MFG323727-L

Simulating Sheet Metal Forming in Fusion 360 Event Simulation

Ebot Ndip-Agbor
Autodesk, Inc.

Lee Taylor
Autodesk, Inc.

Learning Objectives

- Setup dynamic simulations for non-linear problems with contact.
- Define boundary conditions and loads for sheet metal forming problems.
- Setup non-linear materials and contacts for forming problems.
- Correctly interpret results (stresses, strains, and reaction forces) from a sheet metal forming simulation.

Description

Sheet metal forming is an integral part of the manufacturing industry, and sheet metal parts can be found everywhere: cars (fenders, hoods, and doors), home appliances (freezers and sinks), and airplane panels. The complexity involved in simulating a deformation-based process like sheet metal forming, which is characterized by large deformation and an abundant amount of contact, has been stream-lined in Fusion 360. This class will explore several methods for designing, setting up, and running sheeting metal forming simulations for processes like stamping and deep drawing in Fusion 360 Event Simulation. Designers will learn how to set up loading conditions, contact conditions, fixed and displacement boundary conditions, and non-linear material properties for both quasi-static and dynamic-type simulations. Finally, the class will also explore how to interpret sheet metal forming simulation results correctly for use in design and manufacturing.

Speaker(s)

Ebot Ndip-Agbor, Autodesk Inc.

Ebot Ndip-Agbor is a Research Engineer at Autodesk and a member of the Fusion 360 Family simulation team. He has a PhD in Manufacturing and Mechanics from Northwestern University. During his doctoral work, Ebot focused on a wide range topics including: finite element simulation and process parameter optimization in sheet metal forming and GPU-based finite element acceleration in additive manufacturing. At Autodesk, Ebot works primary on Generative Design and development of Fusion 360 Event Simulation.

Lee Taylor, Autodesk Inc.

Lee Taylor is a Distinguished Research Engineer at Autodesk and a member of the MCP organization. He has a PhD in Engineering Mechanics from the University of Texas at Austin and 35 years of experience developing high performance finite element applications. Lee was a researcher for ten years at Sandia National Laboratories and left there to work in the commercial software industry where he developed high-performance, parallel explicit dynamics applications that are widely used. Lee is the author of Autodesk's explicit dynamics software that powers Fusion Event Simulation.

Contents

Introduction

Sheet metal forming is an integral part of the manufacturing industry, and sheet metal parts can be found everywhere: cars (fenders, hoods, and doors), home appliances (freezers and sinks), and airplane panels. The complexity involved in simulating a deformation-based process like sheet metal forming, which is characterized by large deformation and an abundant amount of contact can be challenging. Setting up and running such simulations has been stream-lined in Fusion 360.

In this lab we will be setting up and running simulations for one of the most popular sheet metal forming techniques called deep drawing (see Fig. 1). In deep drawing a punch is used to stamp a flat blank (sheet) constrained by a blank holder into a die to produce a predefined shape.

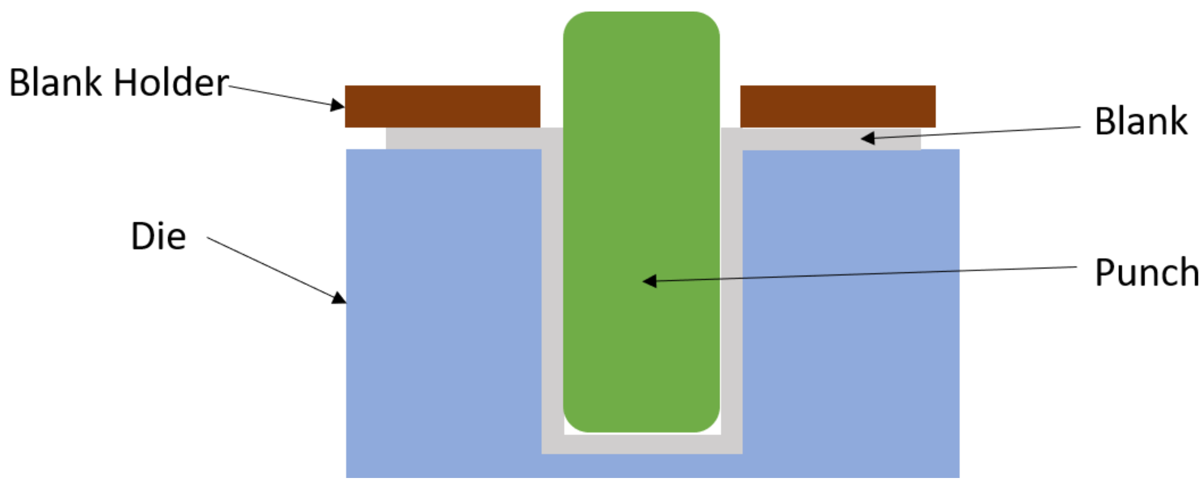


Fig. 1: Deep drawing

For the simulation we will be using the setup specified by Ziaeipoor et al. [1] which is shown in Fig. 2. Simulations will be run for both the full model and the eighth model. The dimensions of the parts are shown in Table 1. Instead of using AL6111T4 that was used in the model by Ziaeipoor et al., we will be using AL5754-O. The true stress – true strain data for AL5754-O obtained from a tensile test and fitted with the Voce Hardening Law is given in Fig. 3.

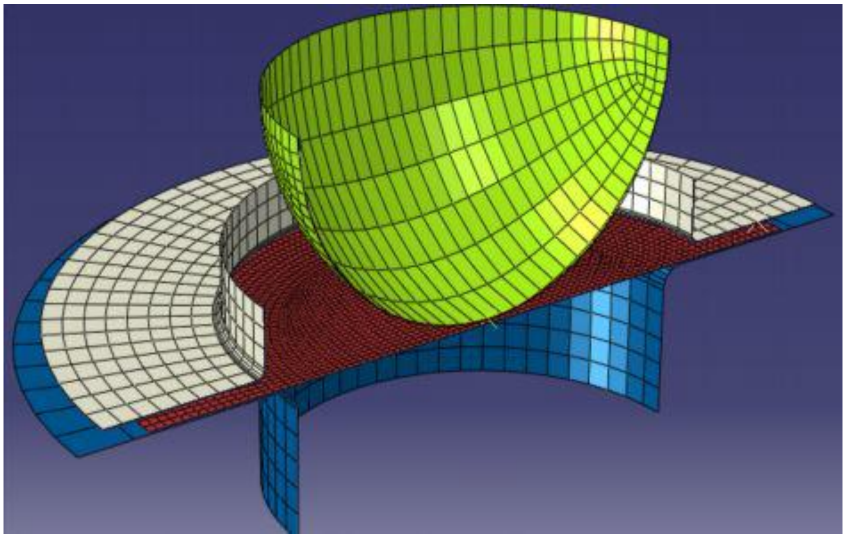


Fig. 2: Deep drawing simulation setup from Ziaeipoor et al. [1]

Table 1. Mechanical Material properties and process parameters for simulation

Material	AL5754-O
Blank Diameter (mm)	177.8
Thickness (mm)	1
Poisson ratio, μ	0.33
Young's modulus, E (GPa)	70
Density (g/cm^3)	2.7
Yield Stress (MPa)	250
Ultimate Tensile Stress, (MPa)	290
Punch travel, (mm)	57
Punch Diameter, (mm)	101.6
Inner Diameter of Blank Holder, (mm)	103.6
Outer Diameter of Blank Holder, (mm)	200
Inner Diameter of Draw binder, (mm)	103.6
Outer Diameter of Die, (mm)	210
Coefficient of Friction	0.05

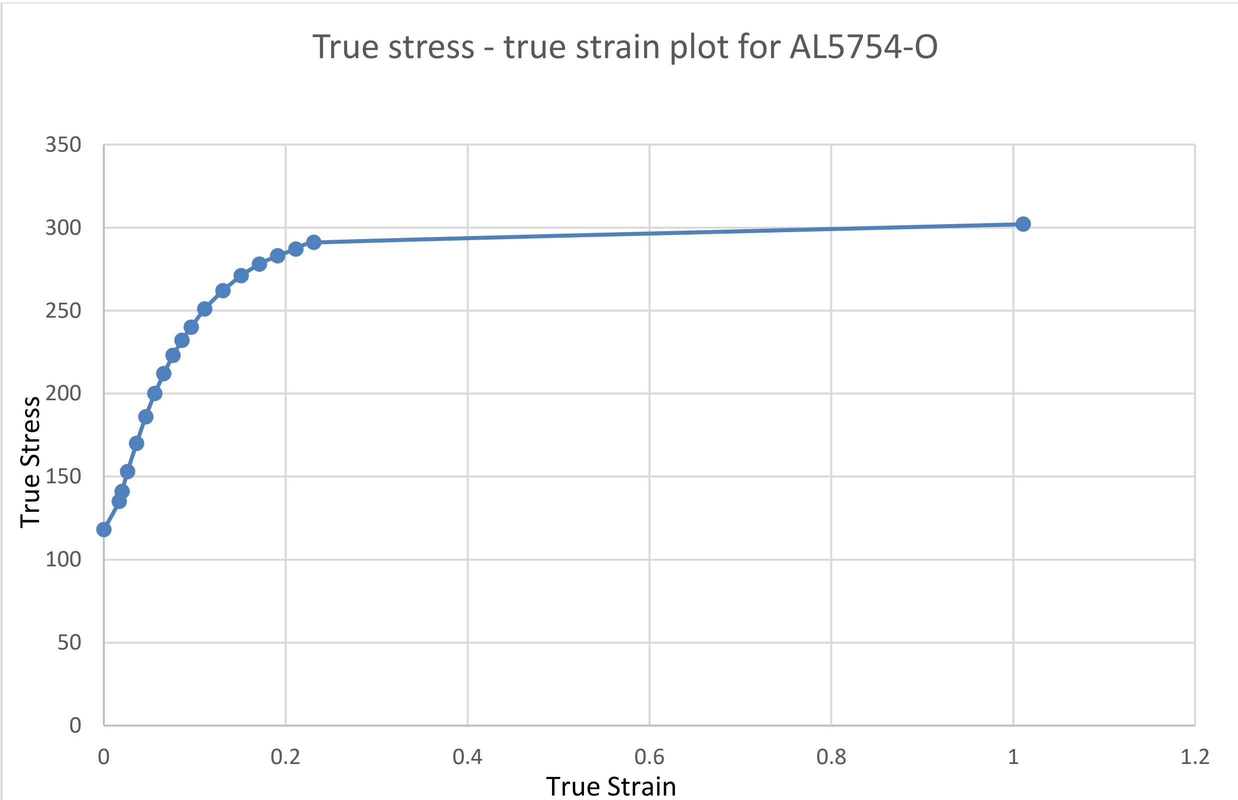
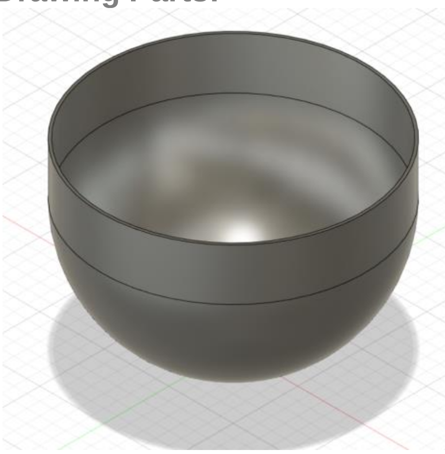


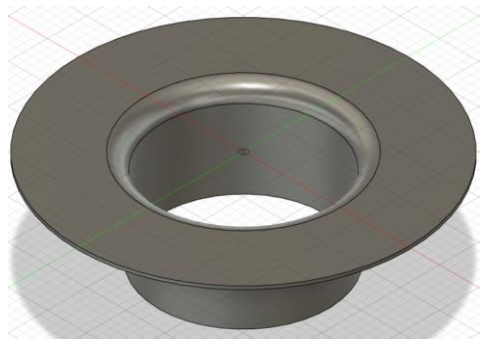
Fig. 3: True stress – True strain plot for AL5754-O

Full Deep Drawing Model in Fusion 360

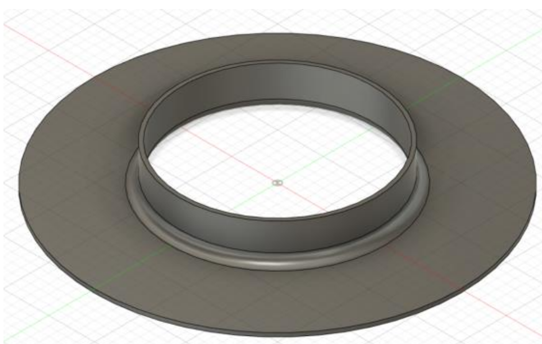
Deep Drawing Parts:



