

ARES Workshop - Ansys FEA

Cas Kent, Ben van der Veer, Hasan al Banna





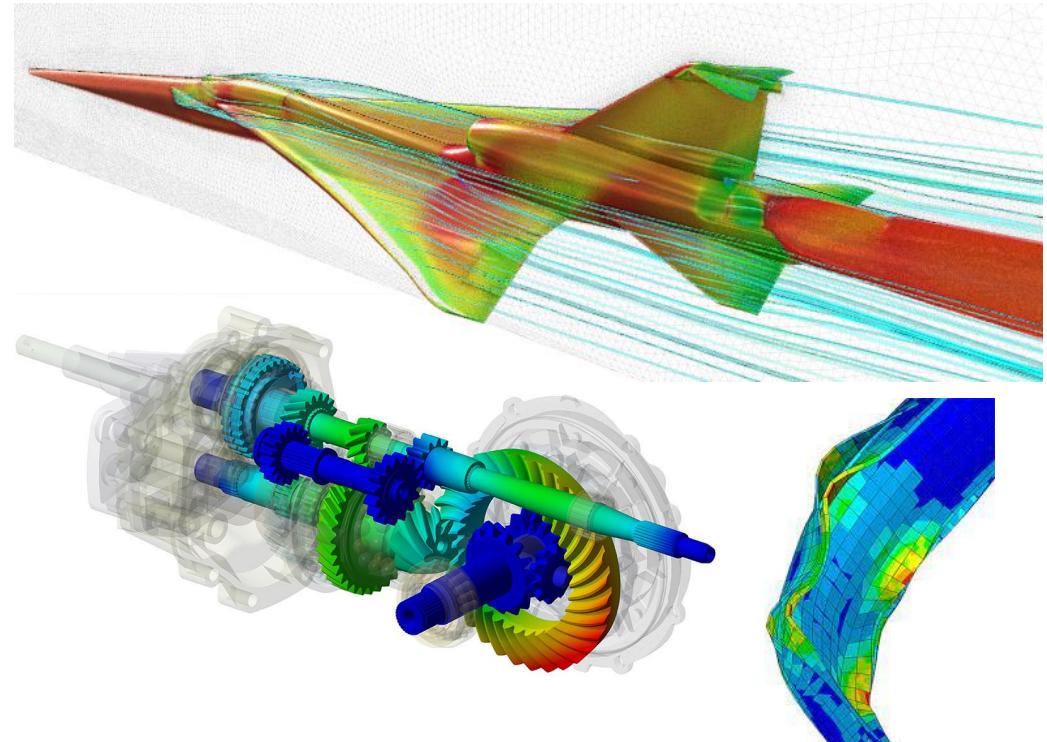
- Industry standard engineering **simulation** software
- Complete package solution
- Global leader in simulation
- Trusted partner of ARES - **LEAP Australia** provide support

leap **australia**
pty ltd

A graphic element consisting of a dark blue wavy line and a yellow wavy line that merges into a single yellow star at the end.

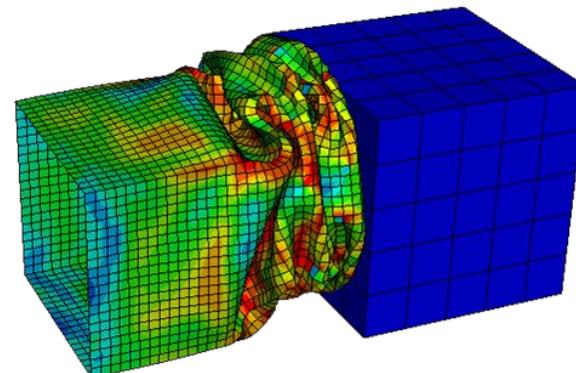
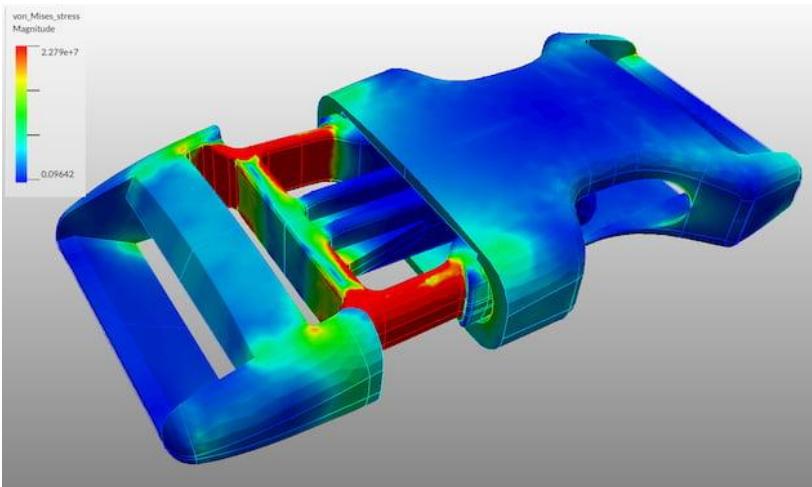


- 3D Design tools (**CAD**)
- Simulation of various physical phenomena, including:
 - Fluid dynamics (**CFD**)
 - Structural mechanics (**FEA**)
 - Thermal
 - Electromagnetic
- Multiphysics simulations
- Model composite materials (carbon fibre)



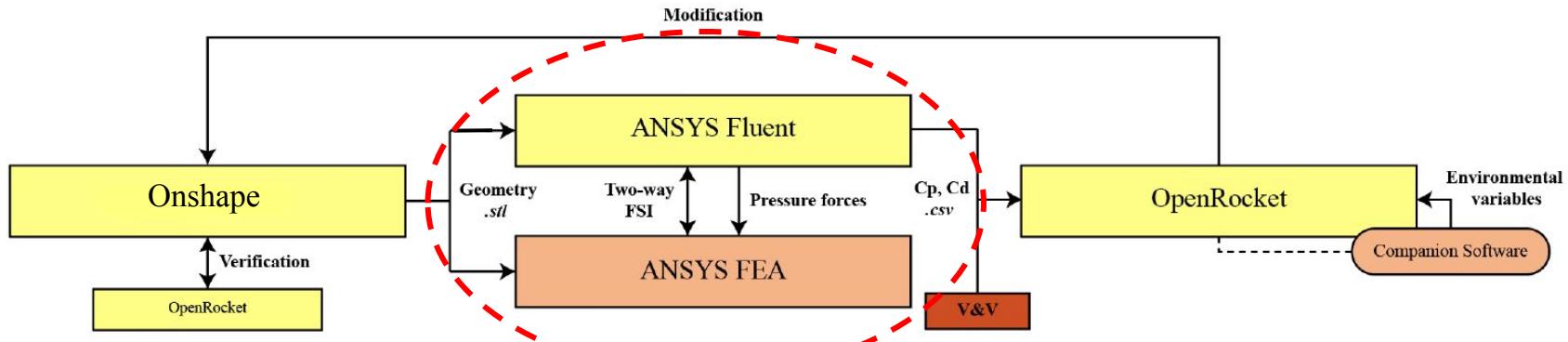
Why FEA?

- Numerically solve solid mechanics problems (**avoid a lot of algebra!**)
- Detailed info about **stress & strain distributions**, and deformation of solid bodies
- Can also find **thermal** distributions



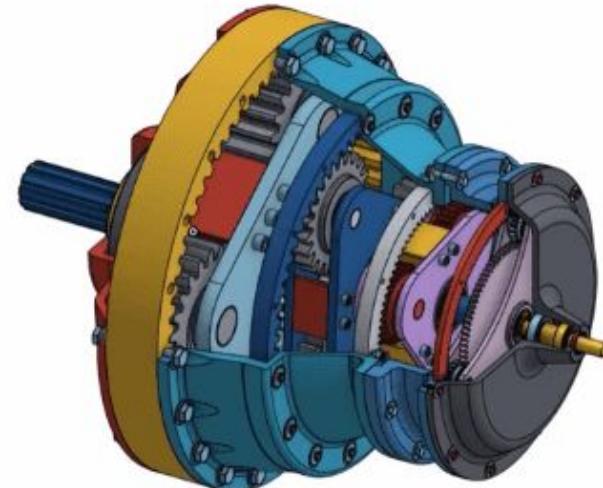
Why FEA?

- Verify structural integrity of components under expected **loading conditions**
- Determine the **likely failure modes and locations**
- Reduces effort and cost by avoiding physical experiments
- Improve accuracy of **CFD** analysis by including deflections from FEA



Software you need

- **Onshape**
 - Online 3D modelling (CAD) software
 - ARES uses this for all of our **3D design projects**



Software you need

- Ansys **SpaceClaim** - CAD software
 - Don't build your model in SpaceClaim!
 - **Prepare the geometry** for CFD
- Ansys **Mechanical** - FEA software
 - Build the **mesh** for FEA
 - Apply loading conditions
 - Solve the FEA



 **Ansys Mechanical**

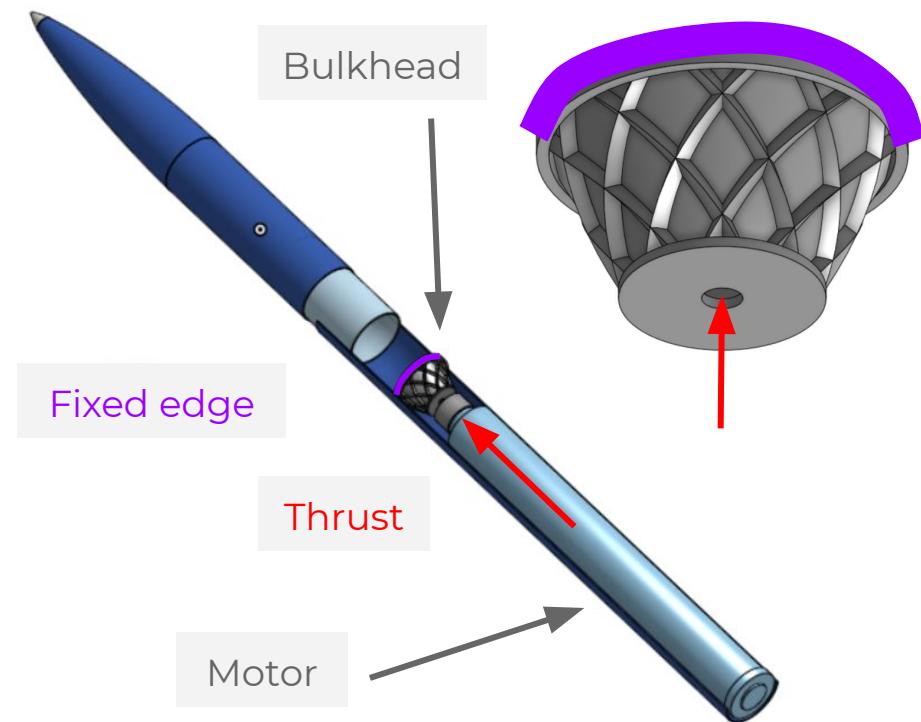
Today's demo

We will be modelling the deformation of a **bulkhead** within a rocket body tube

This bulkhead stops the motor from pushing up through the nose cone

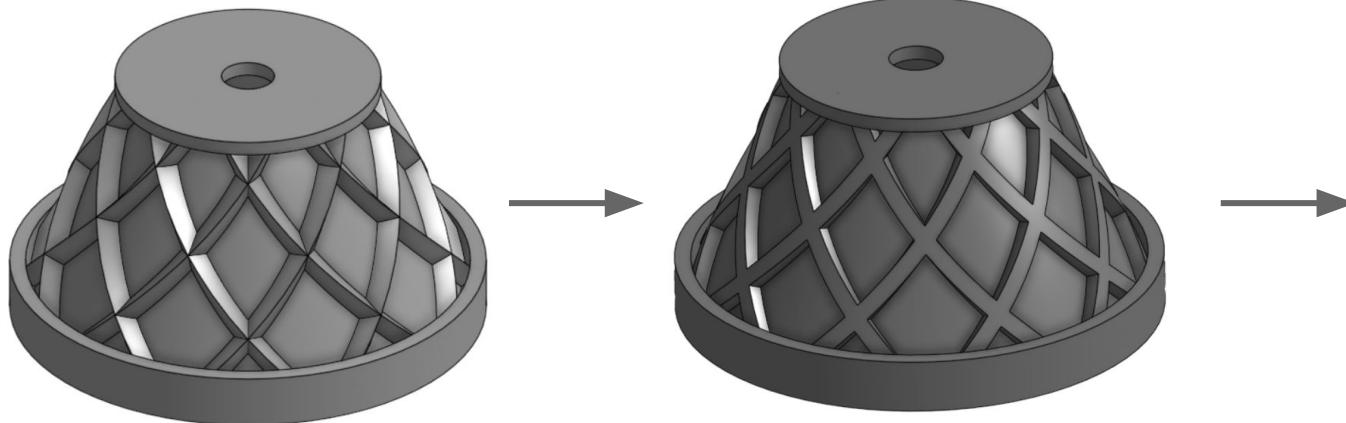
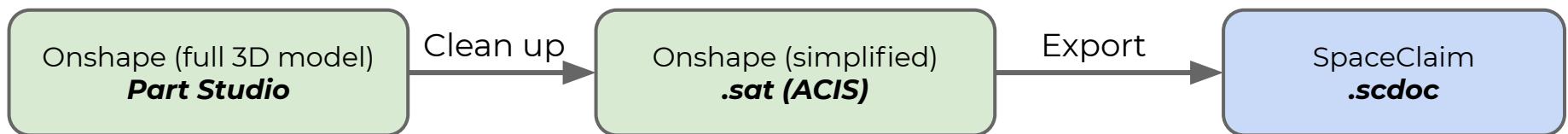
Motor is applying **thrust** to top of bulkhead

Bulkhead is fixed to inside of tube at the **outer edge**

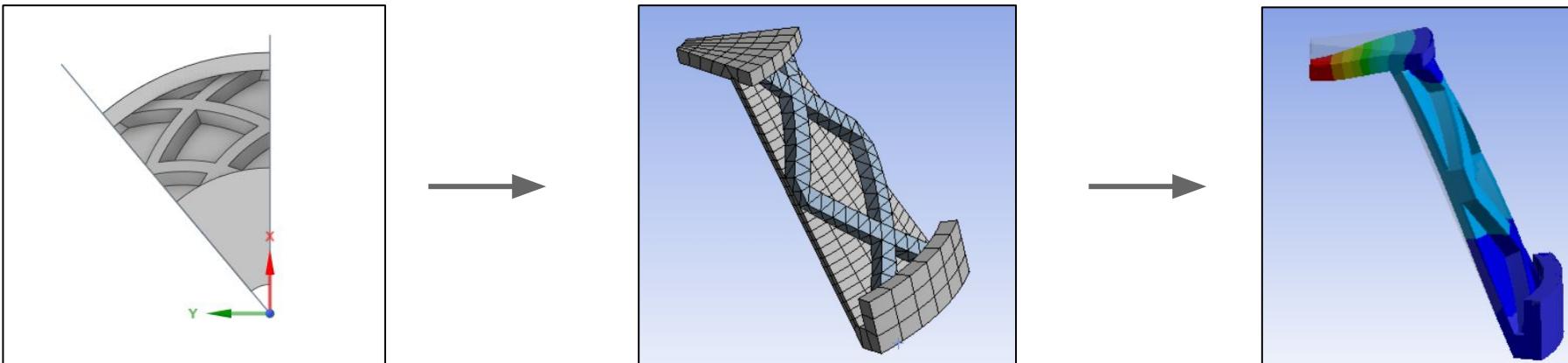




Design Workflow

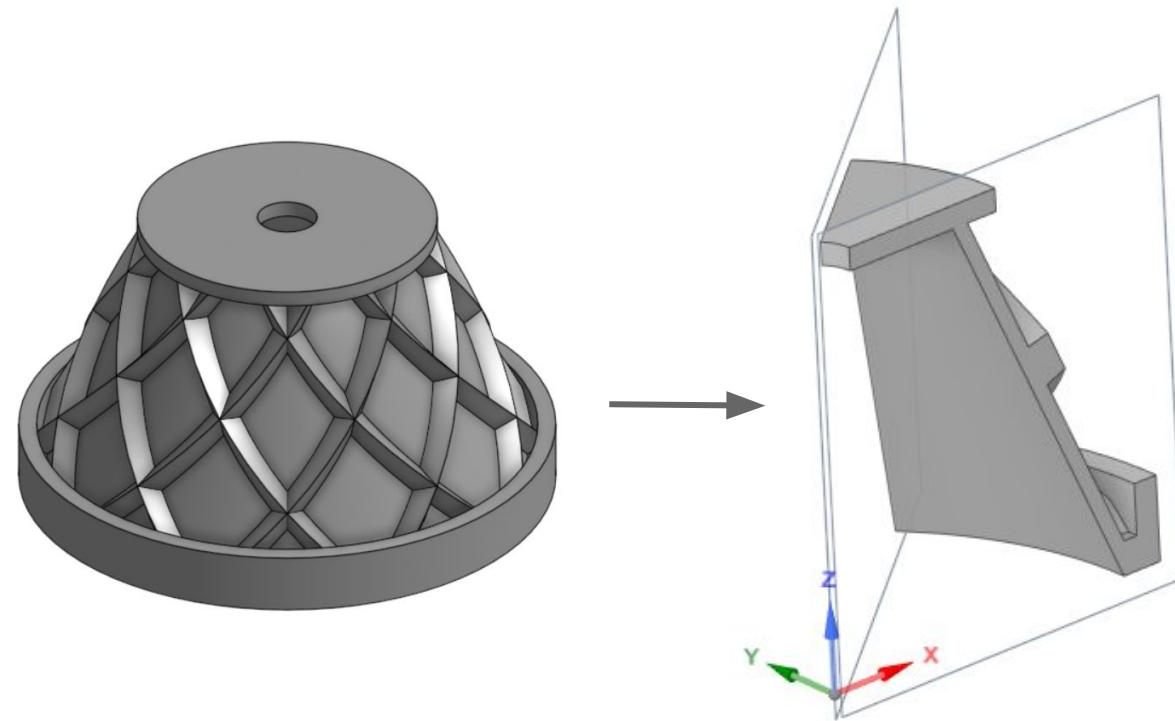


Design Workflow



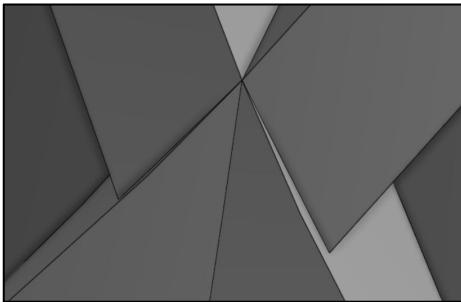
Preparing Geometry

- FEA can't handle complex geometry
- Need to **simplify** the geometry for FEA
- Build a **simplified model in Onshape**
- Exporting might cause more issues, so you need to **double check in Ansys**

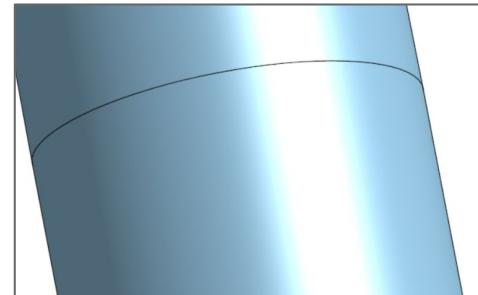
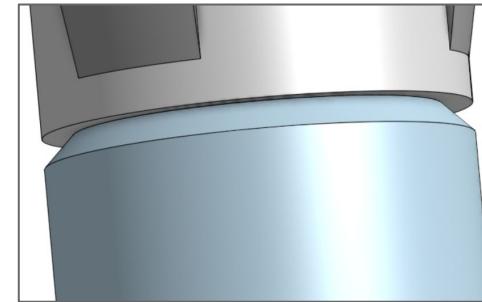


Cleanup - Onshape

Small faces,
manufacturing
features

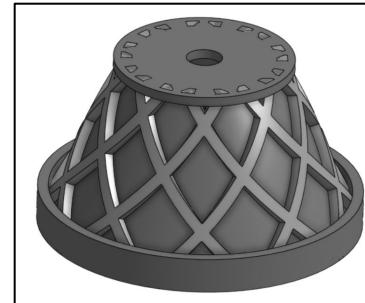


Fillets, chamfers,
rounds

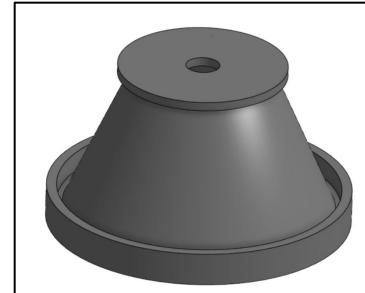


Cleanup - Onshape

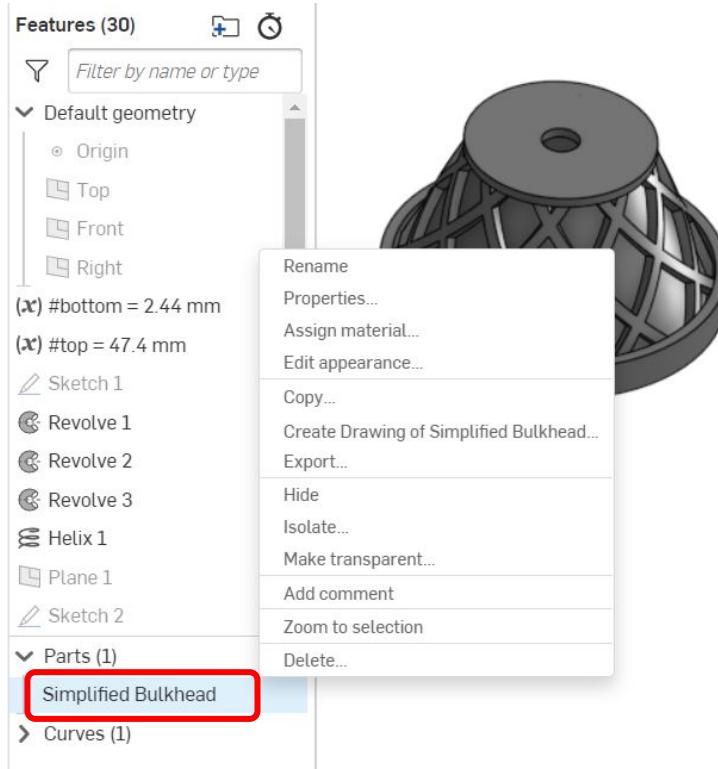
Relevant features?



