MFG463370-L

Stressing Out: Simulation Workspace in Fusion 360

Elizabeth Bishop
University of Warwick, UK

Learning Objectives

- Learn about the basics of setting up a simulation in Fusion 360.
- Learn how to understand which simulation study type to use.
- Explore simulation outputs and how to use them.
- Explore iterative design methods.

Description

Fusion 360 software could be your most powerful tool as a designer, maker, or engineer. Fusion 360 is an all-encompassing piece of software that helps users with processes from start to end, design to manufacture, utilizing CAD, CAM, Simulation, and Rendering to envisage a design. This class will focus on the middle of that process—Simulation for Fusion 360. Simulation lets designers test (simulate) their designs before reaching the manufacturing process. This enables iterative changes without the time and money (and potential waste) that can go into making a product only to find it doesn't meet the standards needed. The Simulation workspace in Fusion 360 can feel a little intimidating at first glance—with so many options to choose from. This class will take you through the steps, showing you which study type of simulation can be used depending on what your aims are. You will leave this class feeling more confident and ready to improve your whole design process!

Speaker

Elizabeth Bishop is a Postgraduate Researcher at the University of Warwick researching Large-Scale Additive Manufacturing (3D Printing). She has been interested in 3D printing for several years now, following a successful project surrounding designing and making a humanitarian rescue UAV. Elizabeth also volunteers as a Maker in Residence in the Engineering Build Space at Warwick University where she explores making, CAD and CAM alongside 3D printing.

@LizBish94

Table of Contents

Introduction

In this class we are going to go through the entire Simulation workspace in Fusion 360. We will go through the basics of setting up a simulation; how to choose the study type; how to review the results; and how these can be used to influence future design decisions. We will cover the following study types in the simulation workspace:

1. Static Stress
2. Modal Frequencies
3. Thermal
4. Thermal Stress
5. Structural Buckling
6. Nonlinear Static Stress
7. Event Simulation
8. Shape Optimization

There is a supplied data set for this class which you will need to upload into your data panel. For each section follow the example through the step-by-step instructions to solve the simulation study.

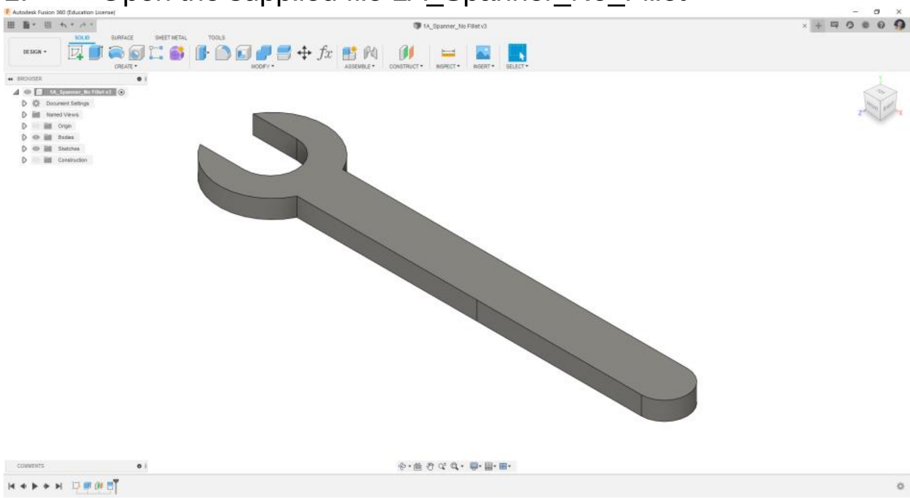
Static Stress

Static stress analyses can be used when a known load is applied in a static (non-dynamic) way to an object. The resultant stress, strain, and deformation results analysed to determine the likelihood of failure of the design. For a Linear Static Stress analysis, it is assumed that the model will behave in an elastic way i.e. will return to the original shape when the load is removed.

Exercise 1A+B: Static Stress, Spanner

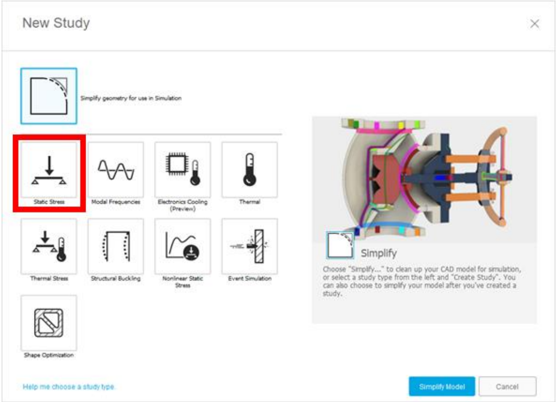
In this exercise we will perform a static stress simulation on a spanner.

1. Open the supplied file 1A_Spanner_No_Fillet

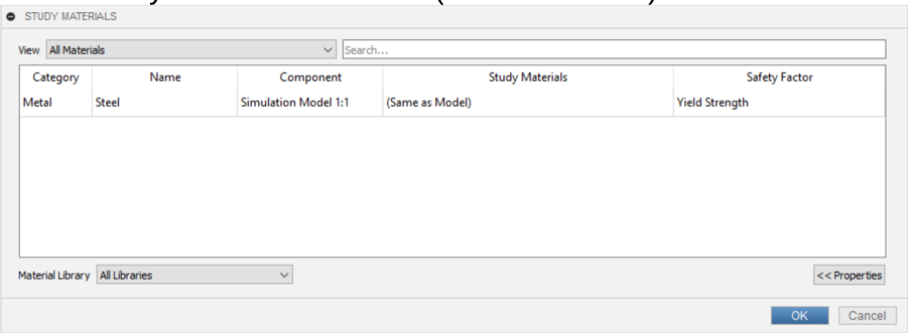


2. Move from the Design Workspace into the Simulation Workspace

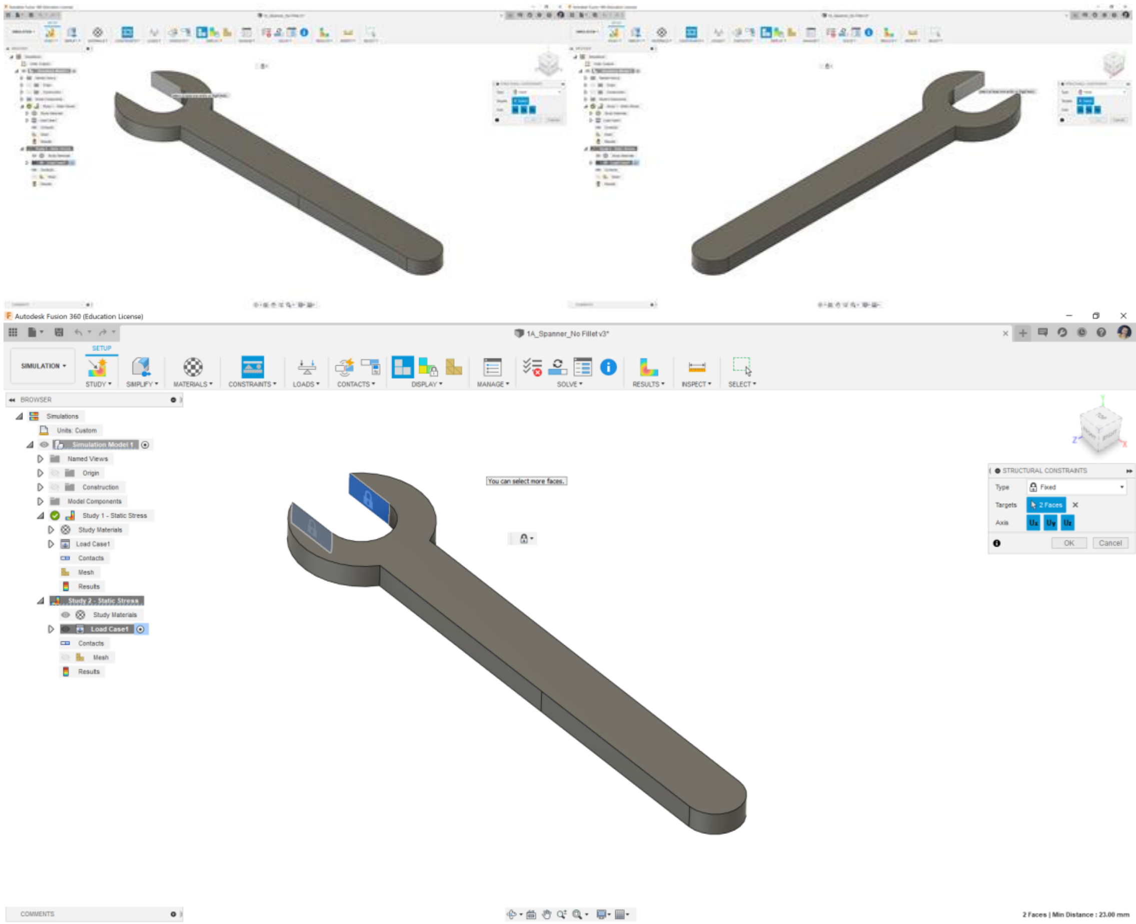
3. Choose a static stress simulation from the study type menu



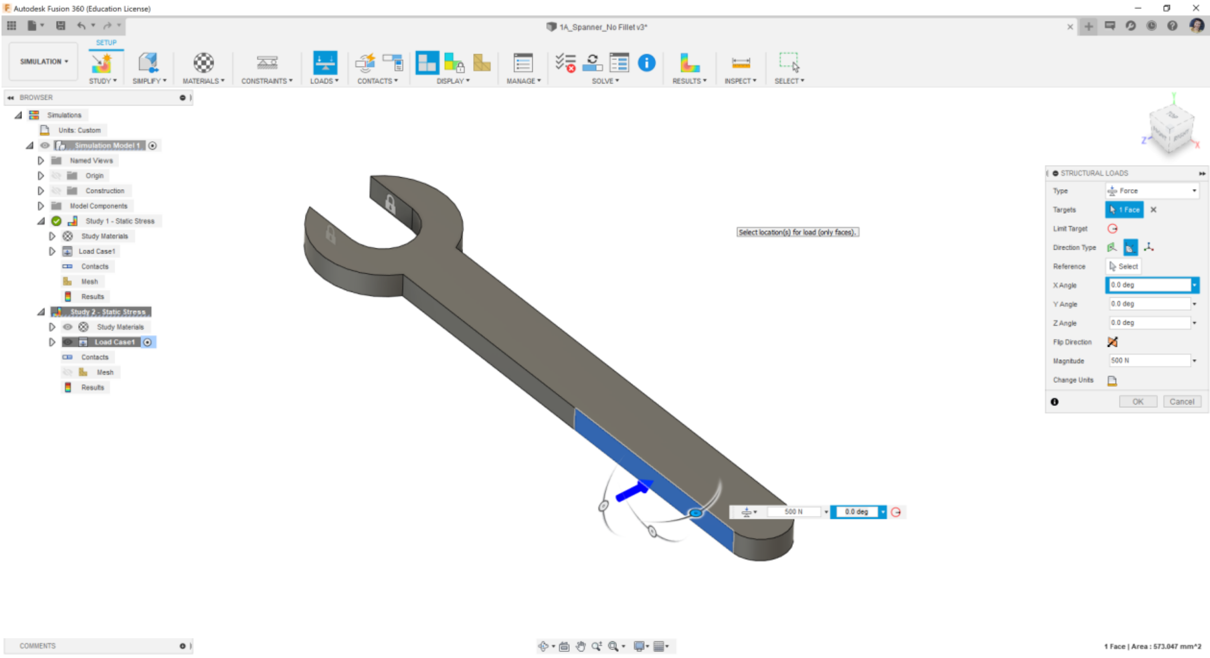
4. Set your material to Steel (Same as Model)



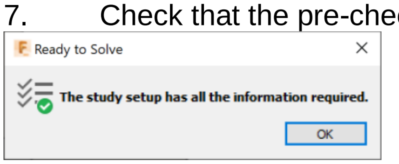
5. Apply the constraints to the two surfaces as below, use a fixed constraint, and click ok



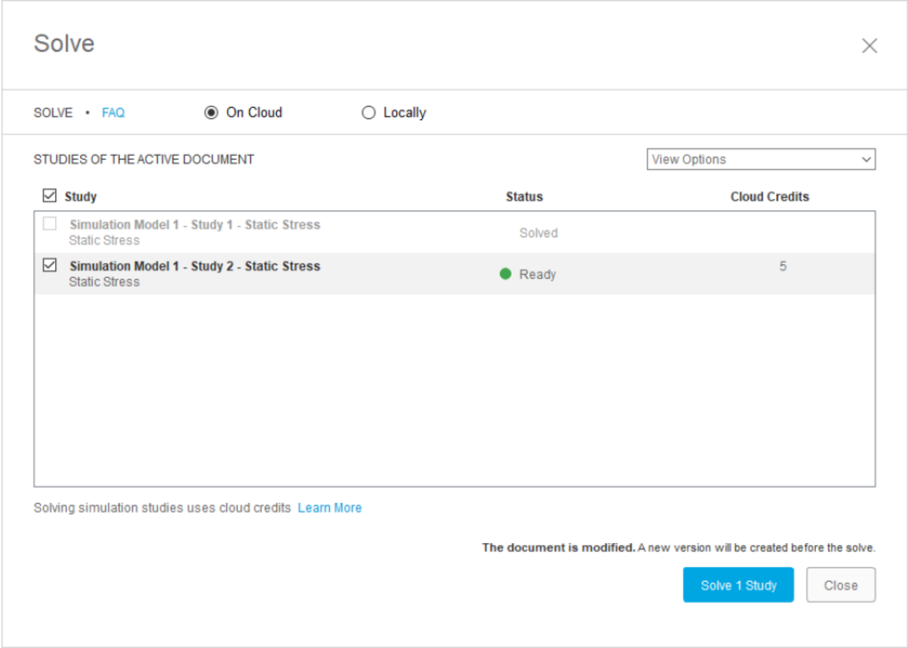
6. Apply the loads. Select the split face. Using a force load, change the direction type to angle and a magnitude of 500 N.



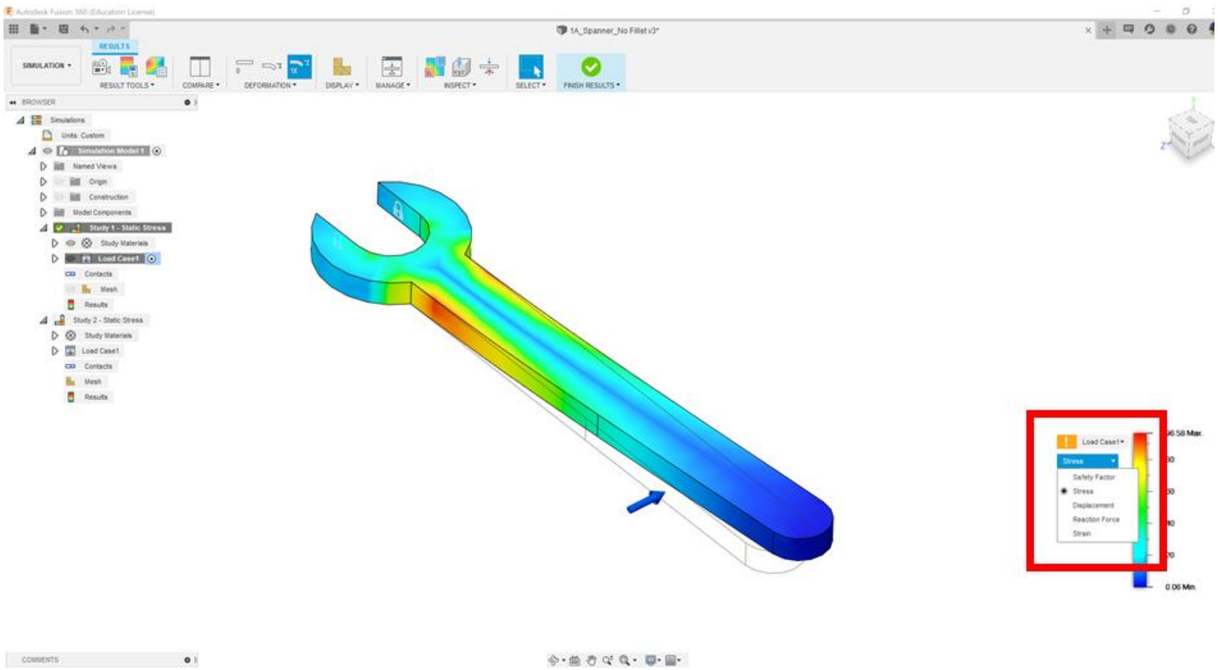
7. Check that the pre-check is ready



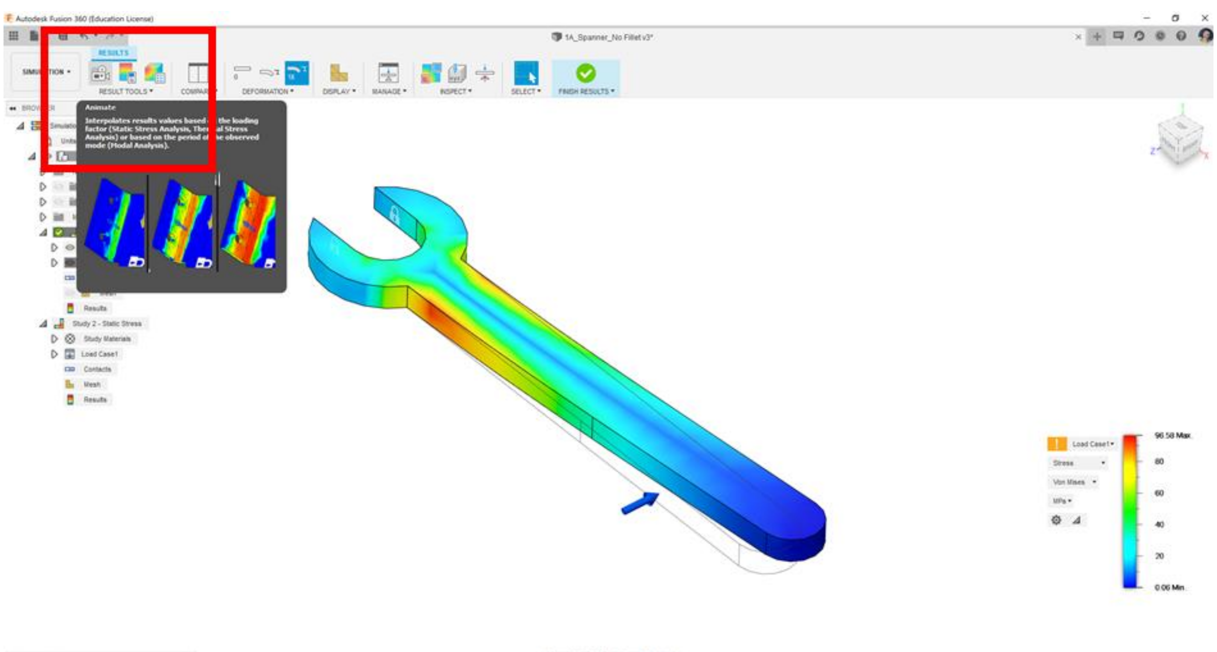
8. Click Solve and choose either on cloud simulation (requires 5 cloud credits) or solve locally.



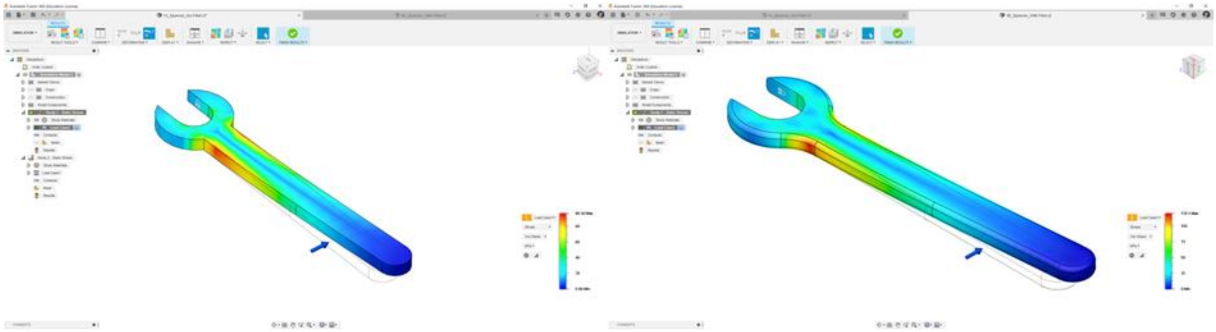
9. Review your results. You can change the result type using the drop-down list



10. You can also animate your results, change the deformation scale, and share your results by generating a report.

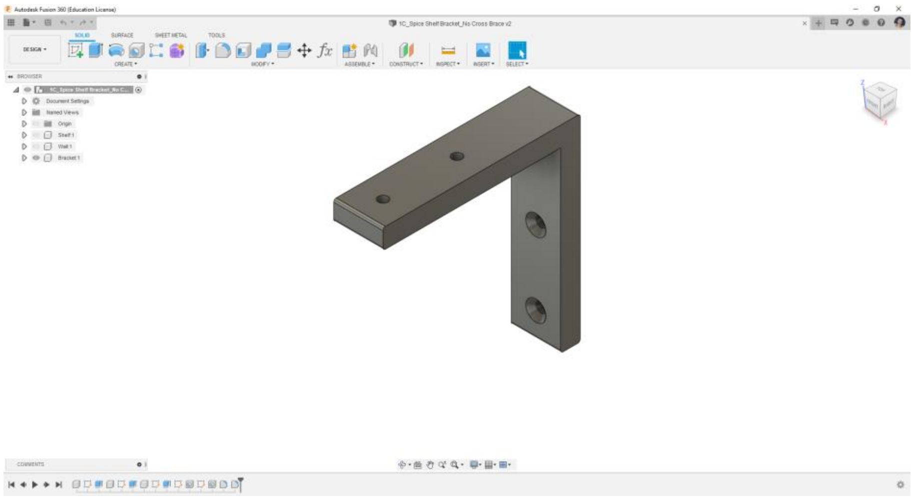


11. Repeat steps 1 to 10 using the supplied file 1B_Spanner_With_Fillet and compare the results



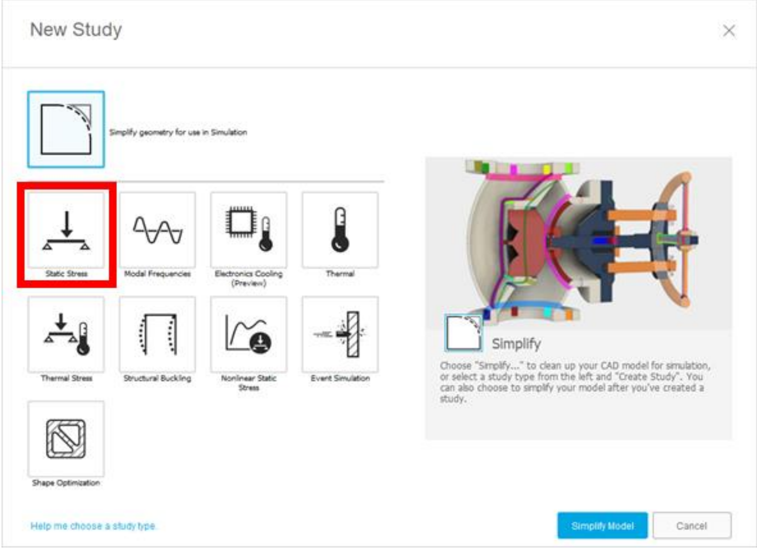
Exercise 1C+D: Static Stress, Spice Shelf

1. Open the supplied file 1C_Spice Shelf Bracket_No Cross Brace

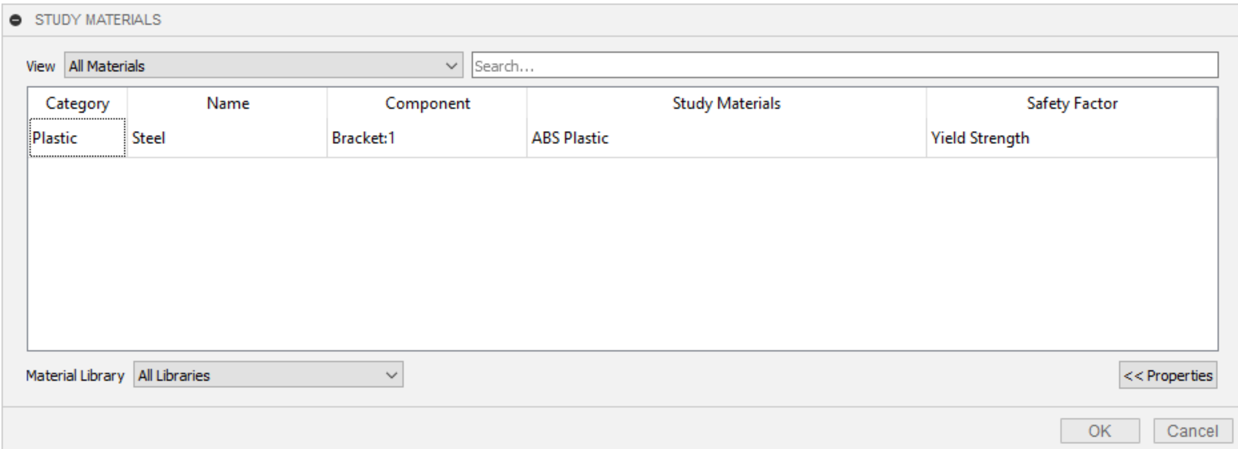


2. Move from the Design Workspace into the Simulation Workspace

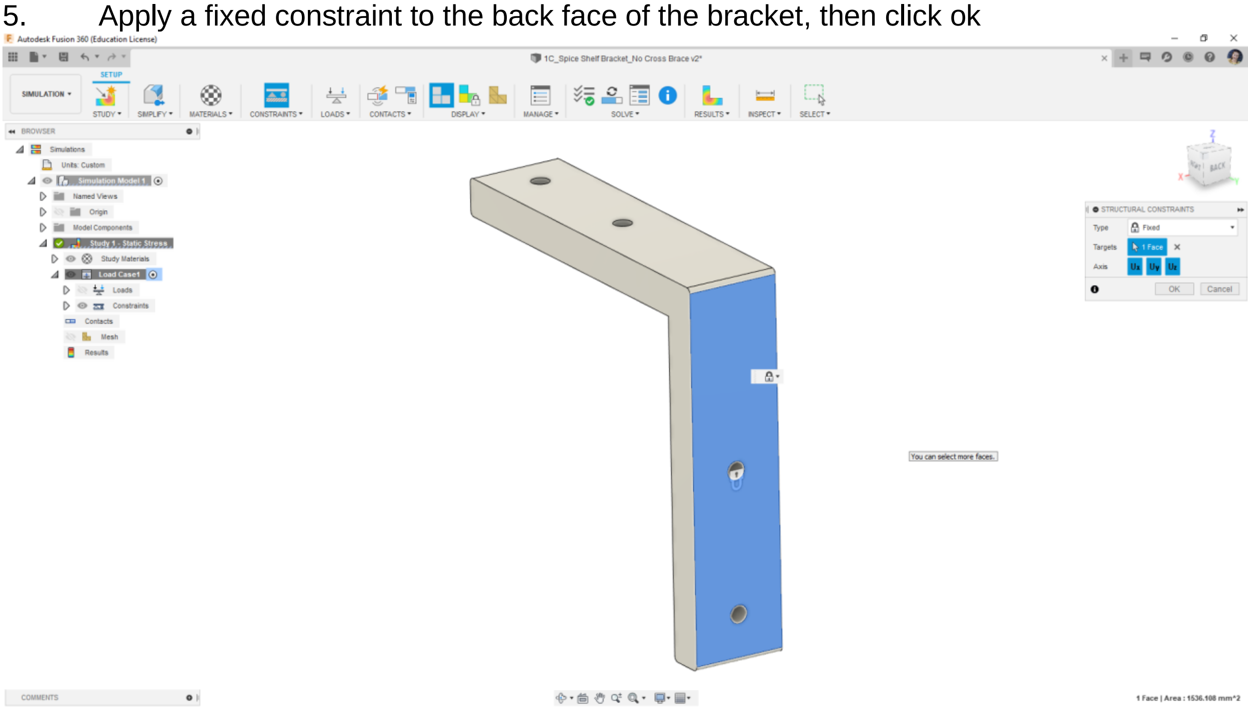
3. Choose a static stress simulation from the study type menu



4. Change the material to ABS Plastic

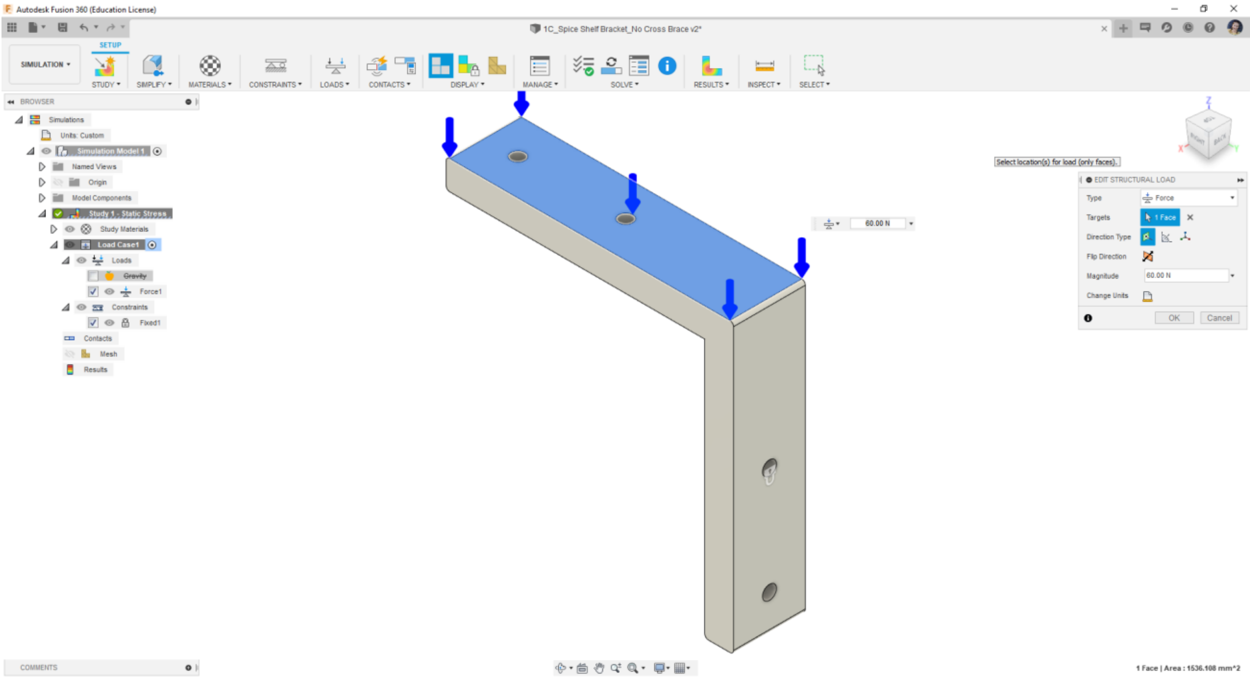


5. Apply a fixed constraint to the back face of the bracket, then click ok

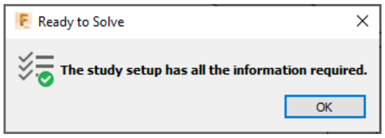


6. Apply a load of 60 N to the top surface, acting downwards. Change the type to Force. Direction type to Normal (acting downwards) and a magnitude of 60 N.

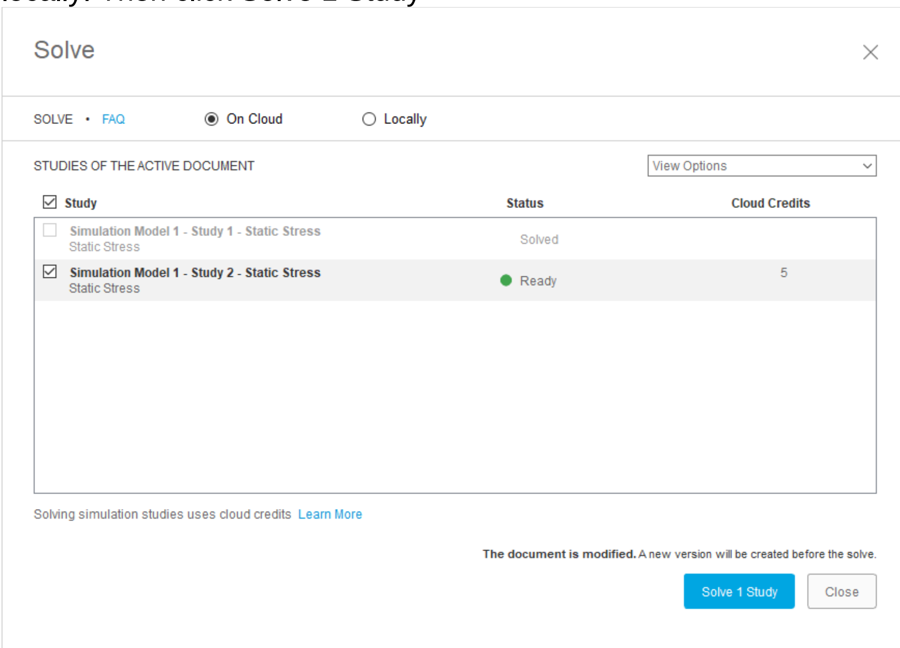
7.



