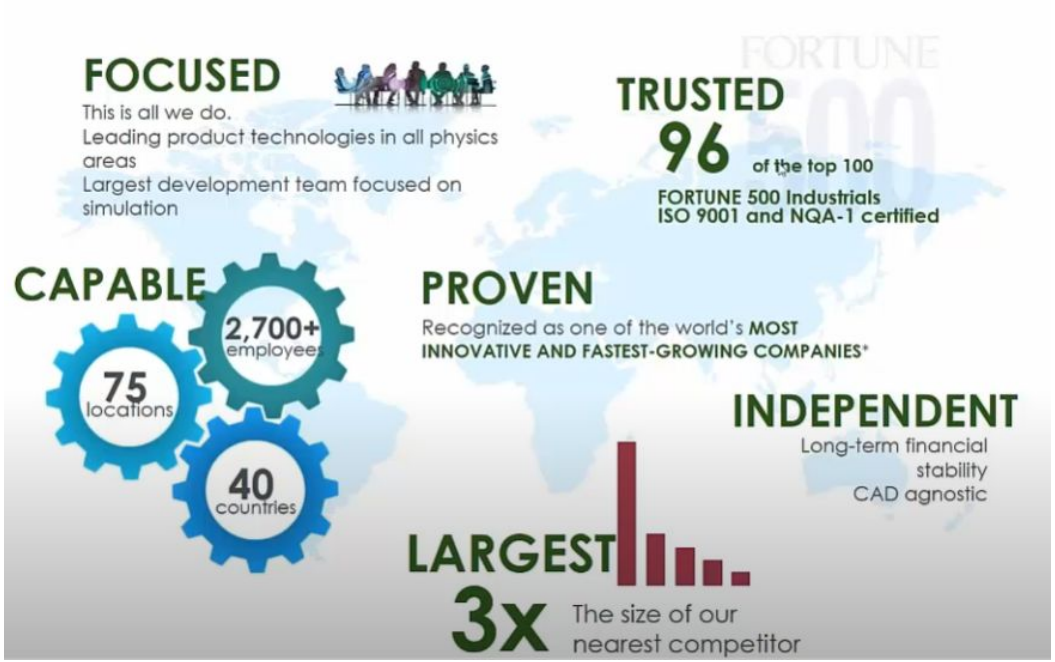
ARES Workshop - Ansys FEA

Cas Kent, Ben van der Veer, Hasan al Banna

Ansys

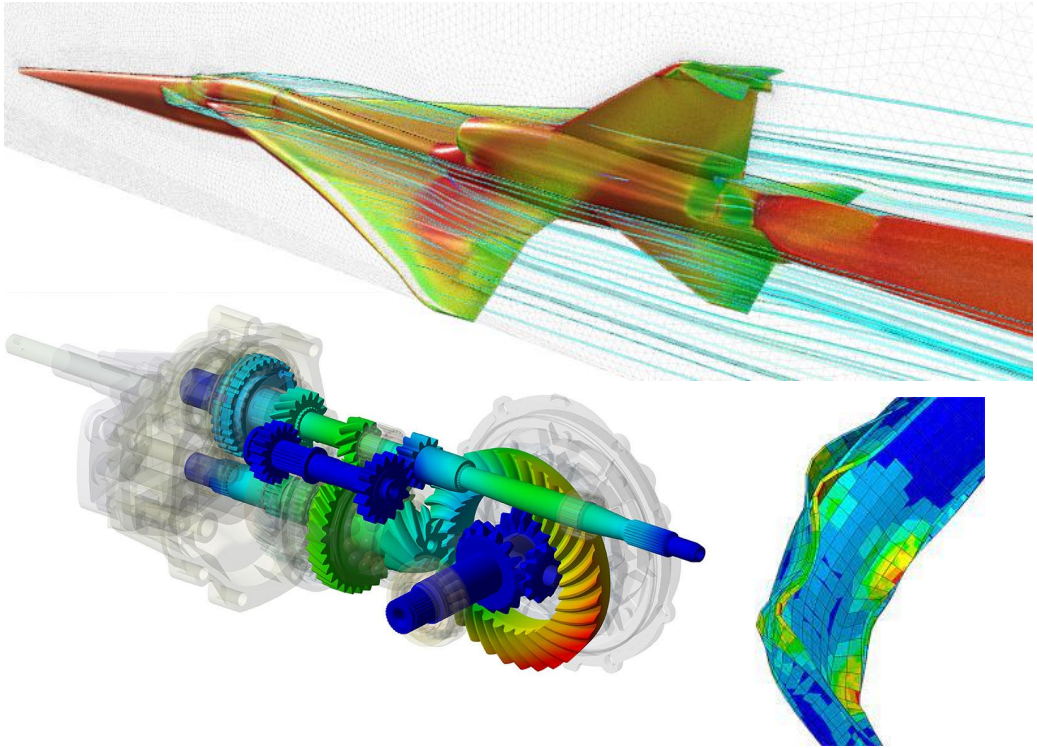
- Industry standard engineering **simulation** software
- Complete package solution
- Global leader in simulation
- Trusted partner of ARES - **LEAP Australia** provide support

leap australia pty ltd



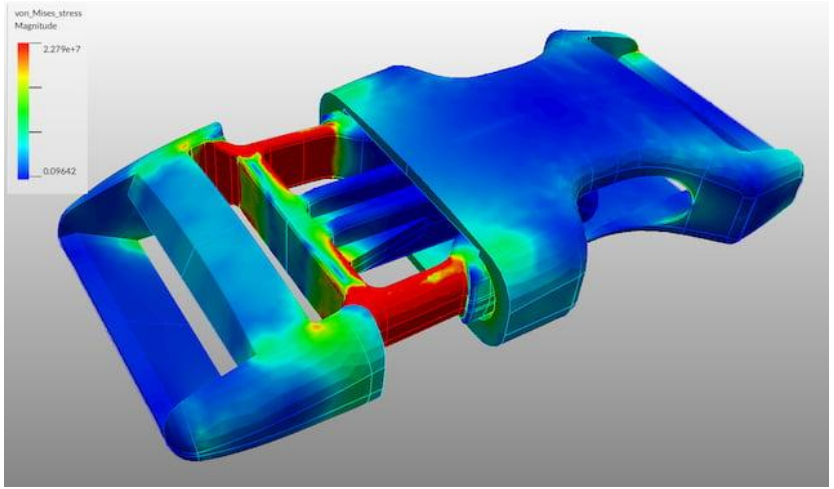
Ansys

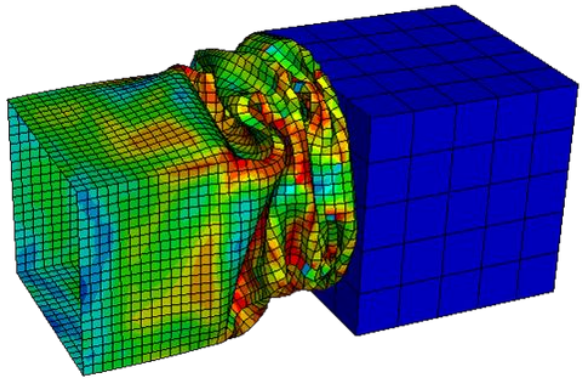
- 3D Design tools (**CAD**)
- Simulation of various physical phenomena, including:
  - Fluid dynamics (**CFD**)
  - Structural mechanics (**FEA**)
  - Thermal
  - Electromagnetic
- Multiphysics simulations
- Model composite materials (carbon fibre)



Why FEA?

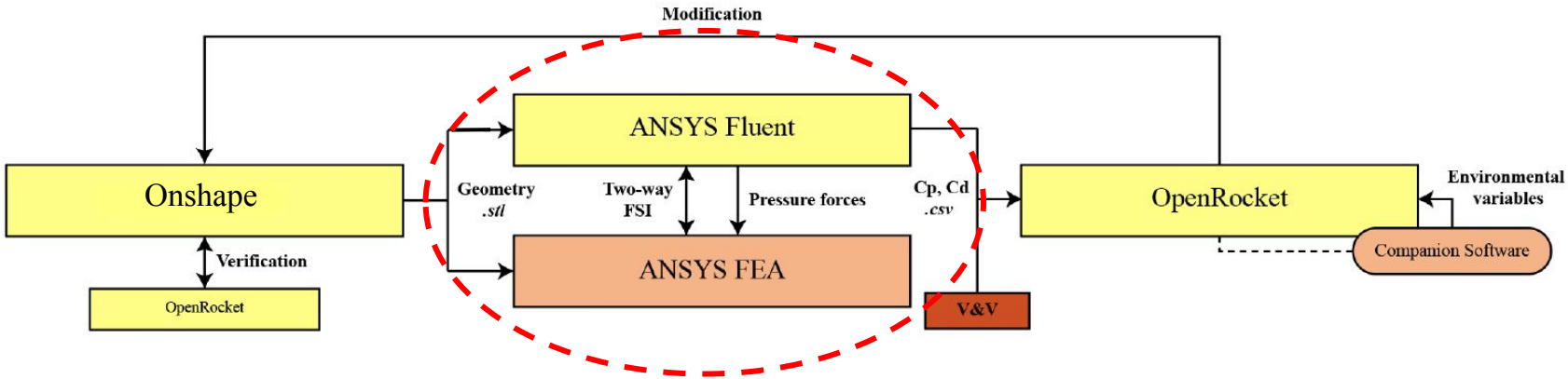
- Numerically solve solid mechanics problems **(avoid a lot of algebra!)**
- Detailed info about **stress & strain distributions,** and deformation of solid bodies
- Can also find **thermal** distributions





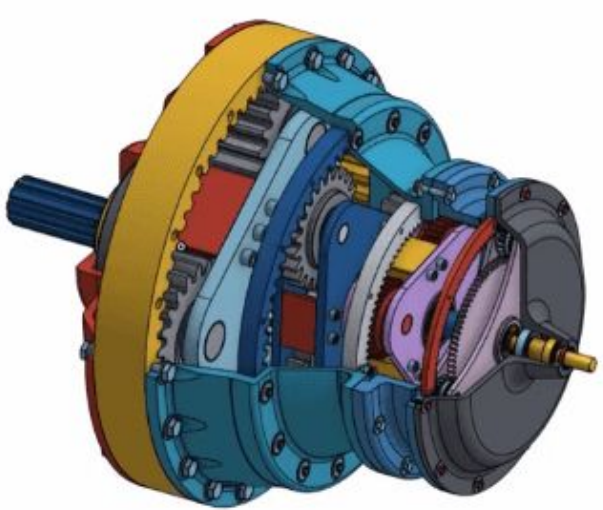
Why FEA?

- **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

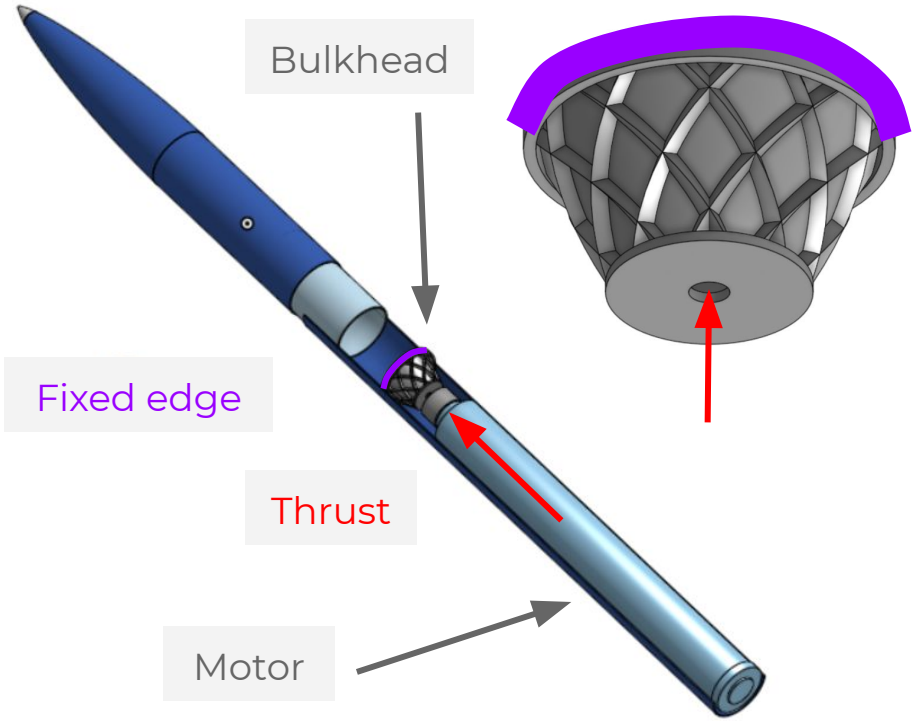
Today's demo

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

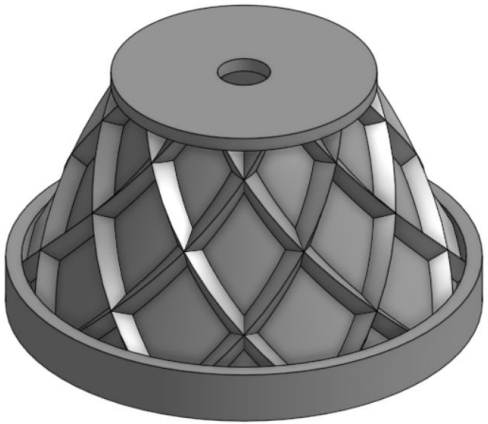
Onshape (full 3D model)
Part Studio

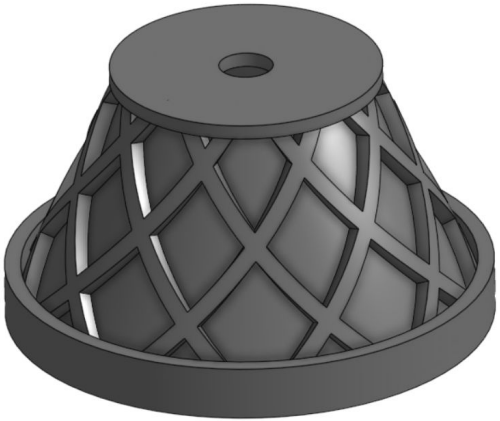
Clean up

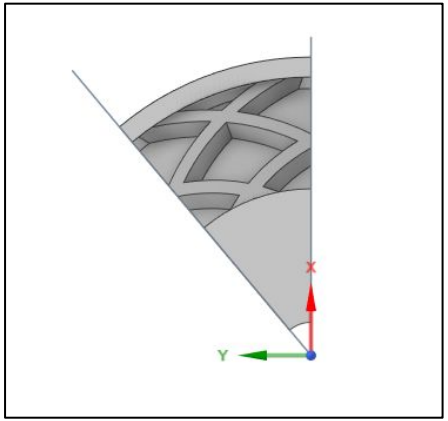
Onshape (simplified)
.sat (ACIS)

Export

SpaceClaim
.scdoc







Design Workflow

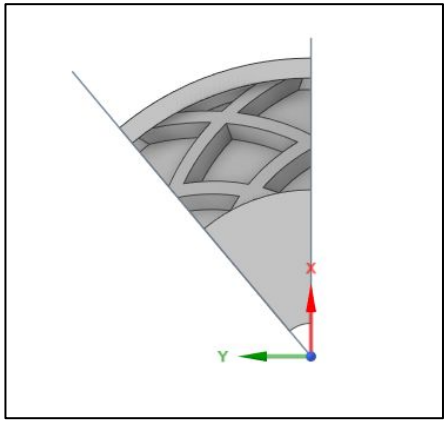
SpaceClaim
.scdoc

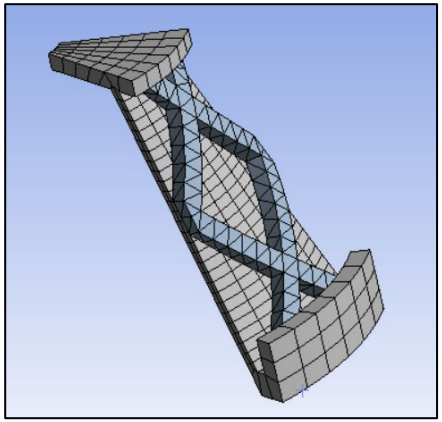
Import to Mechanical

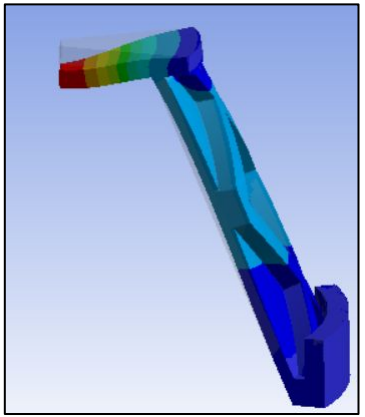
Mechanical (mesh)
.msh

Read mesh

Mechanical (solver)

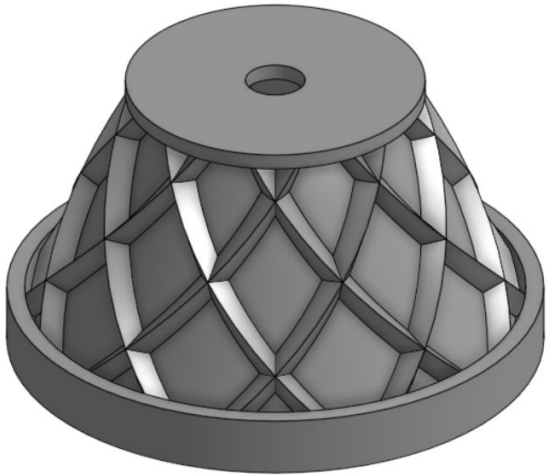


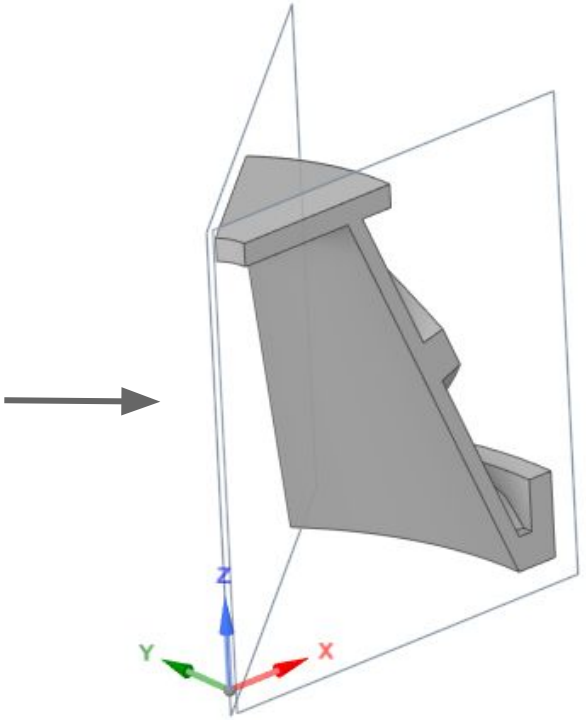




Preparing Geometry

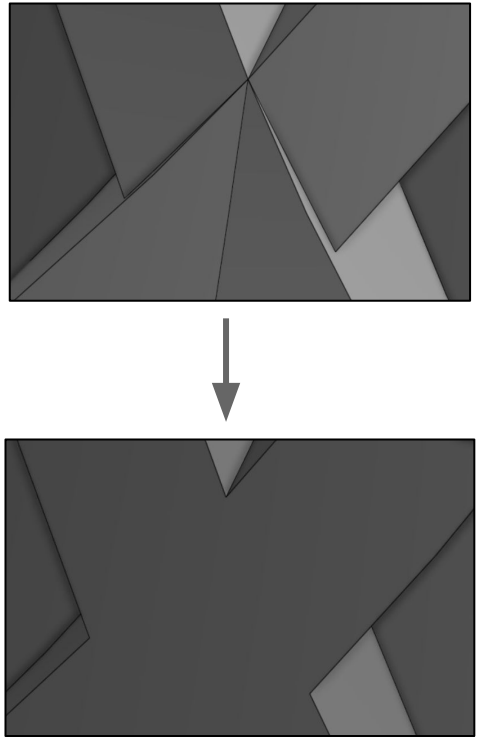
- **FEA can't handle** complex geometry
- Need to **simplify** the geometry for FEA
- Build a **simplified model in Onshape**
- Exporting might cause more issues, so you need to **double check in Ansys**