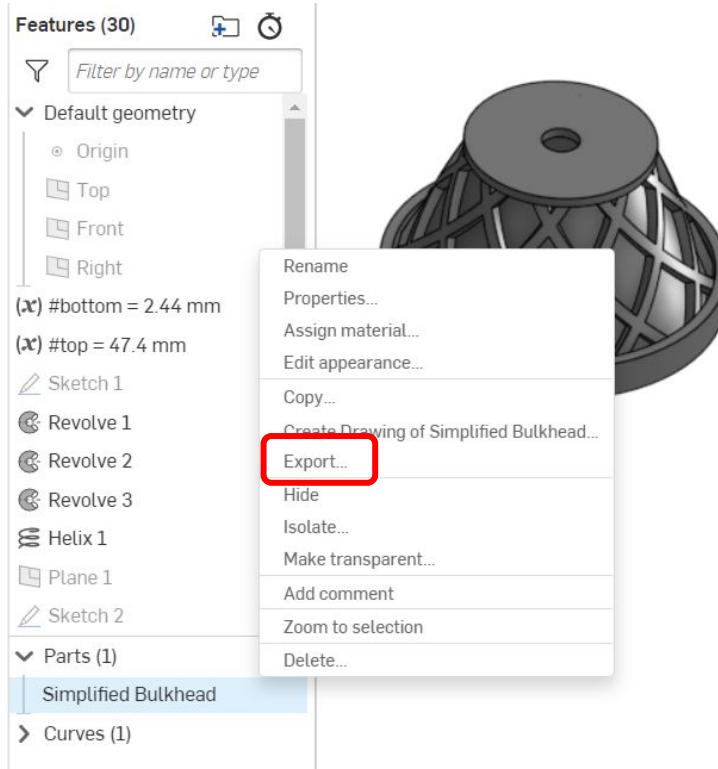
Will this compromise
strength?



Export Onshape



Export Onshape



Export Onshape

The screenshot shows the Onshape interface with a model of a simplified bulkhead. A context menu is open over the model, listing options such as Rename, Properties..., Assign material..., Edit appearance..., Copy..., Create Drawing of Simplified Bulkhead..., Export..., Hide, Isolate..., Make transparent..., Add comment, Zoom to selection, and Delete... The 'Export...' option is highlighted.

Features (30)

- Default geometry
 - Origin
 - Top
 - Front
 - Right
- (x) #bottom = 2.44 mm
- (x) #top = 47.4 mm
- Sketch 1
- Revolve 1
- Revolve 2
- Revolve 3
- Helix 1
- Plane 1
- Sketch 2

Parts (1)

- Simplified Bulkhead

Curves (1)

Export

File name [View export rules](#)

Simplified Model - Simplified Bulkhead

Format

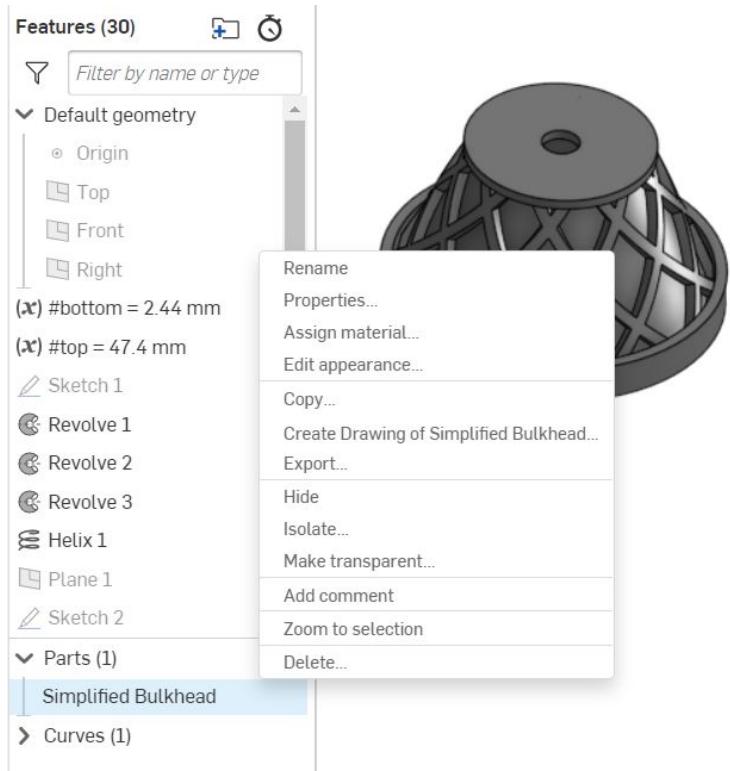
ACIS

Options

Download

OK **Cancel**

Export Onshape



Export

File name [View export rules](#)

Simplified Model - Simplified Bulkhead

Format

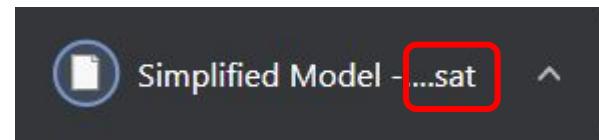
ACIS

Options

Download

OK

Cancel



ARES

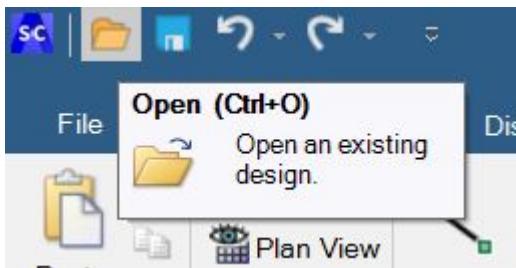
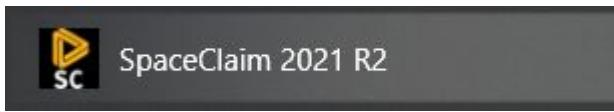
Export Onshape to SpaceClaim



SpaceClaim 2021 R2

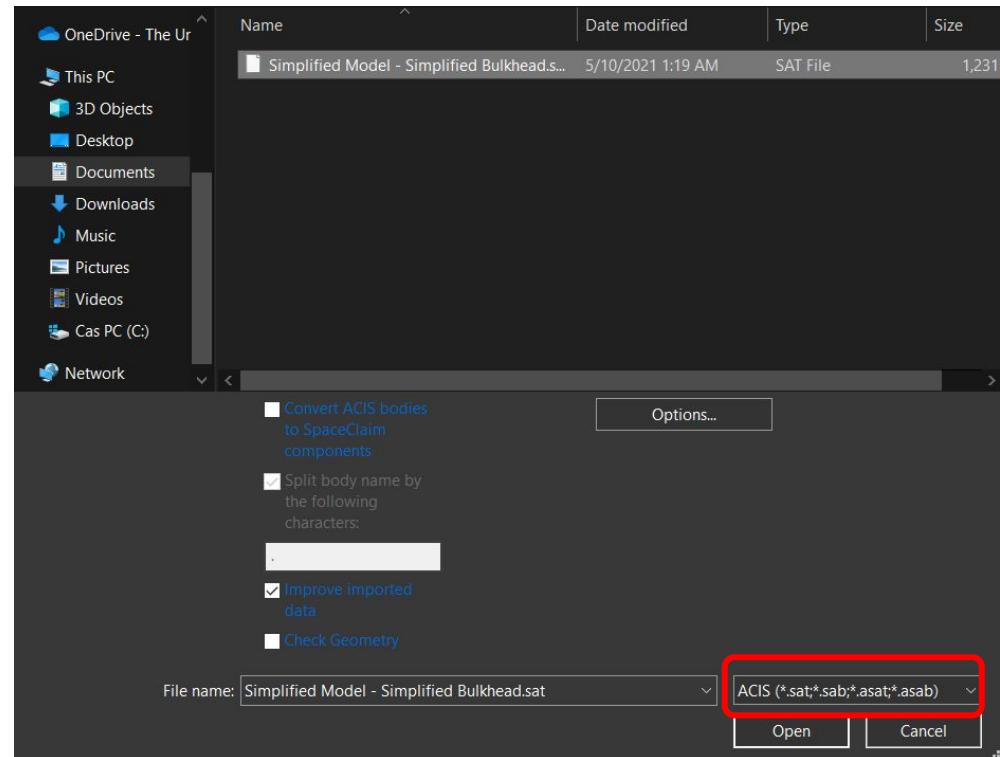
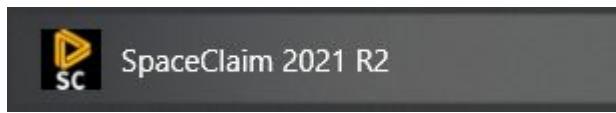


Export Onshape to SpaceClaim



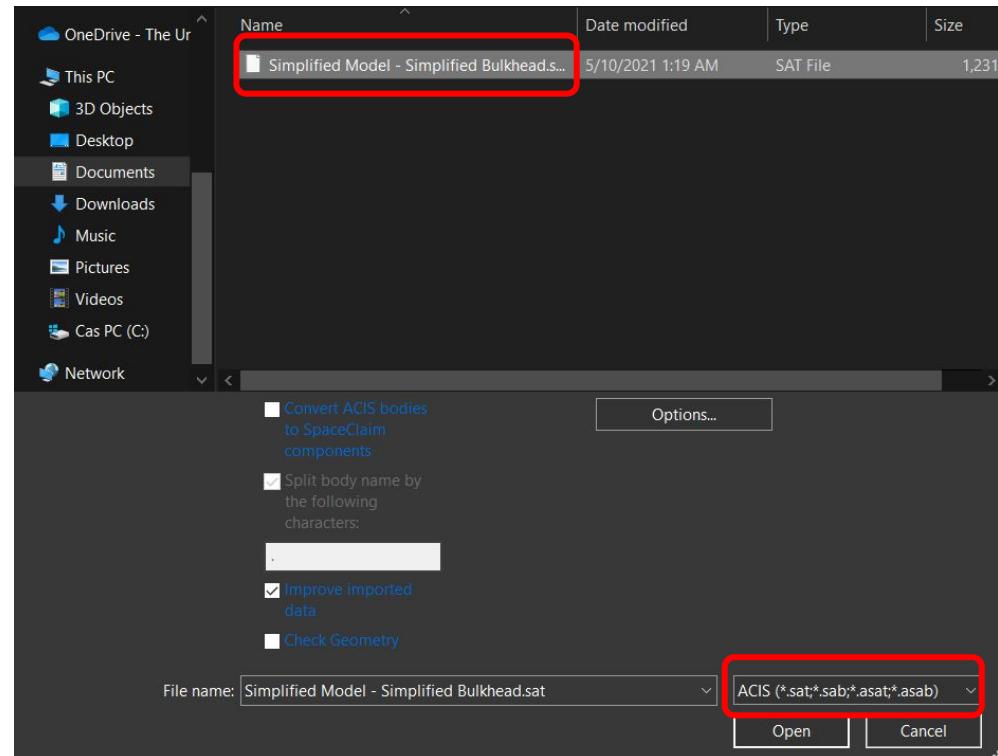
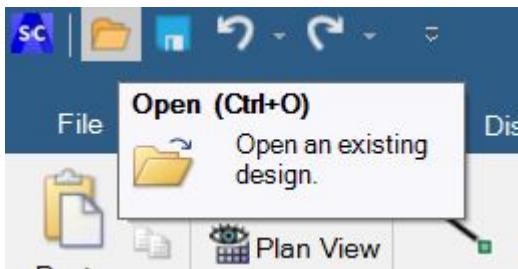


Export Onshape to SpaceClaim



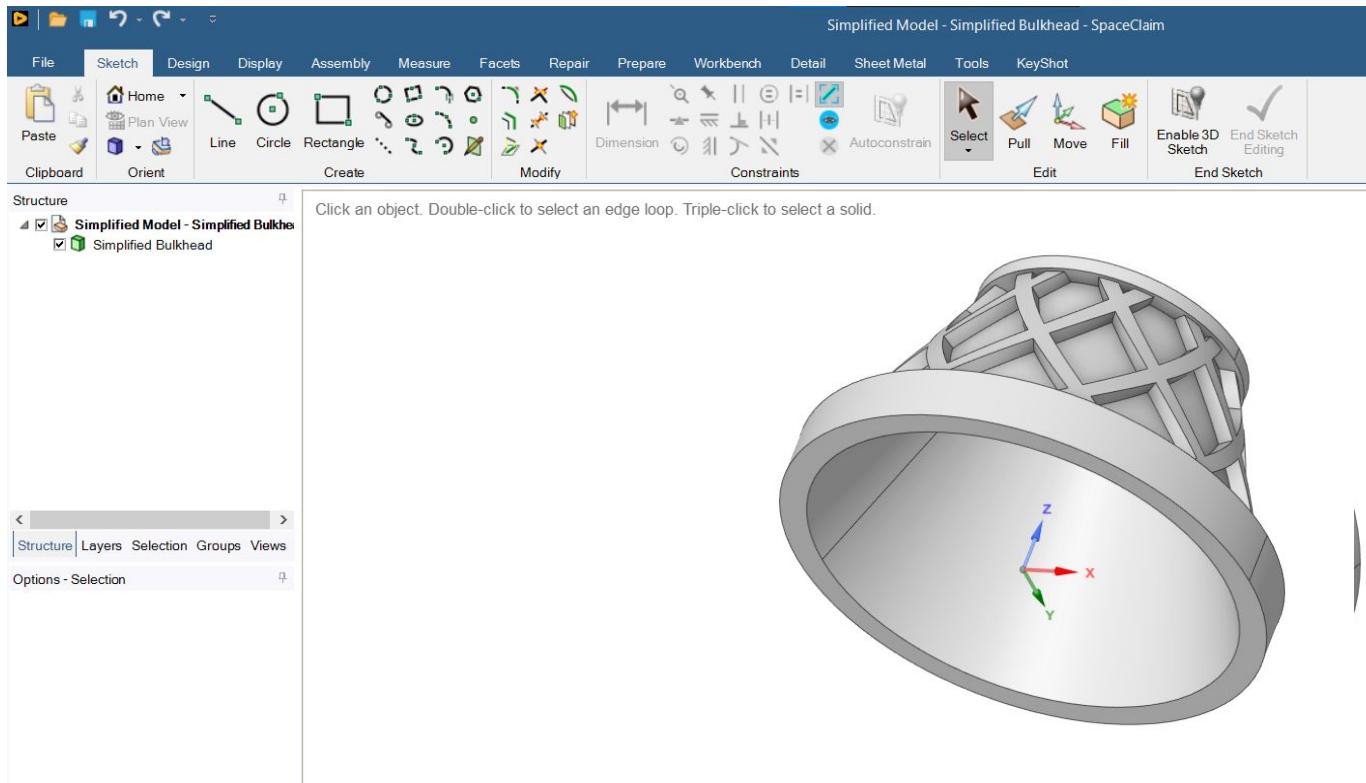


Export Onshape to SpaceClaim



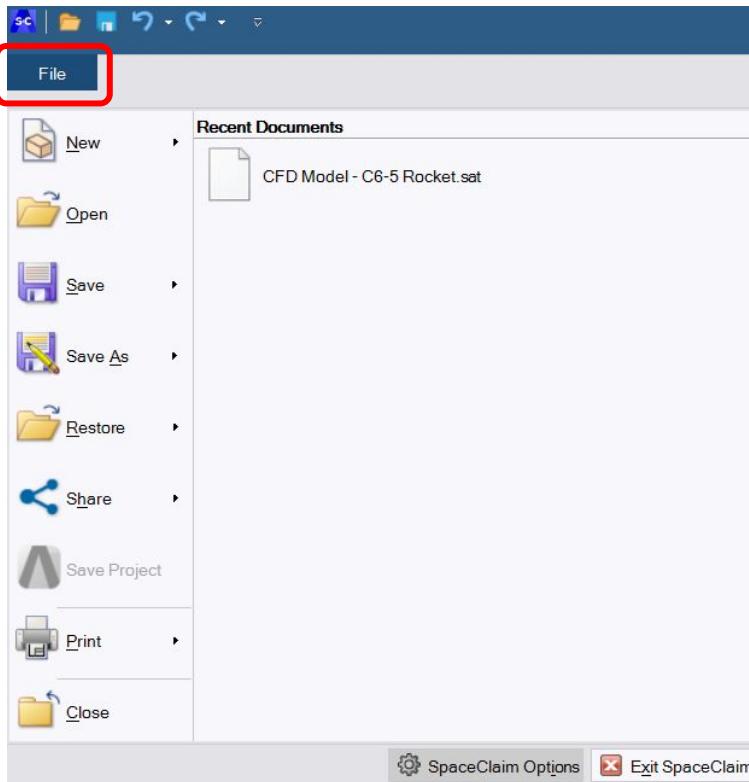


Export Onshape to SpaceClaim



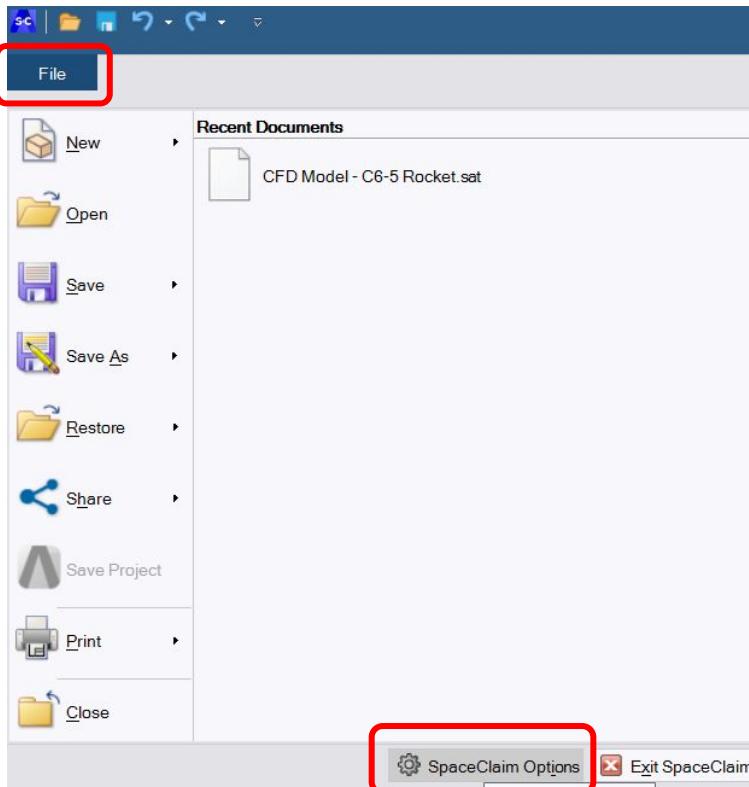


SpaceClaim Mouse Controls



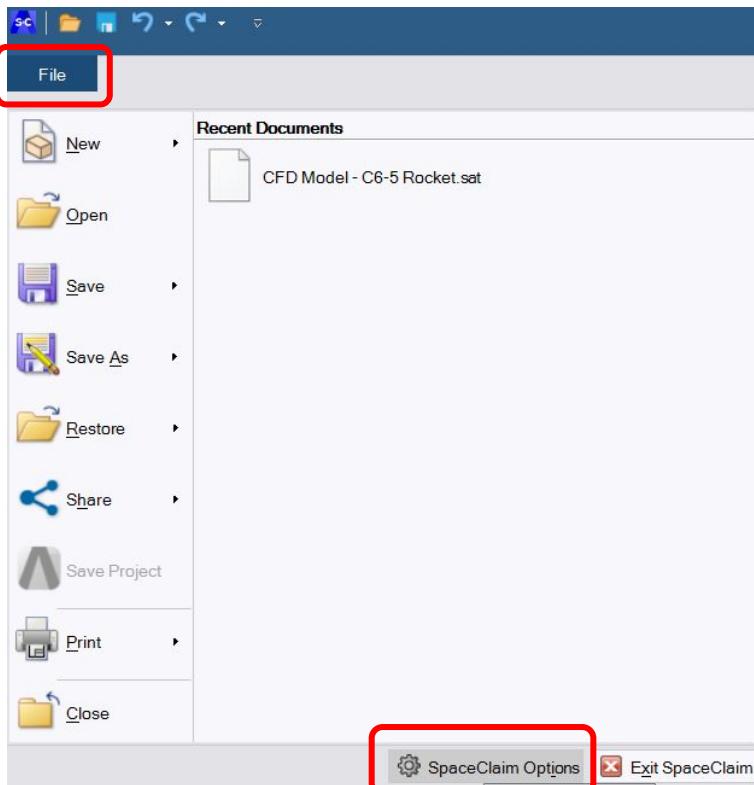


SpaceClaim Mouse Controls





SpaceClaim Mouse Controls



SpaceClaim Options

Change navigation style used in SpaceClaim.

