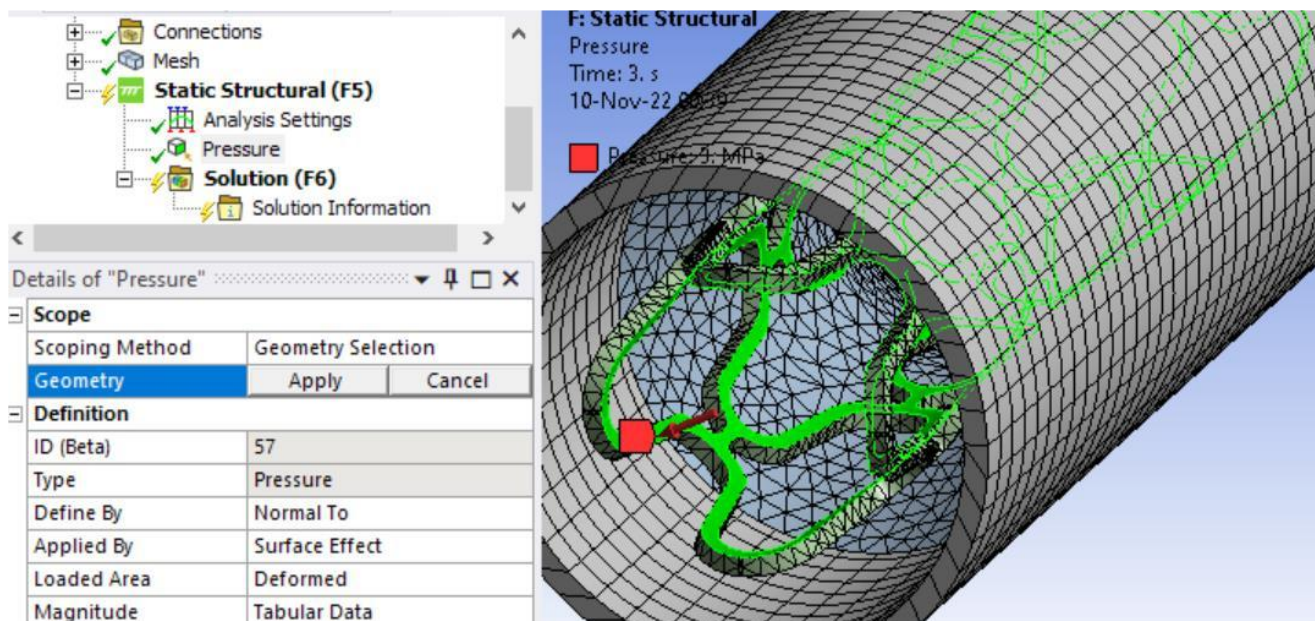
For ex



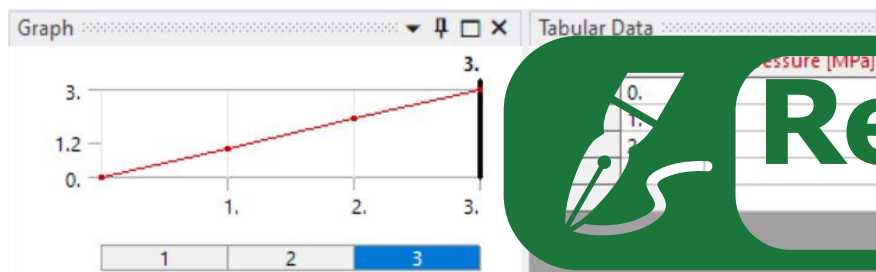
Fro the Environment tab on top, click the Pressure button shown on the lower right here.



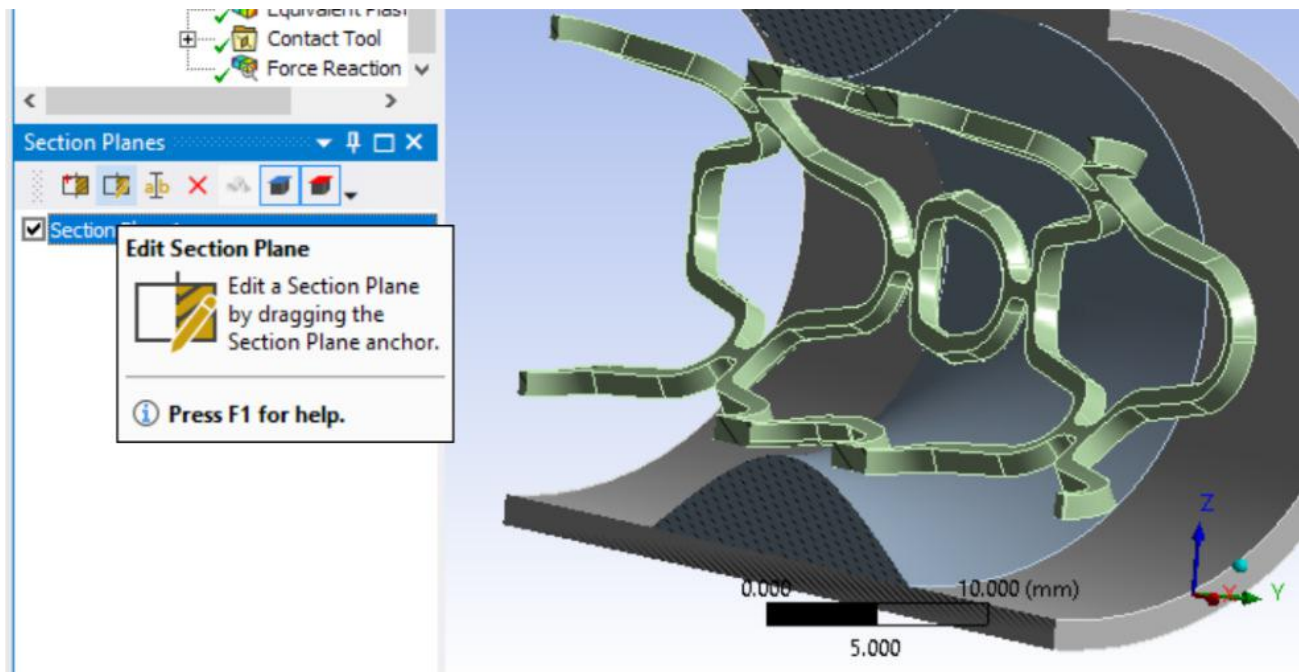
Select the face gren on the inside of the stent, make Magnitude as Tabular, Apply.



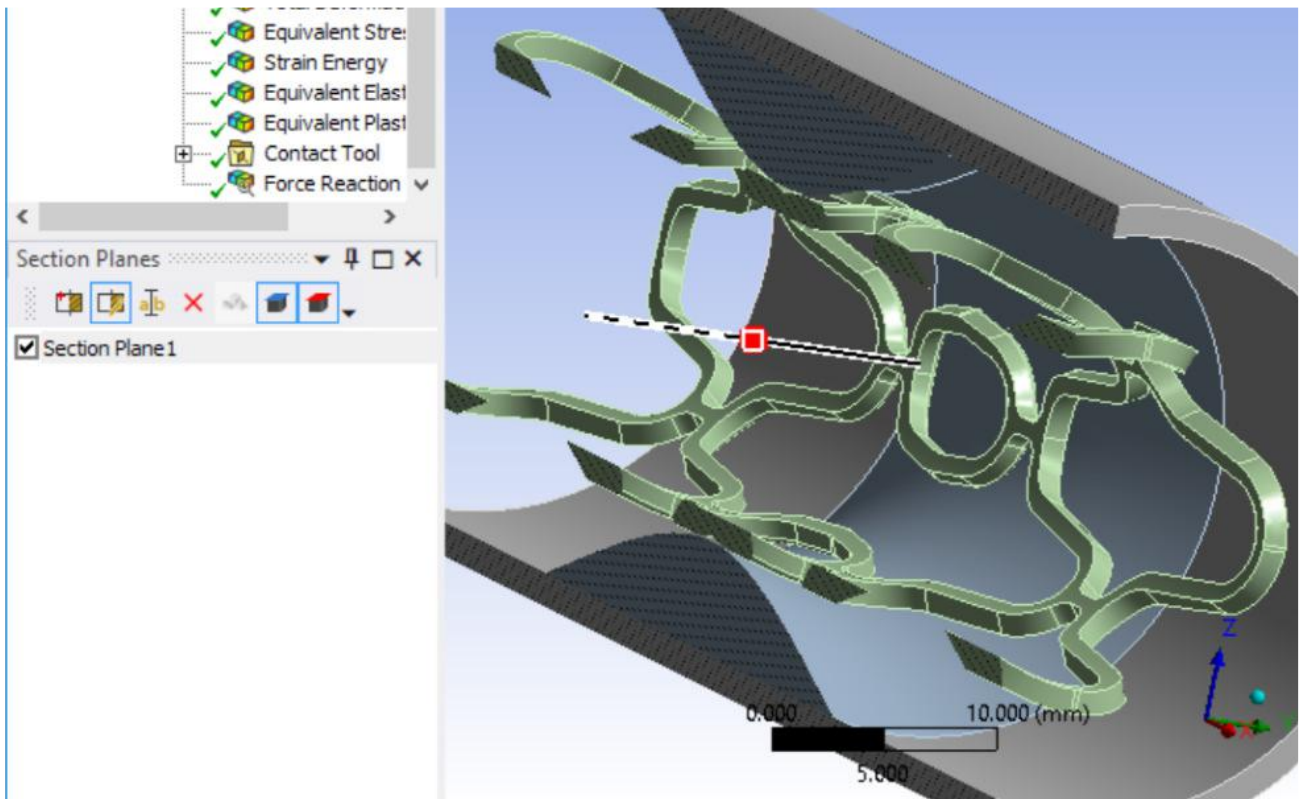
Apply these values in the table.



Rotate the image for a better view, then click the Edit Section Plane button.



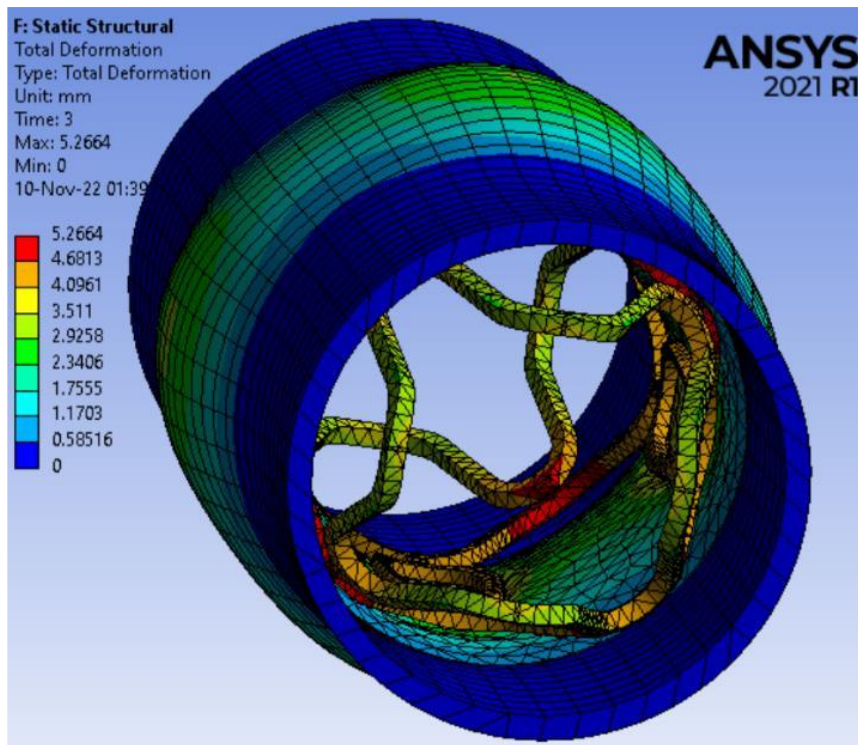
Then move the middle square or box to pull the section to show more struts touching the inner plaque.



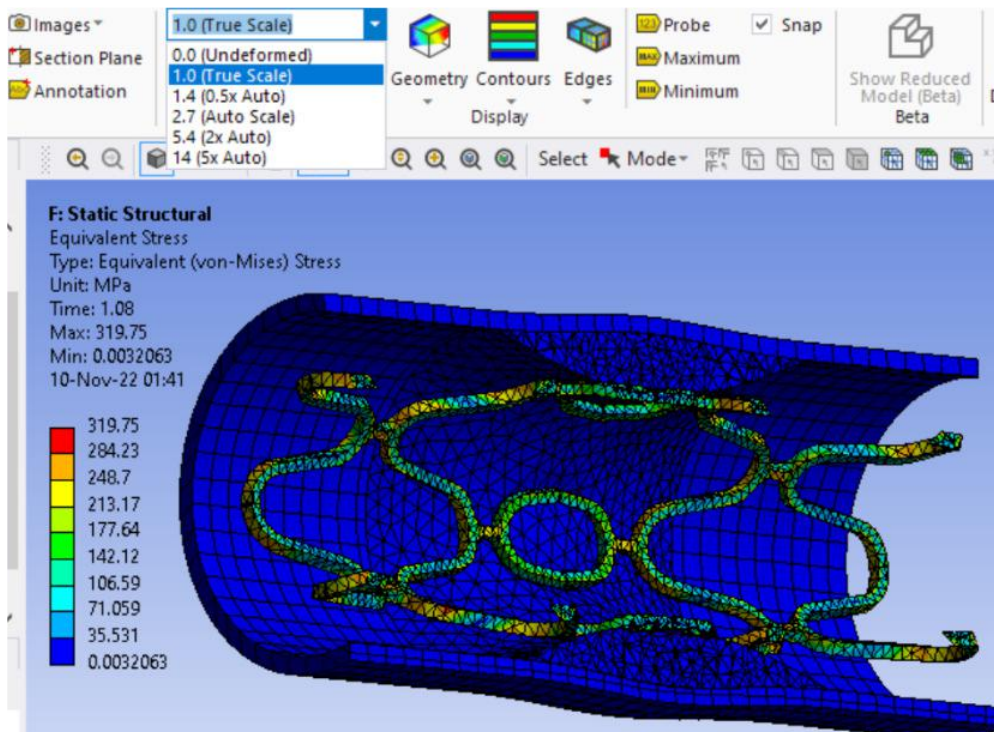
You can uncheck Section Plane1 to obtain a full view of the assy.



This is the Total Deformation plot.

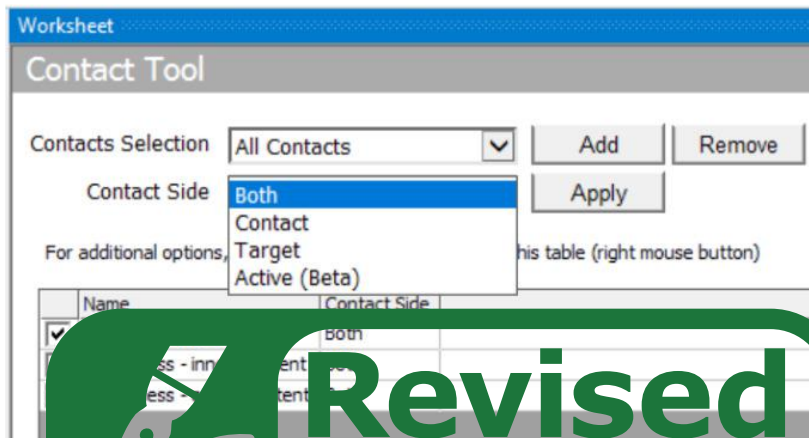


Here is the Equivalent Stress plot after the section view was activated from the Section Planes tab. If needed, from Results tab, Display, change the scale to True Scale = 1. The value being higher than 250 MPa, we can expect plasticity to occur in the stent.

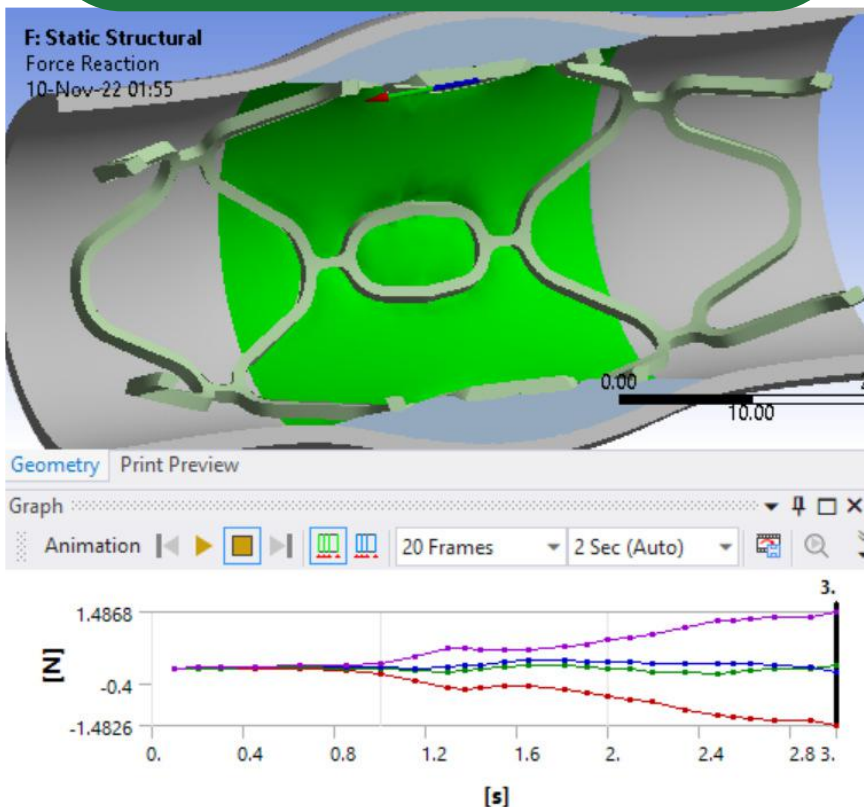




Because different sides of the contact were left active, the above contact results show certain sides. Click the Solution, Contact branch from the tree in the left and see that we can change the Contact Selection and the Contact Side for each one, thus allowing for solution plots on certain contacts and sides.



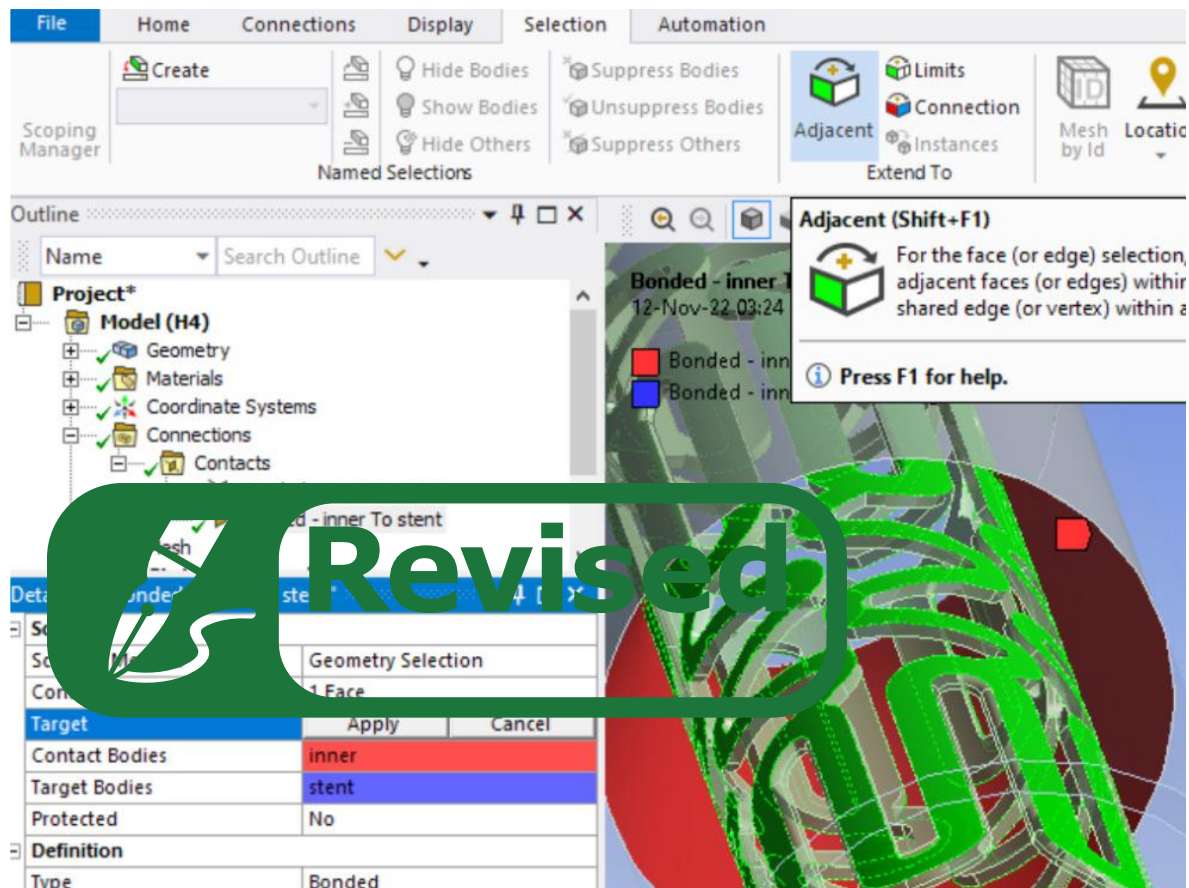
The Contact Results plot looks like here, for the chosen contact.



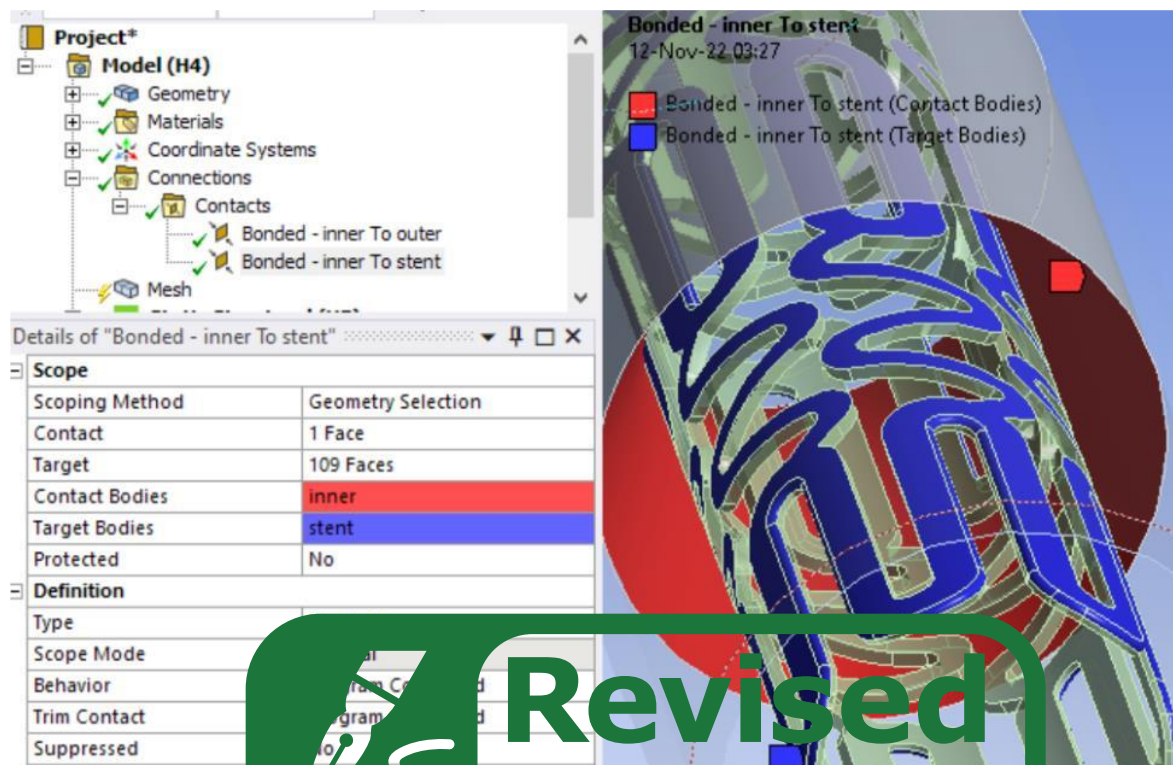
#### Further homework:

- to deny plasticity (or remanent deformations) to occur, replace Structural Steel NL with Structural Steel from Engineering Data, General Materials library; solve and draw the conclusions
- go to Mesh, Element Sizing to be 0.5 mm; solve and draw the conclusions
- make all contacts as Frictionless; solve and draw the conclusions
- double the pressure for each step; solve and draw the conclusions

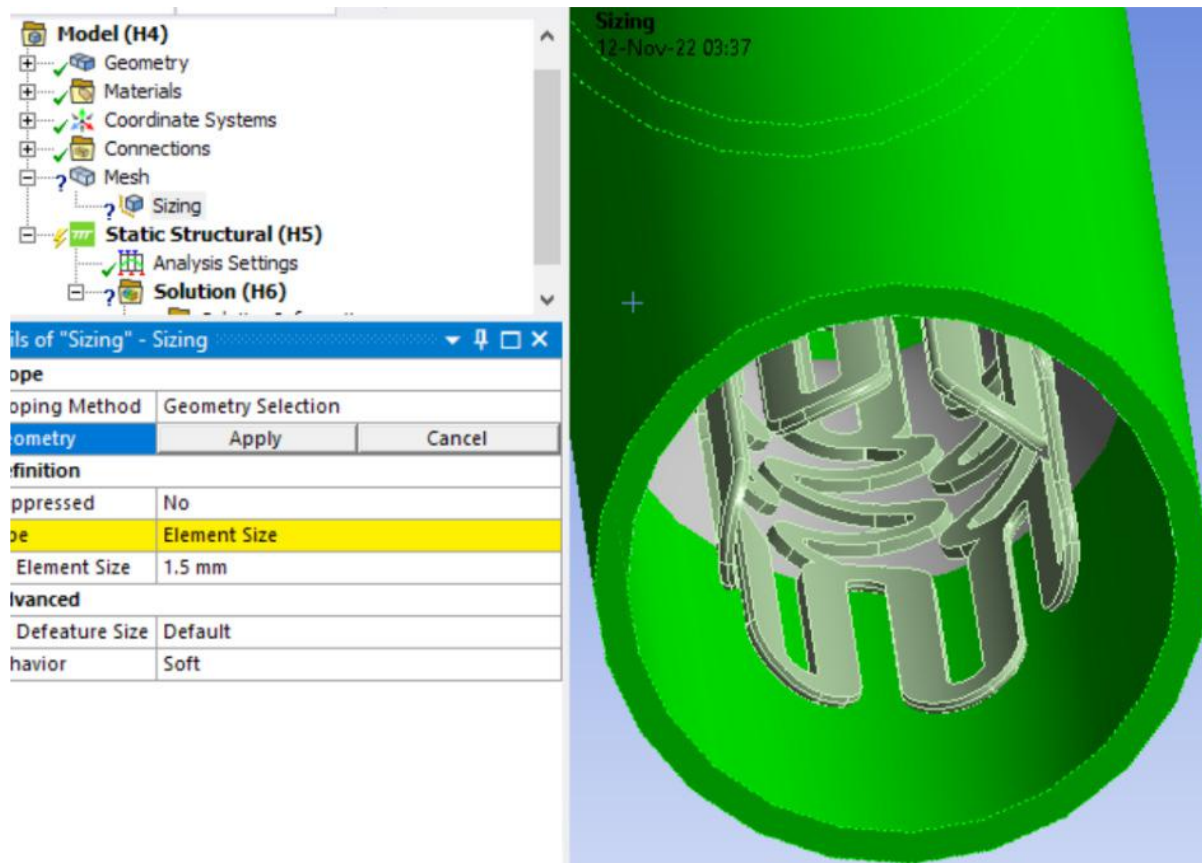
Go to the Selection tab, and click Adjacent once, then Apply.