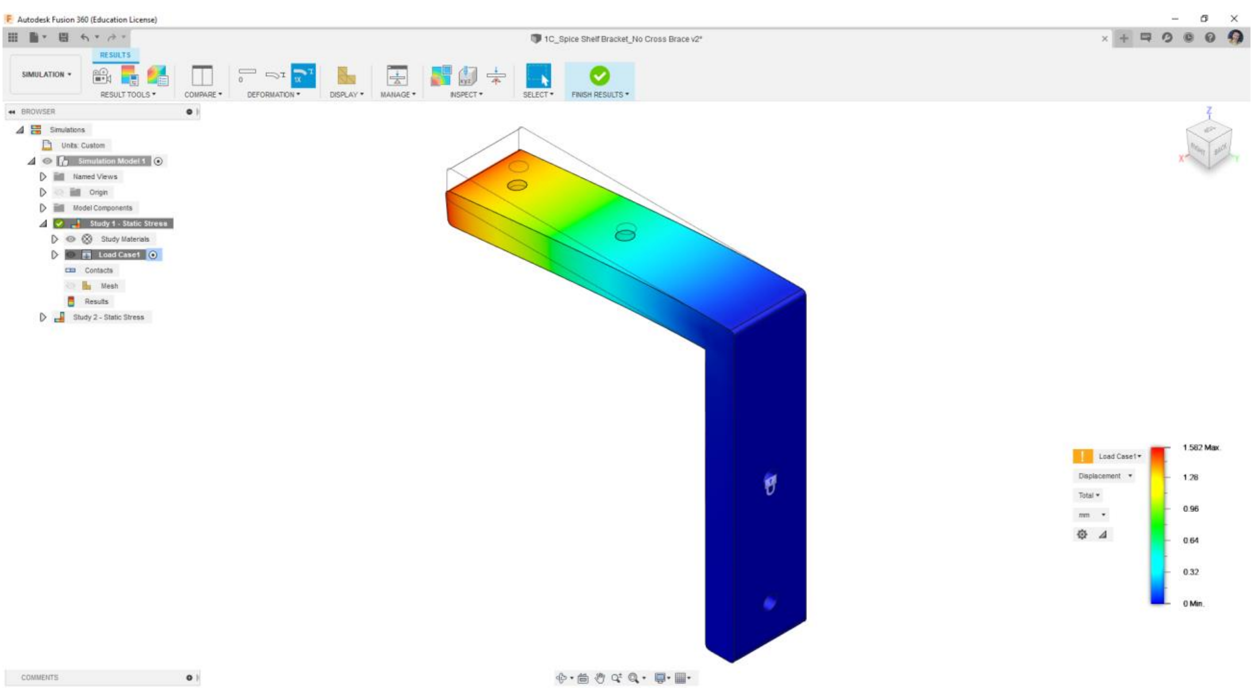
8. Click Pre-Check to ensure that everything is set up for the Simulation



9. Select Solve and either solve on the cloud (using 5 cloud credits) or solve the simulation locally. Then click Solve 1 Study



10. Review the results. Observe that the displacement is quite high.



11. Repeat steps 1 to 10 using the supplied file 1D_Spice Shelf Bracket_Wtih Cross brace and compare the results. Note the reduction in displacement.

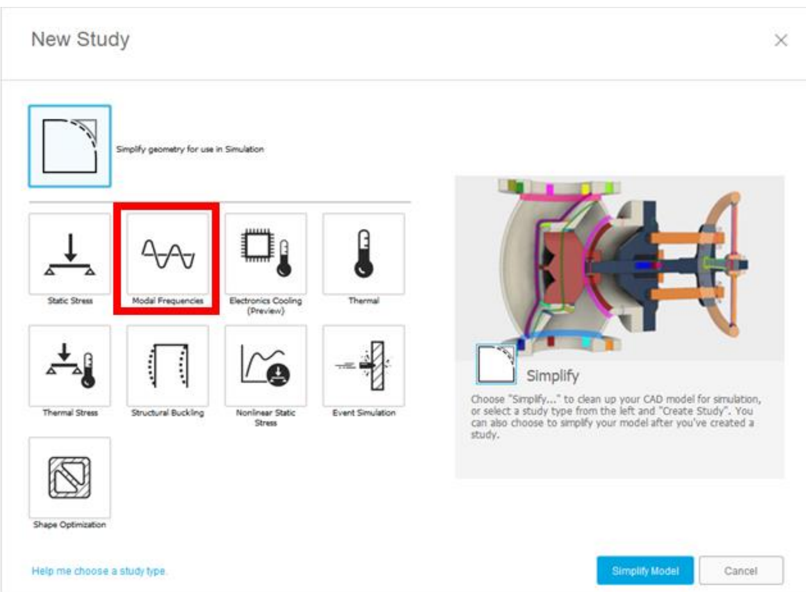
Modal Frequencies

A modal frequency study can be used to determine a part's natural frequency. This is important as you do not want a design to naturally vibrate when in use and to shake itself apart. Structures exhibit multiple natural frequencies, and this study type can be used to determine the fundamental mode (lowest frequency) and its harmonic modes. Engineers need to design components so that the natural frequency is not close to the frequency at which the component operates.

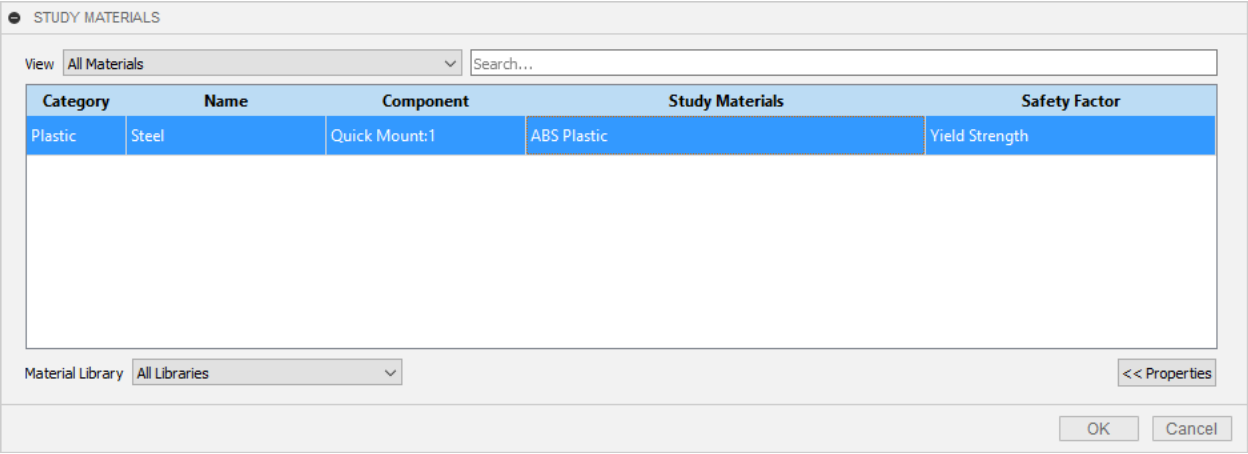
Exercise 2A

1. Open the supplied file 2A_Fan_Mount. We are going to perform a modal frequency study on a mount for a fan, the fan operates at 50 Hz so this is the frequency we want to avoid.
2. Move from the Design Workspace into the Simulation Workspace

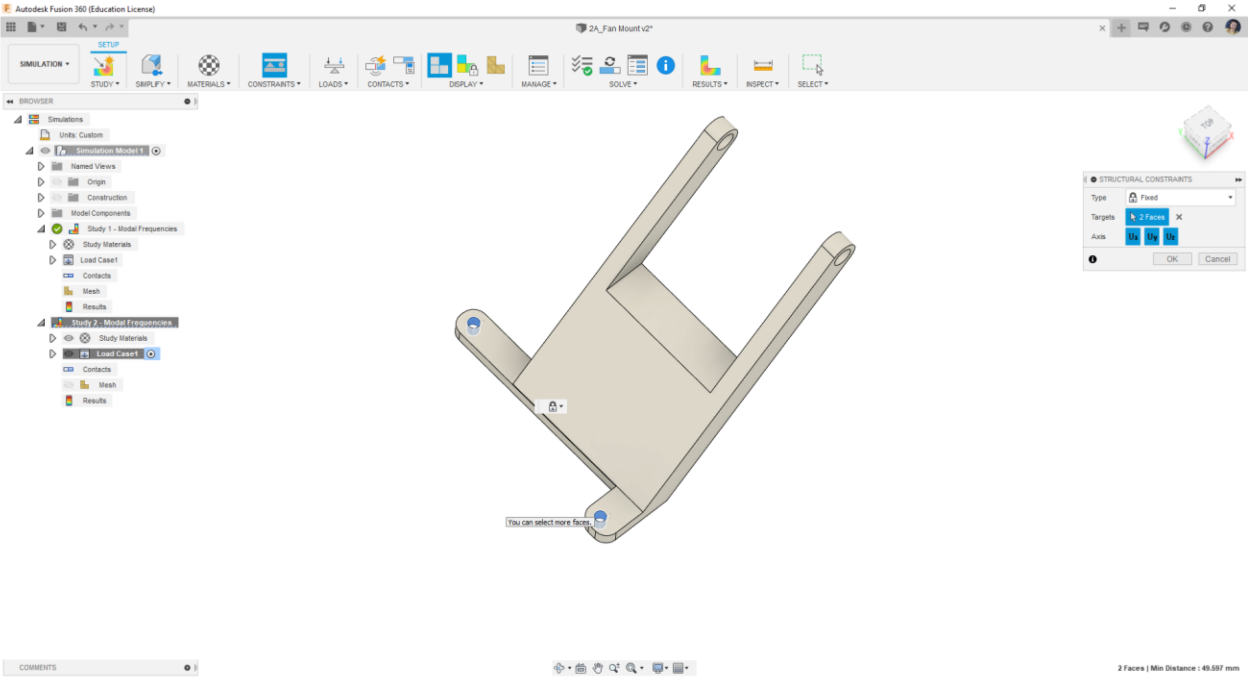
3. Choose a Modal Frequencies simulation from the study type menu



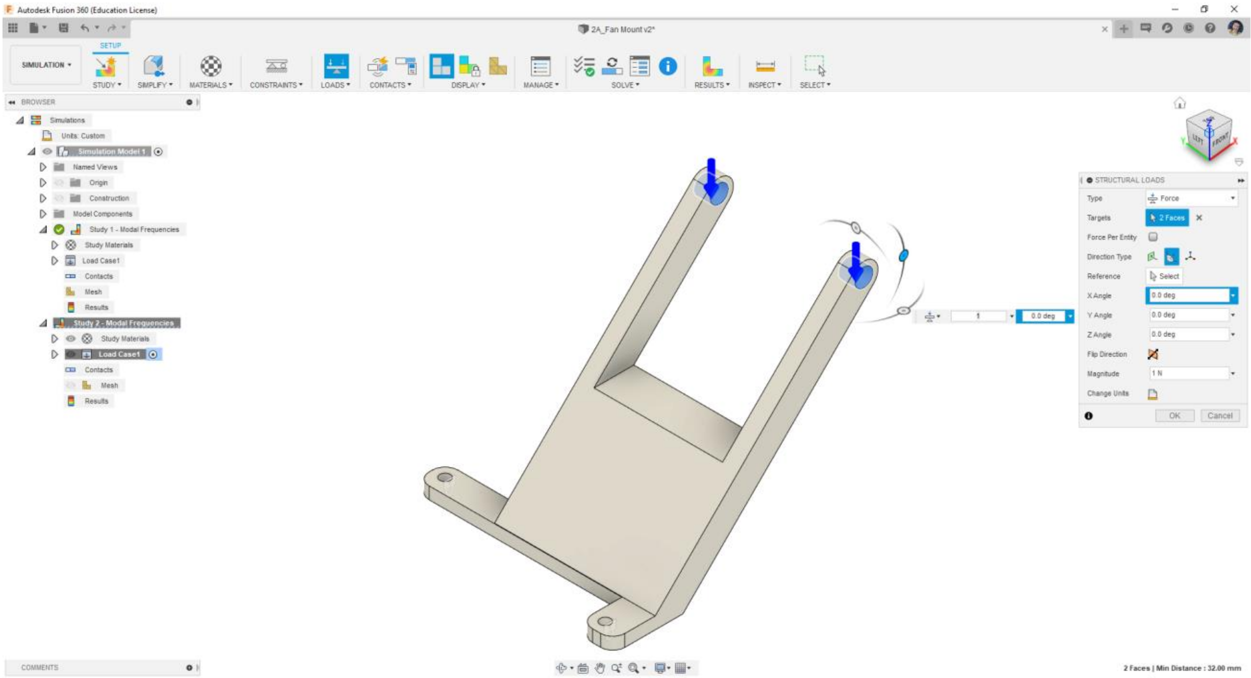
4. Select the material, choose ABS Plastic



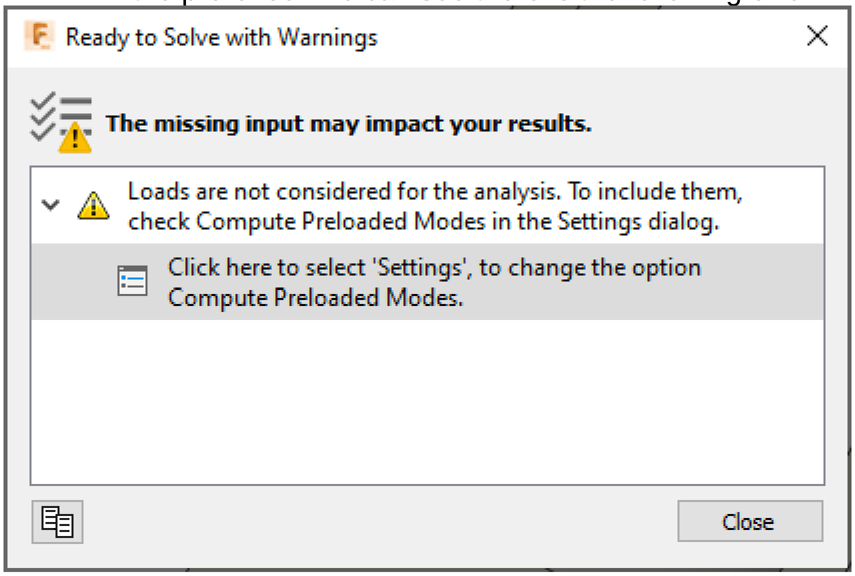
5. Set the contraints. Choose a pin constraint and apply to the two cylindrical pins as shown below



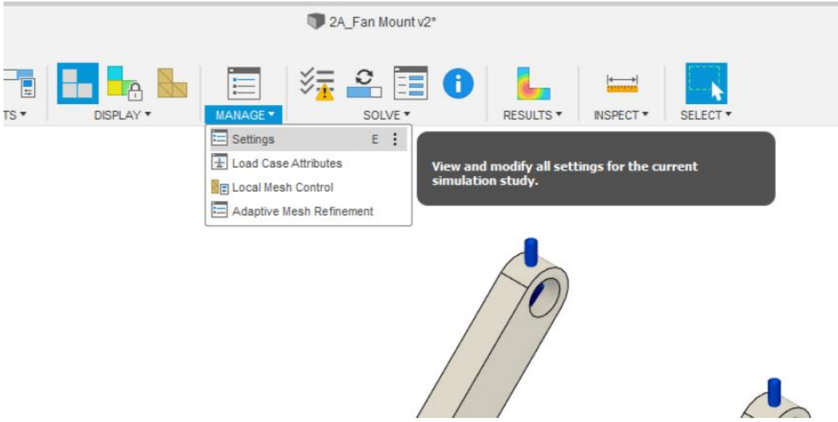
6. Apply the Loads. Apply a load across the two mounting holes for the fan. This will be a load of 1 N and an angle pointing directly downwards.

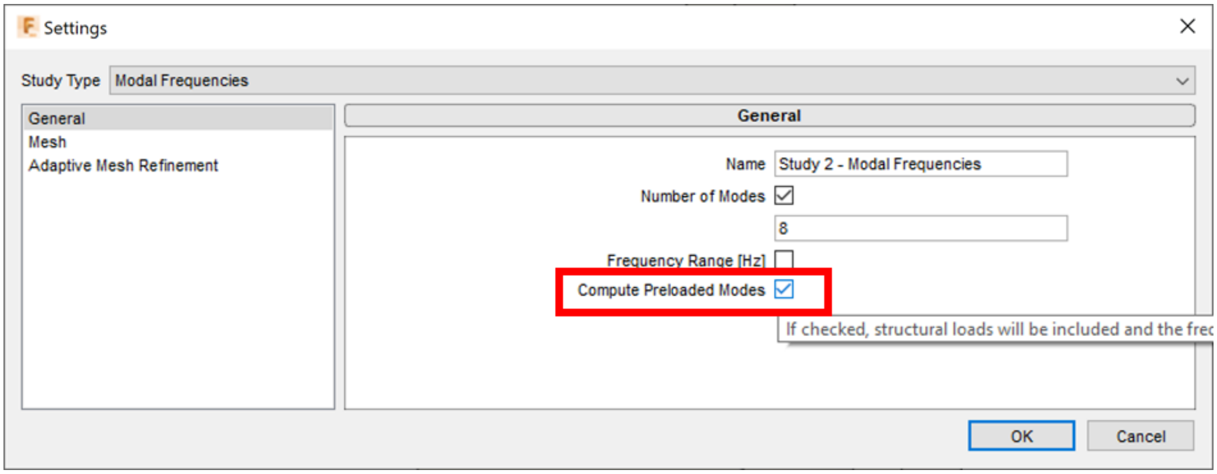


7. In the pre-check we can see there is the following error.

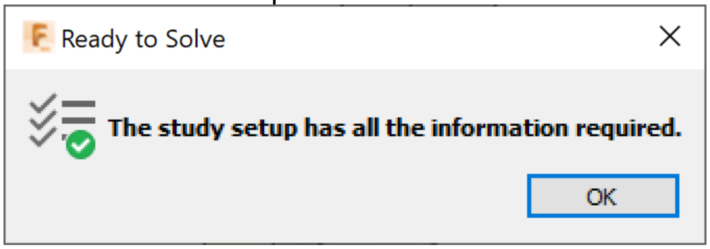


8. Go to Settings and click ‘Apply pre-loads’

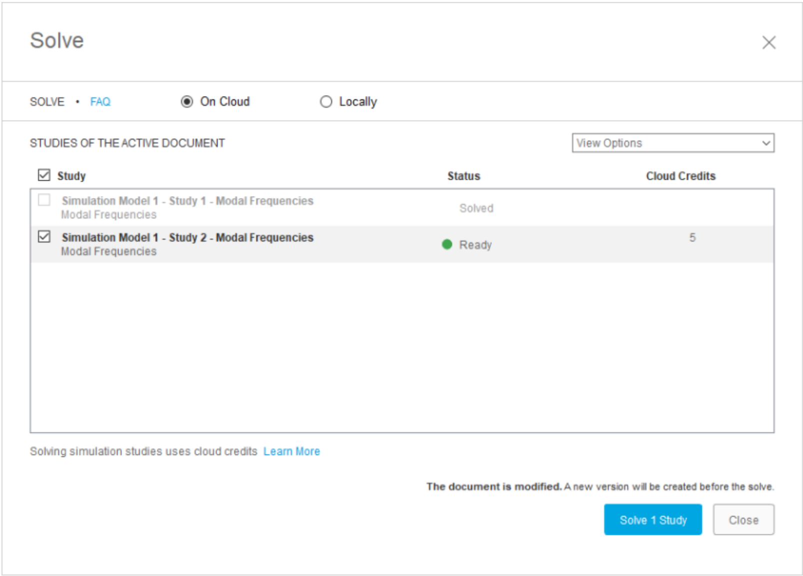




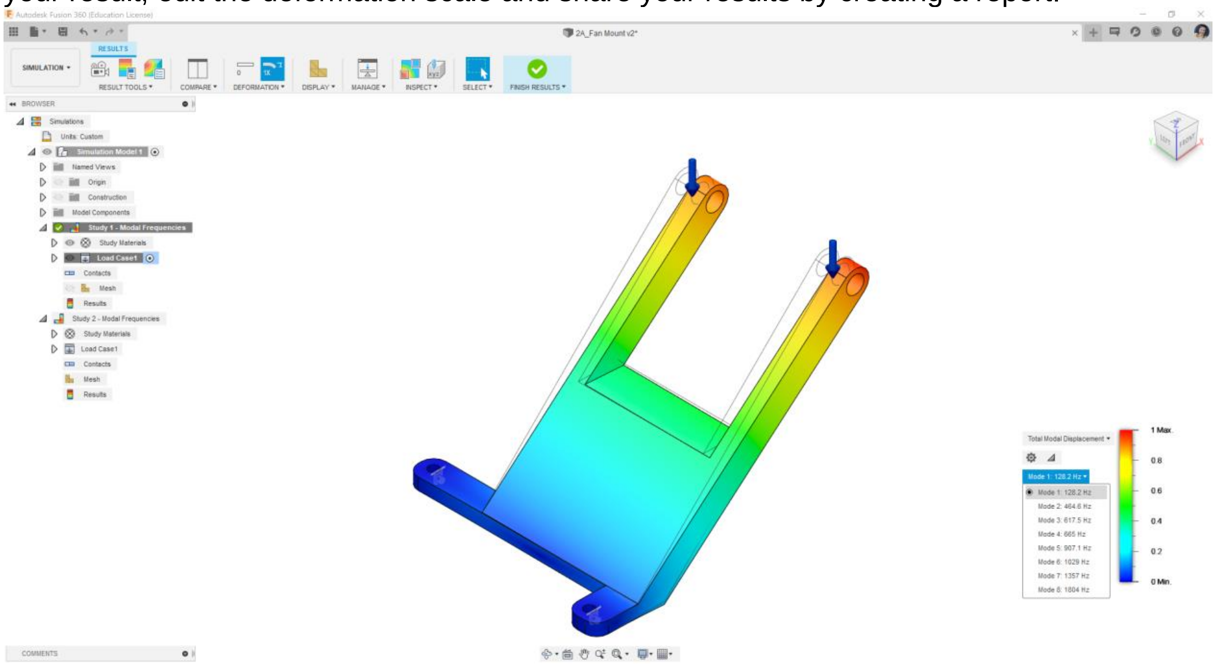
9. Recheck the pre-check



10. Select solve and Solve on the cloud (requires 5 cloud credits) or solve locally



11. Review the results. Look at the different modes using the drop-down list. You can animate your result, edit the deformation scale and share your results by creating a report.



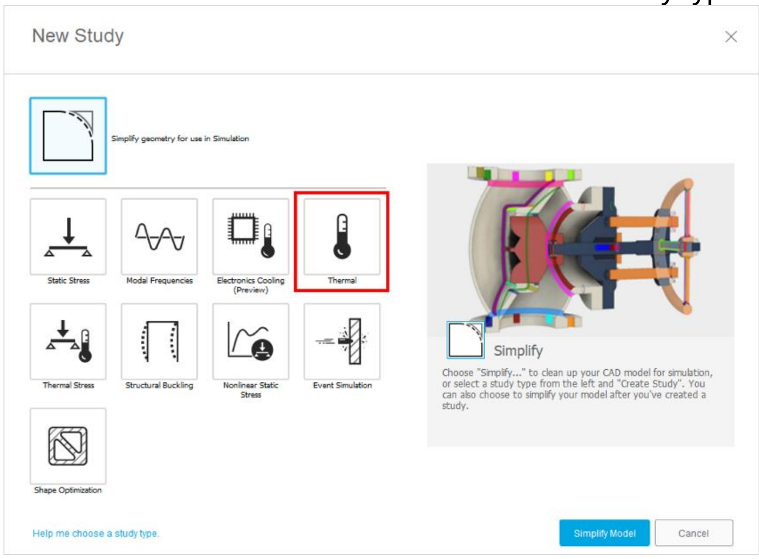
Thermal

Thermal analysis can be used to modelling thermal problems such as cooling fins for heat distribution or to look a\at multiple possible materials for an insulation. In the tutorial we will look at a pipe and different insulation materials and thicknesses.

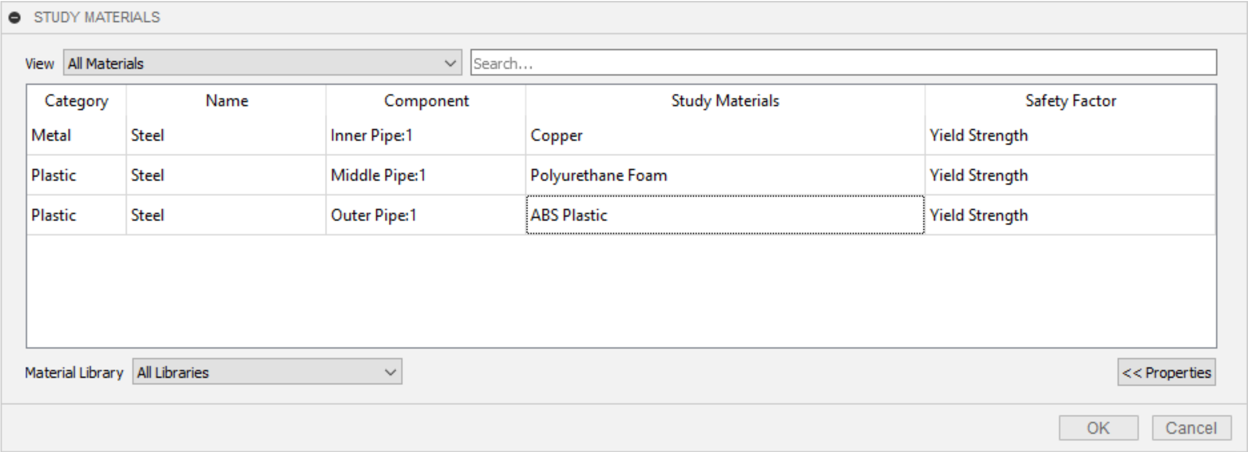
Exercise 3A

1. Open the supplied file 3A_MultiLayer_Pipe_1
2. Move from the Design Workspace into the Simulation Workspace

3. Choose a Thermal simulation from the study type menu

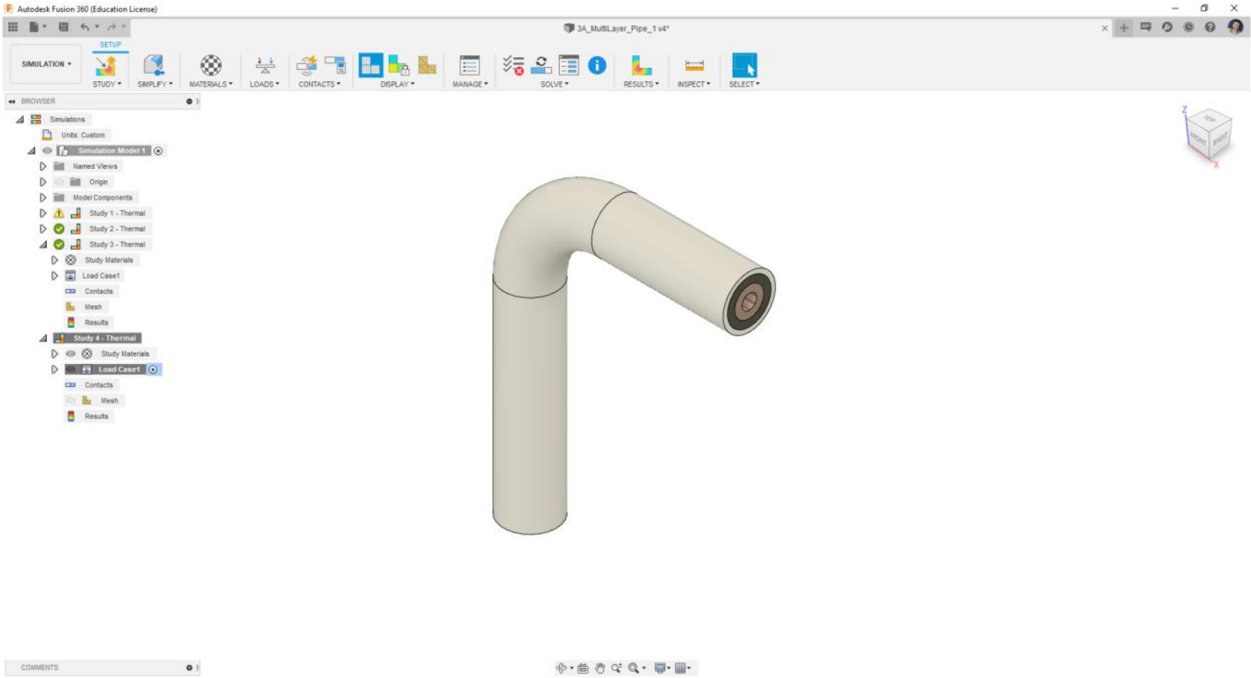


4. Set up the materials. We will need to set three different materials
a. Inner Piper set as Copper
b. Middle Pipe set as Polyurethane Foam
c. Outer Pipe set as ABS Plastic.



Category	Name	Component	Study Materials	Safety Factor
Metal	Steel	Inner Pipe:1	Copper	Yield Strength
Plastic	Steel	Middle Pipe:1	Polyurethane Foam	Yield Strength
Plastic	Steel	Outer Pipe:1	ABS Plastic	Yield Strength

This will also update the appearance of the three pipe sections in the display window

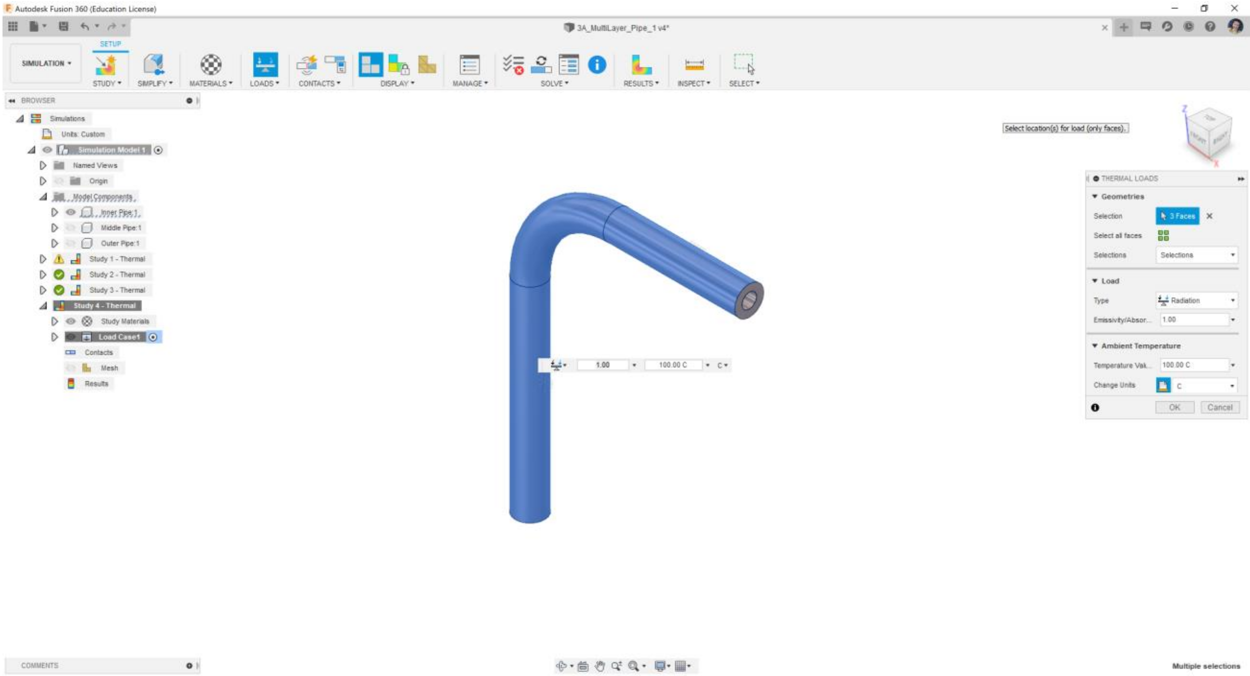


5. Hide the Middle and Outer pipe in the browser tree by clicking the eye next to them. Apply a Load to the inner pipe.

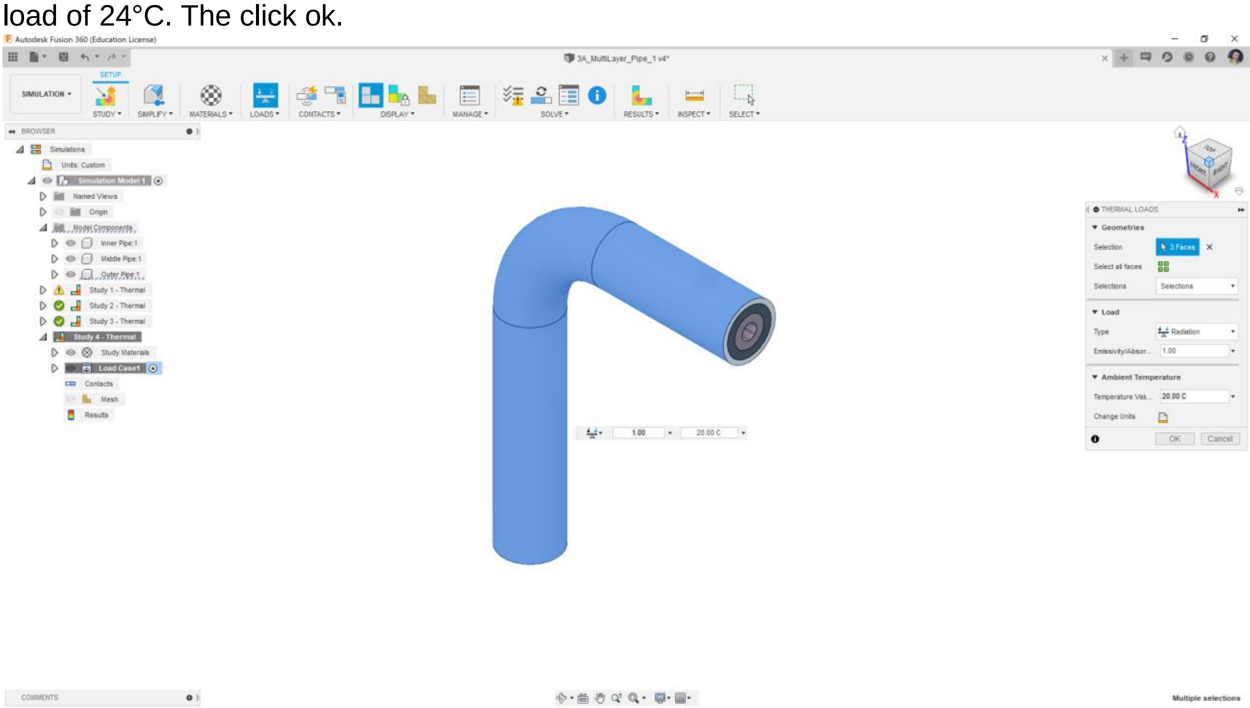
a. Select all three sections of the Inner Pipe.

b. Choose type to Radiation

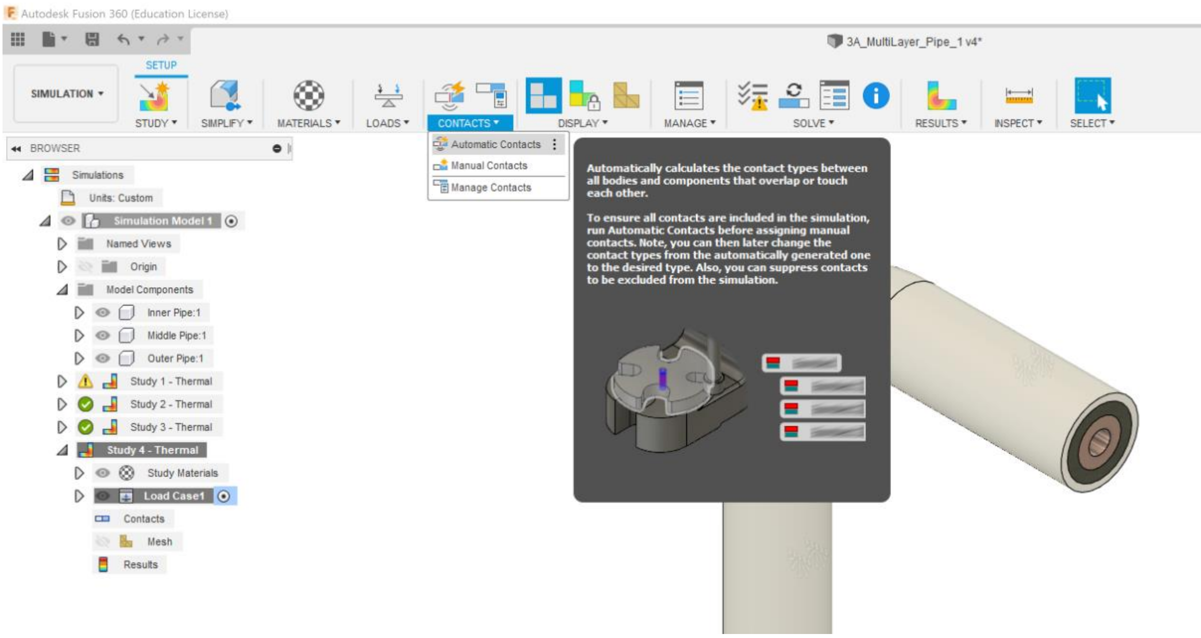
c. Set the value to 100°C



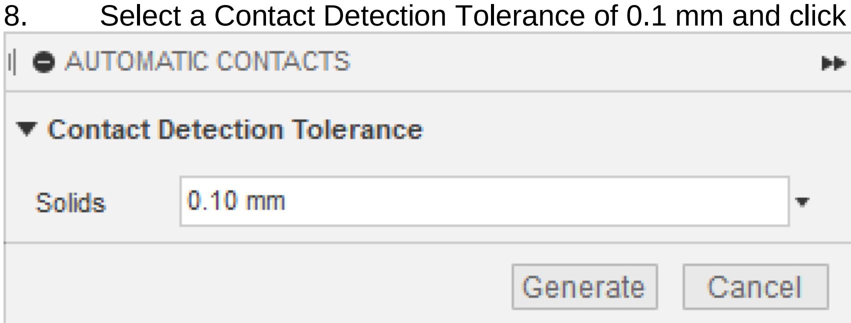
6. Show the Middle and Outer Pipe and set a thermal load to the outer surface of the pipe. This is going to represent room temperature. Choose the type of Radiation and apply the thermal load of 24°C. The click ok.