Navigation

Theme

Theme	Custom
Spin	Default
Pan	Alternative 1
Zoom	Alternative 2
Zoom-in drag direction	Alternative 3
Zoom-in wheel direction	Alternative 4
	Custom

Zoom

Zoom with mouse wheel:

In Zoom mode
 In Spin/Pan/Zoom mode
 Always (hold Ctrl to Query Select)

Speed: Slow — — Fast

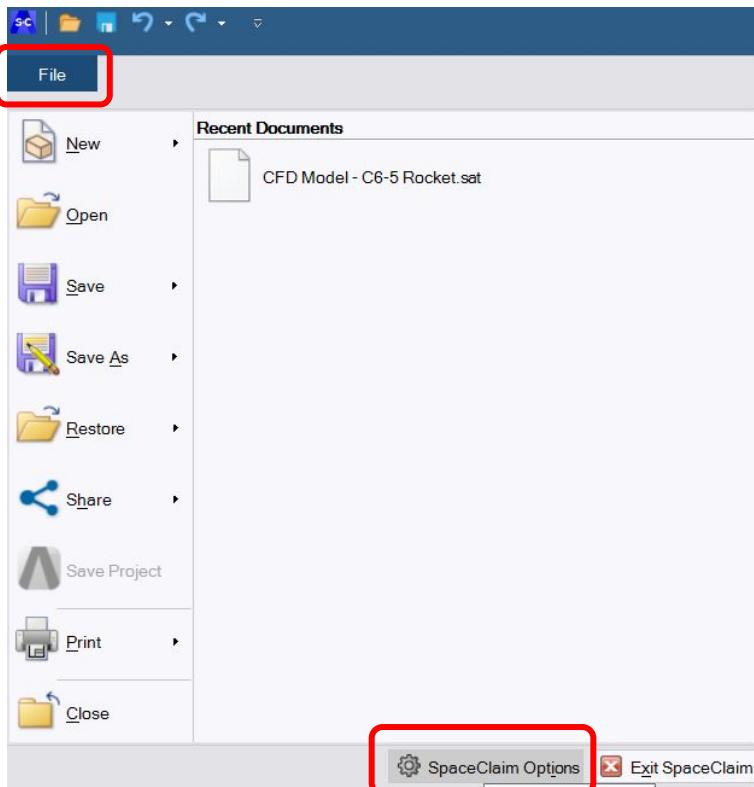
Reset All

SpaceClaim Options

Exit SpaceClaim



SpaceClaim Mouse Controls



SpaceClaim Options

Change navigation style used in SpaceClaim.

Navigation

Theme

Custom
Default
Alternative 1
Alternative 2
Alternative 3
Alternative 4
Custom

The view navigation settings to use for spin, pan, and zoom.
MMB = Middle Mouse Button, RMB = Right Mouse Button, LMB = Left Mouse Button

Zoom

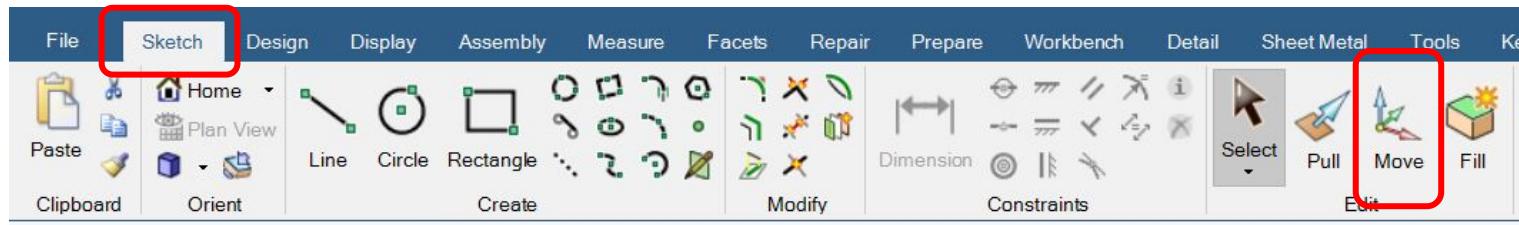
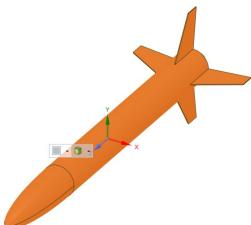
Zoom with mouse wheel:

In Zoom mode
 In Spin/Pan/Zoom mode
 Always (hold Ctrl to Query Select)

Speed: Slow — [Slider] — Fast

This dialog box allows users to change the navigation style in SpaceClaim. It features a sidebar with categories like Popular, Detailing, Appearance, Selection, Snap, Units, Sheet Metal, Navigation, Advanced, File Options, Support Files, Customize, Add-Ins, License, and Resources. The "Navigation" category is selected and highlighted with a red box. On the right, the "Theme" section is expanded, showing a list of options: Custom, Default, Alternative 1, Alternative 2, Alternative 3, Alternative 4, and Custom again. The "Custom" option is highlighted with a red box. Below the theme list, there are sections for "Zoom with mouse wheel" and "Speed" control.

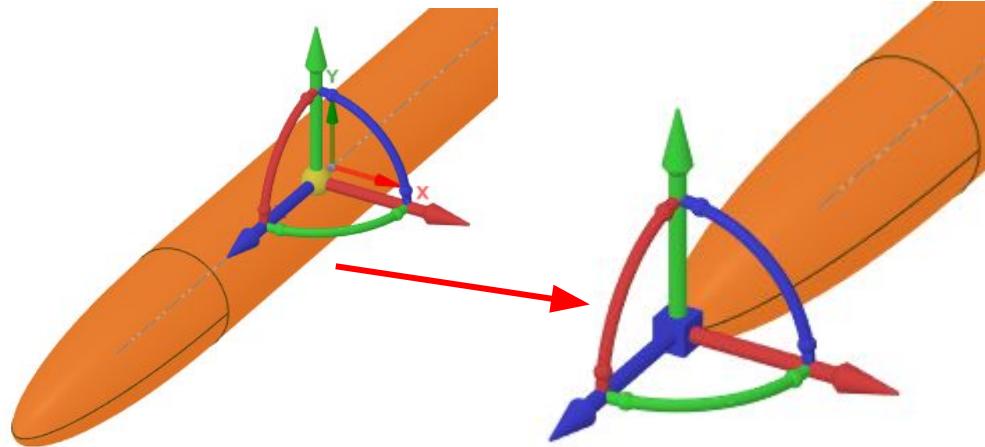
Fix Origin and Orientation



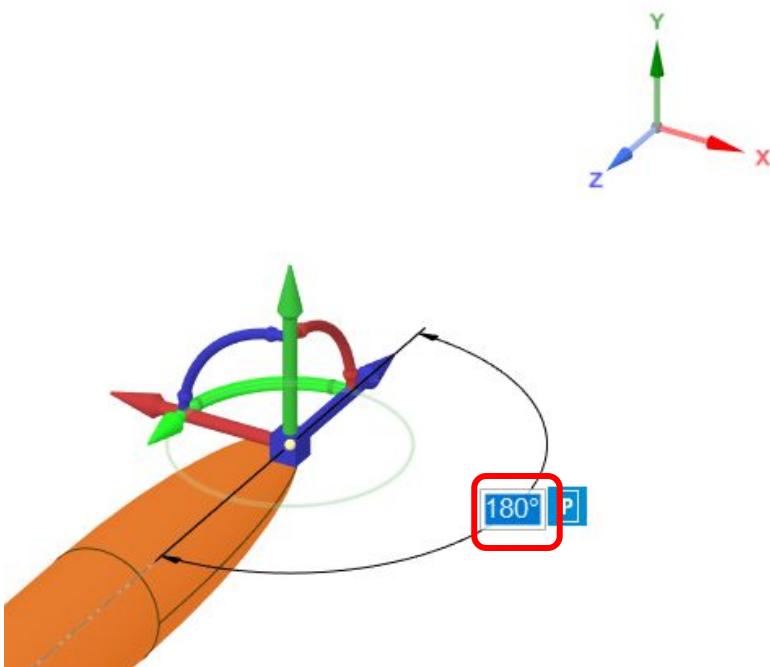
Want gravity (**z**) pointing down
the rocket

Want the origin at the **tip of the nose** cone to match OpenRocket

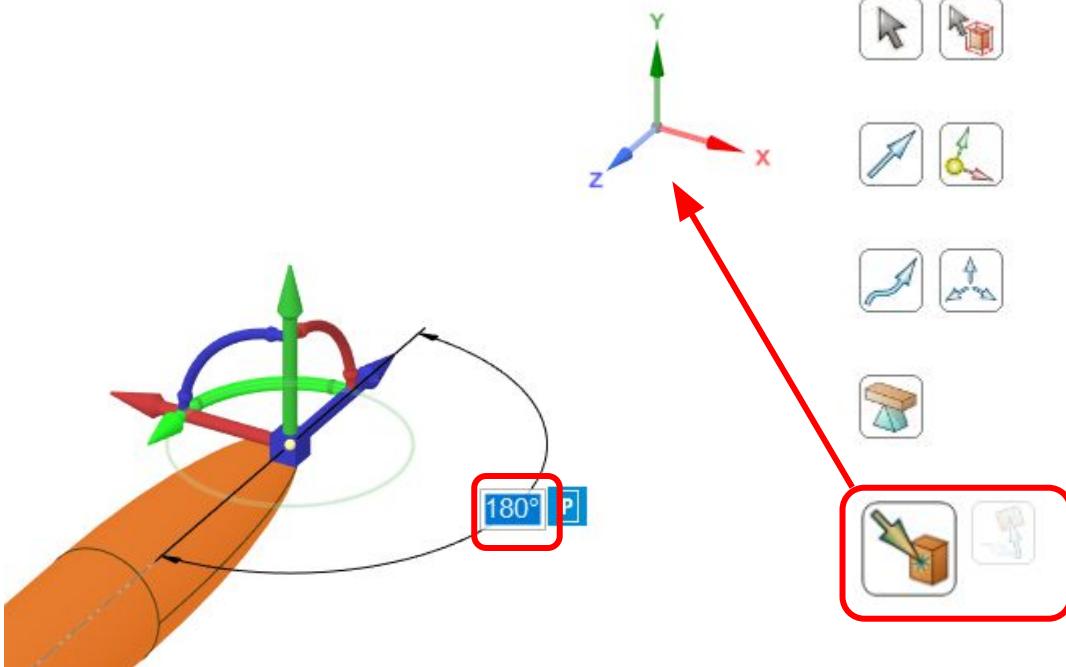
Triple click to select the entire
rocket



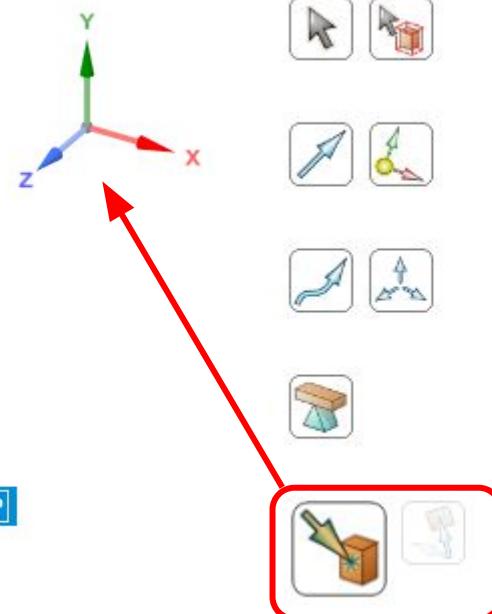
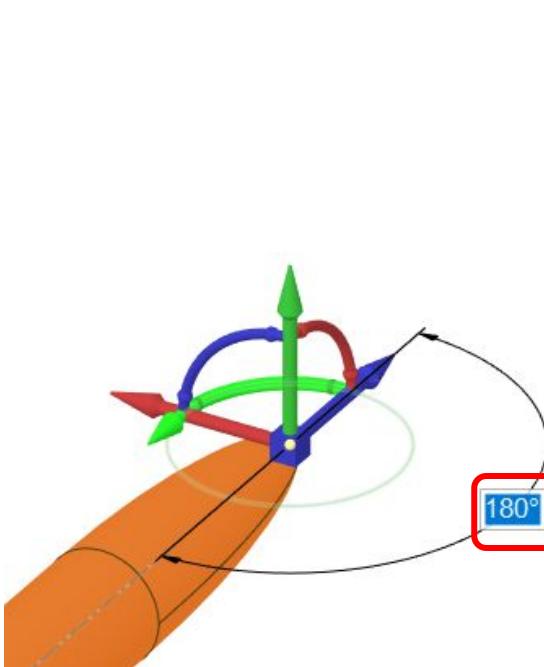
Fix Origin and Orientation



Fix Origin and Orientation

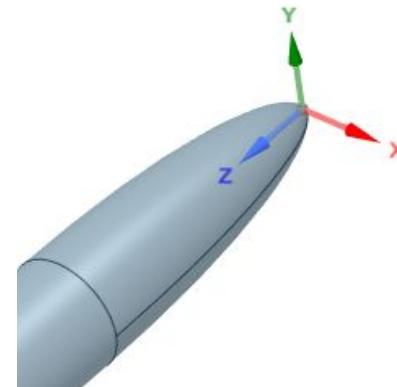


Fix Origin and Orientation



Click a reference to translate up to

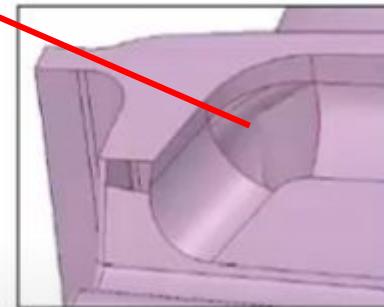
Press **esc** to leave the Move Tool



Cleanup - SpaceClaim



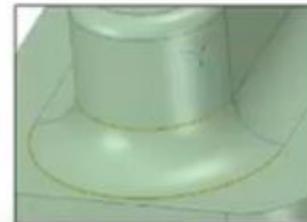
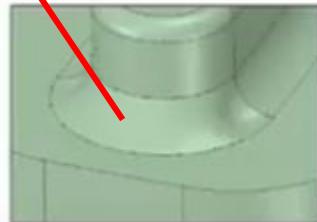
Missing faces



Cleanup - SpaceClaim



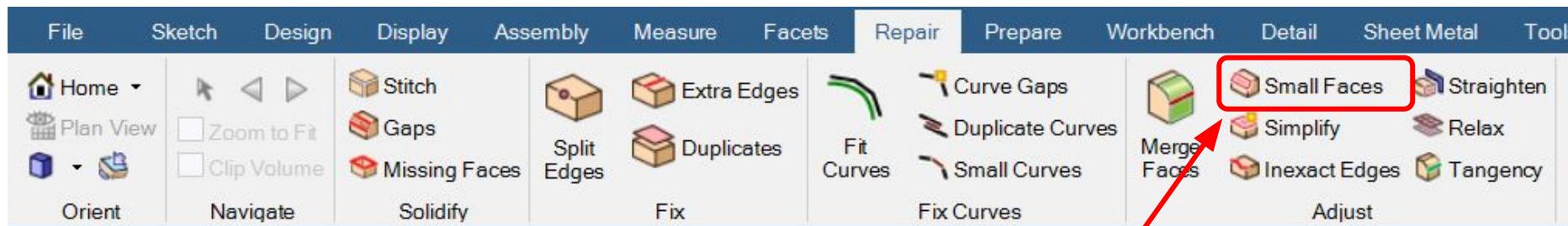
Split edges



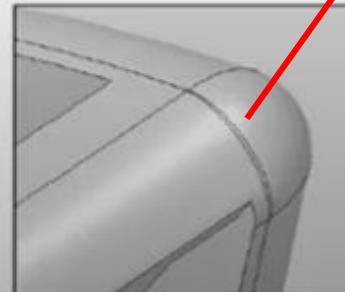
Cleanup - SpaceClaim



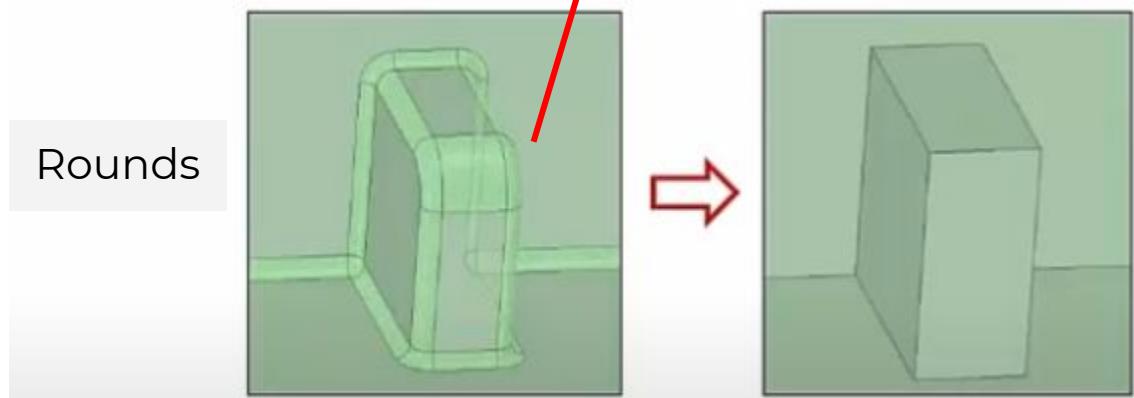
Cleanup - SpaceClaim



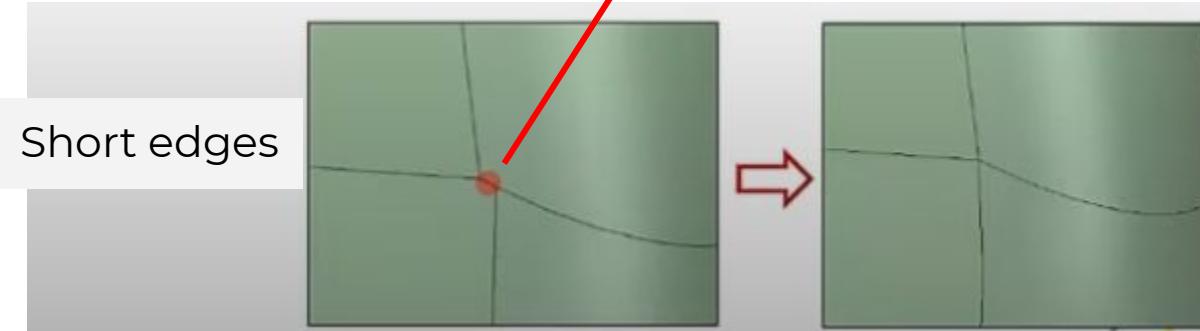
Small faces



Cleanup - SpaceClaim

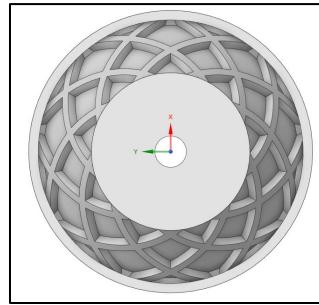


Cleanup - SpaceClaim



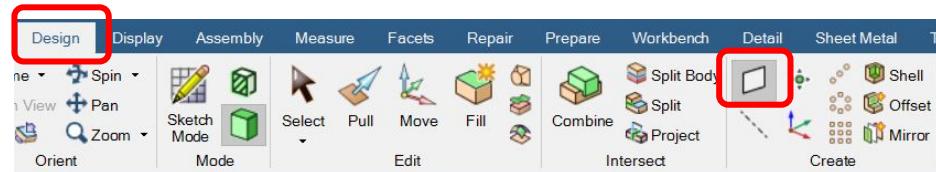
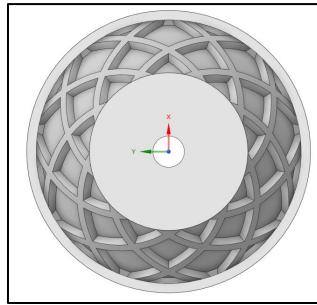
Cleanup - SpaceClaim

Symmetric object
+ symmetric
loading
condition?



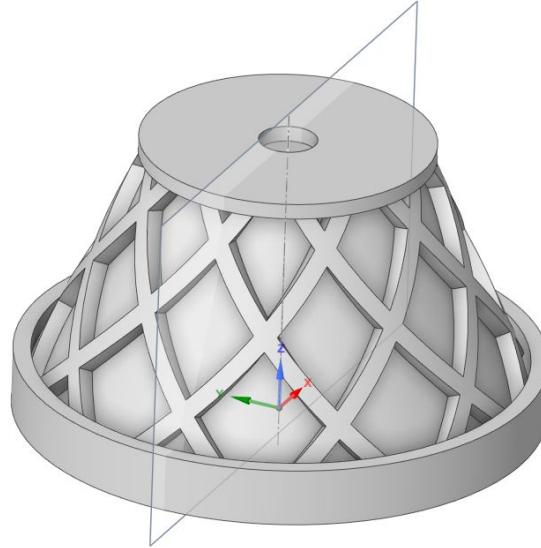
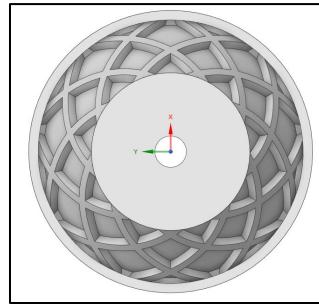
Cleanup - SpaceClaim

Symmetric object
+ symmetric
loading
condition?