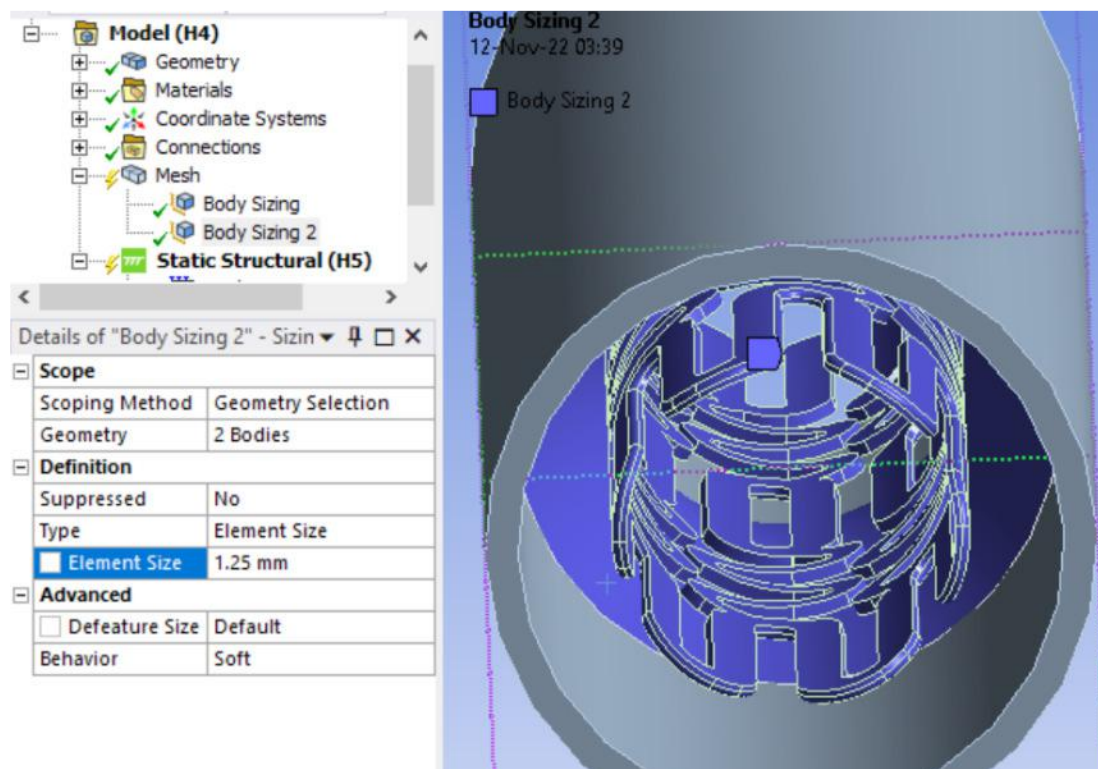
Observe how more faces got into the Target (or blue) side of the contact.



Ctrl+B to select bodies, then click the *outer* part, green here, Apply. Assign 1.5 mm.

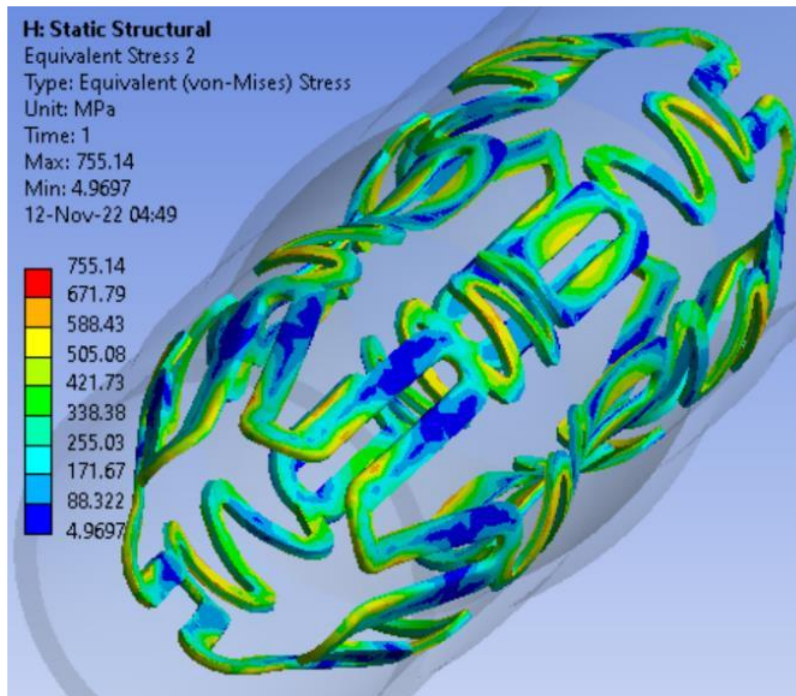


Insert a similar Sizing for the other 2 parts, seen blue here, 1.25 mm.

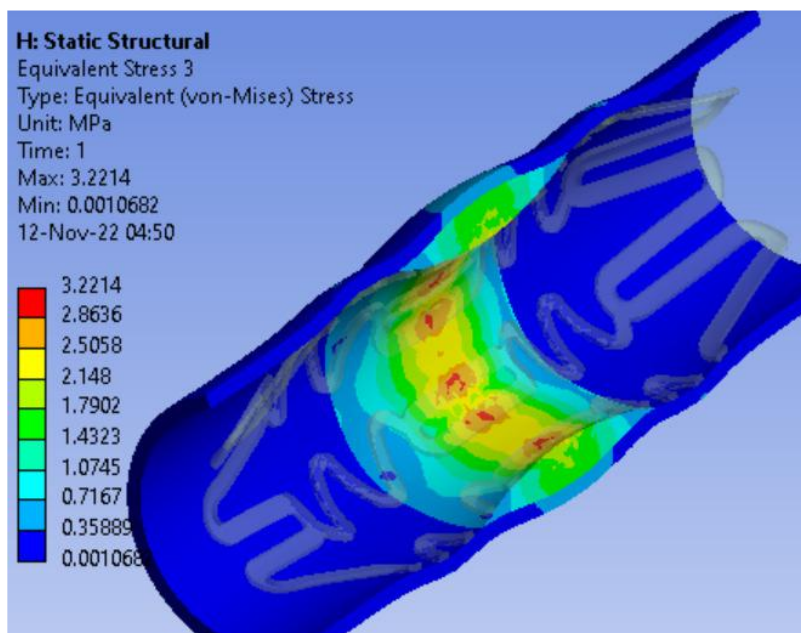




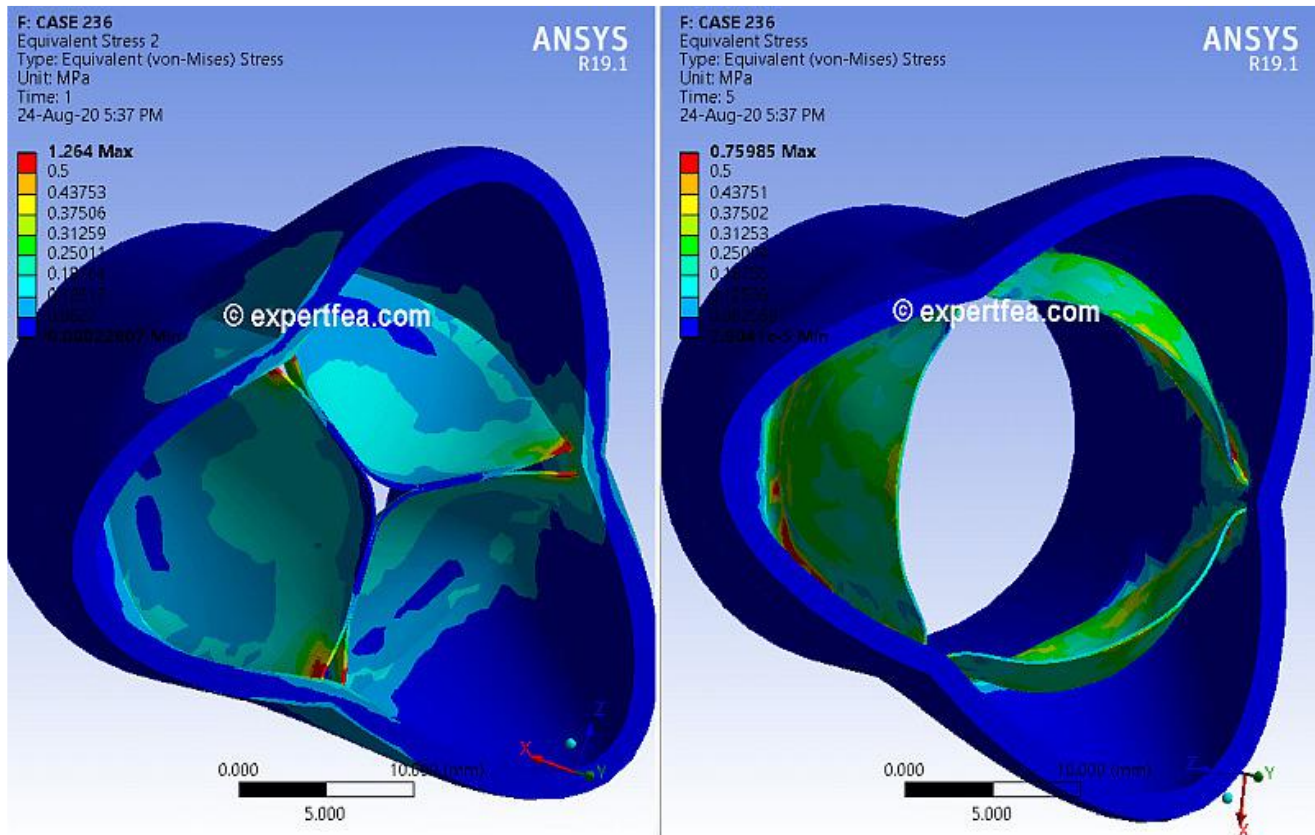
This is the stress only in the stent. The high values are allowed by the Superelasticity parameters we defined at the beginning of the tutorial.



Here is the stress in the 2 parts of the blood vessel. The lower values correspond to materials having behavior near hyperelasticity.



### CASE 236: ANSYS WB Static Structural - FEA simulation of the closing and opening of an aortic valve



**Engineering Data (Materials):** To easily create a material for the aorta with one close to having hyper-elasticity, right click the default Structural Steel, Duplicate.

Rename it *aorta* then insert these values for the respective parameters; you can delete other properties until you have only these ones.

1	Contents of Engineering Data				Source
2	Material				
3	aorta				Fatigue Data at zero mean : .1
4	Structural Steel				Fatigue Data at zero mean : .1
*	Click here to add a new material				

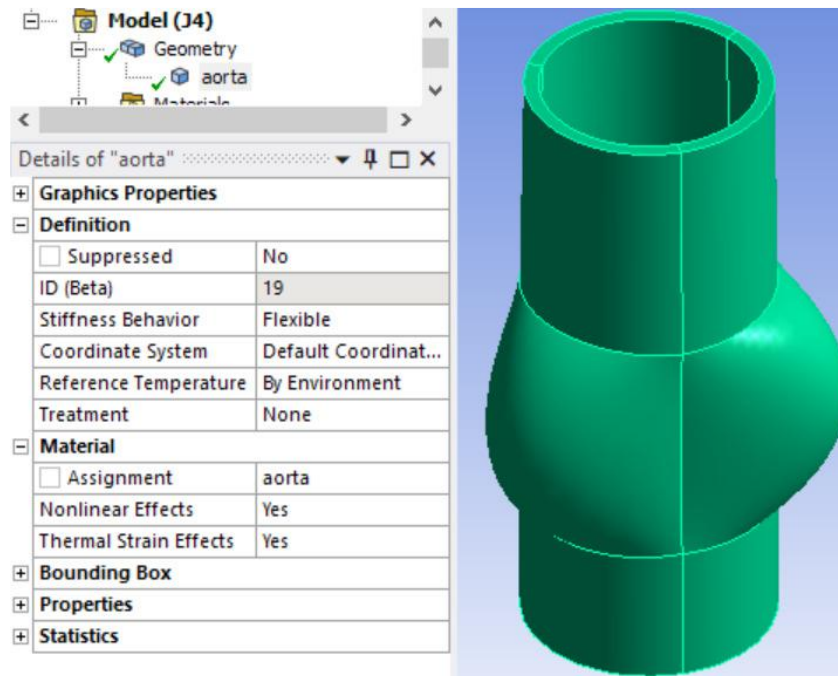
Properties of Outline Row 3: aorta			
	A	B	
1	Property	Value	
2	Material Field Variables	Table	
3	Density	1000	kg m <sup>-3</sup>
4	Isotropic Secant Coefficient of Thermal Expansion		
6	Isotropic Elasticity		
7	Derive from		
8	Young's Modulus		Pa
9	Poisson's Ratio		
10	Bulk Modulus		Pa
11	Shear Modulus		Pa



Geometry cell: Right click, Replace Geometry, Browse for *2019\_feb\_15\_aortic valve version 2.x\_t*

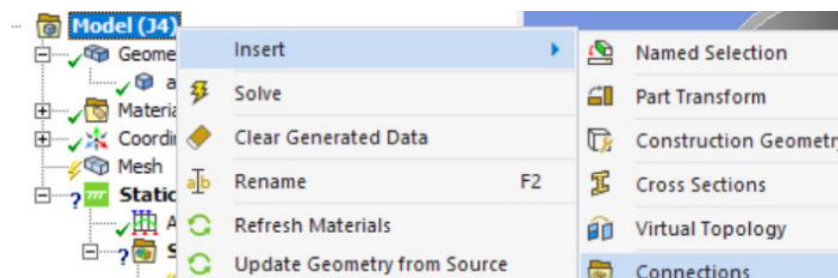
Model cell: Double click and the Static Structural – Mechanical window will open, with the proper FEA utilities.

Geometry branch: Rename the part as *aorta* then apply the respective material to it. This should be the result.



By default, because it is a single part, ANSYS does not assign contacts to it, so we need to do that now.

Right click the Model branch, Insert, Connections.

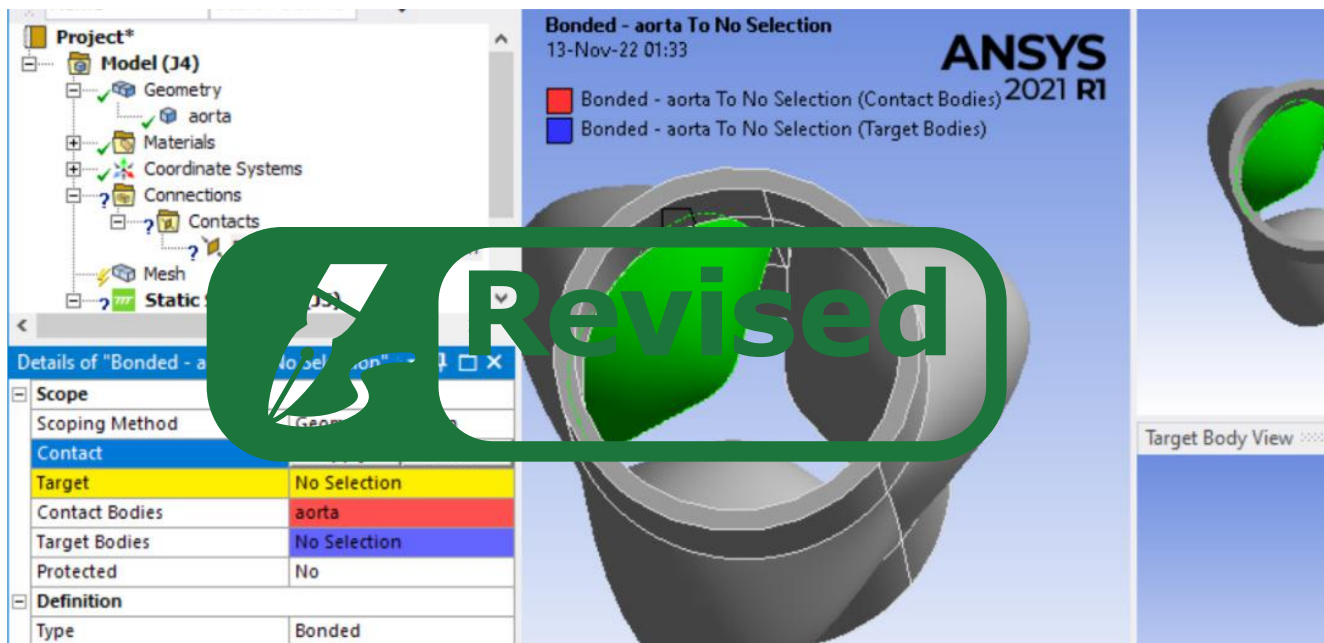


Right click Connections, Insert, Manual Contact Region.

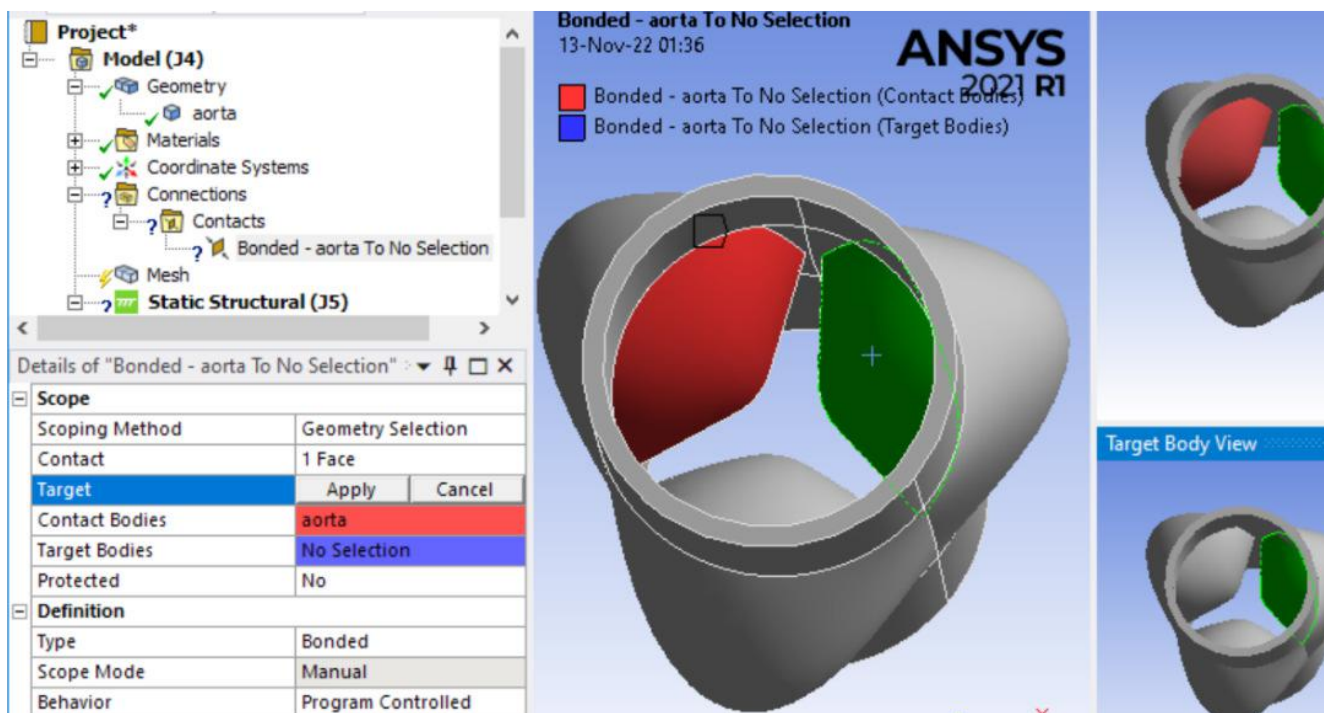




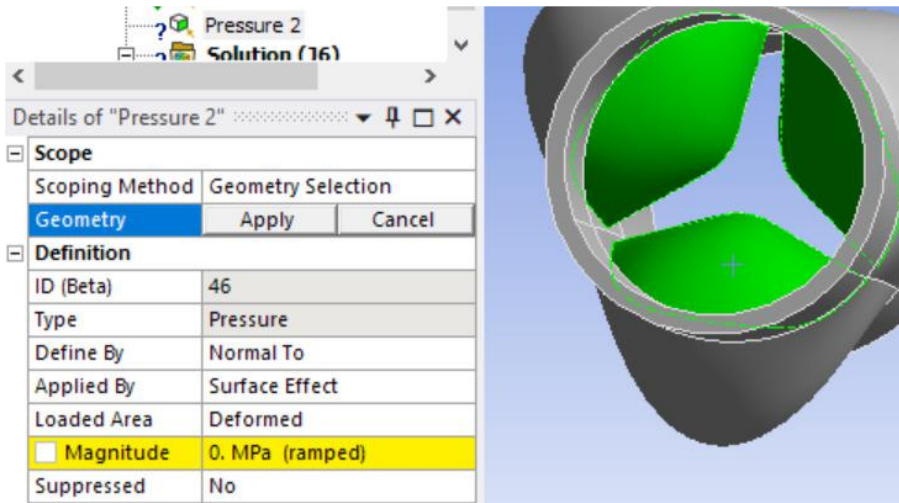
Click Scope, Contact, click one leaflet on the concave side (or bulging towards us, green here), Apply.



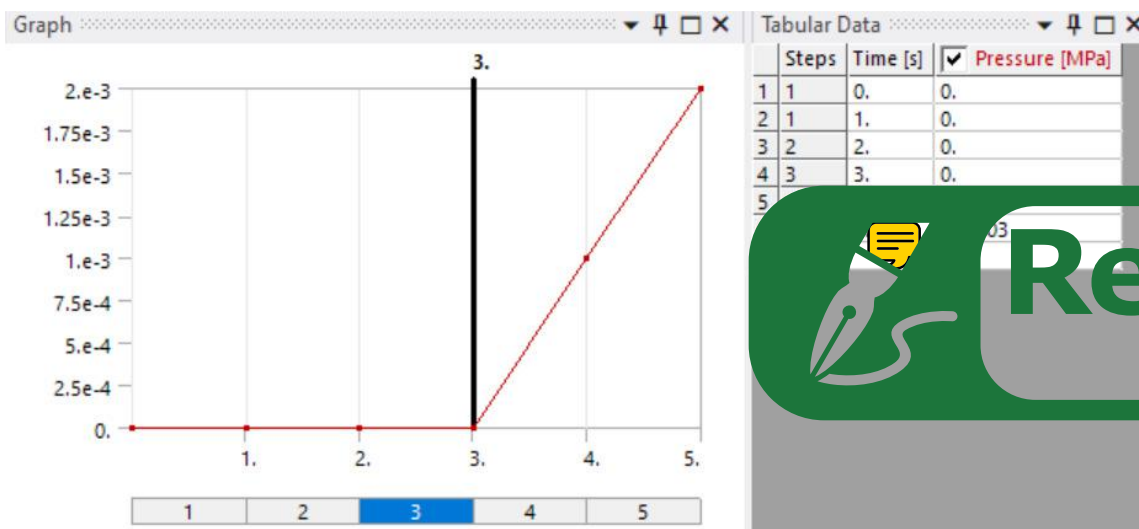
Click Scope, Target, click a neighboring leaflet on the concave side (or bulging towards us, green here), Apply.



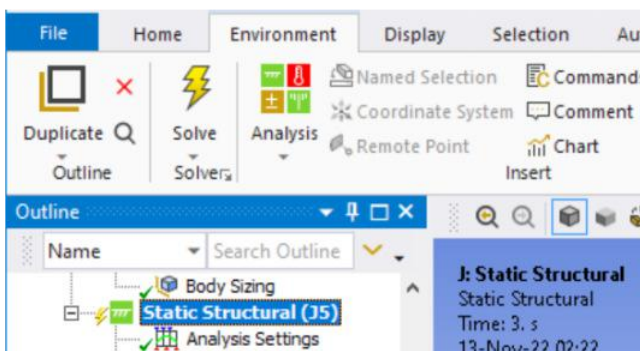
Insert another Pressure item to apply it on the opposite side and direction, to achieve an alternative (or pulsating) movement of the leaflets. Select the concave faces, seen here in green, Apply.