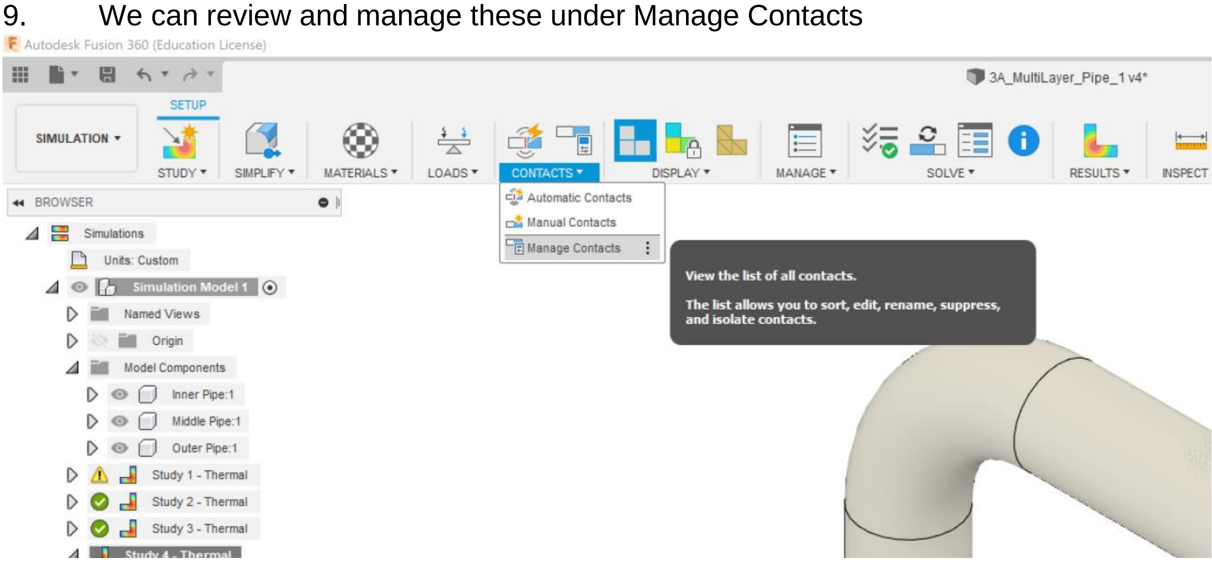
7. In this simulation we have multiple components (or bodies) so we need to apply contacts between the surfaces so the model knows how to behave.

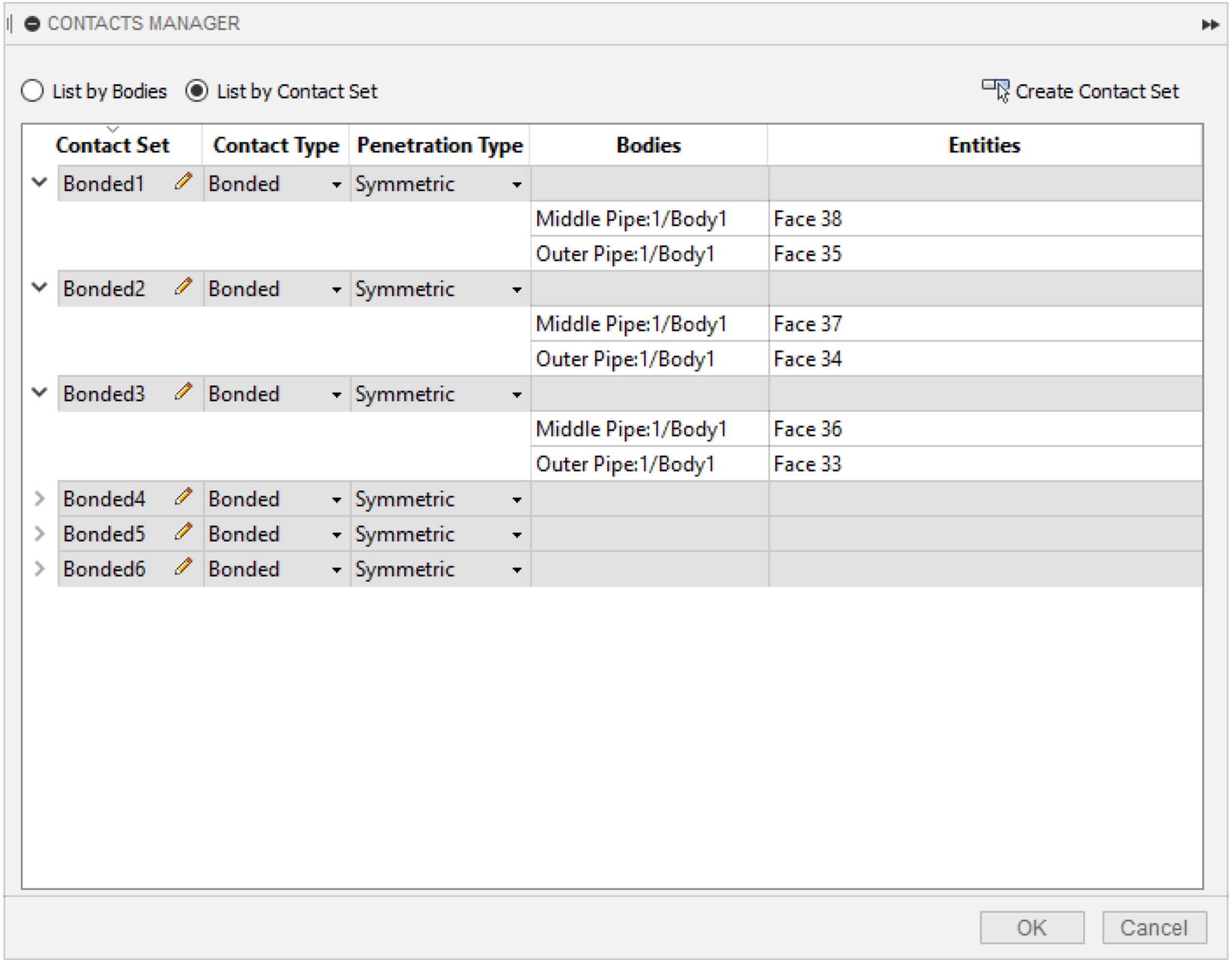


8. Select a Contact Detection Tolerance of 0.1 mm and click Generate

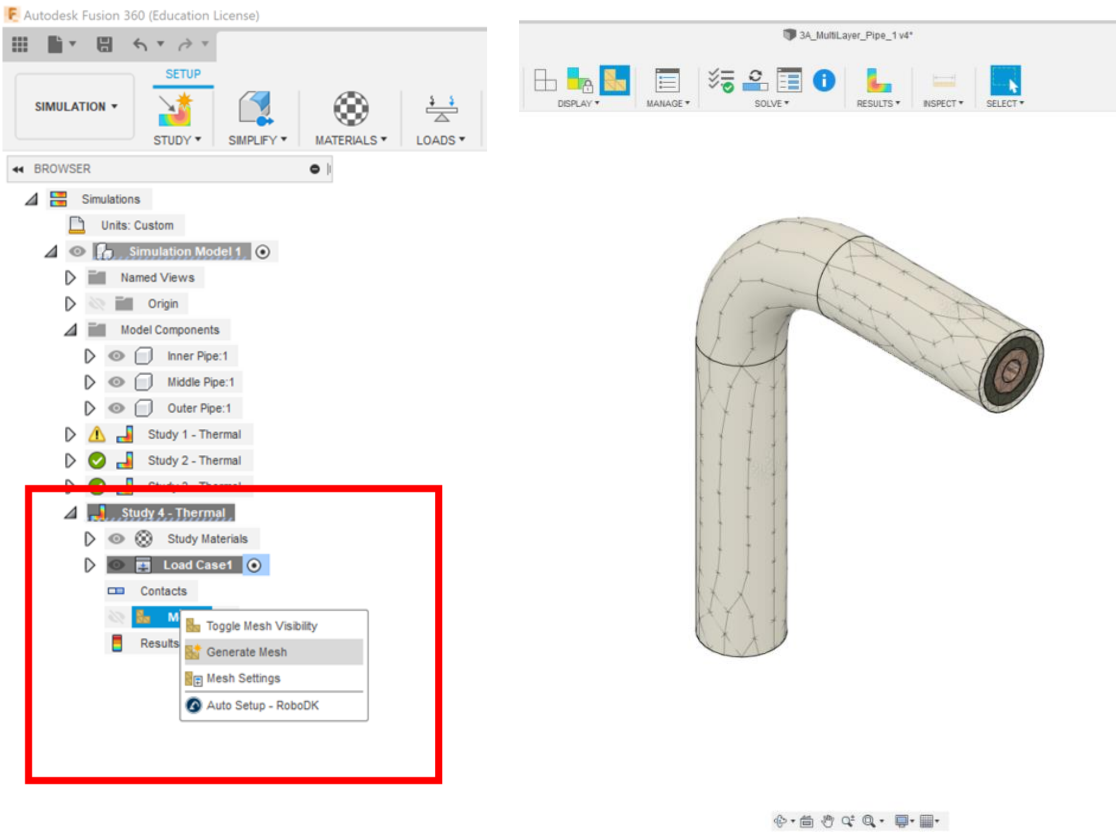


9. We can review and manage these under Manage Contacts



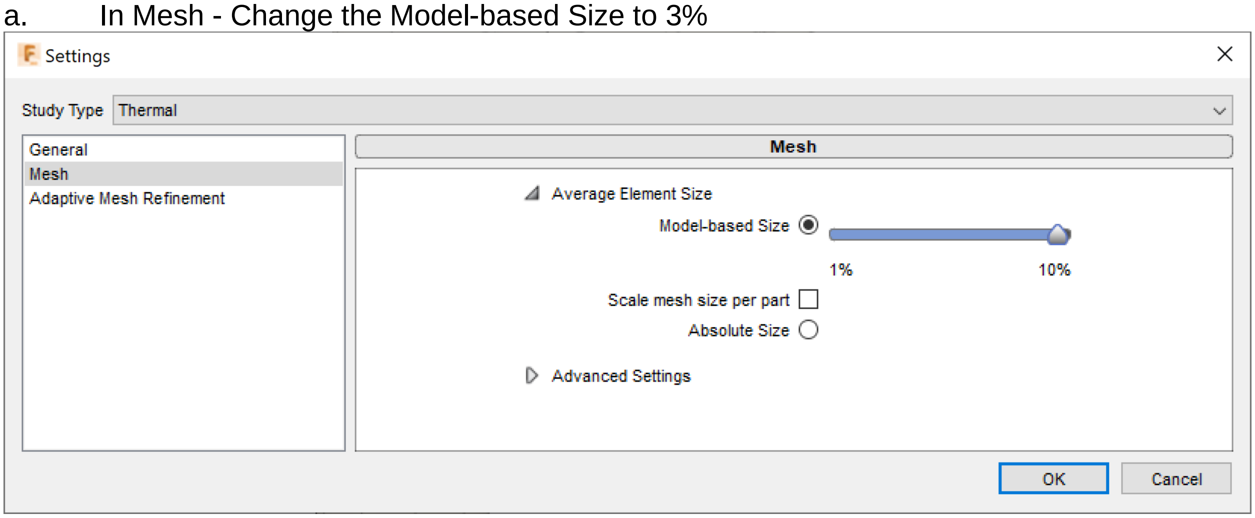


10. Review the Mesh Settings by right clicking on the mesh in the browser tree and Generate the Mesh – we can see that the mesh is quite large and may not be suitable for the simulation.

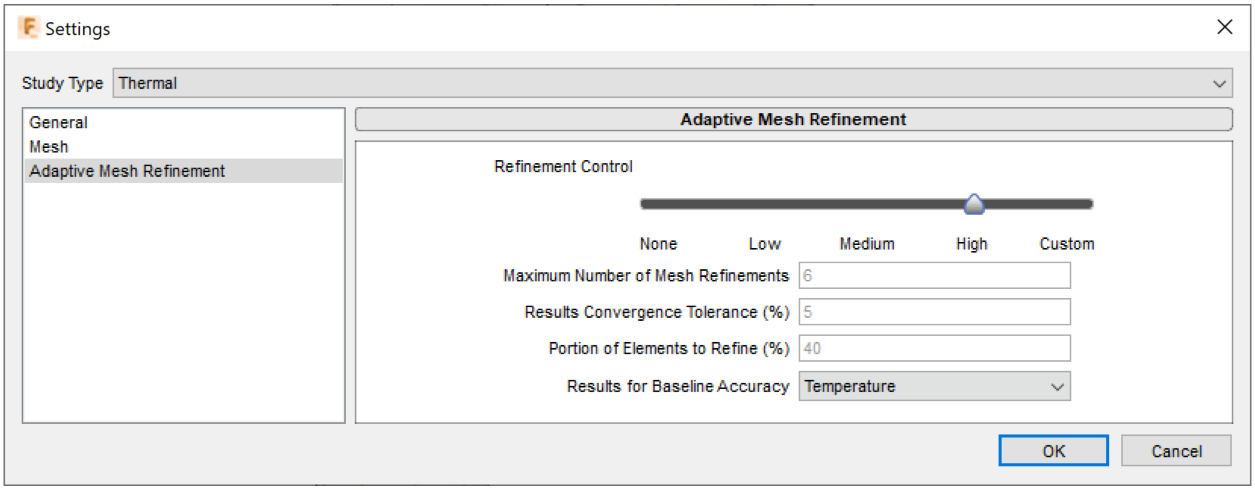


11. Modify the mesh by clicking on Settings in the Tab at the top and selecting the following:

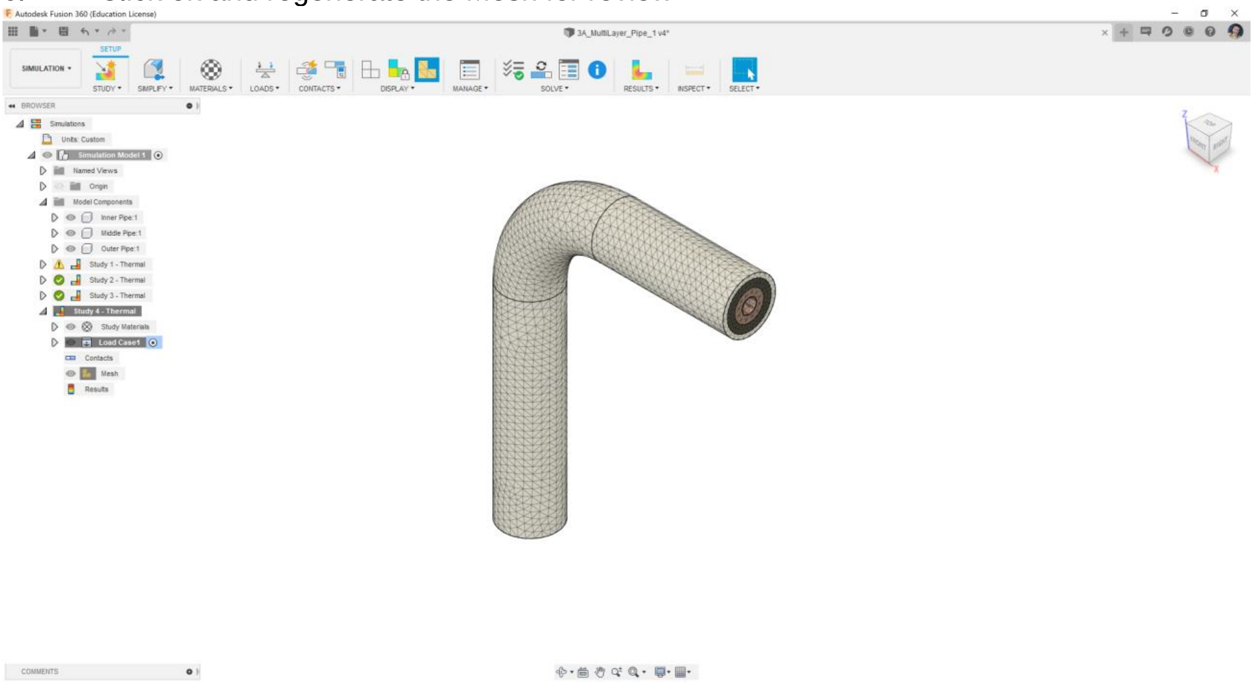
a. In Mesh - Change the Model-based Size to 3%



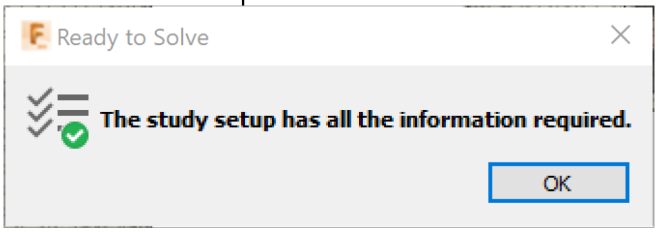
b. In Adaptive Mesh Refinement – Change Refinement Control from None to High and change the Results for Baseline Accuracy from Heat Flux to Temperature



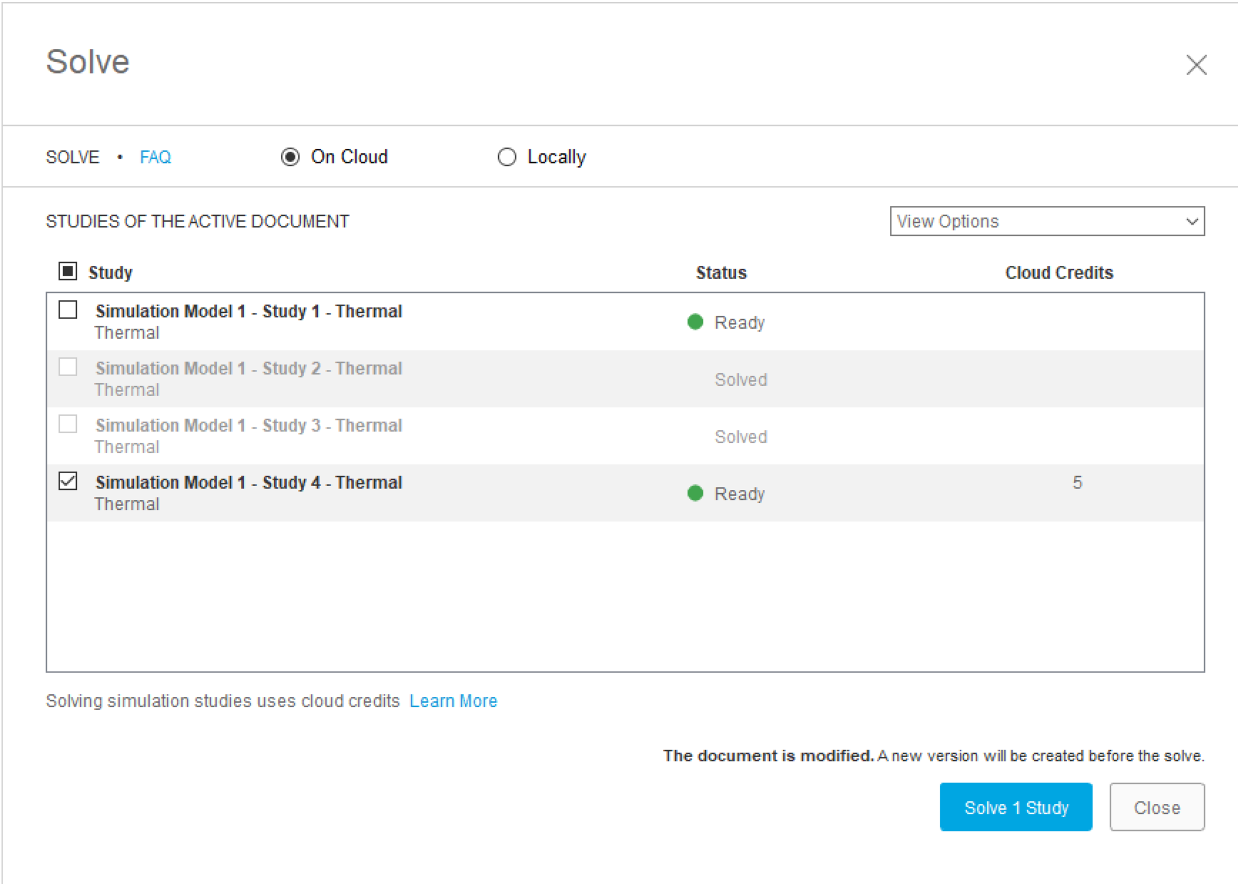
c. Click ok and regenerate the Mesh for review



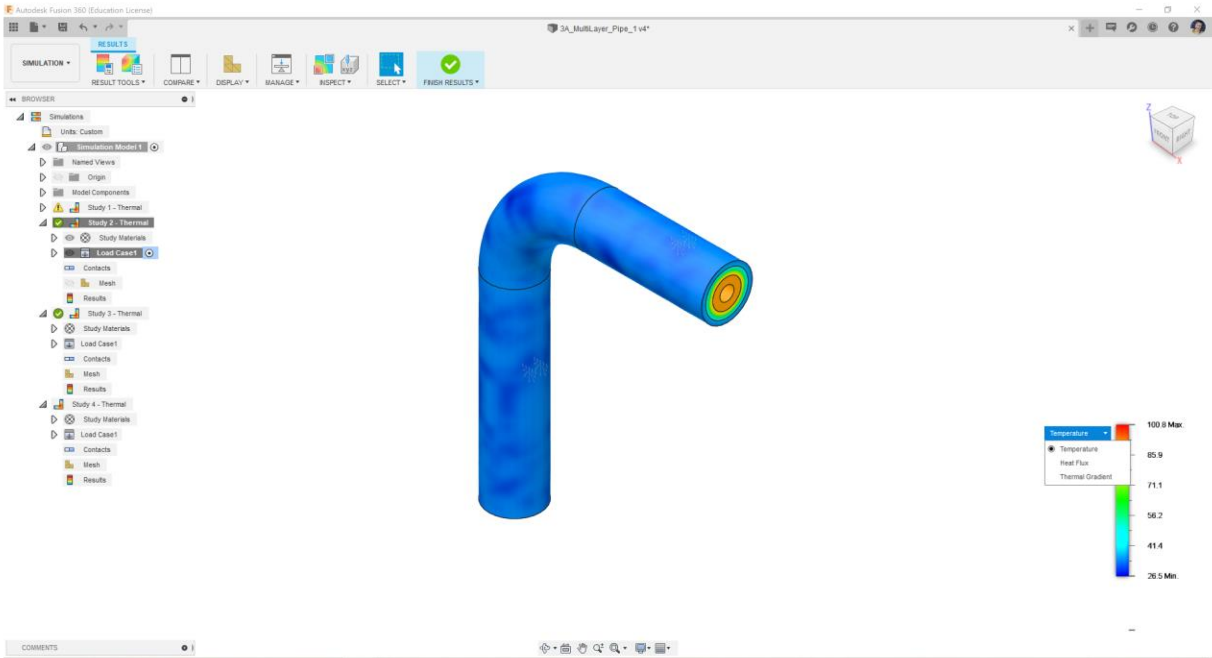
12. Check the pre-check



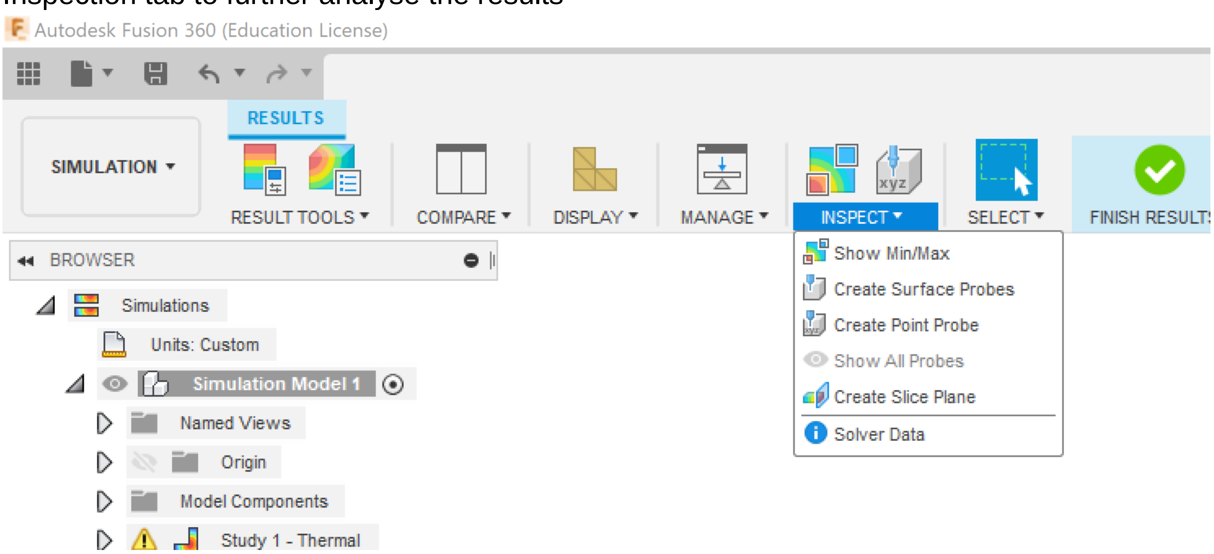
13. Solve the study on the cloud (using 5 cloud credits) or using a local solve



14. Review the results. In an thermal simulation we can review the Temperature Heat Flux and Thermal Gradient.



15. Use the inspection tools such as surface probes, point probes and slice planes in the Inspection tab to further analyse the results



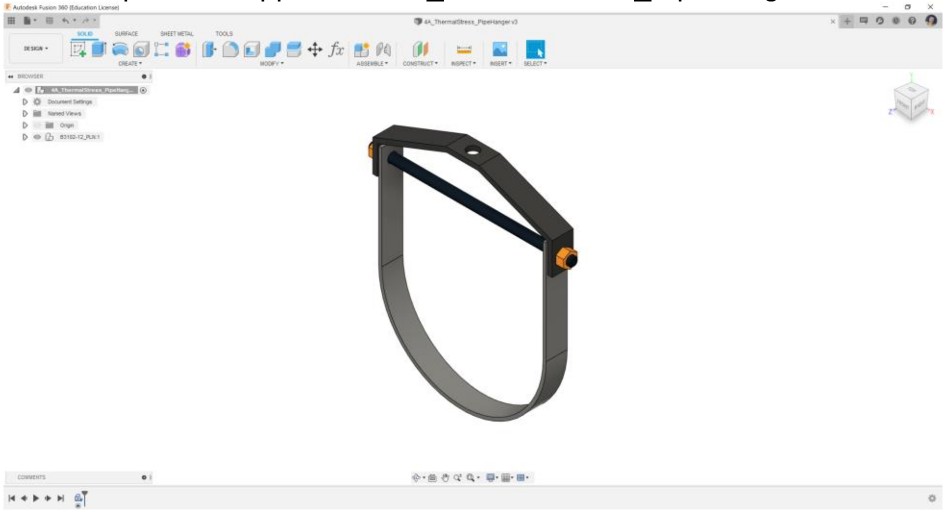
16. Investigate the pipe by selecting different materials for the Middle Pipe and see how the surface temperature changes. You can also change the thickness of the insulation. Compare your results. The supplied file 3B_MultiLayer_Pipe_2 has a thicker insulation layer.

Thermal Stress

Thermal Stress analysis combines both the Static Stress Simulation as well as the Thermal Simulation Types. This exercise will consider the thermal stress on a high temperature steam pipe hanger.

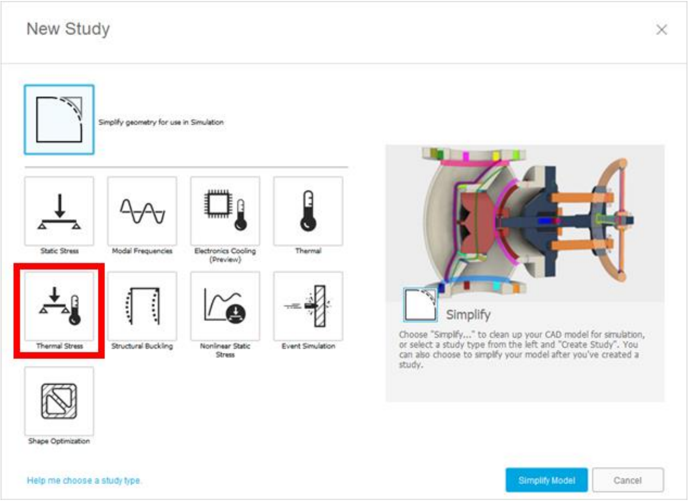
Exercise 4A

1. Open the supplied file 4A_ThermalStress_PipeHanger

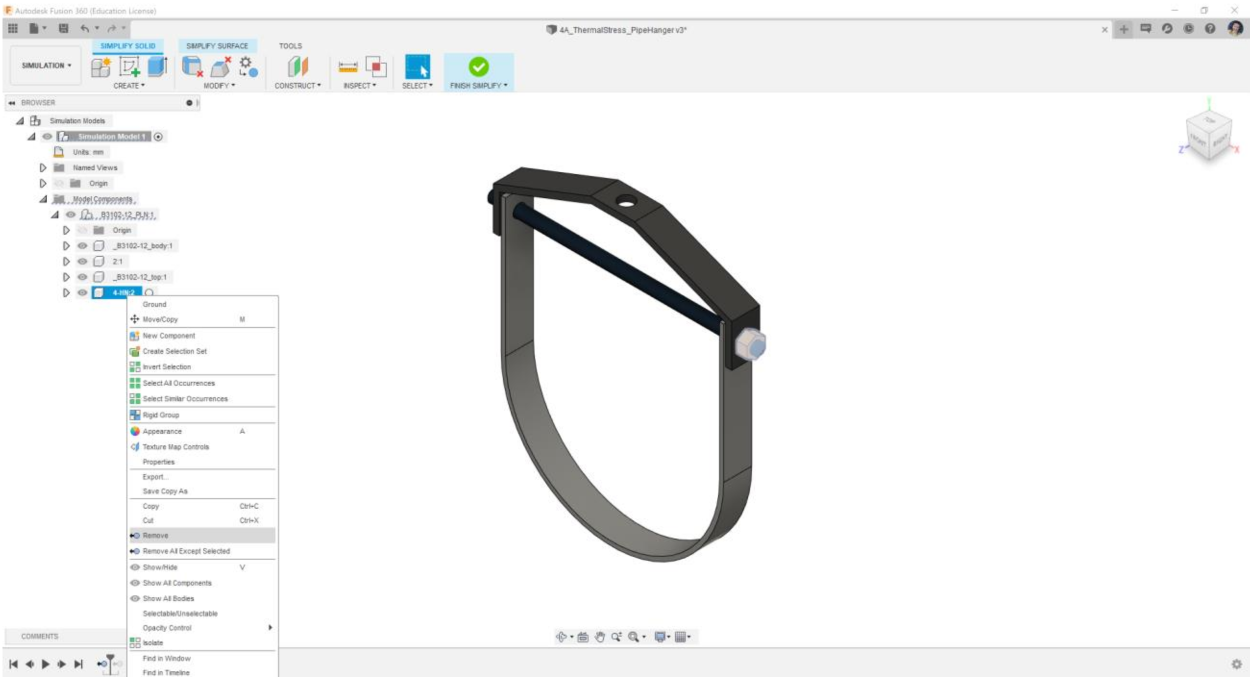


2. Move from the Design Workspace into the Simulation Workspace

3. Choose a Thermal Stress simulation from the study type menu



4. Select Simplify from the top bar to enter the Simplify Workspace. Remove the Nuts from the Model Component list to remove them from the simulation. This removes them from the simulation only, they will still exist in the Design Workspace. When you have deleted them click FINISH SIMPLIFY.