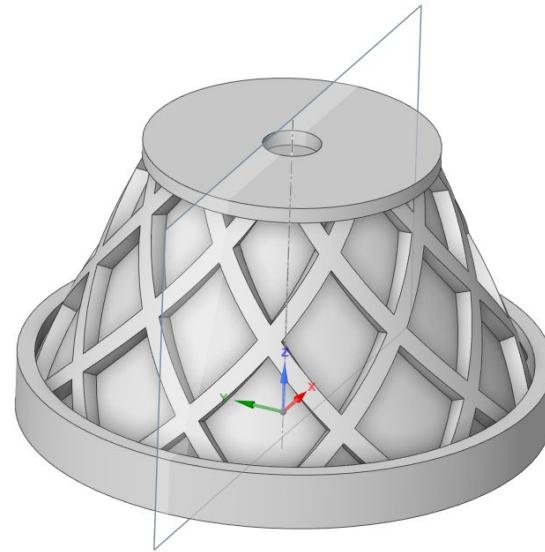
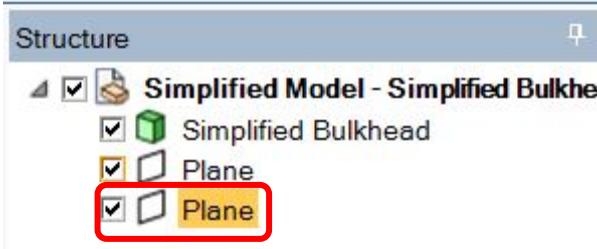
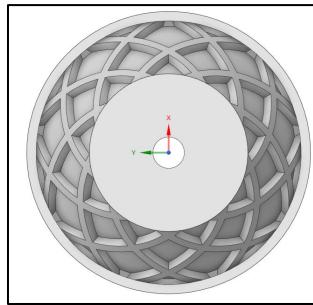
Cleanup - SpaceClaim

Symmetric object
+ symmetric
loading
condition?



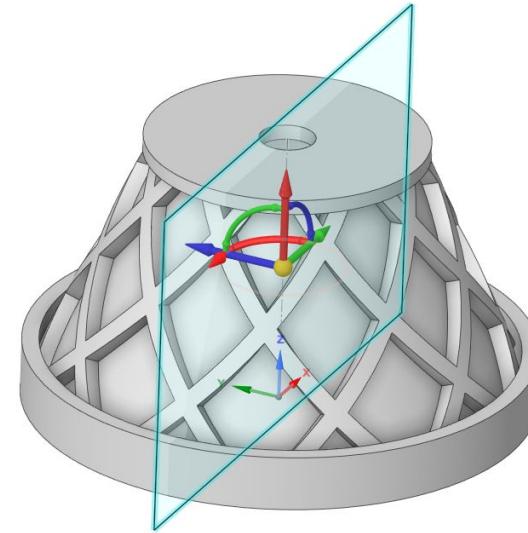
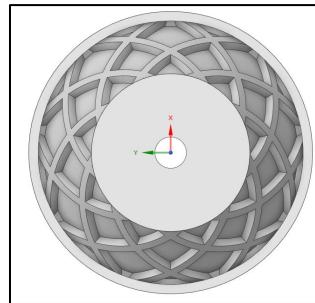
Cleanup - SpaceClaim

Symmetric object
+ symmetric
loading
condition?



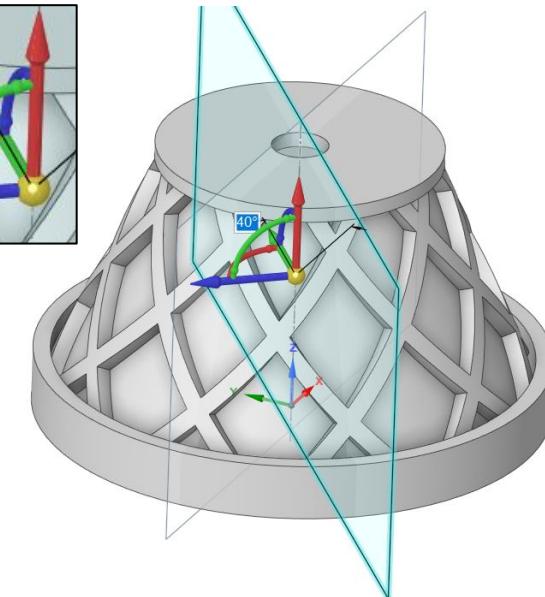
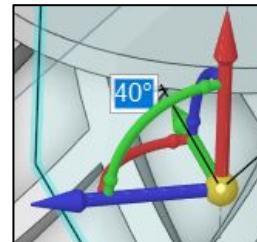
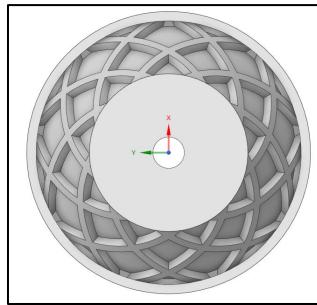
Cleanup - SpaceClaim

Symmetric object
+ symmetric
loading
condition?



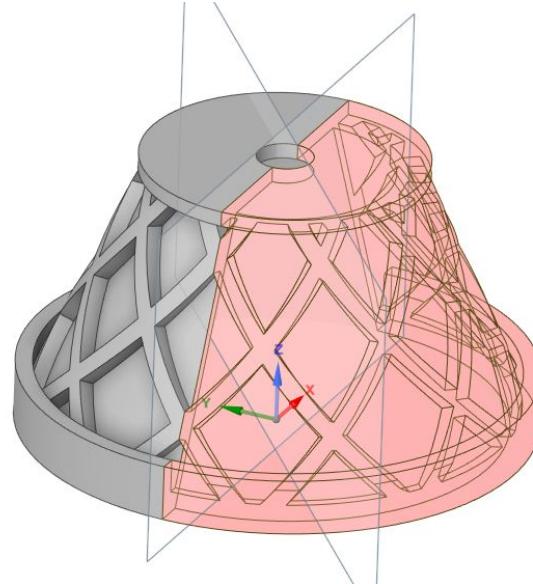
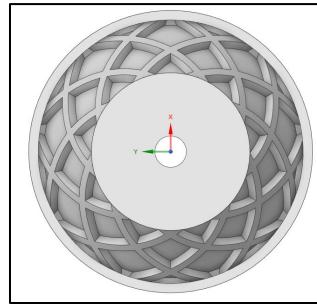
Cleanup - SpaceClaim

Symmetric object
+ symmetric
loading
condition?



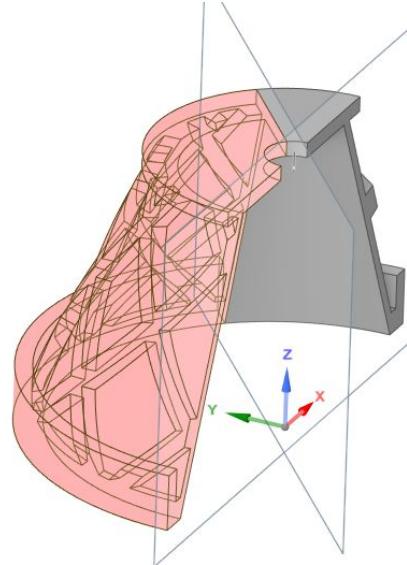
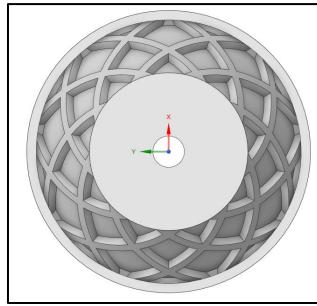
Cleanup - SpaceClaim

Symmetric object
+ symmetric
loading
condition?



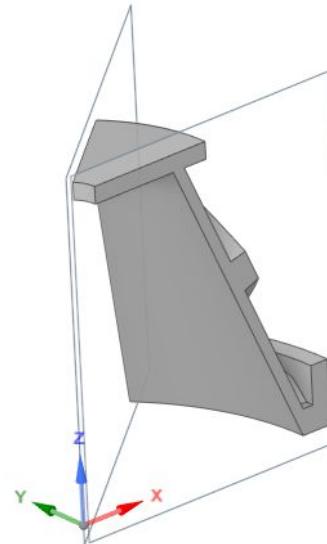
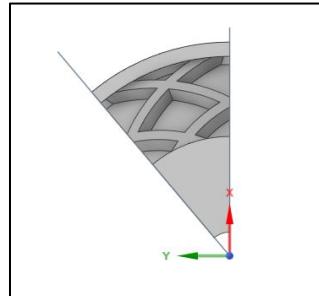
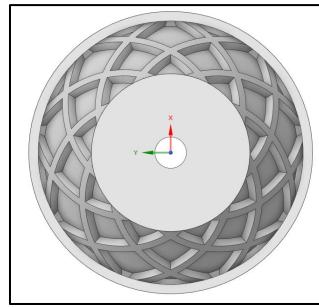
Cleanup - SpaceClaim

Symmetric object
+ symmetric
loading
condition?



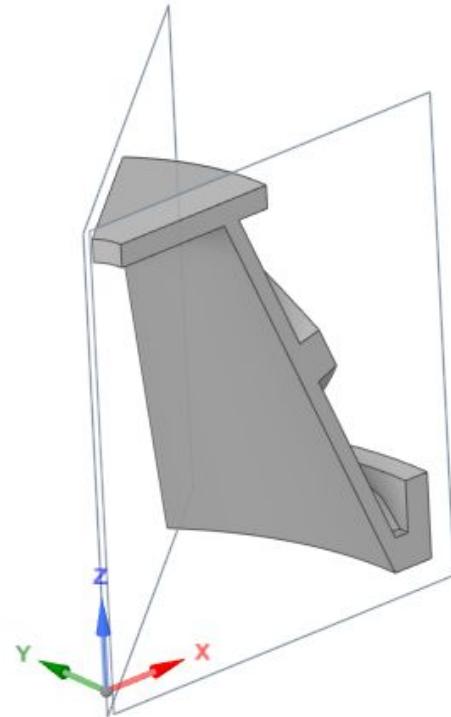
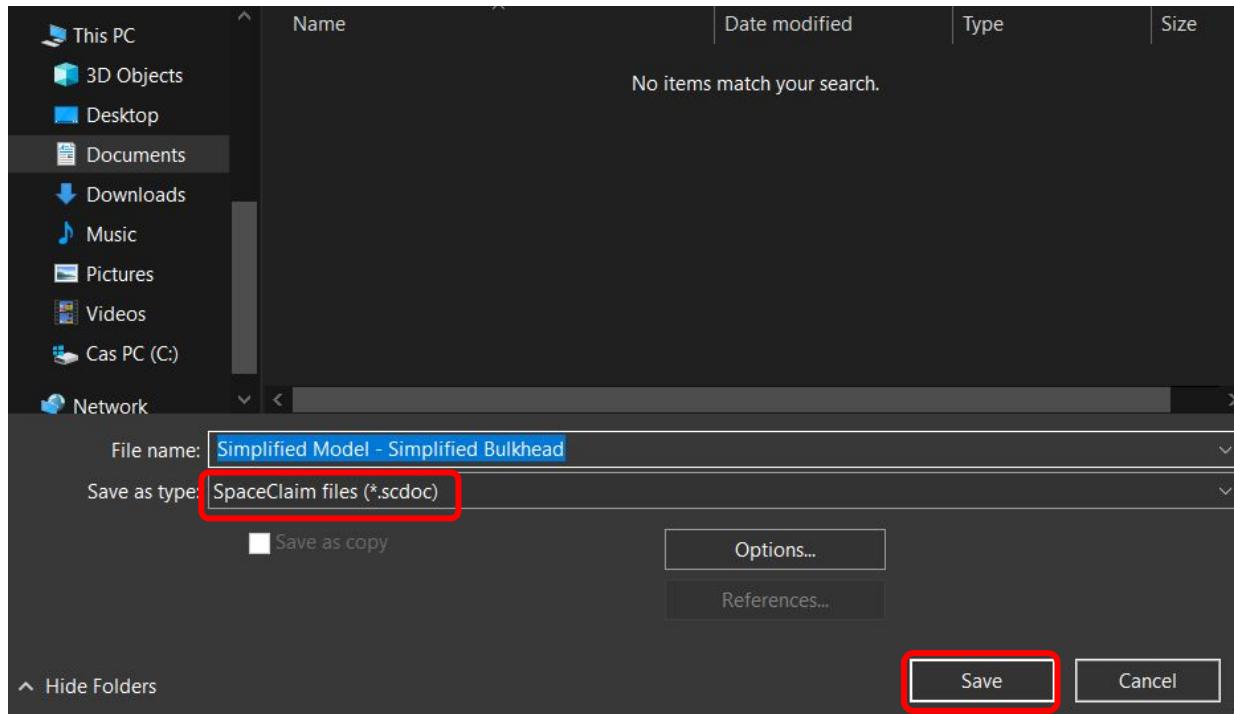
Cleanup - SpaceClaim

Symmetric object
+ symmetric
loading
condition?





Export to Mechanical





Export to Mechanical

The screenshot shows the Mechanical 2021 R2 software interface. At the top is a dark header bar with the ARES logo and the text "Mechanical 2021 R2". Below this is a ribbon menu with tabs: File, Home, Model, Display, Selection, and Automation. The "File" tab is currently selected. On the left is a vertical toolbar with options: Info, What's New?, Save Project, Save Project As..., Archive Project..., Save Database, Refresh All Data, Clear Generated Data, and Import... (which is highlighted with a red rectangle). The main workspace is titled "Import" and contains three items:

- Geometry**: Represented by a cube icon. Description: Open a geometry by selecting one of the following options.
- Mesh (External Model)**: Represented by a cube icon.
- Recent geometry file**: Represented by a circular arrow icon.
- Browse to a geometry file**: Represented by a folder icon.

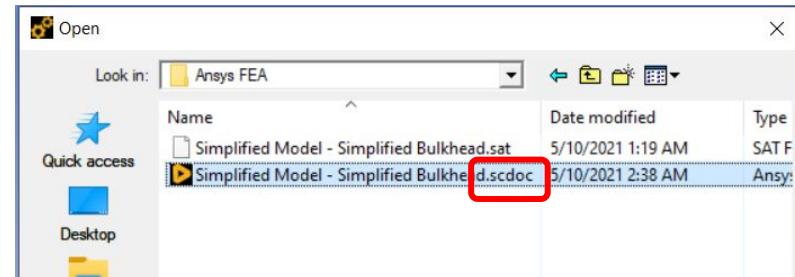


Export to Mechanical

The screenshot shows the Mechanical 2021 R2 software interface. The title bar reads "Mechanical 2021 R2". The ribbon menu includes "File", "Home", "Model", "Display", "Selection", and "Automation". The "File" tab is selected. The main area displays the "Import" screen with the following options:

- Geometry**: Represented by a cube icon.
- Mesh (External Model)**: Represented by a cube icon.
- Recent geometry file**: Represented by a circular arrow icon.
- Browse to a geometry file**: Represented by a folder icon.

On the left sidebar, under the "File" category, the "Import..." button is highlighted with a red box.





Export to Mechanical

The screenshot shows the Mechanical 2021 R2 software interface. The title bar says "Mechanical 2021 R2". The ribbon menu includes "File", "Home", "Model", "Display", "Selection", and "Automation". The "File" tab is selected, showing options like "Info", "What's New?", "Save Project", "Save Project As...", "Archive Project...", "Save Database", "Refresh All Data", "Clear Generated Data", and "Import...". The "Import..." button is highlighted with a red box. The main workspace is titled "Import" and contains sections for "Geometry" and "Mesh (External Model)". Below these are "Recent geometry file" and "Browse to a geometry file". To the right, there is descriptive text about opening geometry.

Import

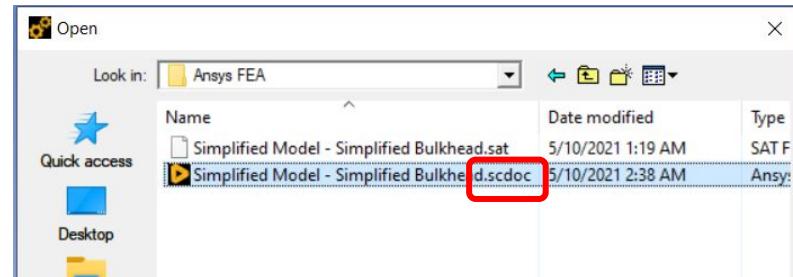
Open a geometry by selecting one of the following options:
After selecting the geometry, the geometry will be imported into the current default options in the Workbench project. You can then edit them there, and then update the geometry by saving it again before opening the geometry.

Geometry

Mesh (External Model)

Recent geometry file

Browse to a geometry file



The screenshot shows the Mechanical ribbon. The "Units" tab is selected, indicated by a blue background. Other tabs include "Worksheet", "Keyframe Animation", "Tags", "Wizard", and "Show Errors". Under the "Unit Systems" section, three options are listed: "Metric (m, kg, N, s, V, A)", "Metric (cm, g, dyne, s, V, A)", and "Metric (mm, kg, N, s, mV, mA)". The first option is checked with a blue checkmark.

Units

Worksheet Keyframe Animation Tags Wizard Show Errors

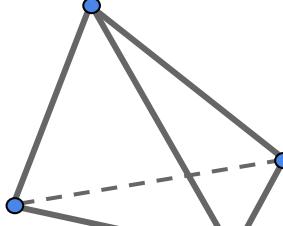
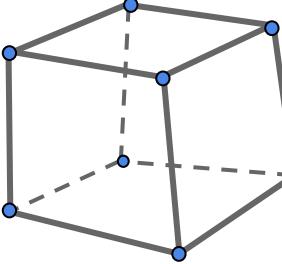
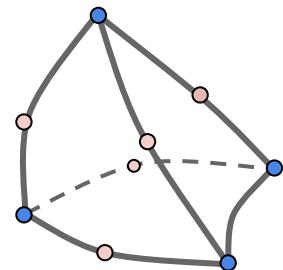
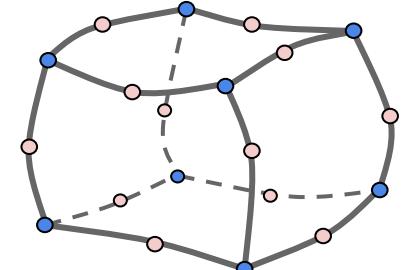
Unit Systems

Metric (m, kg, N, s, V, A)

Metric (cm, g, dyne, s, V, A)

Metric (mm, kg, N, s, mV, mA)

Element Types

	Tetrahedral (tet)	Hexahedral (hex)
Linear	 4 faces 4 nodes	 6 faces 8 nodes
Quadratic	 4 faces 8 nodes	 6 faces 20 nodes

Linear vs Quadratic Mesh

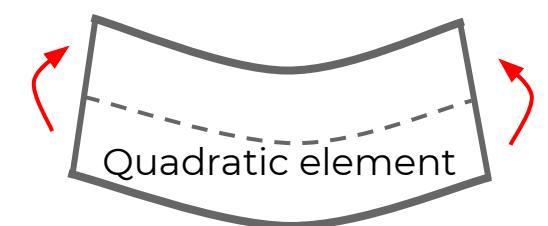
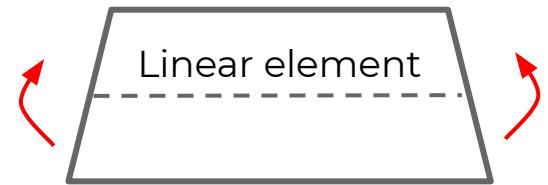
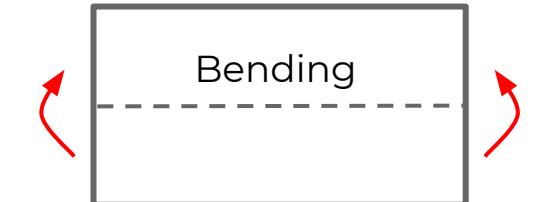
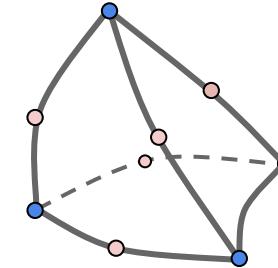
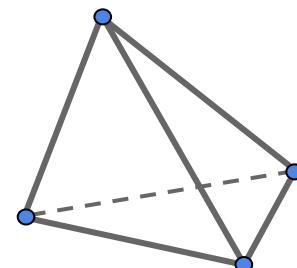
Linear elements suffer from **shear locking** -
unable to represent pure bending

Linear will have **larger error**, especially for
curved geometry and bending

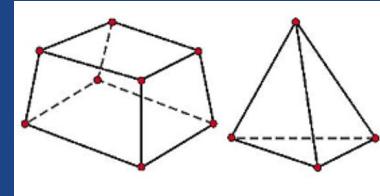
Quadratic may **converge more quickly** as
mesh size reduces

Quadratic more **computationally expensive**
than linear

Quadratic **may be overkill** - if the variable in
question is linear, use linear elements!



Hex vs. Tet Mesh



Hex can be **structured** (uniform), tet is unstructured.

Hex is generally **more accurate** than tet for the same cell count.

For complex geometry, hex mesh is **more difficult** to achieve than tet.