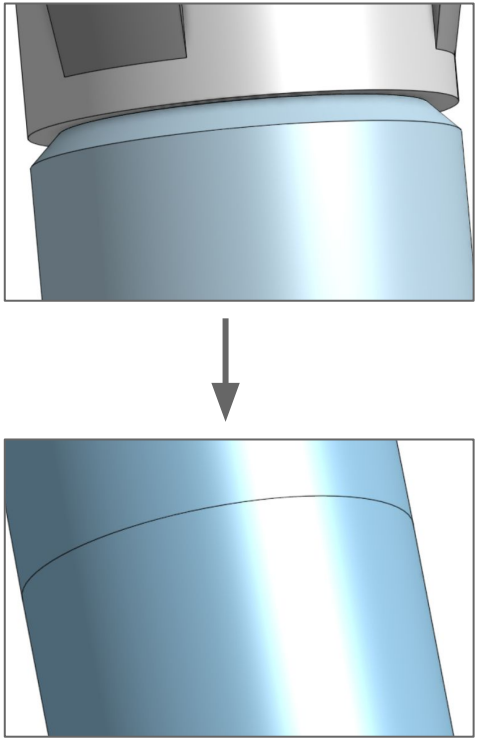


Cleanup - Onshape

Small faces, manufacturing features

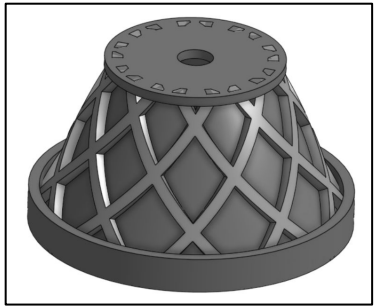


Fillets, chamfers, rounds

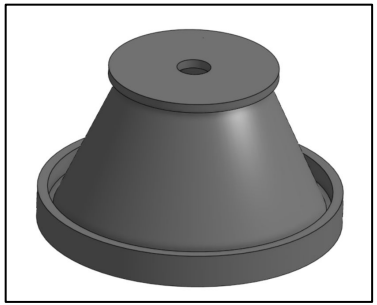


Cleanup - Onshape

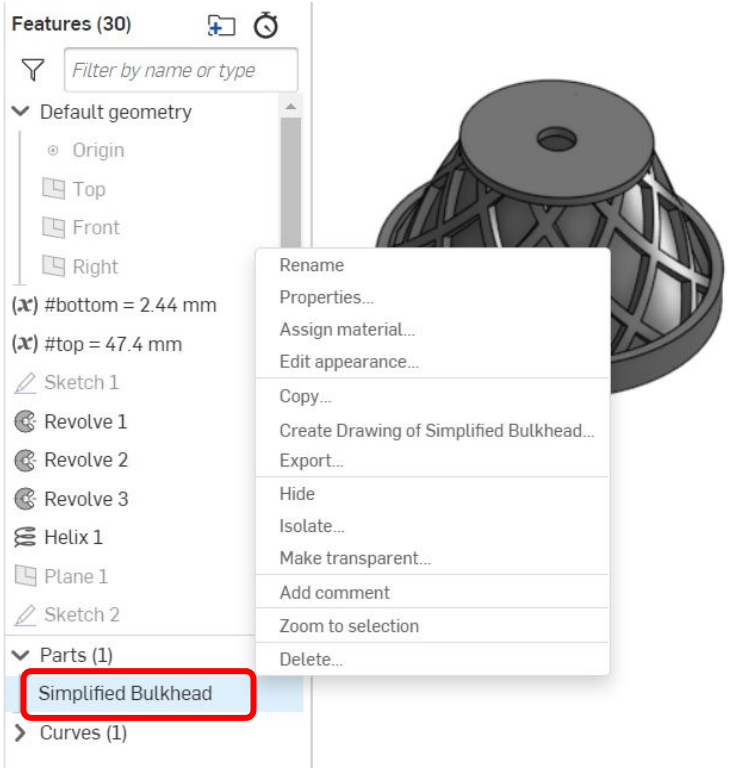
Relevant features?



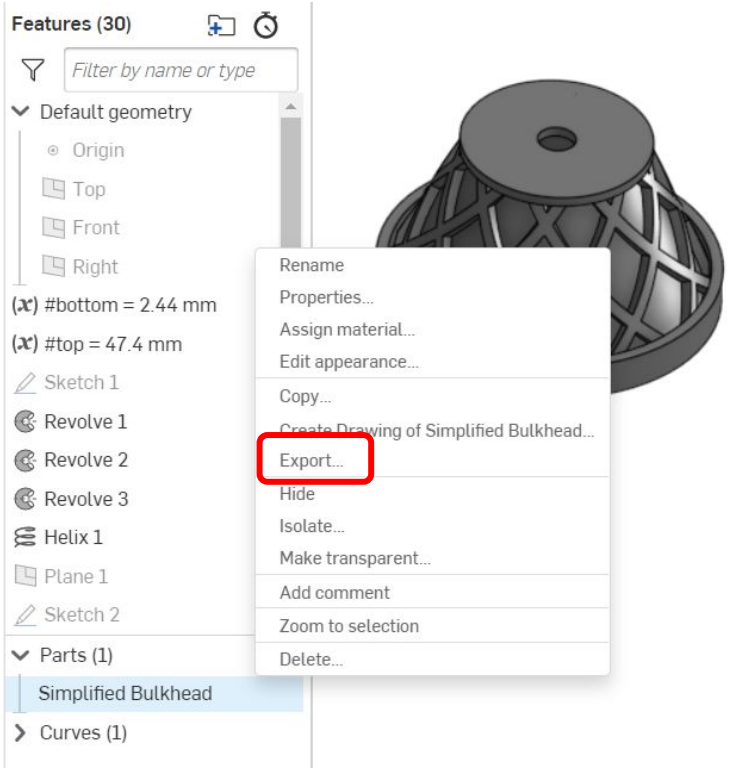
Will this compromise strength?