5. Set the materials for the study using the MATERIALS, change all the components to Steel Alloy.

STUDY MATERIALS

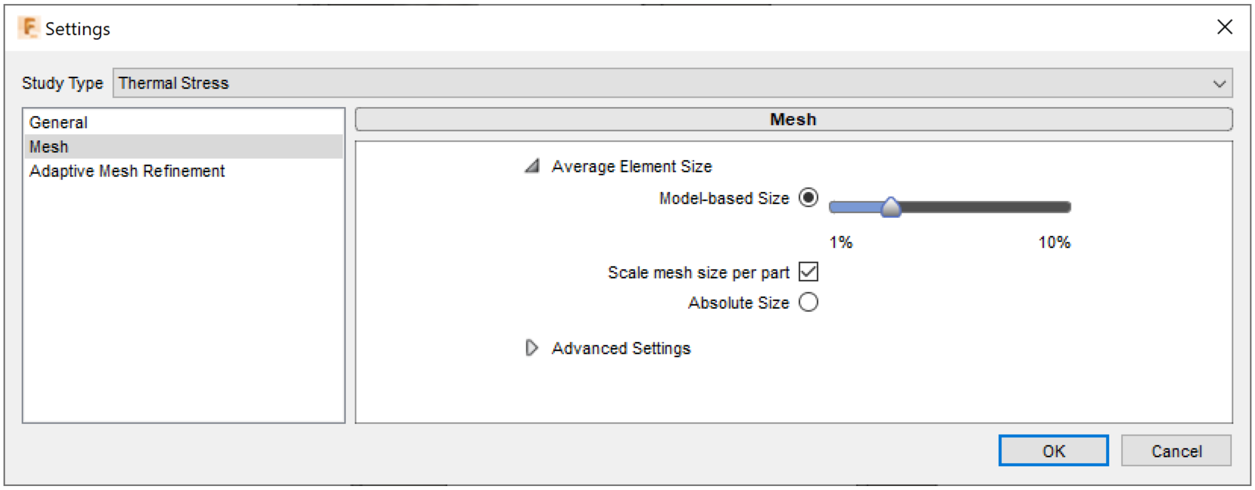
View: All Materials | Search...

Category	Name	Component	Study Materials	Safety Factor
Metal	Steel	/_B3102-12_body:1	Steel, Alloy	Yield Strength
Metal	Steel	/2:1	Steel, Alloy	Yield Strength
Metal	Steel	/_B3102-12_top:1	Steel, Alloy	Yield Strength

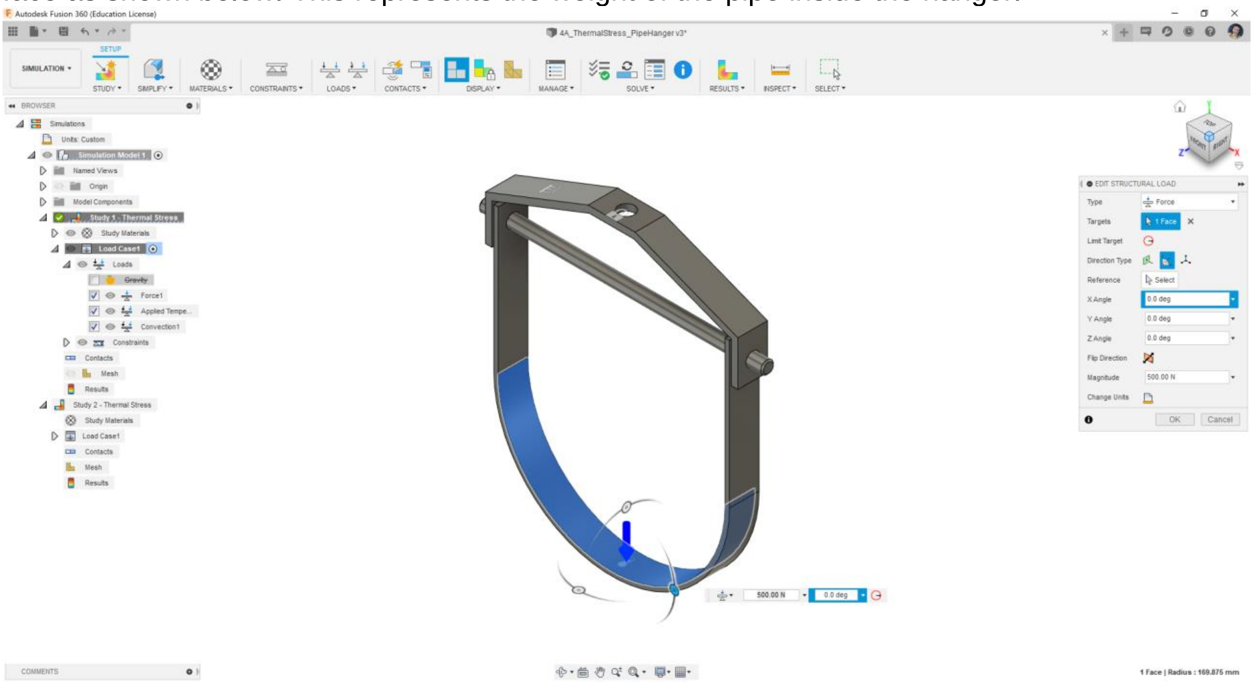
Material Library: All Libraries | << Properties

OK | Cancel

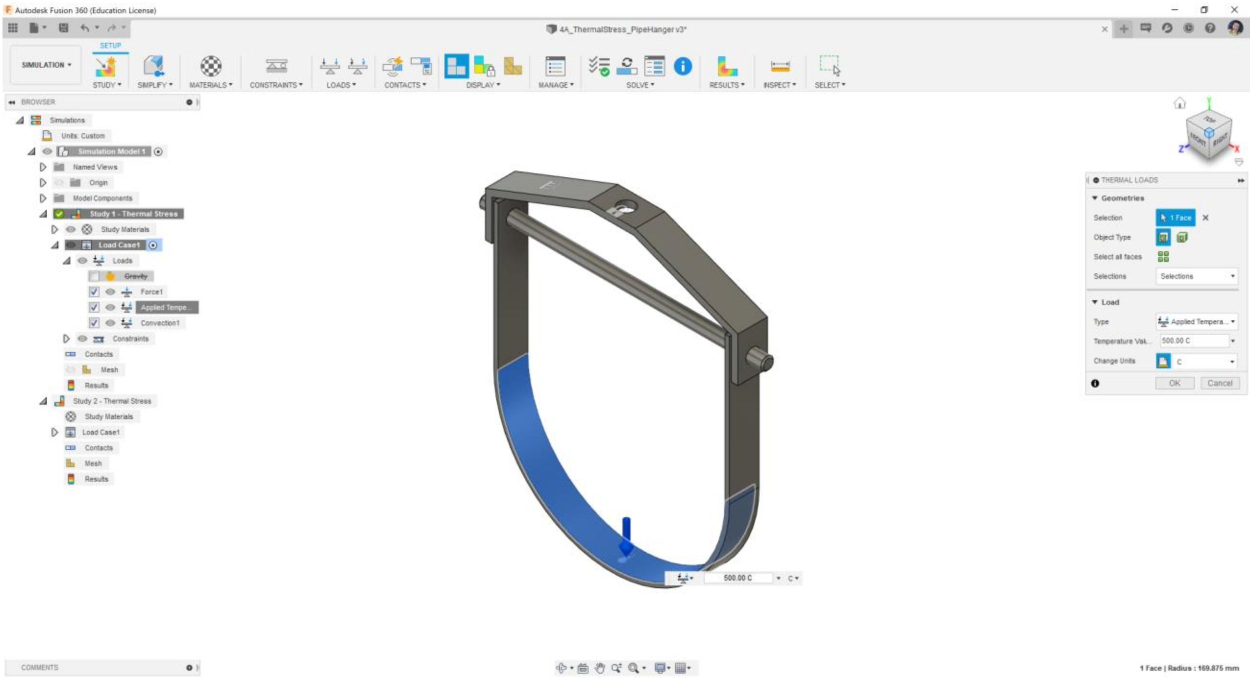
6. Change the Mesh settings using the Settings button in the top bar. Change the Model-based Size to 3%. Tick Scale mesh size per part to ON, and generate the mesh to review.



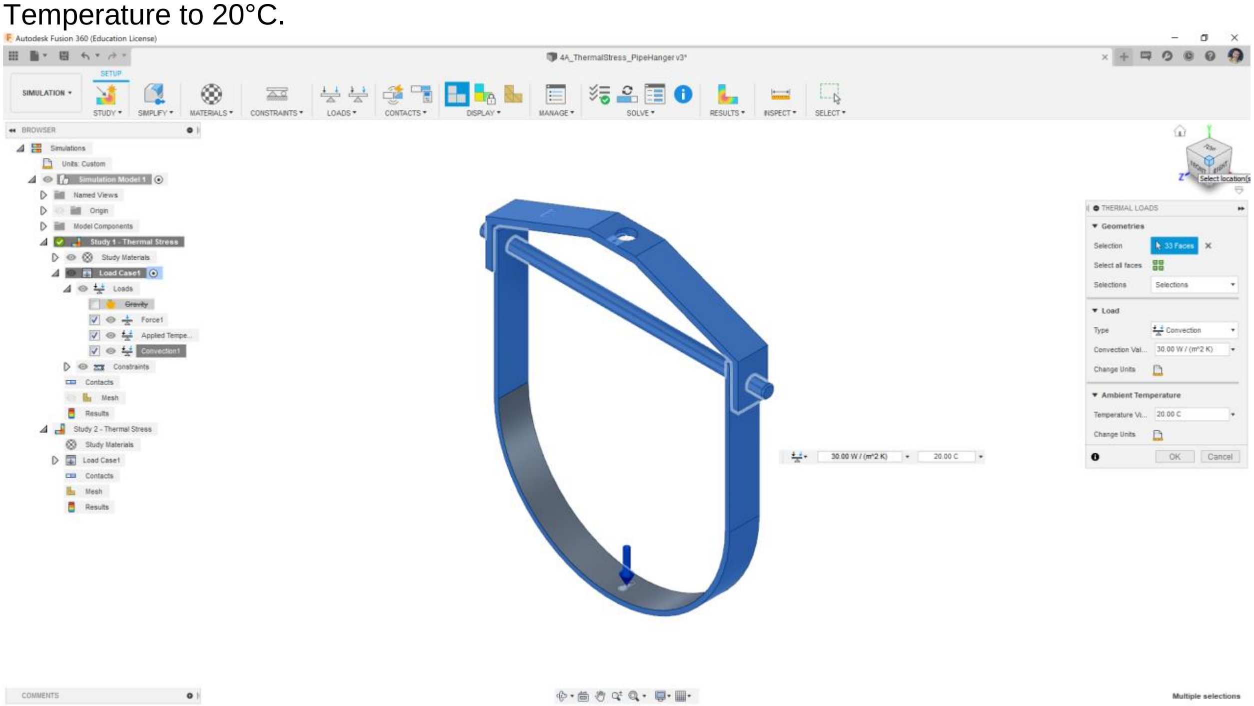
7. Apply the Loads, start with the Structural Load. Apply a Load of 500 N to the curved inside face as shown below. This represents the weight of the pipe inside the hanger.



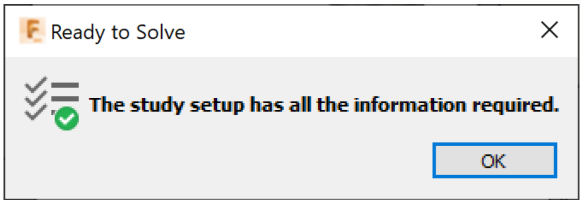
8. Apply a Thermal Load to the same inside face. Change the type to Applied Temperature and set a Magnitude of 500°C.



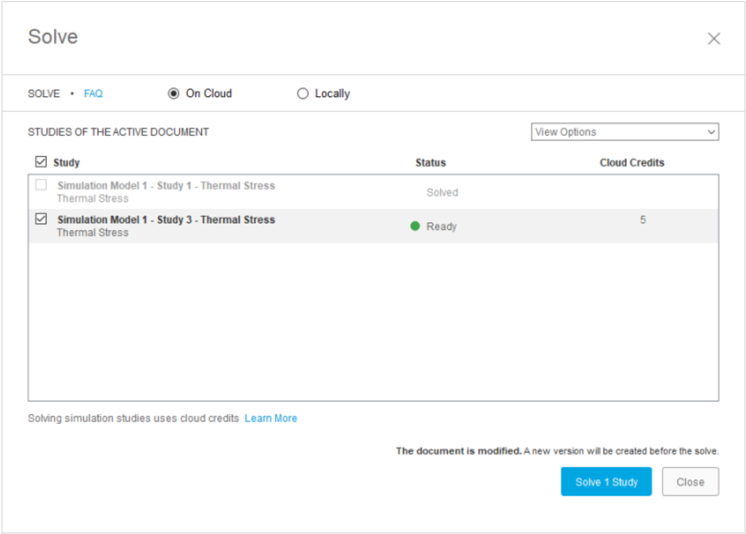
9. Add an additional Thermal Load. Select Load > Thermal Load. Change the Type to Convection. Toggle on Select all Faces and choose all the faces of all the parts (except the face we have already applied a load to). Set the convection value to 30 W / (m^2 K) and the Ambient Temperature to 20°C.



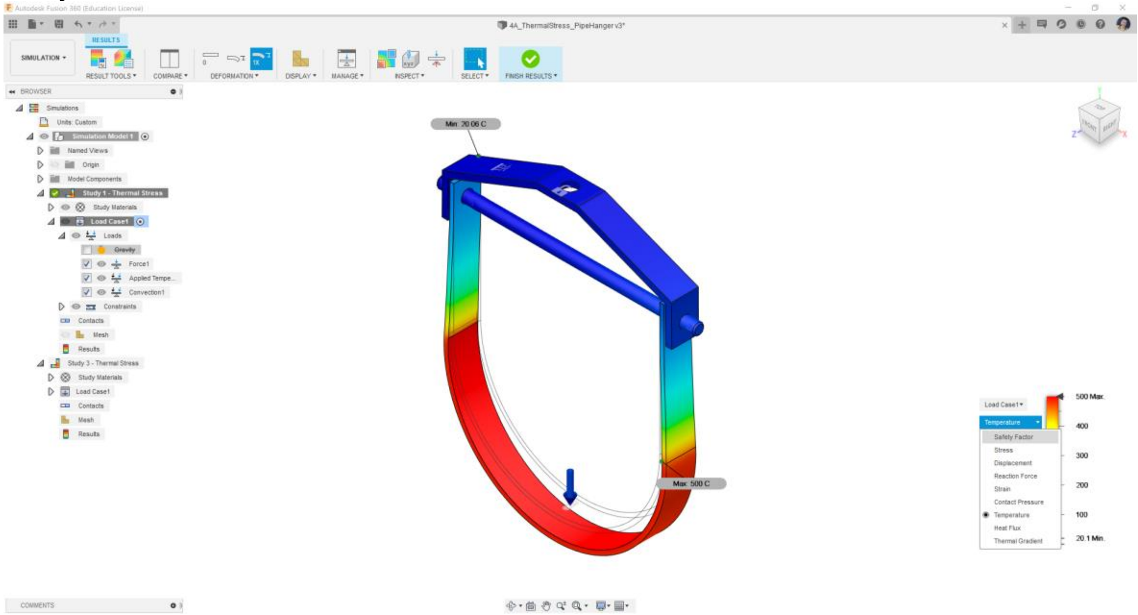
10. Check the pre-check



11. Solve the simulation on the cloud (requires 5 cloud credits) or solve locally.



12. Review the results. With a Thermal Stress Analysis you can review both the results that are available in a Static Stress simulation as well as those available when performing a Thermal analysis.



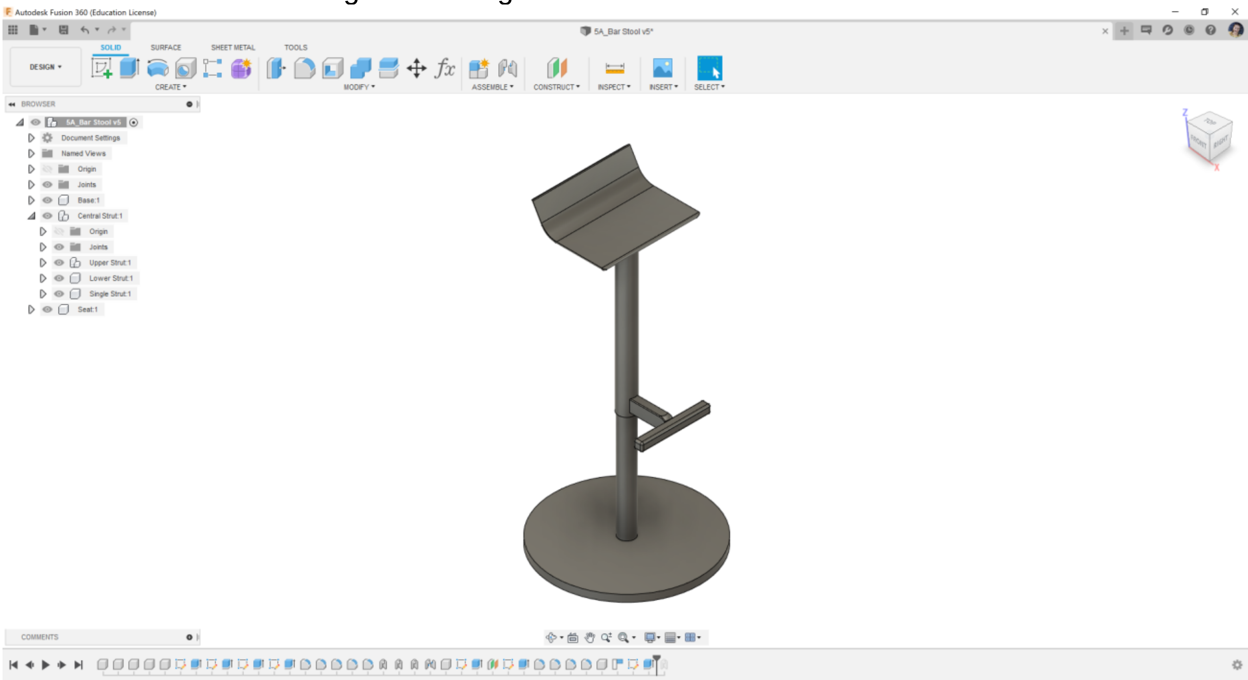
Structural Buckling

Buckling is when there is a sudden change in the shape of an object when subjected to compressive forces. This usually happens when the length is long compared to the cross-sectional area of the object. You may have experienced this when pushing on the end of a long tube, it suddenly pushes out to the side.

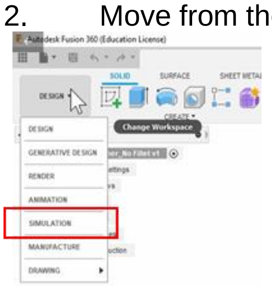
There are two ways to approach buckling in Fusion 360, the results given are in the form of a multiplier based on the inputted force. The first approach is to apply a known load, and the result will be a multiplier of that load at which the object will buckle. Alternatively, we can apply a load of 1 N and the output will be the load at which the part will buckle.

Exercise 5A

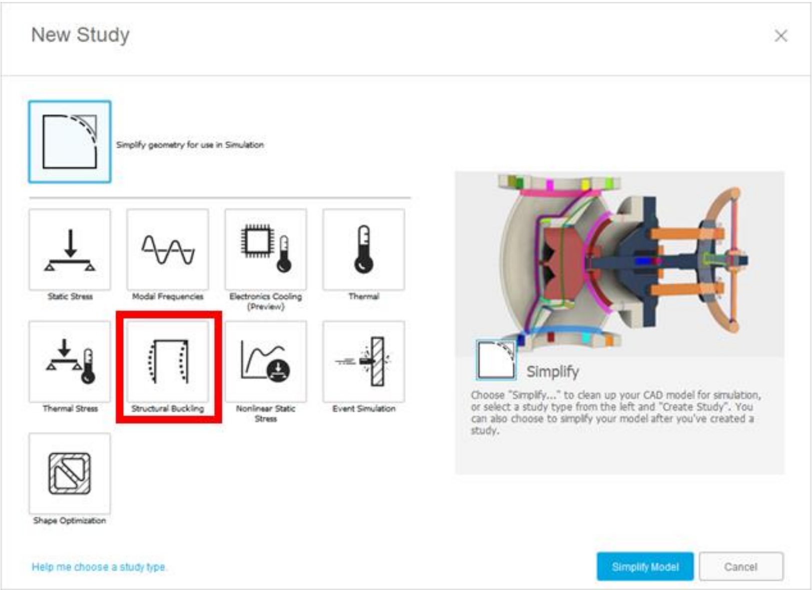
1. Open the supplied file 5A_Bar_Stool. We are going to investigate some different materials for the long tube to see what might be suitable for the design. Our target load is that the stool must be able to take a weight of 110 kg.



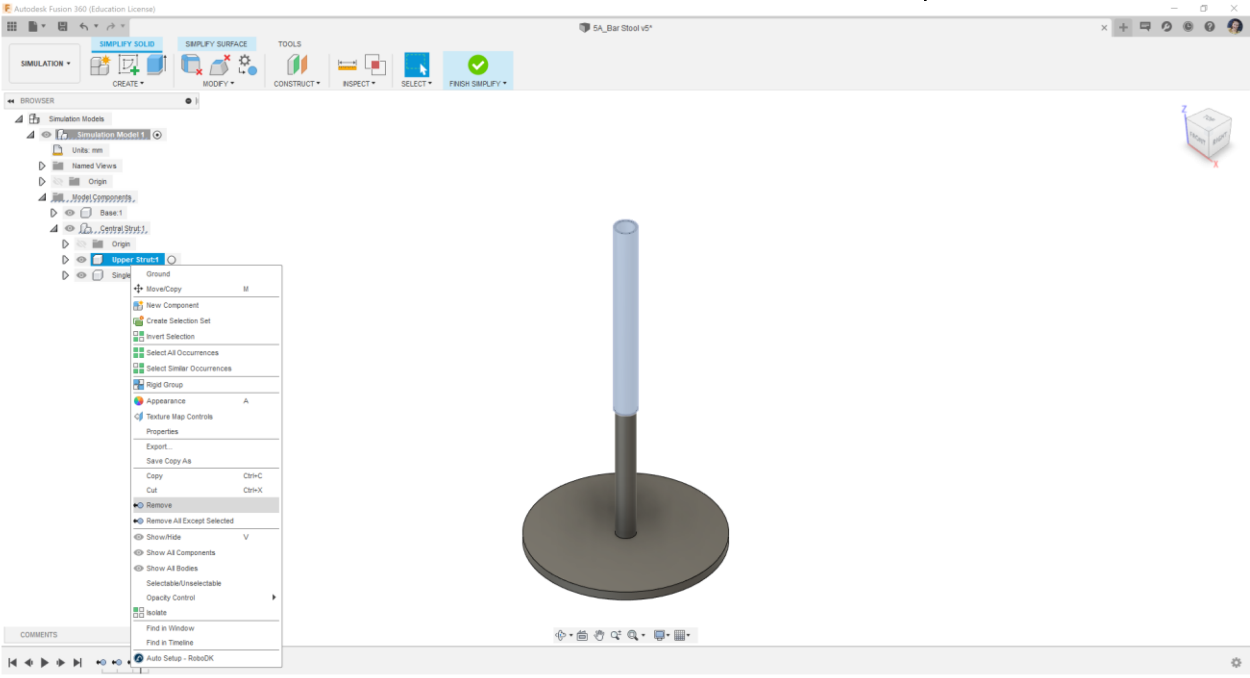
2. Move from the Design Workspace into the Simulation Workspace



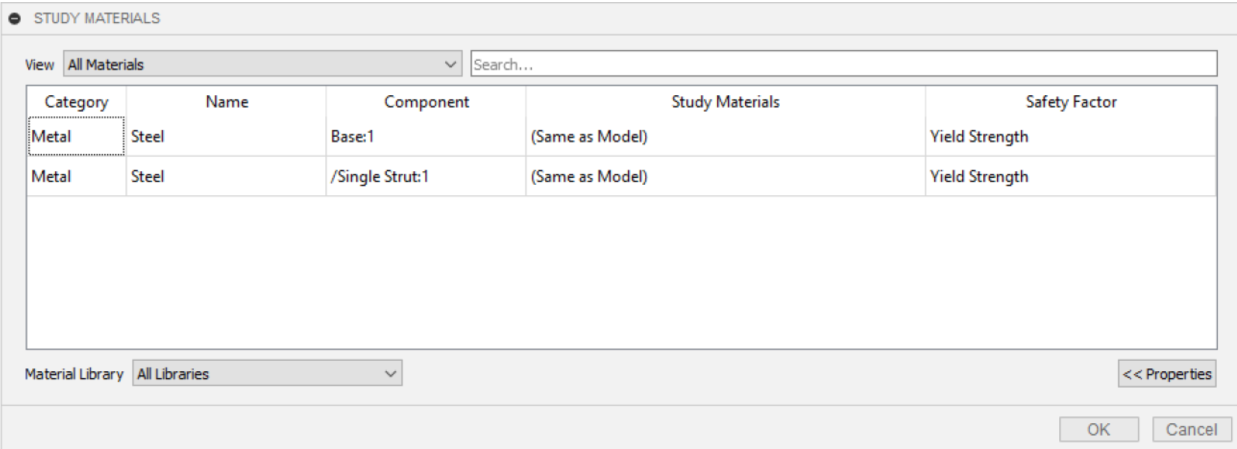
3. Choose a Structural Buckling simulation from the study type menu



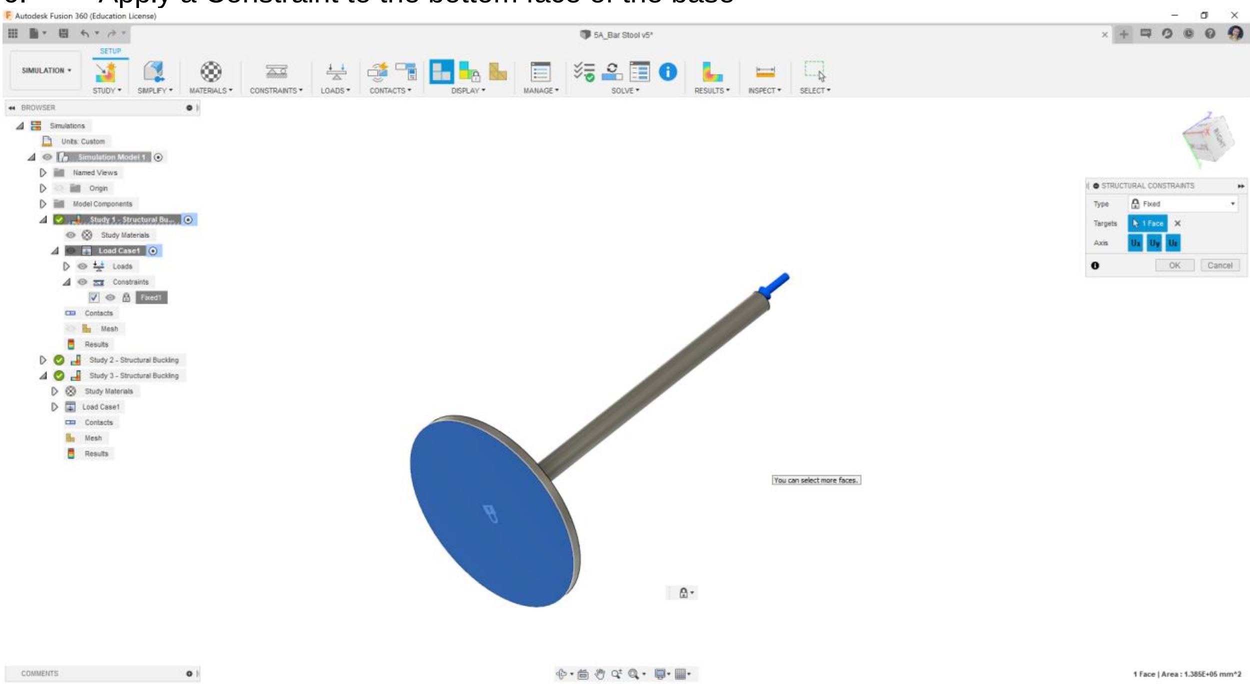
4. In the Simplify Workspace remove the Components: Seat; Upper Strut; and Lower Strut. You can do this by right clicking on them in the browser tree and selecting remove. When you have finished click FINISH SIMPLIFY to return to the Simulation Workspace.



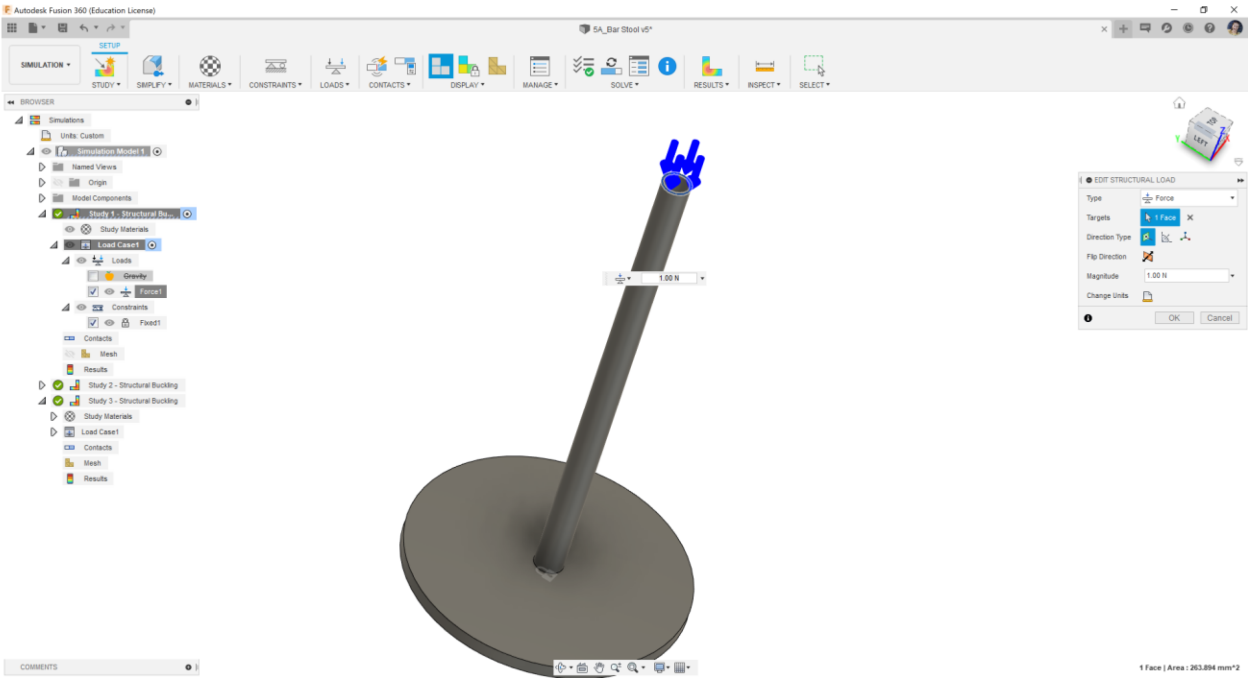
5. Set the Materials for the first study to Steel (Same as Model)



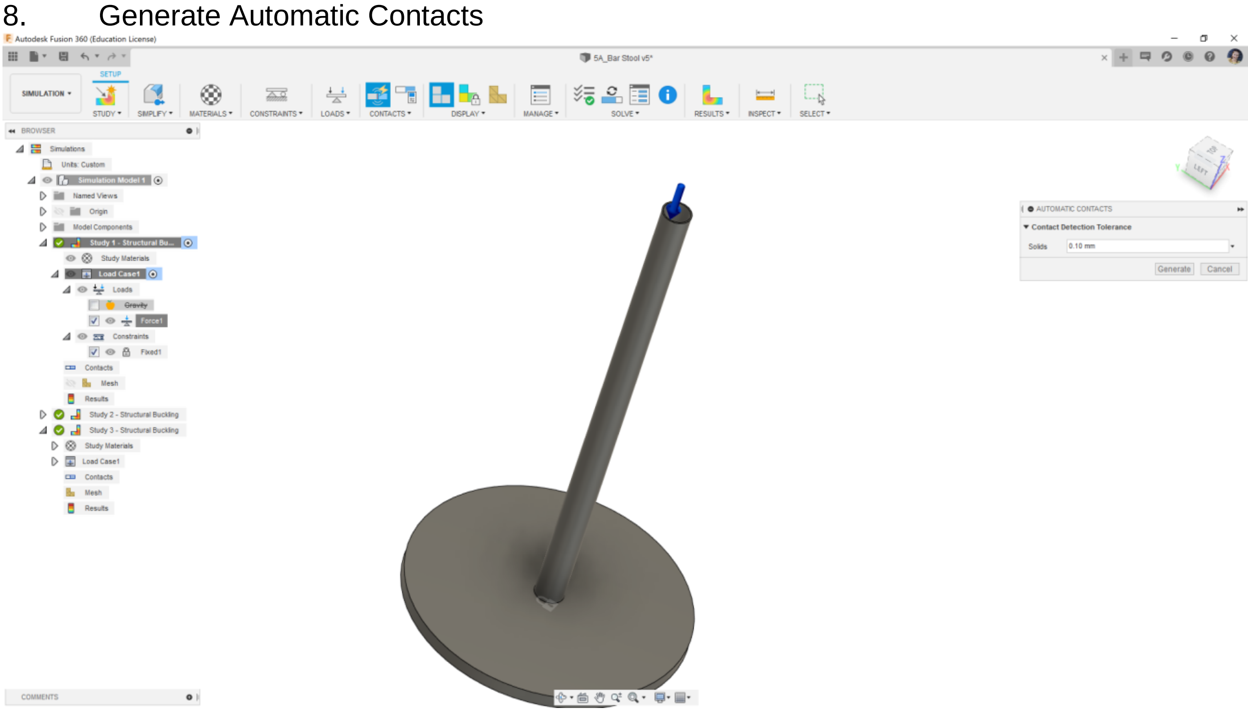
6. Apply a Constraint to the bottom face of the base



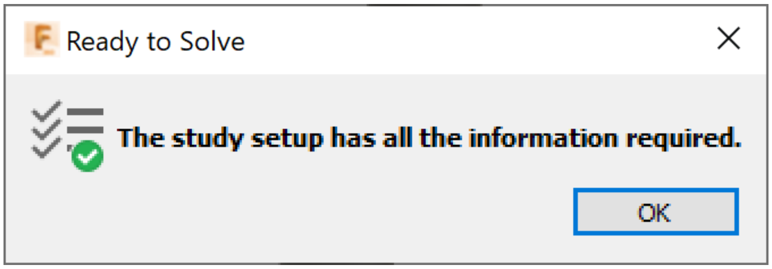
7. Apply a load to the top of the tube – use a Force load of 1 N. This will give our results so we can find the load at which the chair will buckle.