Make Magnitude as Tabular then insert these values in the table. To avoid autocompletion, start inserting the values from the bottom rows going up.

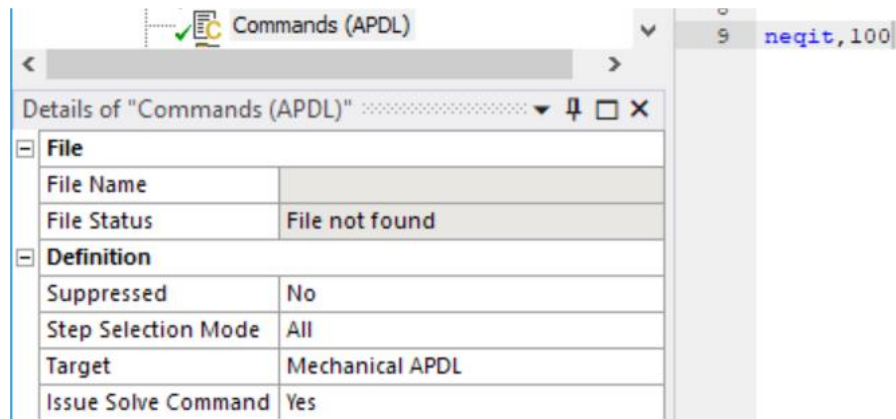


By default, ANSYS assigns 26 equilibrium iterations, so let us extend this value to 100, to help convergence in the solver. Select the Static Structural branch, then click the Commands button from the Environment tab.

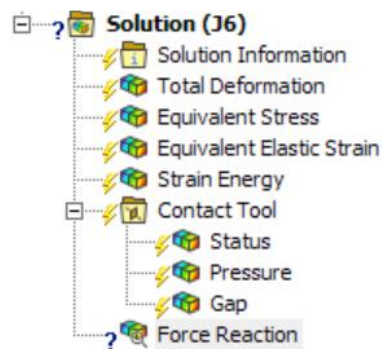


Click inside the 9<sup>th</sup> row and write NEQIT,100. Then assign the details seen below in the Definition branch.

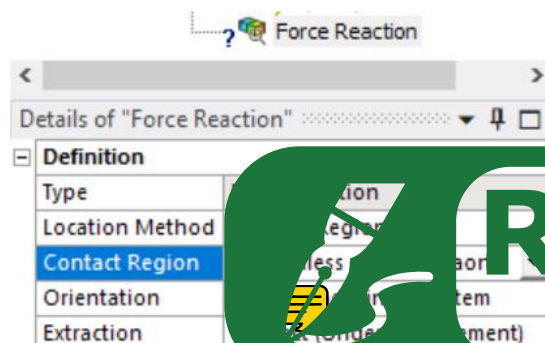
**NEQIT.** Specifies the maximum number of equilibrium iterations for nonlinear analyses



Solution: Insert these items from the respective toolbar.

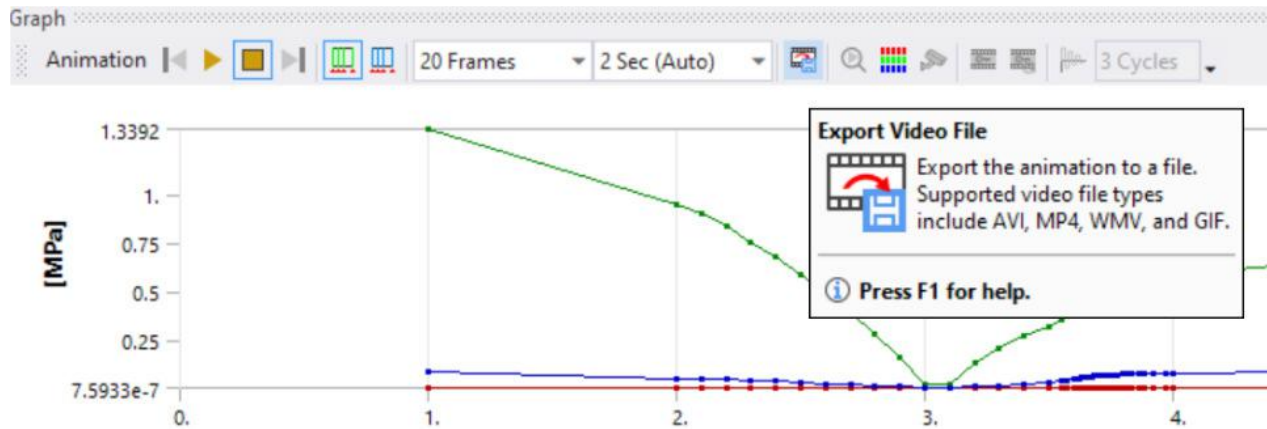


To assign any of the contacts, make Location Method, Contact Region; Contact Region, Frictionless – aorta To aorta.

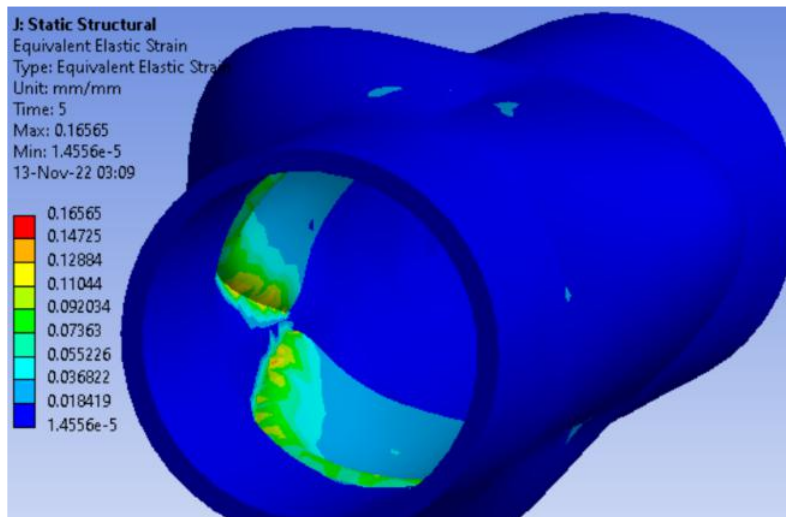




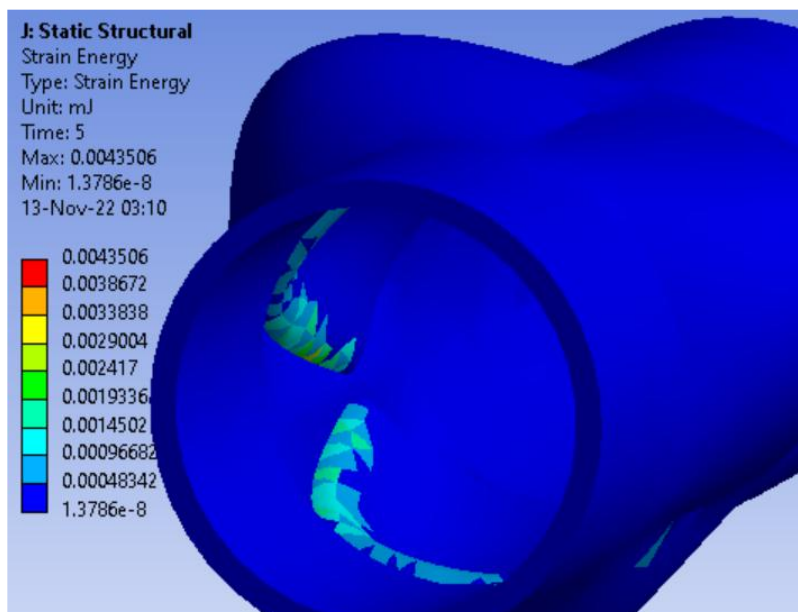
To create animations, you either press the Play button from the Animation toolbar below or Export Video File.



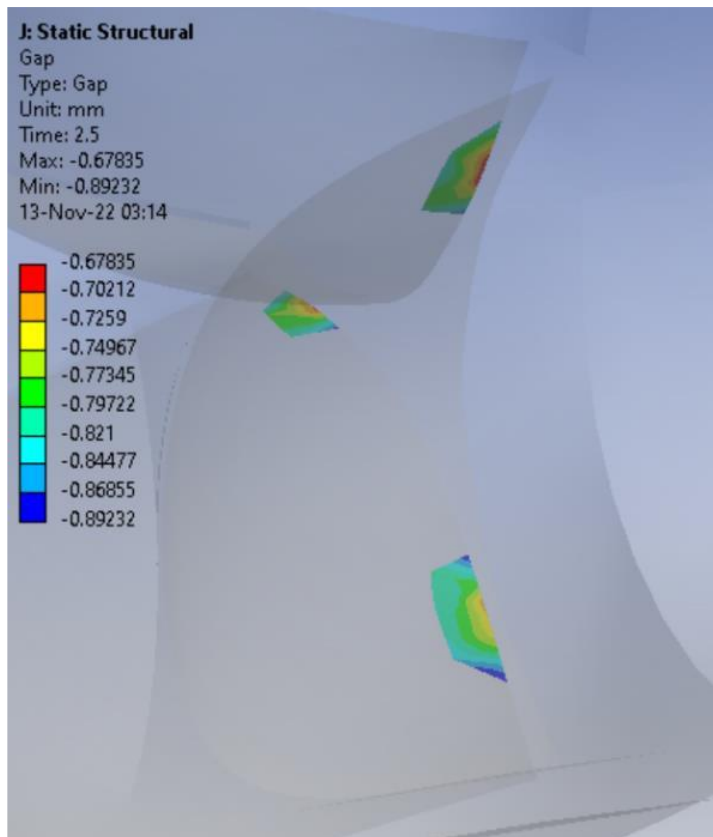
This is the Equivalent Elastic Strain plot.



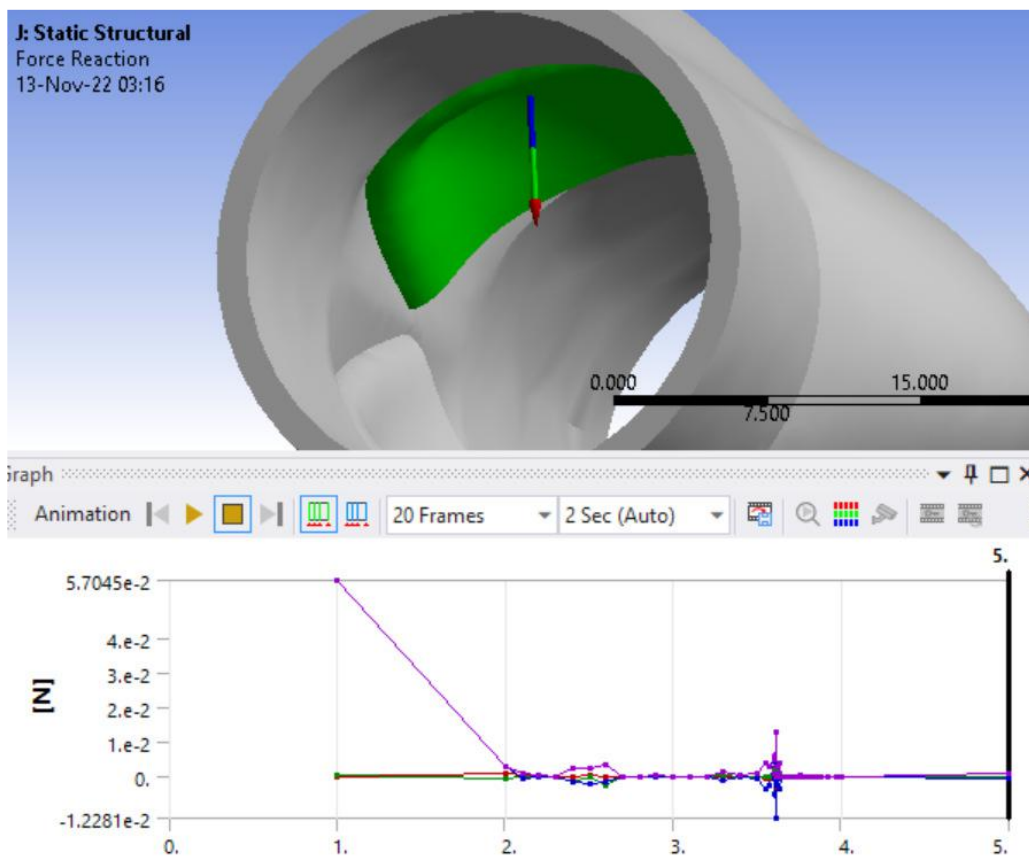
Here is the Strain Energy plot which signals an optimal mesh with such low energy values.



This is the resulting plot, with negative values showing a suction action.

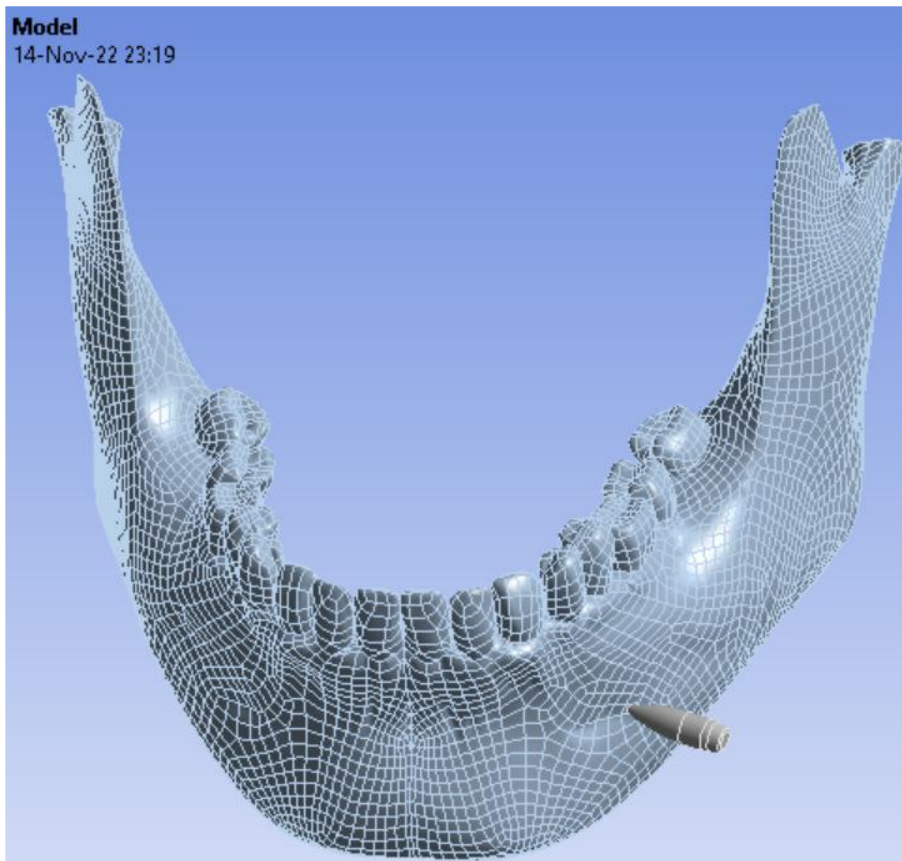


The Force Reaction plot looks like here, with an unexpected peak at the start and other peaks during the solving, which are expected.

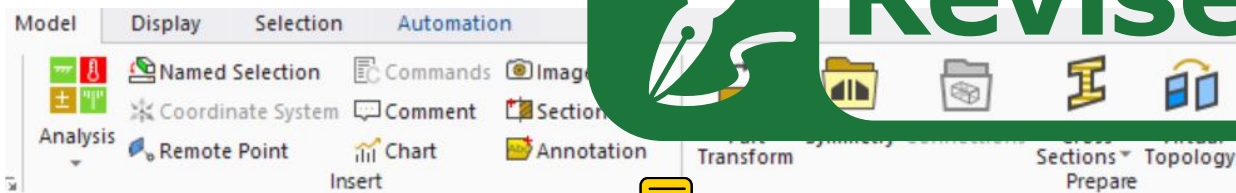


Model: Having this branch selected, you can access the items from the Model tab on the top.

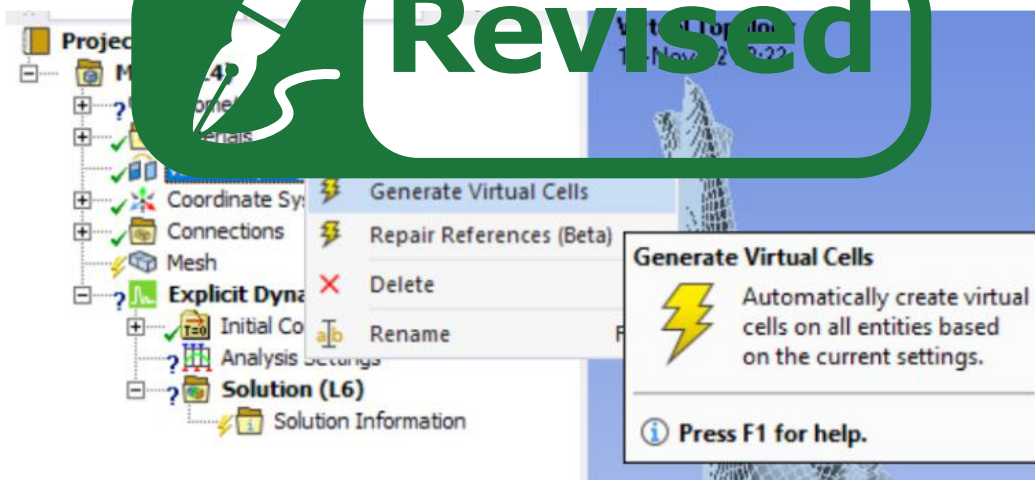
Observe that the initial geometry has quad facets which will impose the mesh shape and density.



To heal and simplify the geometry for the meshing menu.

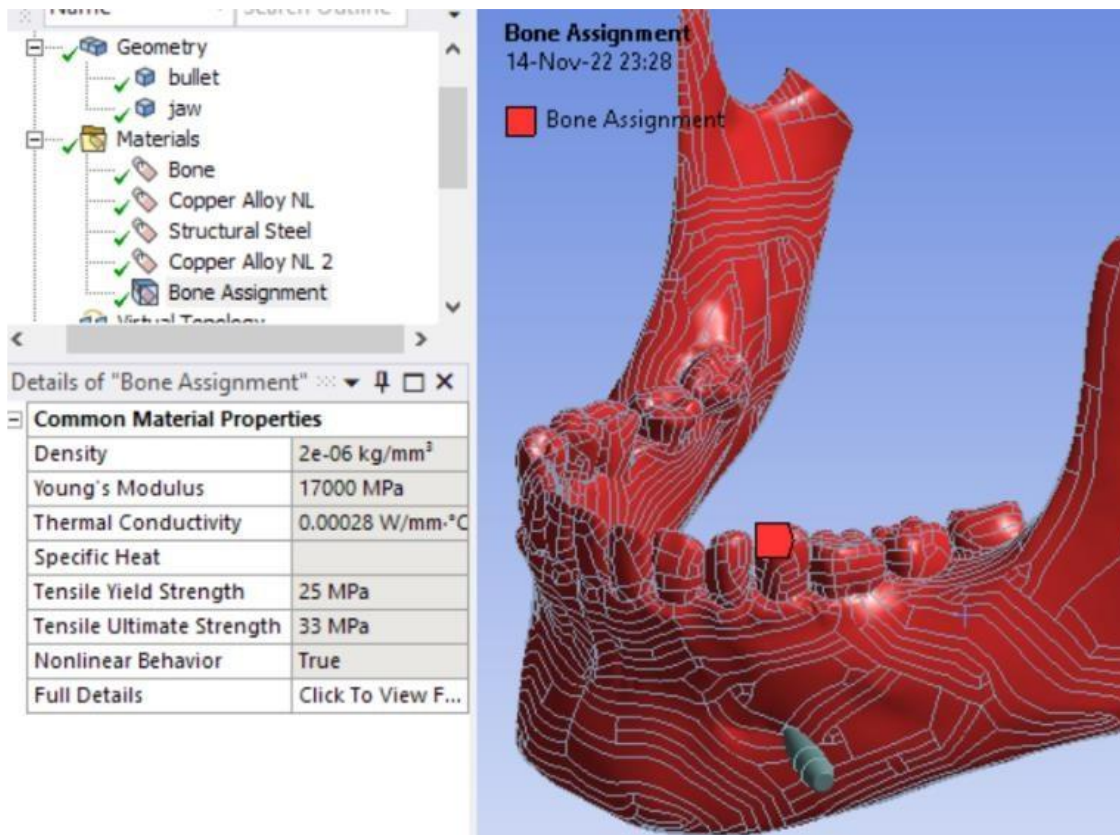


Right click Virtual Cells from the tree in the left, generate Virtual Cells.

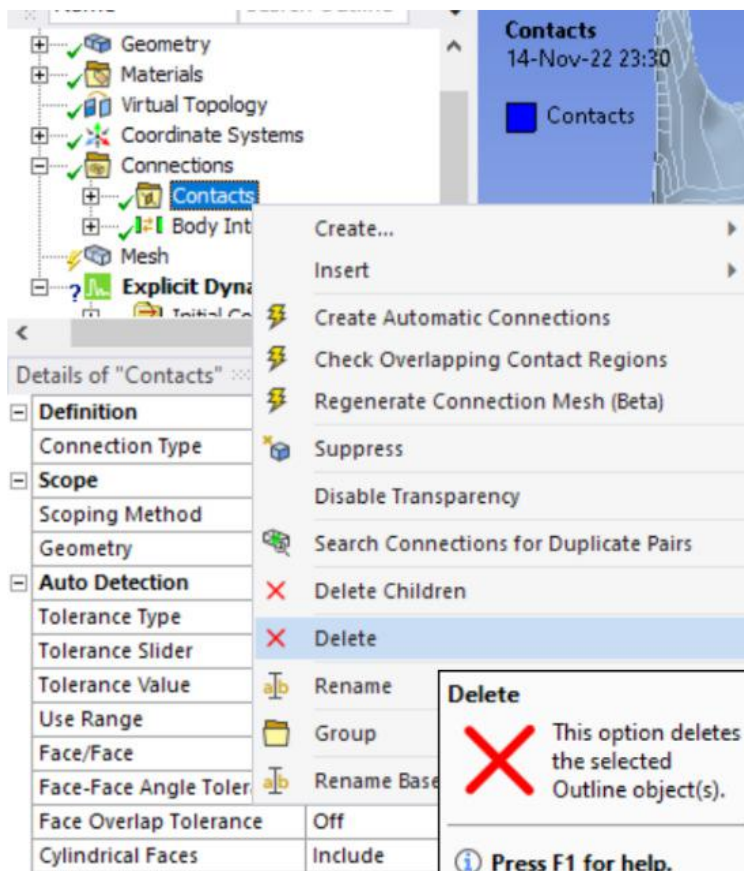




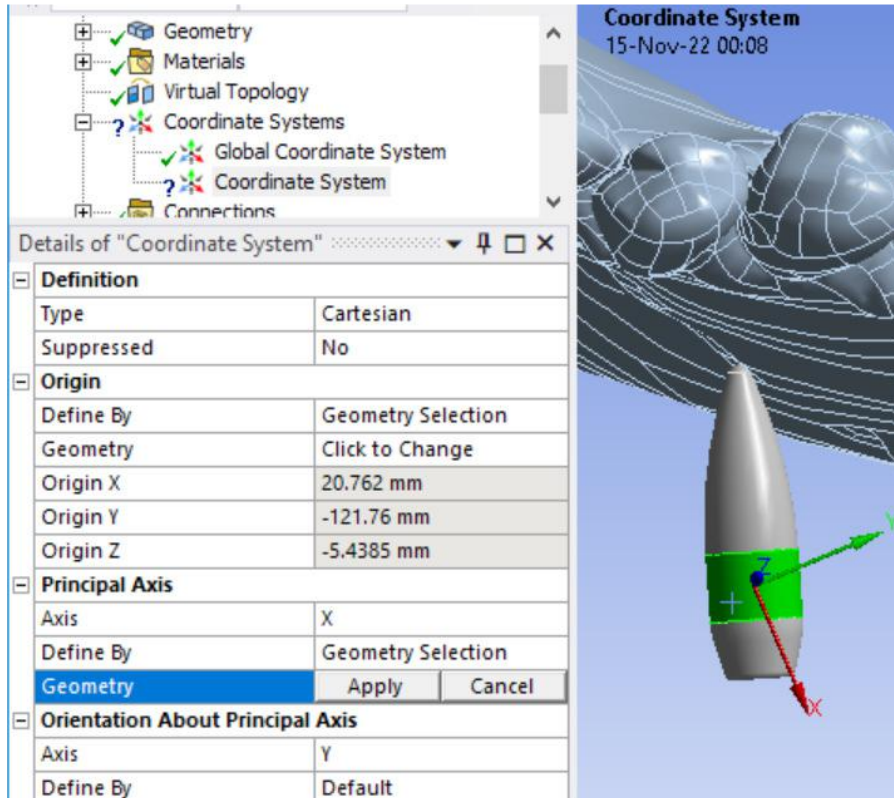
These are the parts properly named and the Bone Assignment on the *jaw*.