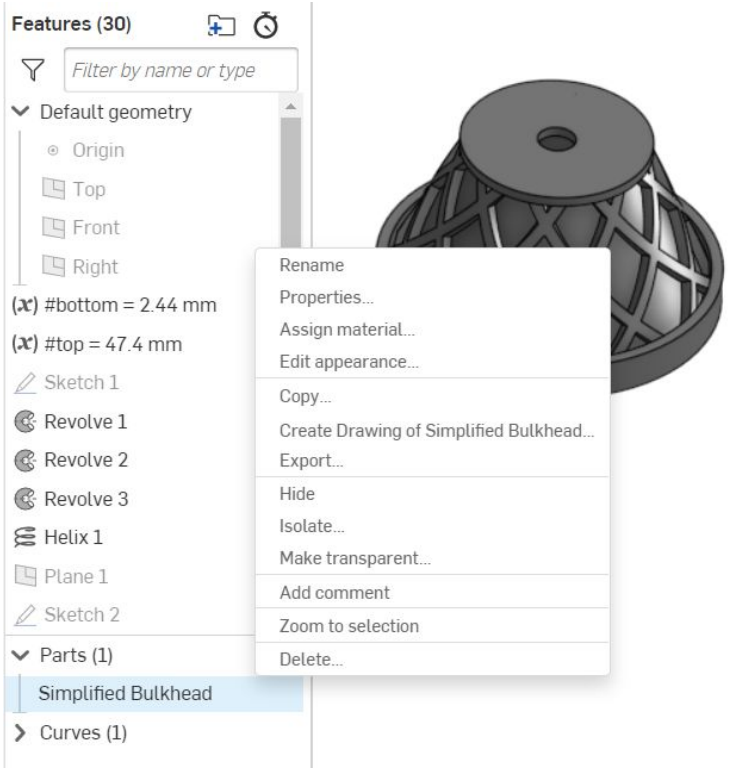
Export Onshape

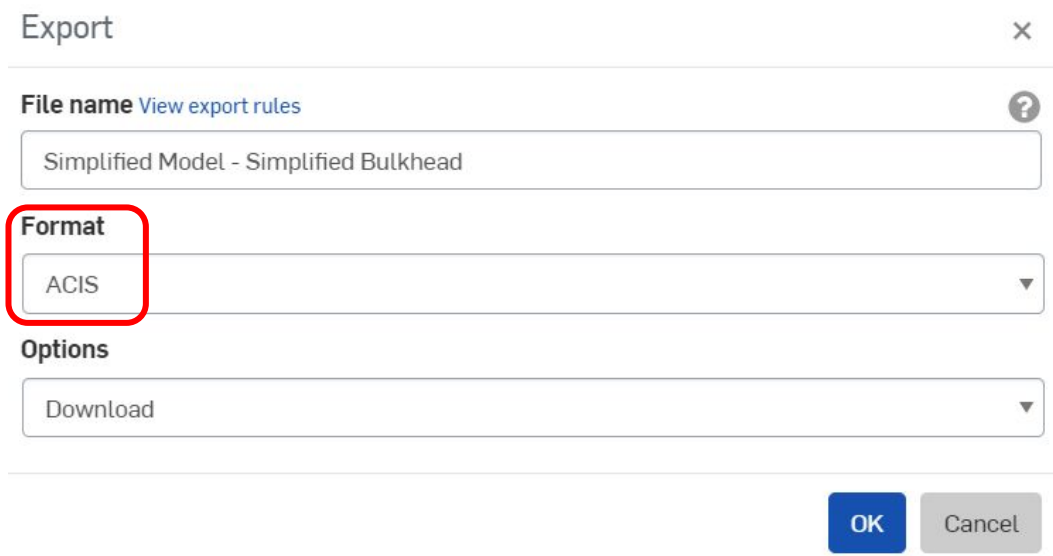


Export Onshape

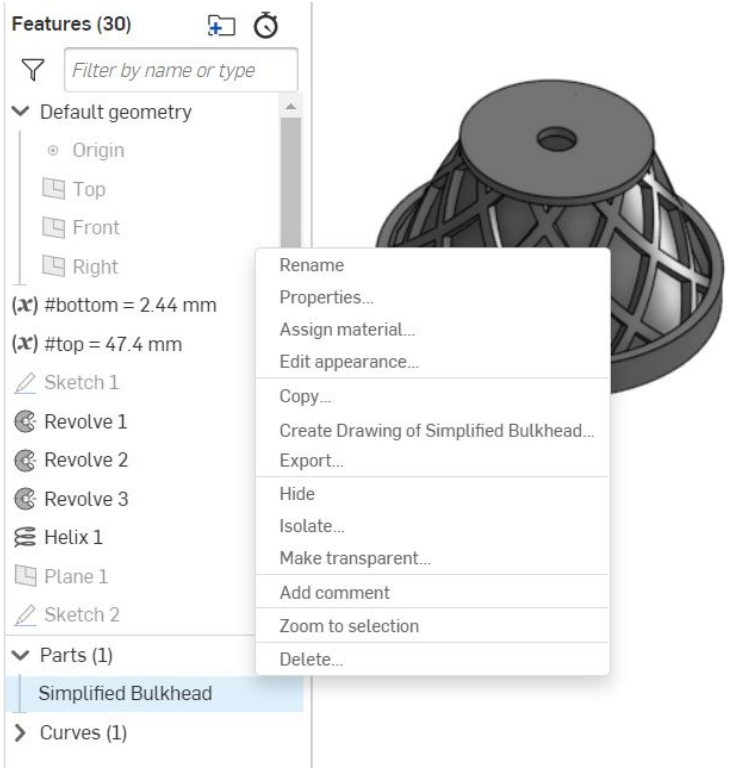


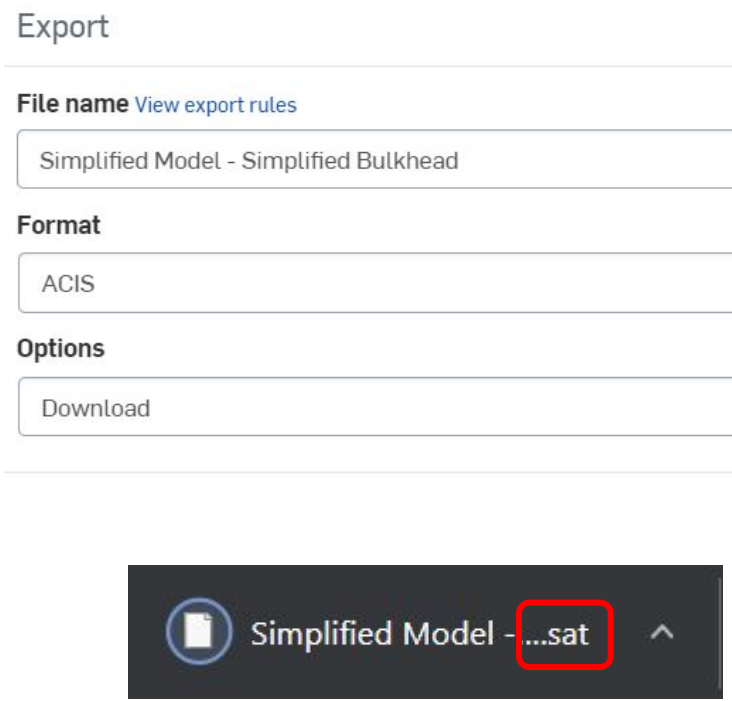
Export Onshape





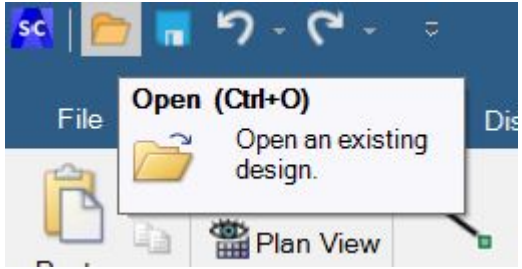
Export Onshape



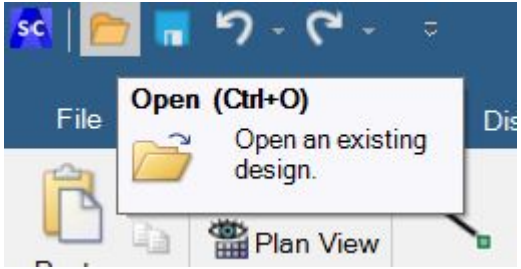


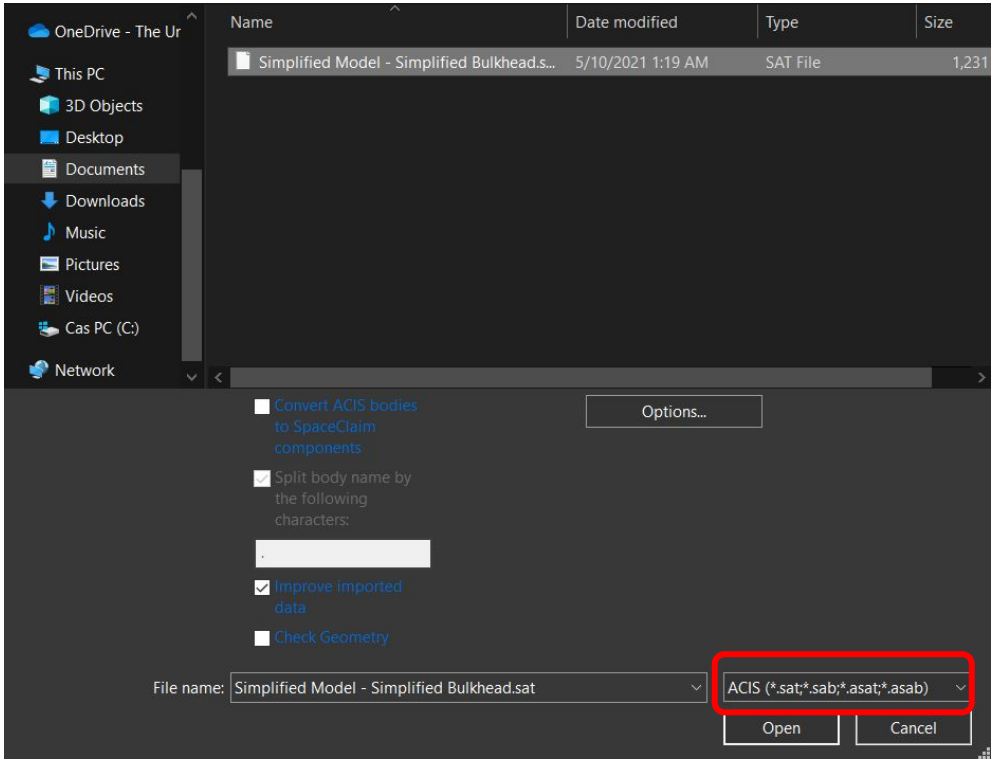
Export Onshape to SpaceClaim

Export Onshape to SpaceClaim

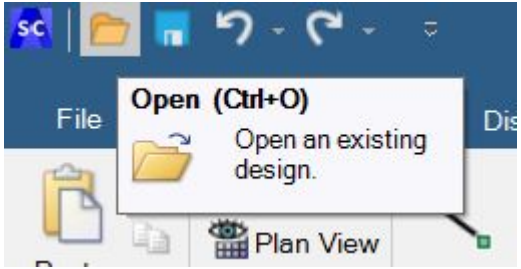


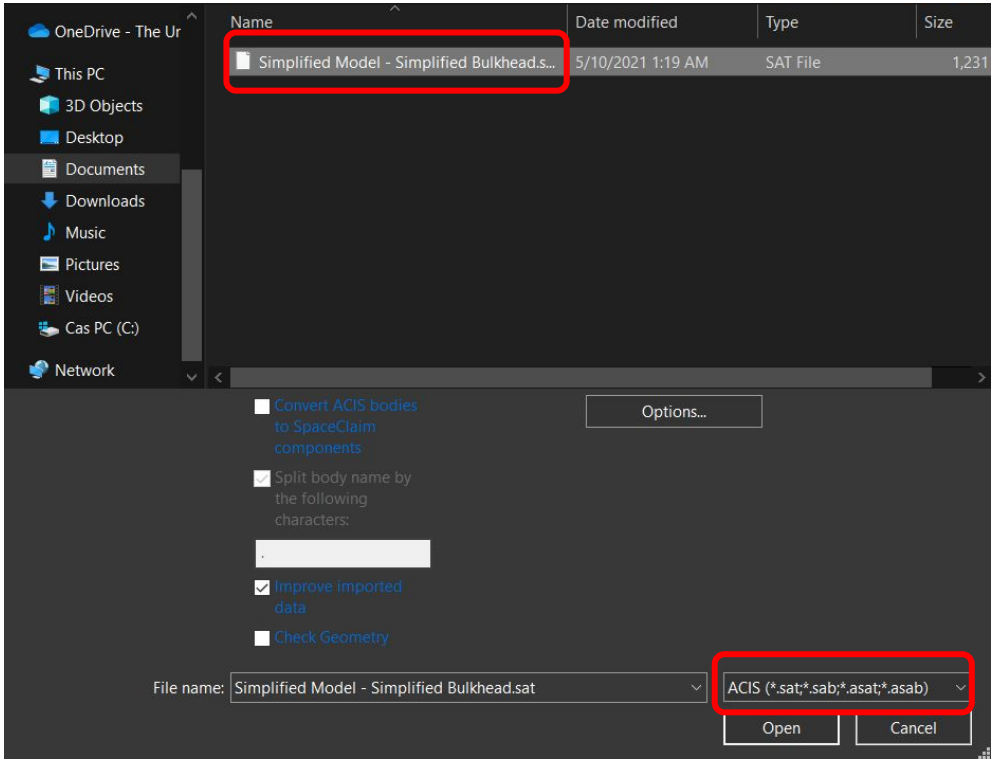
Export Onshape to SpaceClaim



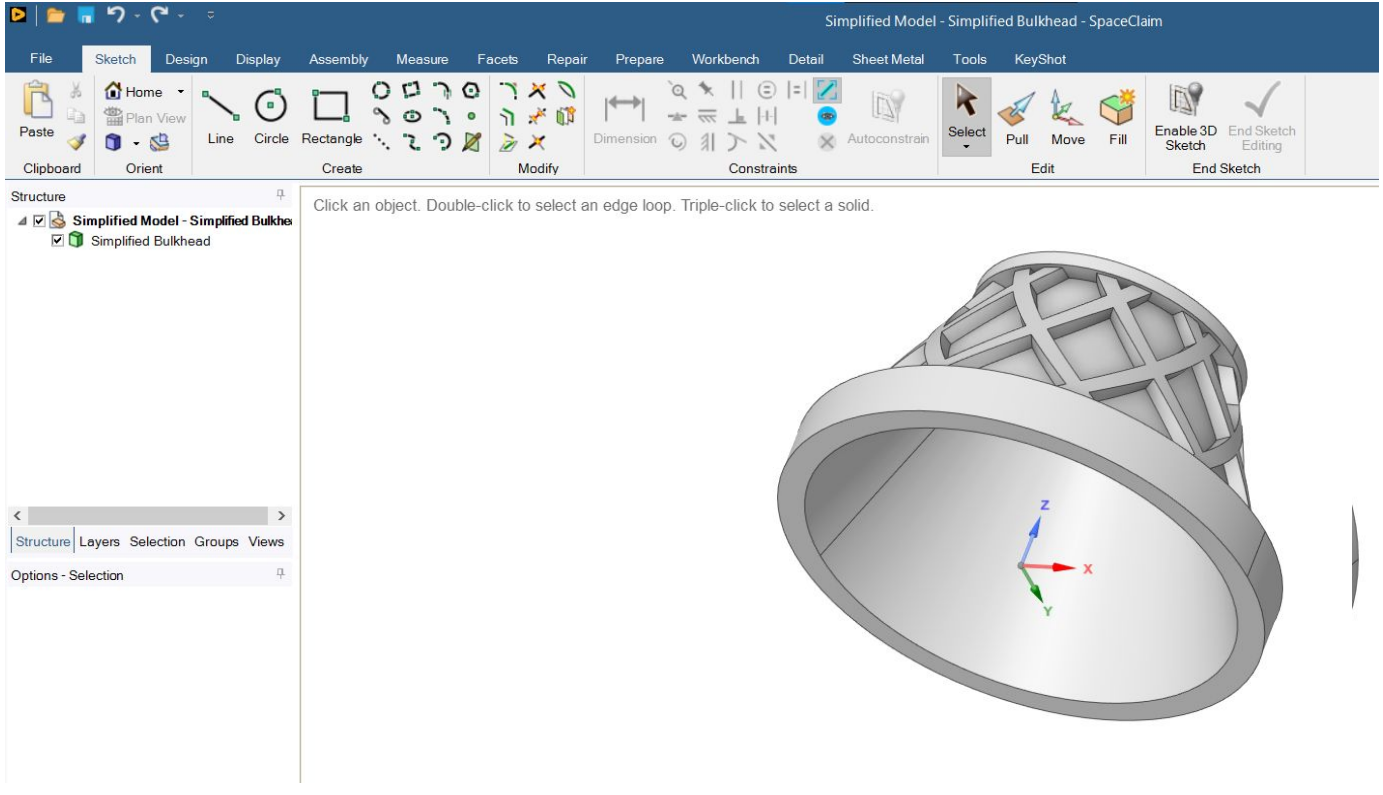


Export Onshape to SpaceClaim

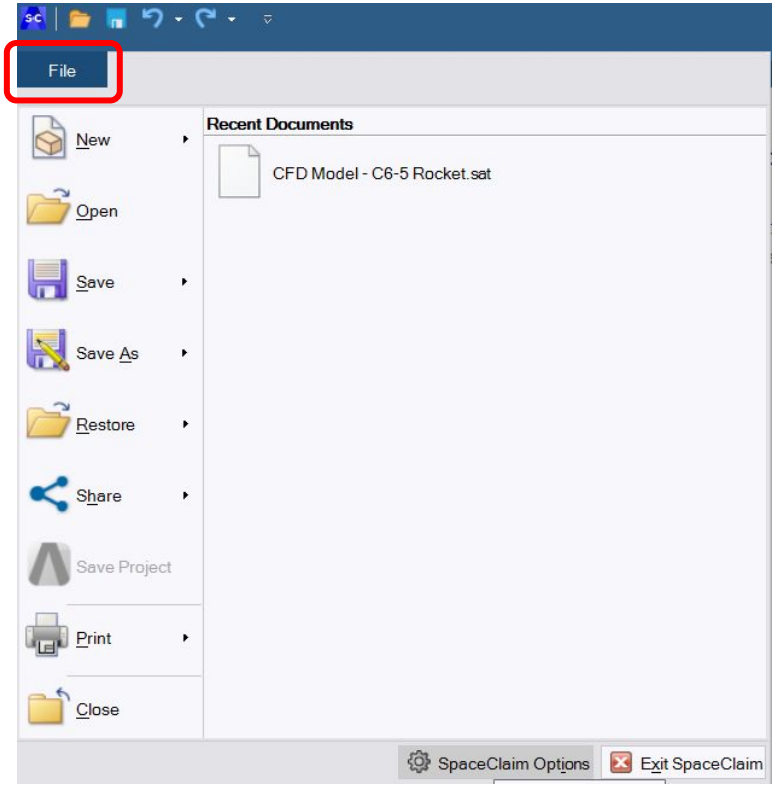




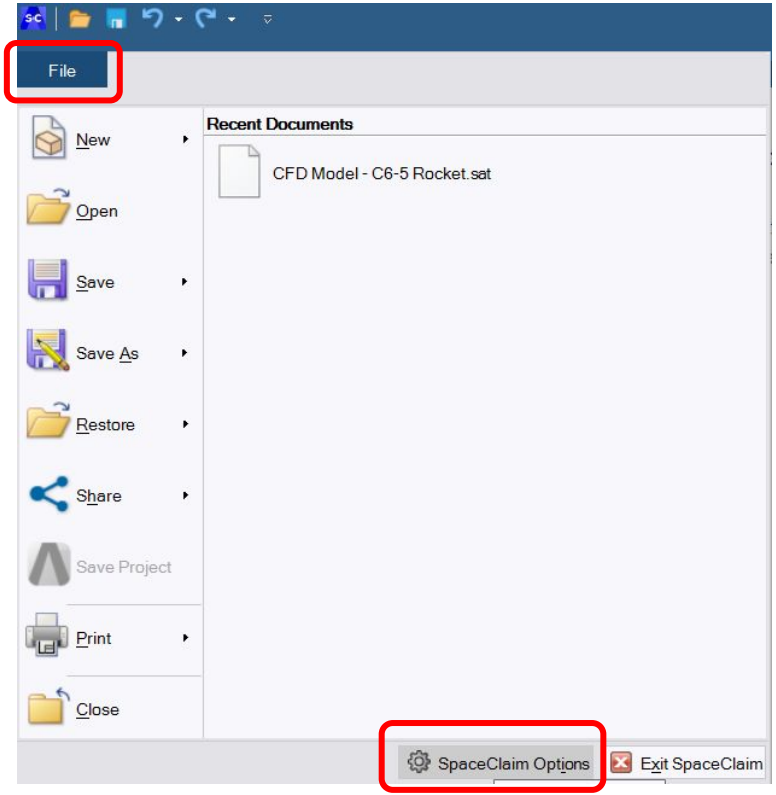
Export Onshape to SpaceClaim



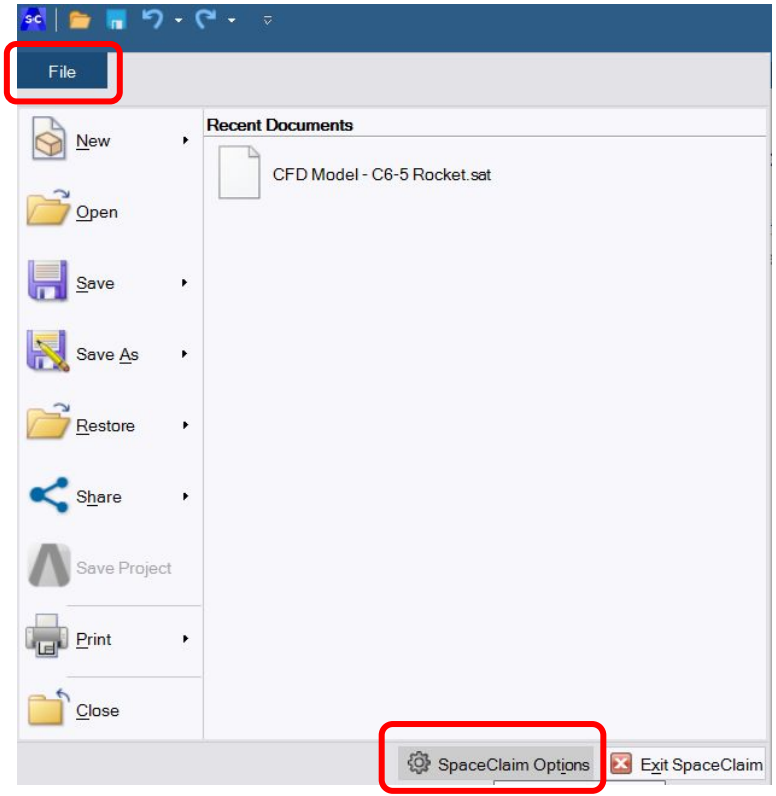
SpaceClaim Mouse Controls

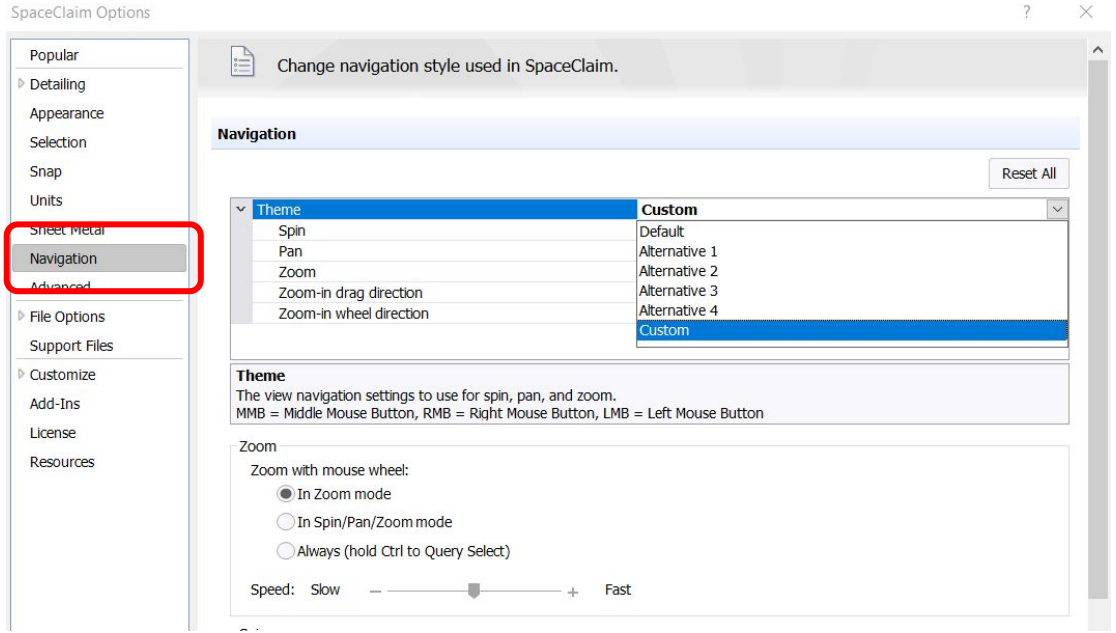


SpaceClaim Mouse Controls

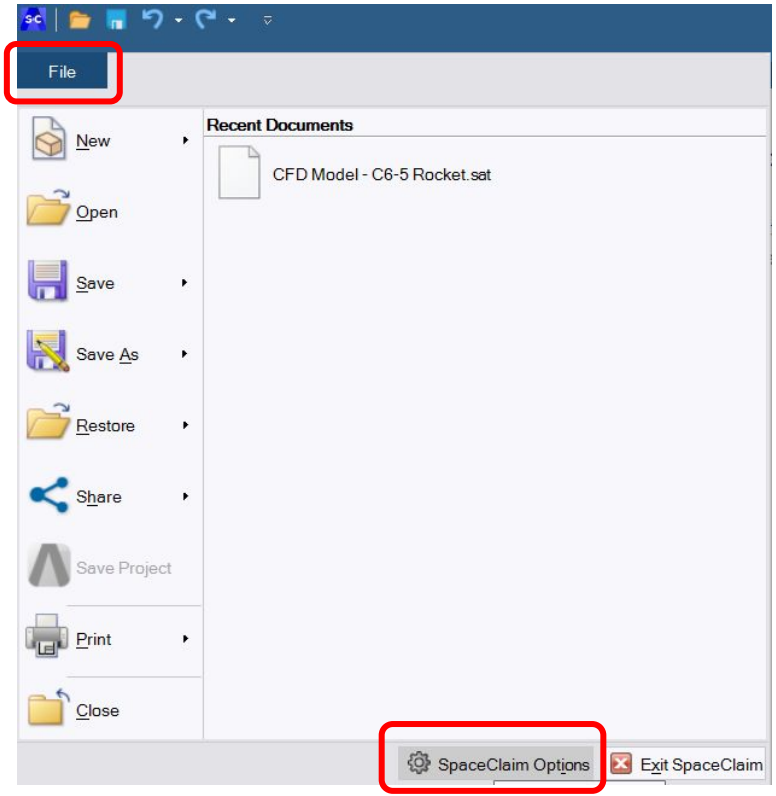


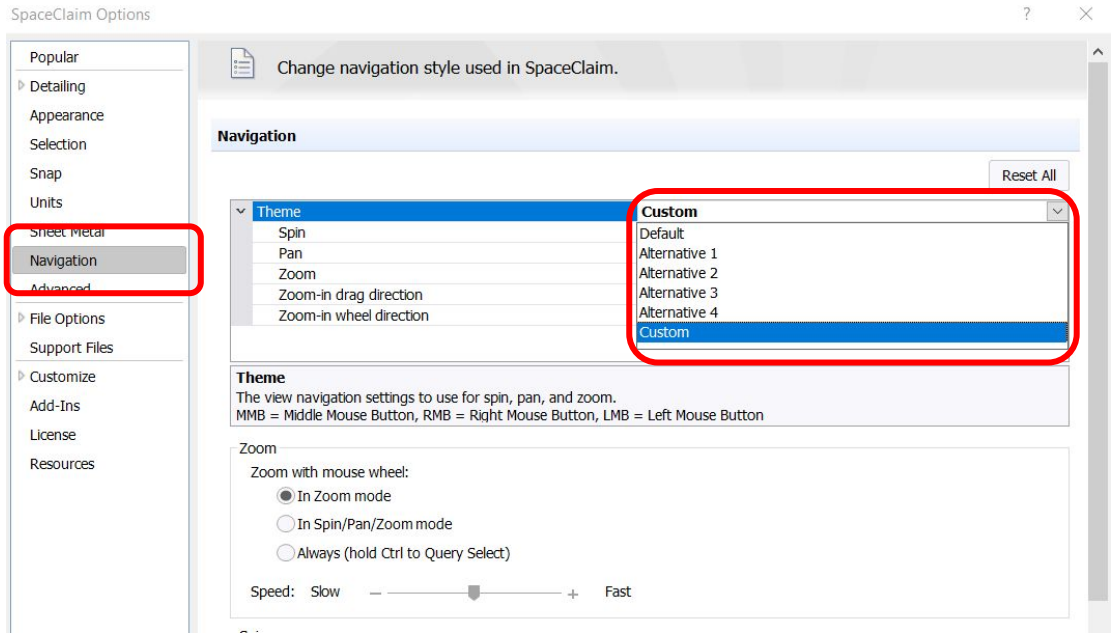
SpaceClaim Mouse Controls



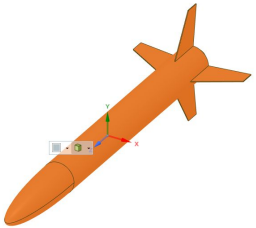


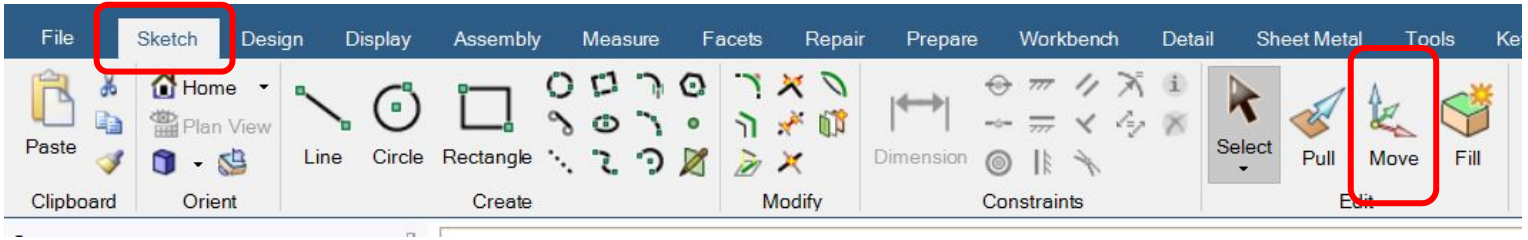
SpaceClaim Mouse Controls





Fix Origin and Orientation

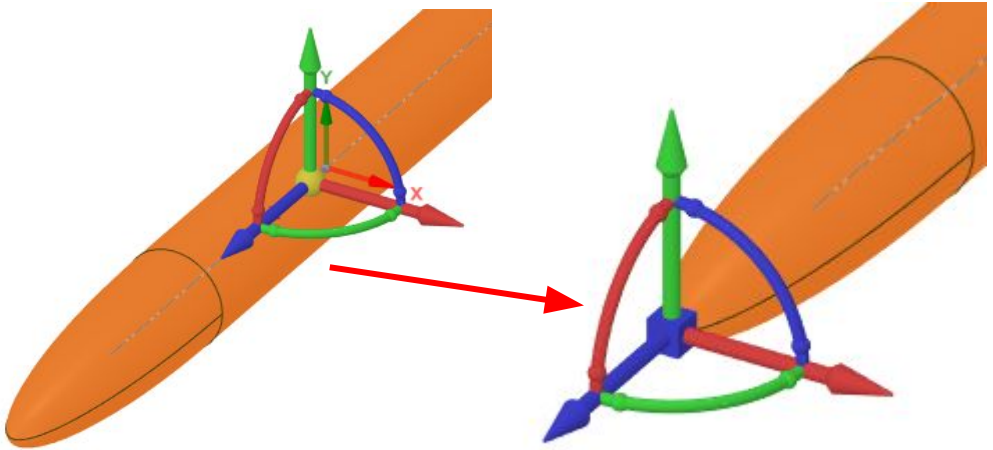




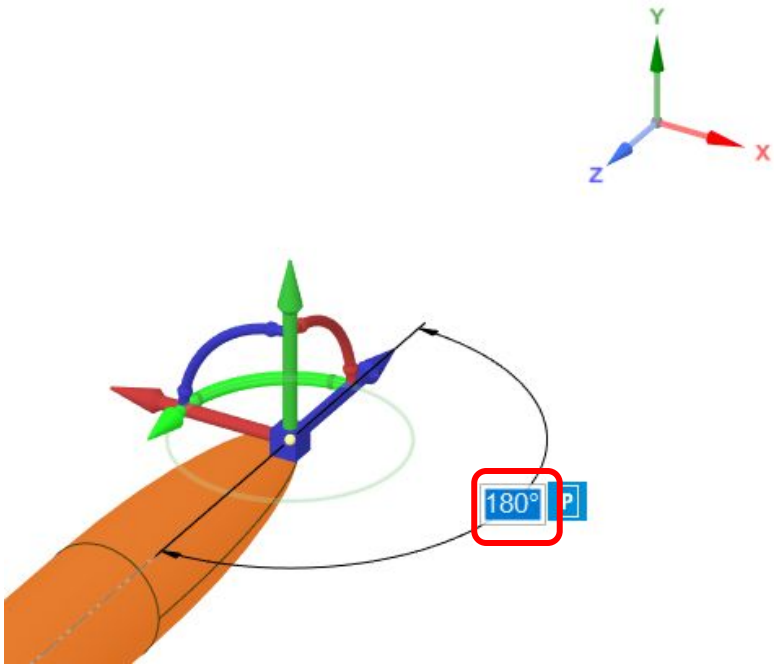