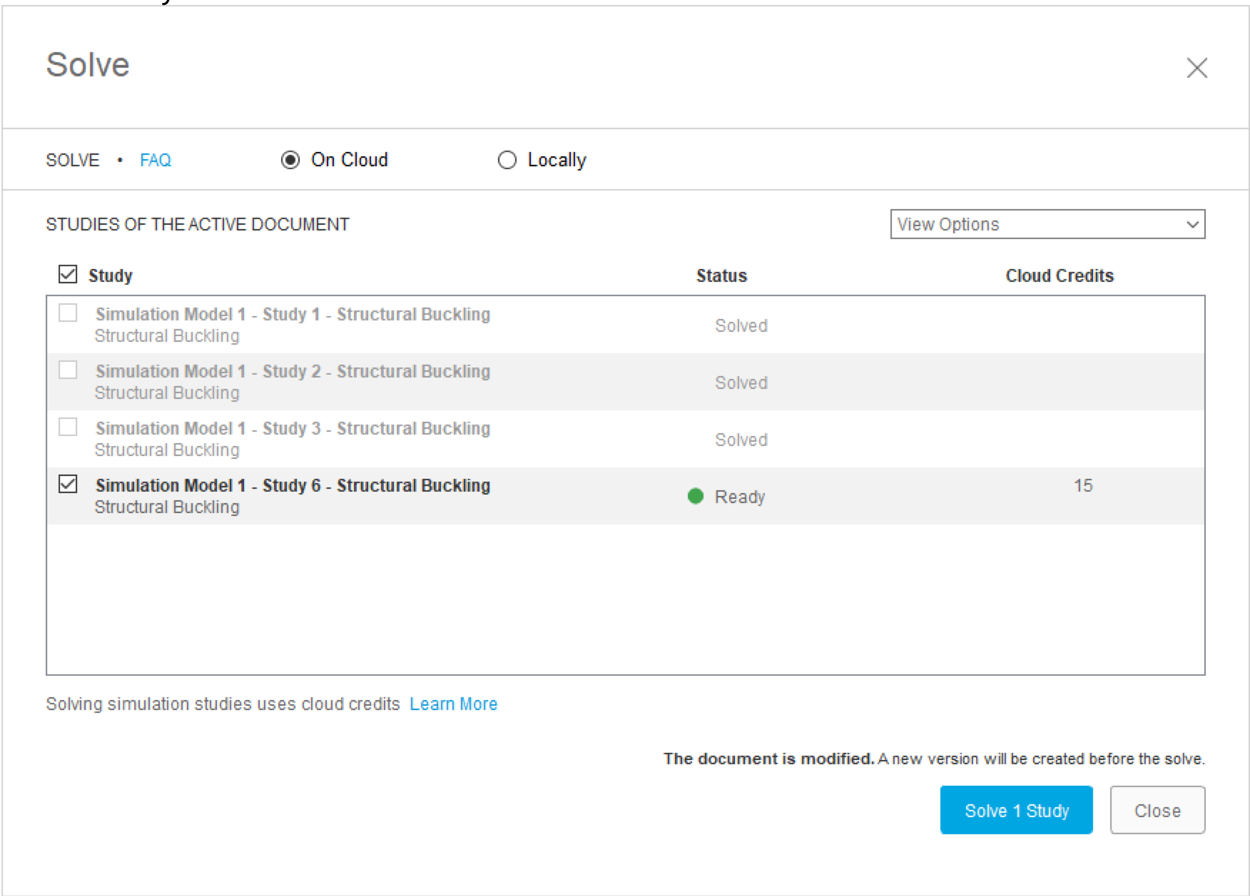
8. Generate Automatic Contacts



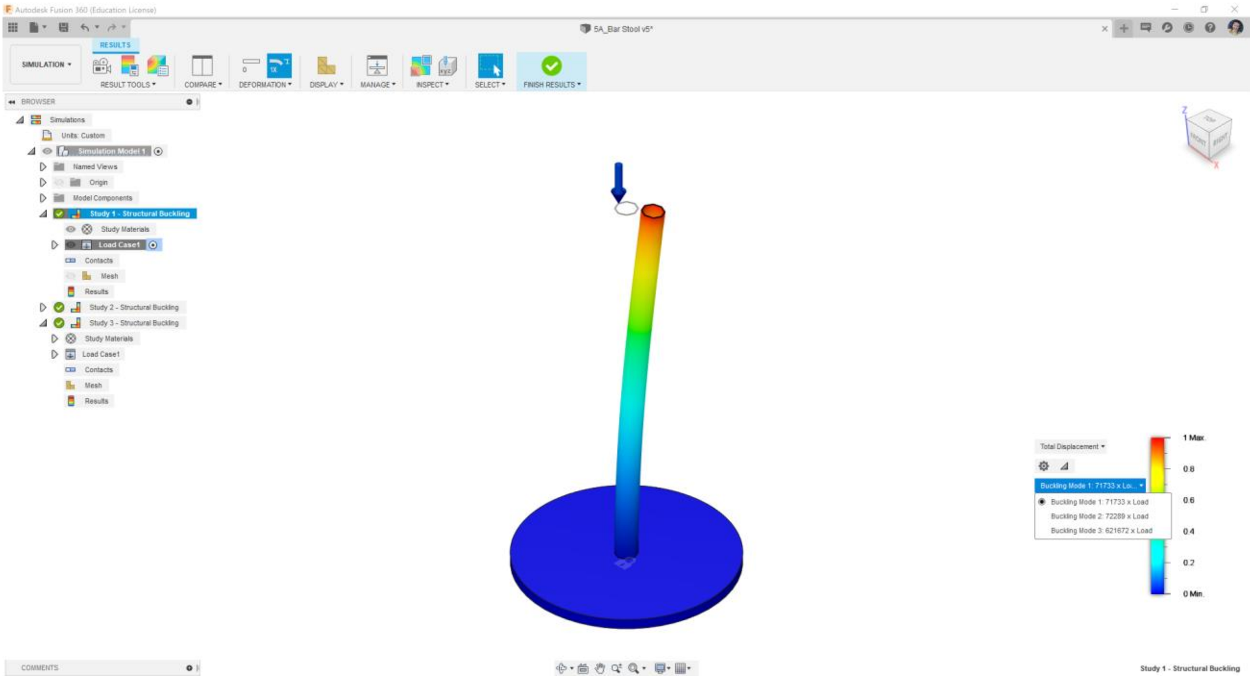
9. Check the Pre-Check



10. Solve the Simulation on the Cloud (requires 15 cloud credits) this study type cannot be solved locally



11. Review the Results

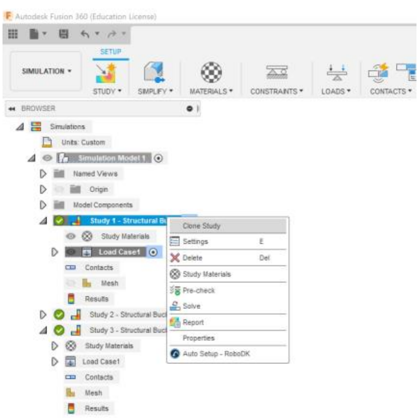


12. Clone the Study by right clicking on the study and selecting clone study, do this twice so you have 3 studies in total. Change the material in the 2 new studies to:

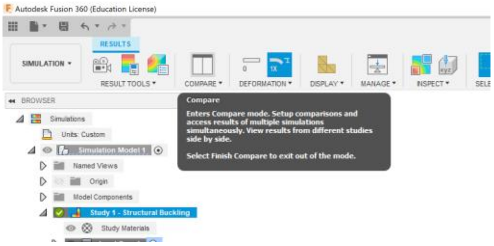
a. Aluminium for the tube

b. ABS Plastic for the tube

Leave the base as Steel as it is not part of the study.

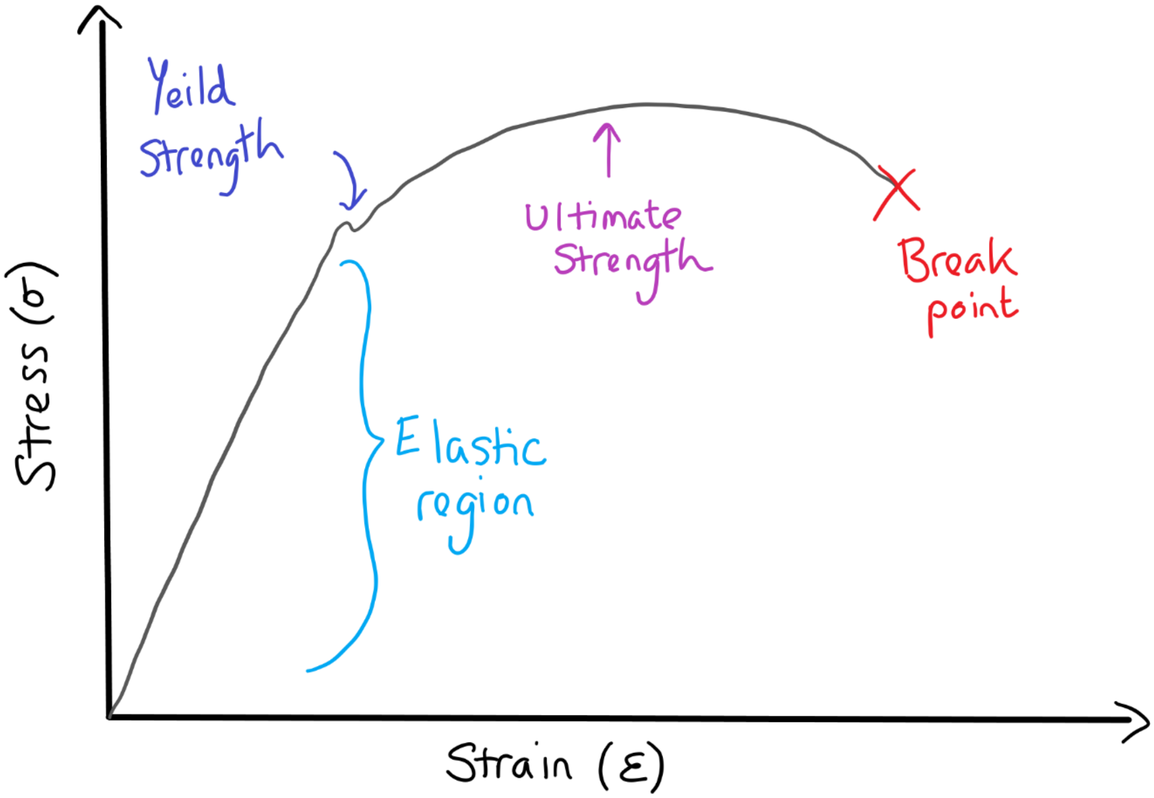


13. Compare the results of the different materials to determine which of the three materials is most suitable. Remember that the chair needs to take a load of 110 kg. Use the Compare option in the top bar to view multiple results at the same time.


Autodesk Fusion 360 (Education License)
RESULTS
SIMULATION
RESULT TOOLS
COMPARE
DEFORMATION
DISPLAY
MANAGE
INSPECT
BROWSER
Simulations
Units: Custom
Simulation Model 1
Named Views
Origin
Model Components
Study 1 - Structural Buckling
Study Materials
Compare
Enters Compare mode. Setup comparisons and access results of multiple simulations simultaneously. View results from different studies side by side.
Select Finish Compare to exit out of the mode.

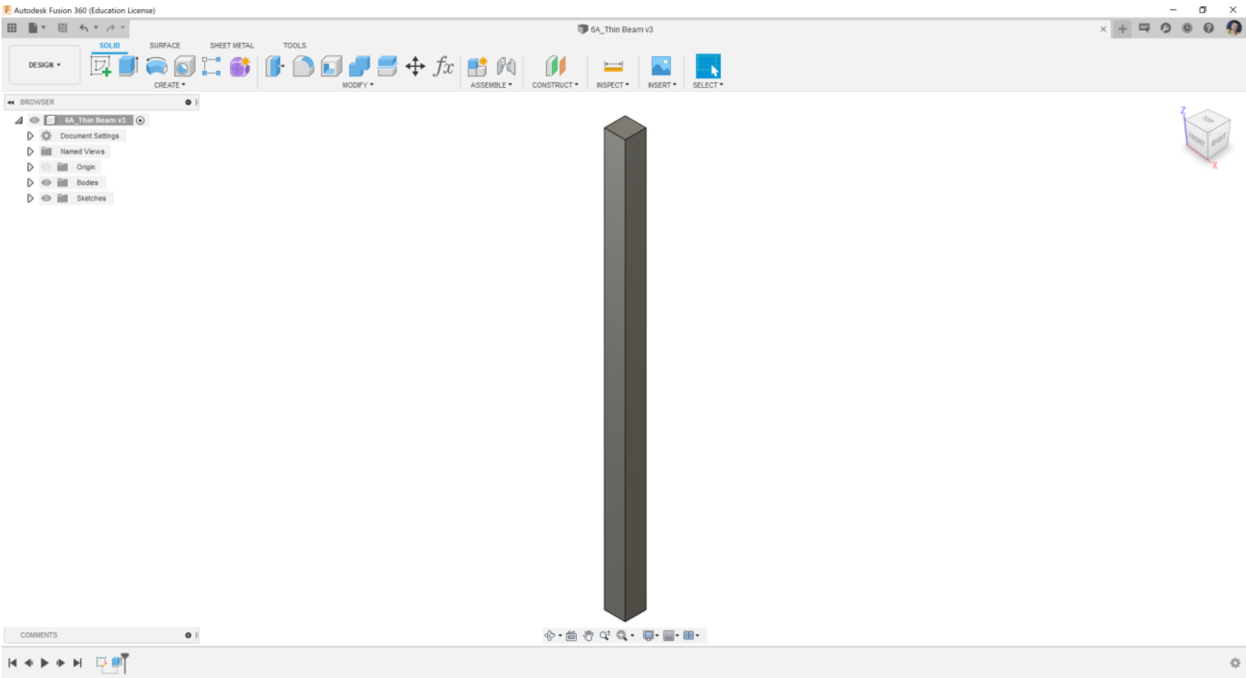
Nonlinear Static Stress

Many materials do not behave in a fully elastic way. i.e. when a load is applied, they do not return to their original shape. Most materials follow a elastic-plastic behaviour and have the following characteristic stress strain curve.



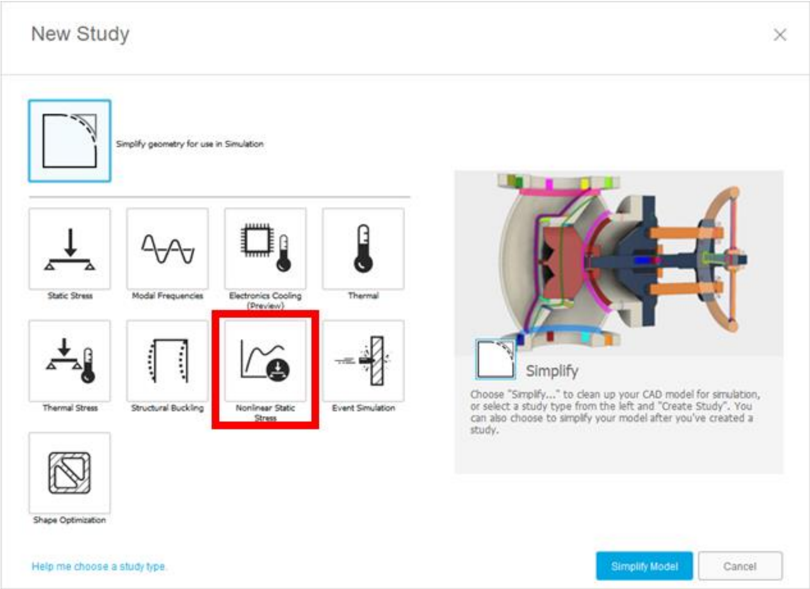
Exercise 6A

1. Open the supplied file 6A_Thin_Beam

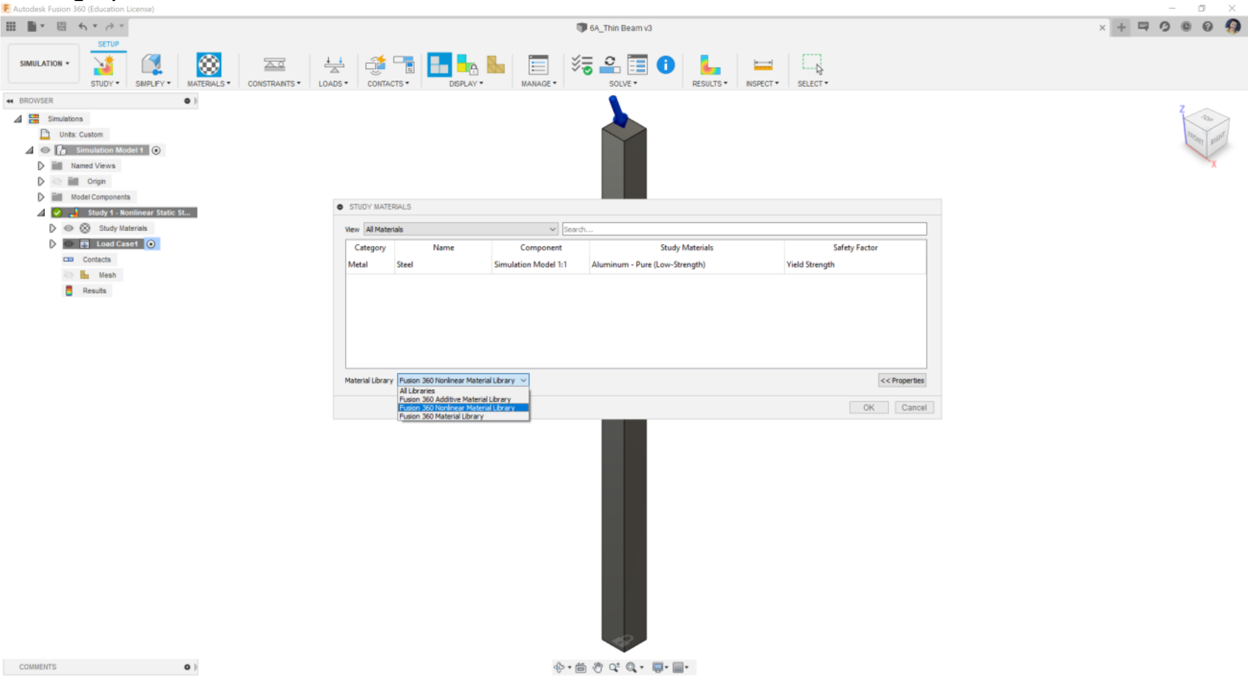


2. Move from the Design Workspace into the Simulation Workspace

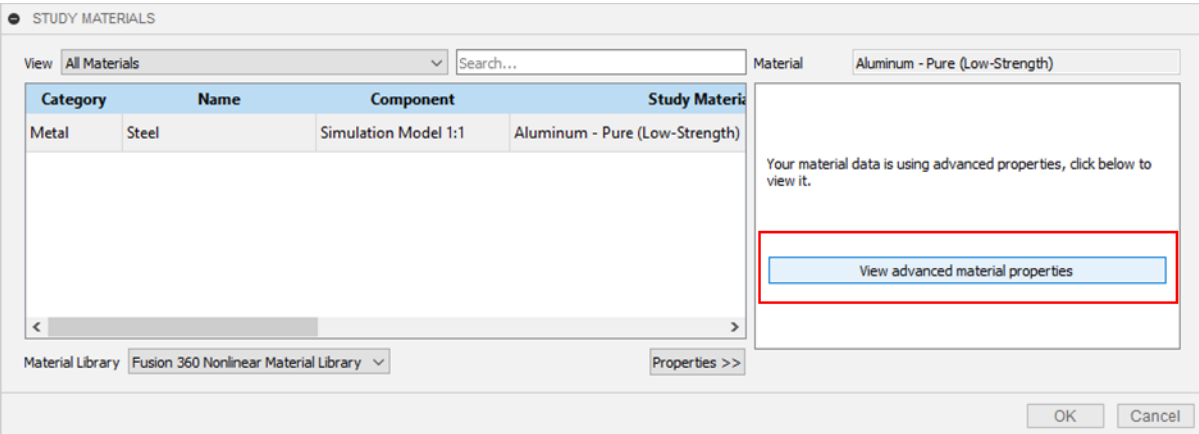
3. Choose a static stress simulation from the study type menu



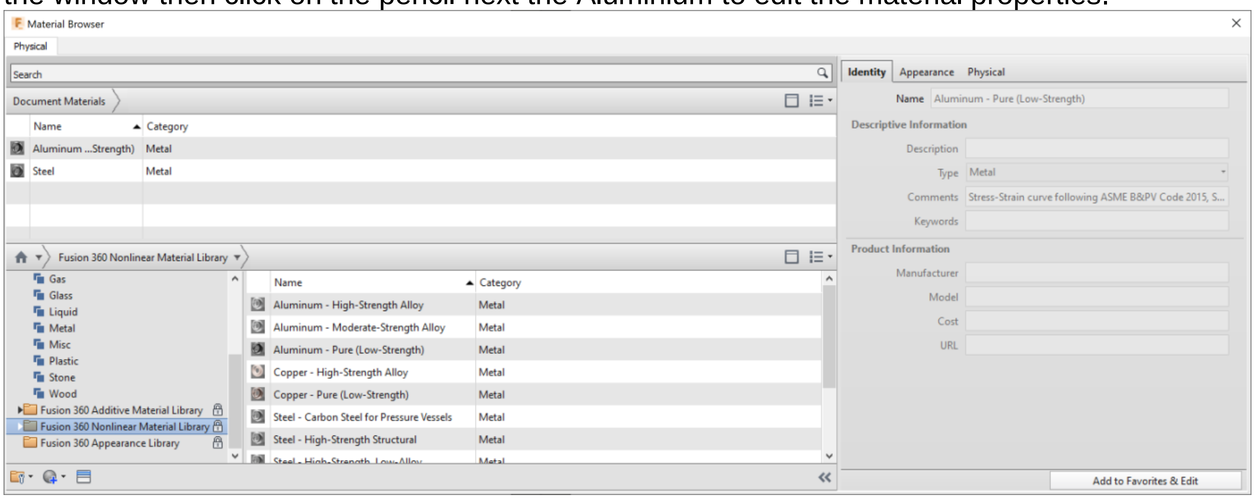
4. Set up the materials. Choose from the Fusion 360 in built library of Nonlinear Materials. This leaves far fewer materials to choose from. Set the material to Aluminium – Pure (low Strength).



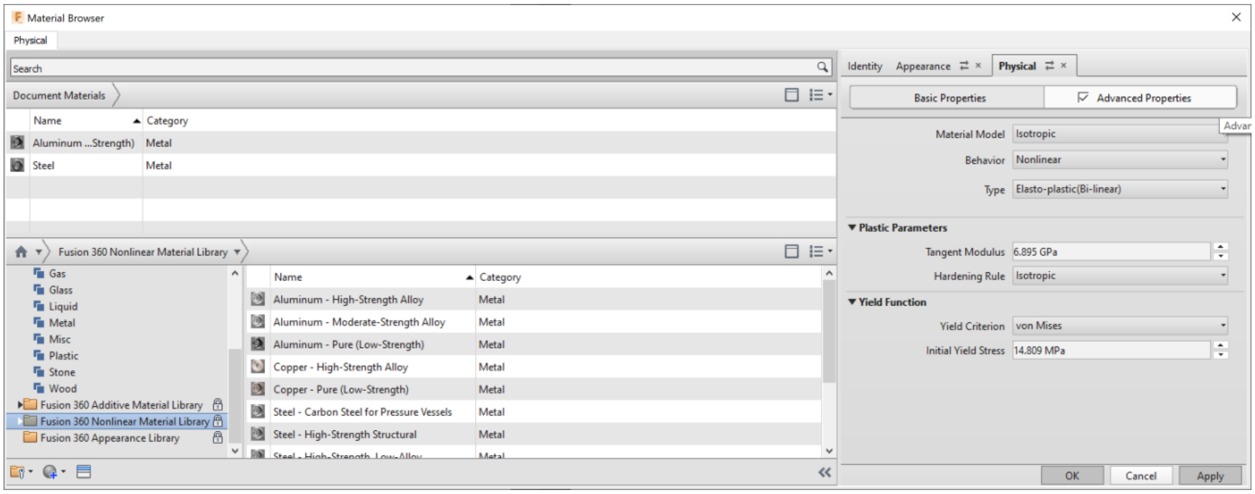
5. Select Properties and then click view advanced material properties.



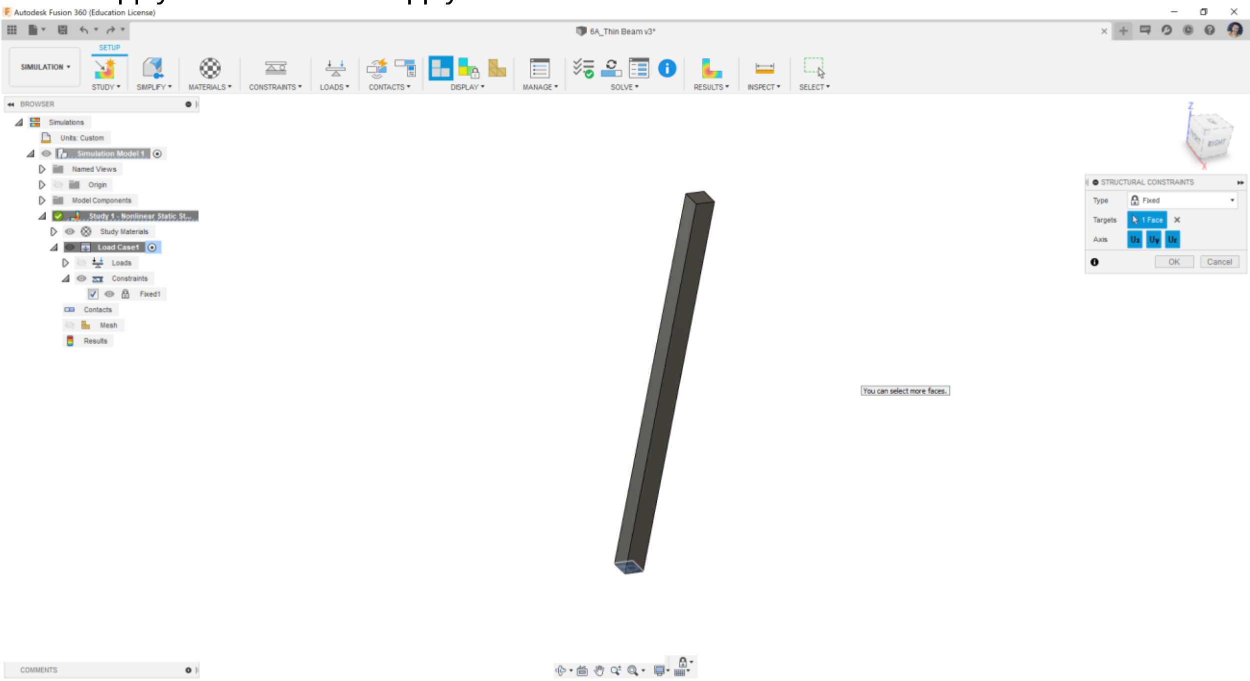
6. This takes you to the Materials Properties Browser. Click on the double arrow to expand the window then click on the pencil next the Aluminium to edit the material properties.



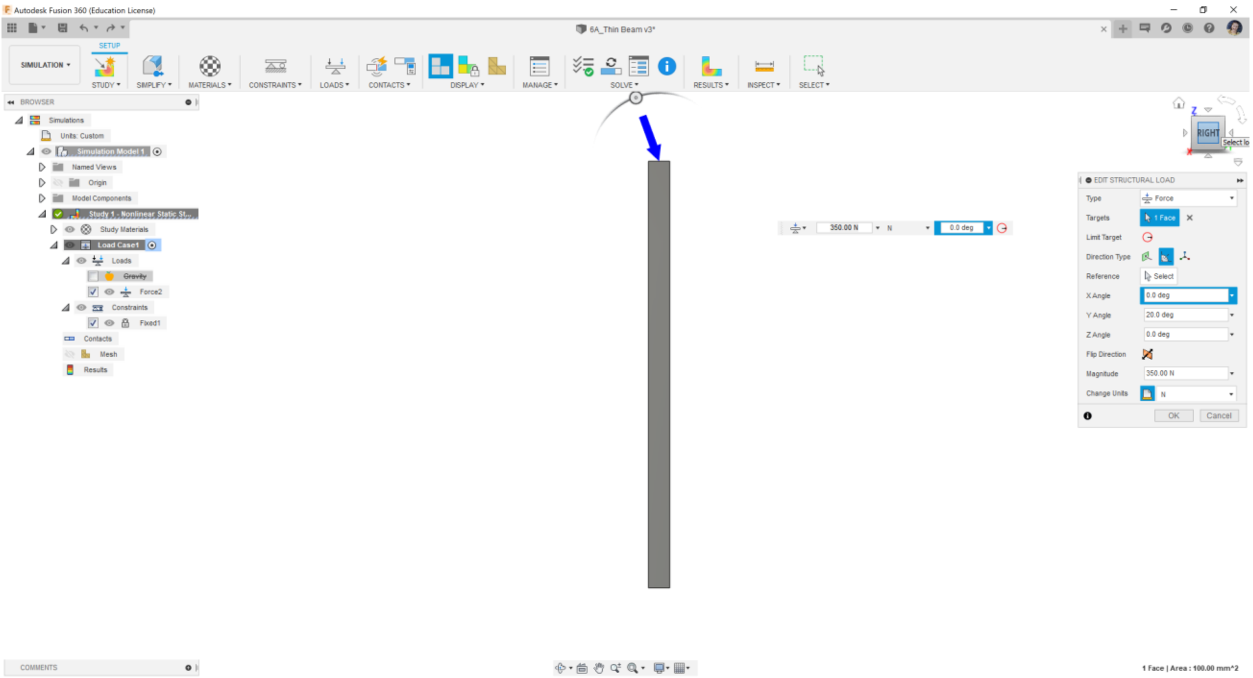
7. Click on the tab called Physical and the Advanced Properties. Here the Material Model, Behaviour and Type can be changed. Change the type to Elasto-plastic(Bi-Linear) this matched the curve shown above. Click Ok, then close the panel.



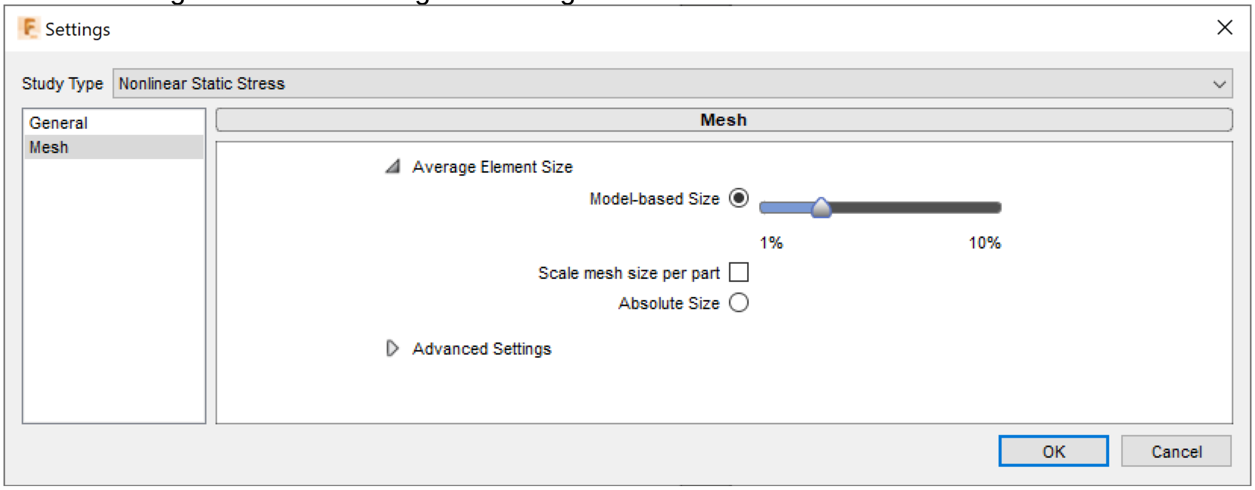
8. Apply the constraints. Apply a fixed constraint to the bottom face.