Hex mesh can be **quicker** to make.

Harder to get high quality hex mesh than tet.

Can also make hybrid meshes with both hex and tet.

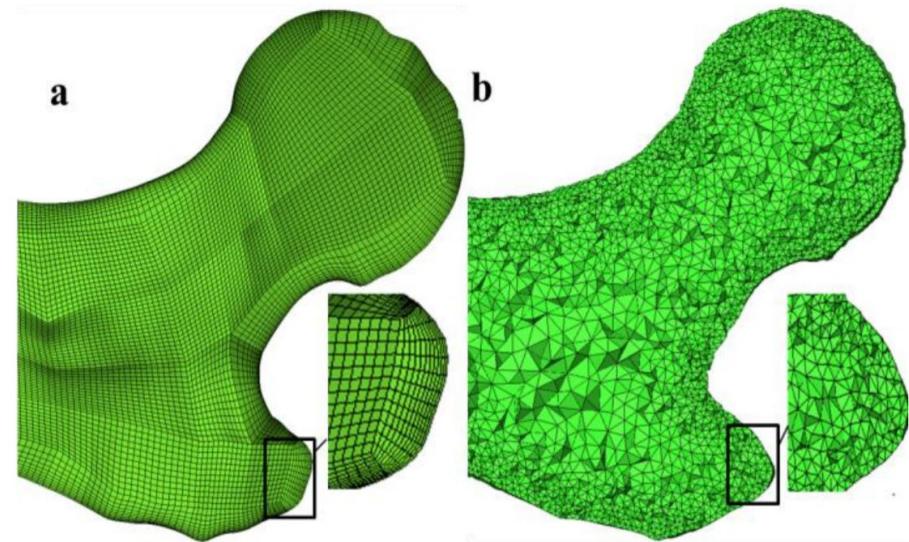


Figure 1: a) Hex, and b) Tet meshes for a femur.

Image source: Mathur, V. and Dragomir-Daescu, D. (2012). Hexahedral vs. Tetrahedral Finite Element Models of the Proximal Femur.

Hex vs. Tet Mesh

Hex can be **structured** (uniform), tet is unstructured.

Hex is generally **more accurate** than tet for the same cell count.

For complex geometry, hex mesh is **more difficult** to achieve than tet.

Hex mesh can be **quicker** to make.

Can also make hybrid meshes with both hex and tet.

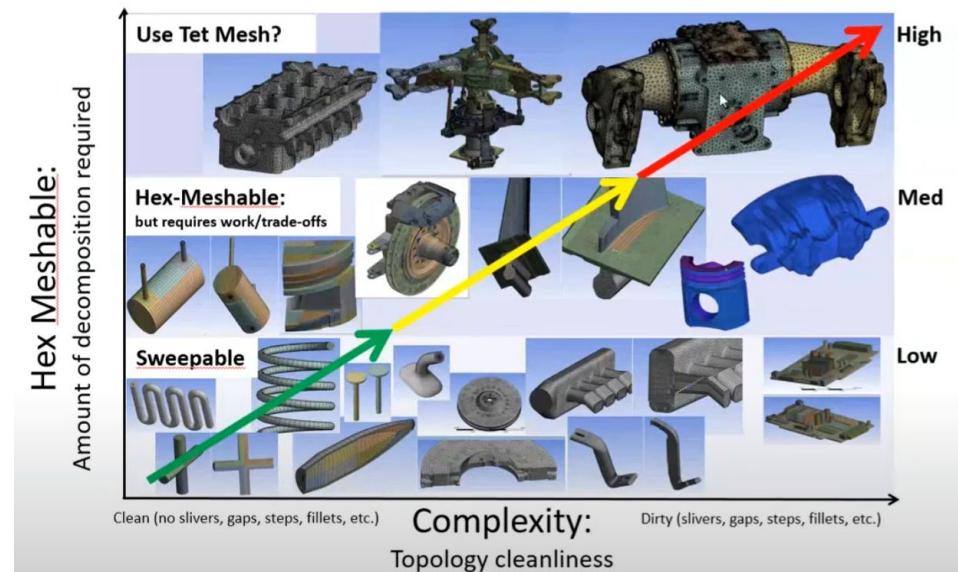
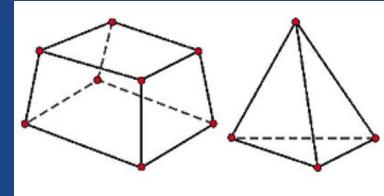


Image source: Leap Australia, *Introduction to ANSYS Meshing, Module 02: Meshing Methods*. <https://www.leapaustralia.com.au/leap-learning-hub/>

Meshing Advice

Mesh **should match the physics of the problem**

Check the **internal mesh** quality (sections)

Convert long parts into **1D beam elements** and thin-walled parts into **surface elements**

Huge mesh with lots of elements isn't necessarily good

Look for symmetrical and **sweepable** surfaces

Remember **symmetry**, and consider **2D solution** for problems which allow it

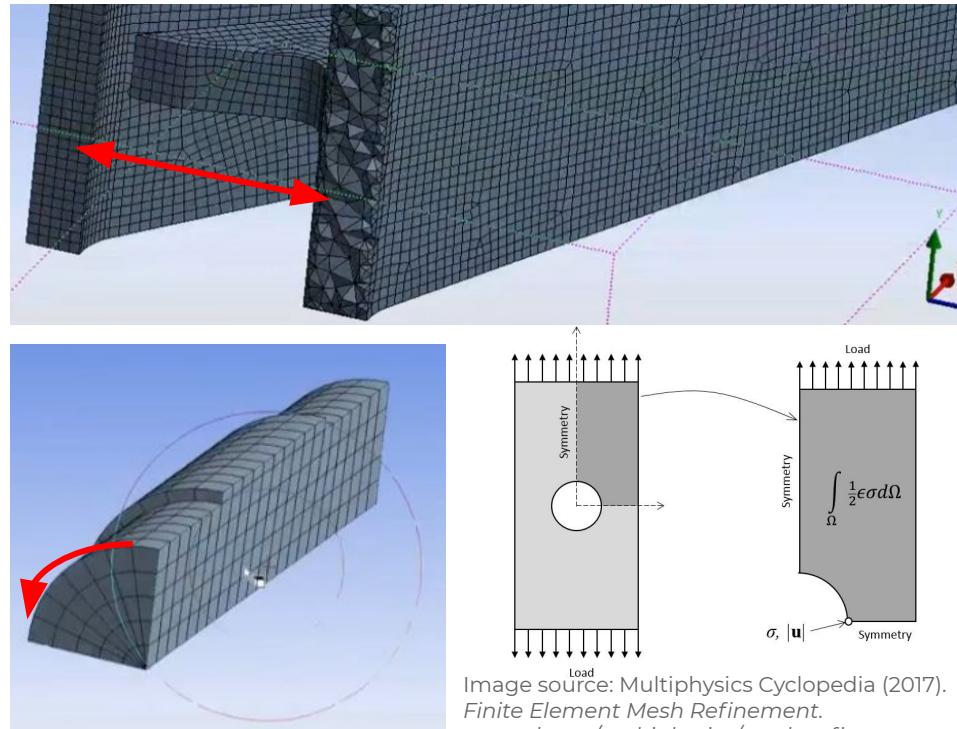


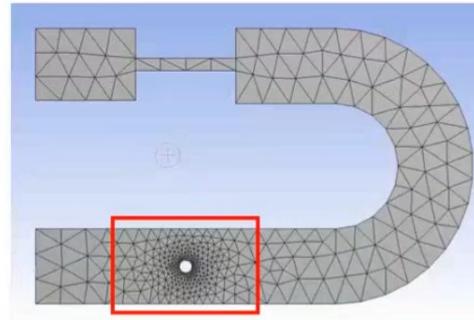
Image source: Multiphysics Cyclopedic (2017).
Finite Element Mesh Refinement.
comsol.com/multiphysics/mesh-refinement

Curvature and Proximity

Curvature sizing limits the angle between adjacent elements

Best used for curved surfaces

Growth rate determines transition to larger mesh away from curvature



Sizing	
<input type="checkbox"/>	Use Adaptive Sizing
<input type="checkbox"/>	Growth Rate
<input type="checkbox"/>	Max Size
<input type="checkbox"/>	Mesh Defeaturing
<input checked="" type="checkbox"/>	Capture Curvature
<input type="checkbox"/>	Curvature Min Size
<input type="checkbox"/>	Curvature Normal Angle
<input type="checkbox"/>	Capture Proximity
<input type="checkbox"/>	Proximity Min Size
<input type="checkbox"/>	Num Cells Across Gap
<input type="checkbox"/>	Proximity Size Function Sources

Curvature and Proximity

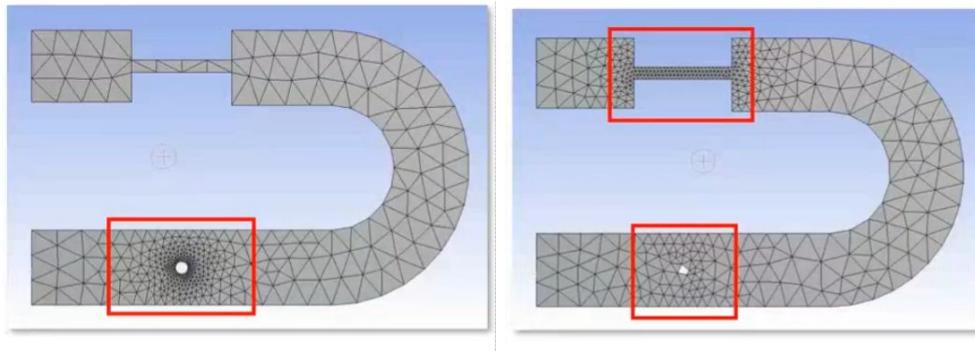
Curvature sizing limits the angle between adjacent elements

Best used for curved surfaces

Growth rate determines transition to larger mesh away from curvature

Proximity sizing ensures a minimum number of elements across **thin faces/narrow gaps**

Note this does not apply to the curved surface



Sizing	
<input type="checkbox"/> Use Adaptive Sizing	No
<input type="checkbox"/> Growth Rate	Default (1.85)
<input type="checkbox"/> Max Size	Default (16.0 mm)
Mesh Defeaturing	No
<input checked="" type="checkbox"/> Capture Curvature	Yes
<input type="checkbox"/> Curvature Min Size	Default (8.e-002 mm)
<input type="checkbox"/> Curvature Normal Angle	Default (70.395°)
Capture Proximity	
<input type="checkbox"/> Capture Proximity	Yes
<input type="checkbox"/> Proximity Min Size	4.0 mm
<input type="checkbox"/> Num Cells Across Gap	Default (3)
Proximity Size Function Sources	Faces and Edges

Defeaturing

Defeaturing ignores features below a certain size

Default feature size is half the minimum **local** element size

In areas of fine local mesh, features are kept

In areas of coarse local mesh, features are omitted

Defeaturing

Defeaturing ignores features below a certain size

Default feature size is half the minimum **local** element size

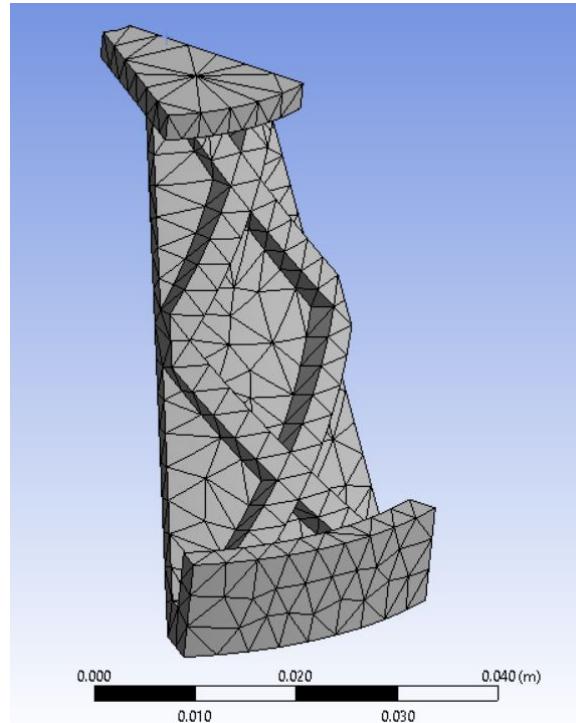
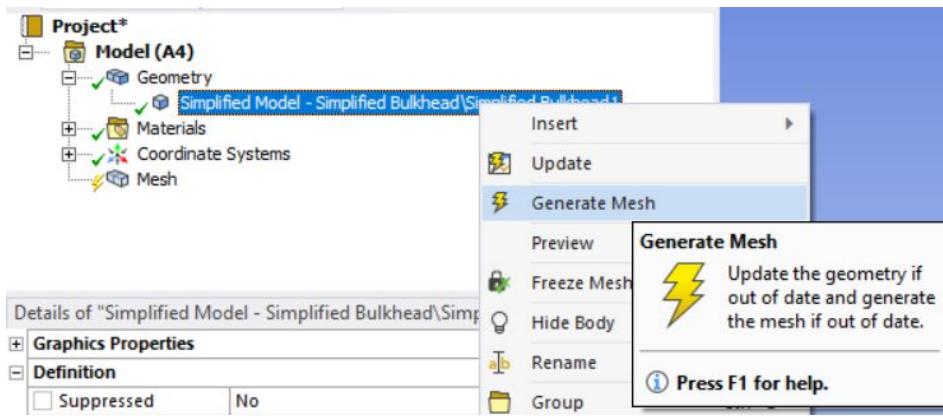
In areas of fine local mesh, features are kept

In areas of coarse local mesh, features are omitted

Default Mesh

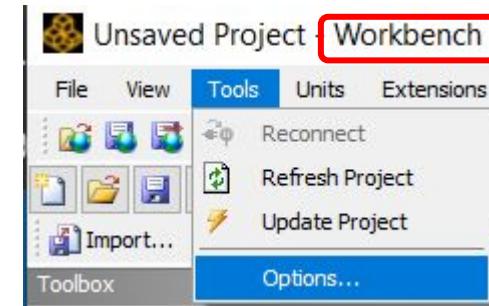
Can use the **default settings** (without any local fine-tuning to match the part geometry)

This mesh is generally quite **non-uniform** or poorly well-aligned with the part and loading



Mechanical Mouse Controls

Mouse controls for Ansys Mechanical edited in **Workbench** under **Graphics Interaction**



The screenshot shows the "Options" dialog box with the "Graphics Interaction" tab selected. The "Graphics Interaction" tab is highlighted in blue. The "Mouse Button Assignment" section contains three dropdown menus: "Mouse Wheel" (set to "Zoom"), "Middle Button" (set to "Rotate"), and "Right Button" (set to "Box Zoom").

Options

- Project Management
- Appearance
- Regional and Language Options
- Graphics Interaction**
- Journals and Logs
- Project Reporting
- Solution Process
- Extensions
- Mechanical APDL
- CFX
- Design Exploration
- Distributed Compute Services