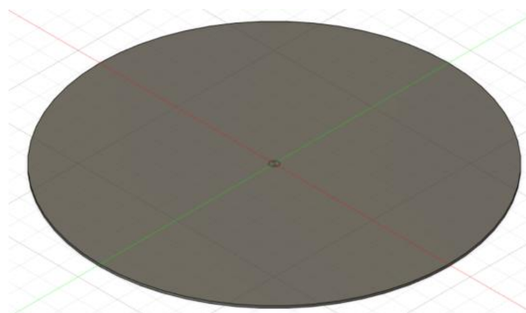
Punch



Die



Blank Holder

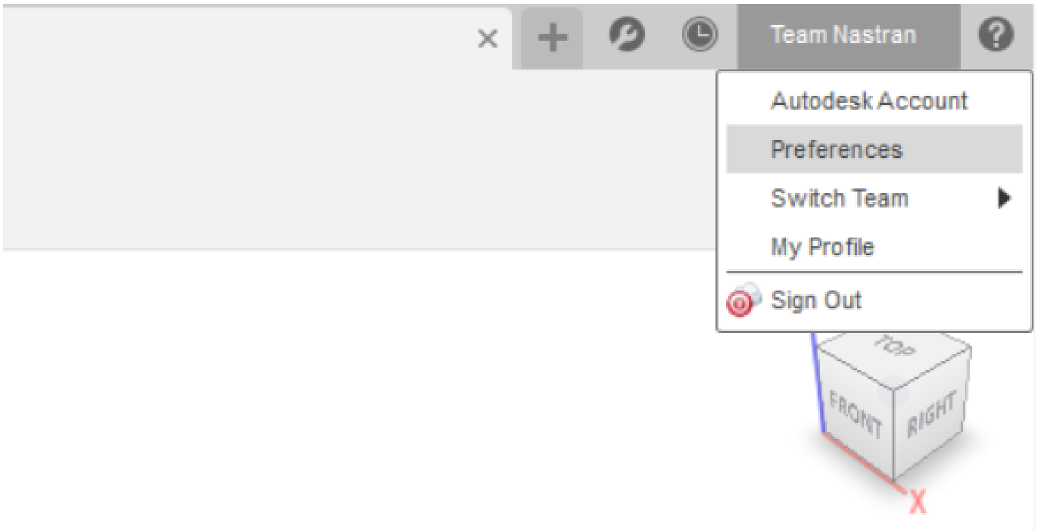


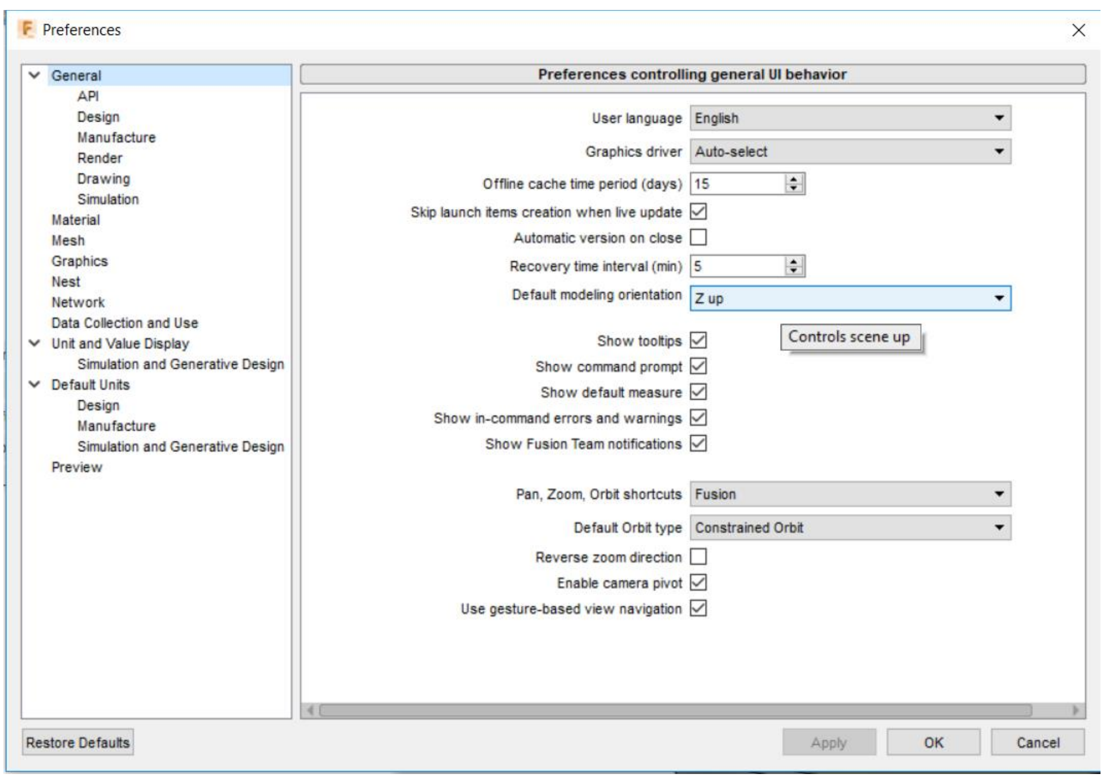
Blank

Step-by-step instructions

Set Orientation

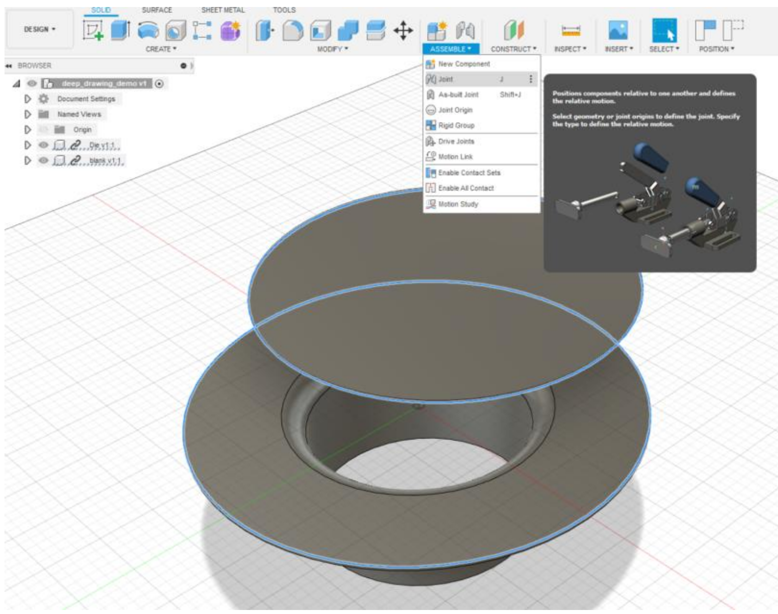
1. Click on **Account** -> **Preferences**. Then click on **General**, under **Default modeling orientation** make sure **Z up** is selected.


Team Nastran
Autodesk Account
Preferences
Switch Team
My Profile
Sign Out
TOP
FRONT
RIGHT
X

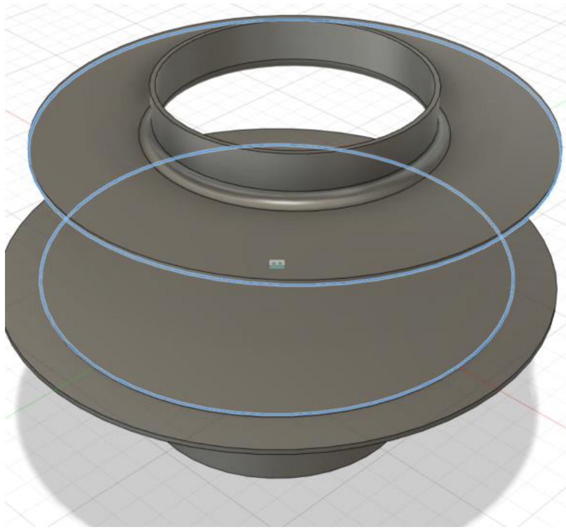

Preferences
General
API
Design
Manufacture
Render
Drawing
Simulation
Material
Mesh
Graphics
Nest
Network
Data Collection and Use
Unit and Value Display
Simulation and Generative Design
Default Units
Design
Manufacture
Simulation and Generative Design
Preview
Preferences controlling general UI behavior
User language English
Graphics driver Auto-select
Offline cache time period (days) 15
Skip launch items creation when live update
Automatic version on close
Recovery time interval (min) 5
Default modeling orientation Z up
Controls scene up
Show tooltips
Show command prompt
Show default measure
Show in-command errors and warnings
Show Fusion Team notifications
Pan, Zoom, Orbit shortcuts Fusion
Default Orbit type Constrained Orbit
Reverse zoom direction
Enable camera pivot
Use gesture-based view navigation
Restore Defaults
Apply
OK
Cancel

Setting up Assembly

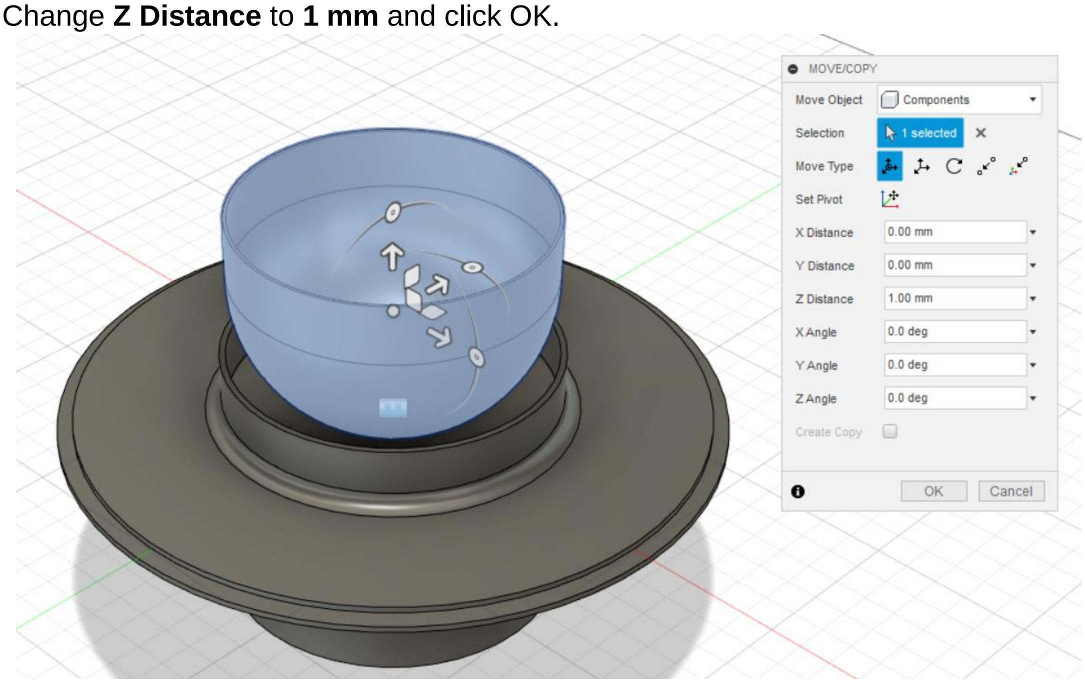
2. Click on File -> New Design.
3. Click on File -> Save, and name the design “Deep_Drawing_Full”.
4. Drag the **Die** model into the design space.
5. Drag the **Blank** model into the design space.
6. Hold the Control key, then select the bottom circular edge of the blank and the top outer edge of the Die. Then click on menu: Assembly -> Joint. Click Ok in the pop up Joint menu.



7. Drag the **Blank Holder** model into the design space.
8. Just like in Step 5. Select the bottom outer edge of the Blank Holder and the top edge of the Blank and make a joint.



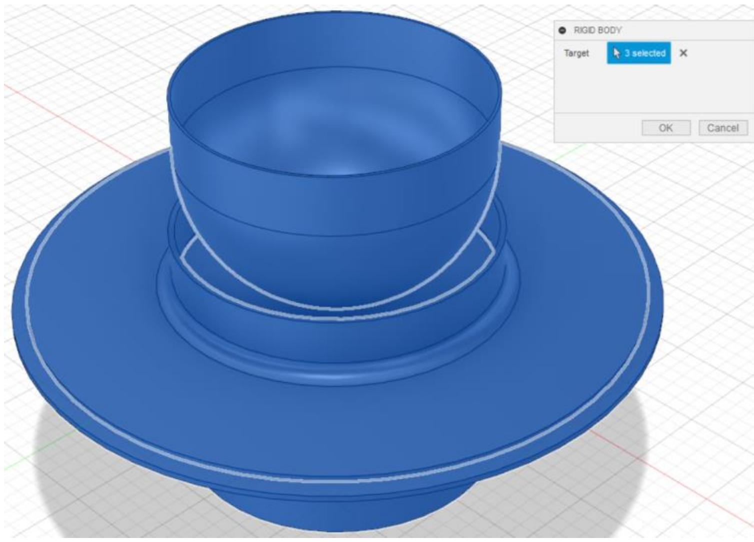
9. Drag the **Punch Model** into design space. In the **Move/Copy** menu that appears. Change **Z Distance** to **1 mm** and click OK.



Setting Up Simulation

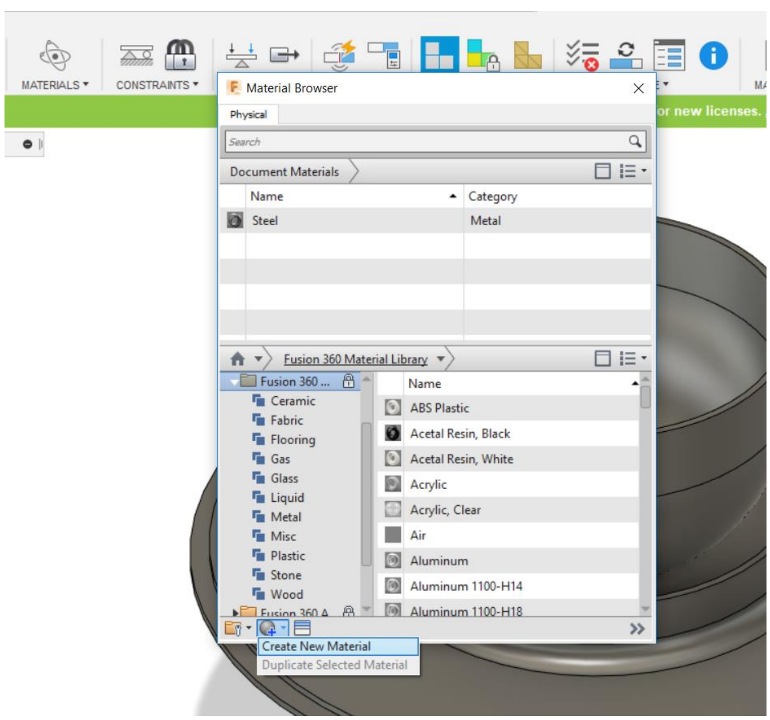
10. Save the design. And Switch to the Simulation environment. Design -> Simulation. In the "New Study" pop-out menu select **Event Simulation (Preview)**. Click on **Create Study**.

11. Under Study, Select **Rigid Bodies** and click the edit pencil icon. Use the cursor to select the punch, blank holder, and die.