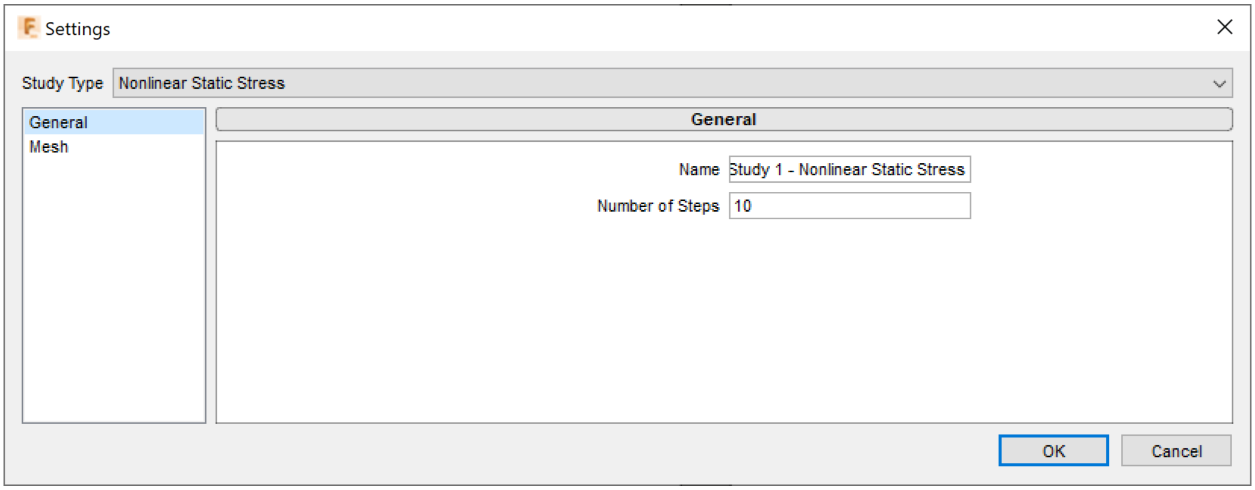
9. Apply a Load to the top face. Apply a Force load of 350 N at an angle of 20° then click ok.



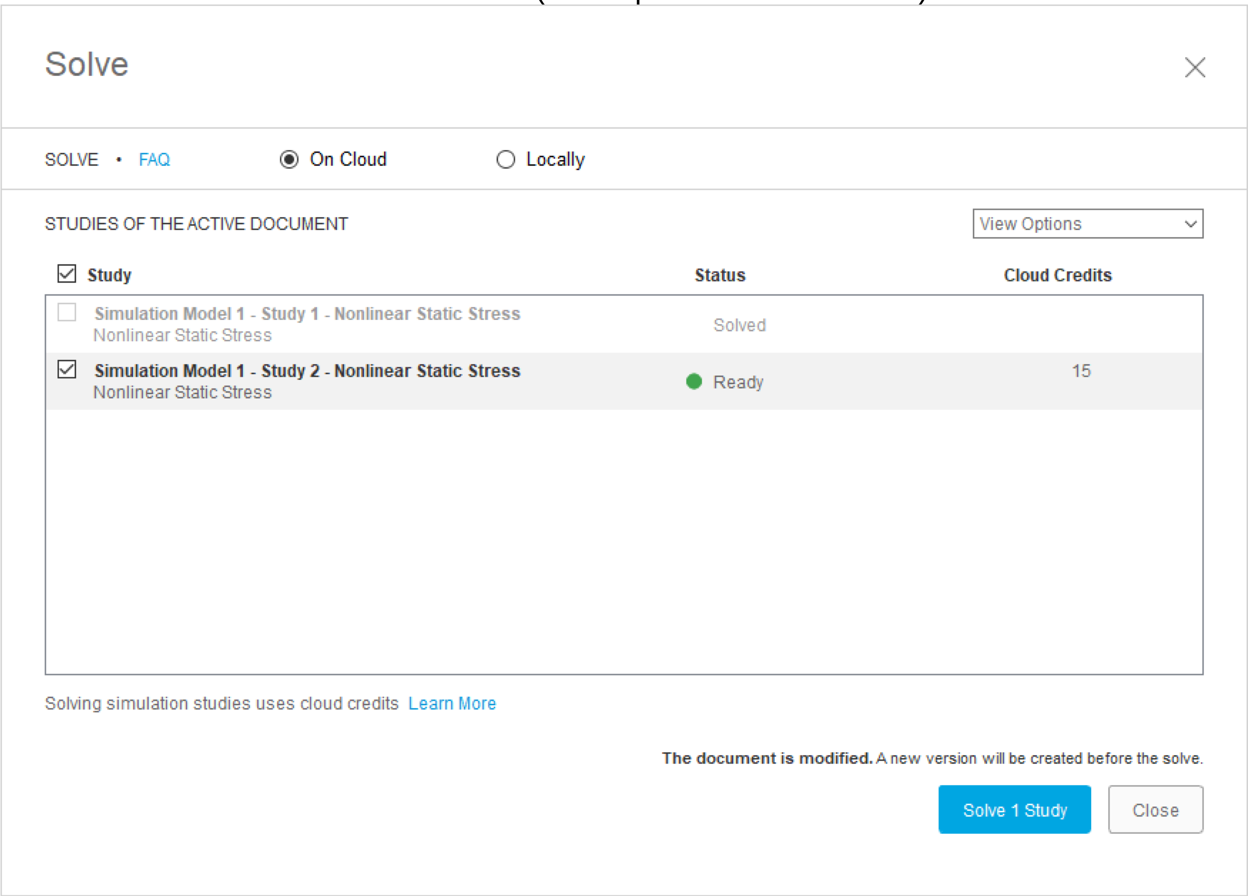
10. Change the Mesh Settings in Settings > Mesh to 3%



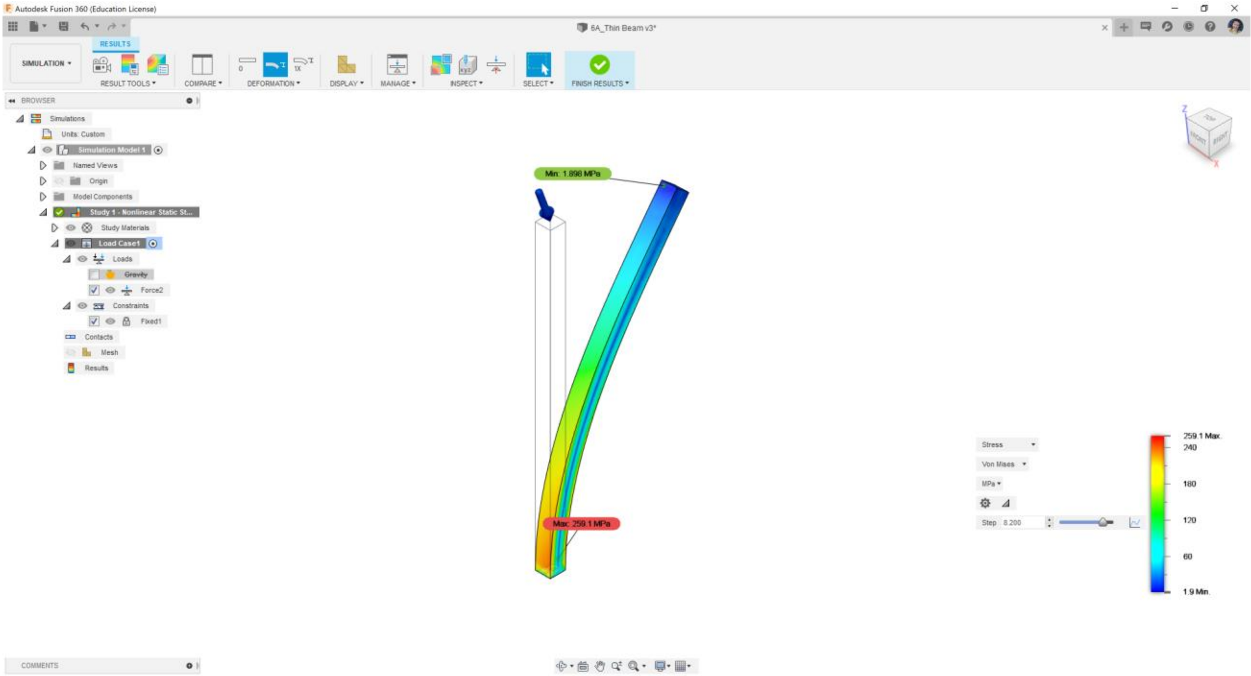
11. Finally, we set up the number of steps in the Simulation. This is how many discrete points in time the simulation will perform over. Set this to 10 Steps in Settings.



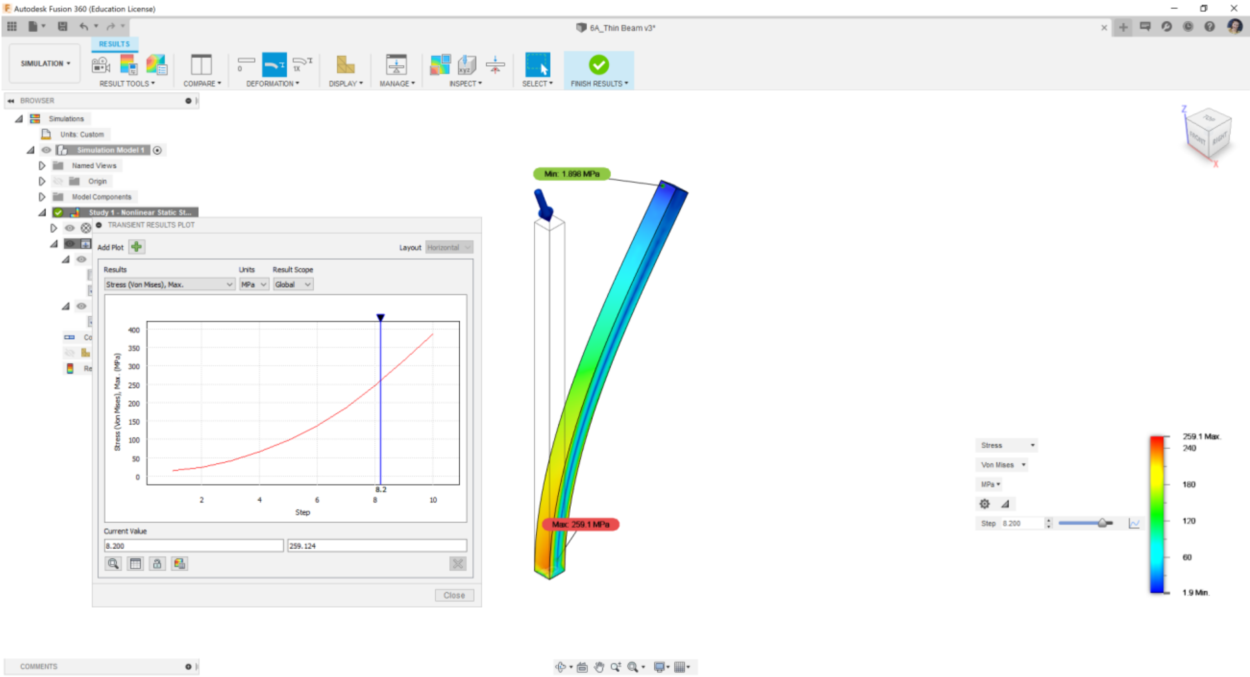
12. Solve the simulation on the cloud (this requires 15 cloud credits).



13. Once the Simulation has solved the results can be reviewed. The results can be reviewd at each of the steps in the simulation.



A 2D chart can also be reviewed over the steps.

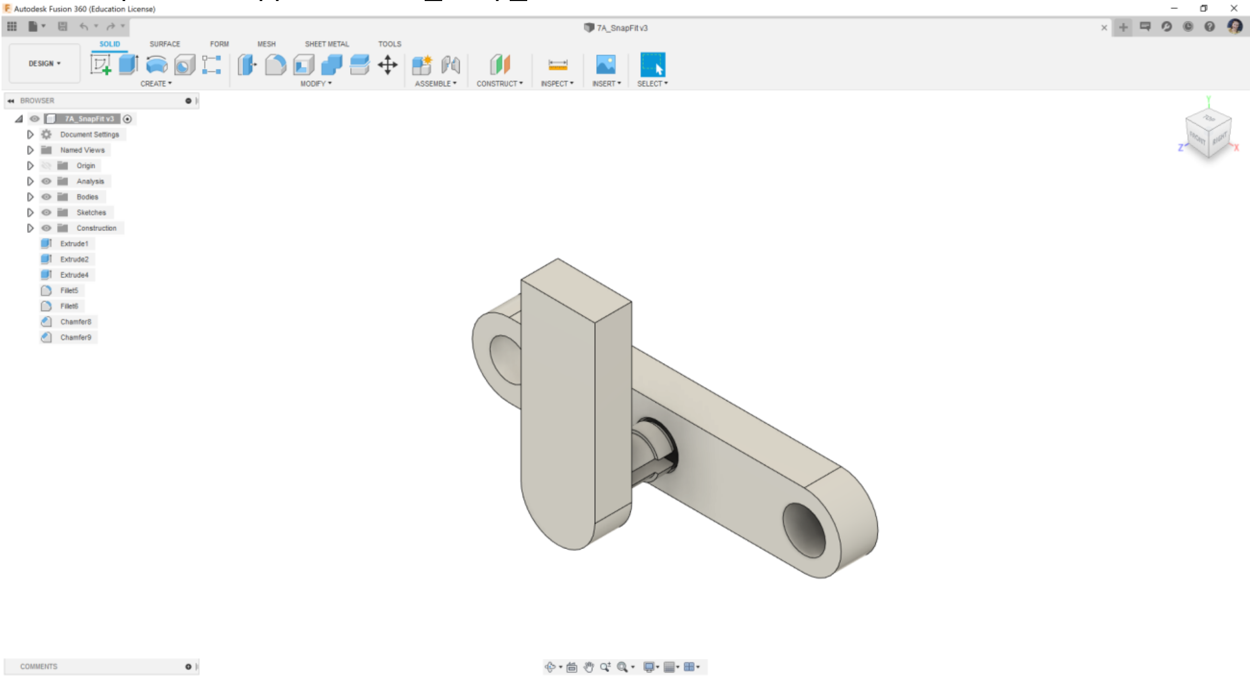


Event Simulation

Event Simulation in Fusion 360 can be used to explore dynamic problems. Event simulation is a fully dynamic tool – it can take into account, the mass; the acceleration; the velocity; the inertia; throughout the simulation. This is really important for simulating impact events, for example a bird strike, or a crash test. We will briefly look at 4 different examples to give an idea of the study type is capable of.

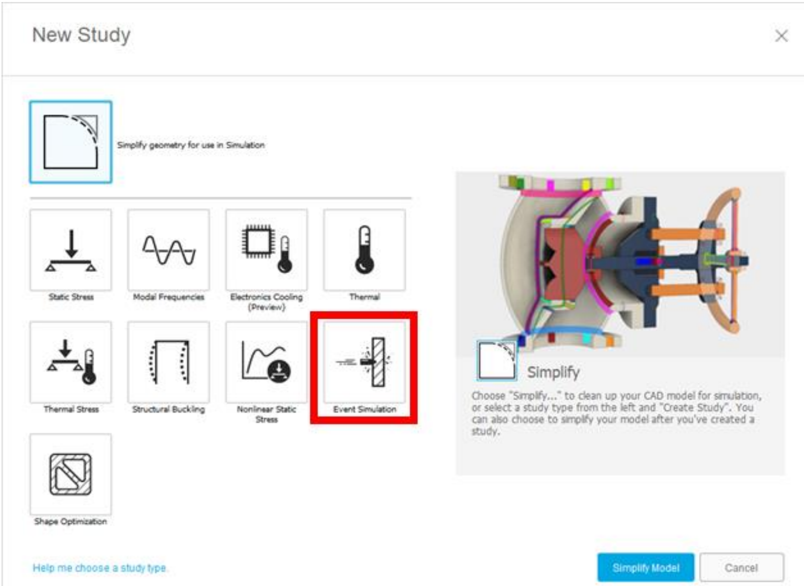
Exercise 7A

1. Open the supplied file 7A_Snap_Fit



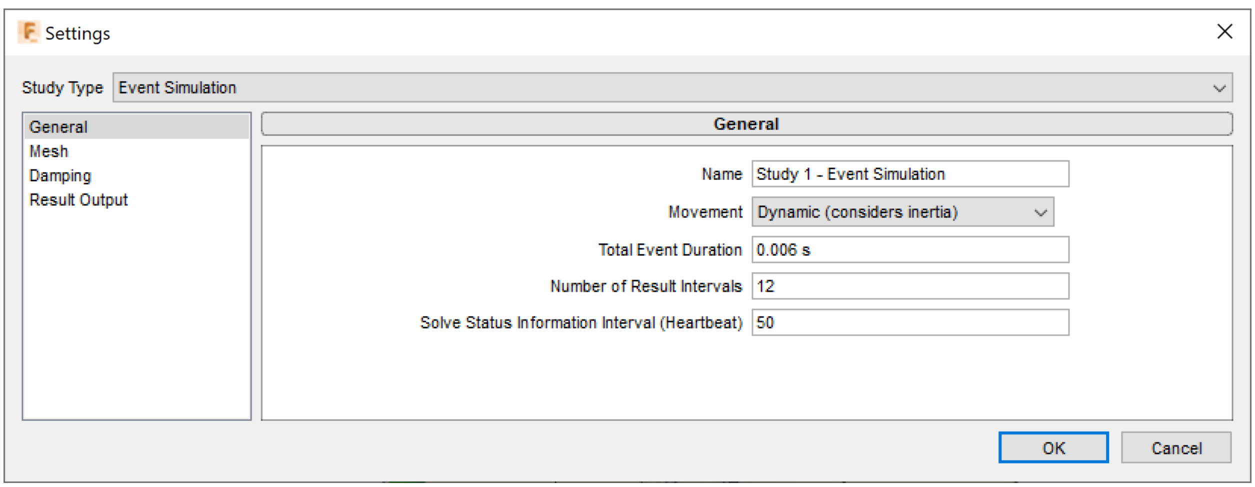
2. Move from the Design Workspace into the Simulation Workspace

3. Choose Event Simulation from the study type menu



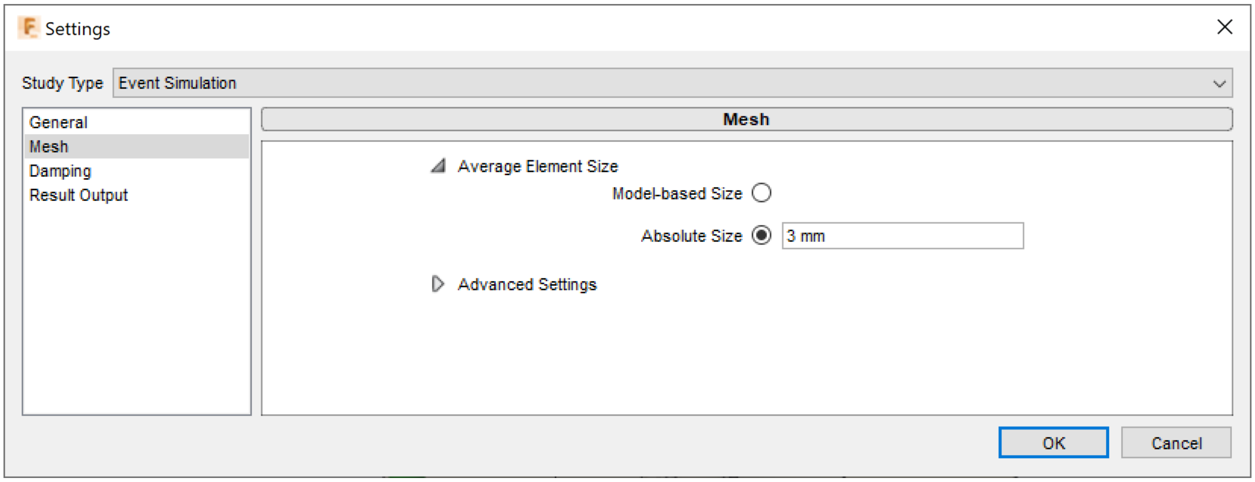
4. Usually we work from left to right along the top bar, but this time we will start by setting up the simulation settings. Click Manage > Settings to open up the settings menu. Chage the following:

a. Movement to Dynamic (considers inertia)
b. Total Event Duration to 0.006 s
c. Number of Result Intervals to 12

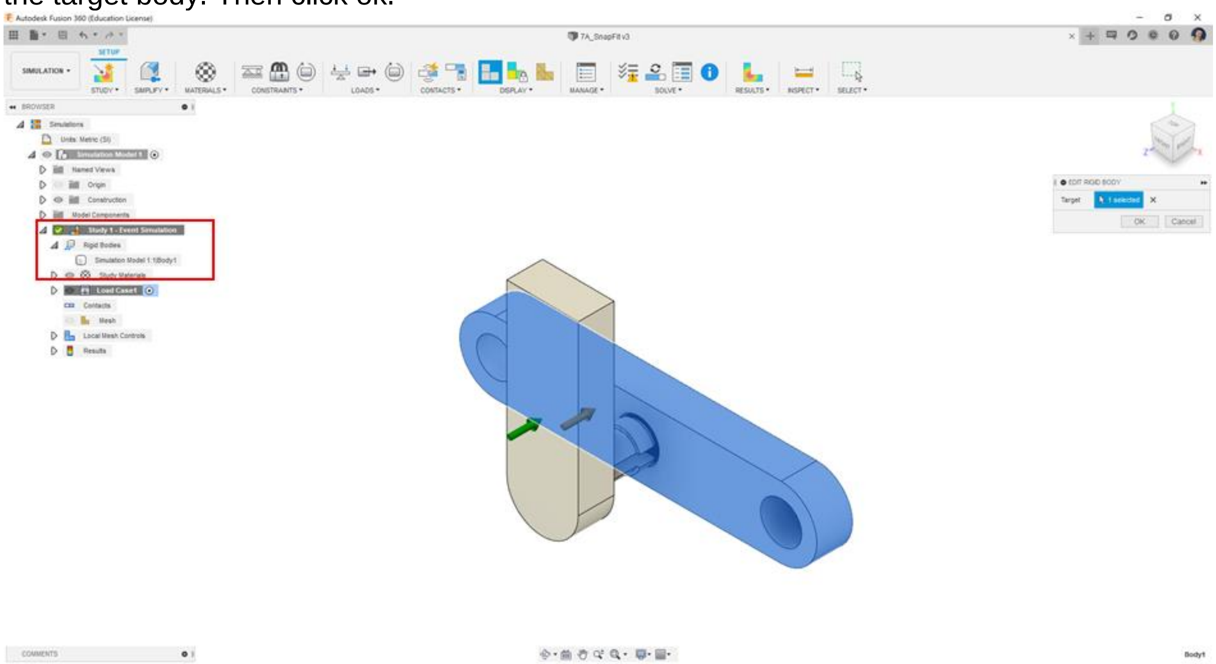


For the Mesh settings:

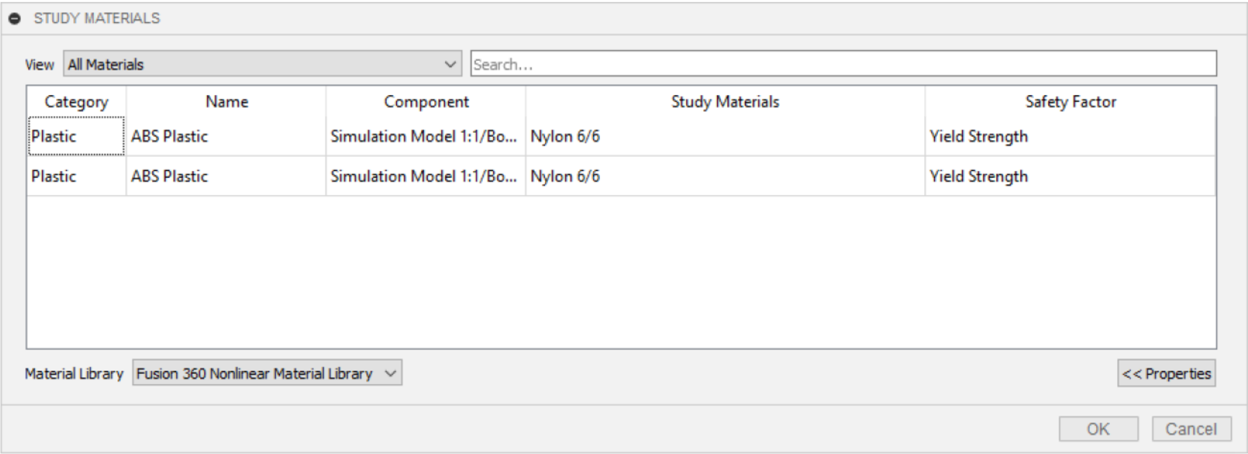
a. Change the mesh to absolute size of 3 mm and then click ok.



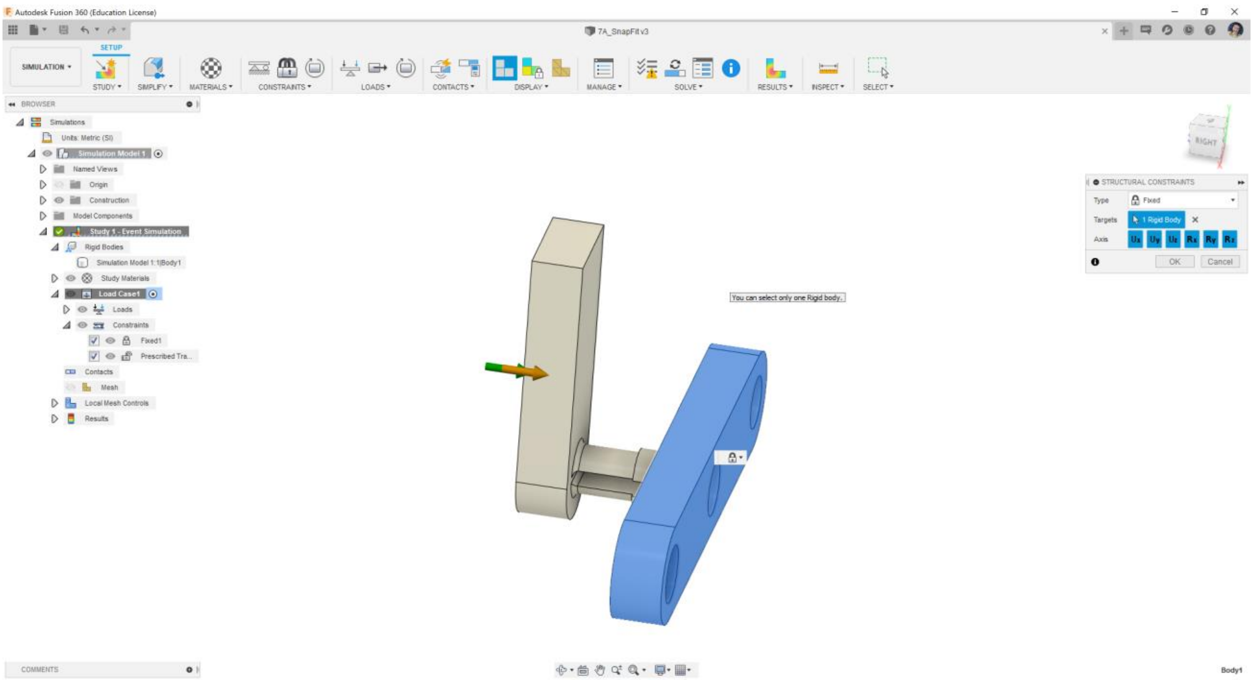
5. Define the stationary and moving bodies. Set Body 1 to be rigid and Body 2 will be moving. We will set Body 1 to be rigid as we are not concerned with its movement or stress in the simulation. In the browser tree select Rigid Bodies, click the pencil to edit, and select Body 1 as the target body. Then click ok.



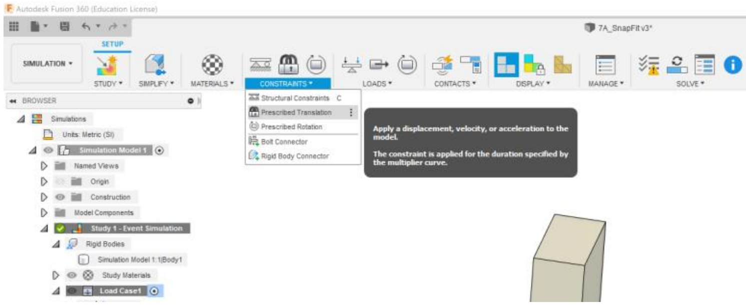
6. Set up the materials for the study. Change the materials for both parts to be Nylon 6/6. Then click ok.



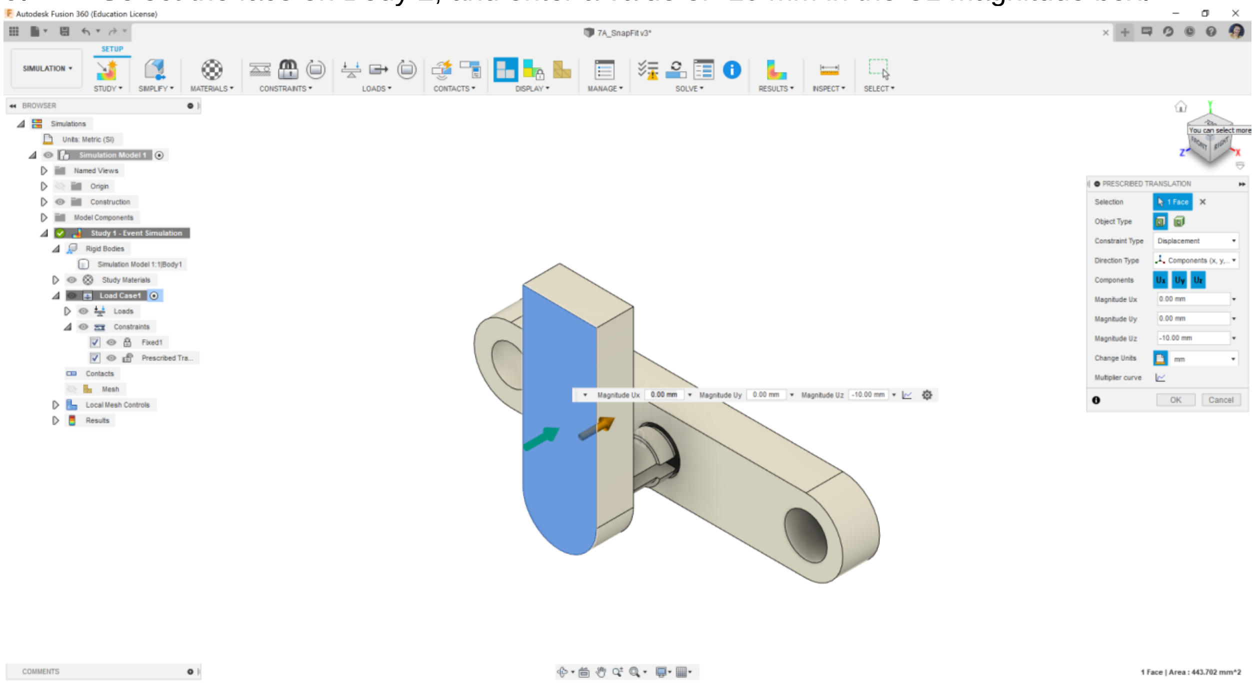
7. Apply the constraints. Start with a constructural constrint. Choose Body 1 and set this to be fixed.



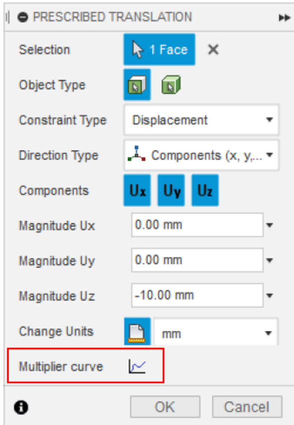
8. Apply a constraint of a prescribed translation. This is to allow the Body 2 to move into Body 1 in a single direction and axis. Choose Constraints > Prescribed Translation

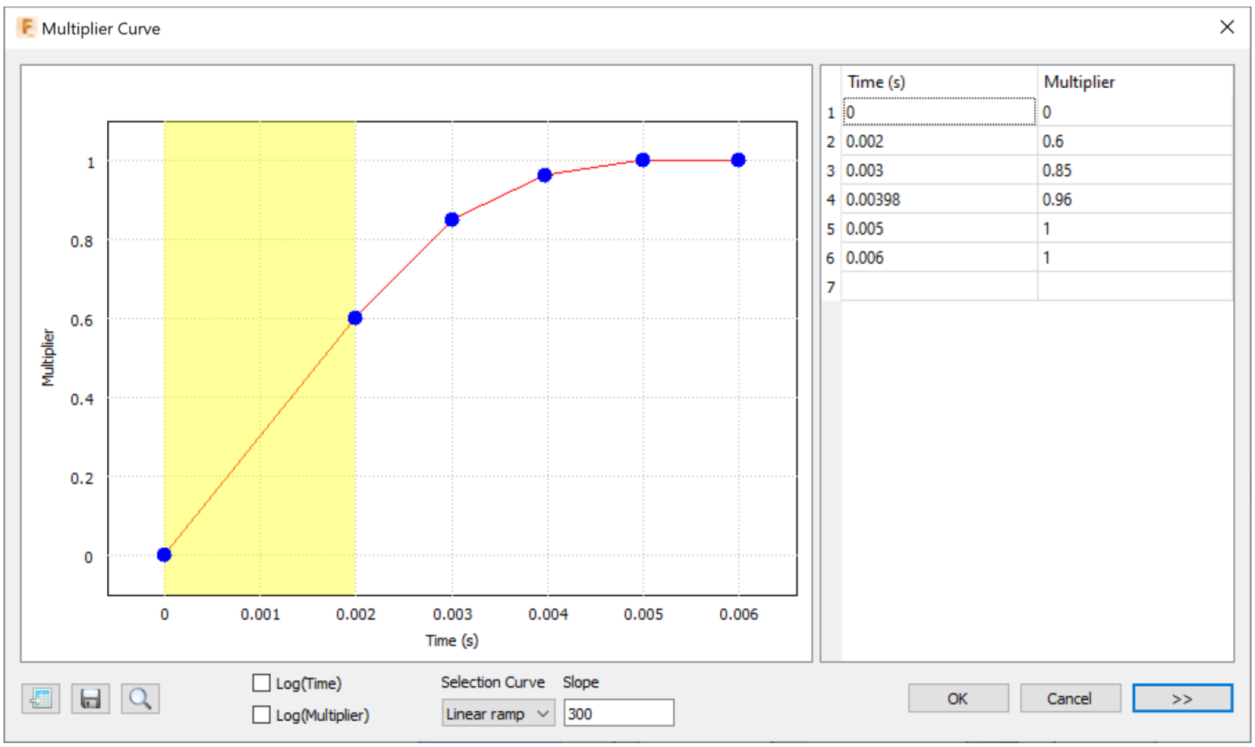


9. Select the face on Body 2, and enter a value of -10 mm in the Uz magnitude box.

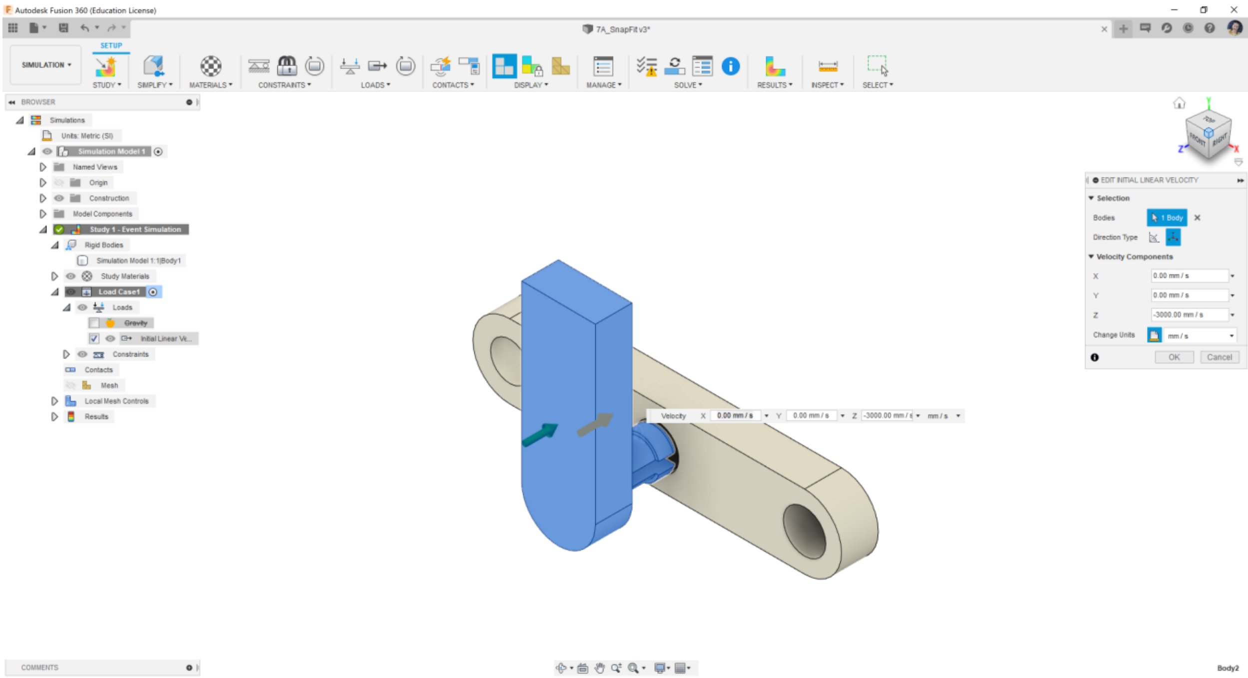


10. Click on the Multiplier Curve and enter the following values. This is where we define how the motion will be over time during the simulation. We will set a movement which will decelerate into the second part over the course of the simulation. Click ok to save the curve, then ok to save the constraint.