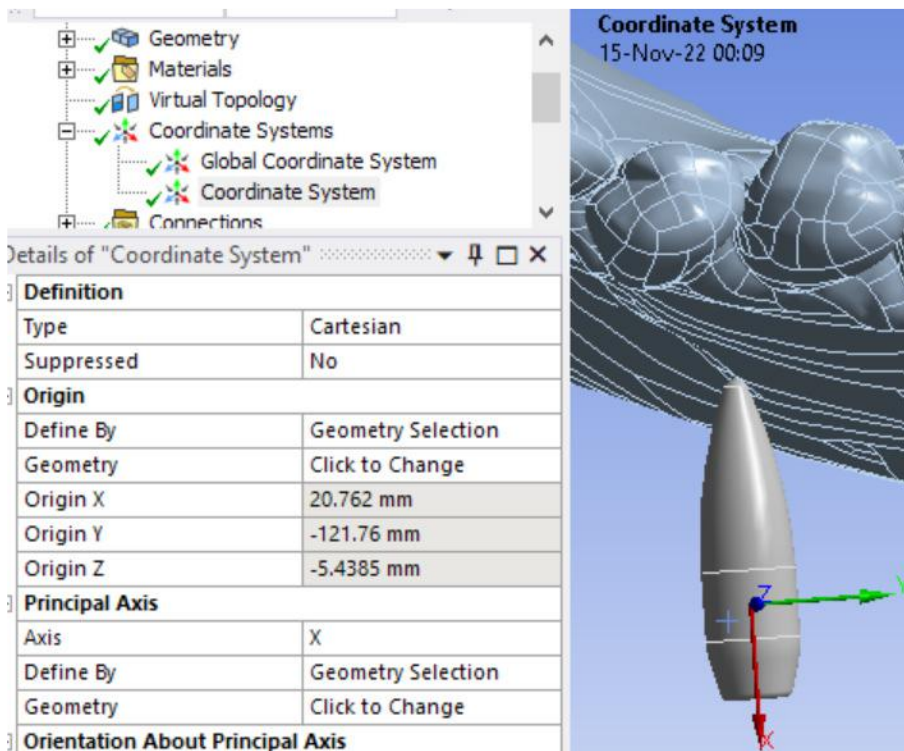
Connections: Because the collision between the bodies are managed by Body Interactions, we right click and Delete the contacts.



To ensure that the CS is aligned to the bullet, go to Principal Axis, Axis, X, Geometry, click the same green face, Apply.

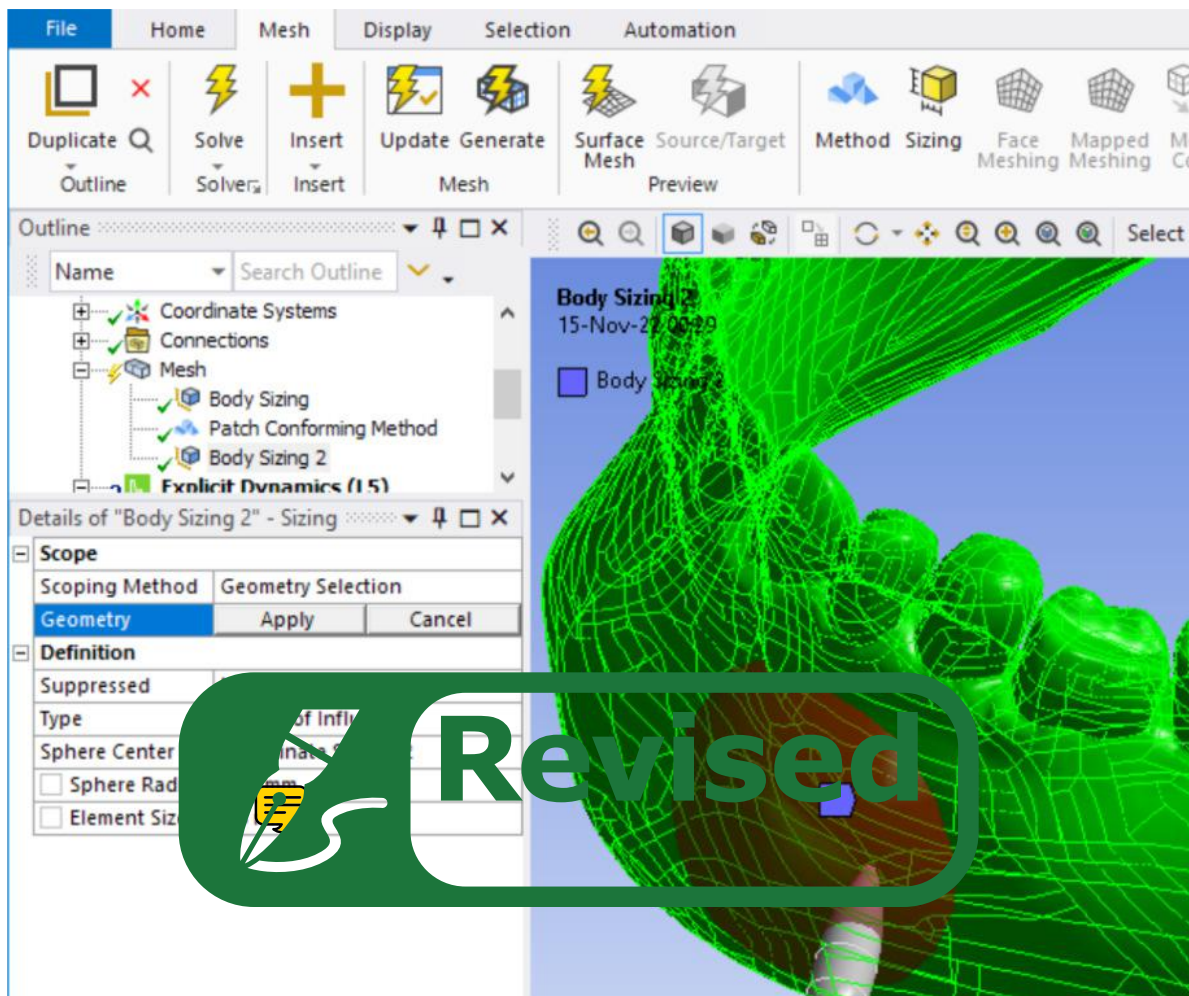


The CS is now properly aligned axially. The red X axis will indicate the velocity vector direction later.

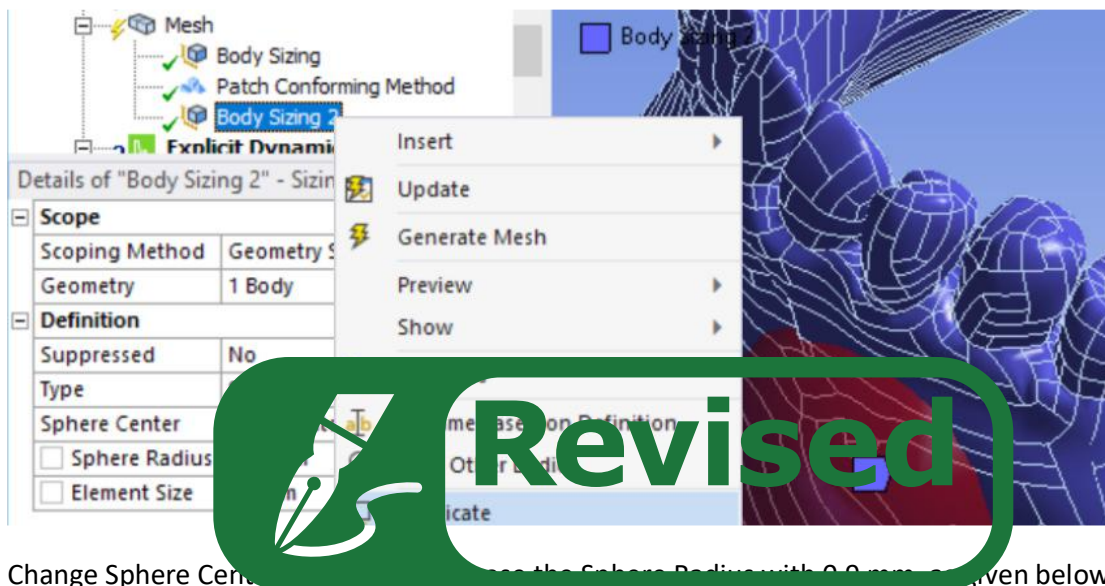




Now back to Mesh; insert a Sizing on the jaw, Apply then insert these details.



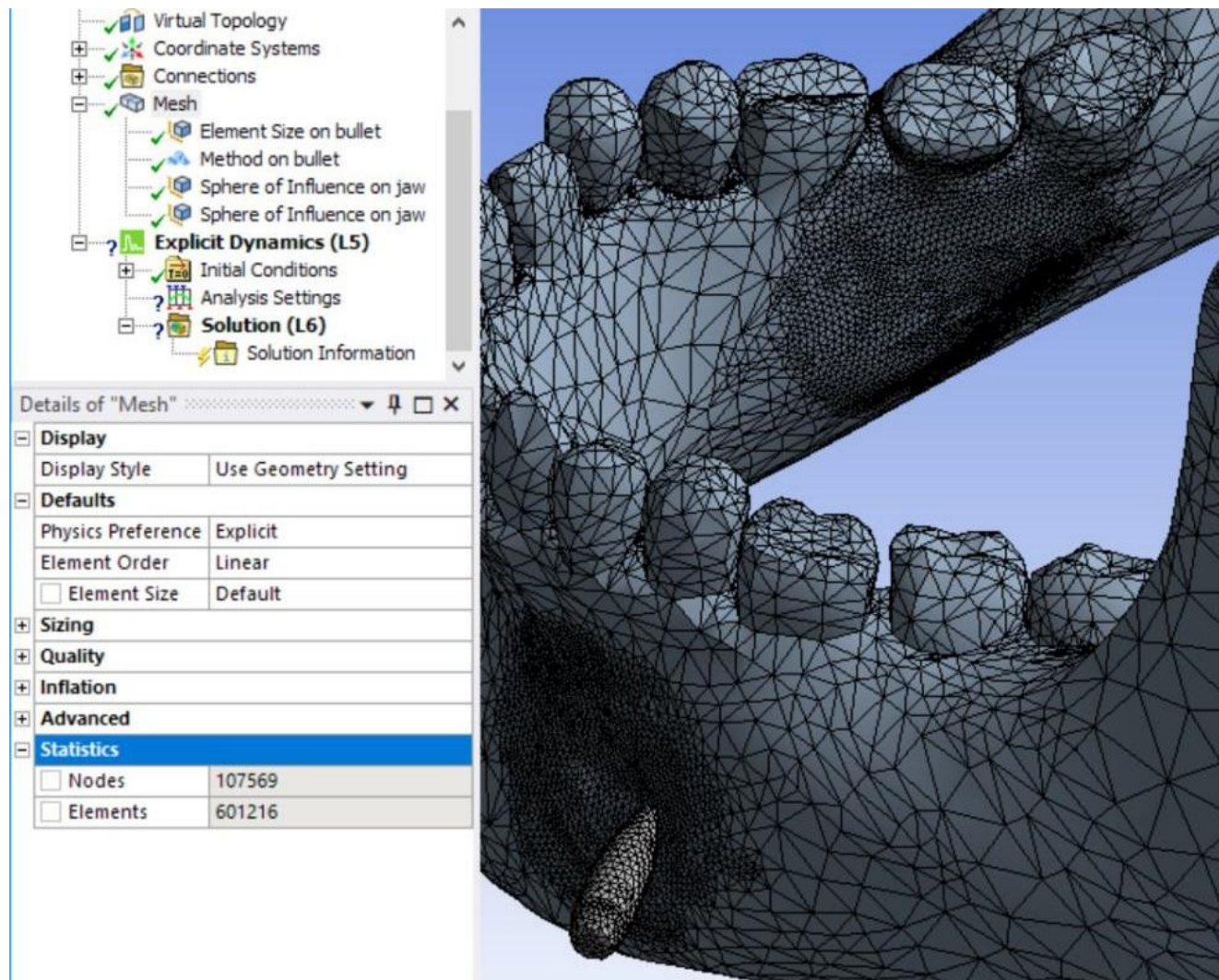
Right click this Body Sizing from the tree, Duplicate.



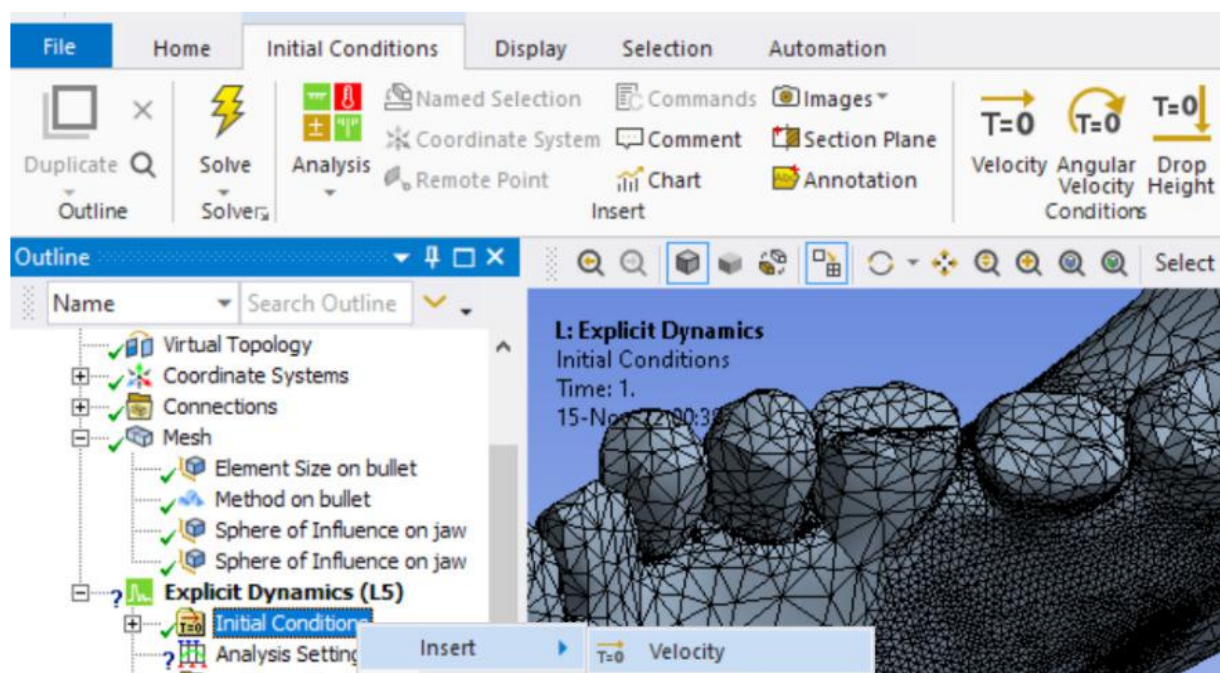
Change Sphere Center to the Jaw and the Sphere Radius with 0.0 mm as given below.



Properly made, the mesh should look like here.

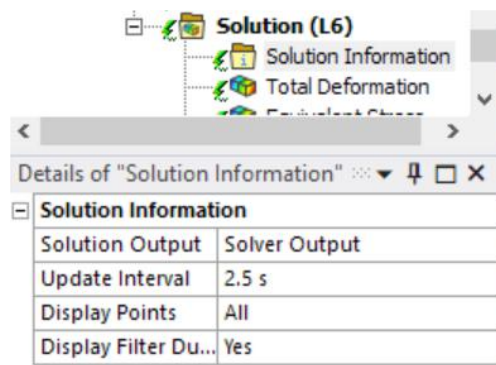


Explicit Dynamics: Right click Initial Conditions, Insert, Velocity.



But the FEA solves with this above value of 10%.

Solution Output, Solver Output shows such a real time log.



The screenshot shows the 'Solution Information' panel with the 'Solver Output' tab selected. The 'Update Interval' is set to 2.5 s, 'Display Points' is 'All', and 'Display Filter During Solve' is 'Yes'.

Solution Output	Solver Output
Update Interval	2.5 s
Display Points	All
Display Filter During Solve	Yes

Details of "Solution Information" ▾ ▴ □ ×

**Solution Information**

Cycle: 1463, Time: 2.214E-05s, Time Inc.: 1.296E-08s, Progress: 8.85%, Est. Clock Time Remaining: 22.0 mins

Cycle: 1464, Time: 2.215E-05s, Time Inc.: 1.296E-08s, Progress: 8.86%, Est. Clock Time Remaining: 22.0 mins

Cycle: 1465, Time: 2.216E-05s, Time Inc.: 1.296E-08s, Progress: 8.86%, Est. Clock Time Remaining: 22.0 mins

Cycle: 1466, Time: 2.218E-05s, Time Inc.: 1.296E-08s, Progress: 8.87%, Est. Clock Time Remaining: 22.0 mins

Cycle: 1467, Time: 2.219E-05s, Time Inc.: 1.296E-08s, Progress: 8.88%, Est. Clock Time Remaining: 22.0 mins

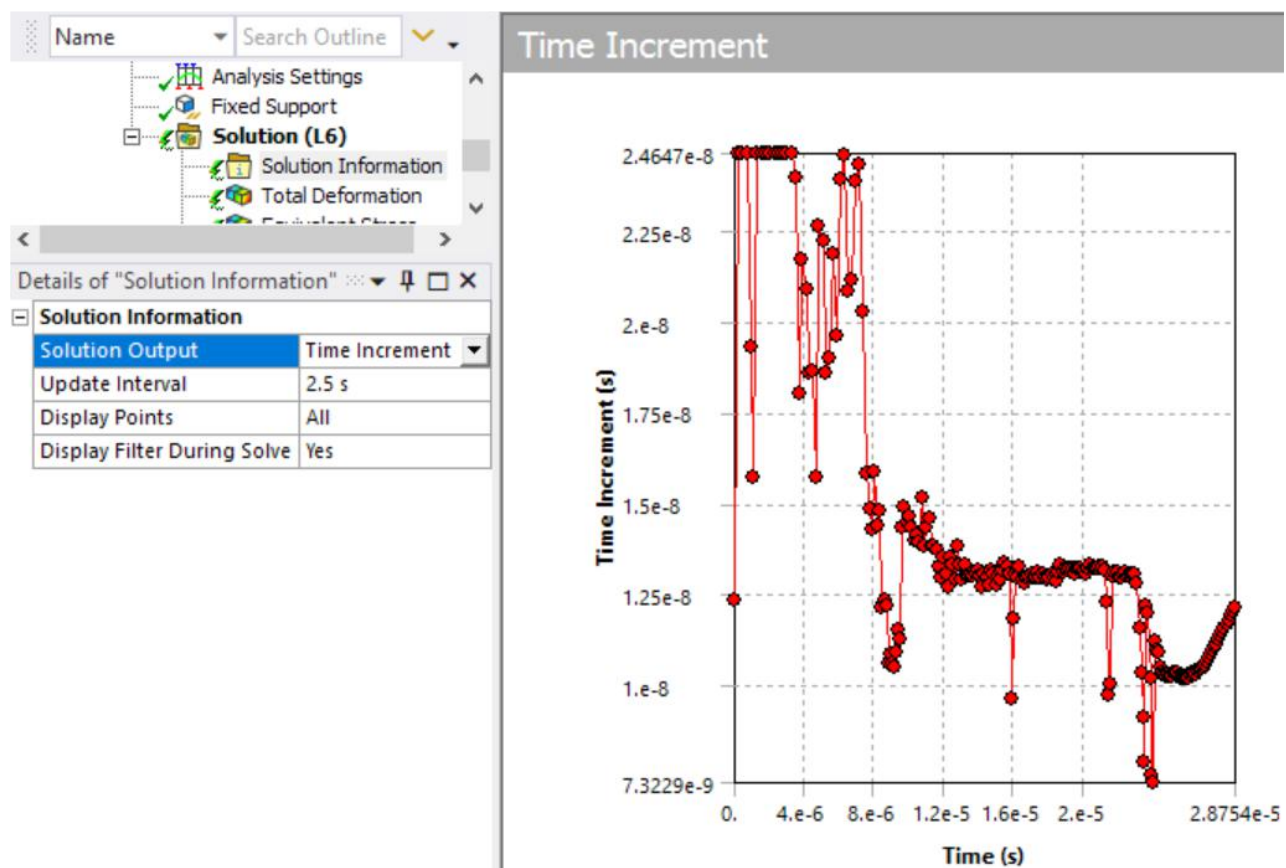
Cycle: 1468, Time: 2.220E-05s, Time Inc.: 1.296E-08s, Progress: 8.88%, Est. Clock Time Remaining: 22.0 mins

Cycle: 1469, Time: 2.221E-05s, Time Inc.: 1.297E-08s, Progress: 8.89%, Est. Clock Time Remaining: 22.0 mins

Cycle: 1470, Time: 2.223E-05s, Time Inc.: 1.297E-08s, Progress: 8.89%, Est. Clock Time Remaining: 22.0 mins

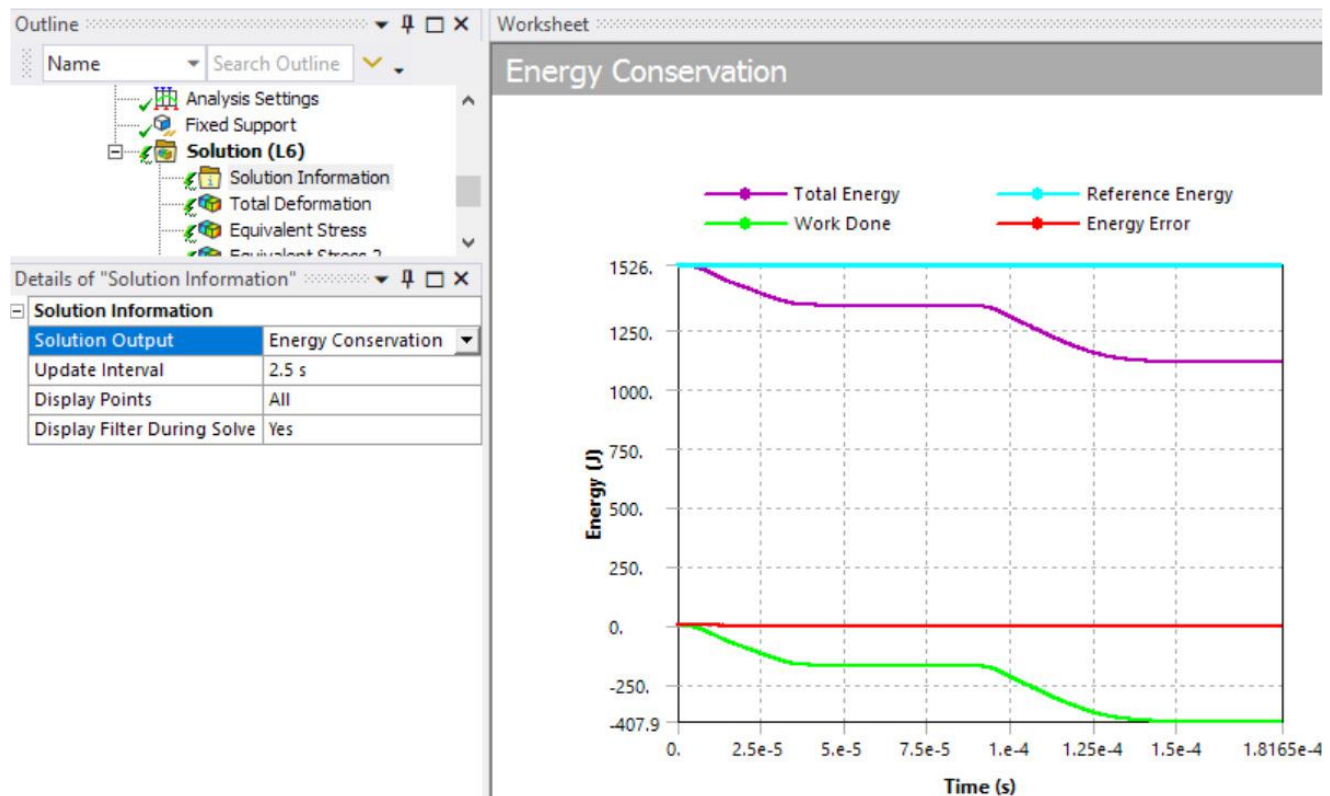
Cycle: 1471, Time: 2.224E-05s, Time Inc.: 1.298E-08s, Progress: 8.89%, Est. Clock Time Remaining: 22.0 mins

Making Solution Output, Time Increment we obtain a plot which signals where “hiccups” are encountered during solving (akin to bisections for Static or Transient, or implicit analyses), which require smaller time steps.

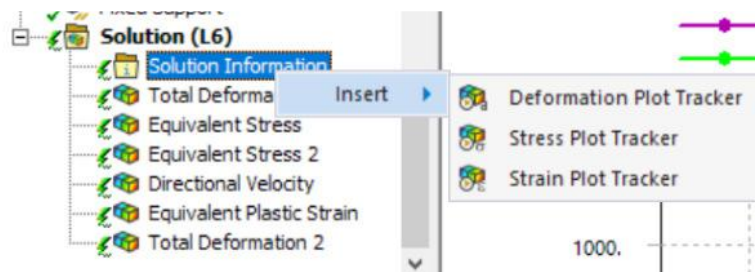




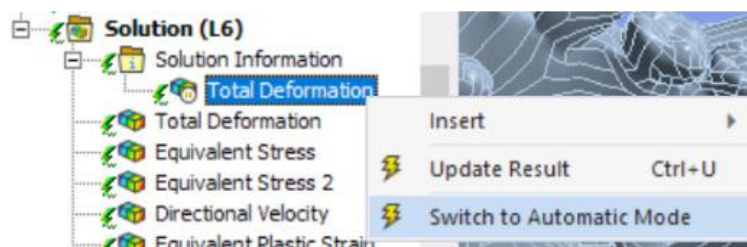
When changing from Time Increment to Energy Conservation we get such a plot in which we look for the Energy Error not to increase or have spikes, since we defined a 0.1 limit on it (review Maximum Energy Error from Analysis Settings).