Want gravity (***z***) pointing down the rocket

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

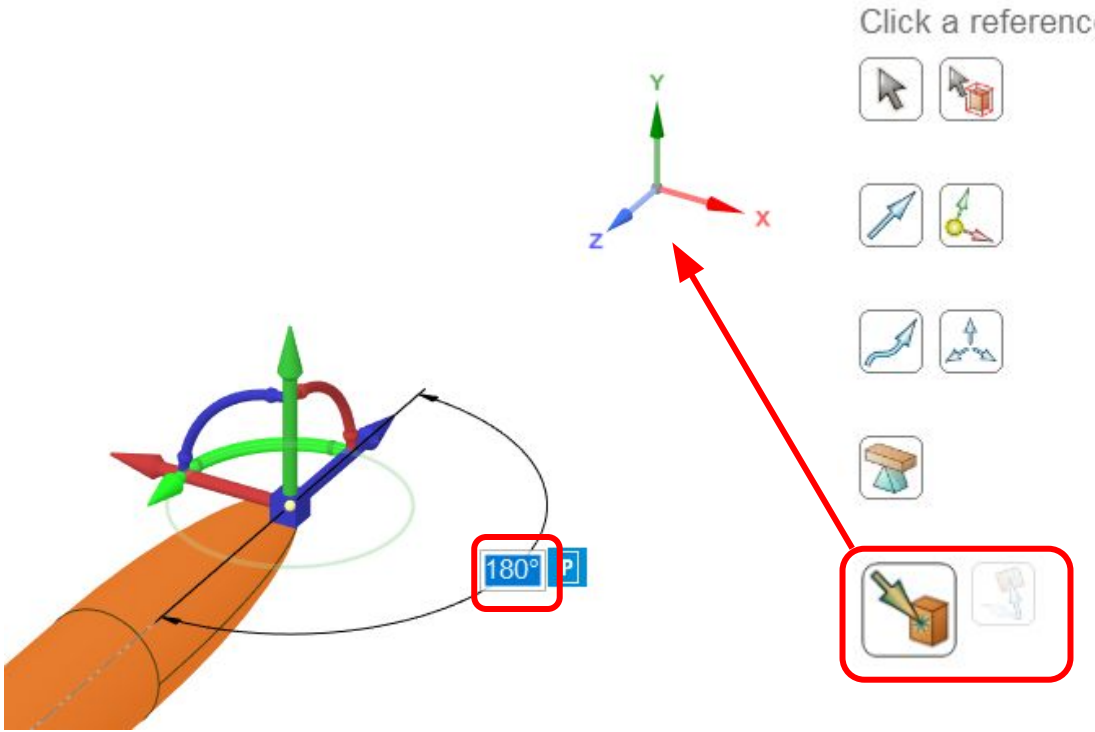
Triple click to select the entire rocket



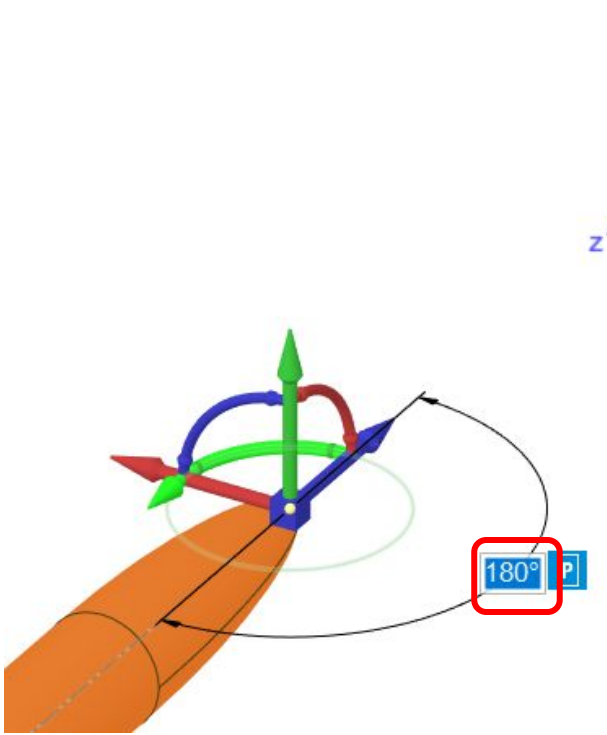
Fix Origin and Orientation

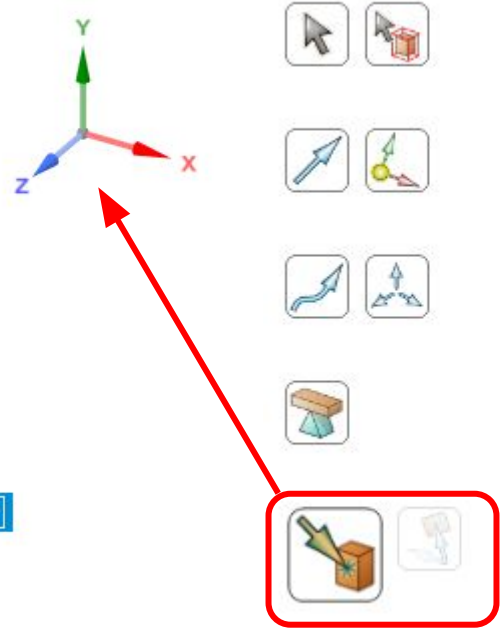


Fix Origin and Orientation

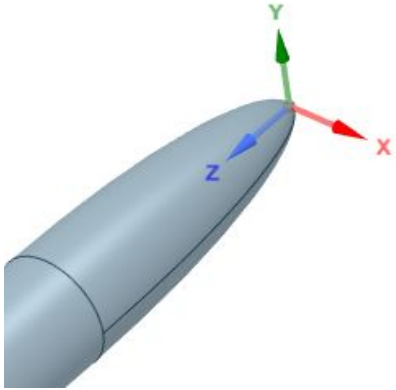


Fix Origin and Orientation

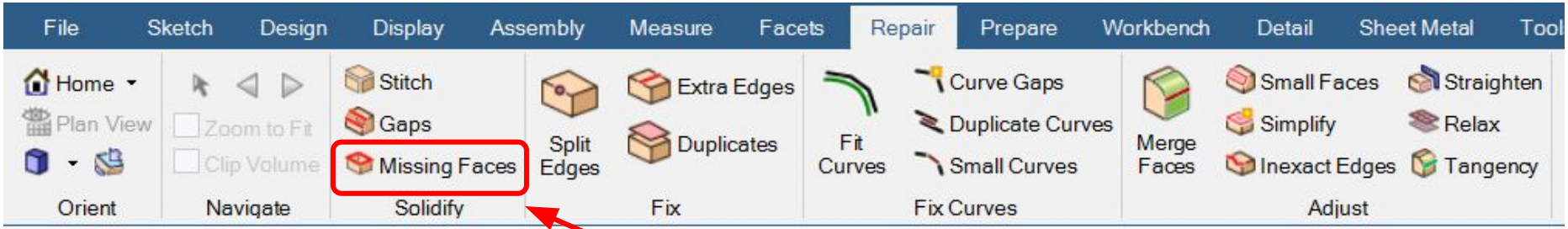




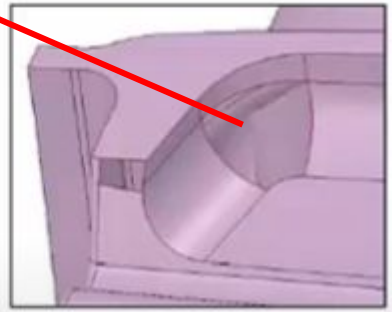
Press **esc** to leave the Move Tool



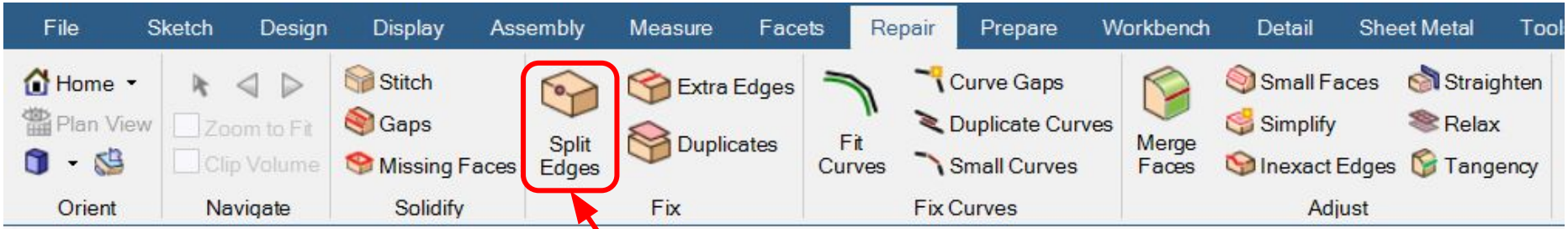
Cleanup - SpaceClaim



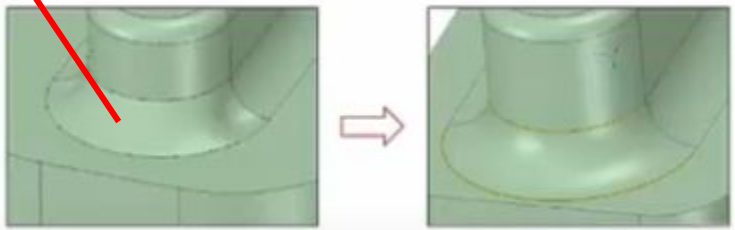
Missing faces



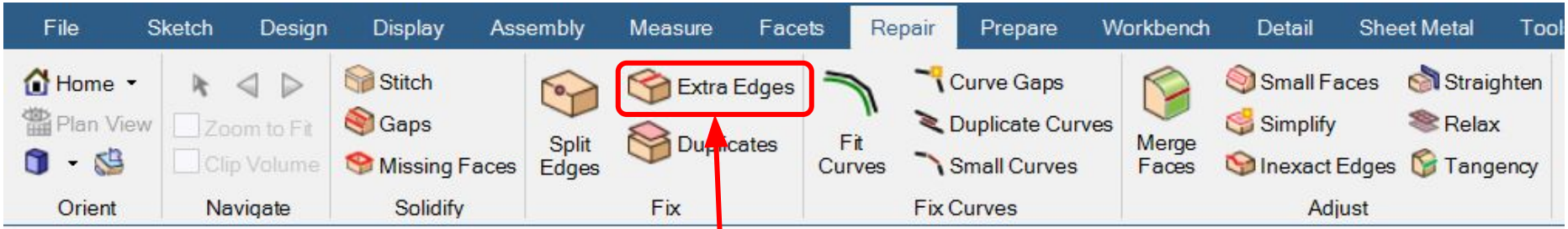
Cleanup - SpaceClaim

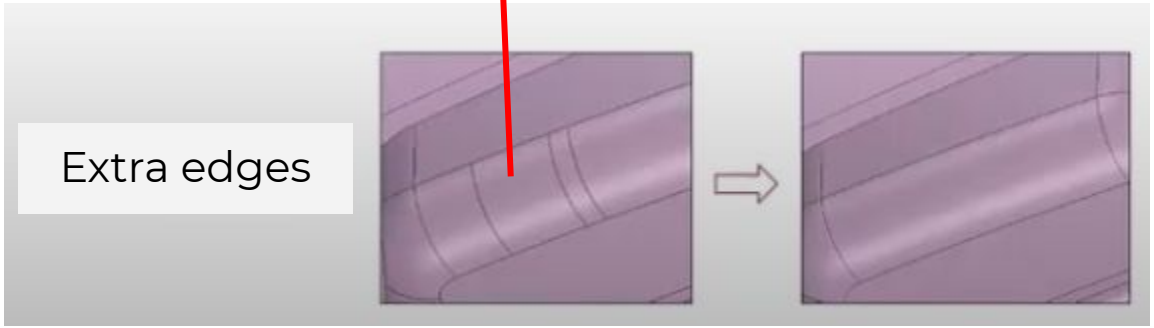


Split edges

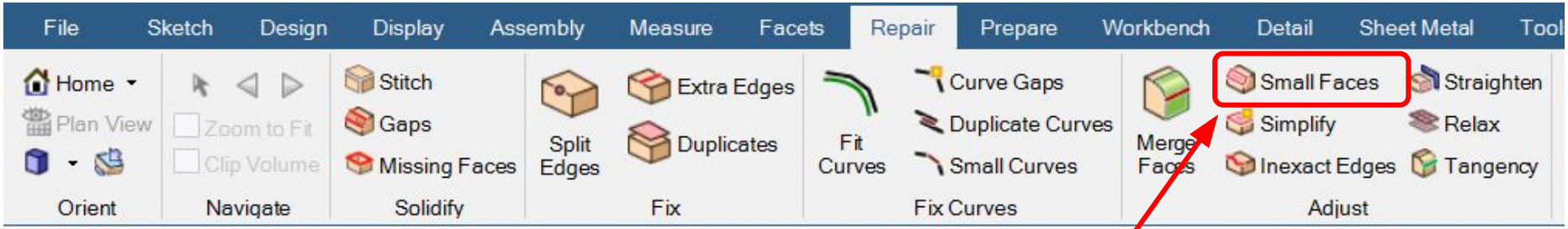


Cleanup - SpaceClaim



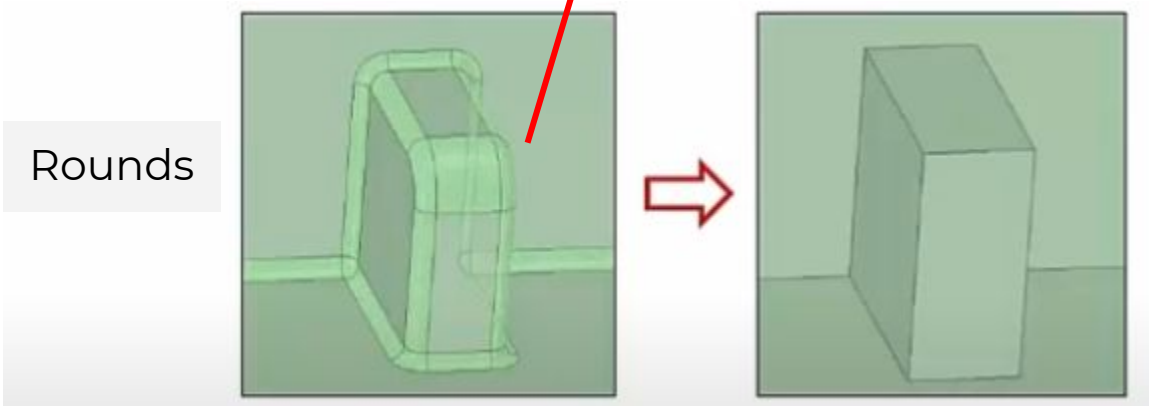


Cleanup - SpaceClaim

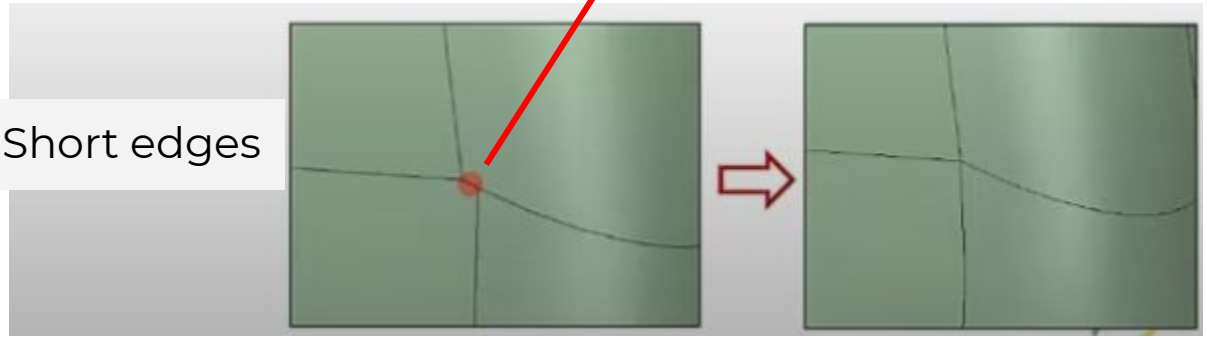


Cleanup - SpaceClaim

Rounds

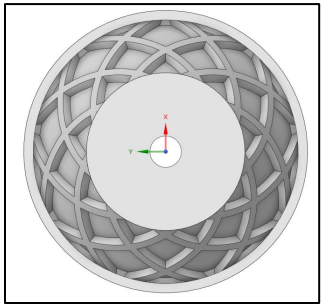


Cleanup - SpaceClaim



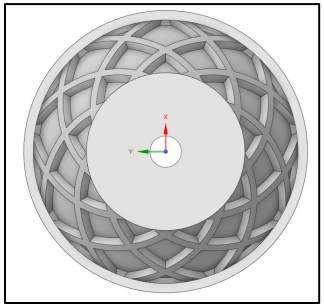
Cleanup - SpaceClaim

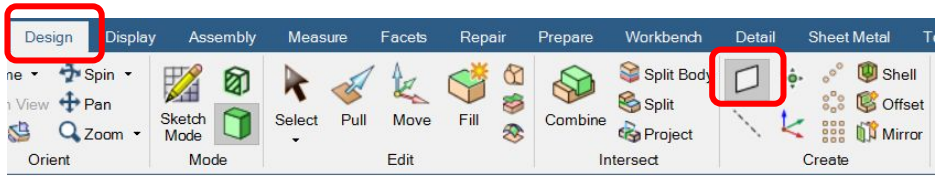
Symmetric object + symmetric **loading condition**?