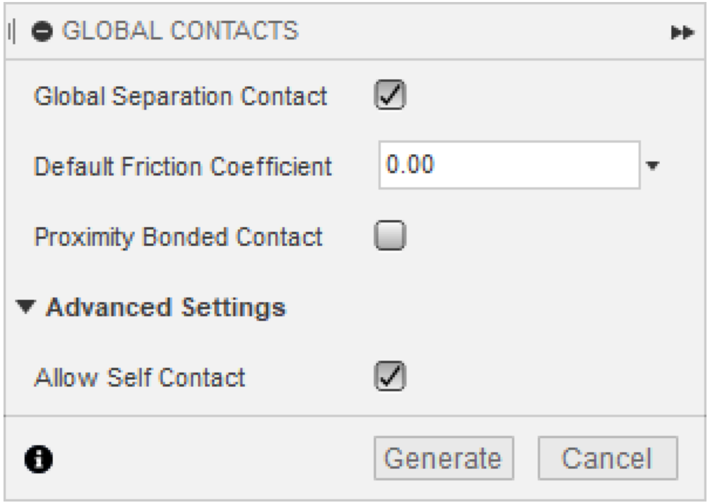


11. Set up the Loads. Start with Load > Initial Linear Velocity. Use Vector for the Direction Type and set a value of -3000 mm/s in the Z component.

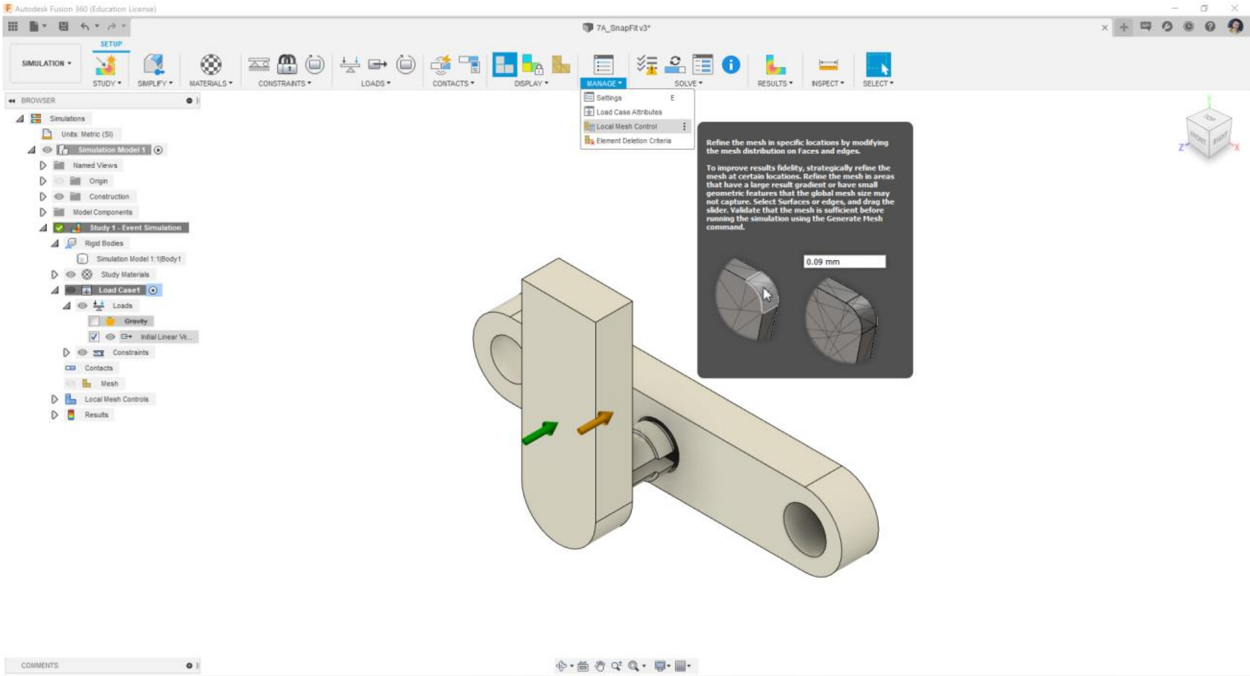


12. Define the contact between the two bodies. If this was a complex problem you may need to enable to check for self contact. In this case we know that the parts will not intersect so we can disable this option which means that the simulation will compute more quickly. Set this in Contacts > Global Contacts.

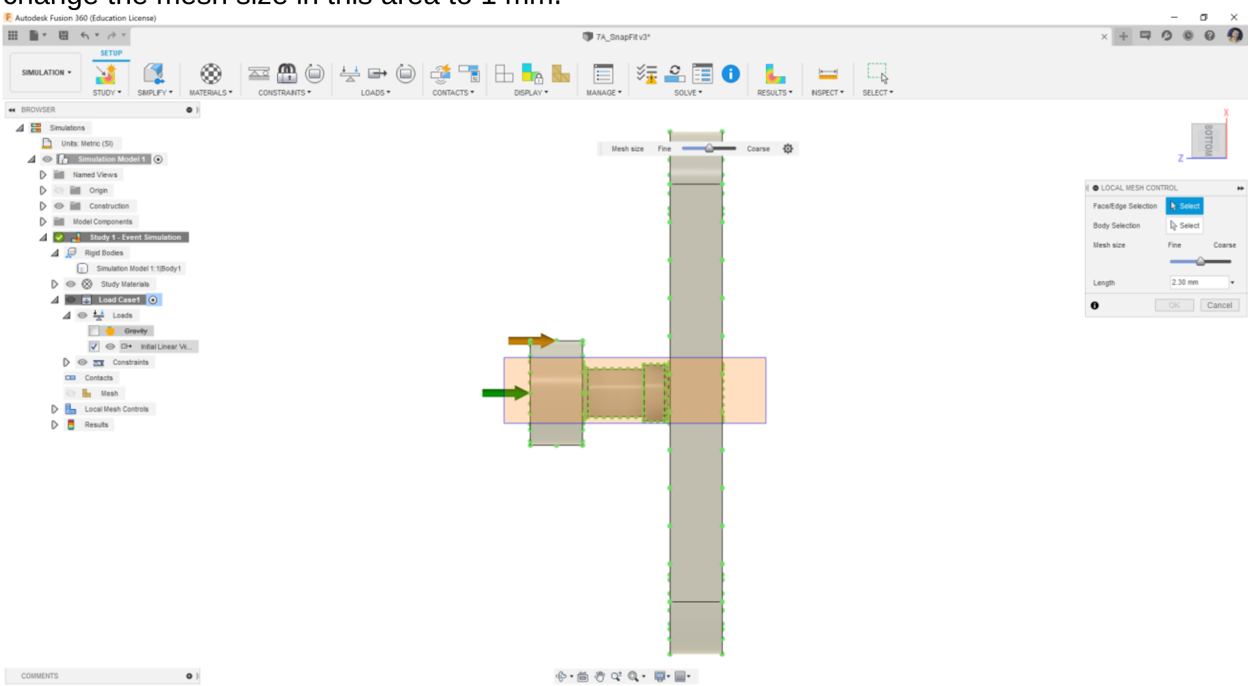
a. Ensure that Global Separation Contact is enabled
b. Proximity Bonded Contact is disabled
c. Under Advanced Settings, Enable Allow Self Contact
d. Click Generate



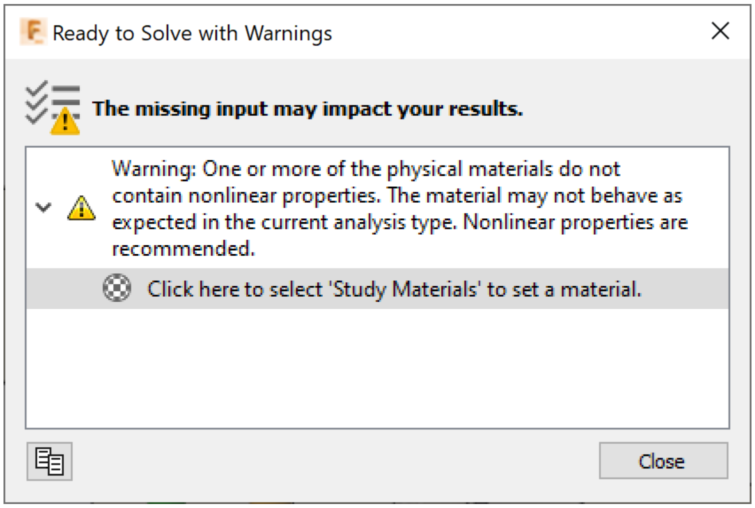
13. Apply Mesh refinement. This allows us to put a finer mesh in an area of interest, without wasting computing power on areas that are not of interest. In Manage select Local Mesh Control.



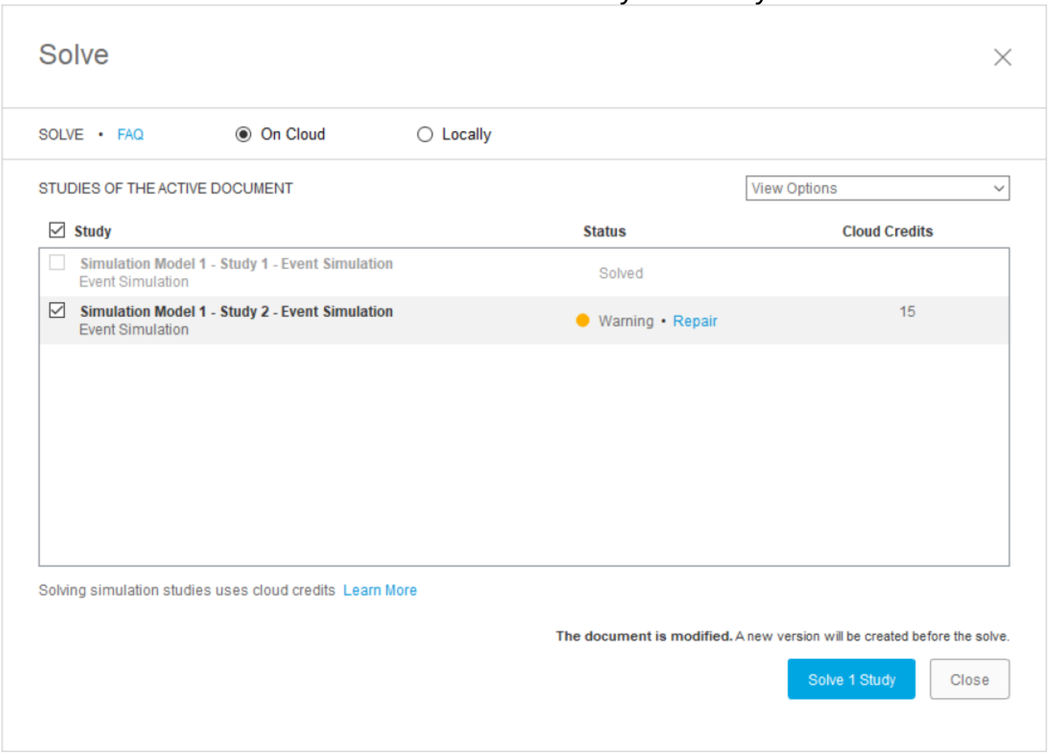
Select the central area (using a left to right selection) which will detect 18 faces/edges. Then change the mesh size in this area to 1 mm.



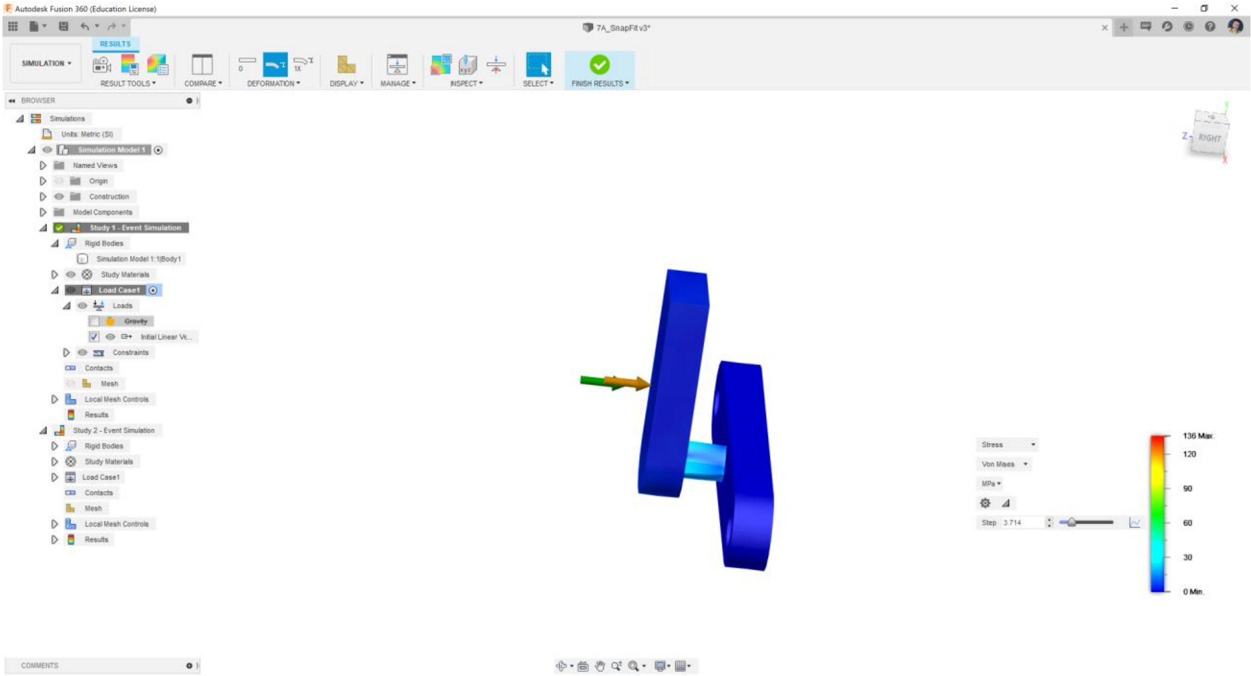
14. Check the pre-check. There may be a warning that a nonlinear material has been selected. In this case this is fine so click ok.



15. Solve the simulation in the Cloud (requires 15 cloud credits). Again, it shows the warning about the nonlinear material. Click Solve 1 Study. This may take some time.



16. Review the results.

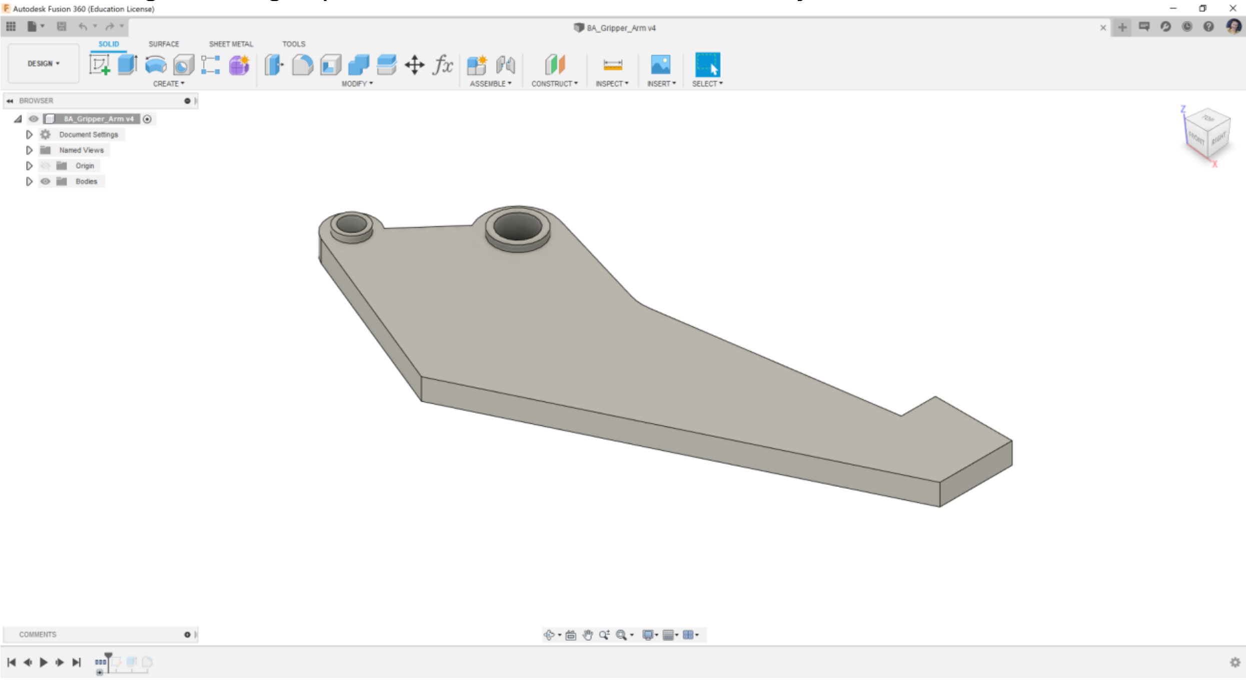

Autodesk Fusion 360 (Education License)
7A_SnapFit v3*
SIMULATION
RESULTS
RESULT TOOLS
COMPARE
DEFORMATION
DISPLAY
MANAGE
INSPECT
SELECT
FINISH RESULTS
BROWSER
Simulations
Units: Metric (SI)
Simulation Model 1
Named Views
Origin
Construction
Model Components
Study 1 - Event Simulation
Rigid Bodies
Simulation Model 1:1|Body1
Study Materials
Load Case1
Loads
Gravity
Initial Linear Ve...
Constraints
Contacts
Mesh
Local Mesh Controls
Results
Study 2 - Event Simulation
Rigid Bodies
Study Materials
Load Case1
Contacts
Mesh
Local Mesh Controls
Results
COMMENTS
Stress
Von Mises
MPa
Step: 3.714
136 Max.
120
90
60
30
0 Min.
RIGHT

Shape Optimization

Shape Optimization is a simulation tool which can be used to remove mass in areas not needed based on a given load. This is for a single material and does not take manufacturing methods into account. We will consider a robot gripper arm and use a mass target for the simulation target to remove material along non-critical load paths. We will then look at using the output to influence future designs.

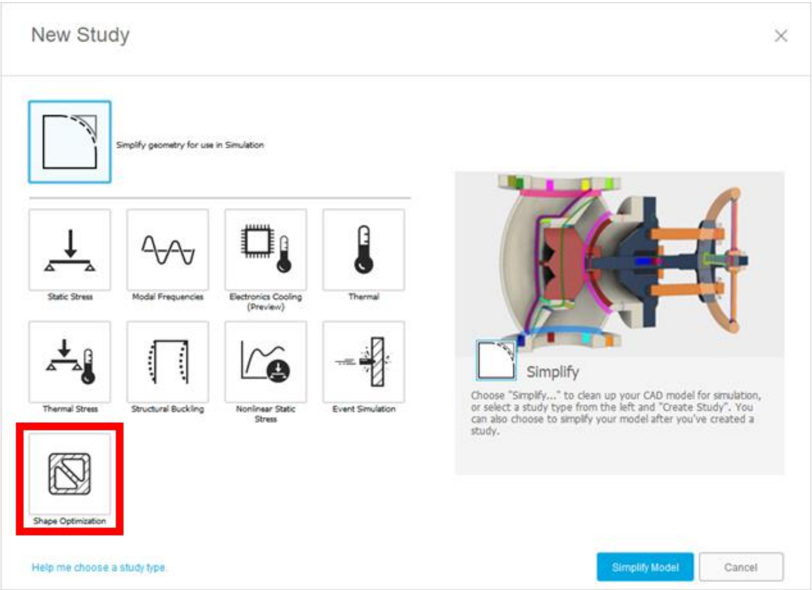
Exercise 8A

1. Open the supplied file 8A_Gripper_Arm. It is important that our new design will comply with the original design specification so we will aim for a safety factor of 2.

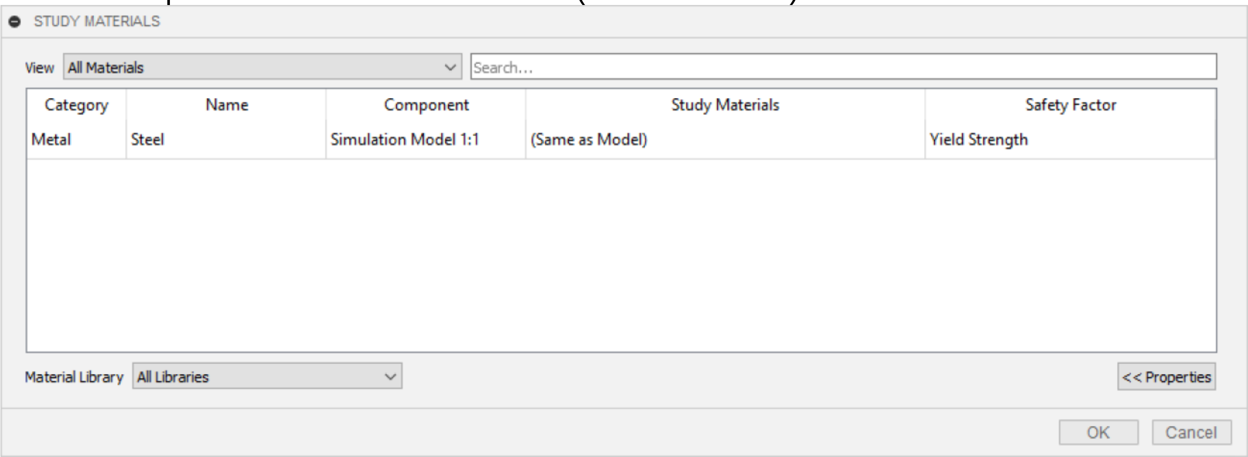


2. Move from the Design Workspace into the Simulation Workspace

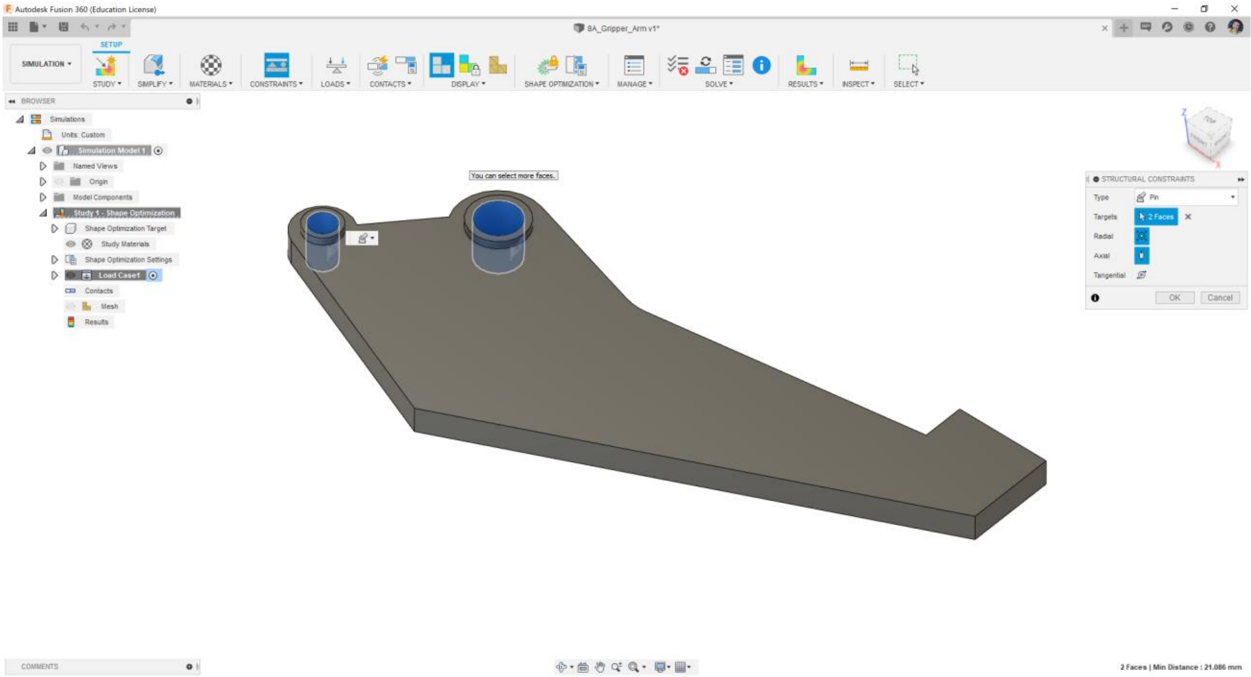
3. Choose a Shape Optimization from the study type menu



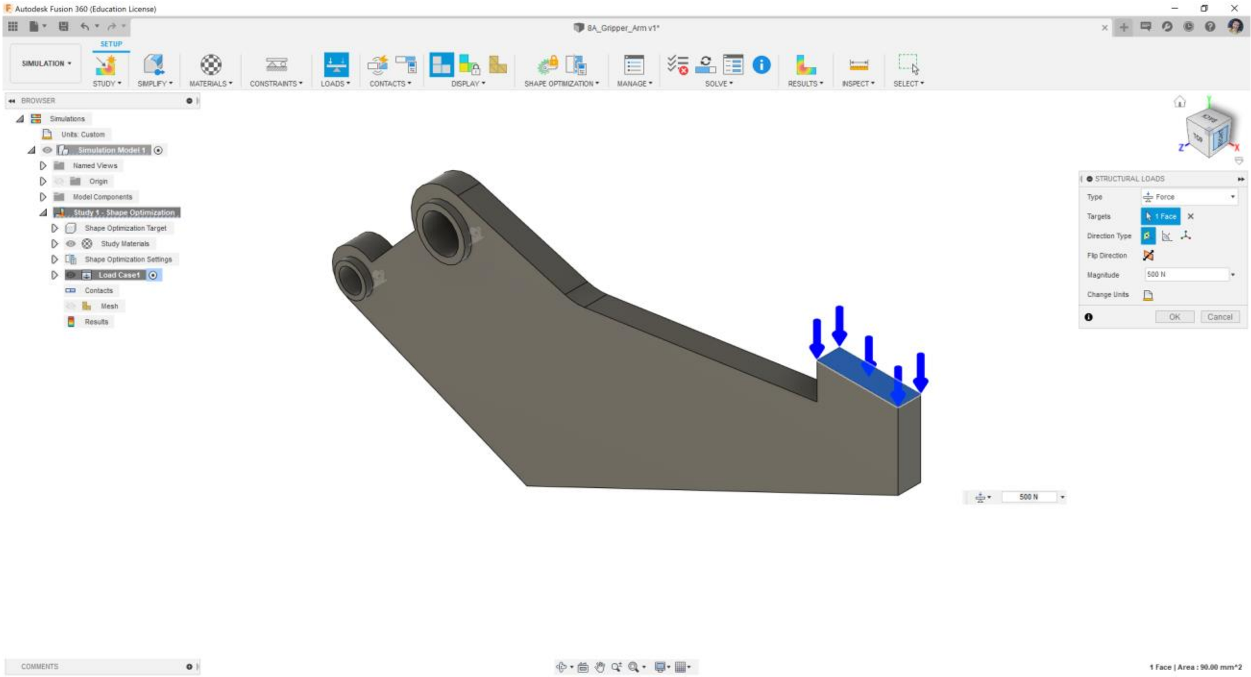
4. Set up the materials. Leave as Steel (Same as Model)



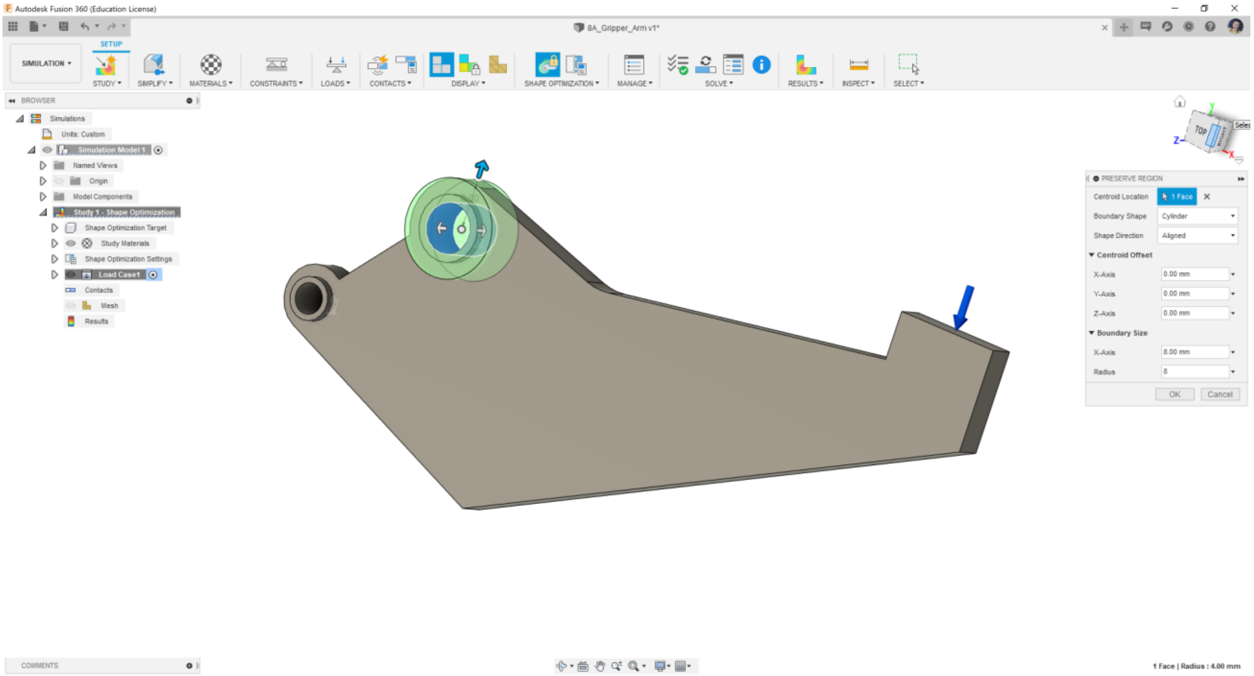
5. Apply the Constraints, change the Type to Pin and select the cylindrical surfaces as below.