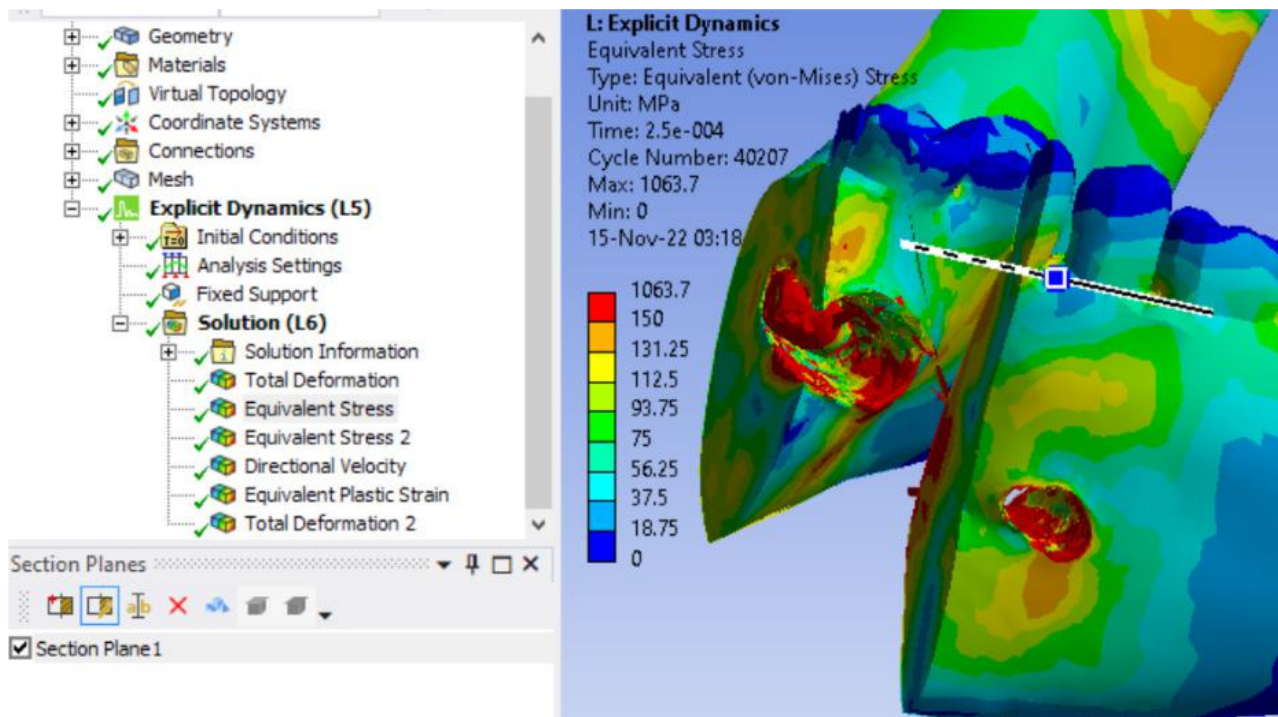
To see the behavior and trend of the analysis during solving, you can insert such intermediate result items when you right click the Solution Information branch.



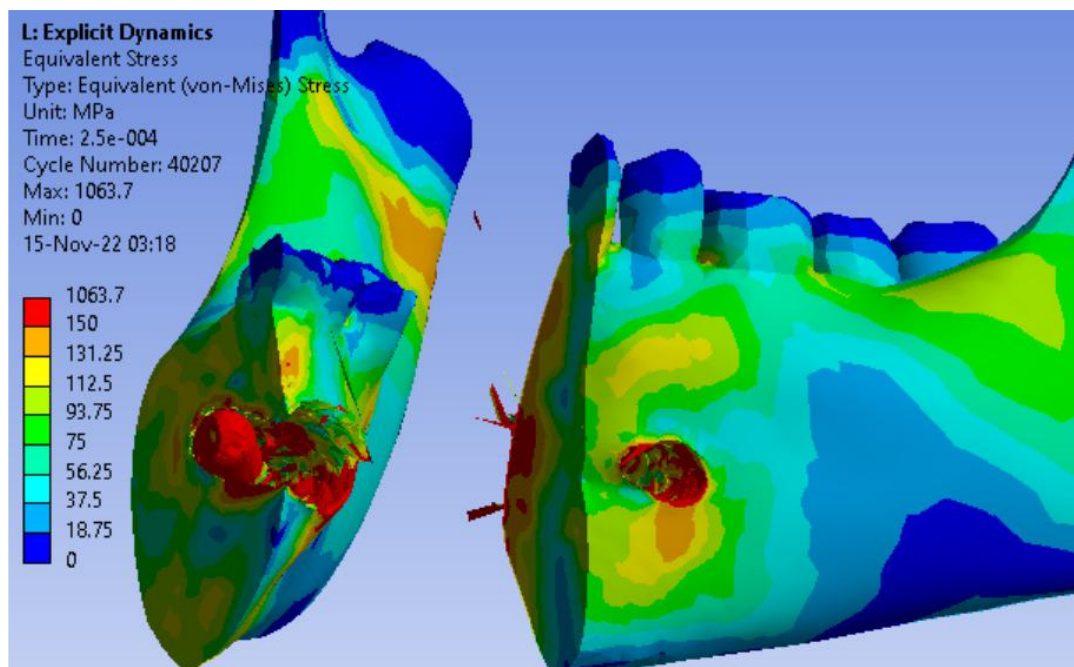
For live results, right click the needed item, Switch to Automatic Mode. Be careful when accessing too often these intermediate items because, occasionally, they may lead to the crash of the solving.



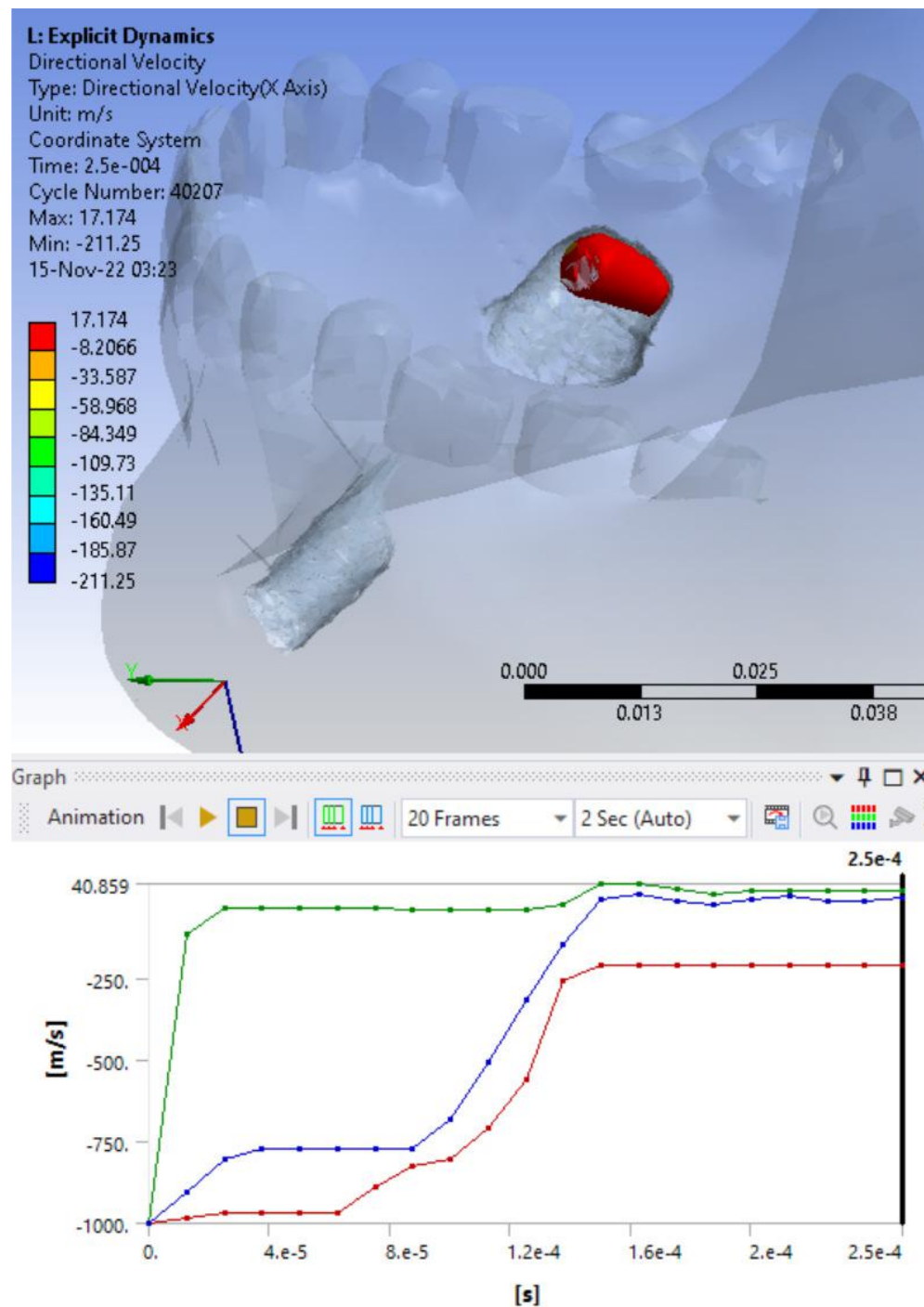




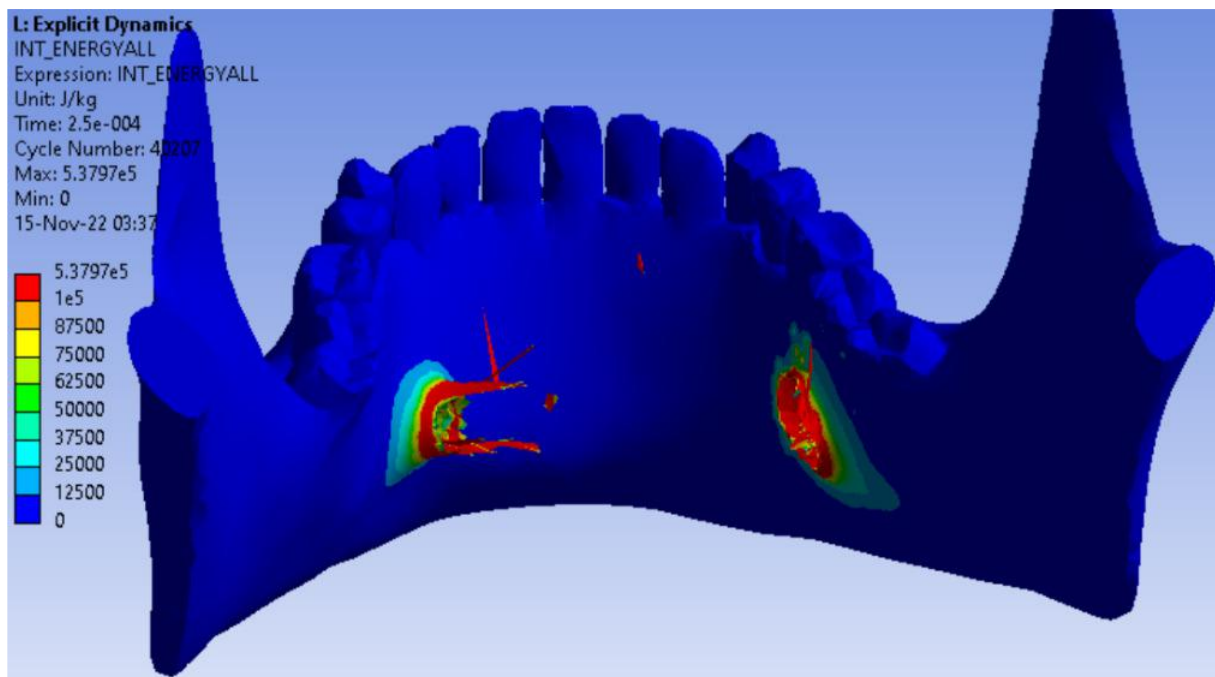
We now have a more revealing stress plot.



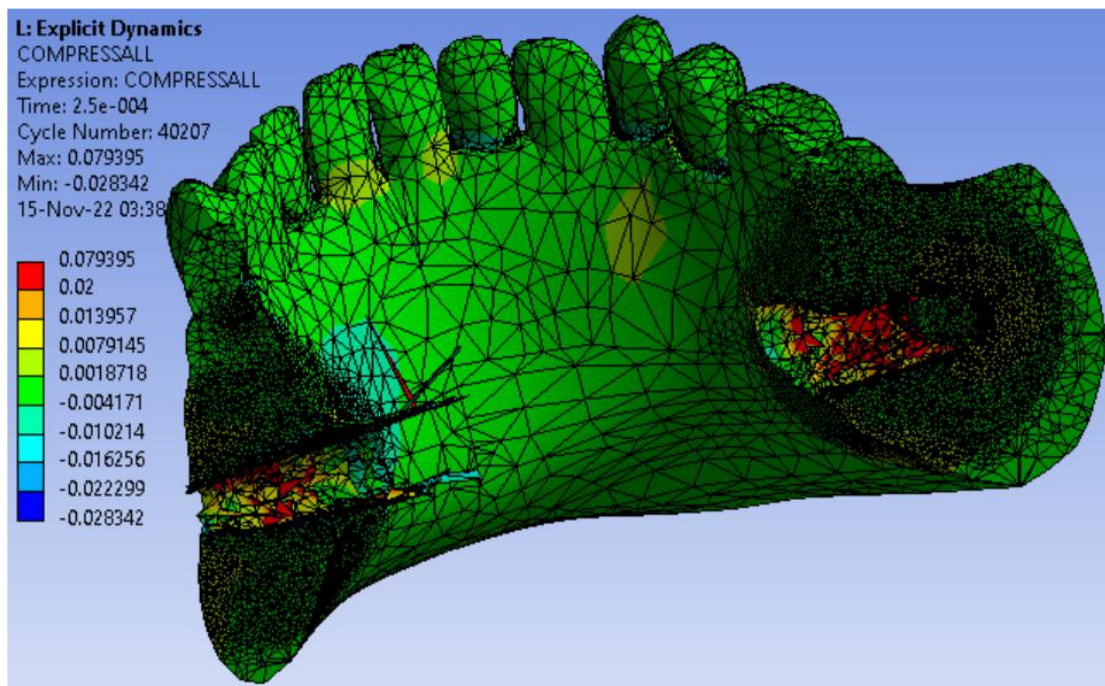
This is the Directional Velocity plot in m/s only on the bullet.



Here is the Internal Energy plot filtered with  $1e5$  J/kg to show the most affected regions.

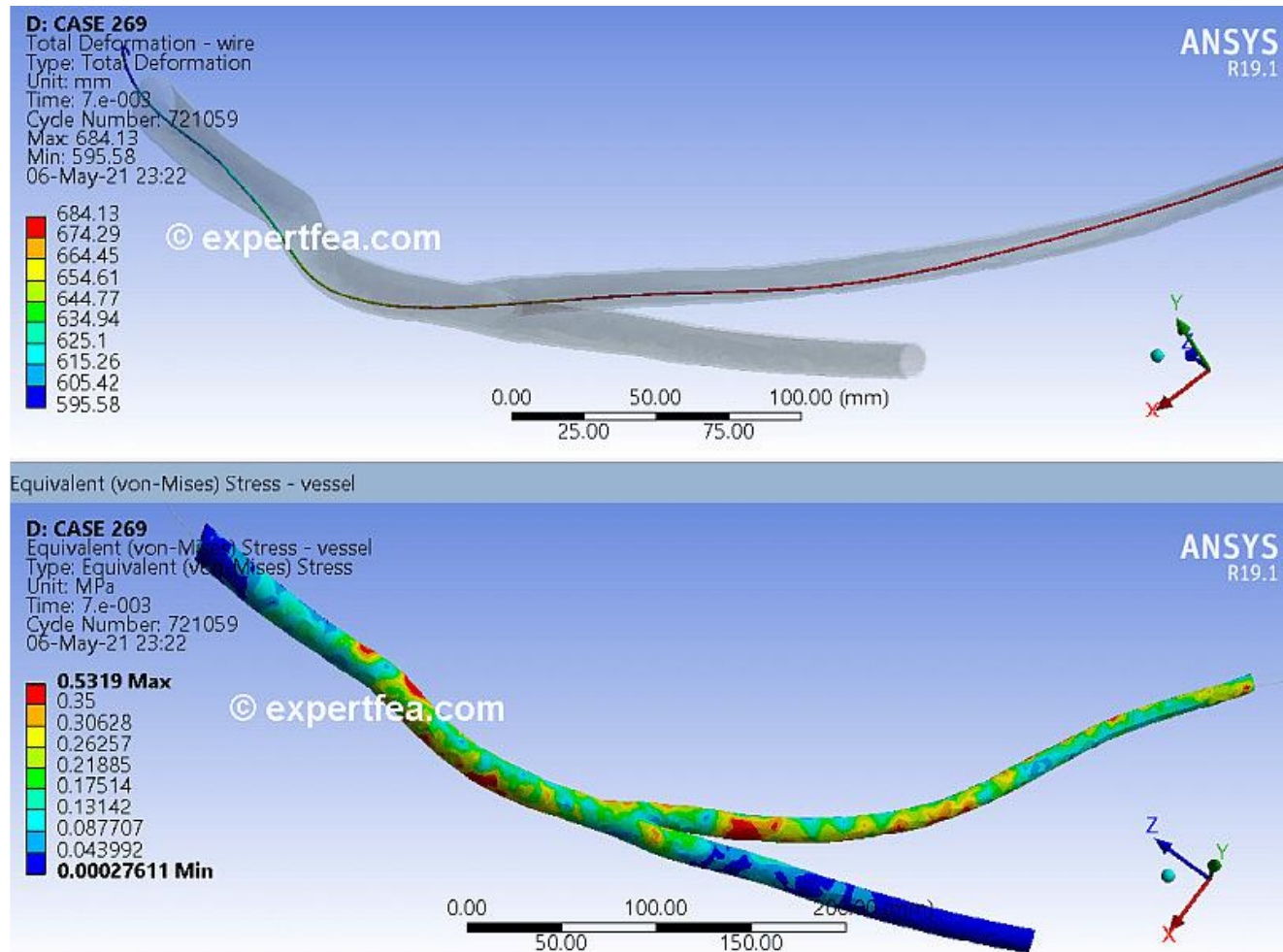


Here is the COMPRESSALL plot while also showing the mesh edges; we filtered with 0.02.



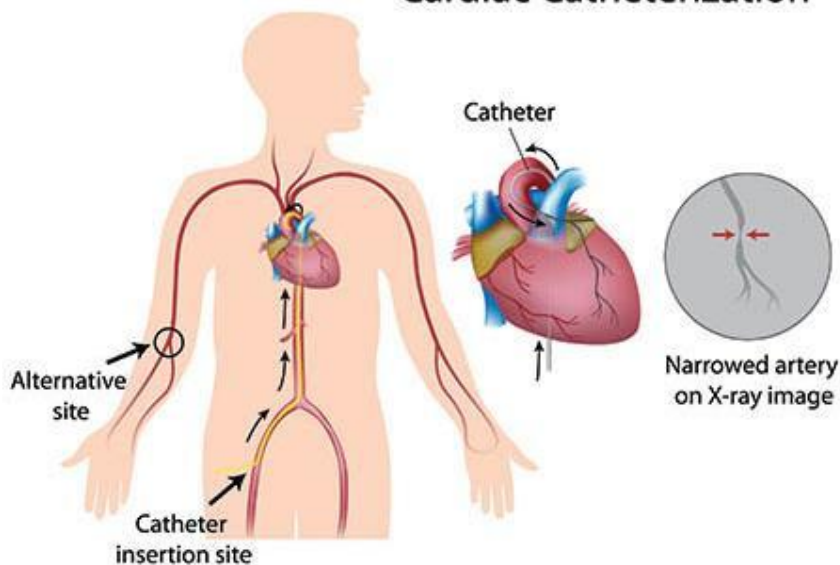


## CASE 269: Insertion simulation of catheter guide wire into a blood vessel - ANSYS Workbench Explicit Dynamics



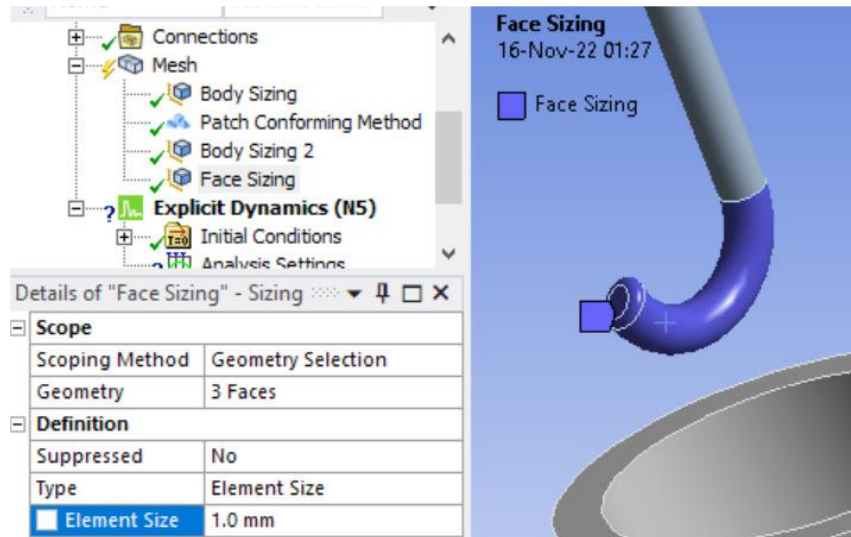
From hearthousenj.com we show an overview of one of the most common catheterization procedures.

### Cardiac Catheterization

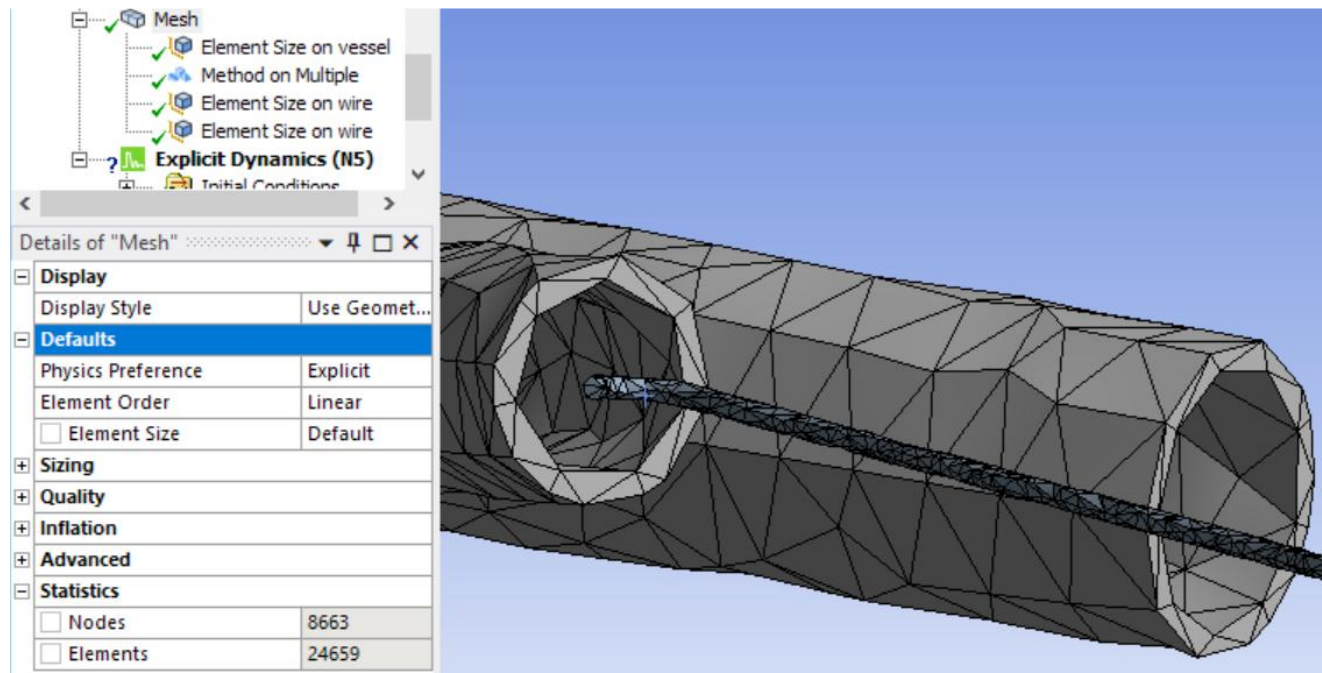


In our geometry, we curled the wire at one end to avoid puncturing the blood vessel.

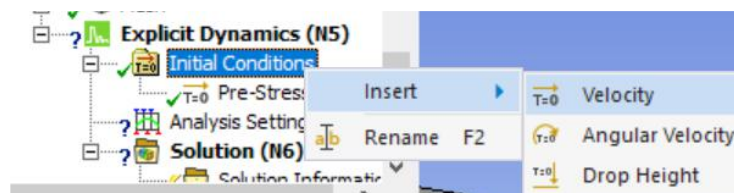
Ctrl+F to select only faces. Right click Mesh, Insert, Sizing then select the 3 faces from the curled end of the wire, where it touches the blood vessel.



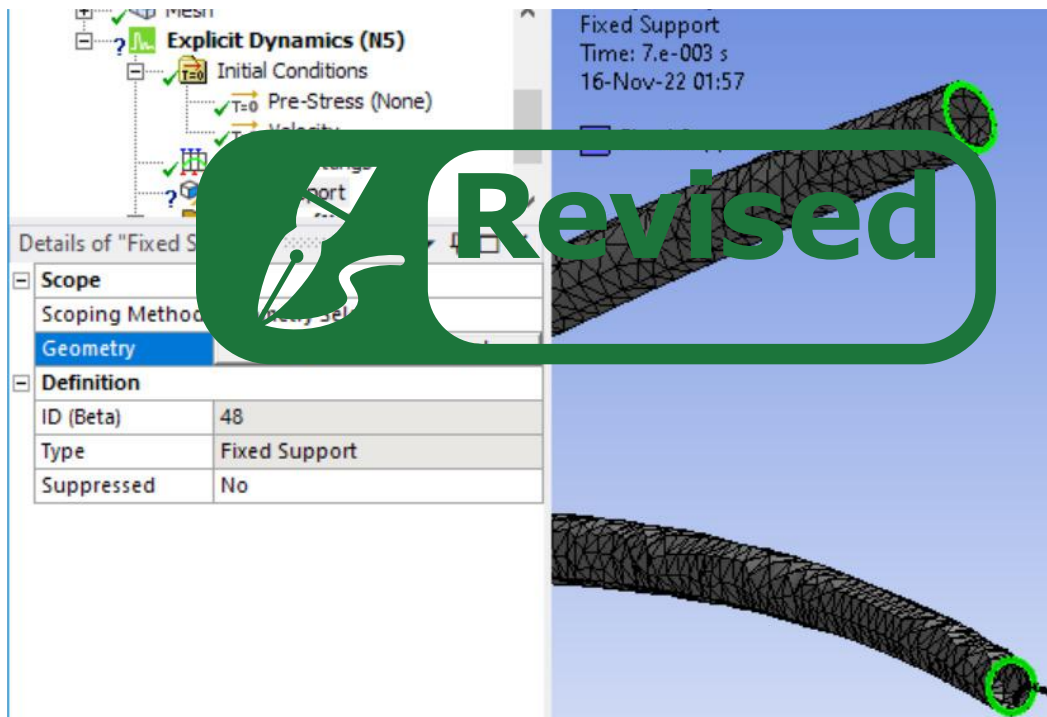
Right click all items under mesh, Rename Based on Definition. Generate Mesh. The mesh looks pretty coarse, since we have to define a long travel for the wire, and we don't want that the solving to take too much to finish.