Cleanup - SpaceClaim

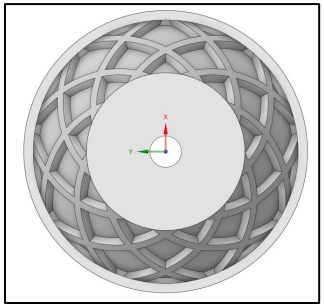
Symmetric object
+ symmetric
loading condition?

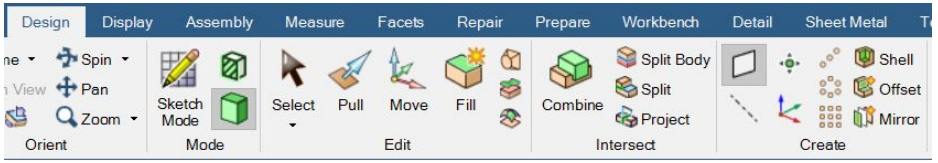


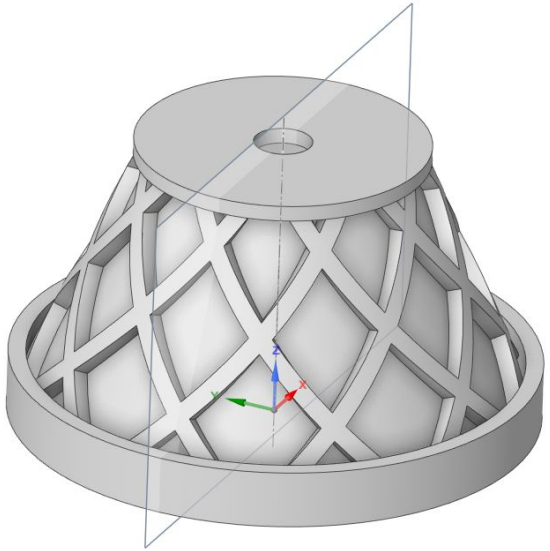


Cleanup - SpaceClaim

Symmetric object
+ symmetric
loading condition?

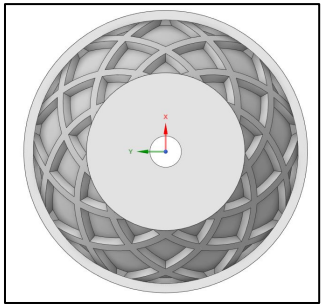


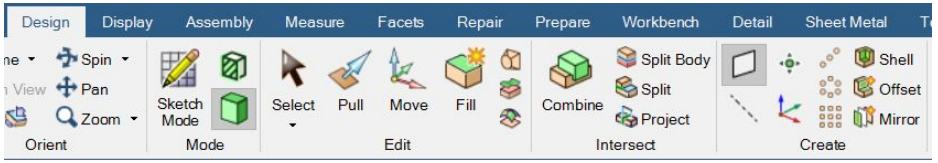


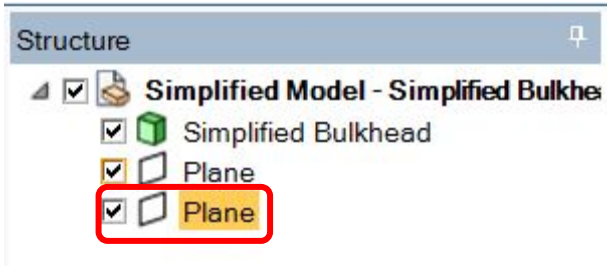


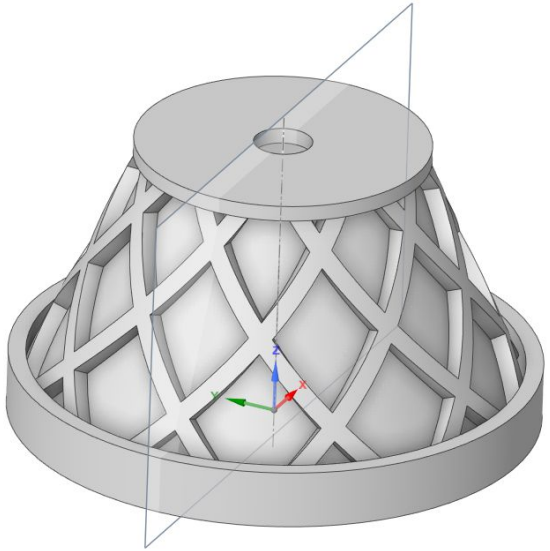
Cleanup - SpaceClaim

Symmetric object
+ symmetric
loading condition?



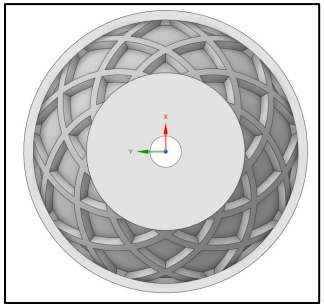


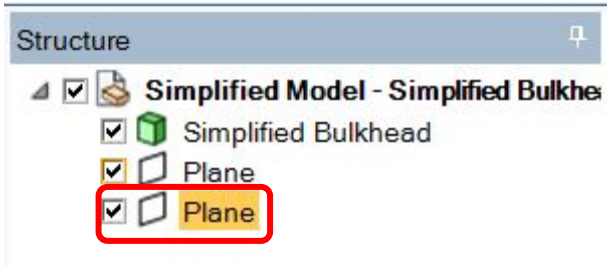


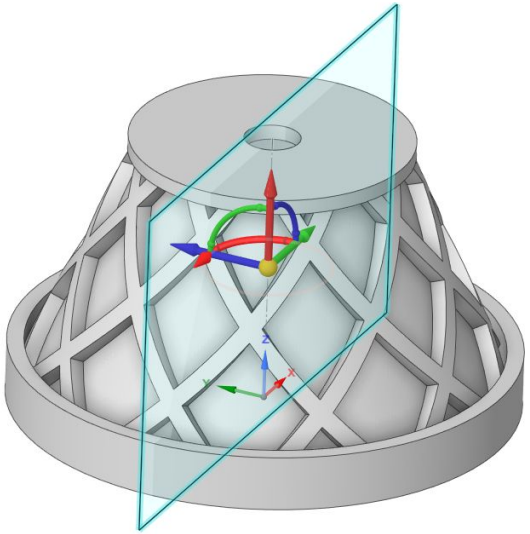


Cleanup - SpaceClaim

Symmetric object
+ symmetric
loading condition?

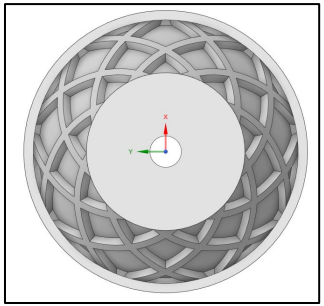


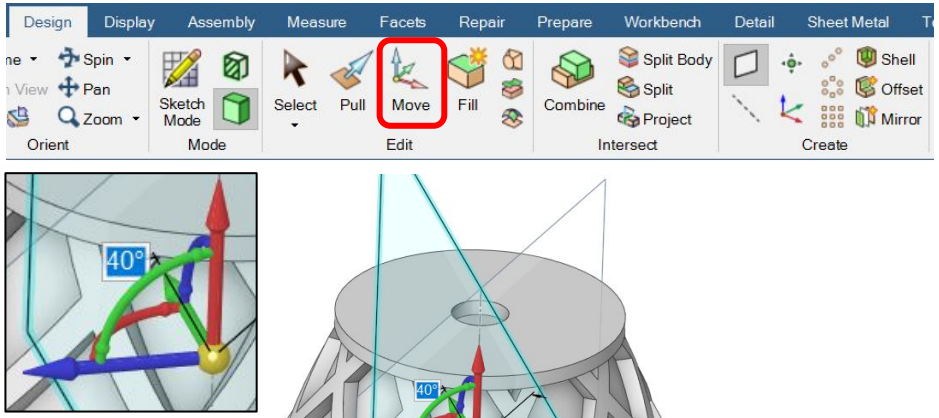


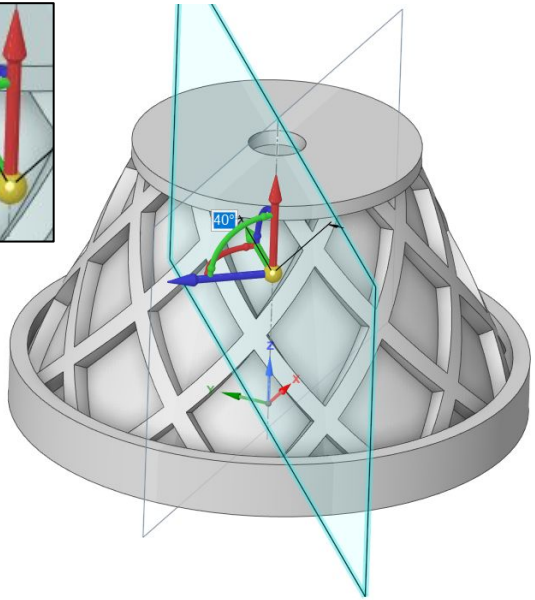


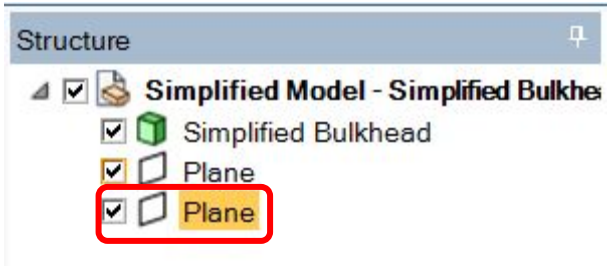
Cleanup - SpaceClaim

Symmetric object
+ symmetric
loading condition?



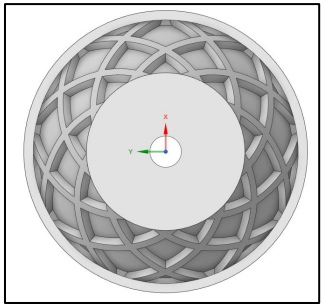


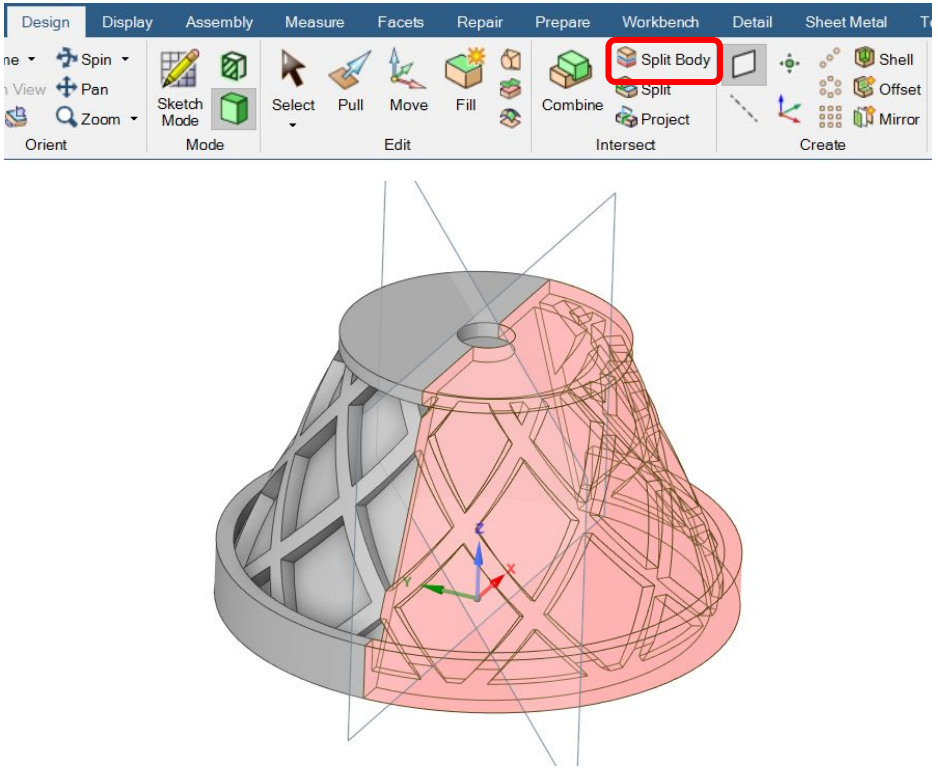




Cleanup - SpaceClaim

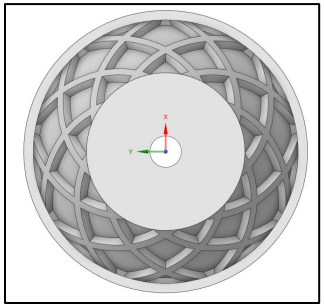
Symmetric object
+ symmetric
loading condition?

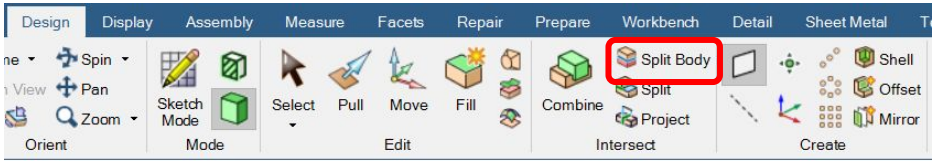


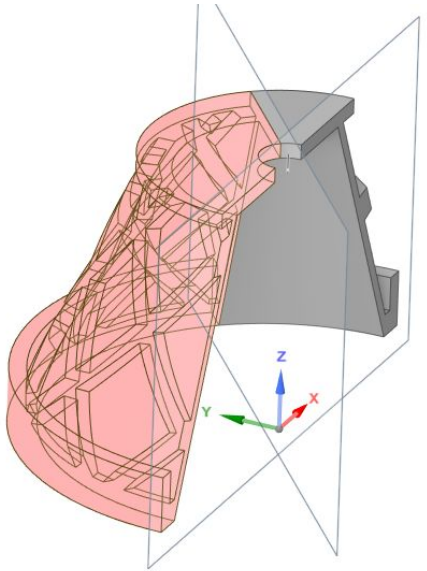


Cleanup - SpaceClaim

Symmetric object
+ symmetric
loading condition?

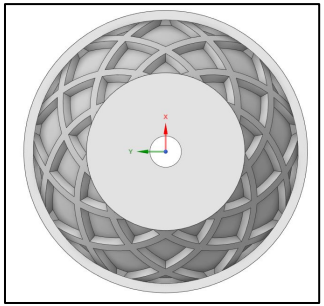


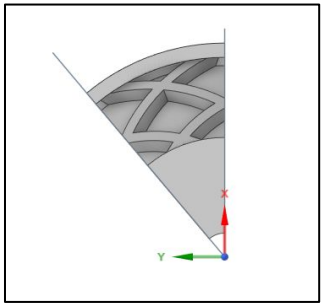


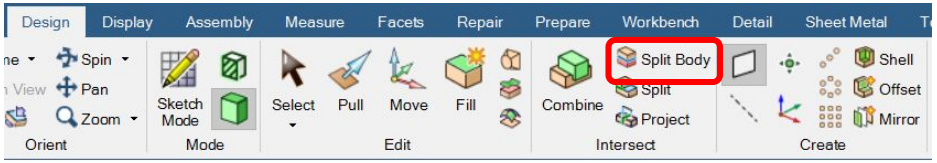


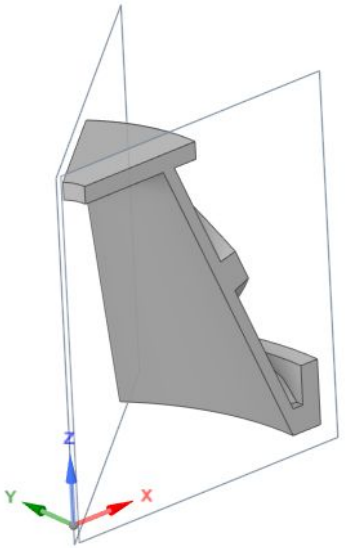
Cleanup - SpaceClaim

Symmetric object
+ symmetric
loading condition?