Graphics Interaction

Mouse Button Assignment

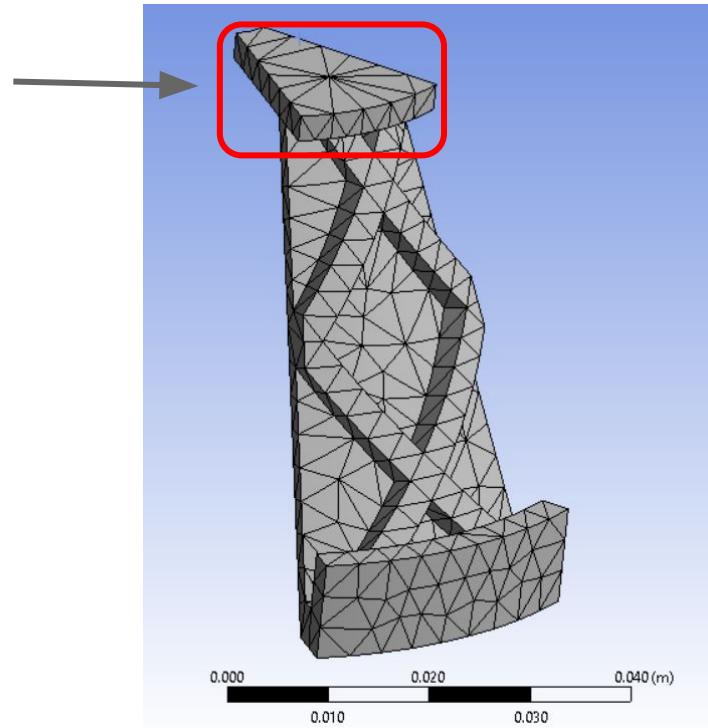
Mouse Wheel: Zoom

Middle Button: Rotate

Right Button: Box Zoom

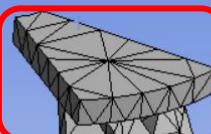
Default Mesh

Doesn't utilise **sweeping**

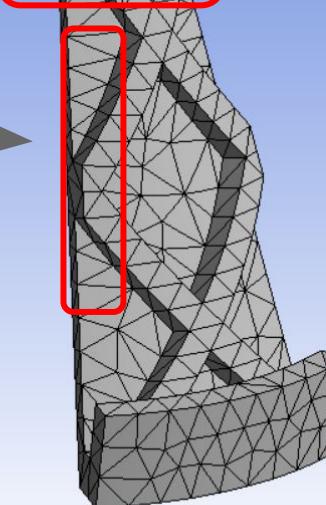


Default Mesh

Doesn't utilise **sweeping**



Doesn't align mesh with
loading direction

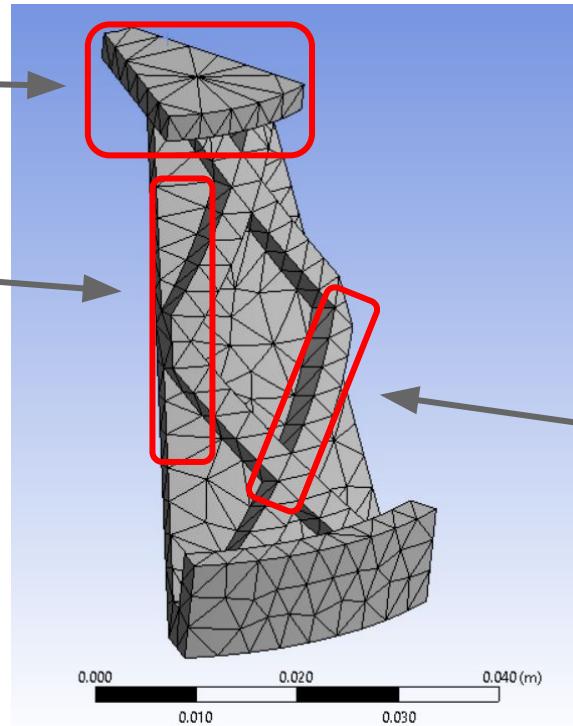


Default Mesh

Doesn't utilise **sweeping**

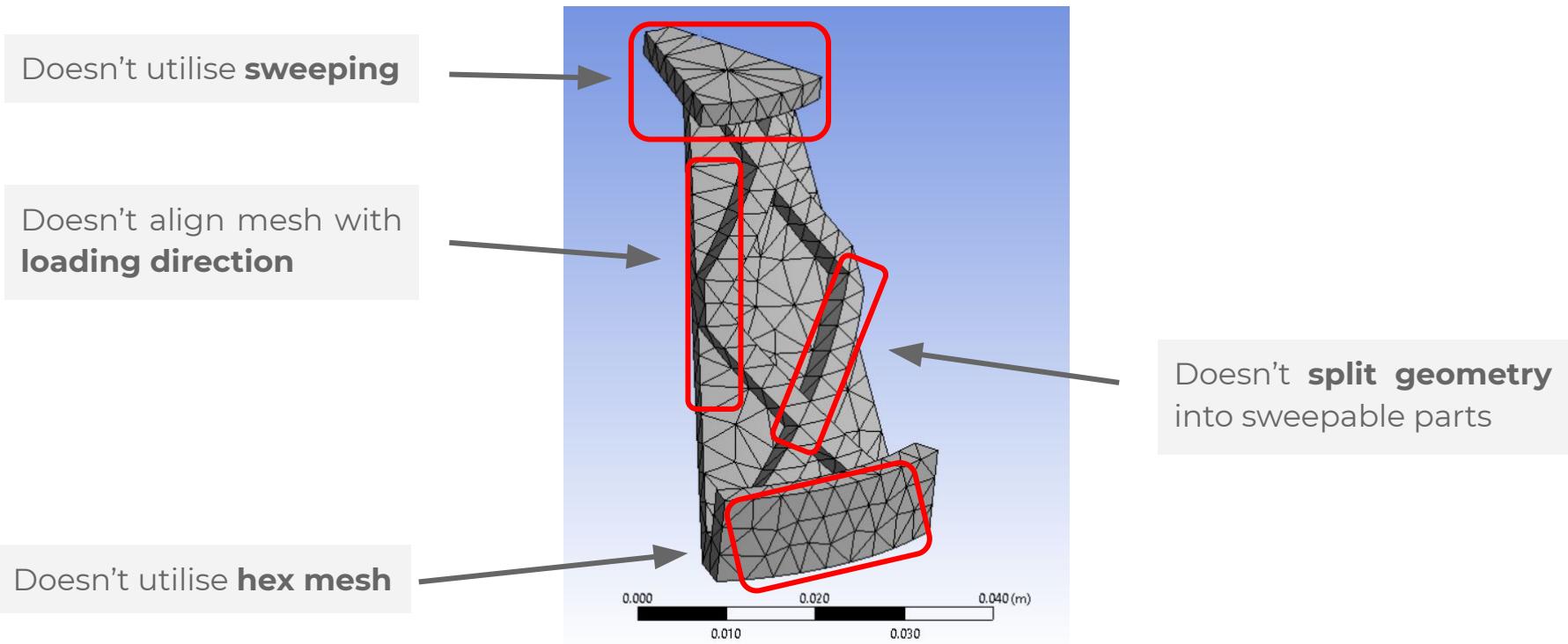


Doesn't align mesh with
loading direction



Doesn't **split geometry**
into sweepable parts

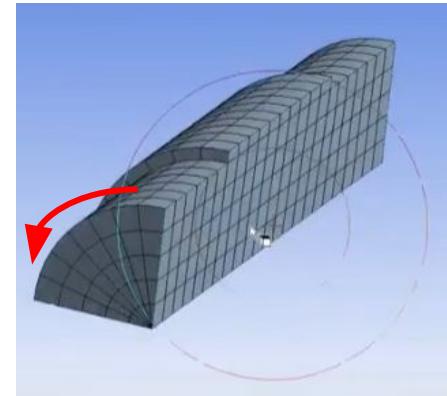
Default Mesh



SpaceClaim - Splitting Geometry

Let's **split** the body up **in SpaceClaim**

This will allow us to **sweep** the mesh



SpaceClaim - Splitting Geometry

Let's **split** the body up **in SpaceClaim**

This will allow us to **sweep** the mesh

Select **Split Body** tool



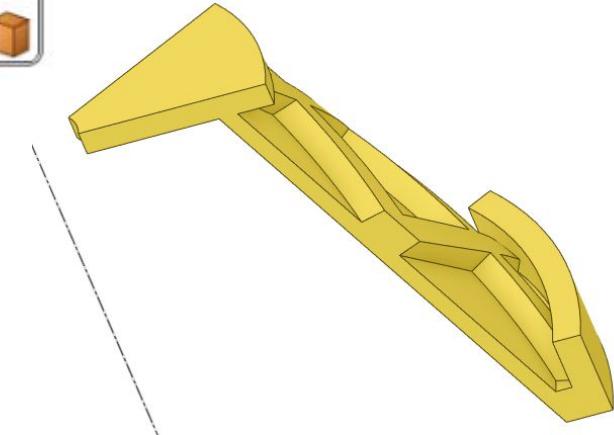
SpaceClaim - Splitting Geometry

Let's **split** the body up **in SpaceClaim**

This will allow us to **sweep** the mesh

Select **Split Body** tool

Select the body



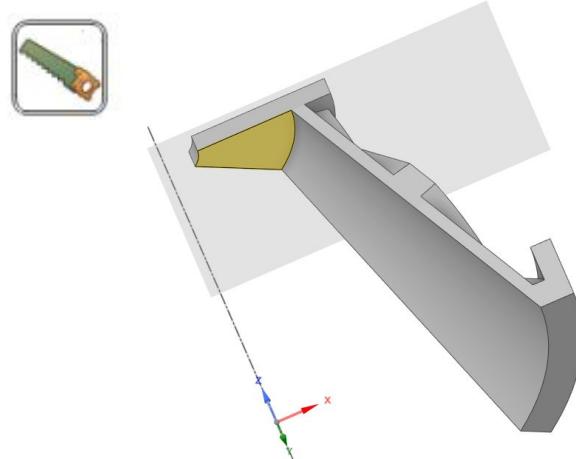
SpaceClaim - Splitting Geometry

Let's **split** the body up **in SpaceClaim**

This will allow us to **sweep** the mesh

Select **Split Body** tool

Select the body and the **face** to split by



SpaceClaim - Splitting Geometry

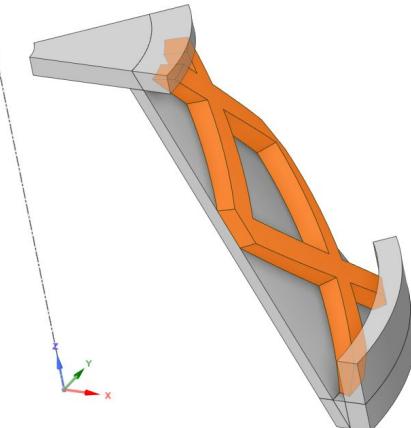
Let's **split** the body up **in SpaceClaim**

This will allow us to **sweep** the mesh

Select **Split Body** tool

Select the body and the **face** to split by

Separate **sweepable** and **non-sweepable** regions



SpaceClaim - Splitting Geometry

Let's **split** the body up **in SpaceClaim**

This will allow us to **sweep** the mesh

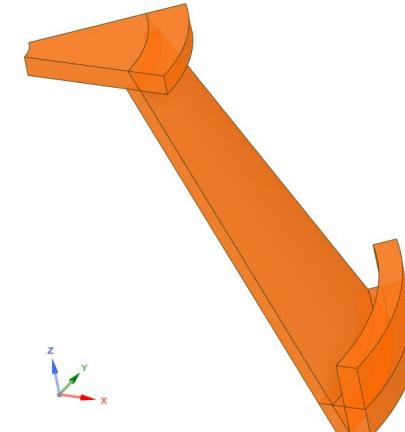
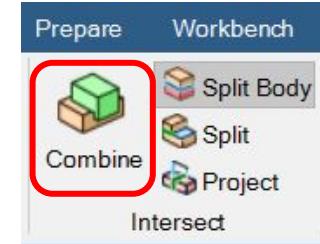
Select **Split Body** tool

Select the body and the **face** to split by

Separate **sweepable** and **non-sweepable** regions

Combine the sweepable regions

- Simplified Bulkhead1
- Simplified Bulkhead11
- Simplified Bulkhead111
- Simplified Bulkhead111
- NON-SWEEPABLE PART
- Simplified Bulkhead1111
- Simplified Bulkhead111
- Simplified Bulkhead1111



SpaceClaim - Splitting Geometry

Let's **split** the body up **in SpaceClaim**

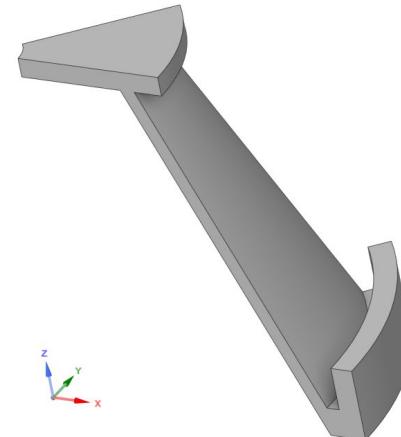
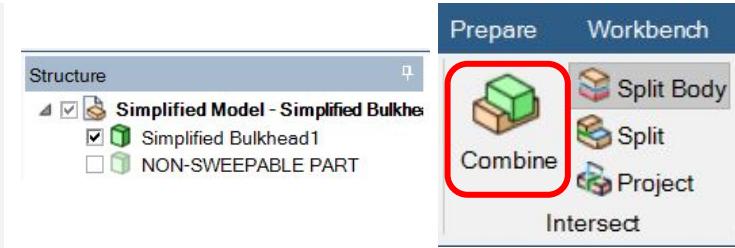
This will allow us to **sweep** the mesh

Select **Split Body** tool

Select the body and the **face** to split by

Separate **sweepable** and **non-sweepable** regions

Combine the sweepable regions



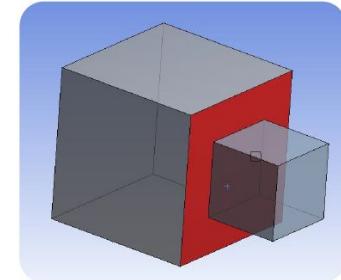
Sharing Topology - Skip this step!

For designs with multiple parts, it's best to **share** any contacting surfaces

When two parts don't share the contacting faces, the **mesh will be discontinuous**

When the parts share surfaces, the **mesh will be continuous** and the shared surface will be an **internal** surface

This is called “conformally meshed”



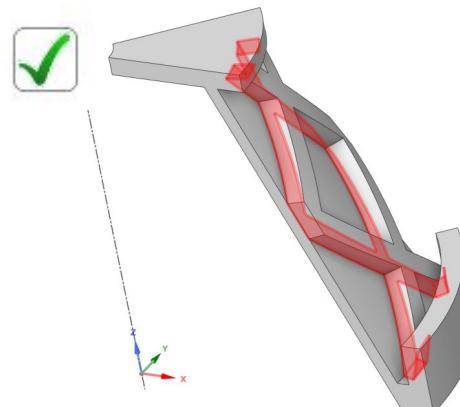
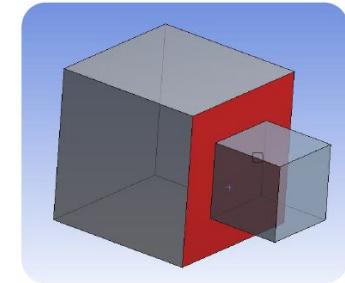
Sharing Topology - Skip this step!

For designs with multiple parts, it's best to **share** any contacting surfaces

When two parts don't share the contacting faces, the **mesh will be discontinuous**

When the parts share surfaces, the **mesh will be continuous** and the shared surface will be an **internal** surface

This is called "conformally meshed"



Sharing Topology - Skip this step!

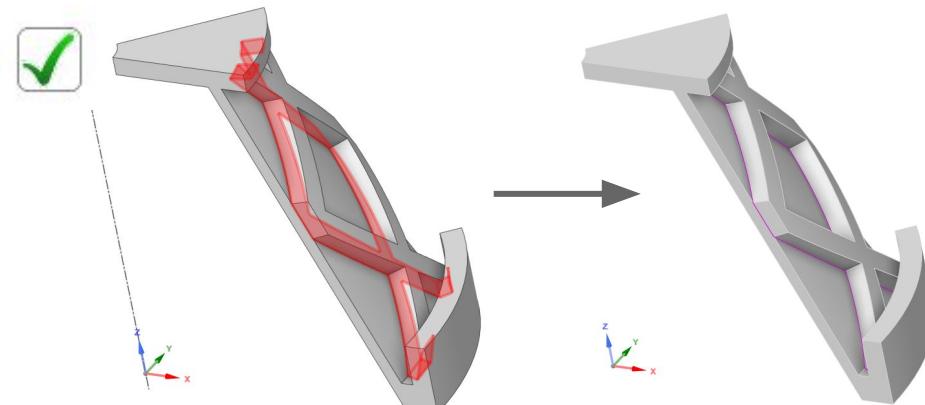
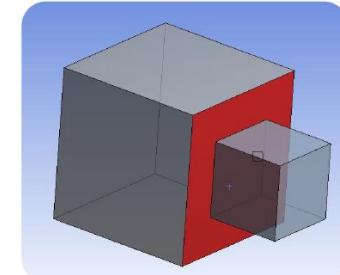
For designs with multiple parts, it's best to **share** any contacting surfaces

When two parts don't share the contacting faces, the **mesh will be discontinuous**

When the parts share surfaces, the **mesh will be continuous** and the shared surface will be an **internal** surface

This is called "conformally meshed"

Skip this step because we want to sweep one part and not the other - **meshes can't be aligned** at the intersection



Midsurfacing (Shell Meshes)

3D meshing needs uniform elements across each feature (ideally **3-4 elements across**)

Geometry with **very thin shells** → **huge number of tiny elements** required for 3D mesh

Modelling thin shells as 3D mesh actually leads to **large errors** in stress, deformation etc.

Much cheaper, **faster** (and more accurate) to model thin shells as **2D elements**

SpaceClaim calls this “**midsurfacing**”

Then use the **extend tool** to meet the faces

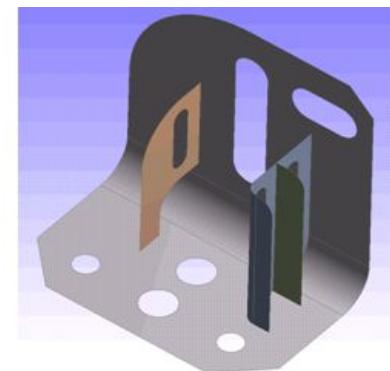
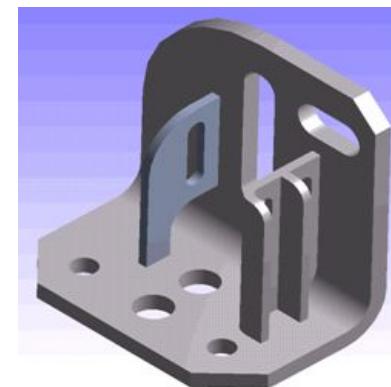
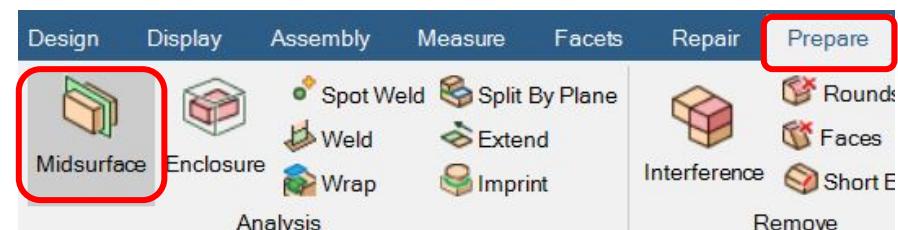


Image source: Ozen Engineering: Automation with the Flexibility to Customize
ozeninc.com/automation-wth-the-flexibility/



Midsurfacing (Shell Meshes)

3D meshing needs uniform elements across each feature (ideally **3-4 elements across**)

Geometry with **very thin shells** → **huge number of tiny elements** required for 3D mesh

Modelling thin shells as 3D mesh actually leads to **large errors** in stress, deformation etc.

Much cheaper, **faster** (and more **accurate**) to model thin shells as **2D elements**

SpaceClaim calls this “**midsurfacing**”

Then use the **extend tool** to meet the faces

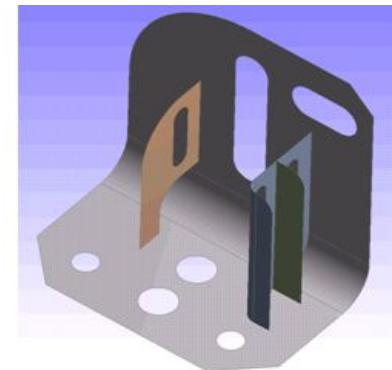
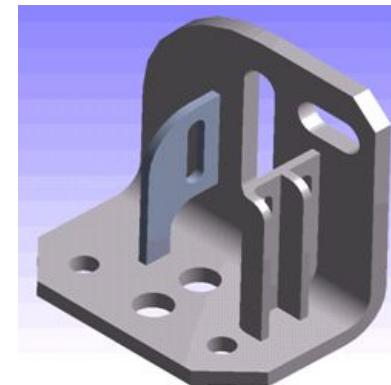


Image source: Ozen Engineering: Automation with the Flexibility to Customize
ozeninc.com/automation-wth-the-flexibility/



Midsurfacing (Shell Meshes)

3D meshing needs uniform elements across each feature (ideally **3-4 elements across**)

Geometry with **very thin shells → huge number of tiny elements** required for 3D mesh

Modelling thin shells as 3D mesh actually leads to **large errors** in stress, deformation etc.

Much cheaper, **faster** (and more accurate) to model thin shells as **2D elements**

SpaceClaim calls this “**midsurfacing**”

Then use the **extend tool** to meet the faces

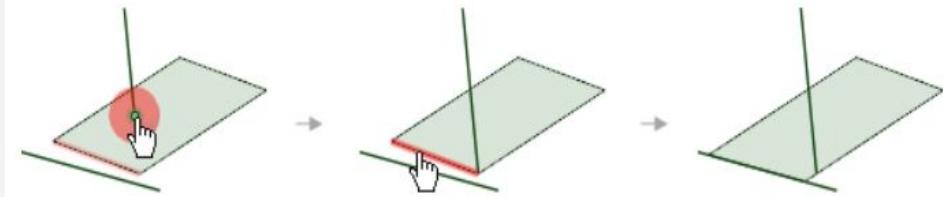


Image source: Ansys SpaceClaim Online Help
<http://help.spaceclaim.com/2015.0/en/Content/Extend.htm>



Beam Extraction

Beams FEA Guide (courtesy Leap Australia)

Trying to model long, thin beams as a solid 3D mesh is undesirable:

- Requires a tiny mesh with many elements. This means **huge solving time**
- Solid mesh gives **massive FEA error** for very long and thin beams

Much more efficient to model beams as **1D objects** which include profile and thickness as internal **parameters**

- More **accurate**
- **Faster** solving, meshing and post-processing

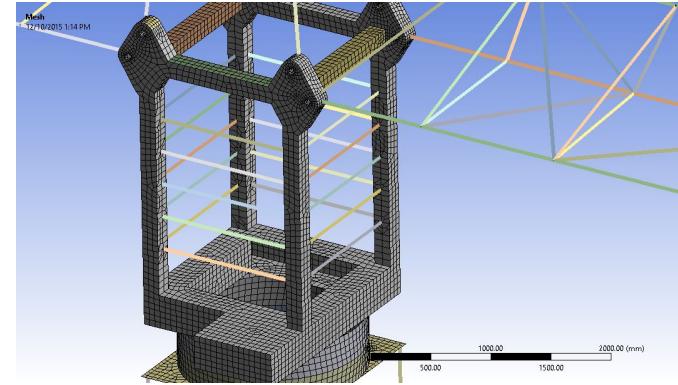


Image source: LEAP Australia: Analysing Large Fabricated Structures
finiteelementanalysis.com.au/featured/analysing-large-fabricated-structures/



Global Mesh Controls

Defaults section:

Physics preference - Mechanical or Nonlinear Mechanical.

Use **nonlinear** if need to consider **plastic deformation**.

Nonlinear option just produces higher-quality elements, to prepare for large distortions

Defaults	
Physics Preference	Mechanical
Element Order	Program Controlled
Element Size	Default (3.5467 mm)

Global Mesh Controls

Defaults section:

Physics preference - Mechanical or Nonlinear Mechanical.

Use **nonlinear** if need to consider **plastic deformation**.

Nonlinear option just produces higher-quality elements, to prepare for large distortions

Element order - linear, quadratic or **program controlled**

(linear for surfaces/beams, quadratic for 3D solids)

Defaults	
Physics Preference	Mechanical
Element Order	Program Controlled
Element Size	Default (3.5467 mm)

Global Mesh Controls

Defaults section:

Physics preference - Mechanical or Nonlinear Mechanical.

Use **nonlinear** if need to consider **plastic deformation**.

Nonlinear option just produces higher-quality elements, to prepare for large distortions

Element order - linear, quadratic or **program controlled**

(linear for surfaces/beams, quadratic for 3D solids)

Element size - used across the entire model (edges, faces, bodies)

Defaults	
Physics Preference	Mechanical
Element Order	Program Controlled
<input type="checkbox"/> Element Size	Default (3.5467 mm)

Global Mesh Controls

Sizing section:

Adaptive sizing - Ansys automatically captures proximity and curvature etc. Generally leads to poor mesh quality. We will set it to **No**

Sizing	
<input checked="" type="checkbox"/> Use Adaptive Sizing	No
<input type="checkbox"/> Growth Rate	Default (1.85)
<input type="checkbox"/> Max Size	5.0 mm
Mesh Defeaturing	Yes
<input type="checkbox"/> Defeature Size	Default (1.5e-002 mm)
Capture Curvature	Yes
<input type="checkbox"/> Curvature Min Size	Default (3.e-002 mm)
<input type="checkbox"/> Curvature Normal Angle	Default (70.395°)
Capture Proximity	Yes
<input type="checkbox"/> Proximity Min Size	Default (3.e-002 mm)
<input type="checkbox"/> Num Cells Across Gap	Default (3)
Proximity Size Function Sources	Faces and Edges
Bounding Box Diagonal	70.934 mm
Average Surface Area	124.15 mm ²
Minimum Edge Length	1.76 mm

Global Mesh Controls

Sizing section:

Adaptive sizing - Ansys automatically captures proximity and curvature etc. Generally leads to poor mesh quality. We will set it to **No**

Growth rate - increase in element size with each successive element, We'll use which **1.2** means each element can be 20% bigger than the last

Sizing	
Use Adaptive Sizing	No
<input type="checkbox"/> Growth Rate	Default (1.85)
<input type="checkbox"/> Max Size	5.0 mm
Mesh Defeaturing	Yes
<input type="checkbox"/> Defeature Size	Default (1.5e-002 mm)
Capture Curvature	Yes
<input type="checkbox"/> Curvature Min Size	Default (3.e-002 mm)
<input type="checkbox"/> Curvature Normal Angle	Default (70.395°)
Capture Proximity	Yes
<input type="checkbox"/> Proximity Min Size	Default (3.e-002 mm)
<input type="checkbox"/> Num Cells Across Gap	Default (3)
Proximity Size Function Sources	Faces and Edges
Bounding Box Diagonal	70.934 mm
Average Surface Area	124.15 mm ²
Minimum Edge Length	1.76 mm

Global Mesh Controls

Sizing section:

Adaptive sizing - Ansys automatically captures proximity and curvature etc. Generally leads to poor mesh quality. We will set it to **No**

Growth rate - increase in element size with each successive element, We'll use which **1.2** means each element can be 20% bigger than the last

Max. size - absolute max. element size. Sometimes mesher will exceed selected “Element size” to improve mech quality.
Must be larger than “Element size”

Sizing	
Use Adaptive Sizing	No
<input type="checkbox"/> Growth Rate	Default (1.85)
<input type="checkbox"/> Max Size	5.0 mm
Mesh Defeaturing	Yes
<input type="checkbox"/> Defeature Size	Default (1.5e-002 mm)
Capture Curvature	Yes
<input type="checkbox"/> Curvature Min Size	Default (3.e-002 mm)
<input type="checkbox"/> Curvature Normal Angle	Default (70.395°)
Capture Proximity	Yes
<input type="checkbox"/> Proximity Min Size	Default (3.e-002 mm)
<input type="checkbox"/> Num Cells Across Gap	Default (3)
Proximity Size Function Sources	Faces and Edges
Bounding Box Diagonal	70.934 mm
Average Surface Area	124.15 mm ²
Minimum Edge Length	1.76 mm

Global Mesh Controls

Sizing section:

Mesh defeaturig - features smaller than or equal to the **Defeature Size** are removed automatically.