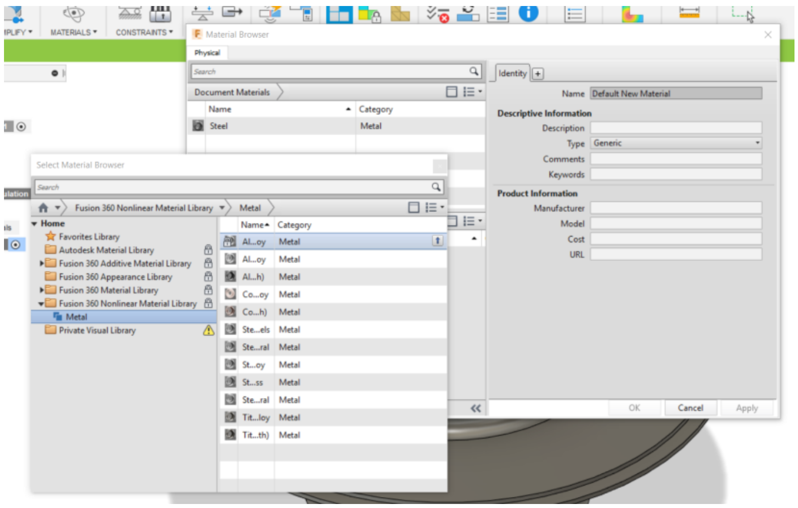


Adding New Non-Linear Material to Study

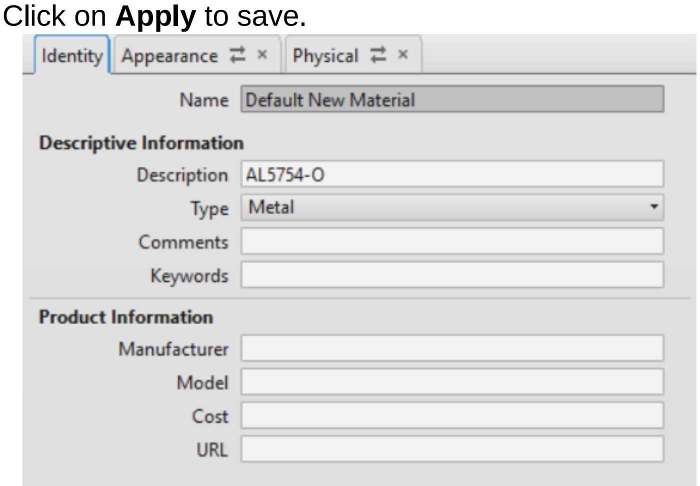
12. Click on **Materials -> Manage Physical Materials.** On the bottom left, click on the sphere to and choose **Create New Material.**

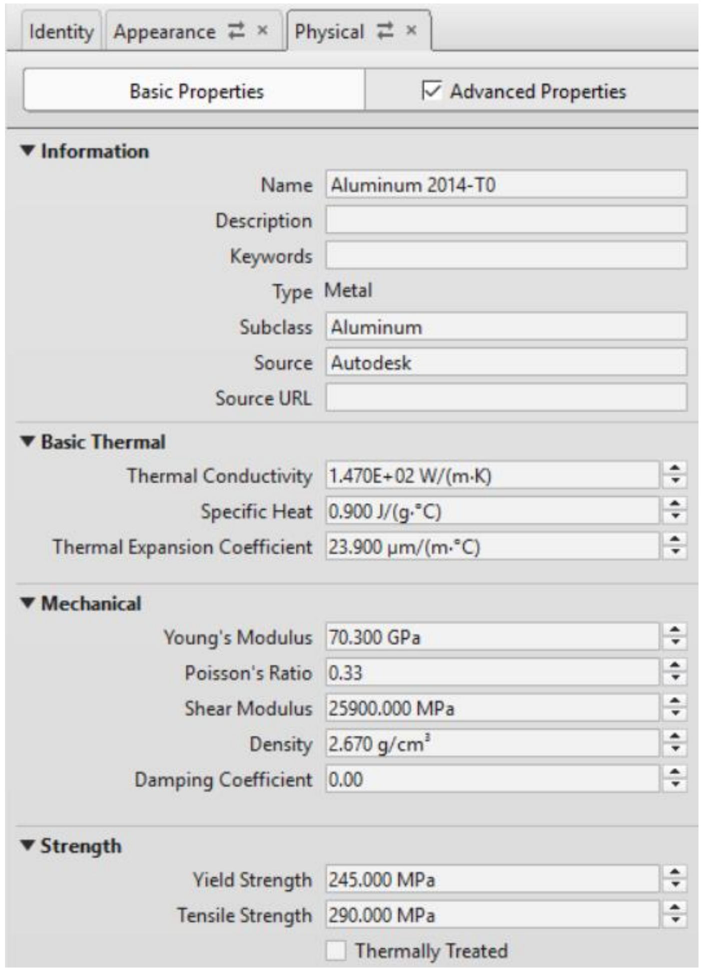


13. Under **Fusion 360 Nonlinear Material Library**, select **Metal**, then double-click on any of the aluminum alloys.



14. Close the **Select Material Browser** window. Fill out the **Identity** and **Physical** tabs as show below and Click on **Apply** to save.





15. Click on the **Advanced Properties** tab.
- For **Material Model** select **Isotropic**.
- For **Behavior** select **Nonlinear**.
- For **Type** select **Plastic**.
- In the **Strain-Stress** table hold the Control key, select all the highlightable entries and right-click, and delete.
- Under **Plastic Parameters** choose **Isotropic.**
- Under **Yield Function**, choose **von Mises** for **Yield Criterion**, and set **Initial Yield Stress** to **118.0 MPa**.

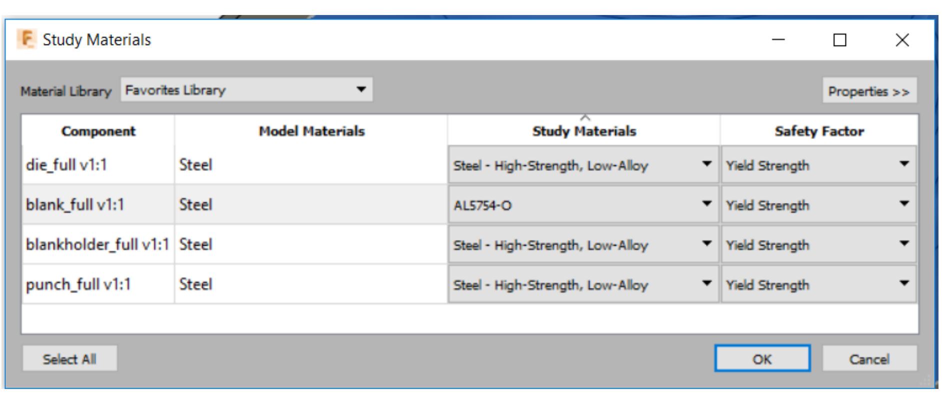
16. Enter the following True Strain – True Stress into the table.

True Strain	True Stress
0.017	135
0.02	141
0.026	153
0.036	170
0.046	186
0.056	200
0.066	212
0.076	223
0.086	232
0.096	240
0.111	251
0.131	262
0.151	271
0.171	278
0.191	283
0.211	287
0.231	291
1.011	302

17. Click on **Apply**. Right-click on **Default New Material** and select **Rename**. Type in **AL5754-OAU.** Close the Material Browser.

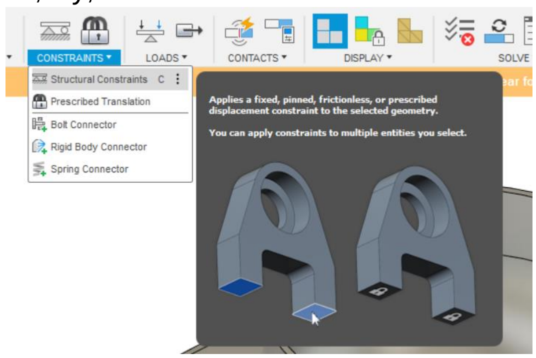
Assign Materials

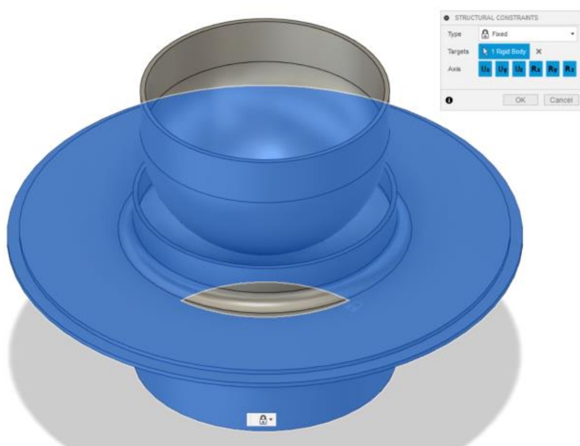
18. Under Study 1, Click on **Study Materials** and Edit. In the **Material Library** dropdown, select Fusion 360 Nonlinear Material Library.
    - Select the **Study Material** for the Die, Punch, and Blank holder as **Steel – High-Strength, Low-Alloy**.
    - In the **Material Library** dropdown, select **Favorites** Library. Then for the Blank Study Material select **AL5754-OAU** (the material we created above).



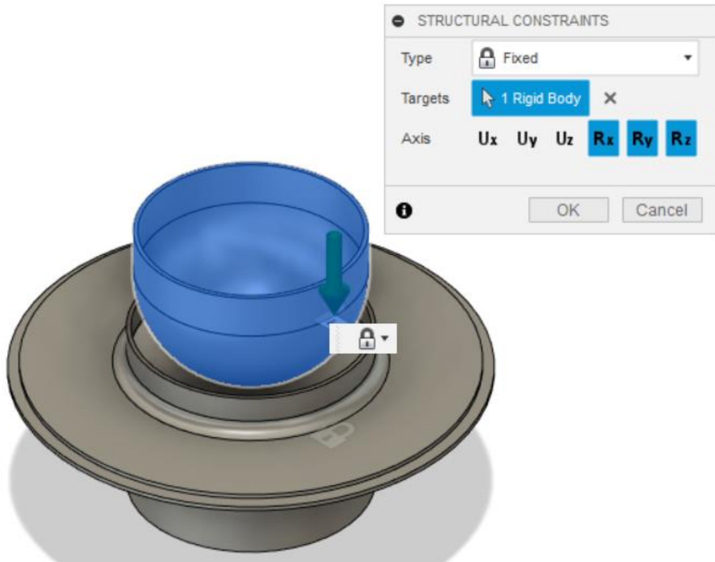
Set Loads and Boundary Conditions

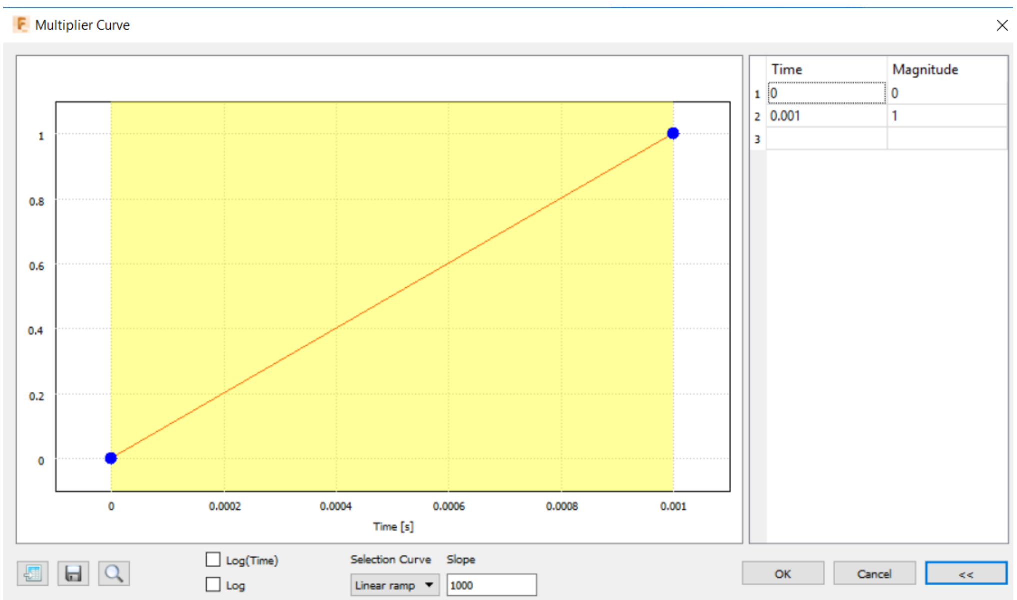
19. On the menu, click on **Constraints**, and select **Structure Constraints**. Select the die and fix it in Ux, Uy, Uz, Rx, Ry, and Rz. Click OK.





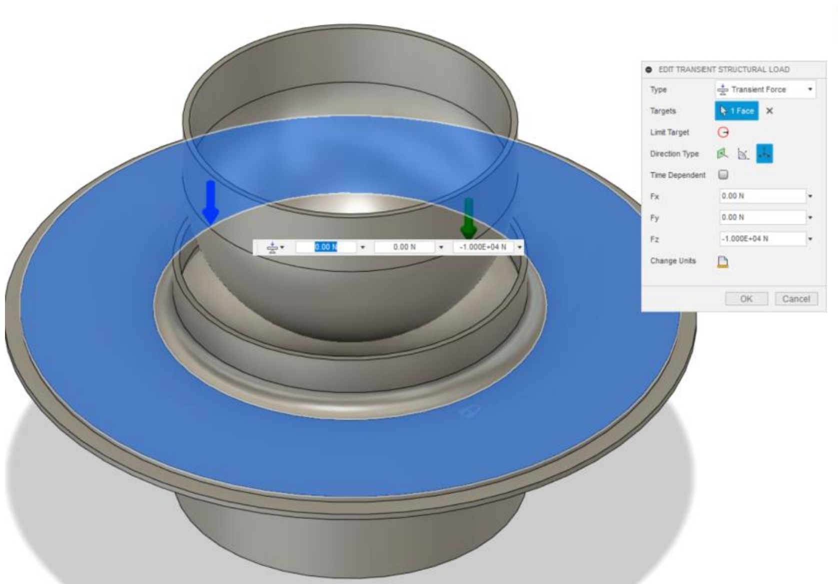
20. On the menu, click on **Constraints**, and select **Prescribed Translation**.
    - Select the Punch model.
    - Make sure only Uz is selected under Components (Unselect Ux and Uy by clicking on the icons)
    - Enter **-57** mm for **Magnitude Uz.**
    - Click on Multiplier curve. **Selection Curve** should be set to **Linear Ramp**. Time – Magnitude table should match the image below.
    - Click OK.
21. On the menu, click on **Constraints**, and select **Structure Constraints**. Select the Punch and fix it in Rx, Ry, and Rz. Click OK. (Sometimes body must be selected from Model Components -> Punch_full -> Bodies -> Body1 before selecting Constraints).





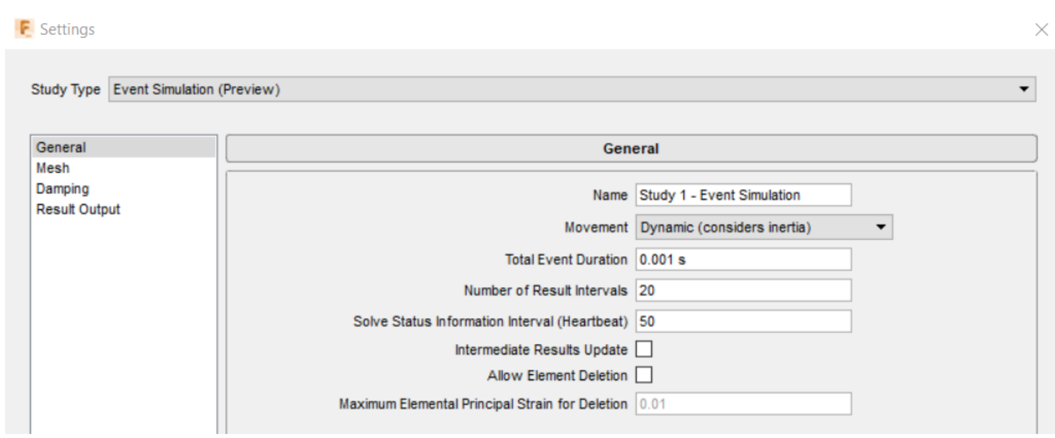
22. On the menu, click on **Loads**, and select **Transient Loads**.
    - Type: Transient Force
    - Targets: Select top face of the blank holder as shown in image below.
    - Direction Type: Vectors
    - Time Dependent: Leave unchecked
    - Fx: 0.0N, Fy: 0.0N, Fz: -10000N

Click Ok.