Explicit Dynamics: Right click Initial Conditions, Insert, Velocity.



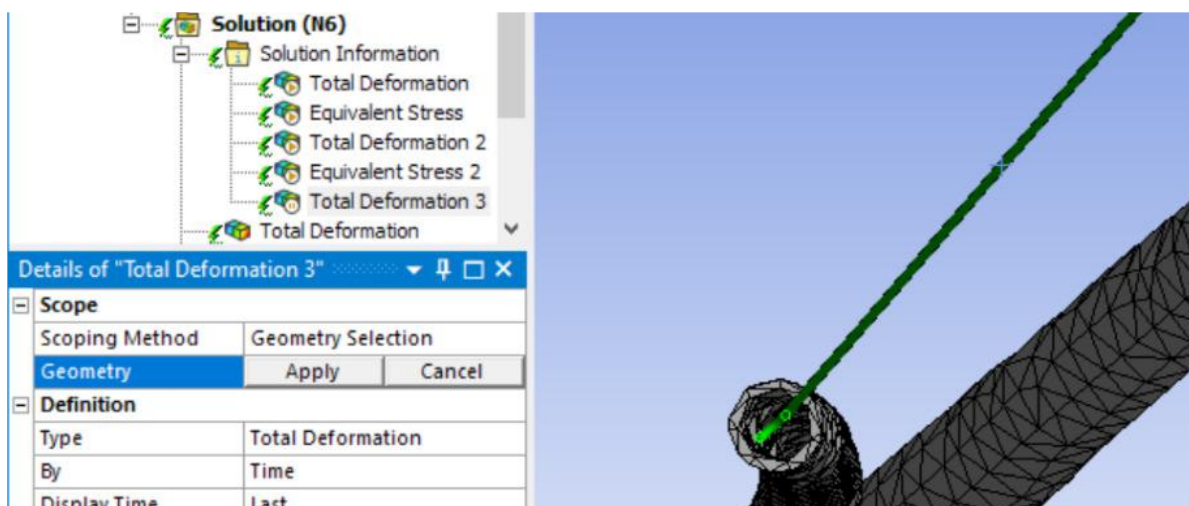
Right click anywhere, Insert, Fixed Support. Select all 3 ends of the blood vessel, Apply.



Solution: Apply these items, on the respective parts, via Rename Based on Definition. We scoped for distinct parts in some of the solution plots, since the material differences can confuse the plots, when shown both parts at the same time. Save, Solve.

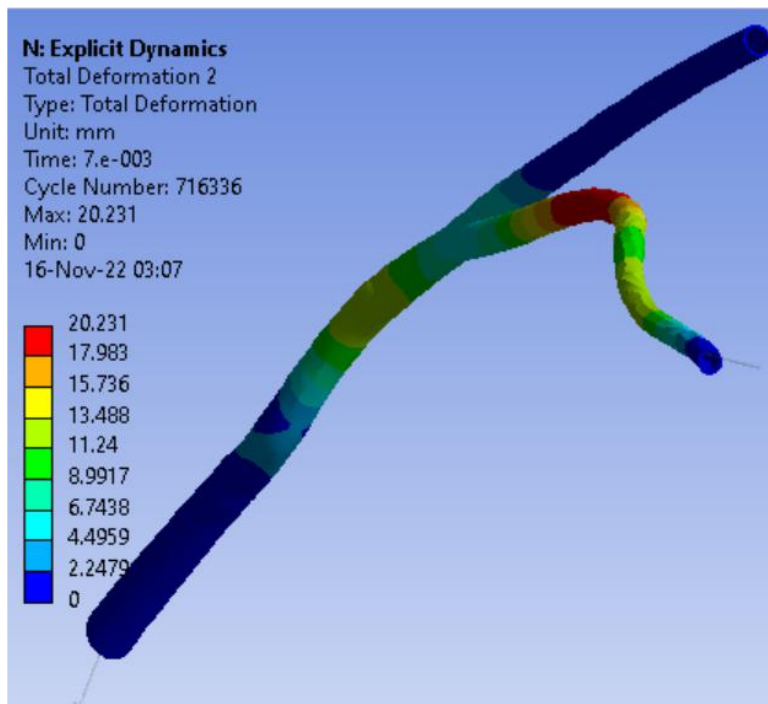


To see the evolution of the simulation, insert Solution Information then deformation, stress or strain items. We scoped here only the wire, because we need it shown only by itself, behind a transparent (unselected) blood vessel.

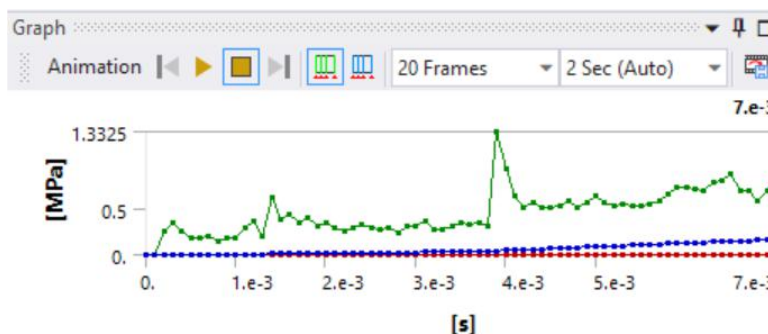
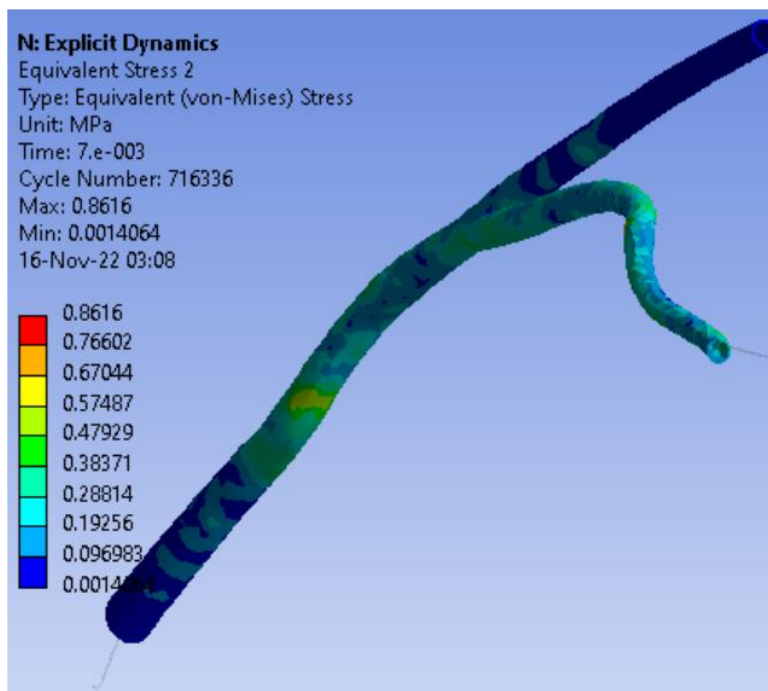




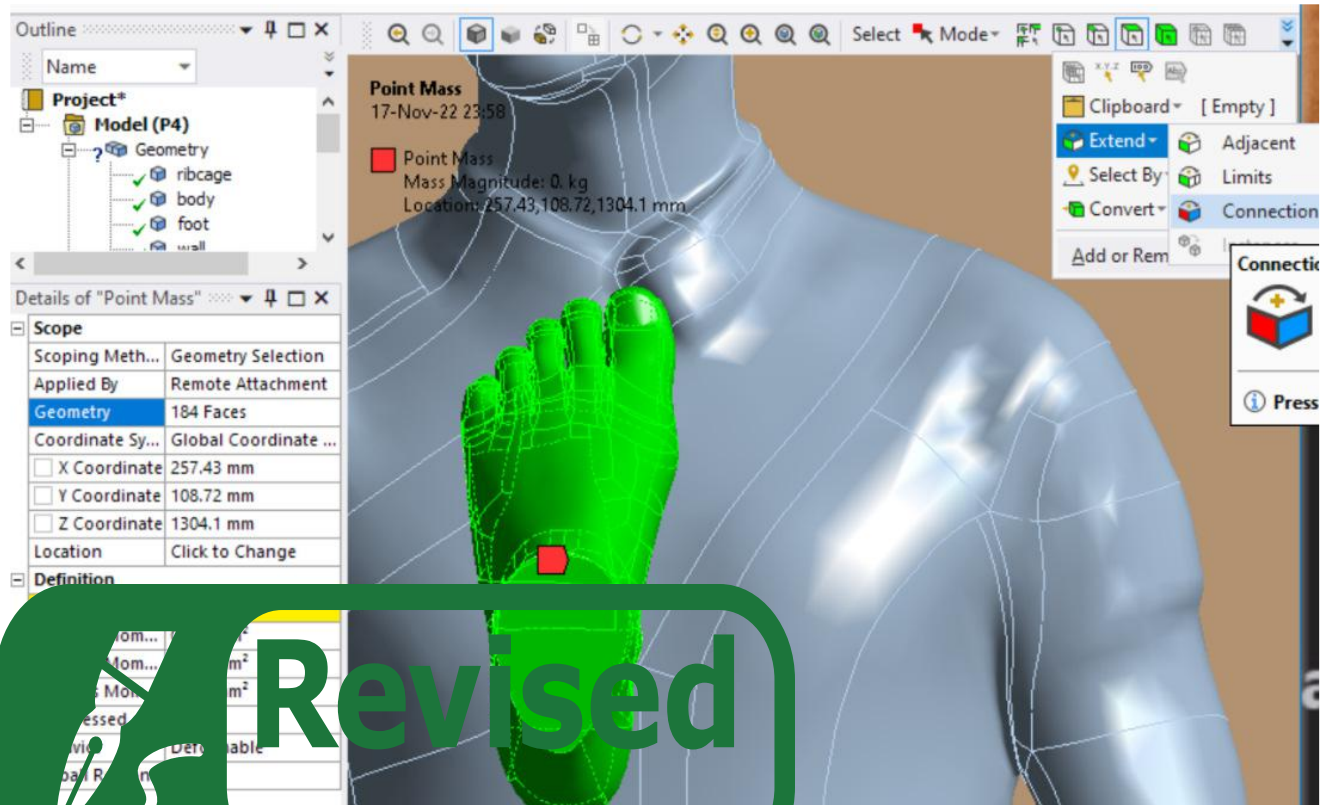
This is the Total Deformation of the blood vessel at the end.



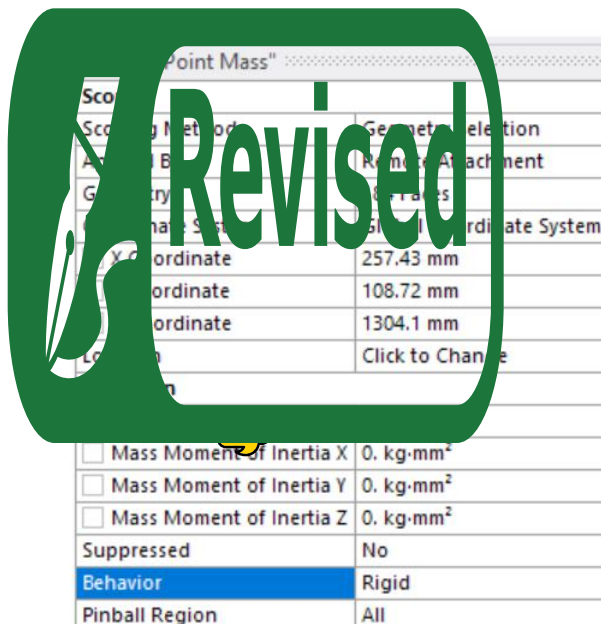
Here is shown the Equivalent Stress in the blood vessel. On the graph, the peak is shown at around 4e-3 s.



Ctrl+F to select only faces, then click the *foot*, go to the top toolbar and click Extend, Connection, then into the Details click Geometry, Apply.



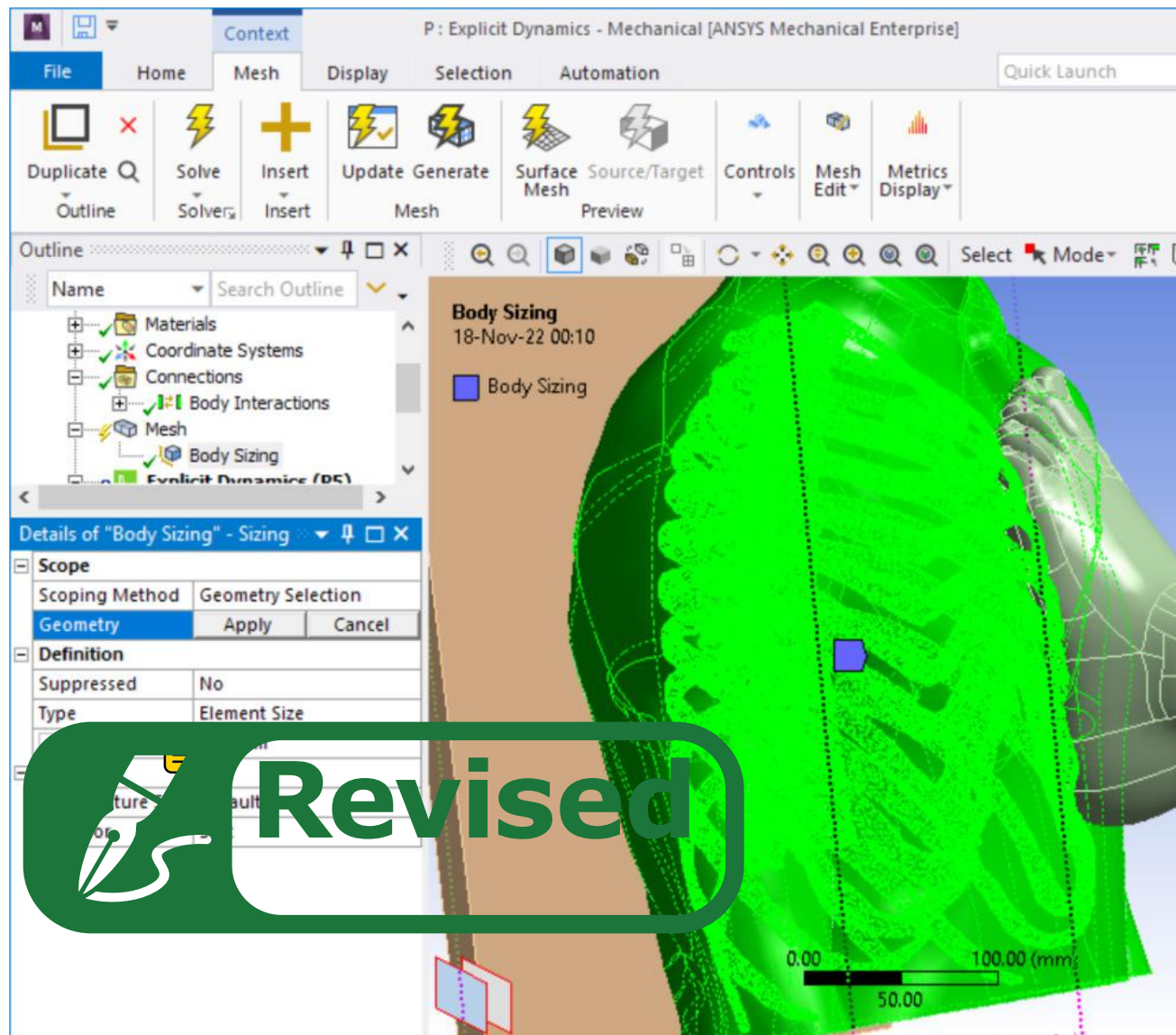
with Mass = 25 Kg, Behavior: Rigid



Because we need a similar sizing on the interface between the ribcage and the body (the ribcage being embedded into a cutout in the body, hence the touching elements should mate properly). Insert, Sizing, 20 mm, Apply.

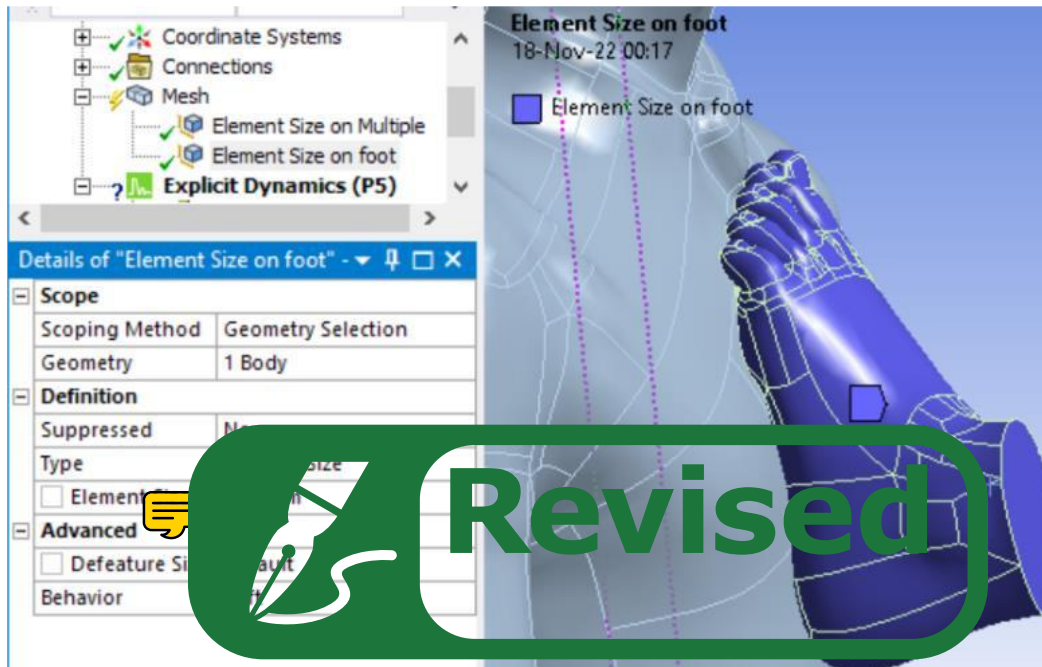
Ctrl+B to select only bodies, click in the middle of the body, it should be colored in green, then if 2 parallel squares appear on the bottom left of the screen, select with Ctrl pressed the one that is not red, to also add the other body to the selection, Geometry, Apply. When one body is overlapping with other bodies, clicking on an area which corresponds to more bodies will always yield parallel planes on the bottom left of the screen to allow for multiple selection by clicking the planes with Ctrl pressed (it works for adding and also for removing parts from the selection).

Another way to select multiple neighboring bodies is to change the selection filter to Box, then dragging the mouse cursor from the right to the left over those bodies, which will encompass them all.

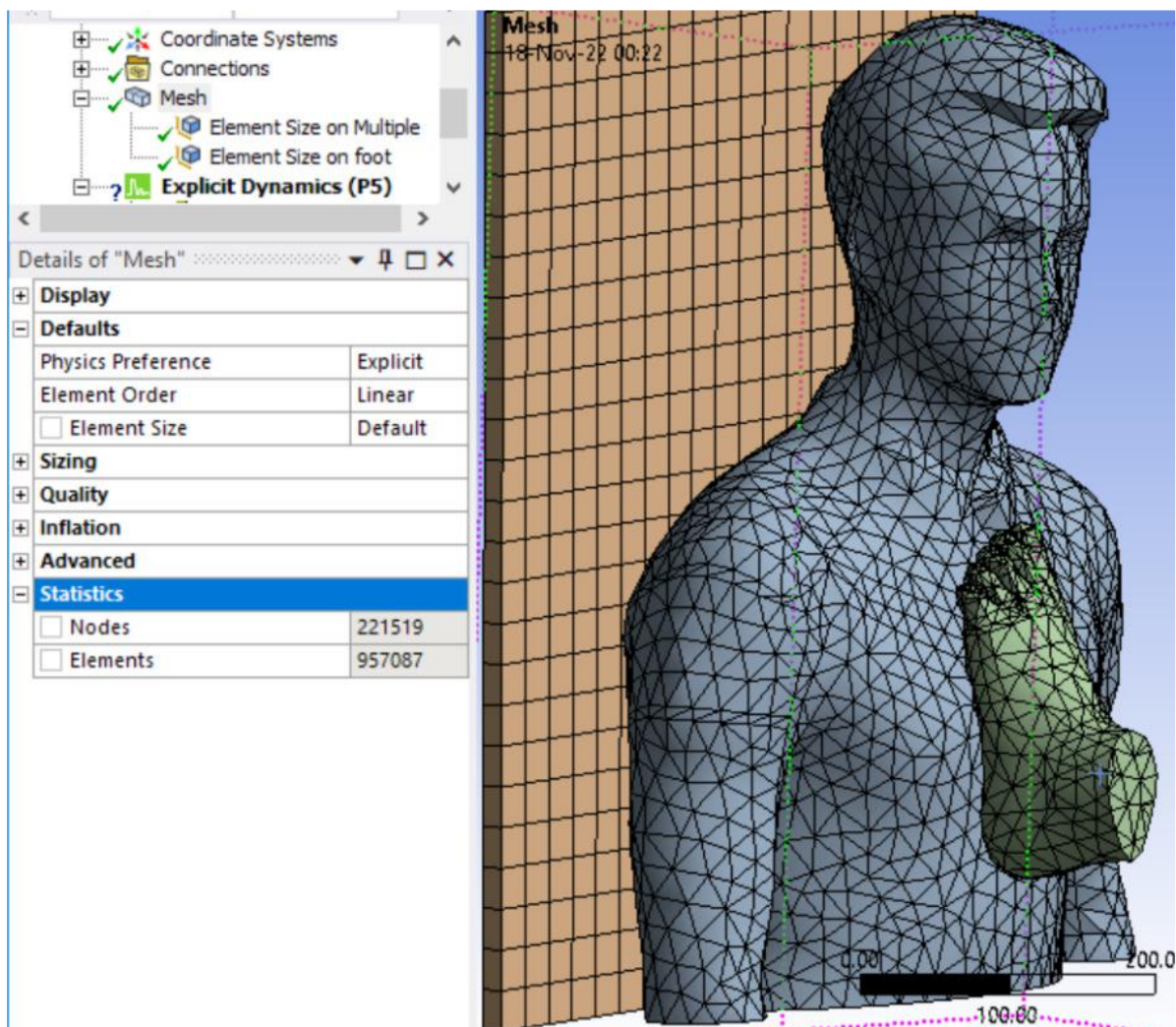




Repeat the sizing, but only for the foot now. We will leave the wall with a default mesh. Rename Based on Definition, Generate Mesh.



The mesh should look like here with the settings mentioned above.



We encountered this error, so let us investigate it.

```

Worksheet
Solver Output

Cycle:      8640, Time:  2.627E-03s, Time Inc.: 1.306E-09s, Progress: 26.27%, Est. Clock Time Remaining: 1.6 hrs
Cycle:      8641, Time:  2.627E-03s, Time Inc.: 1.278E-09s, Progress: 26.27%, Est. Clock Time Remaining: 1.6 hrs
Cycle:      8642, Time:  2.627E-03s, Time Inc.: 1.251E-09s, Progress: 26.27%, Est. Clock Time Remaining: 1.6 hrs
Cycle:      8643, Time:  2.627E-03s, Time Inc.: 1.225E-09s, Progress: 26.27%, Est. Clock Time Remaining: 1.6 hrs
Cycle:      8644, Time:  2.627E-03s, Time Inc.: 1.199E-09s, Progress: 26.27%, Est. Clock Time Remaining: 1.6 hrs
Cycle:      8645, Time:  2.627E-03s, Time Inc.: 1.173E-09s, Progress: 26.27%, Est. Clock Time Remaining: 1.6 hrs
Cycle:      8646, Time:  2.627E-03s, Time Inc.: 1.148E-09s, Progress: 26.27%, Est. Clock Time Remaining: 1.6 hrs
Cycle:      8647, Time:  2.627E-03s, Time Inc.: 1.124E-09s, Progress: 26.27%, Est. Clock Time Remaining: 1.6 hrs
Cycle:      8648, Time:  2.627E-03s, Time Inc.: 1.100E-09s, Progress: 26.27%, Est. Clock Time Remaining: 1.6 hrs
Cycle:      8649, Time:  2.627E-03s, Time Inc.: 1.076E-09s, Progress: 26.27%, Est. Clock Time Remaining: 1.6 hrs
Cycle:      8650, Time:  2.627E-03s, Time Inc.: 1.053E-09s, Progress: 26.27%, Est. Clock Time Remaining: 1.6 hrs
Cycle:      8651, Time:  2.627E-03s, Time Inc.: 1.031E-09s, Progress: 26.27%, Est. Clock Time Remaining: 1.6 hrs
Cycle:      8652, Time:  2.627E-03s, Time Inc.: 1.009E-09s, Progress: 26.27%, Est. Clock Time Remaining: 1.6 hrs
Wrapup timestep controlled by unstructured element: 930859 DT= 9.8710E-07
Cycle:      8653, Time:  2.627E-03s, Time Inc.: 9.871E-10s, Progress: 26.27%, Est. Clock Time Remaining: 1.6 hrs

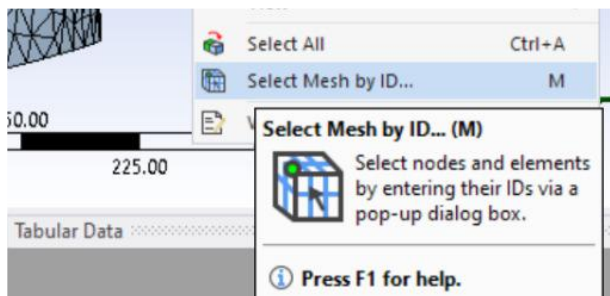
SIMULATION ELAPSED TIME SUMMARY

EXECUTION FROM CYCLE      0 TO      8653
ELAPSED RUN TIME IN SOLVER =      2.86056E+01 Minutes
TOTAL ELAPSED RUN TIME   =      3.41060E+01 Minutes
JOB RAN OVER      8 WORKERS
JOB RAN USING Intel MPI
JOB RAN USING DECOMPOSITION AUTO

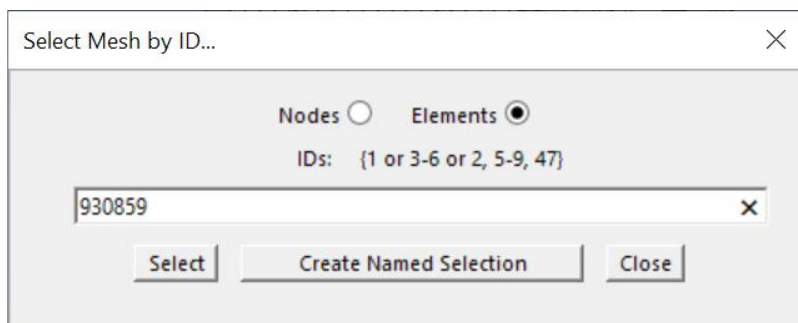
Problem terminated .... time step too small

```

We see that the element named 930859 behaved abnormally. Right click anywhere in the screen, Select Mesh by ID...