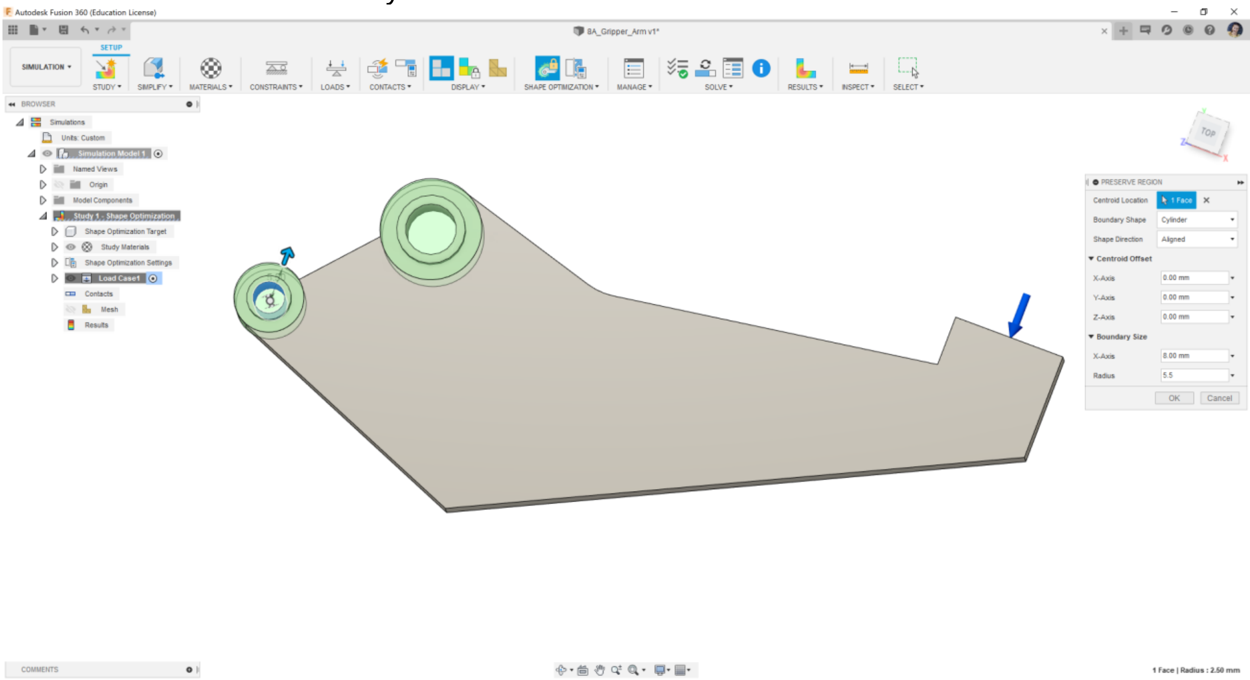
6. Apply a Load to the gripper arm end. Use a Force Load of 500 N to the face as below.



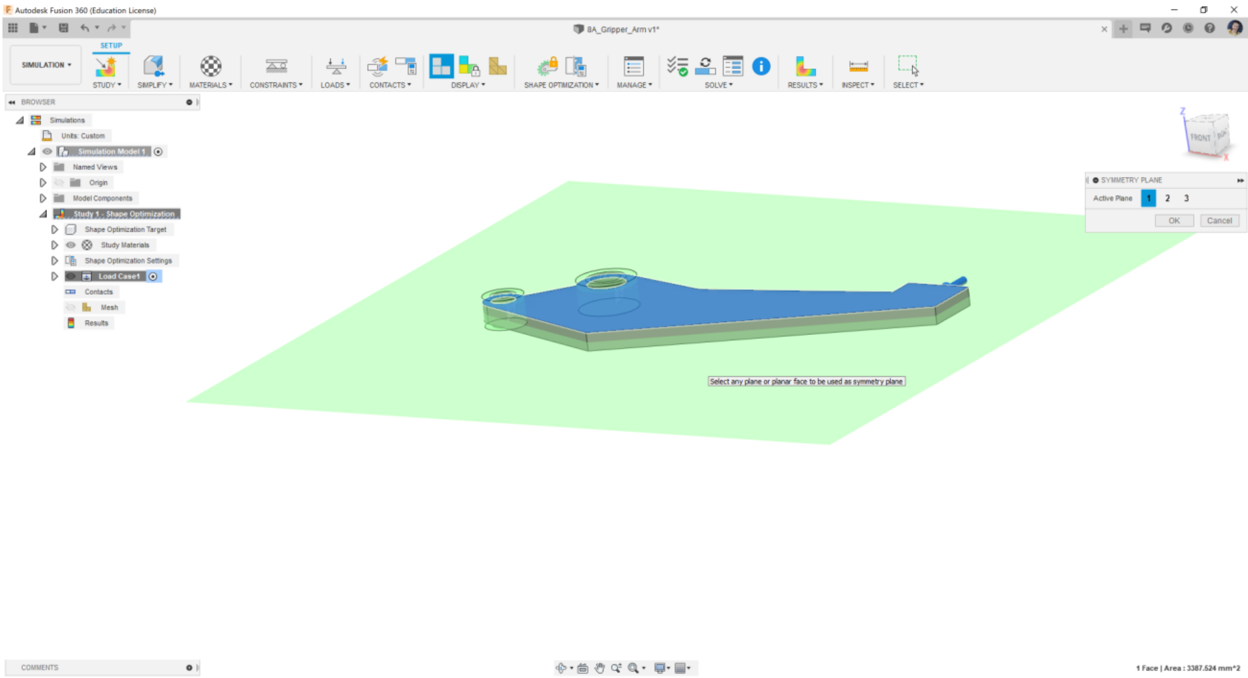
7. Next, we want to set the Preserve Regions – these are the areas which will be kept when the simulation is performed. From the top bar select Shape Optimization > Preserve Region. Select the First Cylindrical surface and set a radius of 8 mm.



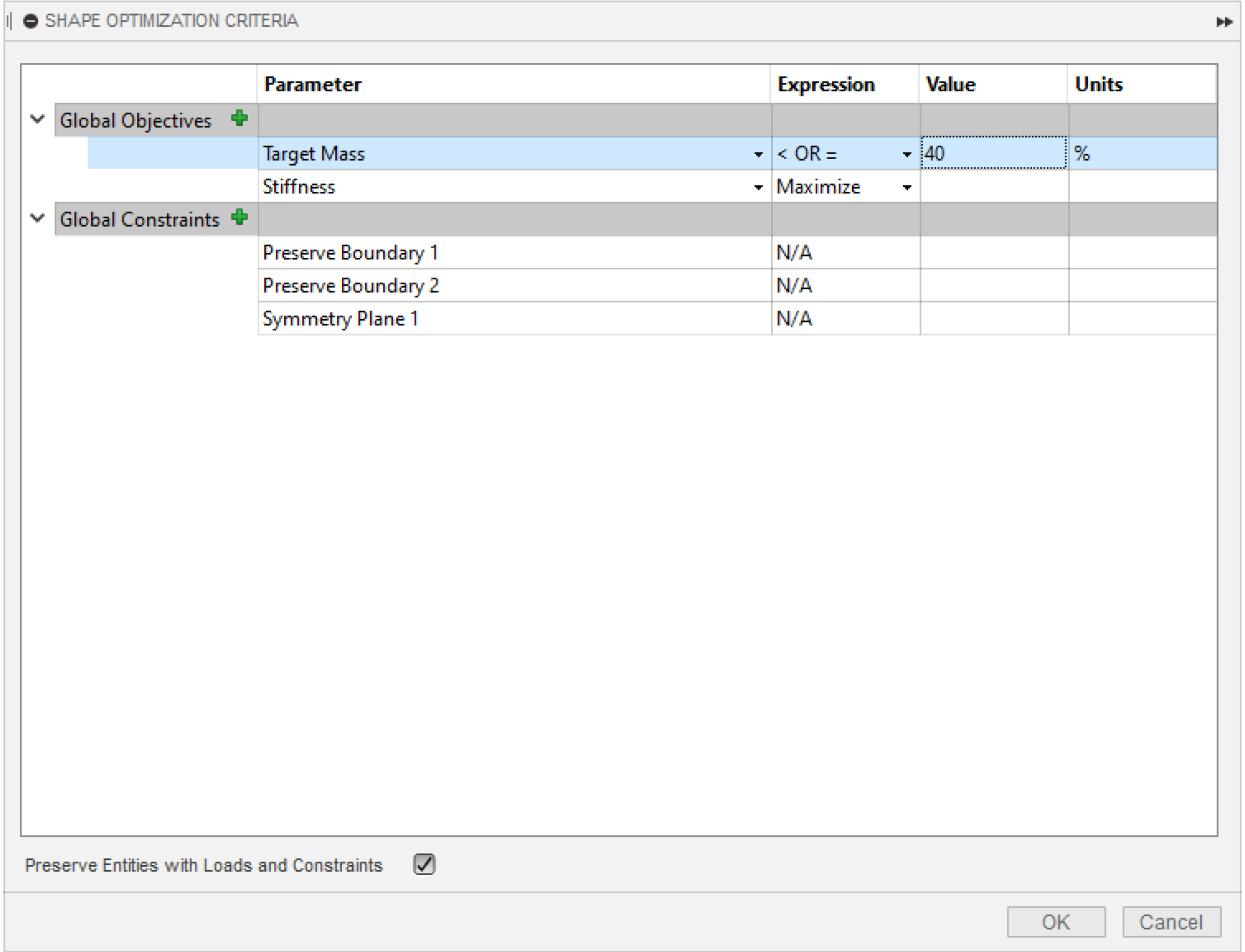
Do the same for the second cylinder surface and set a radius of 5.5 mm.



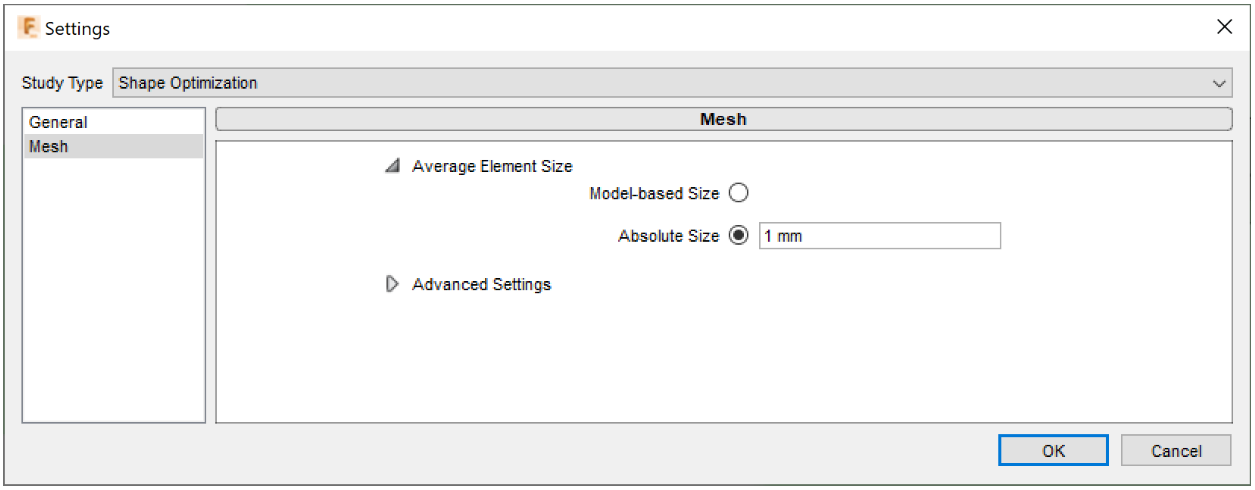
8. Under shape optimisation select a symmetry plane. This is so that when material is removed it is symmetric across this plane.



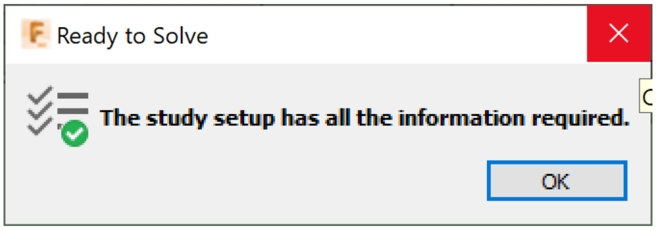
9. Now set the criteria for the study. Select Shape Optimization > Shape Optimization Criteria. Set the Target Mass to be 40%. Click OK.



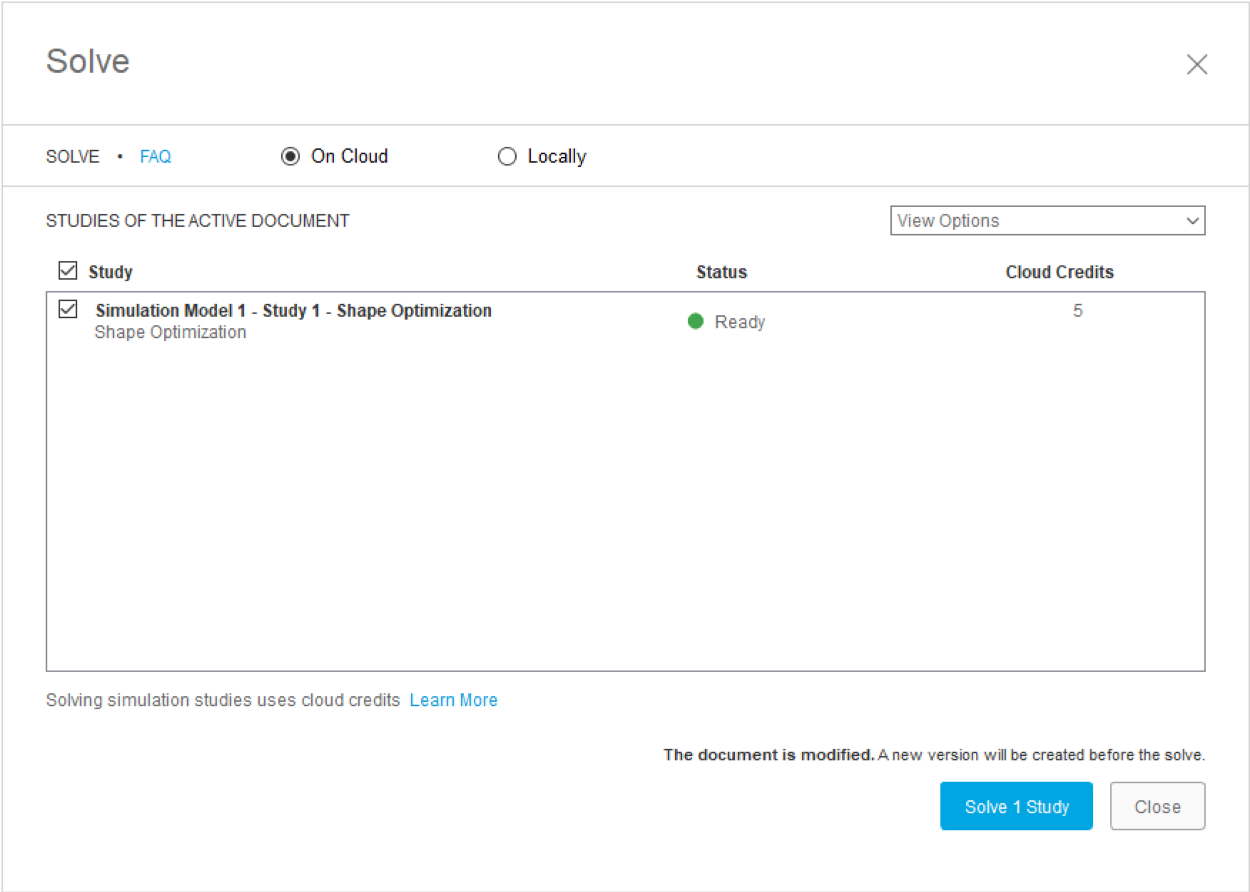
10. Set the Mesh Settings under Manage > Settings. Choose a absolute mesh size of 1 mm then click ok.



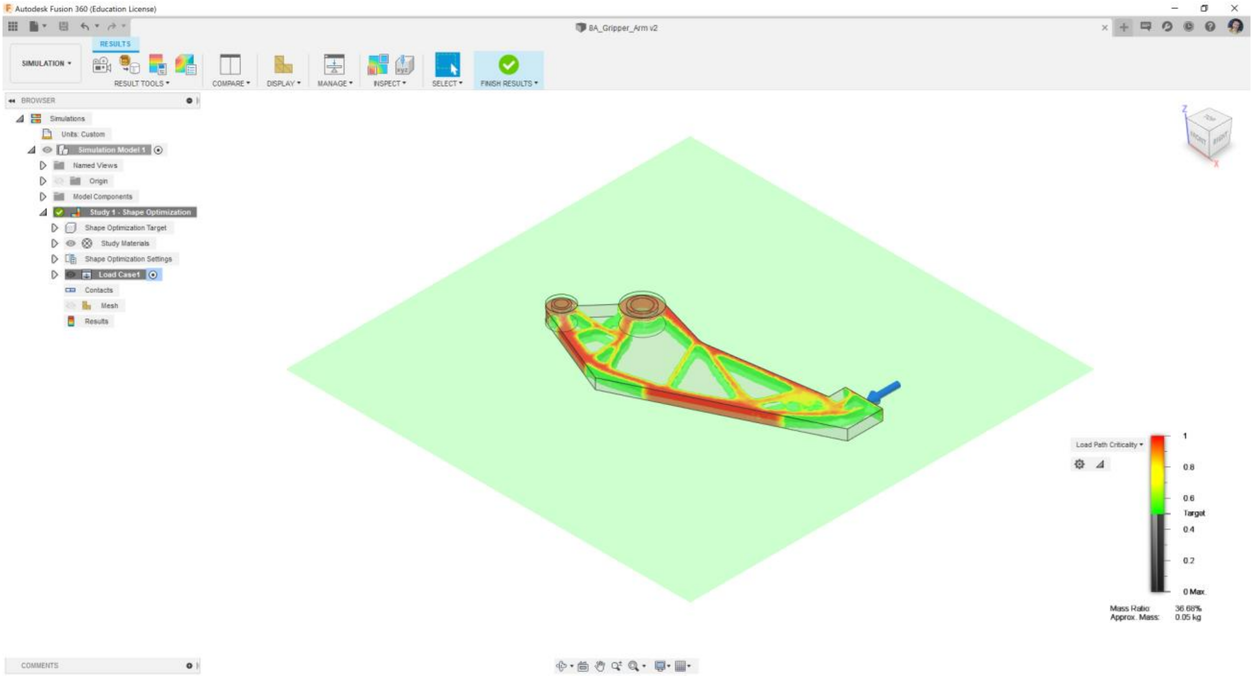
11. Check the Pre-Check.



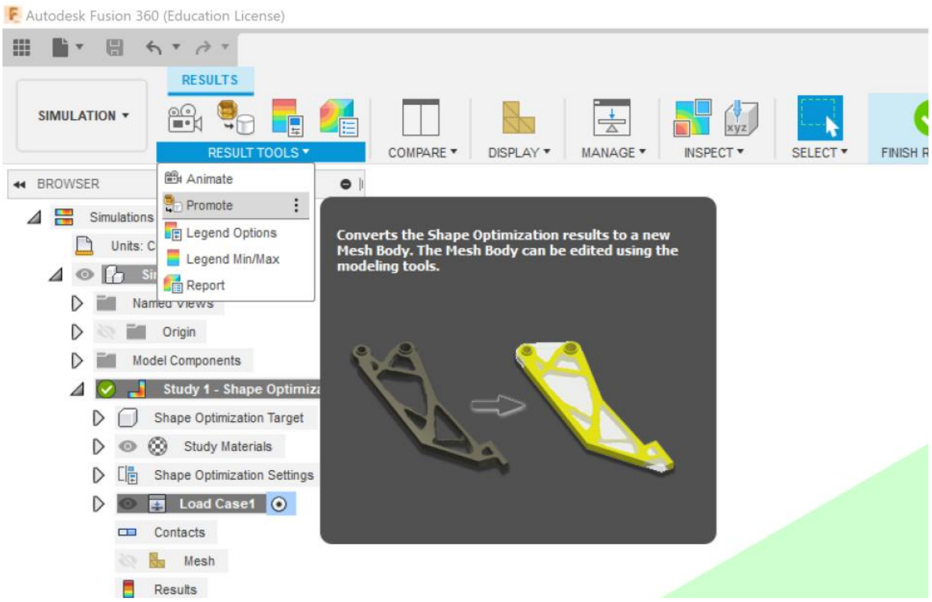
12. Solve the study on the cloud (requires 5 cloud credits) this type of study cannot be solved locally. Click Solve 1 Study.



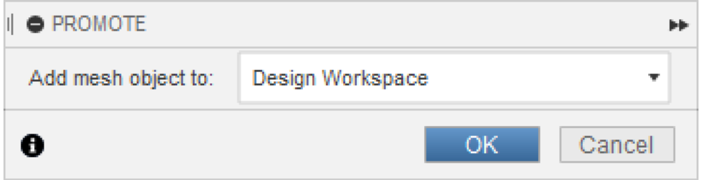
13. Once the simulation is complete, we can review the results. The results are shown as Load Path Criticality. You can drag the arrow to view how the material was removed over the study, you can also go beyond what the mass request was in the criteria.



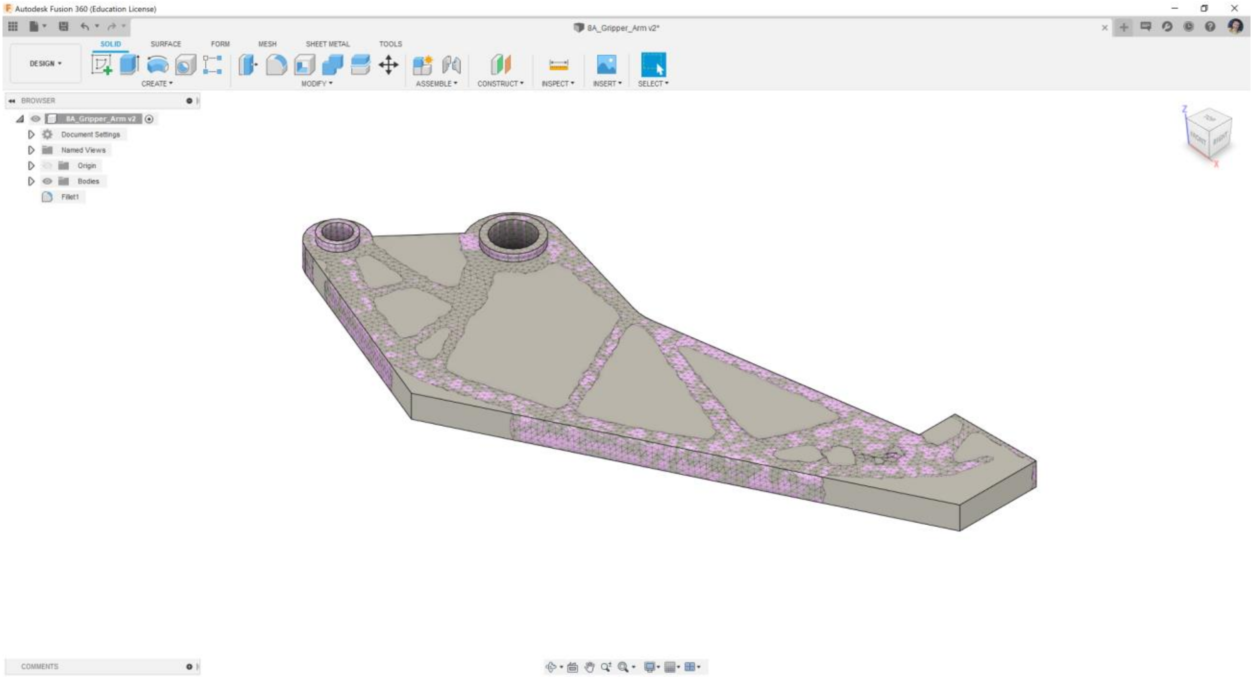
14. We can use this output in our design to influence new designs. In the toolbar under Results Tools select Premote.



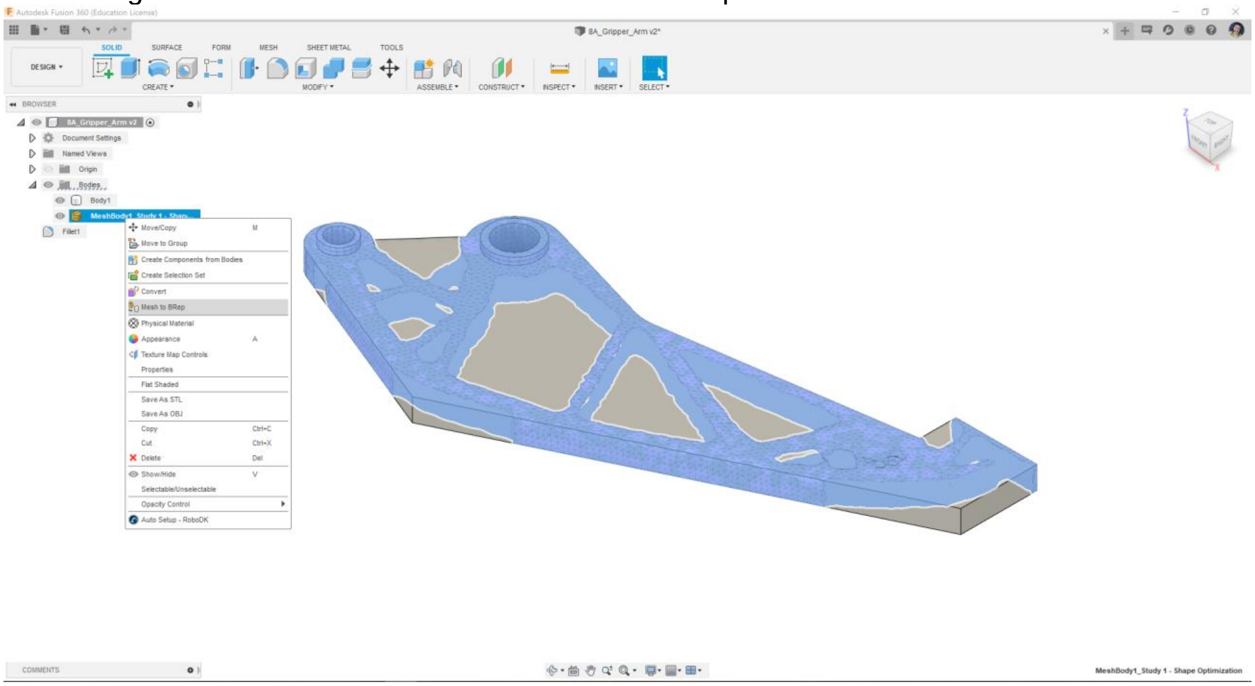
We are going to add this mesh to our design workspace



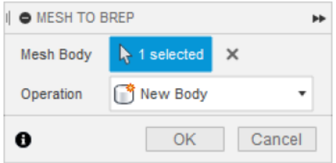
15. Click Finish Results to move back into the Design Workspace. You can see we now have the original body and the new mesh body.



16. Option 1 – Convert Mesh to BRep
Right click on the Mesh and select Mesh to BRep



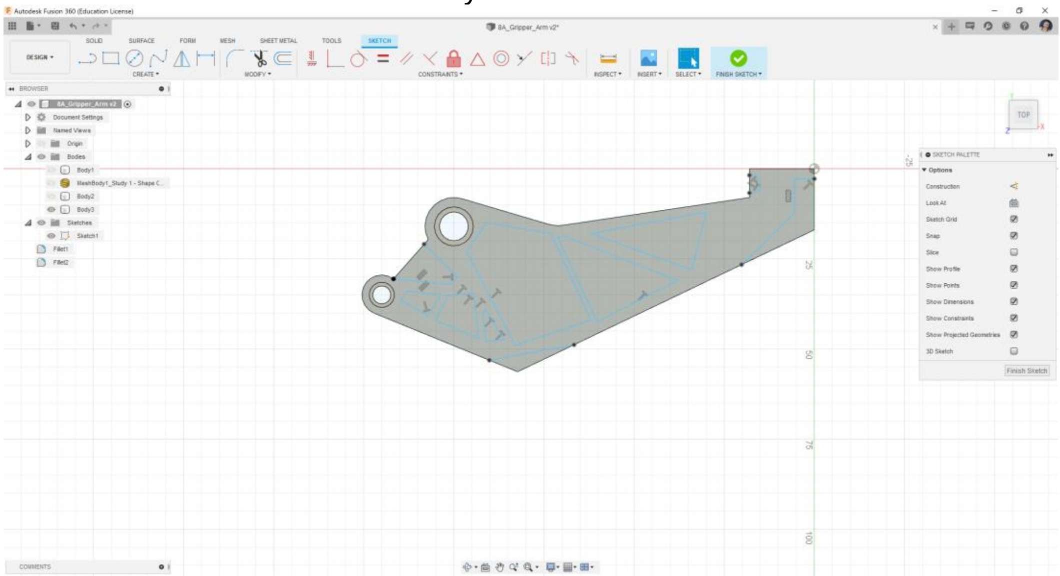
Choose the operation type as New Body and select OK.



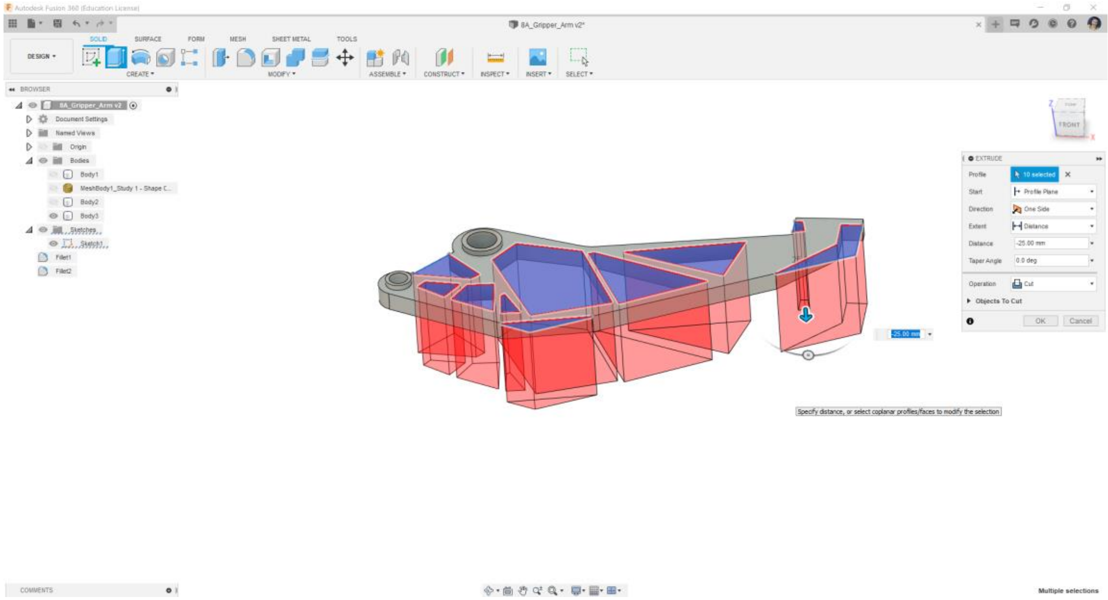
This gives you a new TSpline body that can be manipulated as you would a design in the Design Workspace. This can be re-simulated using a Satic Stress Simulation (see Exercise 1 for how to do this). Check that no areas exceed the safety factor of 2.

17. Option 2 – Use the Mesh as a guide to edit the original Body and simulate using a Static Stress Simulation to check the safety factor does not fall below 2.

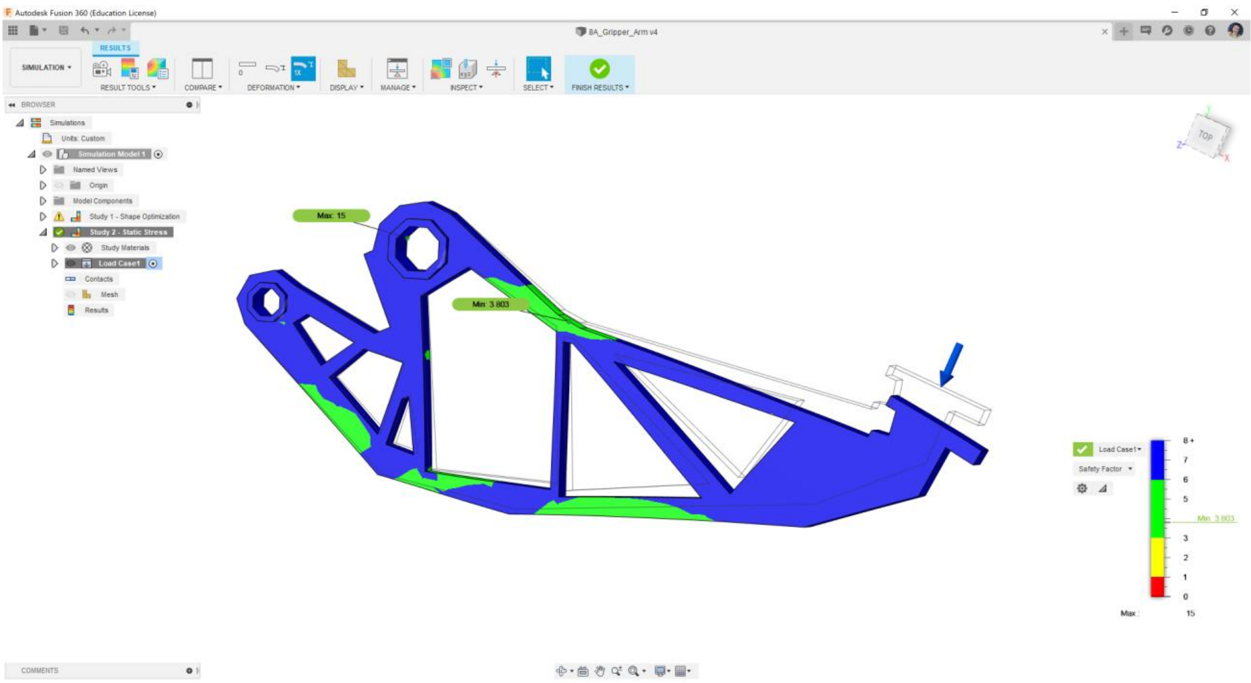
   a. Copy Body 1 and paste so we have a new body to work with.
   b. Create a sketch on the top surface and using the mesh as a guide create areas which will be removed from the body.



   c. Extrude cut the sketch through the part and fillet any sharp corners where needed.



d. Move back into the Simulation workspace and set up a new study as Static Stress type. Perform a static stress simulation to check that the safety factor does not drop below 2. If the results indicate it does, then go back and edit the design and repeat until happy with the design.



Conclusions

In this class we have covered the entire Simulation workspace in Fusion 360. Including the basics of setting up a simulation; how to choose the study type; how to review the results; and how these can be used to influence future design decisions. The following study types in the simulation workspace were covered:

1. Static Stress – A Spanner and Spice Shelf Bracket
2. Modal Frequencies – A Fan Bracket
3. Thermal – A Pipe and its insulation options
4. Thermal Stress – A Pipe hanger and how it reacts to a hot steam pipe
5. Structural Buckling – A bar stool style chair
6. Nonlinear Static Stress – Simple beam example
7. Event Simulation – Dynamic examples
8. Shape Optimization – Removing mass on a robot gripper arm