Setting Up Simulation Parameters

23. Right-click Study 1, and select Settings. Under **General** set the following settings.
    - Movement: Dynamics
    - Total Event Duration: 0.001s
    - Number of Result Intervals: 20
    - Solve Status Information Interval (Heartbeat): 50
    - Intermediate Results Update: unchecked
    - Allow Element Deletion: unchecked

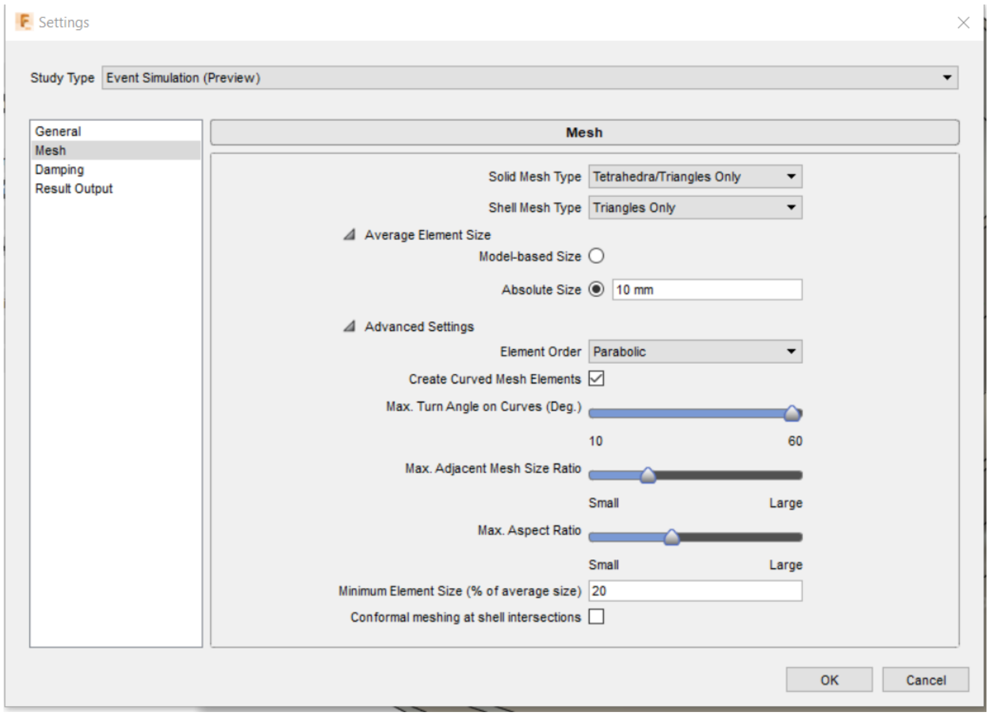


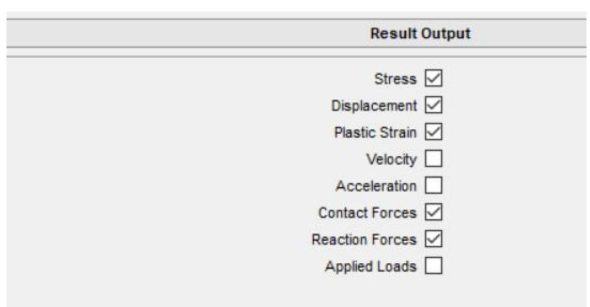
Under **Mesh**,

- Solid Mesh Type: Tetrahedra/Triangles Only
- Shell Mesh Type: Triangles Only
- Model-based size: unchecked

- Absolute Size: 10mm
- Element Order: Parabolic
- Create Curved Mesh Elements: checked

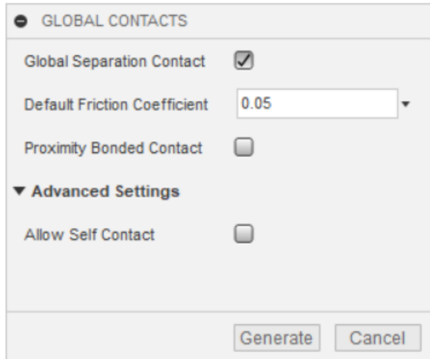
Under Results Output Check **Stress, Displacement, Plastic** Strain (this comes back as strain)**, Contact Forces, and Reaction Forces.**



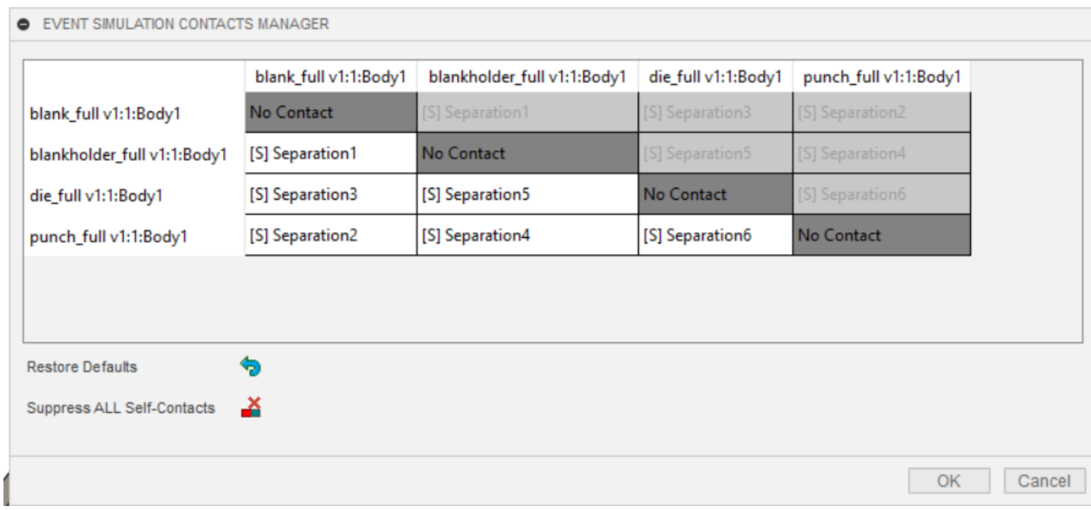


Setting Up Contact

24. Under Study 1. Right-click **Contacts** and select **Global Contacts**. For **Default Friction Coefficient** enter 0.05. Proximity Bonded Contact should be unchecked. Under Advanced Settings, make sure **Allow Self Contact** is unchecked. Click on Generate.

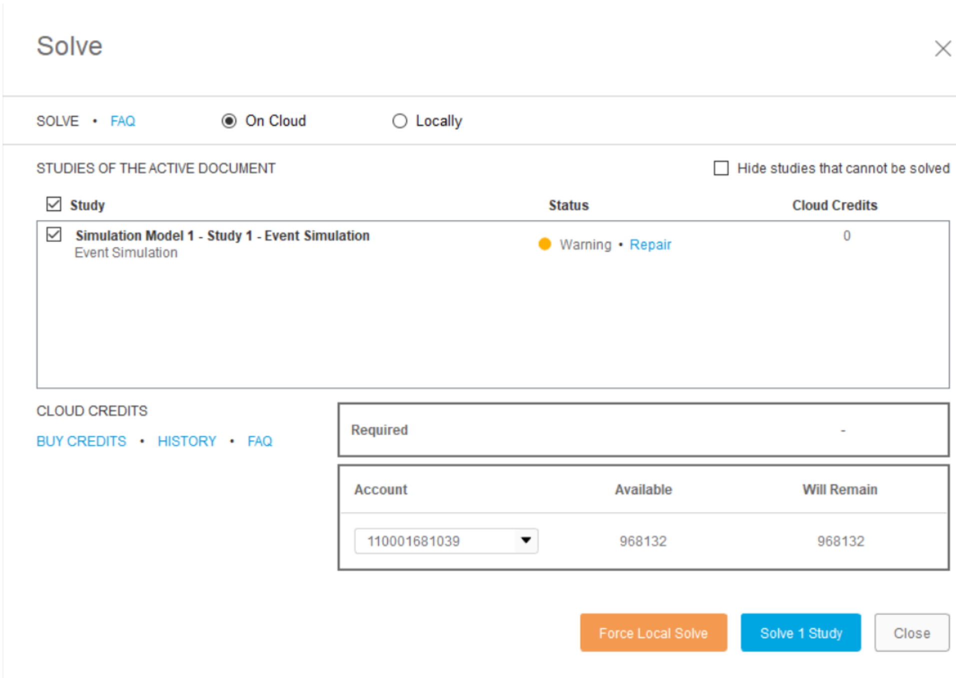


Right-click on **Contacts** again and select **Manage Contacts**. The contact conditions should look like the image below.



Performing Cloud Solve

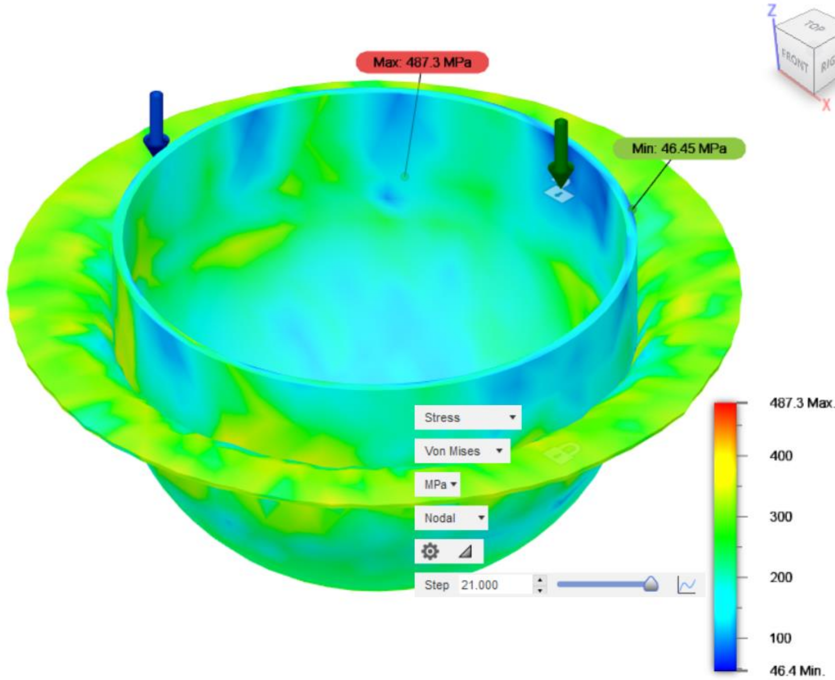
25. In menu, under **Solve**, select **Solve**. Then in the pop up window select **Solve**.

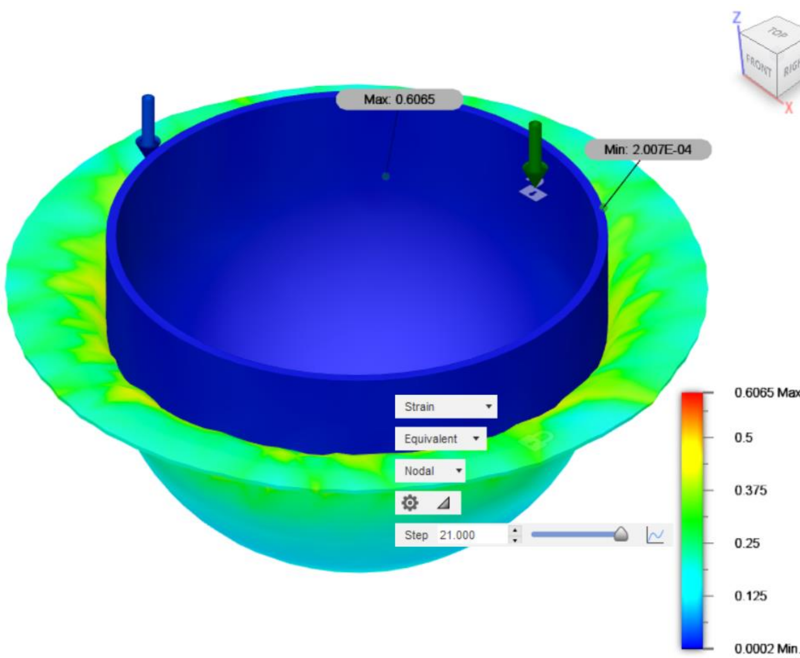


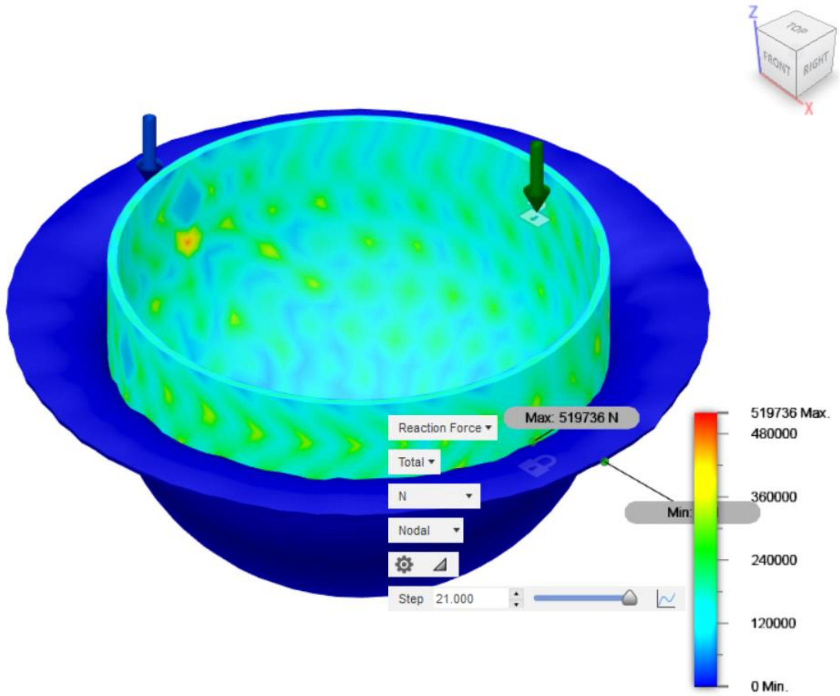
The mesh can be view now by toggling the visibility (eye) to the left of the **Mesh,** under Study 1

Viewing Results and Generating Animation

26. Right-click Results in the Study tree, and click on **View Results.**
27. Under Model Components, use visibility toggle to hide die and blank holder.
28. In the menu, make sure **Deformation -> Actual** is selected.
29. Use horizontal scroll bar to go over simulation steps and use drop down menus to change responses being displayed.

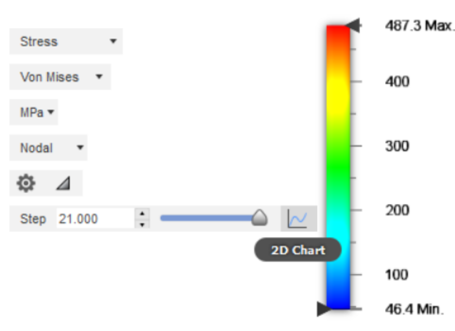

Max: 487.3 MPa
Min: 46.45 MPa
Stress
Von Mises
MPa
Nodal
Step 21.000
487.3 Max.
400
300
200
100
46.4 Min.