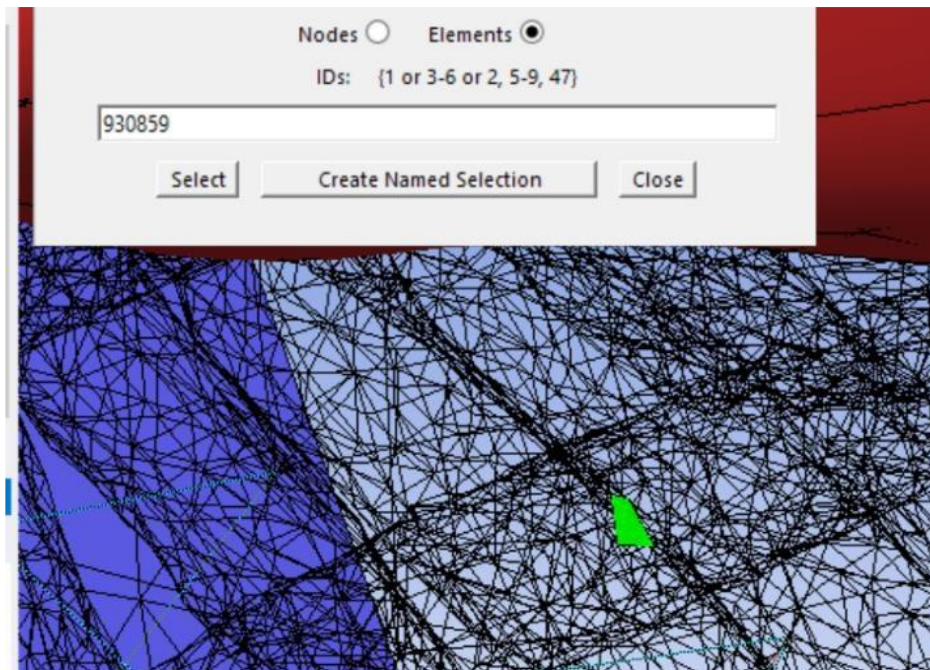


You can click Select to temporarily look at the Element at fault or Create Named Selection to have it saved separately.

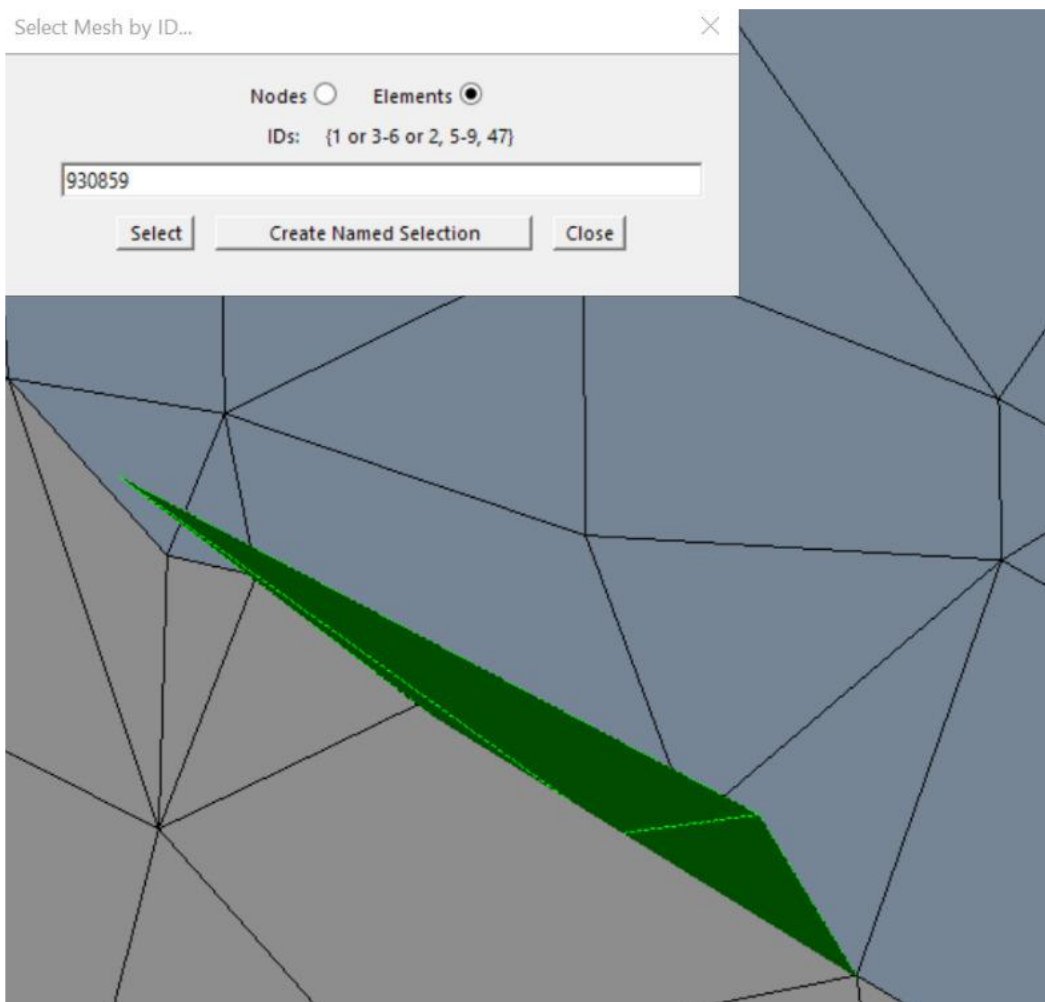




Let us click Select then rotate the assy until we get a better view of that element.

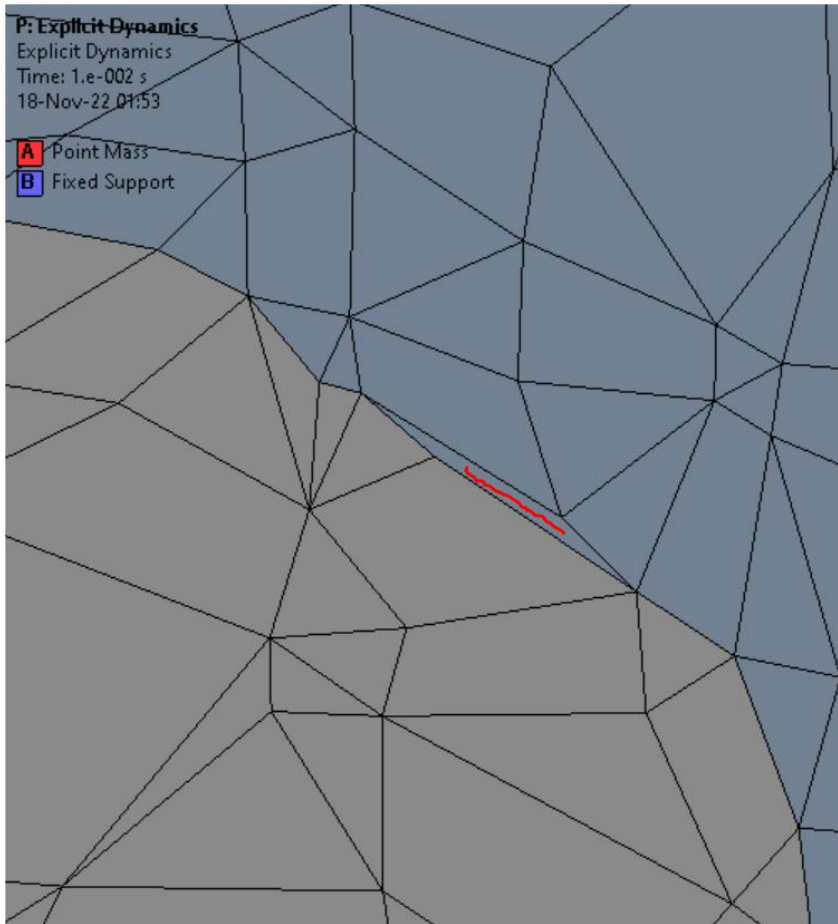


Create a Section Plane through element 930859. It seems to belong to the body, not the ribcage, but we are not 100% sure, since it's located at their interface.

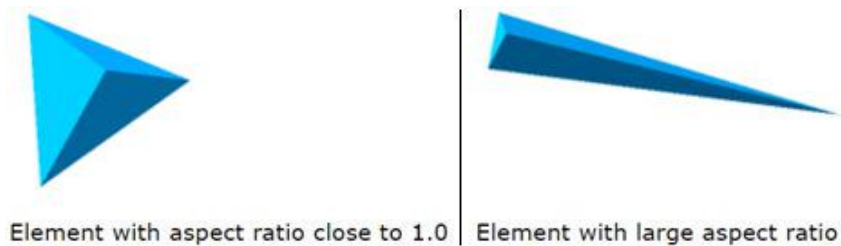




If we click somewhere else to get rid of the element selection, we can see that it pertains to the body part; we marked it with a red line.



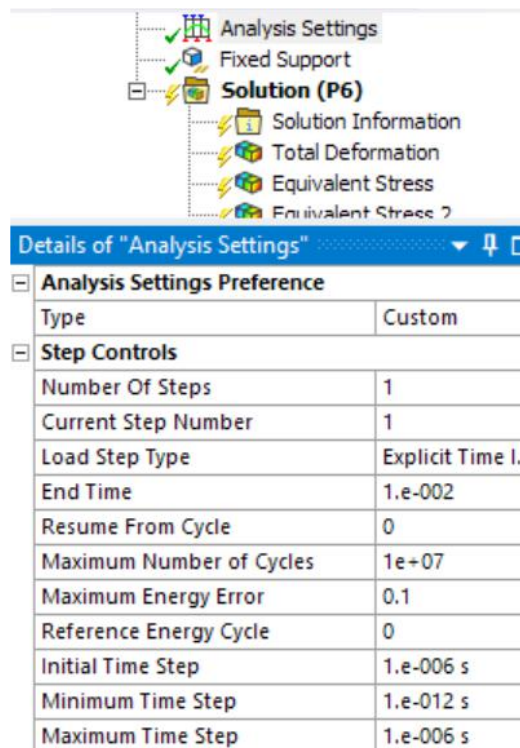
Judging from the excessively elongated shape, its Aspect Ratio is extremely high, which is undesirable since we need it closer to 1, towards an ideal tetrahedron.



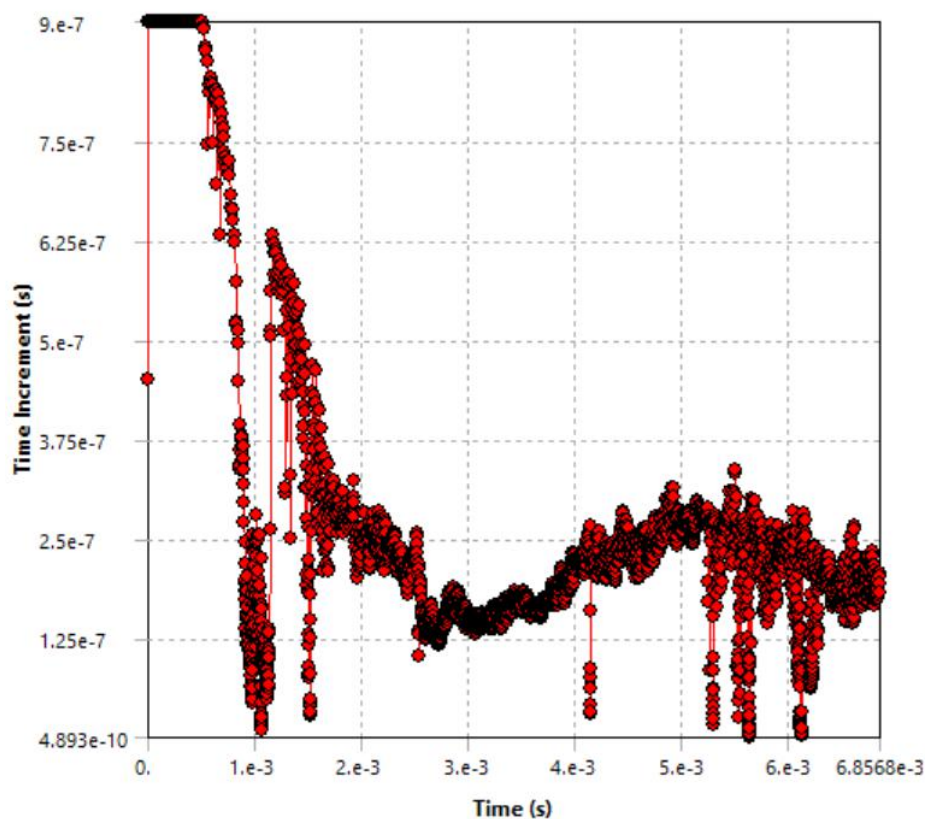
This leads to the conclusion that the mesh needs to be refined in the body, but because it has the same Mesh Sizing as the ribcage, both parts need remeshed accordingly. But we already have a fine mesh for an Explicit analysis, with more than 200.000 nodes, so let us refrain for the moment from further refinement.

We digressed a little to show additional options, as seen above.

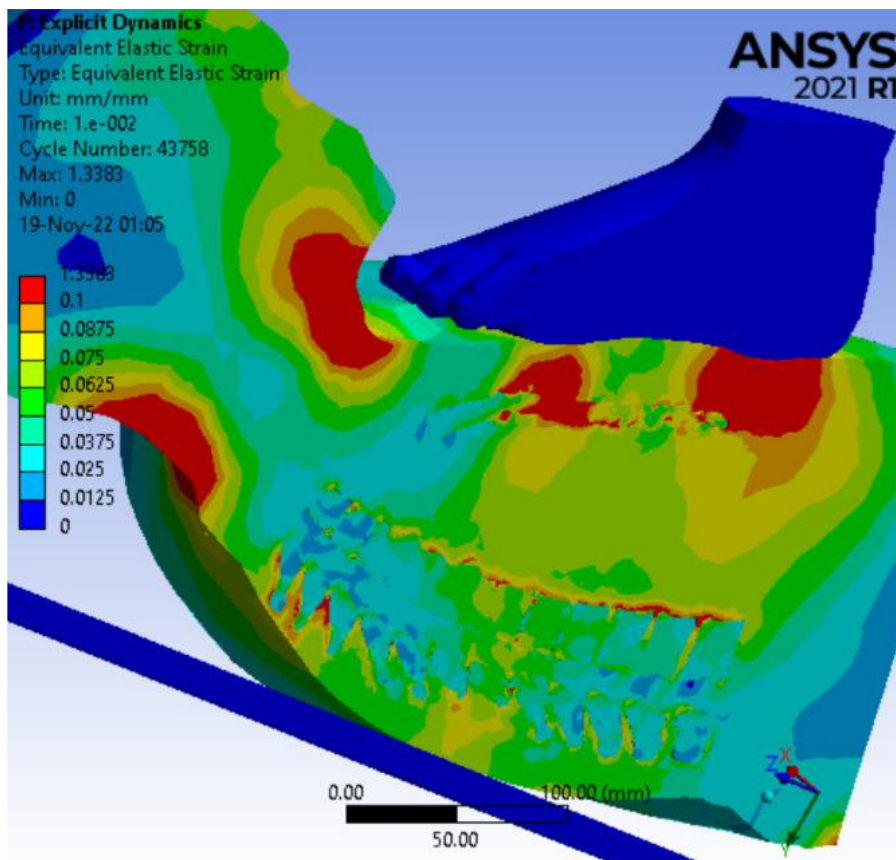
Go to Analysis Settings, make Minimum Time Increment =  $1\text{e-}12$  s and resolve the FEA. We could have chosen to Resume From Cycle 9026 (the last one seen above), but the FEA did not continue when we tested, so we sometimes need to start from the beginning, especially simulations with high nodes count.



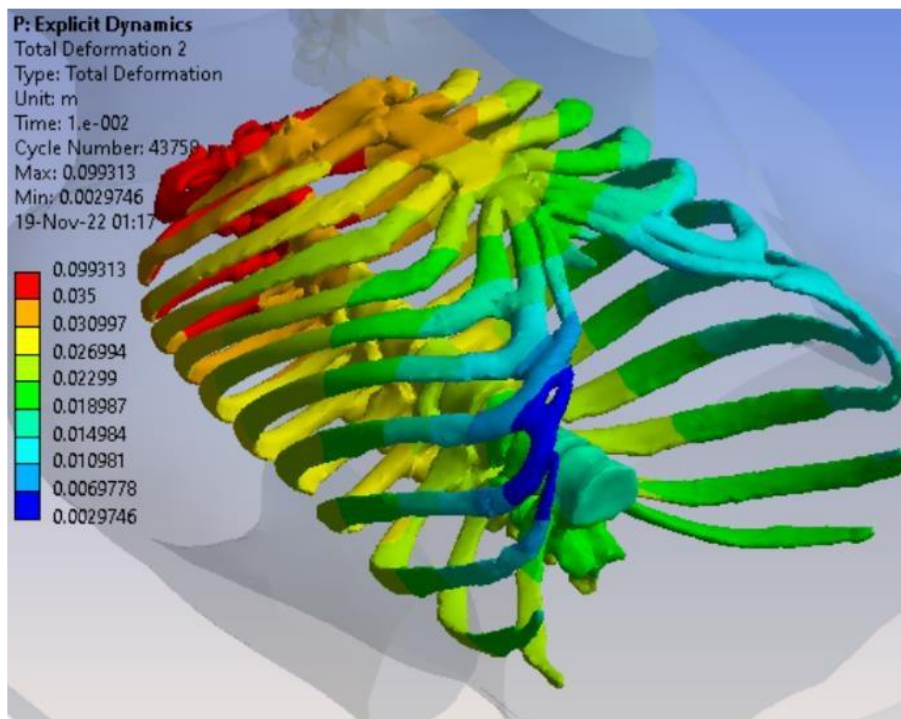
After around 1.5 hours we are at 70% of the solving and, after some dives on the time increment, the FEA works fine.



Here is the Equivalent Strain in the assy, filtered with 0.1 mm/mm to see where the “hotspots” occur.

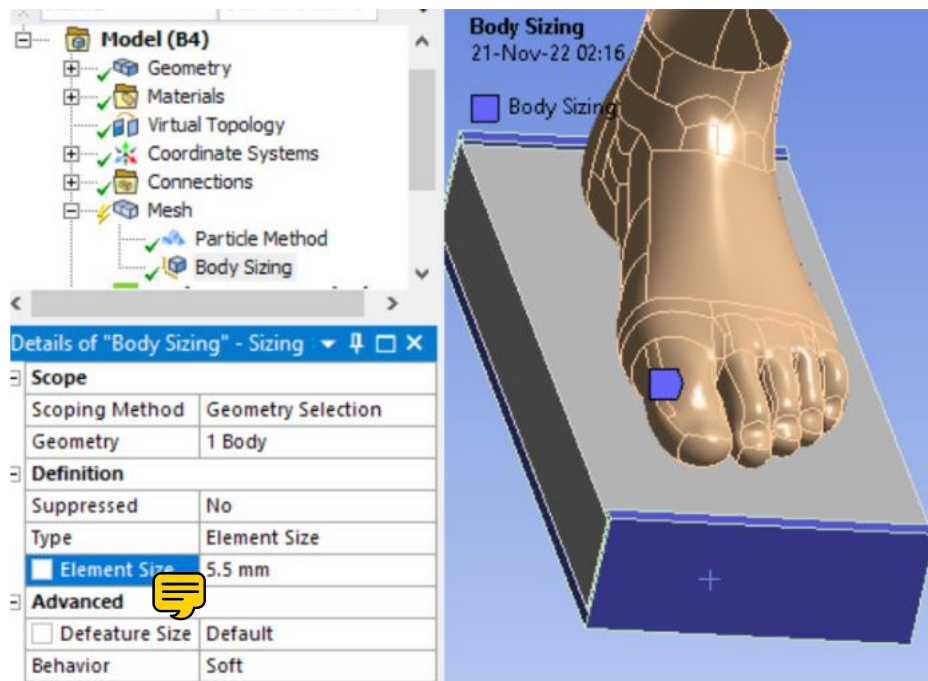


This is the Equivalent Strain plot only in the ribcage, filtered with 0.035 m to easily see the maximum values.

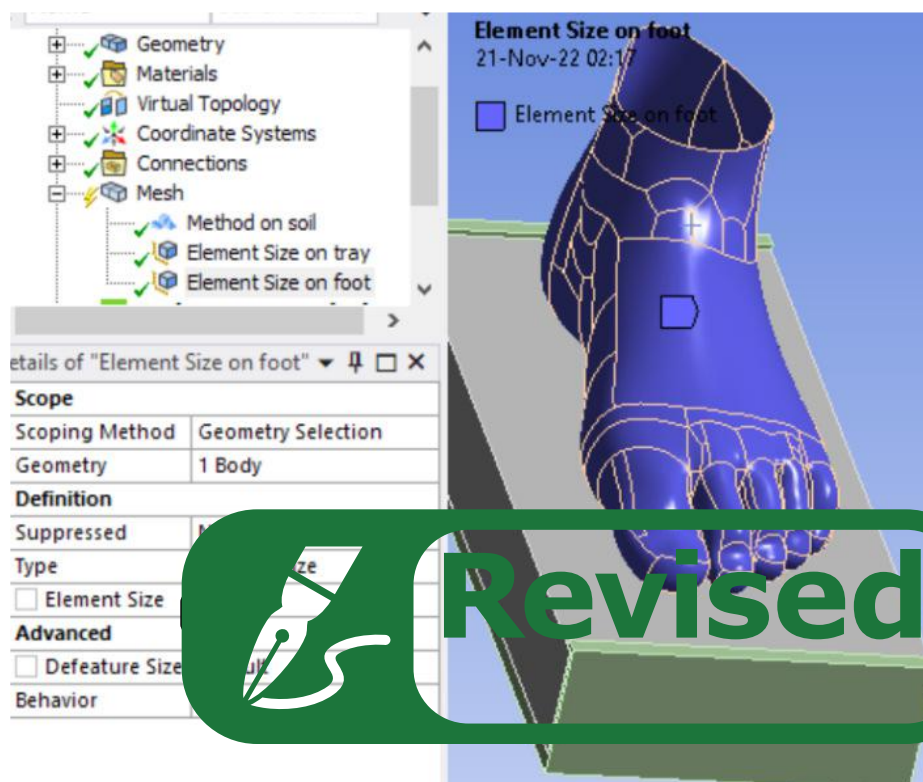




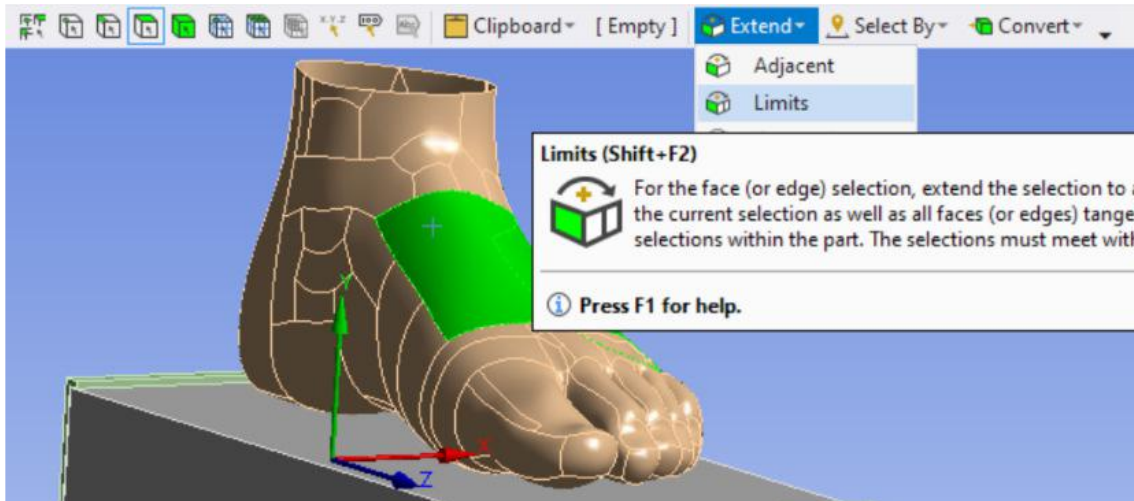
Apply a 5.5 mm Sizing on the tray, blue here.