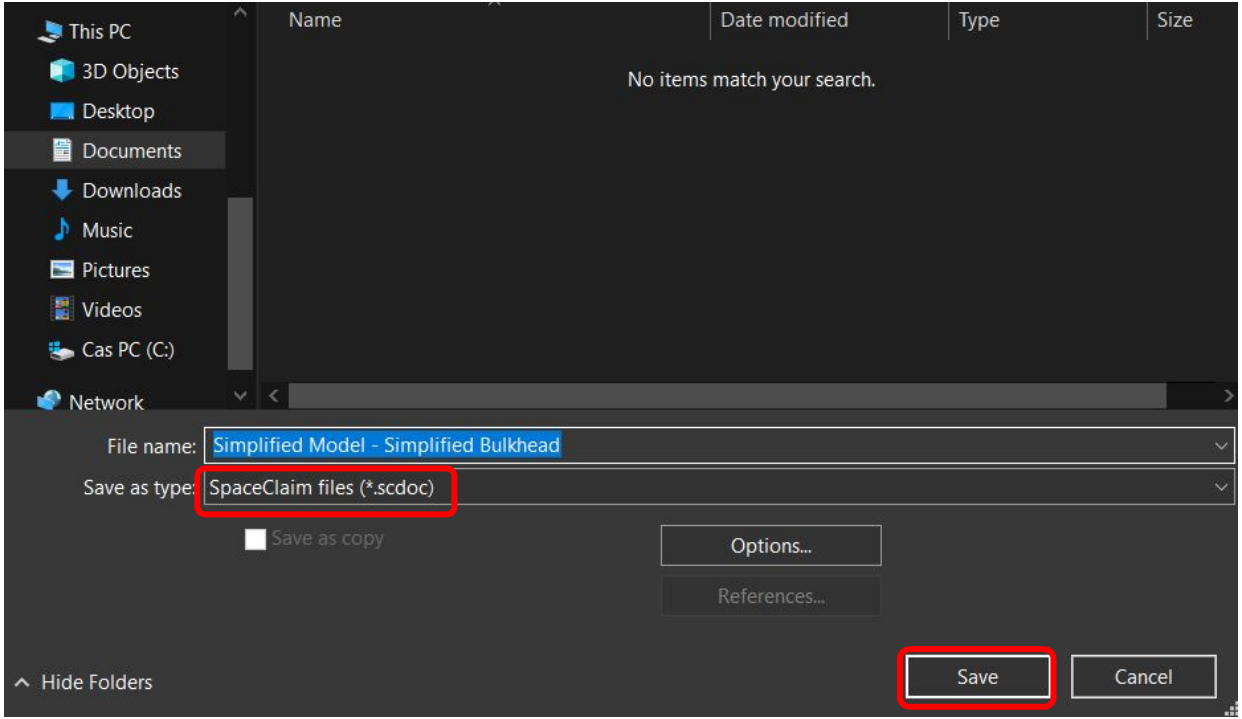


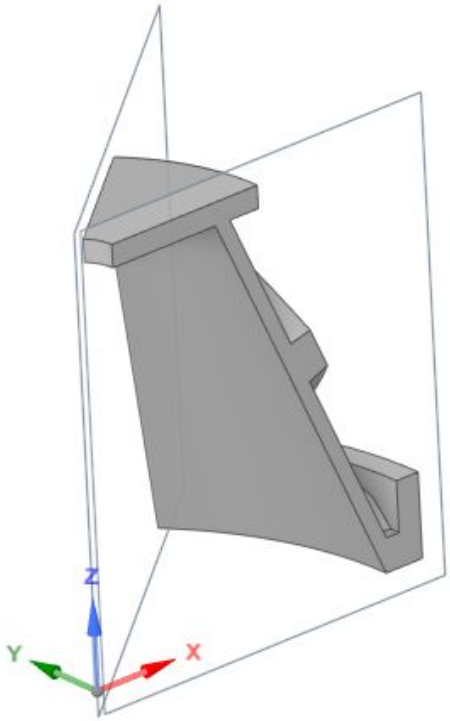




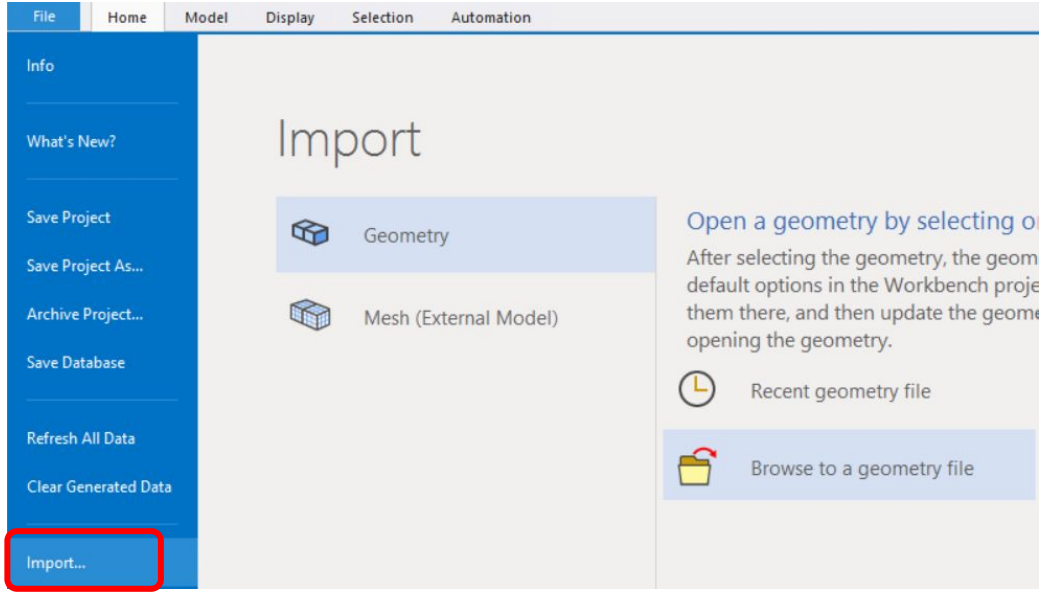


Export to Mechanical

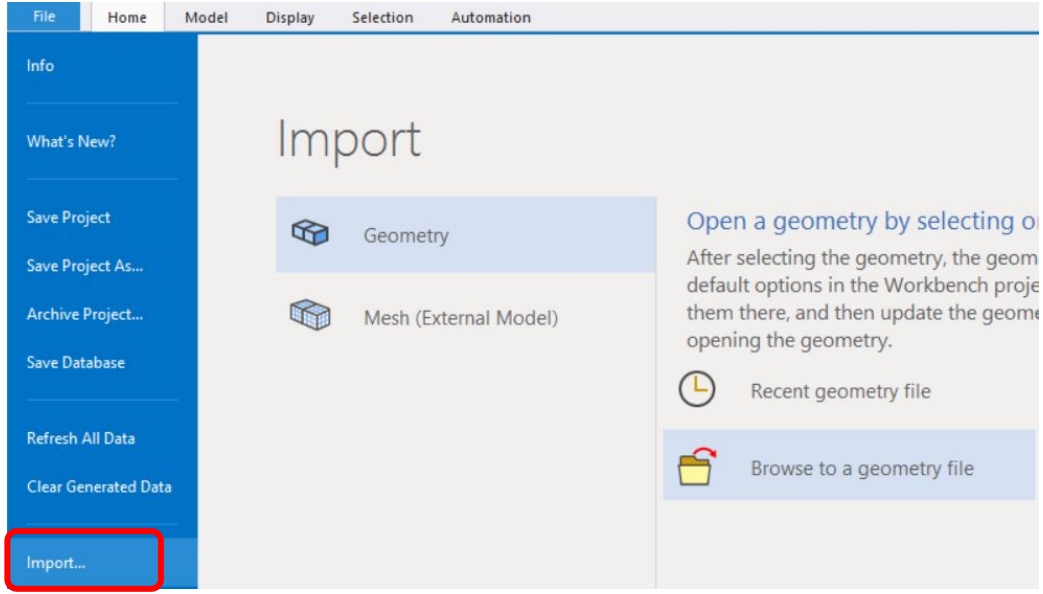


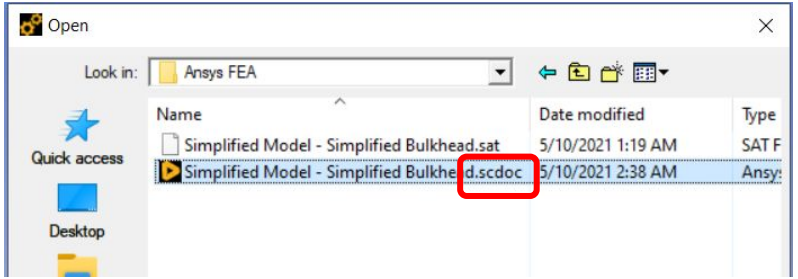


Export to Mechanical

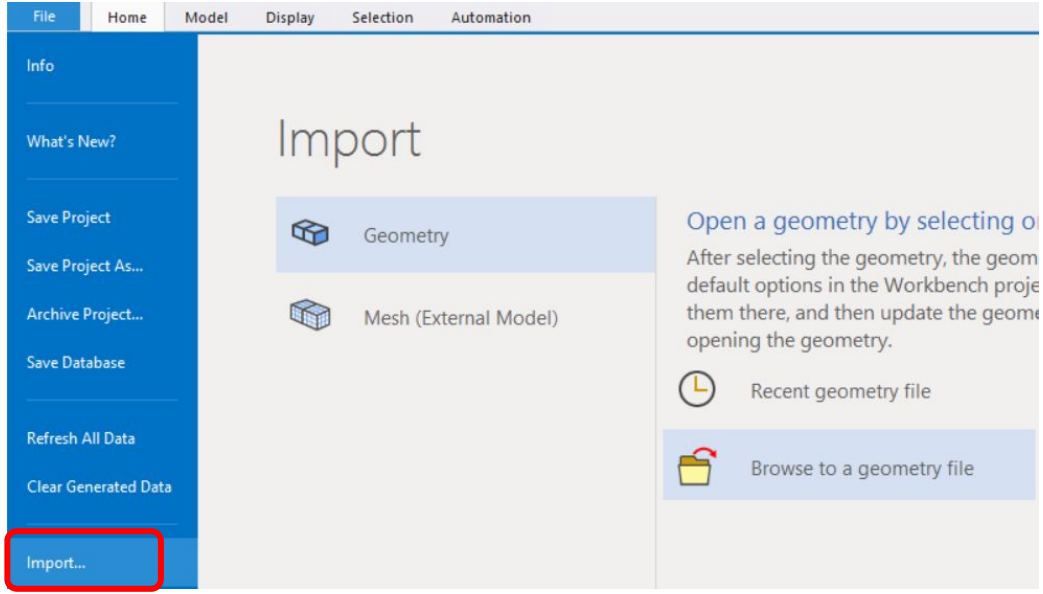


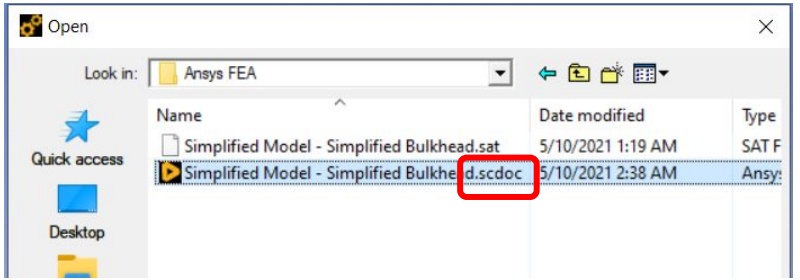
Export to Mechanical

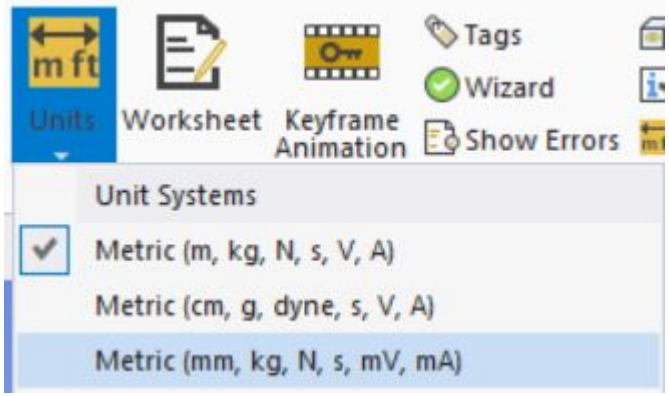




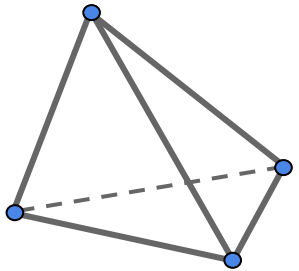
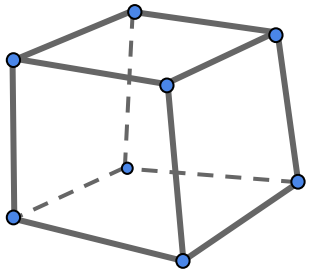
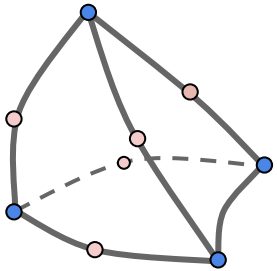
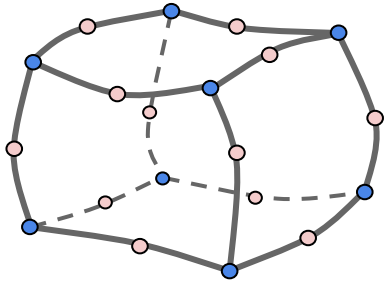
Export to Mechanical







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

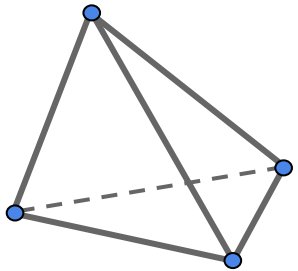
Linear elements suffer from **shear locking** - unable to represent pure bending

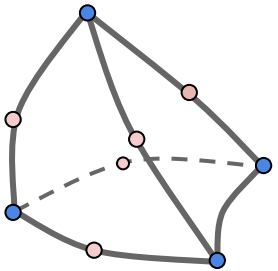
Linear will have **larger error**, especially for **curved geometry** and bending

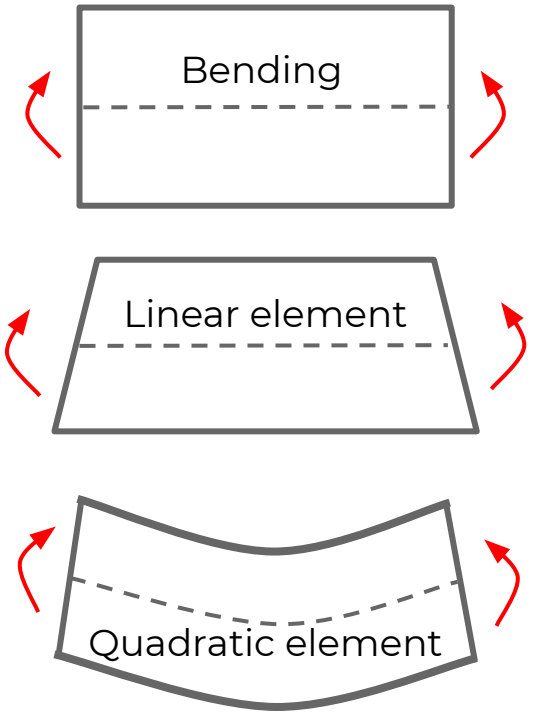
Quadratic may **converge more quickly** as mesh size reduces

Quadratic more **computationally expensive** than linear

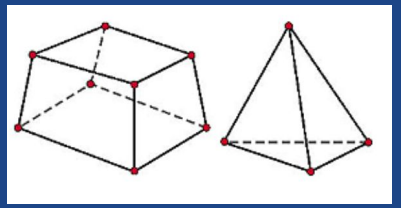
Quadratic **may be overkill** - if the variable in question is linear, use linear elements!







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.

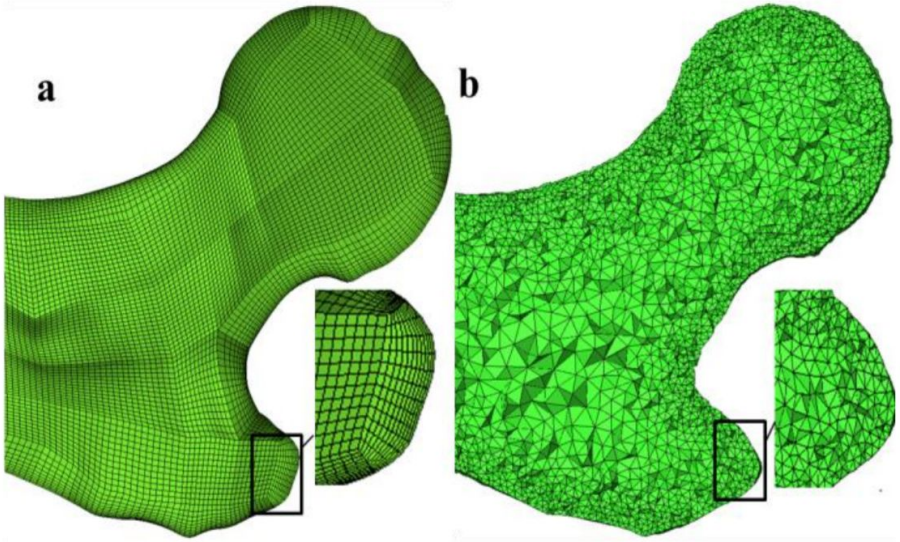
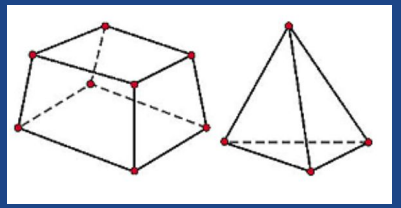


Figure 1: a) Hex, and b) Tet meshes for a femur.

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.