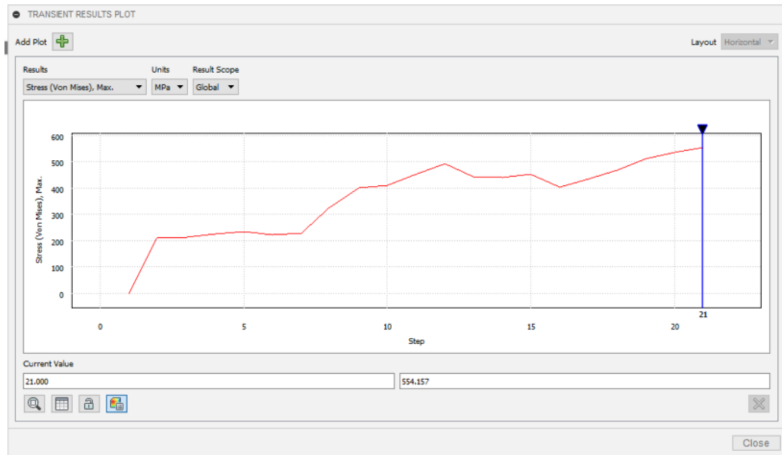

Max: 0.6065
Min: 2.007E-04
Strain
Equivalent
Nodal
Step 21.000
0.6065 Max.
0.5
0.375
0.25
0.125
0.0002 Min.

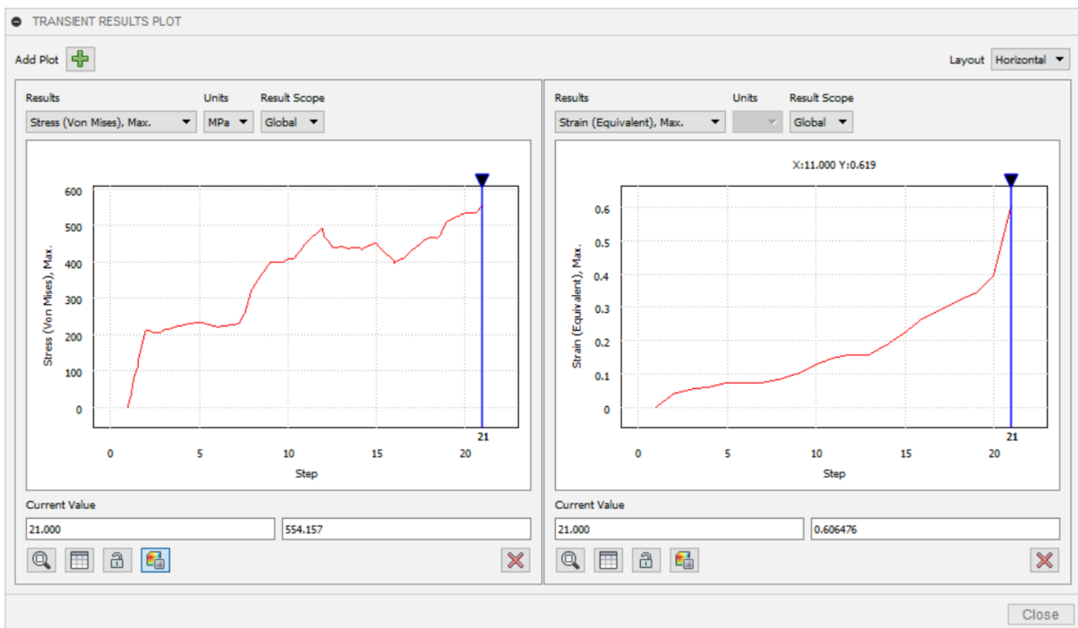


30. Click on **Results Tools** menu, click on **Animate**. Select number of steps and play. If everything looks good, select record and save animation.

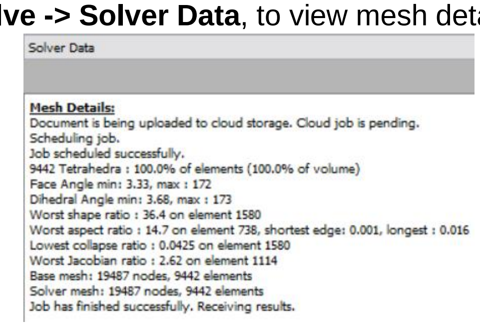
31. Use the **2D Chart** icon in the results view to plot get 2D plots of the responses. Use Add Plot button to view multiple 2D plots at a time. Use slider to query response values at the output times specified.







32. On the menu click on **Solve -> Solver Data**, to view mesh details.


Solver Data

Mesh Details:
Document is being uploaded to cloud storage. Cloud job is pending.
Scheduling job.
Job scheduled successfully.
9442 Tetrahedra : 100.0% of elements (100.0% of volume)
Face Angle min: 3.33, max : 172
Dihedral Angle min: 3.68, max : 173
Worst shape ratio : 36.4 on element 1580
Worst aspect ratio : 14.7 on element 738, shortest edge: 0.001, longest : 0.016
Lowest collapse ratio : 0.0425 on element 1580
Worst Jacobian ratio : 2.62 on element 1114
Base mesh: 19487 nodes, 9442 elements
Solver mesh: 19487 nodes, 9442 elements
Job has finished successfully. Receiving results.

33. To view solver output, on menu click **on Solve -> Solver Data**. Change the drop down from Solver Data to **Solver Output**. This shows detail solver output which includes total run time of the simulation, time step used, etc. The Kinetic, Internal, Viscous, and Total Energies and External Work are also displayed at each heartbeat interval.

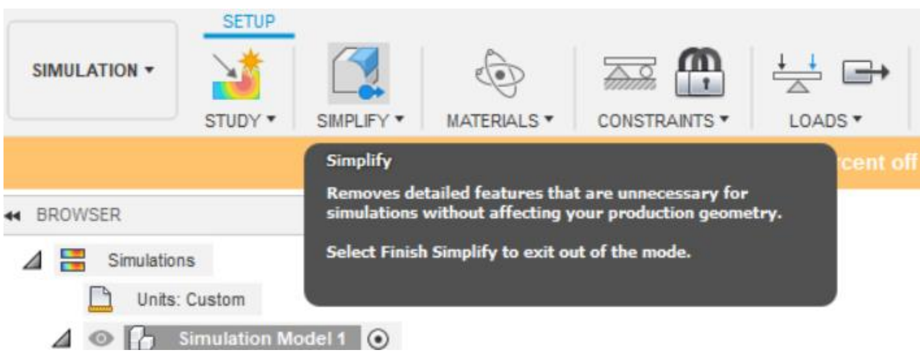
Eighth Symmetry Model

Due to symmetric nature of the cup being formed, the simulation time can be reduced by forming an eighth of the blank with symmetric boundary conditions at the edges to capture the correct responses for the full-scale model.

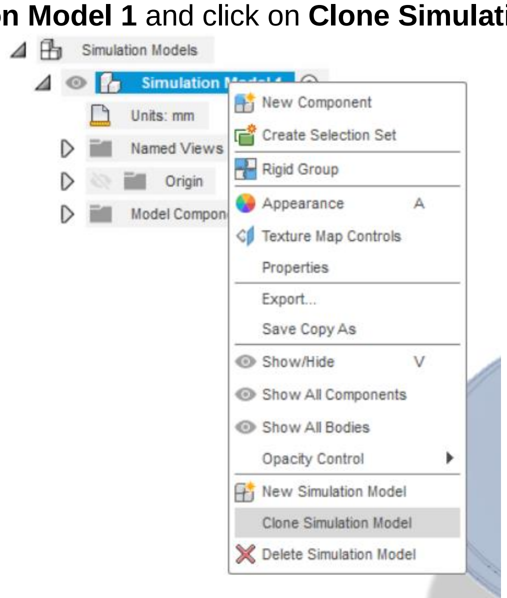
Step-by-step instructions

Setting up Assembly

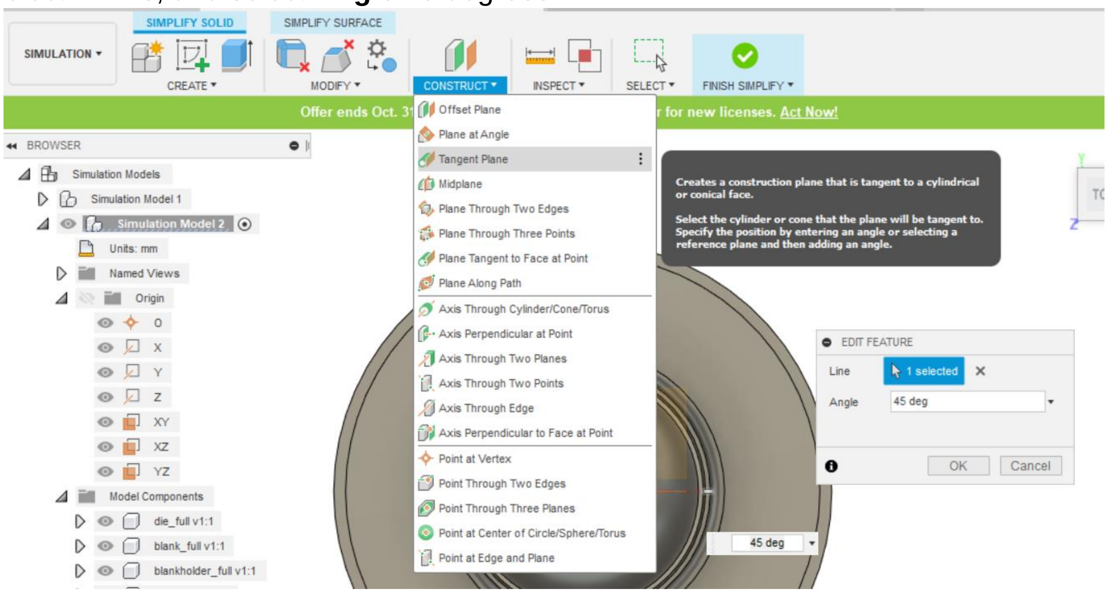
34. On menu click on **Simplify**.



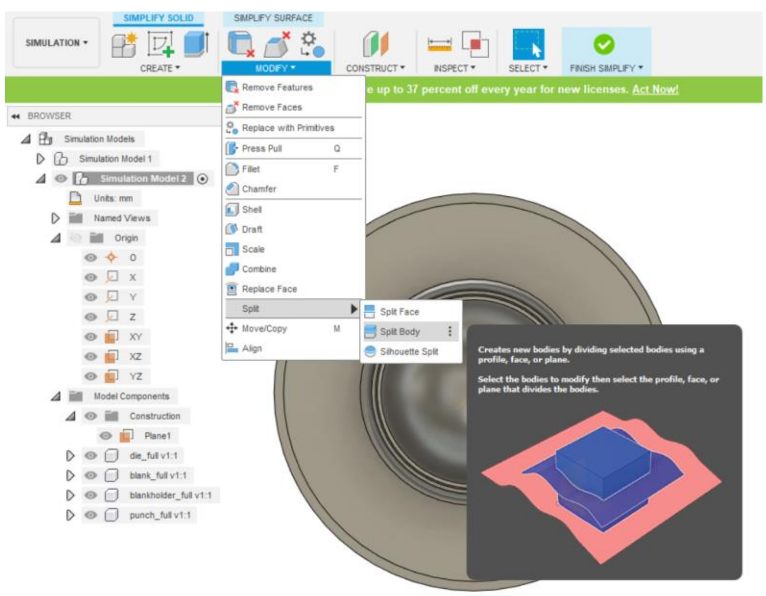
35. Right-click on **Simulation Model 1** and click on **Clone Simulation Model**. Click OK.

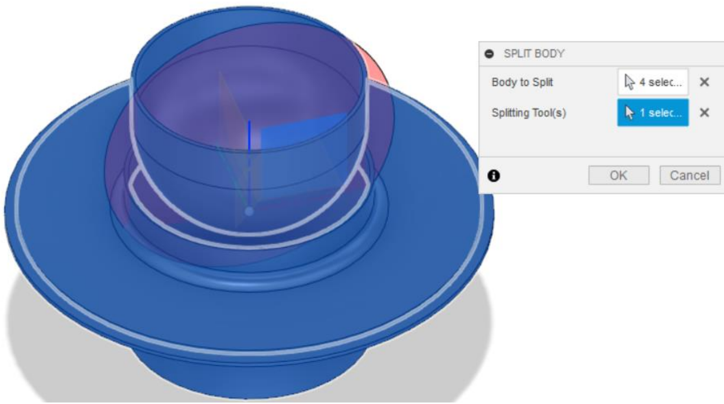


36. On the menu click on **Construct -> Plane At Angl**e. Under **Simulation Model 2** tree, select **Z** Axis, and select **Angle 45** degrees.

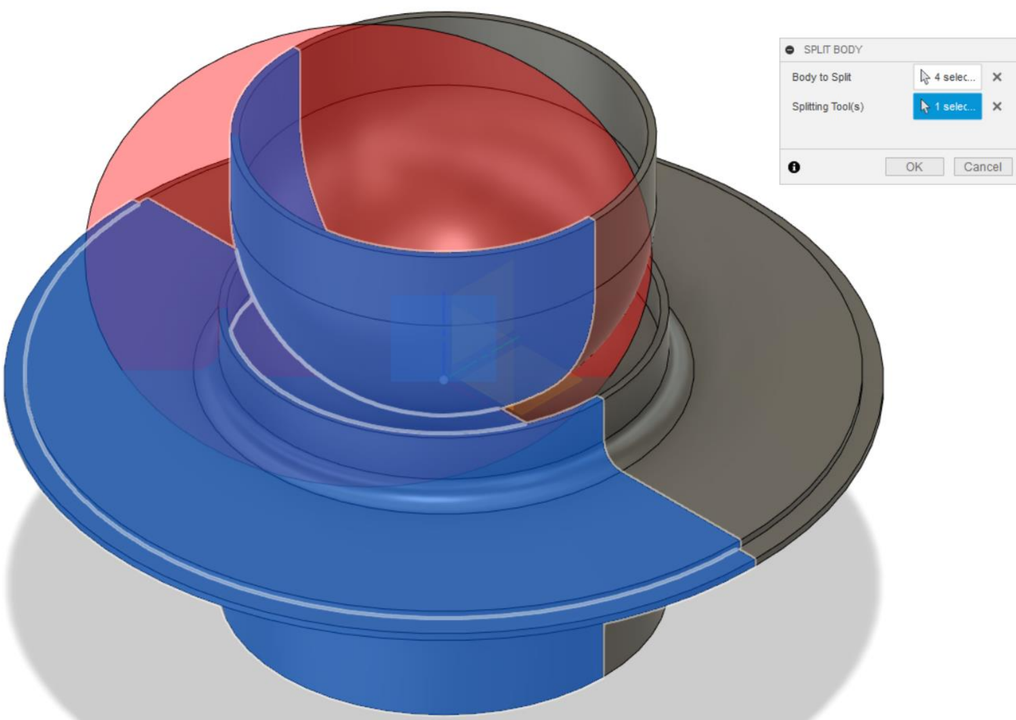


37. On menu click on **Modify -> Split -> Split Body**. For **Bodies to Split**, select all the bodies in the model (die, punch, blank holder, blank), and for **Splitting Tool** select **XZ Plane**.