Sizing	
Use Adaptive Sizing	No
<input type="checkbox"/> Growth Rate	Default (1.85)
<input type="checkbox"/> Max Size	5.0 mm
<input checked="" type="checkbox"/> Mesh Defeaturing	Yes
<input type="checkbox"/> Defeature Size	Default (1.5e-002 mm)
Capture Curvature	Yes
<input type="checkbox"/> Curvature Min Size	Default (3.e-002 mm)
<input type="checkbox"/> Curvature Normal Angle	Default (70.395°)
Capture Proximity	Yes
<input type="checkbox"/> Proximity Min Size	Default (3.e-002 mm)
<input type="checkbox"/> Num Cells Across Gap	Default (3)
Proximity Size Function Sources	Faces and Edges
Bounding Box Diagonal	70.934 mm
Average Surface Area	124.15 mm ²
Minimum Edge Length	1.76 mm

Global Mesh Controls

Sizing section:

Mesh defeaturig - features smaller than or equal to the **Defeature Size** are removed automatically.

Capture Curvature - limits the angle between adjacent elements. **Curvature Min Size** becomes the minimum element size at curved regions. **Curvature Normal Angle** is the maximum angle allowed between elements.

Sizing	
Use Adaptive Sizing	No
<input type="checkbox"/> Growth Rate	Default (1.85)
<input type="checkbox"/> Max Size	5.0 mm
Mesh Defeaturing	Yes
<input type="checkbox"/> Defeature Size	Default (1.5e-002 mm)
Capture Curvature	Yes
<input type="checkbox"/> Curvature Min Size	Default (3.e-002 mm)
<input type="checkbox"/> Curvature Normal Angle	Default (70.395°)
Capture Proximity	Yes
<input type="checkbox"/> Proximity Min Size	Default (3.e-002 mm)
<input type="checkbox"/> Num Cells Across Gap	Default (3)
Proximity Size Function Sources	Faces and Edges
Bounding Box Diagonal	70.934 mm
Average Surface Area	124.15 mm ²
Minimum Edge Length	1.76 mm

Global Mesh Controls

Sizing section:

Mesh defeaturing - features smaller than or equal to the **Defeature Size** are removed automatically.

Capture Curvature - limits the angle between adjacent elements. **Curvature Min Size** becomes the minimum element size at curved regions. **Curvature Normal Angle** is the maximum angle allowed between elements.

Capture Proximity - ensures a minimum number of elements across thin faces/narrow gaps. **Proximity Min Size** becomes the minimum element size for these regions. **Num Cells Across Gap** is the minimum number of layers of elements generated in the gaps.

Sizing	
Use Adaptive Sizing	No
<input type="checkbox"/> Growth Rate	Default (1.85)
<input type="checkbox"/> Max Size	5.0 mm
Mesh Defeaturing	Yes
<input type="checkbox"/> Defeature Size	Default (1.5e-002 mm)
Capture Curvature	Yes
<input type="checkbox"/> Curvature Min Size	Default (3.e-002 mm)
<input type="checkbox"/> Curvature Normal Angle	Default (70.395°)
Capture Proximity	Yes
<input type="checkbox"/> Proximity Min Size	Default (3.e-002 mm)
<input type="checkbox"/> Num Cells Across Gap	Default (3)
Proximity Size Function Sources	Faces and Edges
Bounding Box Diagonal	70.934 mm
Average Surface Area	124.15 mm ²
Minimum Edge Length	1.76 mm

Global Mesh Controls

Quality section:

Check Mesh Quality - If can't generate a mesh that passes all error limits, error message is printed and meshing fails

Quality	
Check Mesh Quality	Yes, Errors
Error Limits	Aggressive Mechanical
<input type="checkbox"/> Target Quality	Default (0.050000)
Smoothing	Medium
Mesh Metric	None

Global Mesh Controls

Quality section:

Check Mesh Quality - If can't generate a mesh that passes all error limits, error message is printed and meshing fails

Error Limits - Set to **Aggressive Mechanical** (makes a mesh of similar quality to Nonlinear Mechanical)

Quality	
Check Mesh Quality	Yes, Errors
Error Limits	Aggressive Mechanical
<input type="checkbox"/> Target Quality	Default (0.050000)
Smoothing	Medium
Mesh Metric	None

Global Mesh Controls

Quality section:

Check Mesh Quality - If can't generate a mesh that passes all error limits, error message is printed and meshing fails

Error Limits - Set to **Aggressive Mechanical** (makes a mesh of similar quality to Nonlinear Mechanical)

Target Quality - target for each element. element quality is related to **uniformity/skewness of elements**. Aims for the target specified by adjusting tet elements. If target can't be met, a warning is given but mesh still generated

Quality	
Check Mesh Quality	Yes, Errors
Error Limits	Aggressive Mechanical
<input type="checkbox"/> Target Quality	Default (0.050000)
Smoothing	Medium
Mesh Metric	None

Global Mesh Controls

Quality section:

Check Mesh Quality - If can't generate a mesh that passes all error limits, error message is printed and meshing fails

Error Limits - Set to **Aggressive Mechanical** (makes a mesh of similar quality to Nonlinear Mechanical)

Target Quality - target for each element. element quality is related to **uniformity/skewness of elements**. Aims for the target specified by adjusting tet elements. If target can't be met, a warning is given but mesh still generated

Smoothing - improve element quality by moving locations of nodes

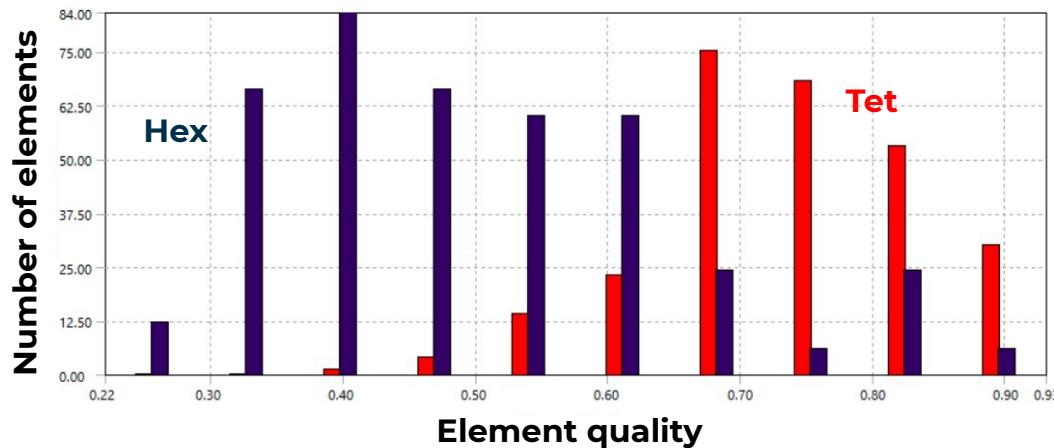
Quality	
Check Mesh Quality	Yes, Errors
Error Limits	Aggressive Mechanical
<input type="checkbox"/> Target Quality	Default (0.050000)
Smoothing	Medium
Mesh Metric	None

Global Mesh Controls

Quality section:

Mesh Metric - view some metrics to measure the mesh quality yourself. E.g. Element Quality (0-1), Aspect Ratio, Maximum Corner Angle, Skewness (0-1), ...

Quality	
Check Mesh Quality	Yes, Errors
Error Limits	Aggressive Mechanical
<input type="checkbox"/> Target Quality	Default (0.050000)
Smoothing	Medium
Mesh Metric	None

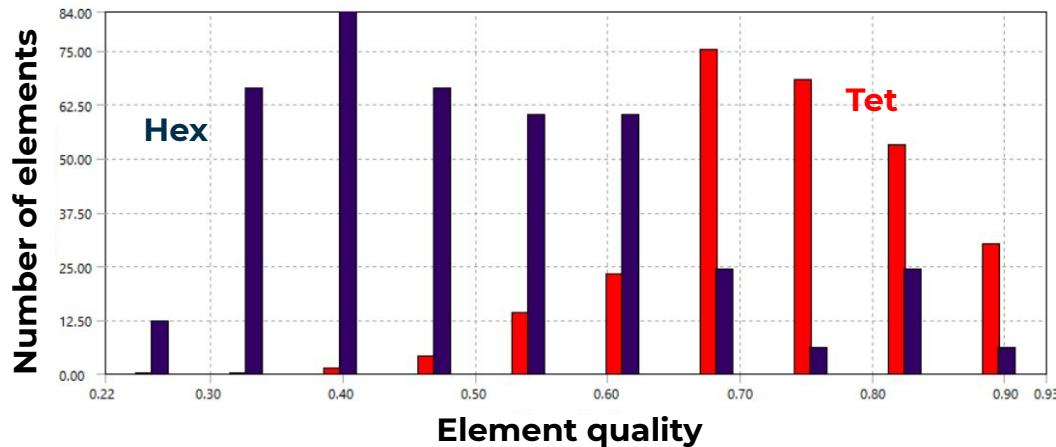


Sweep Meshing

Quality section:

Mesh Metric - view some metrics to measure the mesh quality yourself. E.g. Element Quality (0-1), Aspect Ratio, Maximum Corner Angle, Skewness (0-1), ...

Quality	
Check Mesh Quality	Yes, Errors
Error Limits	Aggressive Mechanical
<input type="checkbox"/> Target Quality	Default (0.050000)
Smoothing	Medium
Mesh Metric	None



Contacts

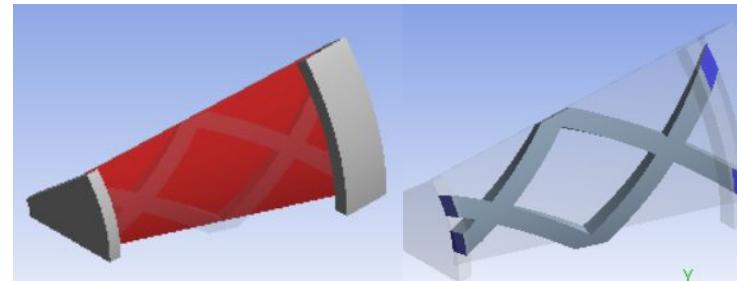
Define the types of contact between solid bodies

The “cone” body has been split from “hatch” body

Want them to be **bonded** - they are the **same part**



Target	7 Faces
Contact Bodies	Simplified Model - Simplified Bulkhead\Simplifi...
Target Bodies	Simplified Model - Simplified Bulkhead\Simplifi...
Protected	No
Definition	
Type	Bonded

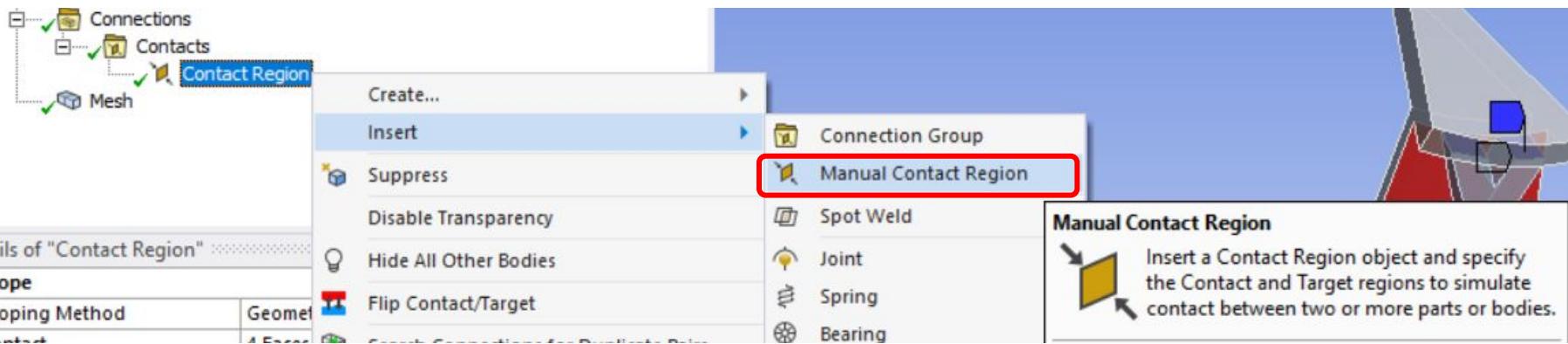


Contacts

Define the types of contact between solid bodies

The “cone” body has been split from “hatch” body

Want them to be **bonded** - they are the **same part**



Contacts

Define the types of contact between solid bodies

The “cone” body has been split from “hatch” body

Want them to be **bonded** - they are the **same part**

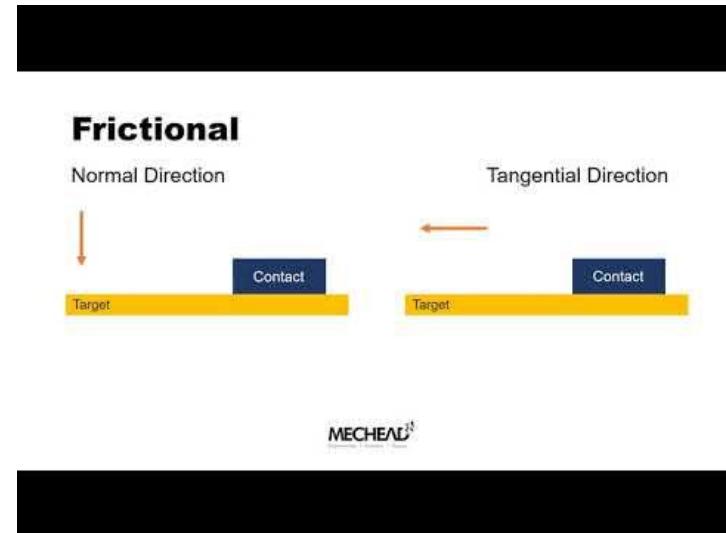
Other options include:

No separation - slide without resistance but can't separate

Frictionless - can **separate and slide** without resistance

Frictional - can separate and slide with **resistance**

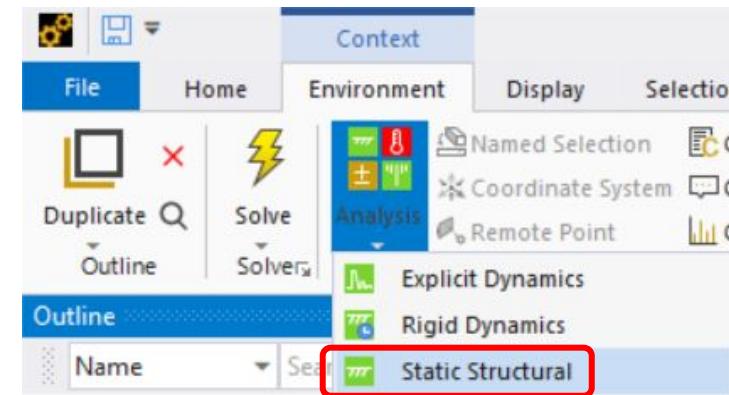
Rough - can **separate** but cannot slide





BCs - Supports

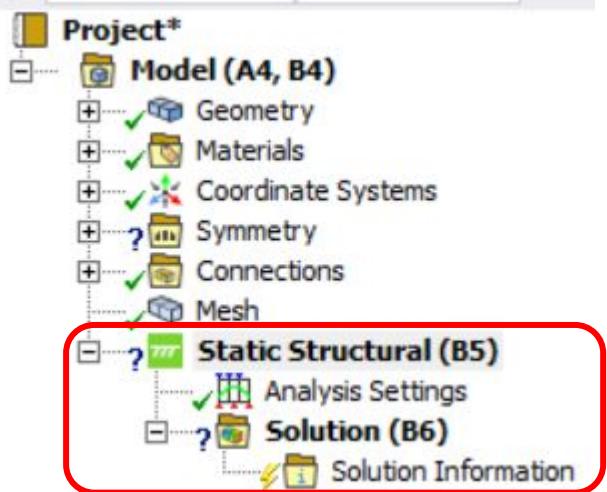
Insert a **Static Structural Analysis**





BCs - Supports

Insert a **Static Structural Analysis**

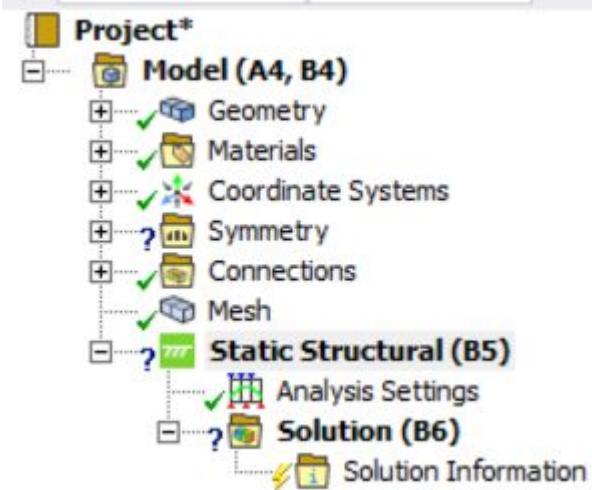




BCs - Supports

Insert a **Static Structural Analysis**

Insert a **Fixed Support** boundary condition



The ribbon menu shows the following tabs:

- File
- Home
- Environment (Selected)
- Display
- Selection
- Automation

The Tools ribbon contains the following buttons:

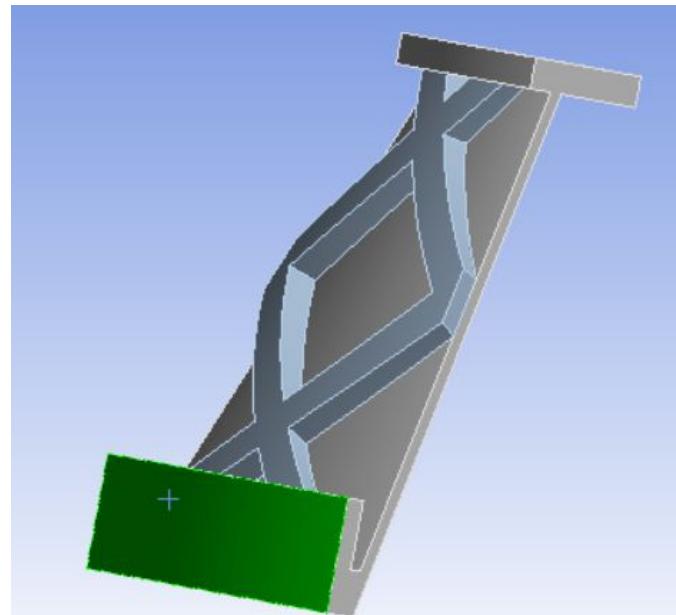
Duplicate	Named Selection	Commands	Inertial	Force	Fixed
Outline	Coordinate System	Comment	Loads	Moment	Frictionless
Solve	Remote Point	Chart	Section Plane	Pressure	Displacement
Solvers	Analysis	Insert	Annotation	Structural	

BCs - Supports

Insert a **Static Structural Analysis**

Insert a **Fixed Support** boundary condition

Select the appropriate faces for the fixed support

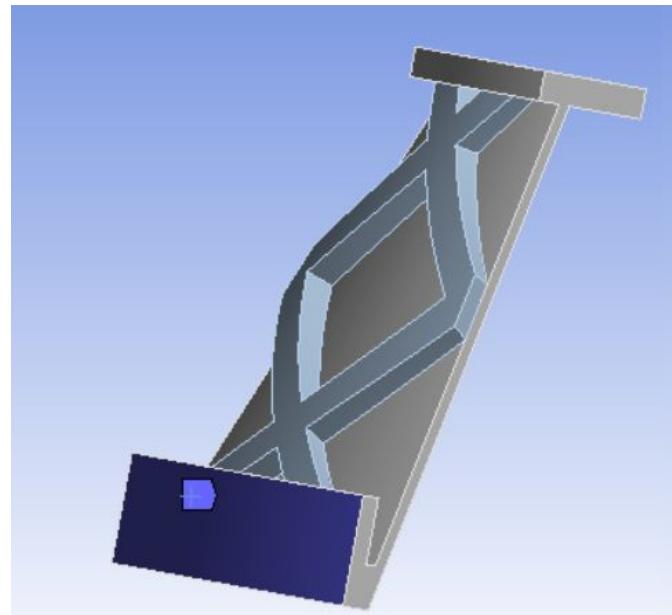
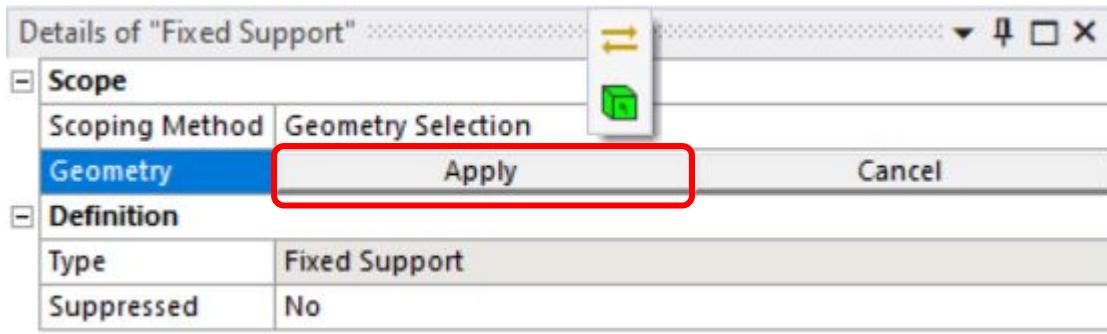