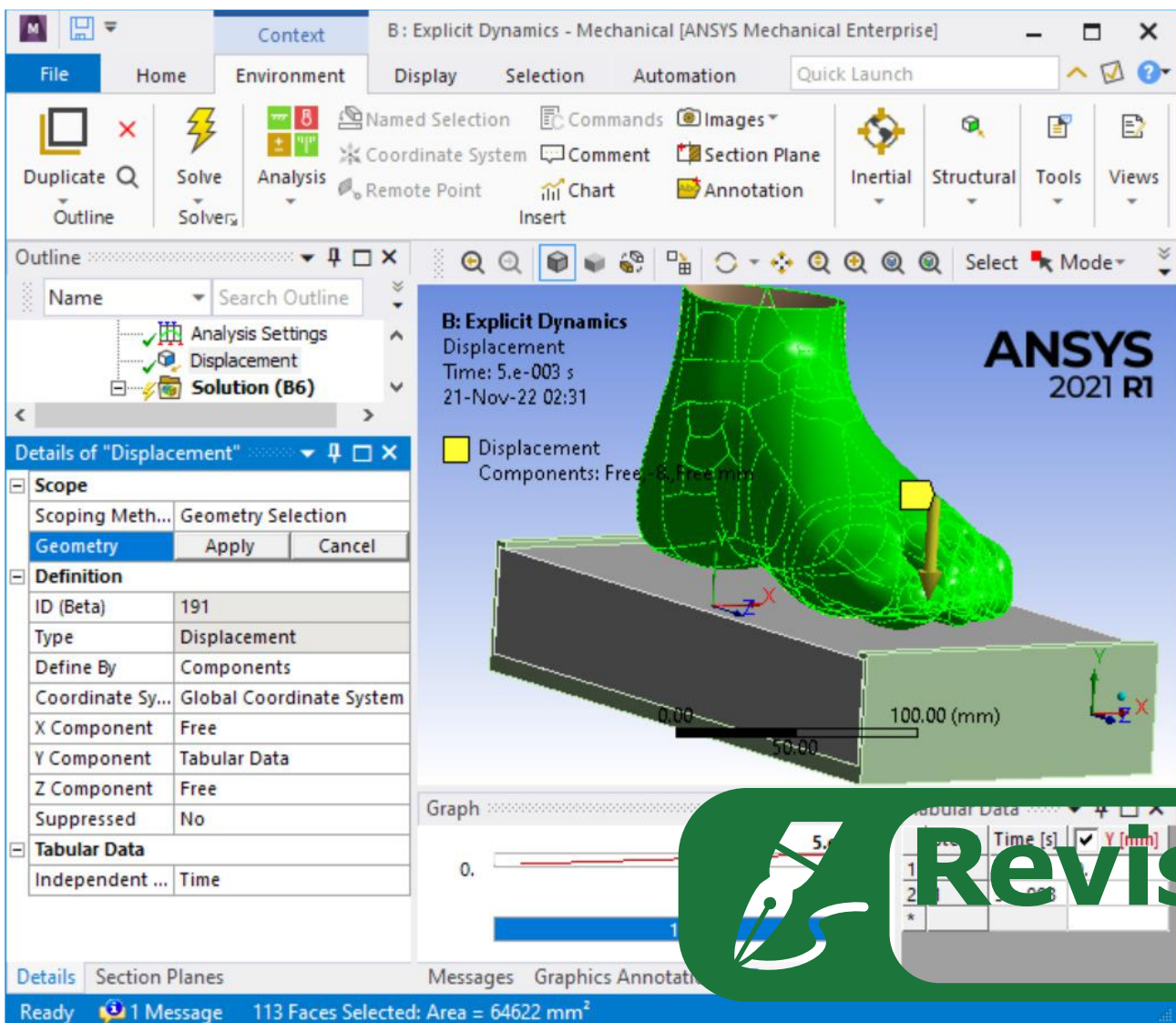
Insert a 3.5 mm Sizing on the foot. Right click all, Rename Based on Definition. Right click, Generate Mesh.



From the Environment toolbar Insert a Displacement condition on the foot: select any face from it, Extend, Limits.



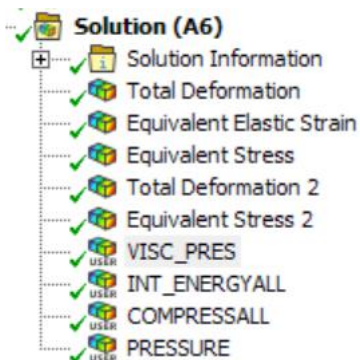
After all the faces are enclosed, click Apply then chose Tabular Data for the Y Component, -8 mm. We did not make the other components 0 to avoid rigid behavior in the foot.



After the solving has finished, click the Solution branch then the Worksheet button from the top bar, on the right to access more results. Select the items below, right click, Create User Defined results, then Evaluate All results once they are created in the left tree.

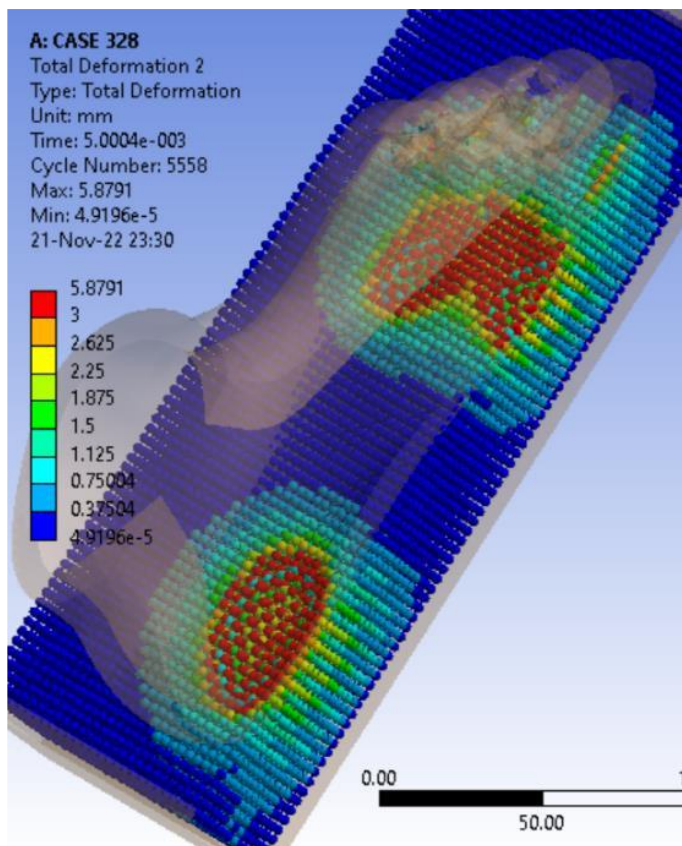
Worksheet				
Result Summary				
Type	Data Type	Data Style	Component	Expression
STRAIN_YZ	Element Nodal	Scalar		STRAIN_YZ
STRAIN_ZX	Element Nodal	Scalar		STRAIN_ZX
EFF_STN	Element Nodal	Scalar		EFF_STN
EFF_PL_STN	Element Nodal	Scalar	ALL	EFF_PL_STN
VISC PRES	Element Nodal	Scalar		VISC PRES
YLD_STRESS	Element Nodal	Scalar		YLD_STRESS
INT_ENERGY	Element Nodal	Scalar	ALL	INT_ENERGY
DENSITY	Element Nodal	Scalar		DENSITY
MASS	Element Nodal	Scalar	ALL	MASSALL
COMPRESS	Element Nodal	Scalar	ALL	COMPRESSA
TEMPERATURE	Element Nodal	Scalar	ALL	TEMPERATU
DAMAGE	Element Nodal	Scalar	ALL	DAMAGEALL
SOUNDSPEED	Element Nodal	Scalar		SOUNDSPEE
EXT_BND_FX	Element Nodal	Scalar		EXT_BND_FX
EXT_BND_FY	Element Nodal	Scalar		EXT_BND_FY
EXT_BND_FZ	Element Nodal	Scalar		EXT_BND_FZ
TIMESTEP	Element Nodal	Scalar		TIMESTEP
MASS_SCALE	Element Nodal	Scalar		MASS_SCALE
BOND_STATUS	Elemental	Scalar		BOND_STATI
RB_CONTACT_ENERGY	Element Nodal	Scalar		RB Contac
EXT_BND_MX	Element Nodal	Scalar		EXT_BND_M
EXT_BND_MY	Element Nodal	Scalar		EXT_BND_M
EXT_BND_MZ	Element Nodal	Scalar		EXT_BND_M
TC_Mass	Element Nodal	Scalar		TC_Mass
STATUS	Elemental	Scalar		STATUS
THICKNESS	Element Nodal	Scalar		THICKNESS
PRESSURE	Element Nodal	Scalar		PRESSURE
P_STRAIN_1	Element Nodal	Scalar		P_STRAIN_1
P_STRAIN_2	Element Nodal	Scalar		P_STRAIN_2
P_STRAIN_3	Element Nodal	Scalar		P_STRAIN_3

Properly made, all the resulting items should look like here.

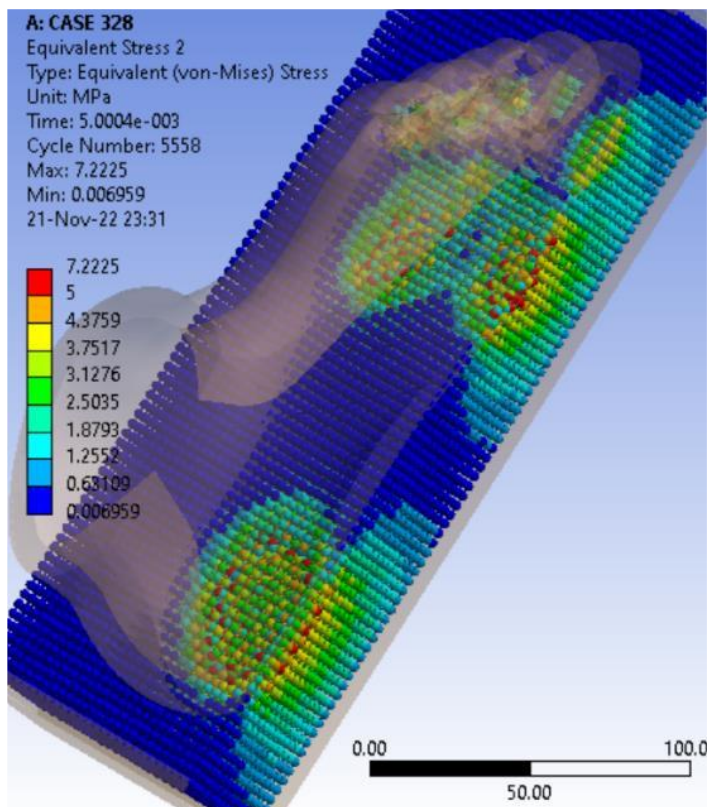




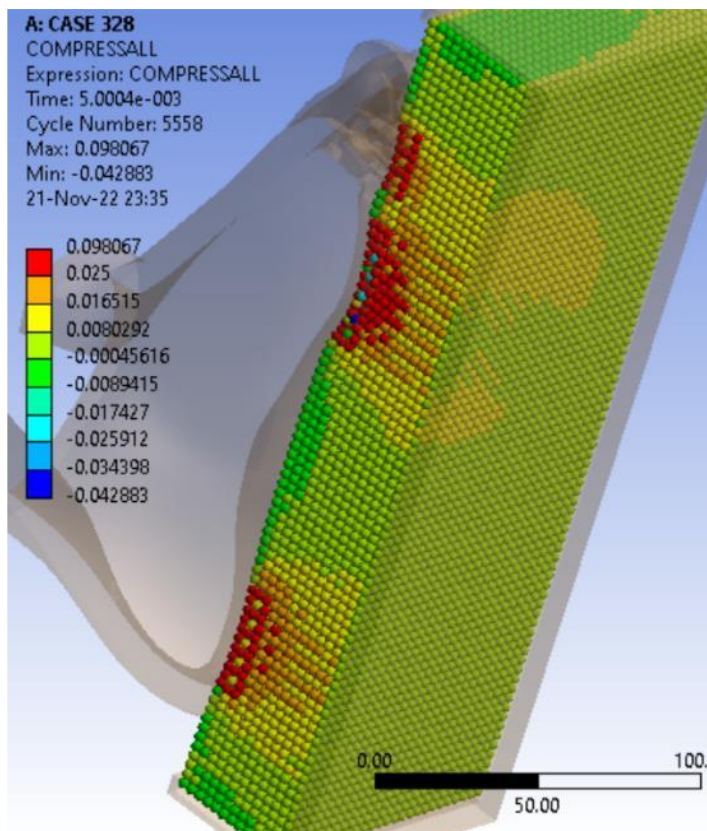
This is the Total Deformation in the soil, filtered with 3 mm for better viewing.



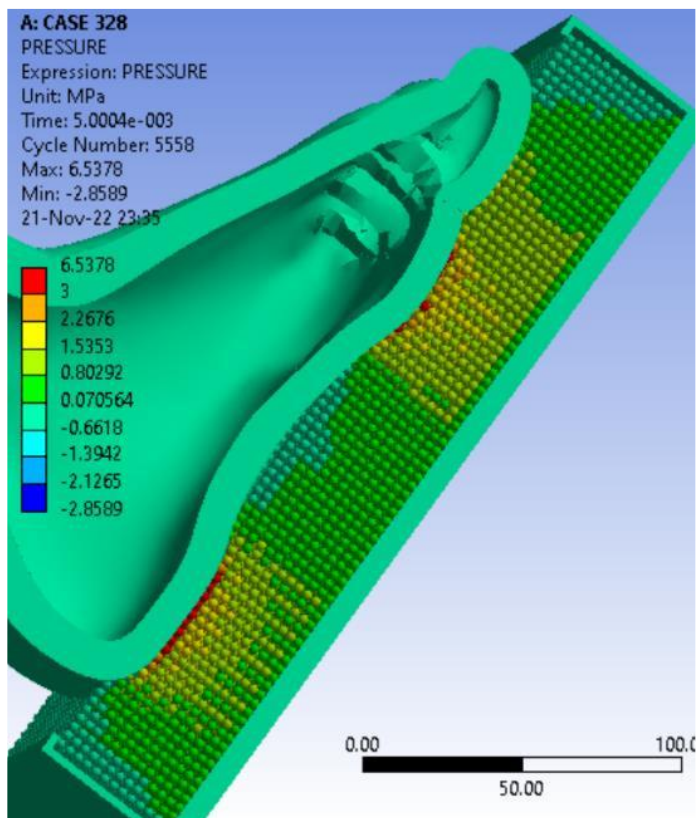
We show here the Equivalent Stress plot only in the soil, filtered with 5 MPa.



This is the COMPRESSALL plot in the soil, filtered with 0.025.

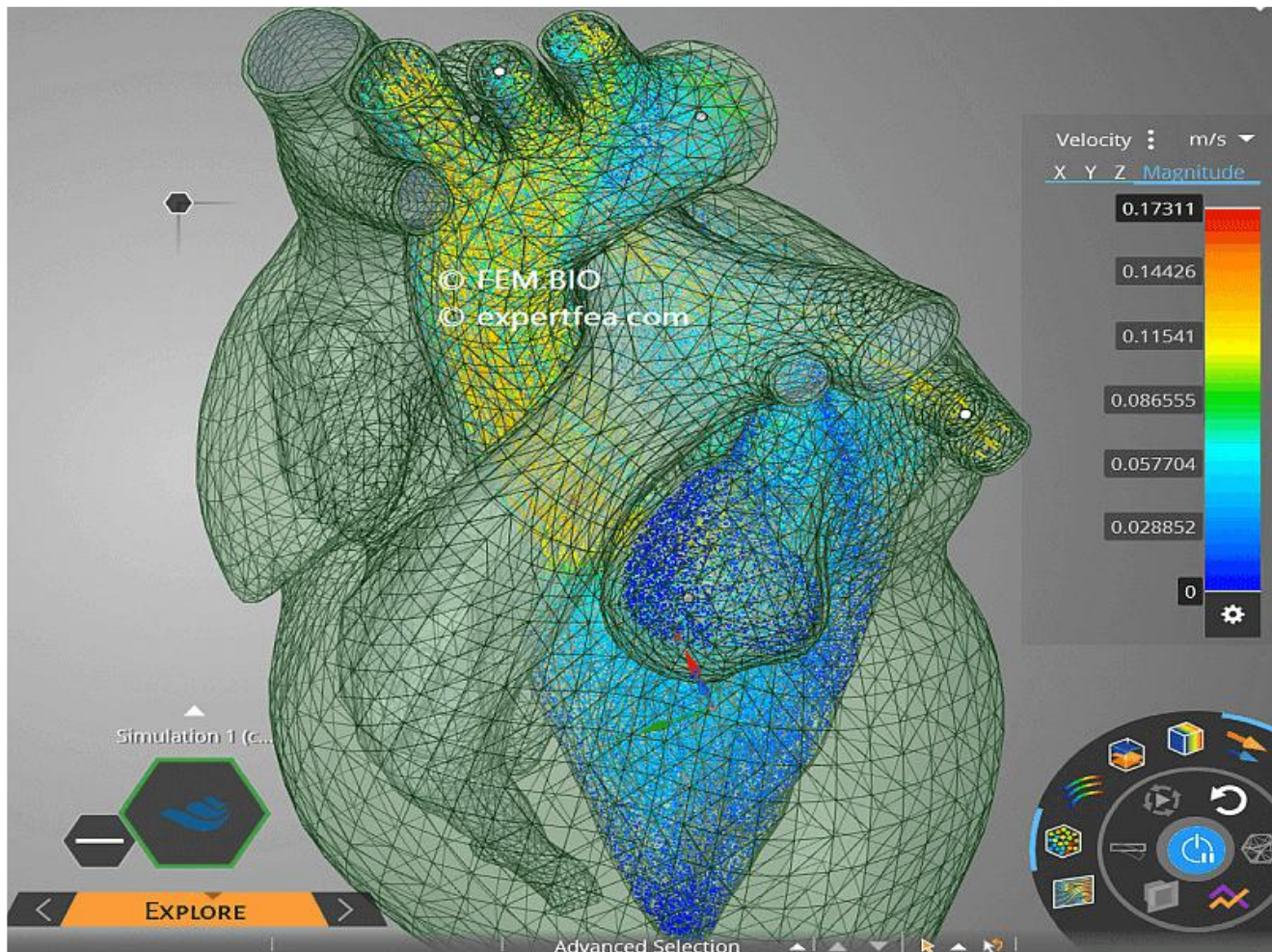


Below we show the PRESSURE plot in the assy, filtered with 3 MPa.





## CASE 322: Hemodynamics of the Arterial vs Venous Blood Flow Through the Human Heart - ANSYS Discovery 2021 R1



Locate the Discovery application inside your installed ANSYS folders. Fastest is to search for “discovery” on the respective bottom toolbar in Windows. Open it.

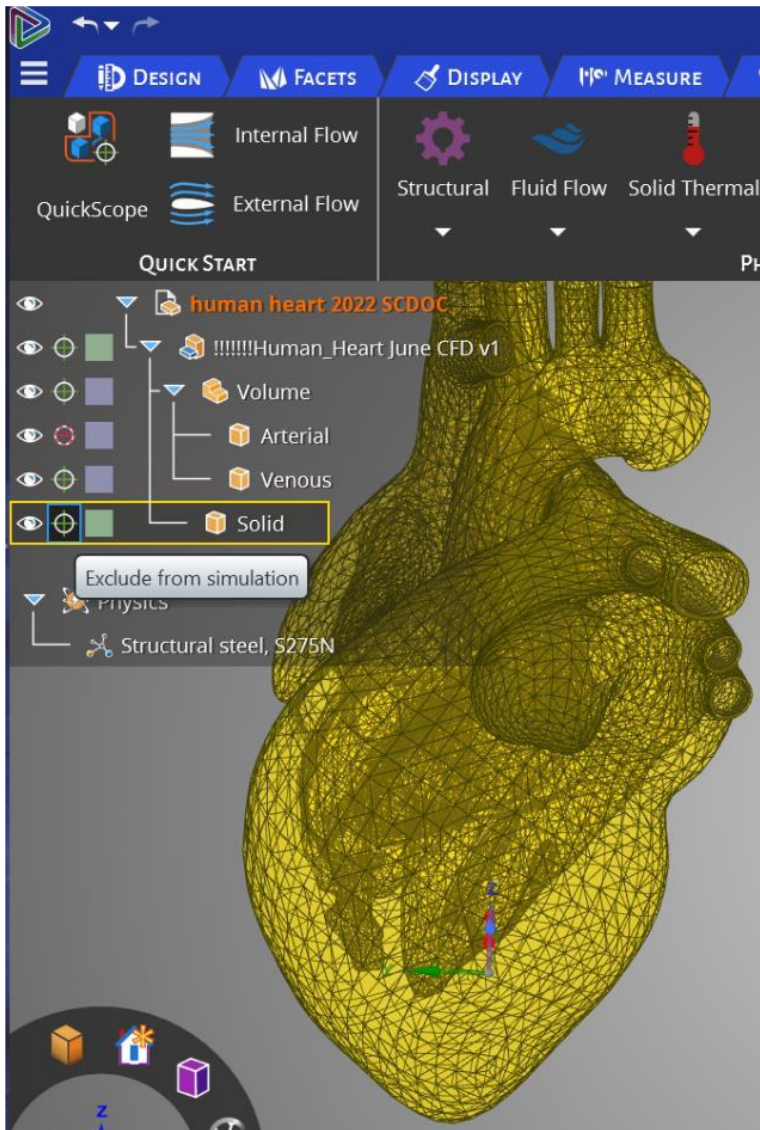


The Discovery package from ANSYS is revolutionary in the sense that it is simple and intuitive, addressing the most needed and fundamental options in simulations – all these being available to experts or non-experts in simulations in REAL TIME solving, so we can change certain geometry or pre-processing options and we will obtain the results in a LIVE manner. This is why, in previous versions, this app was also called Discovery LIVE.



Simulation of venous (or oxygen-poor blood) flow

Suppress Arterial and Solid by clicking on the small corsshairs on the right of the eye icon.



To be 100% clear, they should look like here in the end.



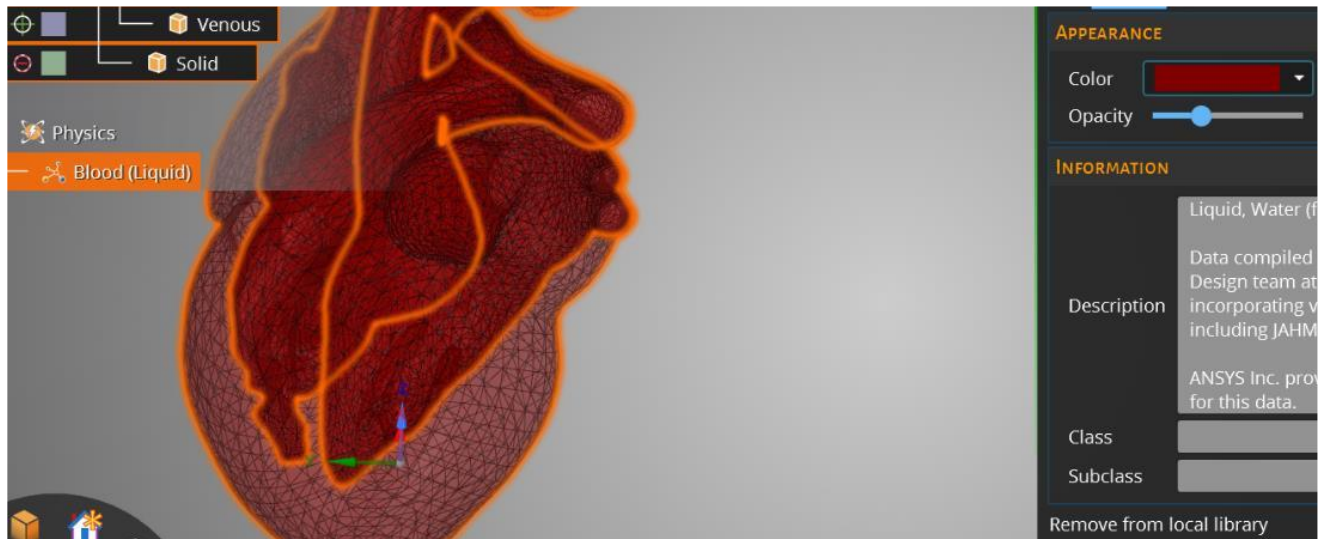
Unless we do not simulate Iron Man's body parts, from the Physics branch, right click Structural Steel, Edit.



Since we do not care to have a solid body, we will select the drop-down list. Air bodies will be shown highlighted in orange since this is the only material available to us at this moment.



For cosmetic purposes, you can change its Appearance.



And the results are updated in real-time.



#### Further homework:

- perform a similar simulation for the oxygen-rich blood flow (you can use the shown diagram or get inspiration from our YouTube simulation with the name: *Hemodynamics of the Arterial vs Venous Blood Flow Through the Human Heart - ANSYS Discovery 2021 R1*) ; solve and draw the conclusions
- change Flow Inlet velocities to 0.1 m/s; solve and draw the conclusions
- change Outlet Pressure to 0 Pa; solve and draw the conclusions
- to be physiologically correct, for the oxygen-rich CFD simulation make inlet Velocity = 0.15 m/s and outlet Pressure = 11.000 Pa; solve and draw the conclusions

Here is the end of the 1<sup>st</sup> volume regarding the modeling and simulations in Biomechanics with ANSYS, published on FEM.BIO and expertfea.com; thank you for being with us in this amazing endeavour! ❤️