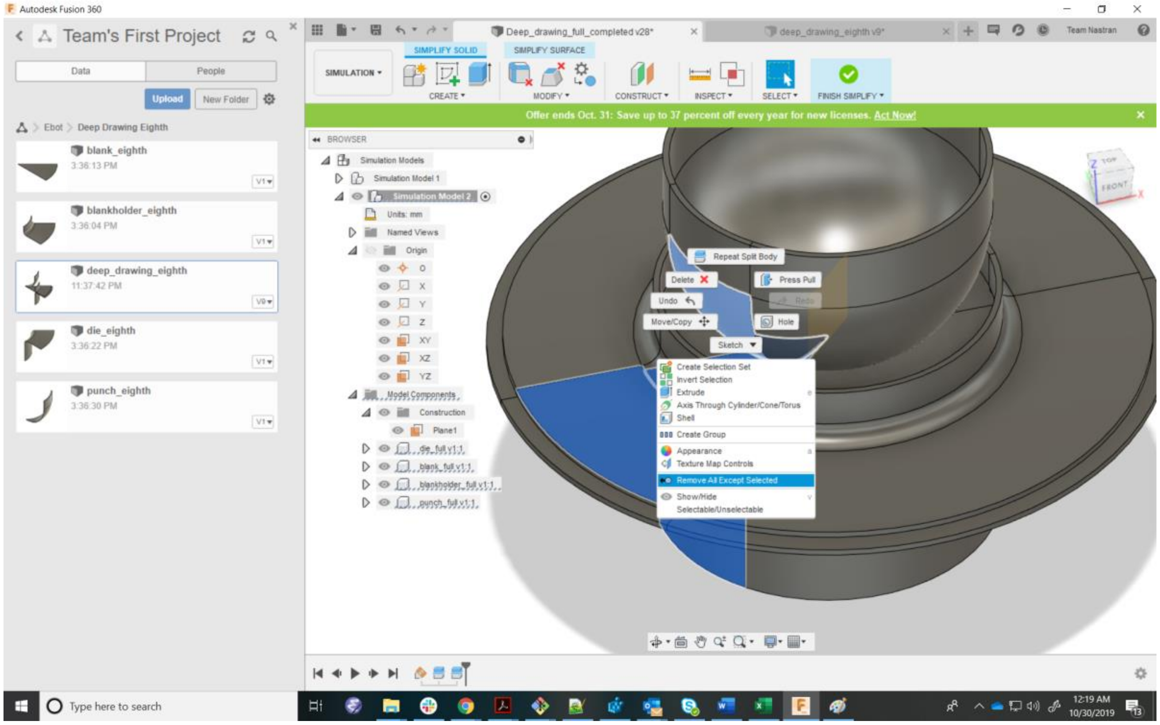




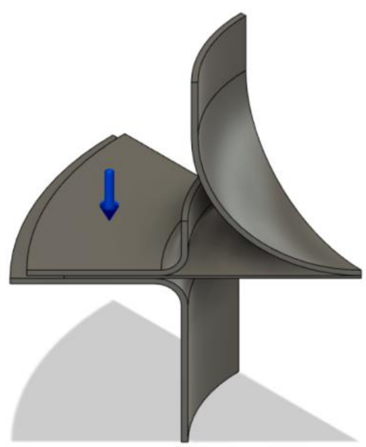
38. Just like in Step 29, select **Split Body** again but this time select only the split bodies on the front side as shown below. For Splitting Tool, select the plane that was created in Step 28 (can be found under Model Components -> Construction).



39. Hold Control and select a face from each component in the smaller wedge, then right-click and select **Remove All Except Selected**.

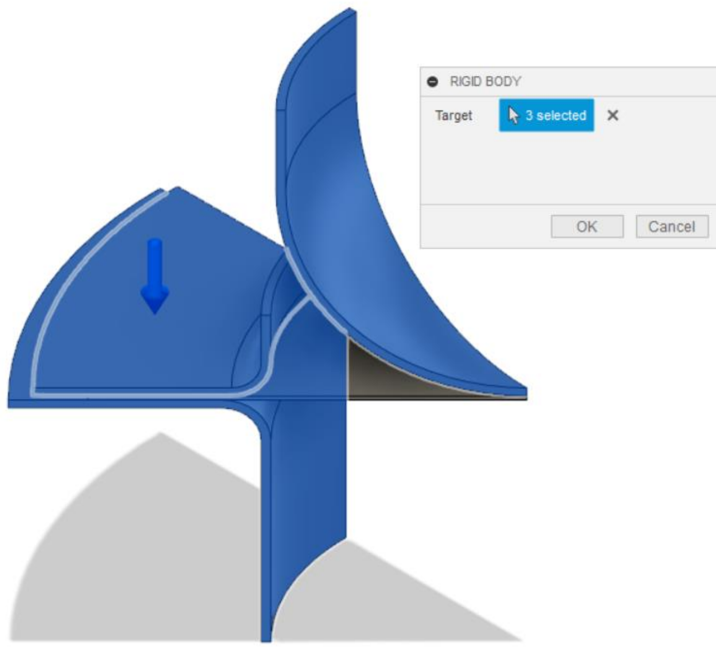


40. On the menu, click on Finish Simplify. You should now see the eighth model.



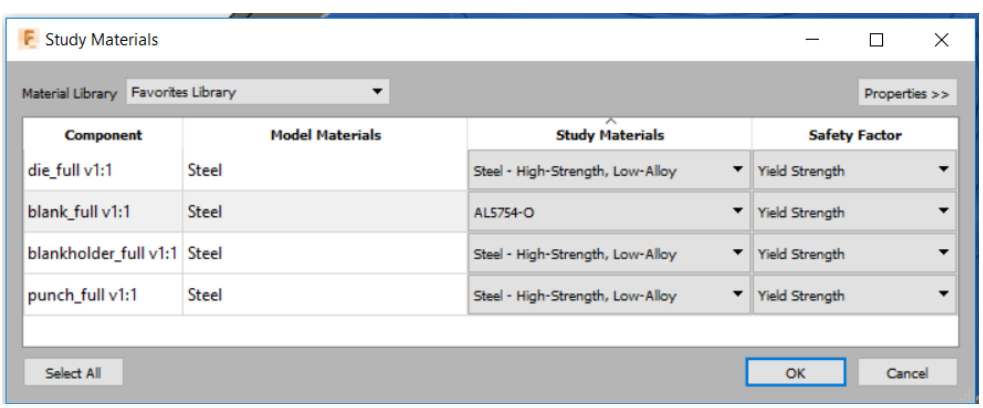
Setting Up Simulation

41. Under Study, Select **Rigid Bodies** and click the edit pencil icon. Use the cursor to select the punch, blank holder, and die.



Assign Materials

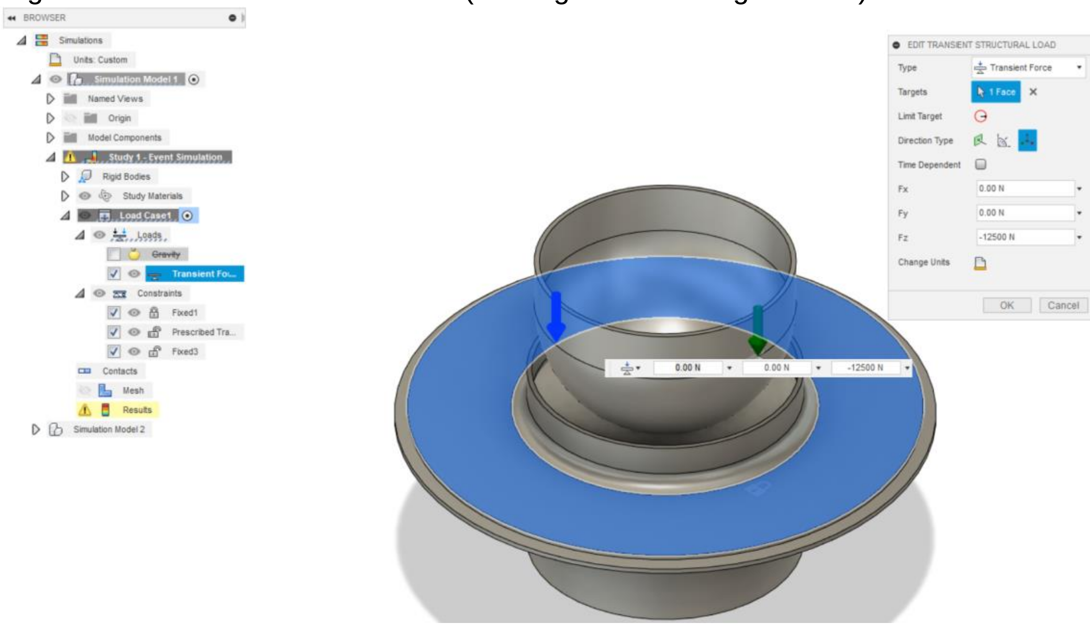
42. Under Study 1, Click on **Study Materials** and Edit. In the **Material Library** dropdown, select Fusion 360 Nonlinear Material Library.
    - Select the **Study Material** for the Die, Punch, and Blank holder as **Steel – High-Strength, Low-Alloy**.
    - In the **Material Library** dropdown, select **Favorites** Library. Then for the Blank Study Material select **AL5754-OAU** (the material we created above).



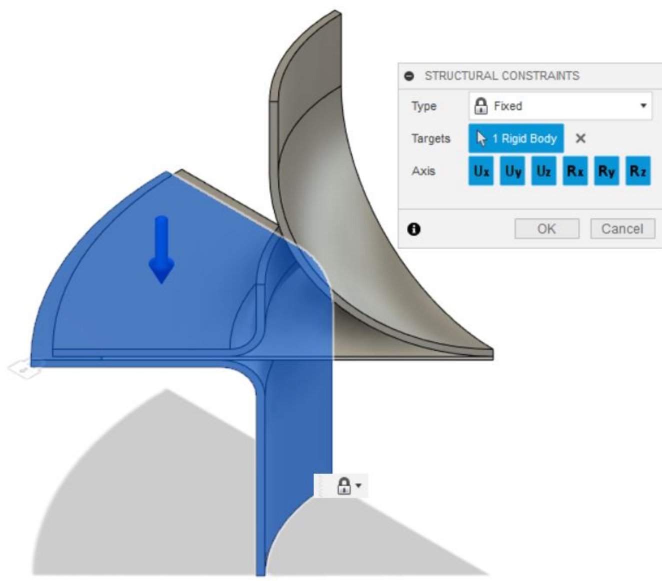
Set Loads and Boundary Conditions

Transient Force on Blank holder should be preserved after the simplify, but other boundary conditions would need to be redone.

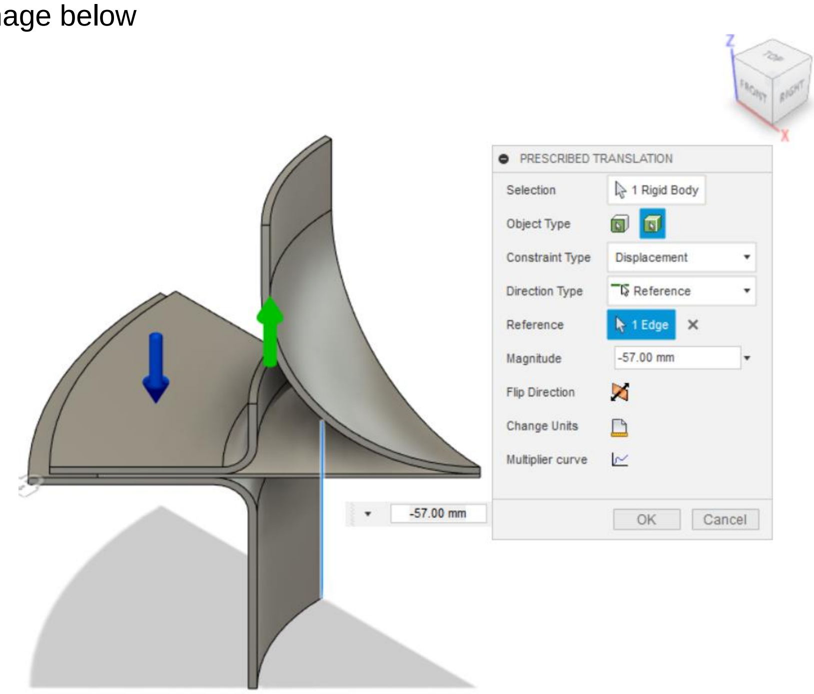
43. Edit the transient force by clicking on pencil icon as shown in below. Change the magnitude of the **Fz** force to **-1250N** (one eighth of the original load).



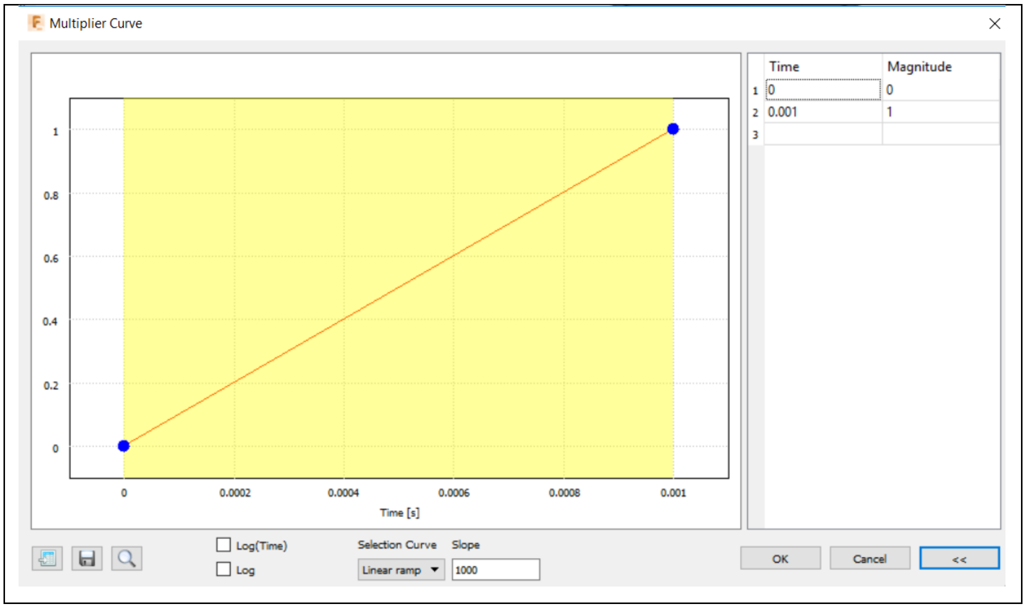
44. Delete all existing constraints (they should all have warnings).
45. On the menu, click on **Constraints**, and select **Structure Constraints**. Select the die and fix it in Ux, Uy, Uz, Rx, Ry, and Rz. Click OK.