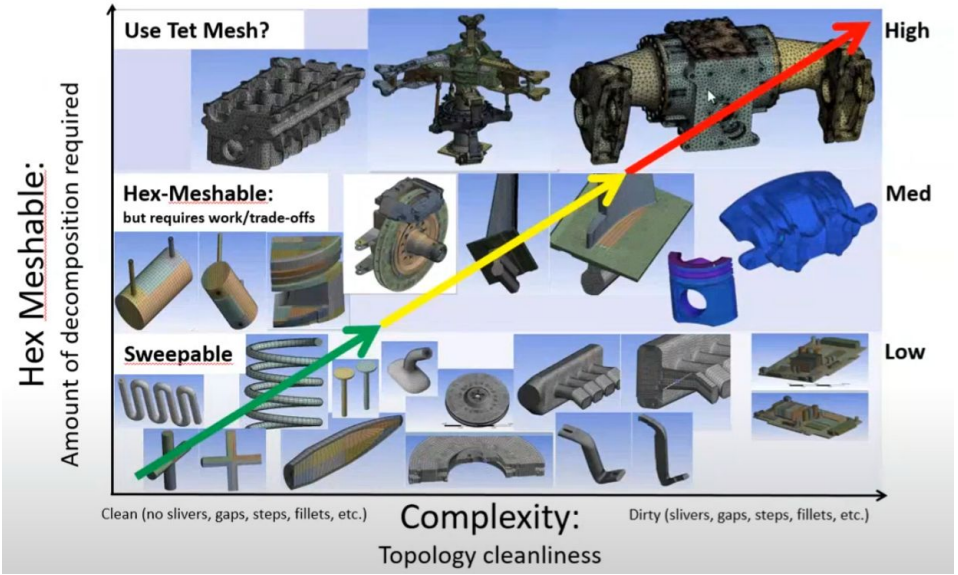


Image source: Leap Australia, *Introduction to ANSYS Meshing, Module 02: Meshing Methods.* https://www.leapaust.com.au/leap-learning-hub/

Meshing Advice

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

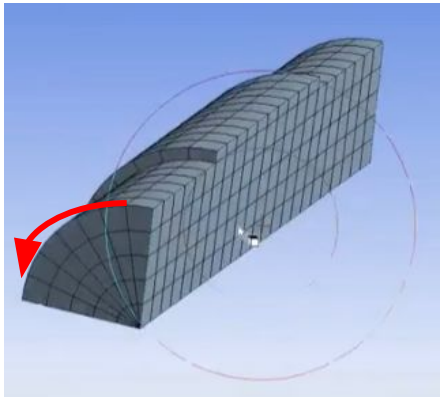
Check the **internal mesh** quality (sections)

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



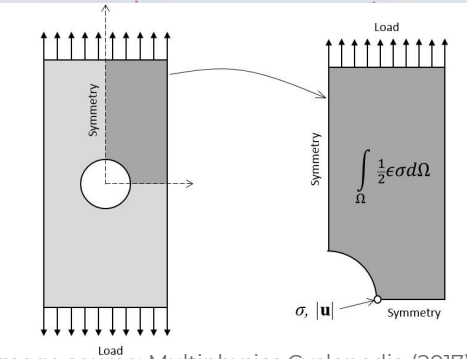


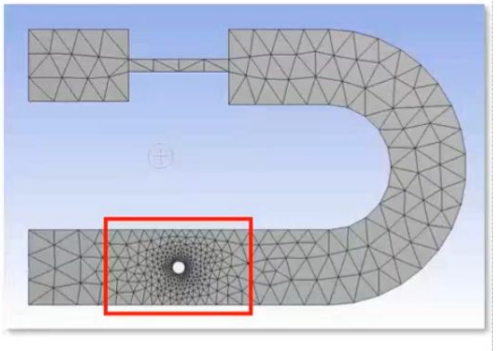
Image source: Multiphysics Cyclopedia (2017). *Finite Element Mesh Refinement.* *comsol.com/multiphysics/mesh-refinement*

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

Curvature and Proximity

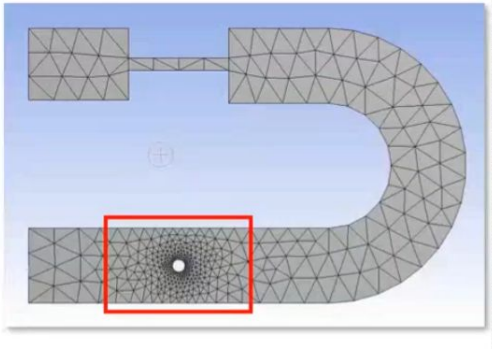
Curvature sizing limits the angle between adjacent elements

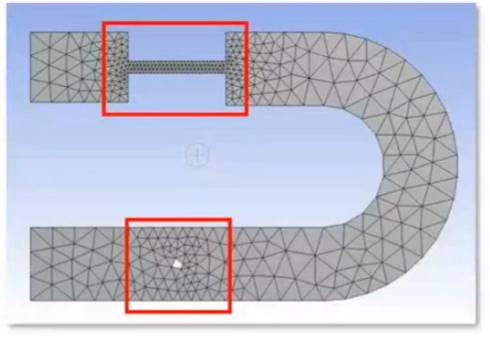
Best used for curved surfaces

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	
Use Adaptive Sizing	No
Growth Rate	Default (1.85)
Max Size	Default (16.0 mm)
Mesh Defeaturing	No
Capture Curvature	Yes
Curvature Min Size	Default (8.e-002 mm)
Curvature Normal Angle	Default (70.395°)
Capture Proximity	Yes
Proximity Min Size	4.0 mm
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

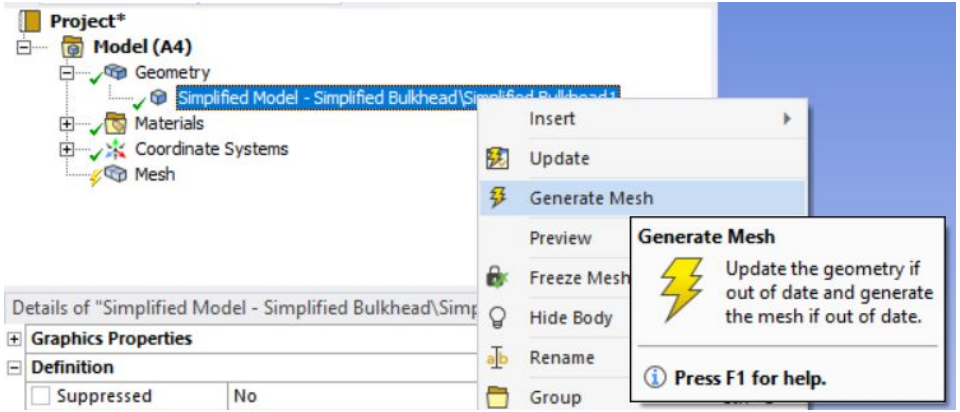
In areas of fine local mesh, features are kept

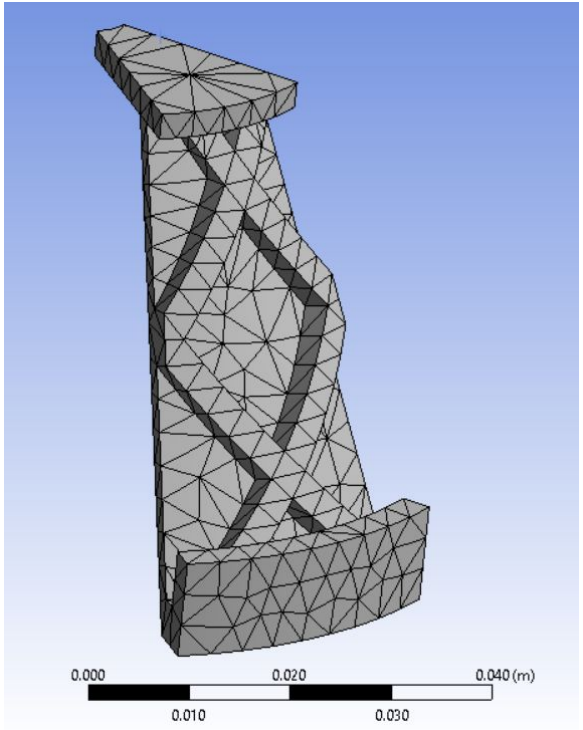
In areas of coarse local mesh, features are omitted

Default Mesh

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

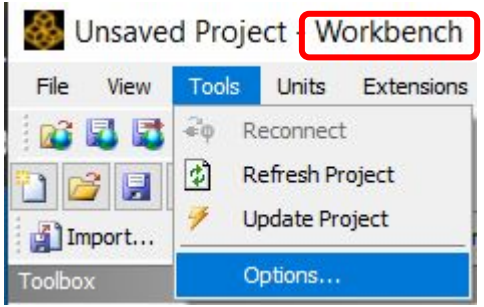
This mesh is generally quite **non-uniform** or poorly well-aligned with the part and loading

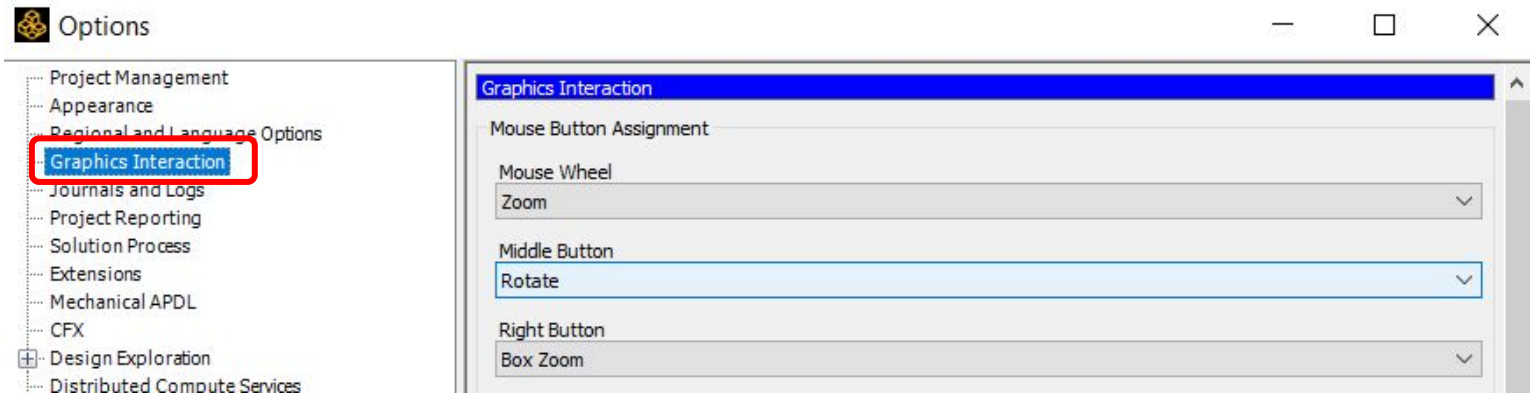




Mechanical Mouse Controls

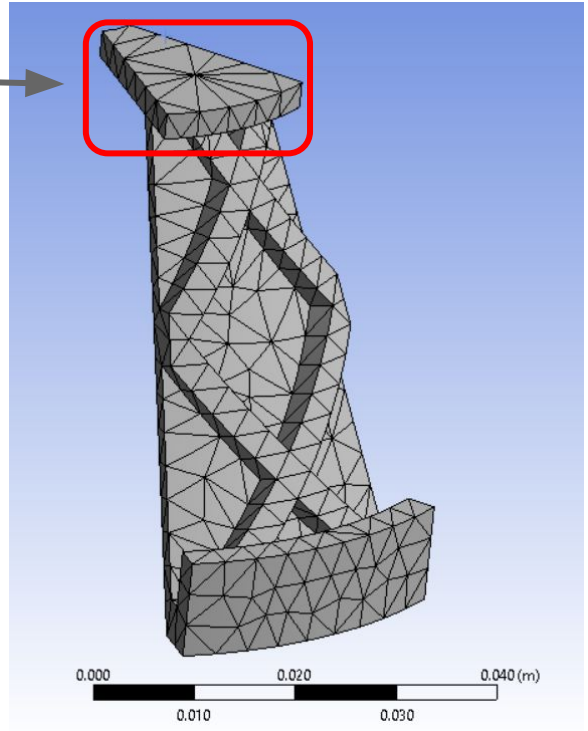
Mouse controls for Ansys Mechanical edited in **Workbench** under **Graphics Interaction**





Default Mesh

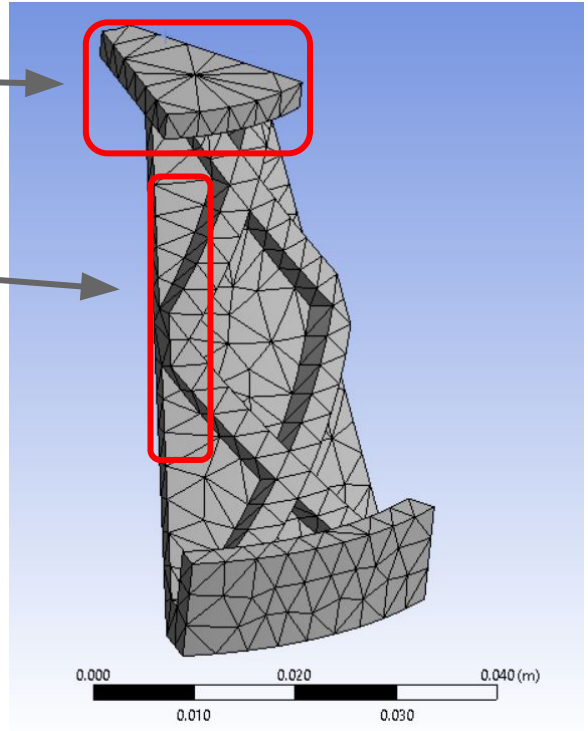
Doesn't utilise **sweeping**



Default Mesh

Doesn't utilise **sweeping**

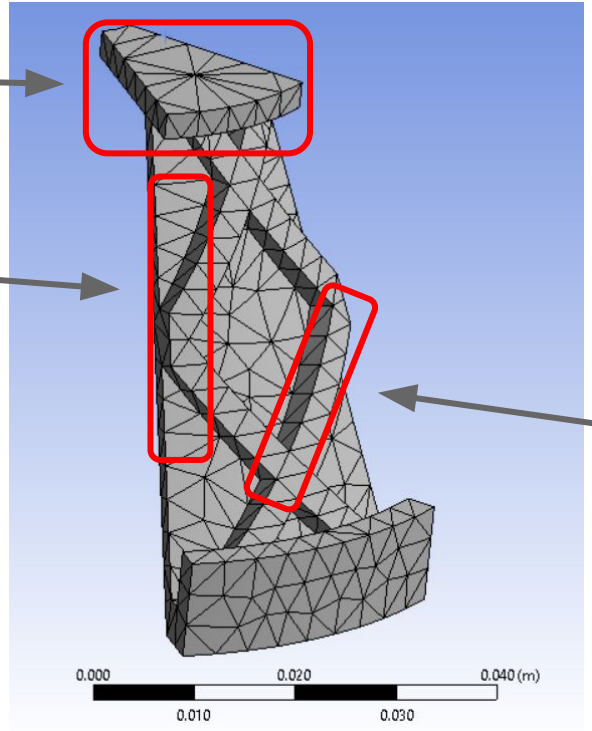
Doesn't align mesh with **loading direction**



Default Mesh

Doesn't utilise **sweeping**

Doesn't align mesh with **loading direction**



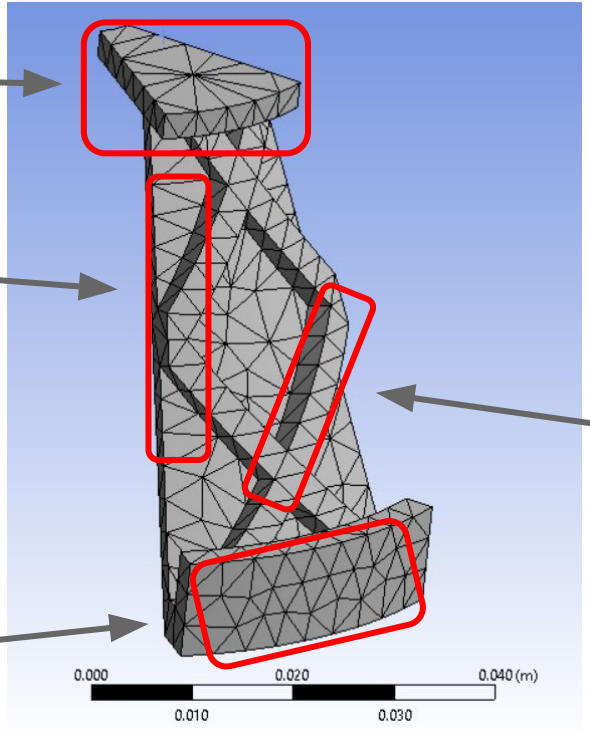
Doesn't **split geometry** into sweepable parts

Default Mesh

Doesn't utilise **sweeping**

Doesn't align mesh with **loading direction**

Doesn't utilise **hex mesh**

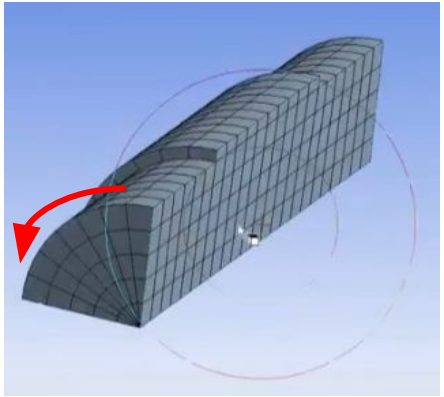


Doesn't **split geometry** into sweepable parts

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

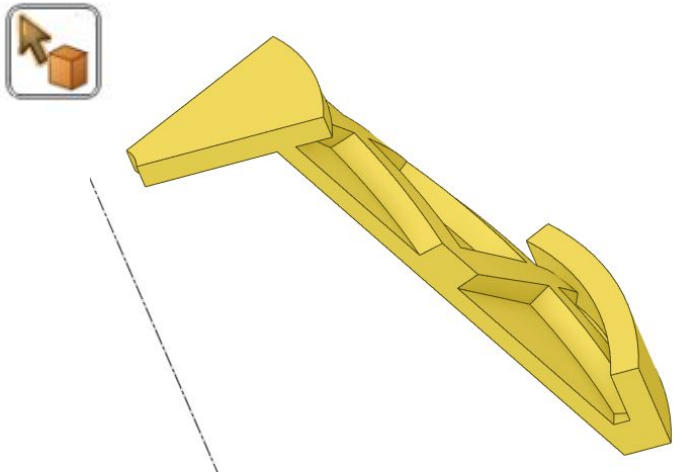
SpaceClaim - Splitting Geometry

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

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

Select **Split Body** tool

Select the body



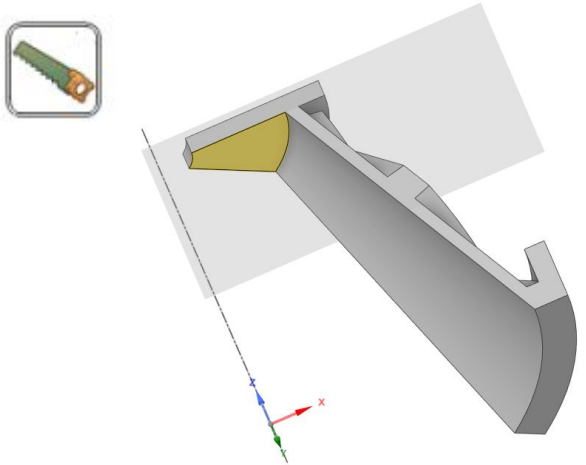
SpaceClaim - Splitting Geometry

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

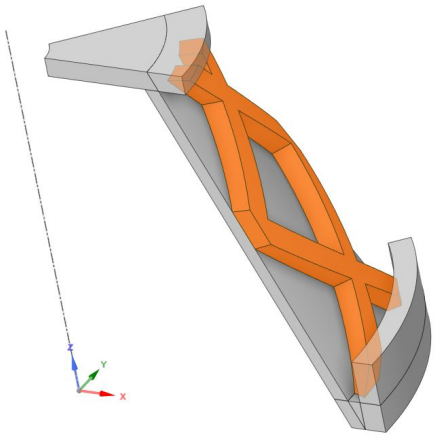
Let's **split** the body up **in SpaceClaim**

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**

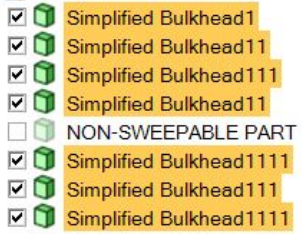
This will allow us to **sweep** the mesh

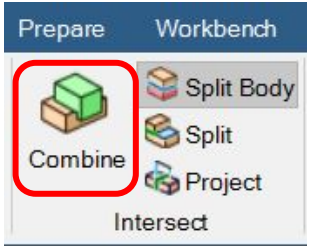
Select **Split Body** tool

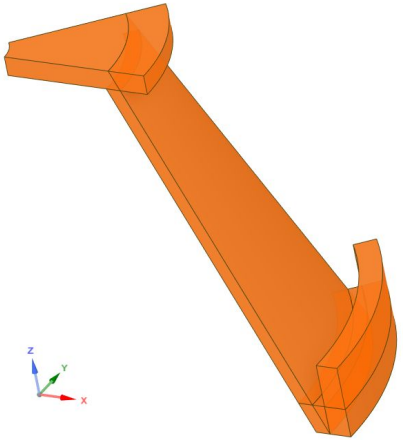
Select the body and the **face** to split by

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

Combine the sweepable regions







SpaceClaim - Splitting Geometry

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

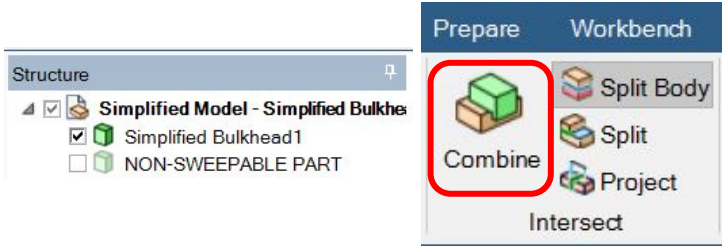
This will allow us to **sweep** the mesh

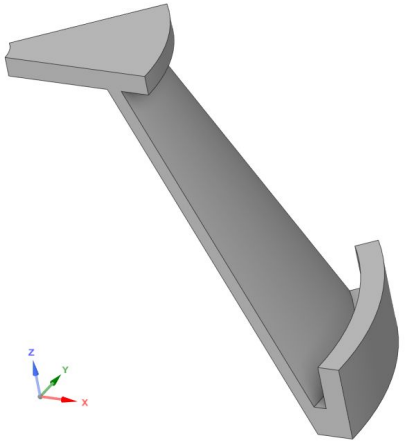
Select **Split Body** tool

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

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

Combine the sweepable regions





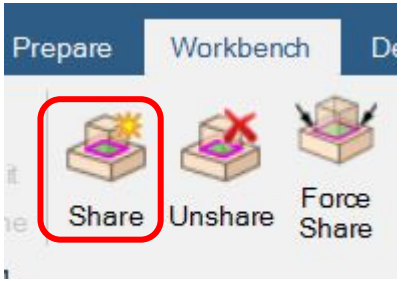
Sharing Topology - Skip this step!