BCs - Supports

Insert a **Static Structural Analysis**

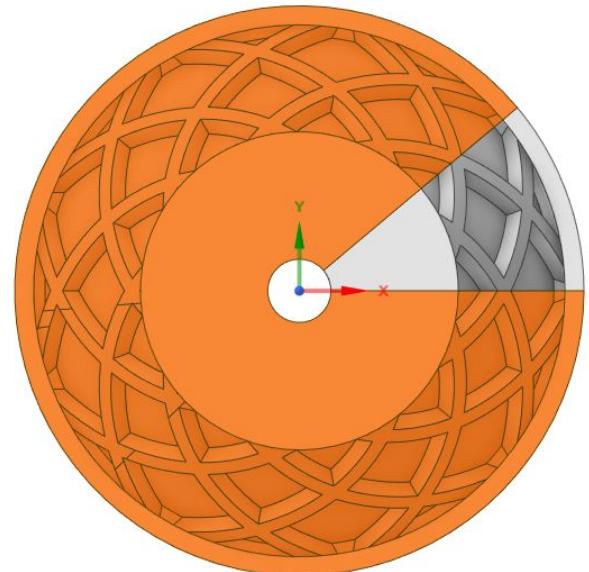
Insert a **Fixed Support** boundary condition

Select the appropriate faces for the fixed support



BCs - Symmetry

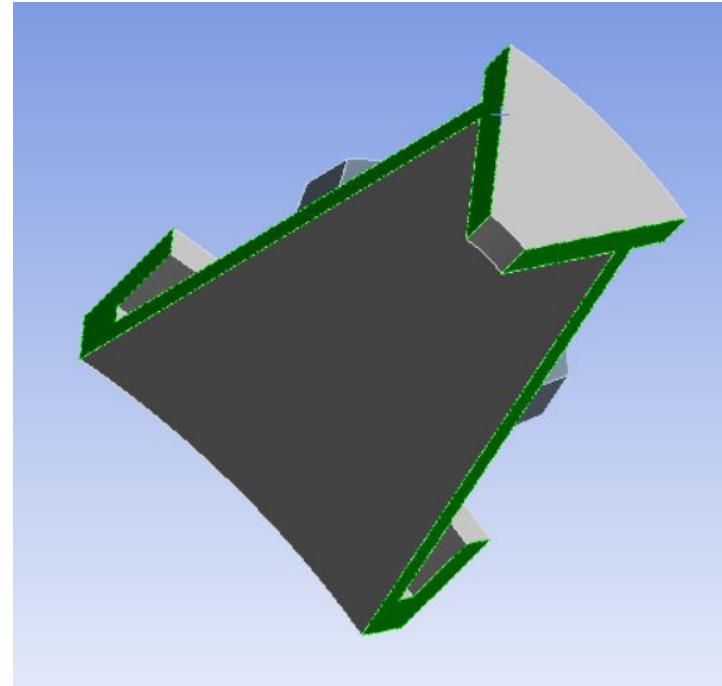
The wedge we're modelling is **constrained** by the rest of the bulkhead around it



BCs - Symmetry

The wedge we're modelling is **constrained** by the rest of the bulkhead around it

These faces can't move outside their own planes

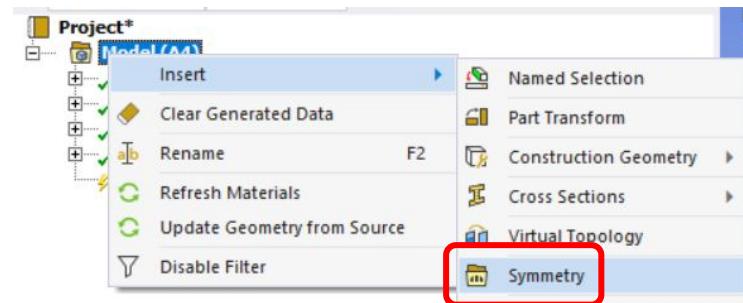


BCs - Symmetry

The wedge we're modelling is **constrained** by the rest of the bulkhead around it

These faces can't move outside their own planes

Symmetry BCs limit the motion of a face to a single plane

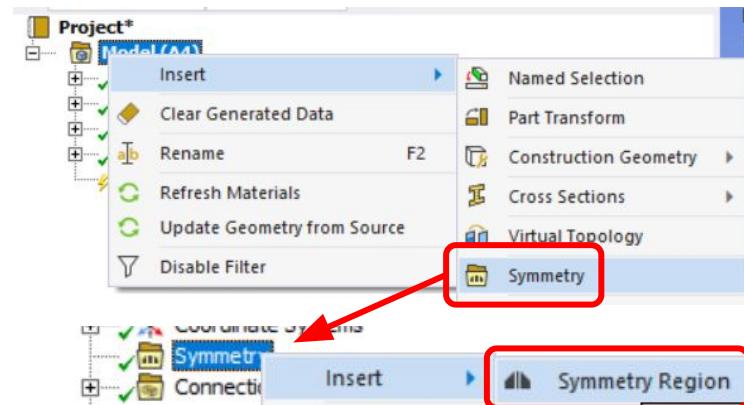


BCs - Symmetry

The wedge we're modelling is **constrained** by the rest of the bulkhead around it

These faces can't move outside their own planes

Symmetry BCs limit the motion of a face to a single plane



BCs - Symmetry

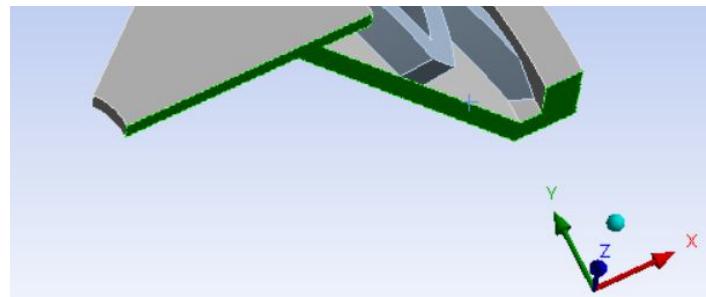
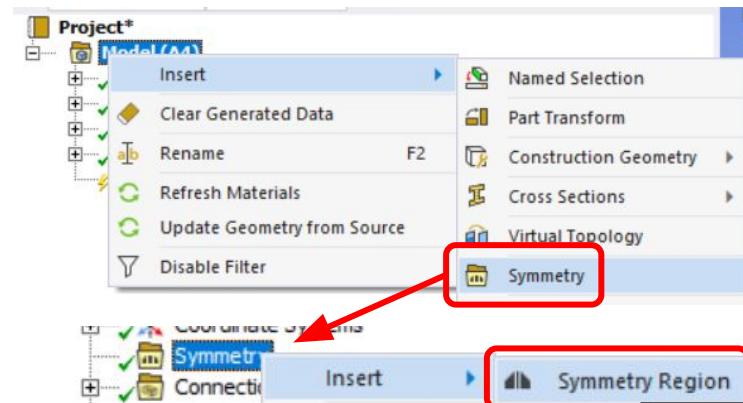
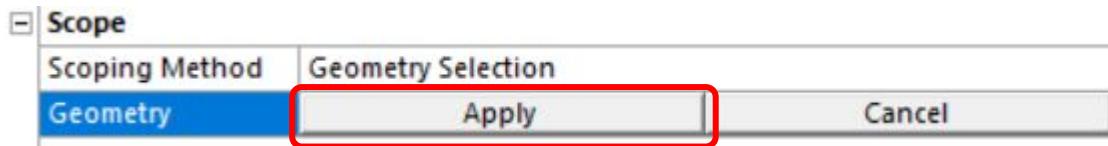
The wedge we're modelling is **constrained** by the rest of the bulkhead around it

These faces can't move outside their own planes

Symmetry BCs limit the motion of a face to a single plane

Select "Geometry" option on bottom left

Select the **face** which is normal to the **y** axis



BCs - Symmetry

The wedge we're modelling is **constrained** by the rest of the bulkhead around it

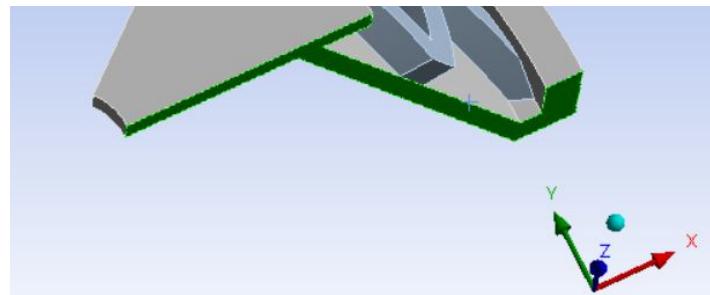
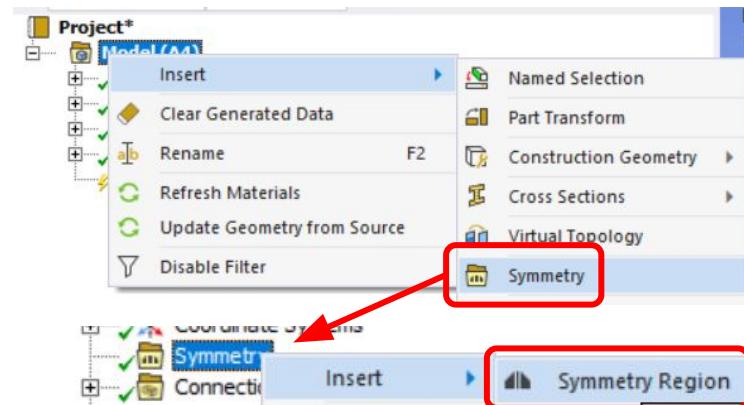
These faces can't move outside their own planes

Symmetry BCs limit the motion of a face to a single plane

Select “Geometry” option on bottom left

Select the face which is normal to the **y** axis

Scope	
Scoping Method	Geometry Selection
Geometry	1 Face





BCs - Symmetry

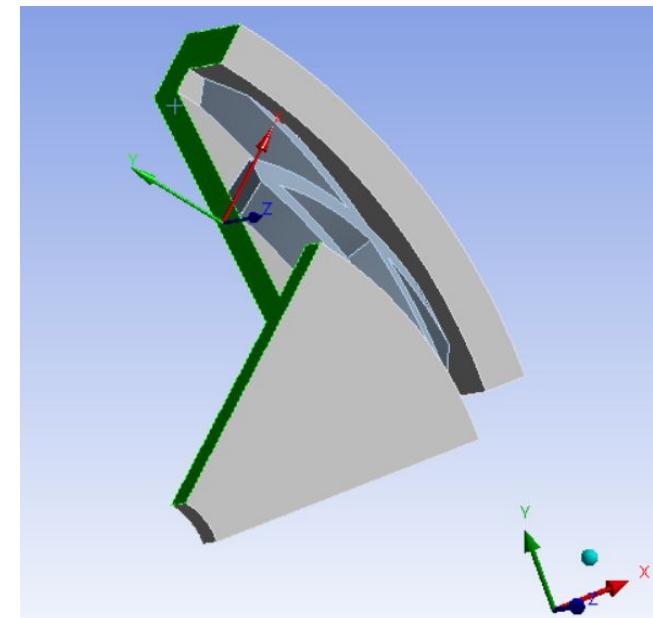
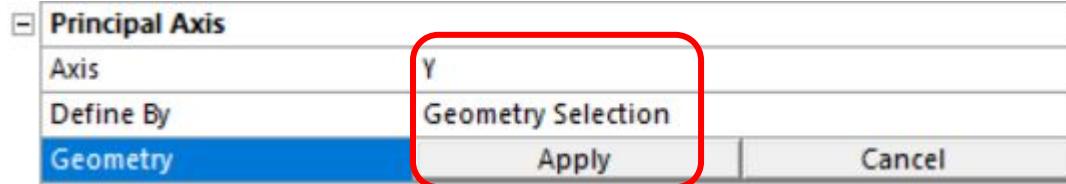
Need a **coordinate system** normal to the other face



BCs - Symmetry

Need a **coordinate system** normal to the other face

Want principal **y** axis normal to the **face** shown

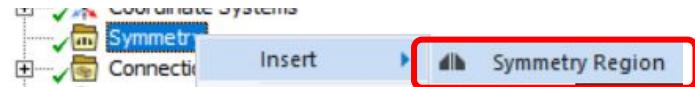


BCs - Symmetry

Need a **coordinate system** normal to the other face

Want principal **y** axis normal to the **face** shown

Insert another **symmetry region**



BCs - Symmetry

Need a **coordinate system** normal to the other face

Want principal **y** axis normal to the **face** shown

Insert another **symmetry region**

Select the **y** axis we created as the normal



Definition	
Scope Mode	Manual
Type	Symmetric
Coordinate System	Coordinate System
Symmetry Normal	Y Axis
Suppressed	No

BCs - Symmetry

Need a **coordinate system** normal to the other face

Want principal **y** axis normal to the **face** shown

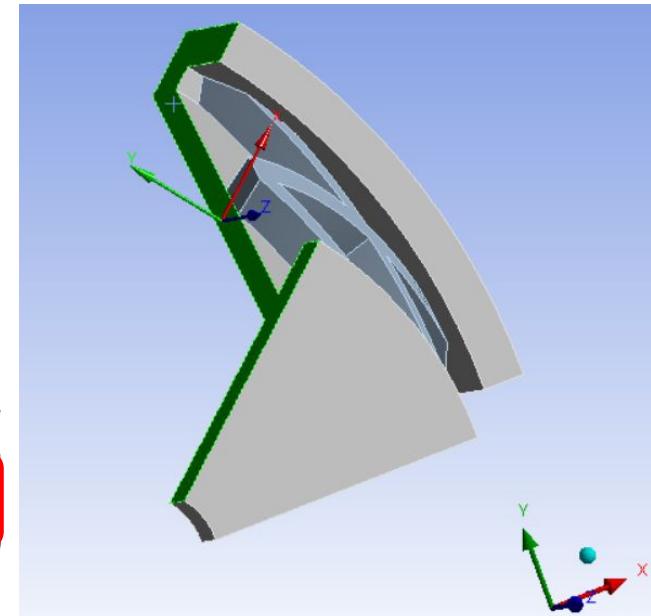
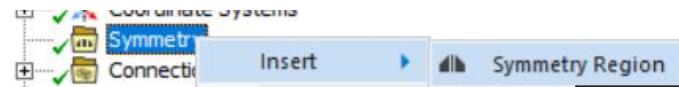
Insert another **symmetry region**

Select the **y** axis we created as the normal

Select the other **face**

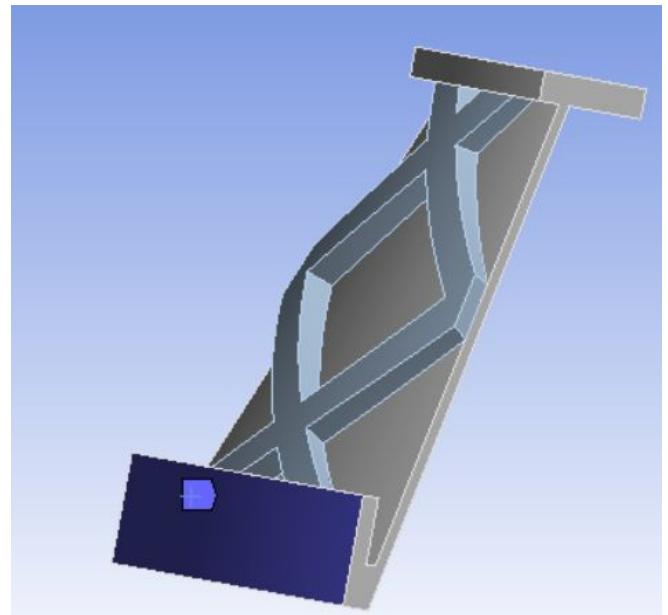
Definition	
Scope Mode	Manual
Type	Symmetric
Coordinate System	Coordinate System
Symmetry Normal	Y Axis
Suppressed	No

Scope	
Scoping Method	Geometry Selection
Geometry	Apply



Forces

Ensure the body is **constrained** by enough **supports**

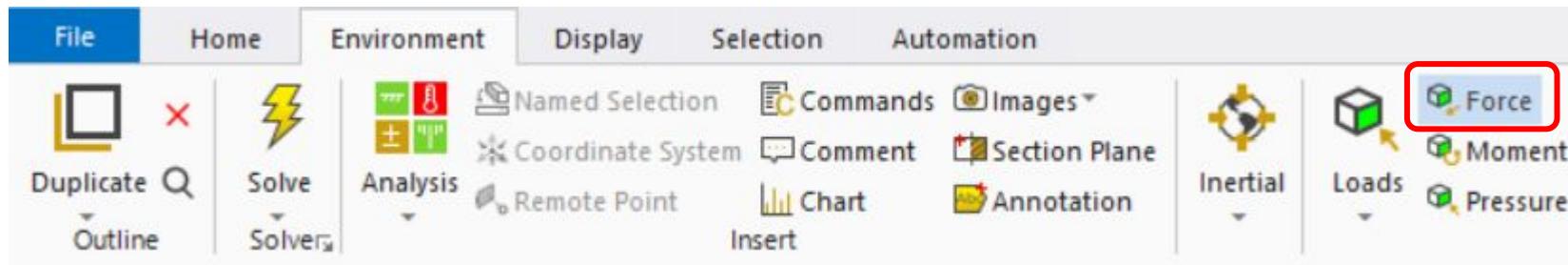




Forces

Ensure the body is **constrained** by enough **supports**

Insert a **force**



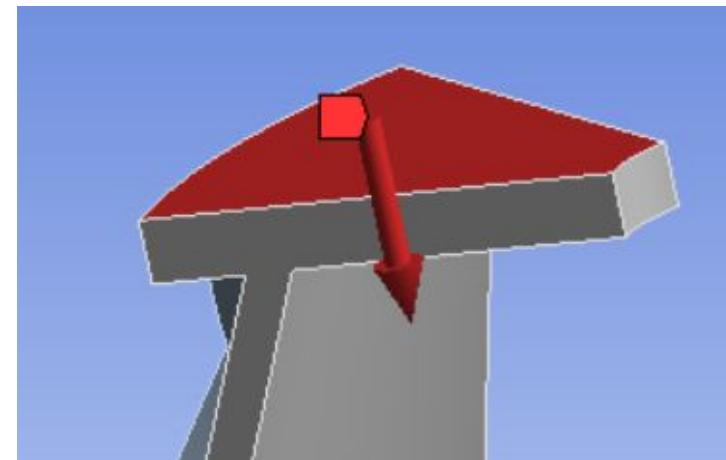
Forces

Ensure the body is **constrained** by enough **supports**

Insert a **force**

Select the **face**, **magnitude** and **direction**

Details of "Force"	
<input type="button" value="▼"/>	
Scoping Method	Geometry Selection
<input checked="" type="button" value="Geometry Selection"/>	
Geometry	1 Face
<input type="button" value="■"/>	
<input type="button" value="X"/>	
<input type="button" value="Scope"/>	
Type	Force
Define By	Vector
Applied By	Surface Effect
<input type="checkbox"/> Magnitude	10. N (ramped)
Direction	Click to Change
Suppressed	No



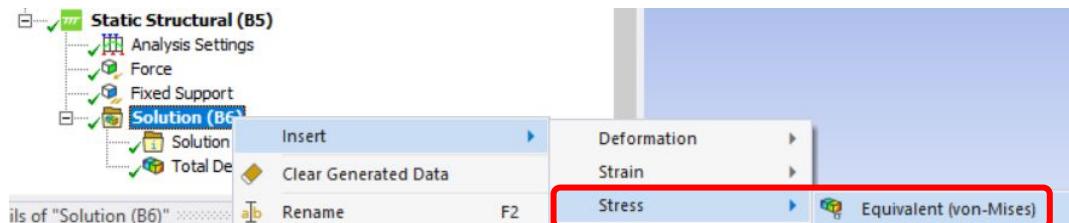
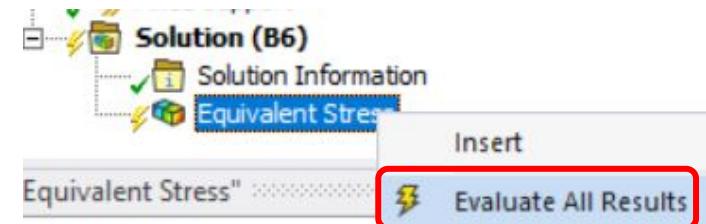
Solve

Set up BCs and loading conditions

Add a solution to the **Static Structural** analysis

Solve the analysis

Adjust **scaling** if needed



That's it for this year!

See you next year! :)