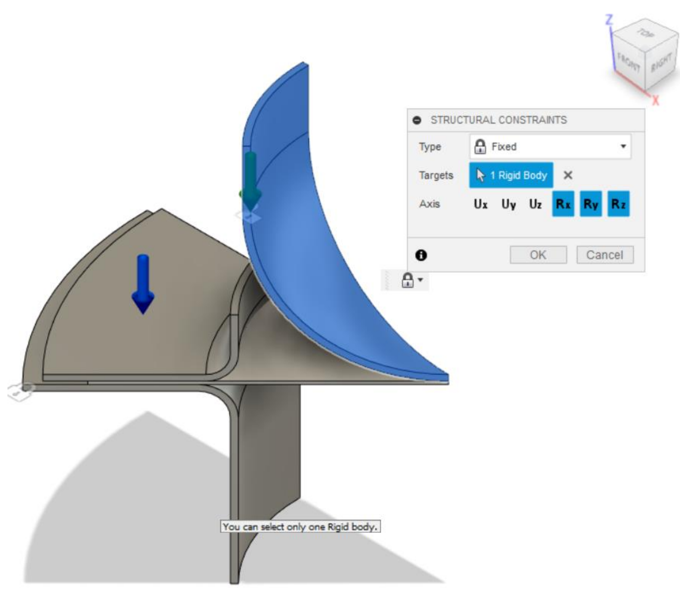
46. On the menu, click on **Constraints**, and select **Prescribed Translation**.
    - Select the Punch model.
    - Constraint Type: Displacement
    - Direction Type: Reference
    - For Reference Edge select the far side straight vertical edge of the die as shown in the image below



- Magnitude: **-57mm**
- Click on Multiplier curve. **Selection Curve** should be set to **Linear Ramp**. Time – Magnitude table should match the image below.
- Click OK. If done correctly prescribed displacement should point downward in the model.

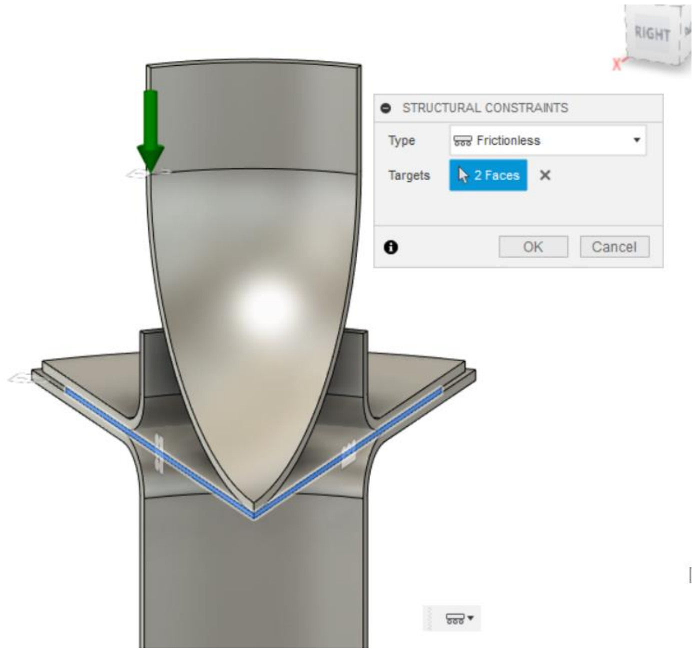


47. On the menu, click on **Constraints**, and select **Structure Constraints**. Select the Punch and fix it in Rx, Ry, and Rz. Click OK. (Sometimes body must be selected from Model Components -> Punch_full -> Bodies -> Body3 before selecting Constraints).



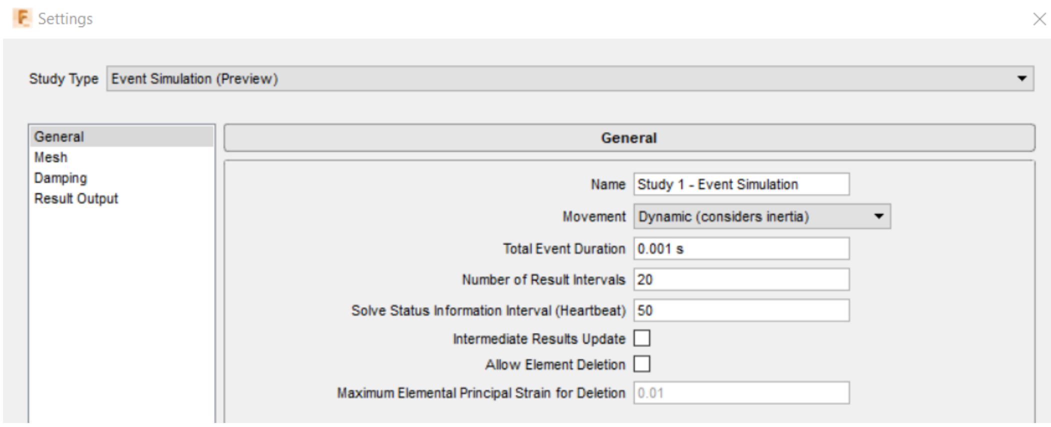
Symmetry Boundary Conditions

48. On the menu, click on **Constraints**, and select **Structure Constraints**. Select both inside edges of the blank as shown in the image below, and Select Type **Frictionless**. Click OK.



Setting Up Simulation Parameters

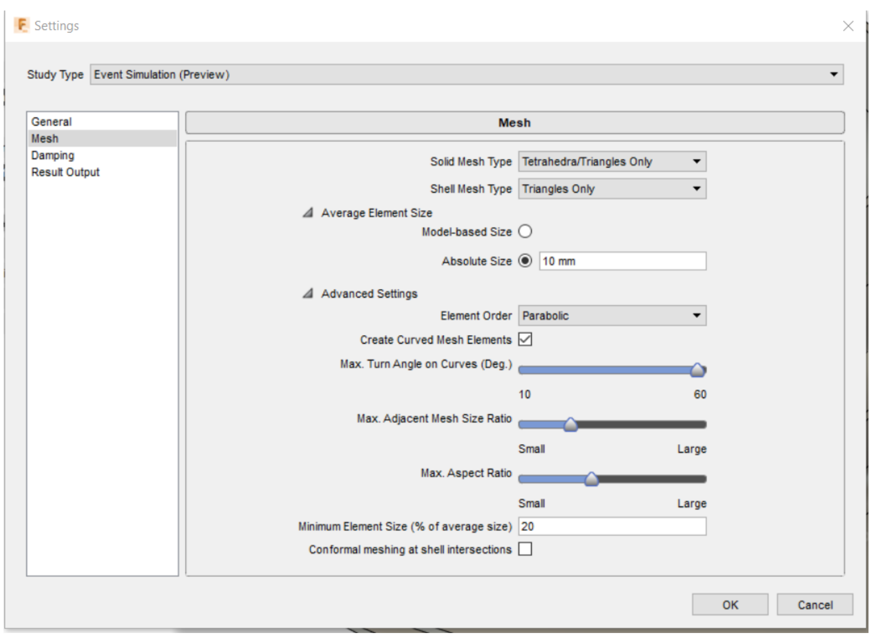
49. Right-click Study 1, and select Settings. Under **General** set the following settings.
    - Movement: Dynamics
    - Total Event Duration: 0.001s
    - Number of Result Intervals: 20
    - Solve Status Information Interval (Heartbeat): 50
    - Intermediate Results Update: unchecked
    - Allow Element Deletion: unchecked

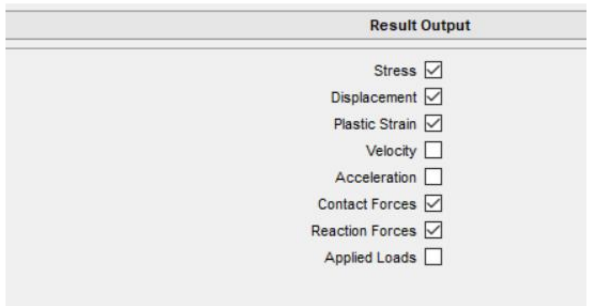


Under **Mesh**,

- Solid Mesh Type: Tetrahedra/Triangles Only
- Shell Mesh Type: Triangles Only
- Model-based size: unchecked
- Absolute Size: 10mm
- Element Order: Parabolic
- Create Curved Mesh Elements: checked

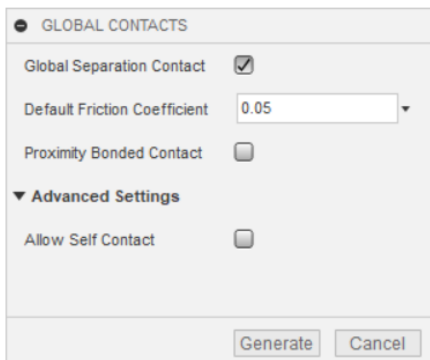
Under Results Output Check **Stress, Displacement, Plastic Strain, Contact Forces, and Reaction Forces.**



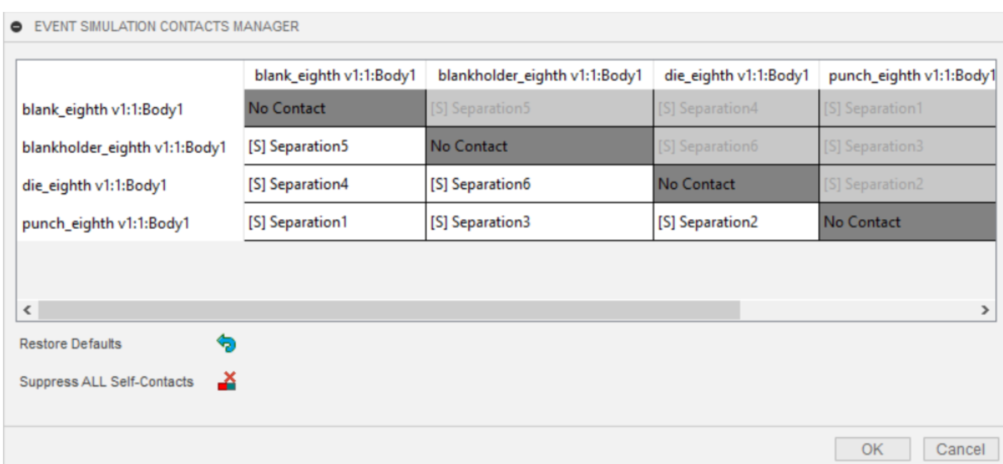


Setting Up Contact

50. Under Study 1. Right-click **Contacts** and select **Global Contacts**. For **Default Friction Coefficient** enter 0.05. Proximity Bonded Contact should be unchecked. Under Advanced Settings, make sure **Allow Self Contact** is unchecked. Click on Generate.

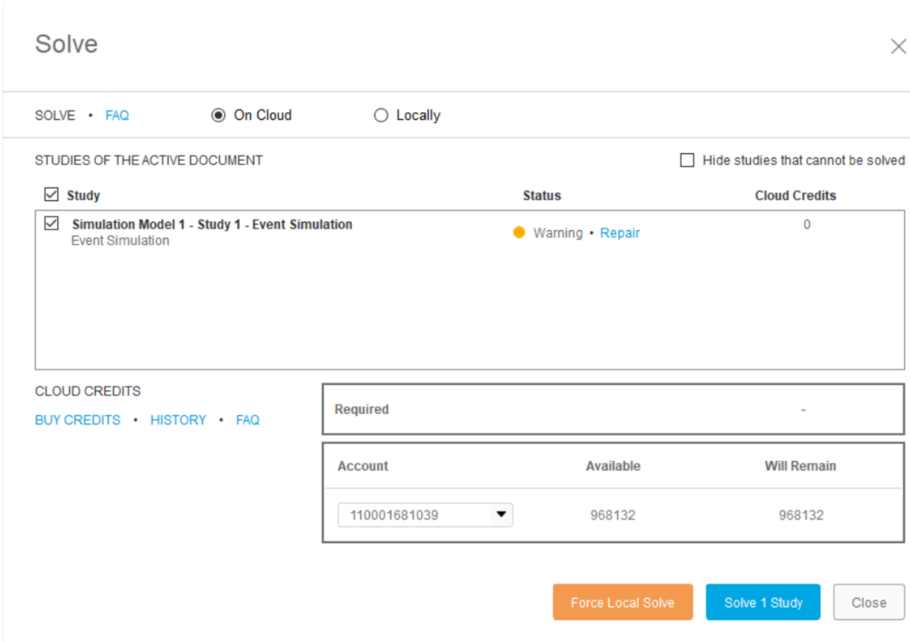


Right-click on **Contacts** again and select **Manage Contacts**. The contact conditions should look like the image below.



Performing Cloud Solve

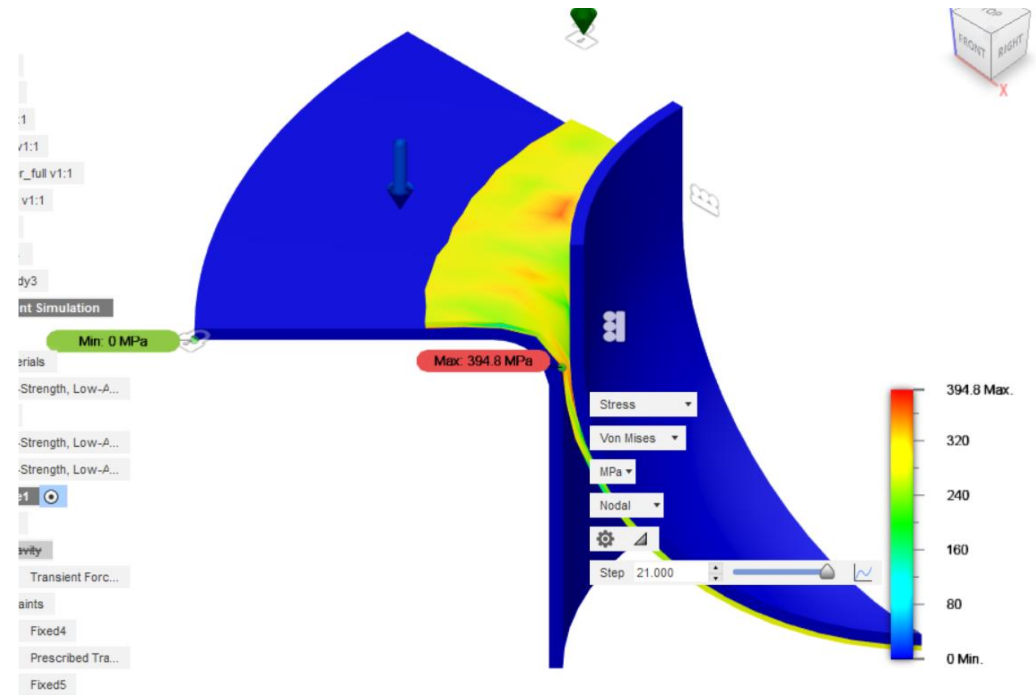
51. In menu, under **Solve**, select **Solve**. Then in the pop up window select **Solve**.

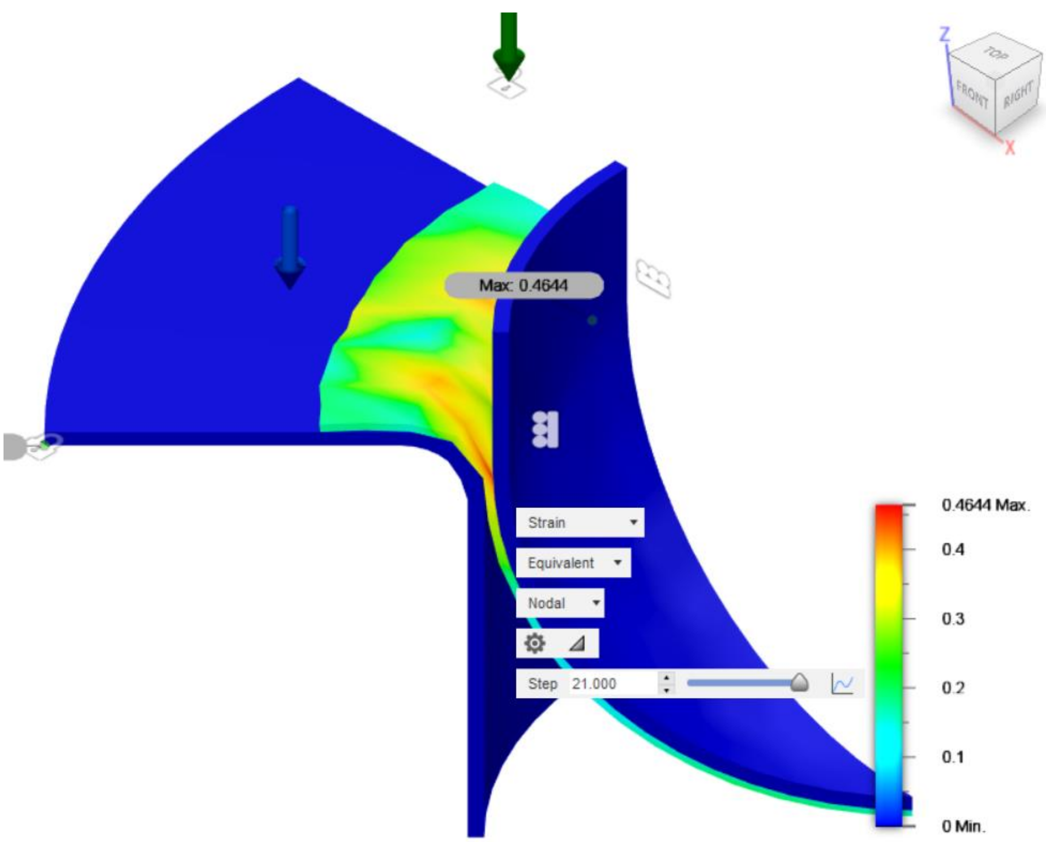


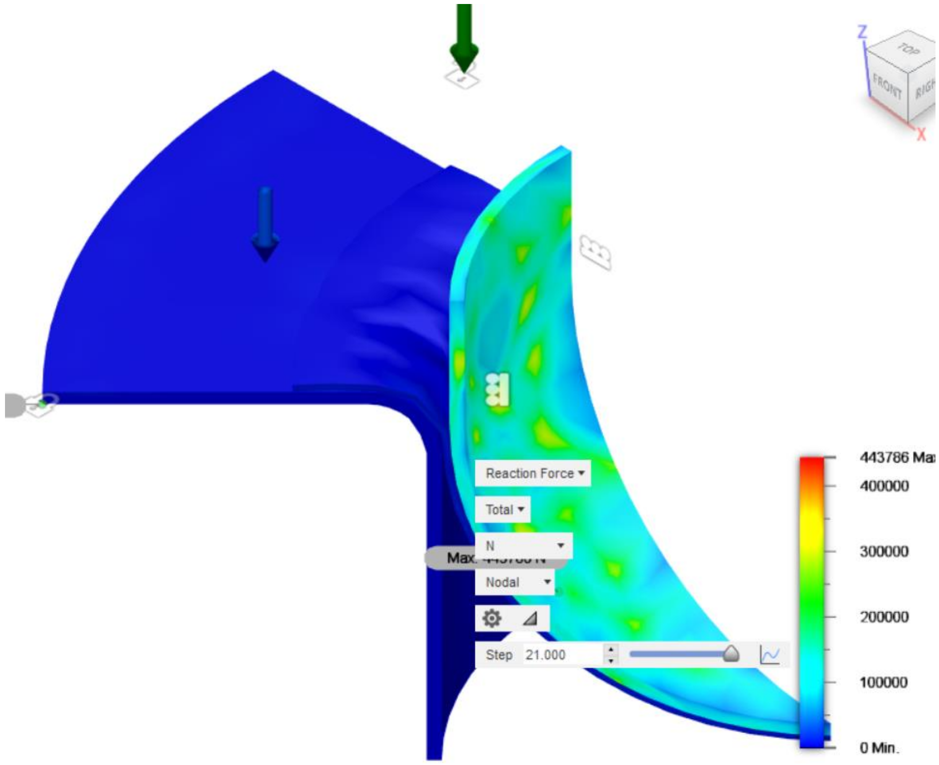
The mesh can be view now by toggling the visibility (eye) to the left of the **Mesh,** under Study 1.

Viewing Results and Generating Animation

52. Right-click Results in the Study tree, and click on **View Results.**
53. Under Model Components, use visibility toggle to hide blank holder.
54. In the menu, make sure **Deformation -> Actual** is selected.
55. Use horizontal scroll bar to go over simulation steps and use drop down menus to change responses being displayed.

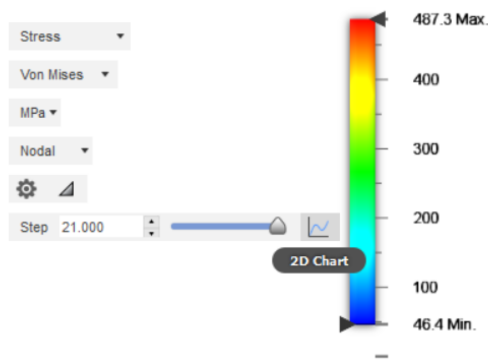

Min: 0 MPa
Max: 394.8 MPa
nt Simulation
Strength, Low-A...
Transient Forc...
Fixed4
Prescribed Tra...
Fixed5
Stress
Von Mises
MPa
Nodal
Step 21.000
394.8 Max.
320
240
160
80
0 Min.

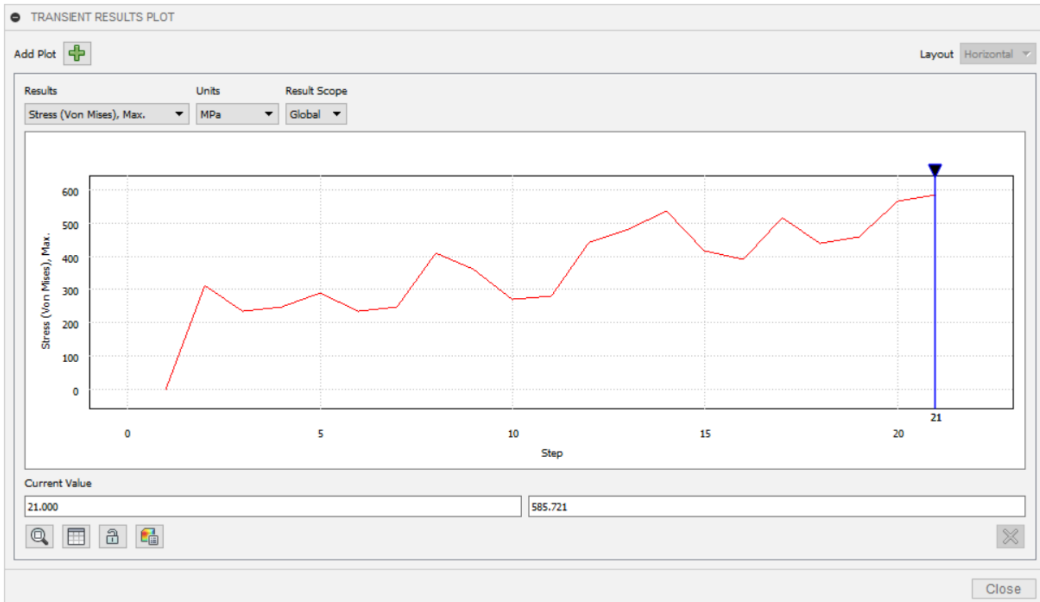

Max: 0.4644
Strain
Equivalent
Nodal
Step 21.000
0.4644 Max.
0.4
0.3
0.2
0.1
0 Min.

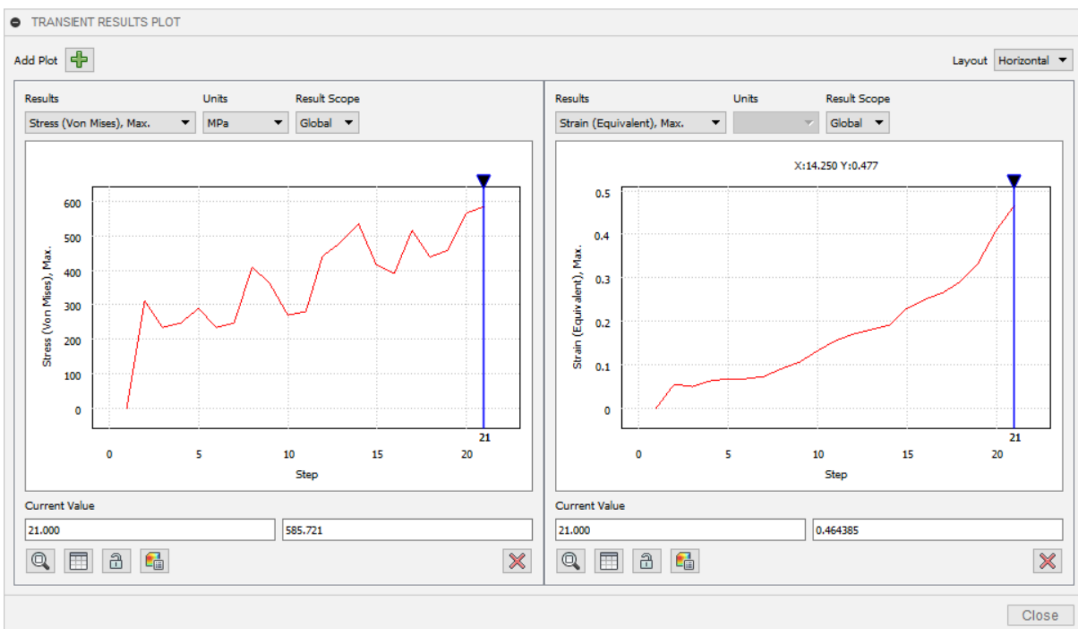


56. Click on **Results Tools** menu, click on **Animate**. Select number of steps and play. If everything looks good, select record and save animation.

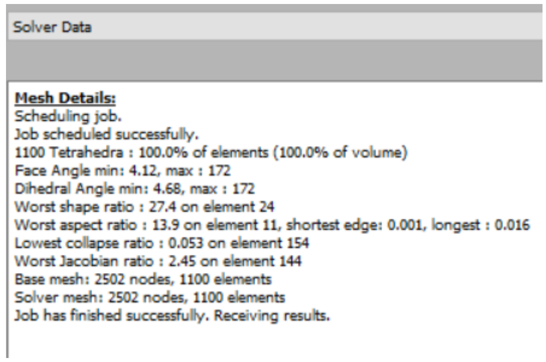
57. Use the **2D Chart** icon in the results view to plot get 2D plots of the responses. Use Add Plot button to view multiple 2D plots at a time. Use slider to query response values at the output times specified.







58. On the menu click on **Solve -> Solver Data**, to view mesh details.



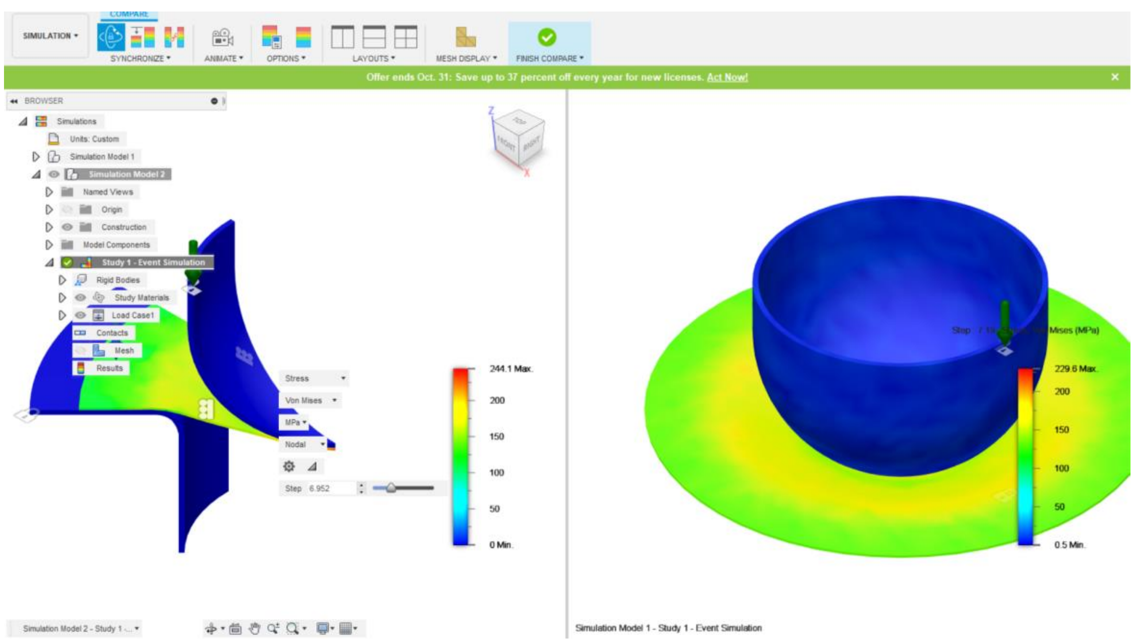
Solver Data

Mesh Details:
Scheduling job.
Job scheduled successfully.
1100 Tetrahedra : 100.0% of elements (100.0% of volume)
Face Angle min: 4.12, max : 172
Dihedral Angle min: 4.68, max : 172
Worst shape ratio : 27.4 on element 24
Worst aspect ratio : 13.9 on element 11, shortest edge: 0.001, longest : 0.016
Lowest collapse ratio : 0.053 on element 154
Worst Jacobian ratio : 2.45 on element 144
Base mesh: 2502 nodes, 1100 elements
Solver mesh: 2502 nodes, 1100 elements
Job has finished successfully. Receiving results.

59. To view solver output, on menu click **on Solve -> Solver Data**. Change the drop down from Solver Data to **Solver Output**. This shows detail solver output which includes total run time of the simulation, time step used, etc. The Kinetic, Internal, Viscous, and Total Energies and External Work are also displayed at each heartbeat interval.

Comparing to Full Scale Model

60. On the menu, click on **Compare**. In the split window, change the second results window to Simulation Component 1 – Study 1. Compare different responses side-by-side.



References

1. S.Jamshidifard, H. Ziaeipoor, H. Moosavi, H. Khademizadeh. “Investigation of hydrostatic counter pressure effect on thickness distribution in Hydromechanical deep drawing process with hemispherical punch”. Proceedings of the 2nd International Conference on Manufacturing Engineering, Quality and Production Systems