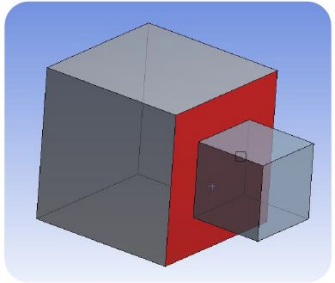
For designs with multiple parts, it's best to **share** any contacting surfaces

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"





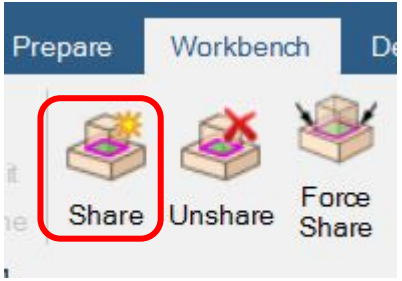
Sharing Topology - Skip this step!

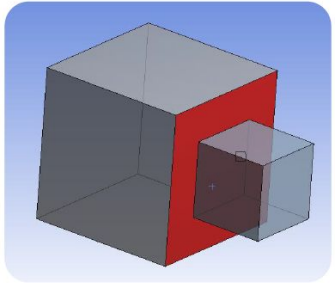
For designs with multiple parts, it's best to **share** any contacting surfaces

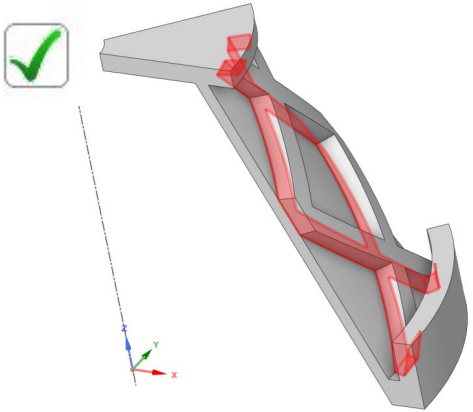
When two parts don't share the contacting faces, the **mesh will be discontinuous**

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!

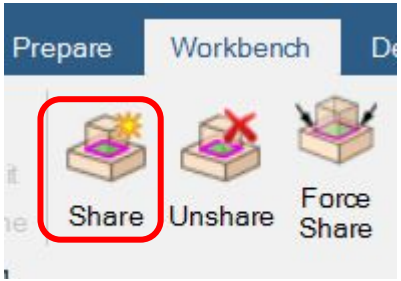
For designs with multiple parts, it's best to **share** any contacting surfaces

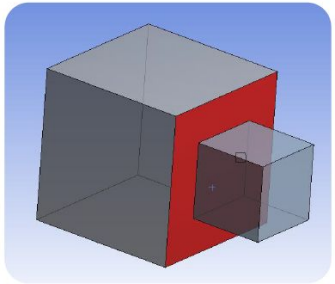
When two parts don't share the contacting faces, the **mesh will be discontinuous**

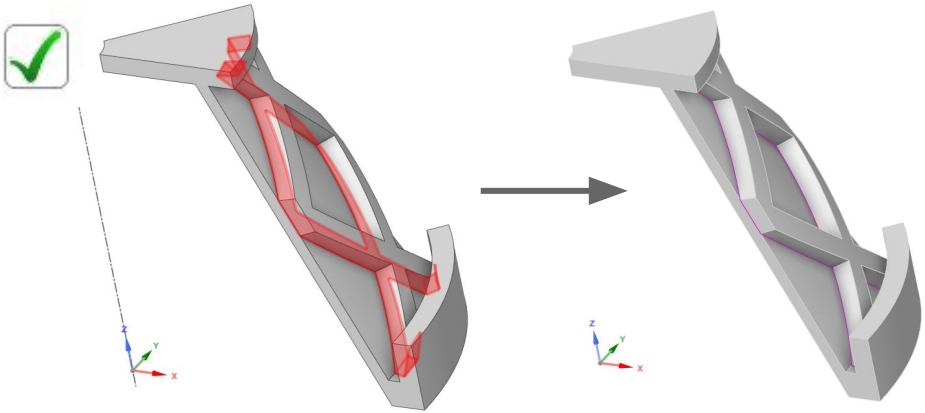
When the parts share surfaces, the **mesh will be continuous** and the shared surface will be an **internal** surface

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**)

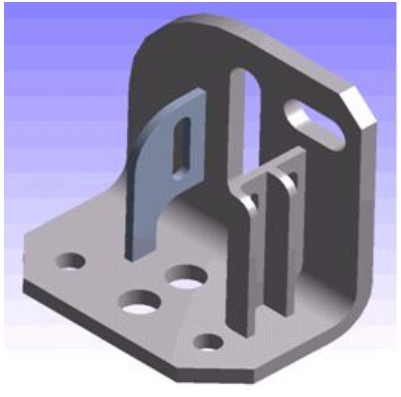
Geometry with **very thin shells → huge number of tiny elements** required for 3D mesh

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



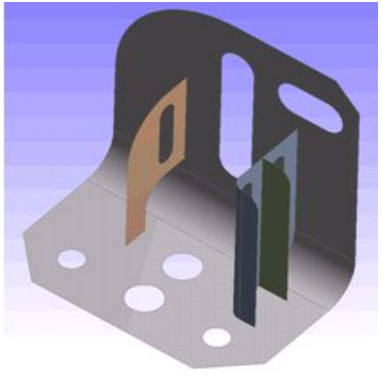
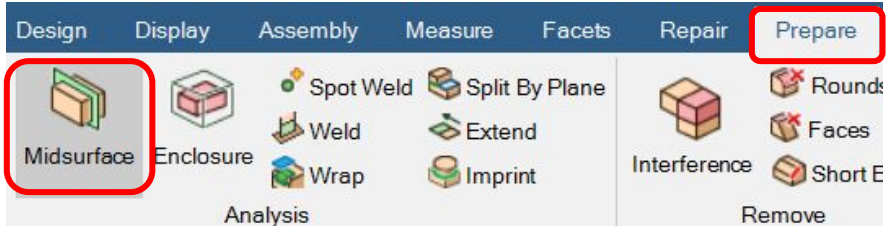


Image source: Ozen Engineering: Automation with the Flexibility to Customize
ozeninc.com/automation-wth-the-flexibility/



Midsurfacing (Shell Meshes)

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

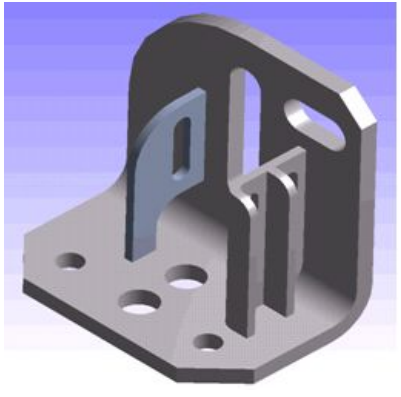
Geometry with **very thin shells → huge number of tiny elements** required for 3D mesh

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



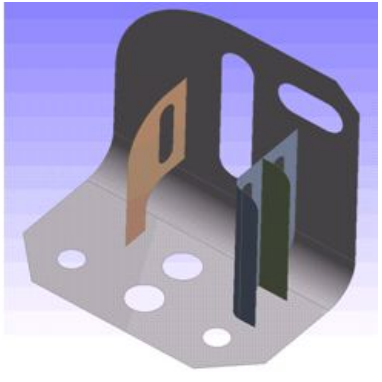
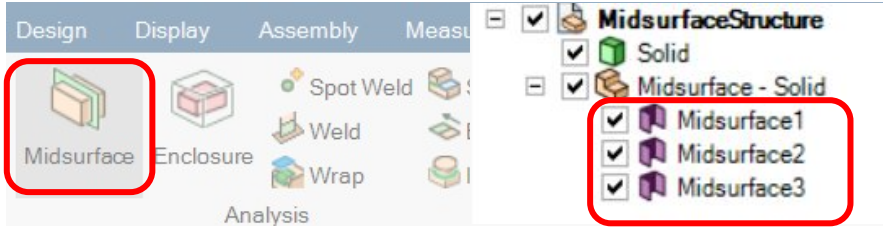


Image source: Ozen Engineering: Automation with the Flexibility to Customize
ozeninc.com/automation-wth-the-flexibility/



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

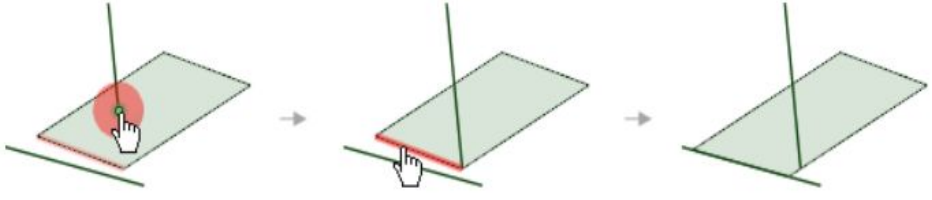


Image source: Ansys SpaceClaim Online Help
http://help.spaceclaim.com/2015.0.0/en/Content/Extend.htm

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

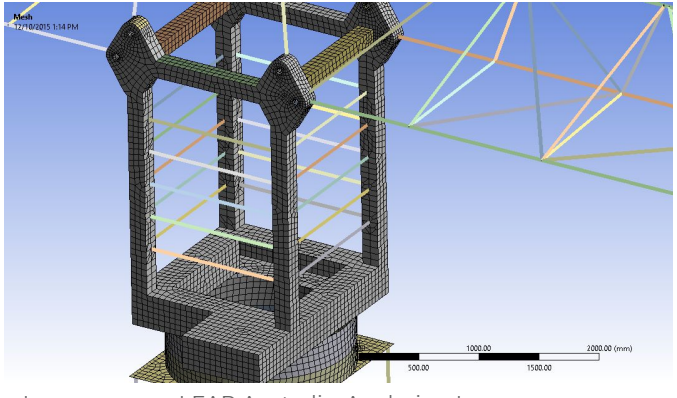
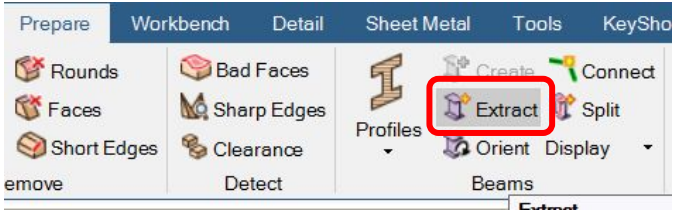


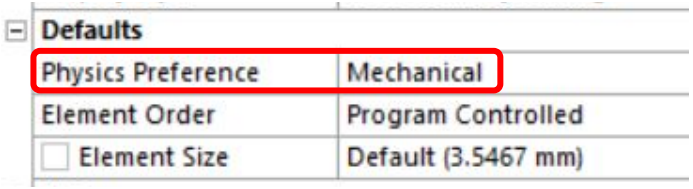
Image source: LEAP Australia: Analysing Large Fabricated Structures finiteelementanalysis.com.au/featured/analysing-large-fabricated-structures/



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

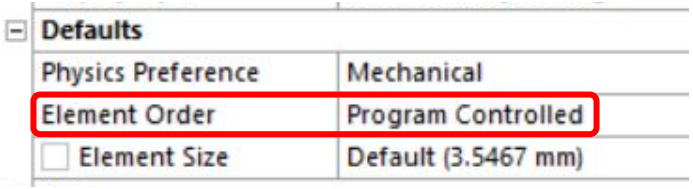


Global Mesh Controls

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)



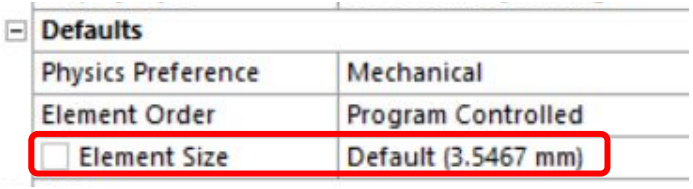
Global Mesh Controls

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)



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	
Use Adaptive Sizing	No
Growth Rate	Default (1.85)
Max Size	5.0 mm
Mesh Defeaturing	Yes
Defeature Size	Default (1.5e-002 mm)
Capture Curvature	Yes
Curvature Min Size	Default (3.e-002 mm)
Curvature Normal Angle	Default (70.395°)
Capture Proximity	Yes
Proximity Min Size	Default (3.e-002 mm)
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
Growth Rate	Default (1.85)
Max Size	5.0 mm
Mesh Defeaturing	Yes
Defeature Size	Default (1.5e-002 mm)
Capture Curvature	Yes
Curvature Min Size	Default (3.e-002 mm)
Curvature Normal Angle	Default (70.395°)
Capture Proximity	Yes
Proximity Min Size	Default (3.e-002 mm)
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
Growth Rate	Default (1.85)
Max Size	5.0 mm
Mesh Defeaturing	Yes
Defeature Size	Default (1.5e-002 mm)
Capture Curvature	Yes
Curvature Min Size	Default (3.e-002 mm)
Curvature Normal Angle	Default (70.395°)
Capture Proximity	Yes
Proximity Min Size	Default (3.e-002 mm)
Num Cells Across Gap	Default (3)
Proximity Size Function Sources	Faces and Edges
Bounding Box Diagonal	70.934 mm
Average Surface Area	124.15 mm²
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.

Sizing	
Use Adaptive Sizing	No
Growth Rate	Default (1.85)
Max Size	5.0 mm
Mesh Defeaturing	Yes
Defeature Size	Default (1.5e-002 mm)
Capture Curvature	Yes
Curvature Min Size	Default (3.e-002 mm)
Curvature Normal Angle	Default (70.395°)
Capture Proximity	Yes
Proximity Min Size	Default (3.e-002 mm)
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.

Sizing	
Use Adaptive Sizing	No
Growth Rate	Default (1.85)
Max Size	5.0 mm
Mesh Defeaturing	Yes
Defeature Size	Default (1.5e-002 mm)
Capture Curvature	Yes
Curvature Min Size	Default (3.e-002 mm)
Curvature Normal Angle	Default (70.395°)
Capture Proximity	Yes
Proximity Min Size	Default (3.e-002 mm)
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
Growth Rate	Default (1.85)
Max Size	5.0 mm
Mesh Defeaturing	Yes
Defeature Size	Default (1.5e-002 mm)
Capture Curvature	Yes
Curvature Min Size	Default (3.e-002 mm)
Curvature Normal Angle	Default (70.395°)
Capture Proximity	Yes
Proximity Min Size	Default (3.e-002 mm)
Num Cells Across Gap	Default (3)
Proximity Size Function Sources	Faces and Edges
Bounding Box Diagonal	70.934 mm
Average Surface Area	124.15 mm²
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
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
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
Target Quality	Default (0.050000)
Smoothing	Medium
Mesh Metric	None

Global Mesh Controls

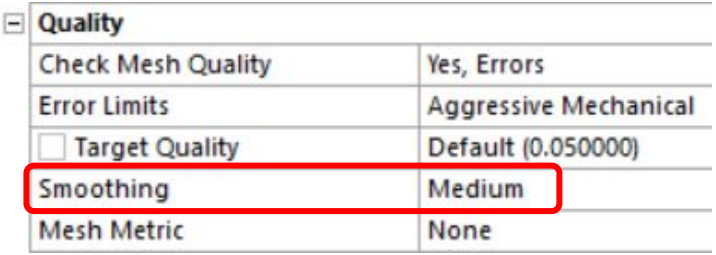
Quality section:

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

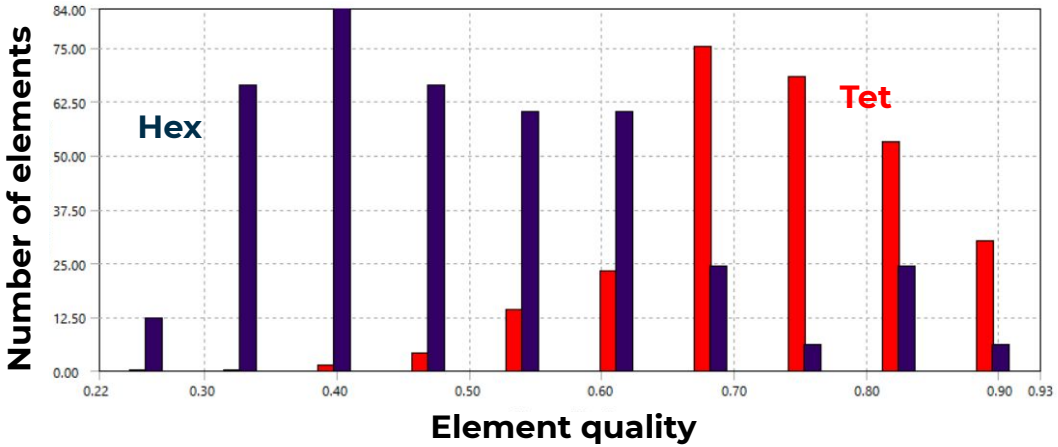


Global Mesh Controls

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
Target Quality	Default (0.050000)
Smoothing	Medium
Mesh Metric	None

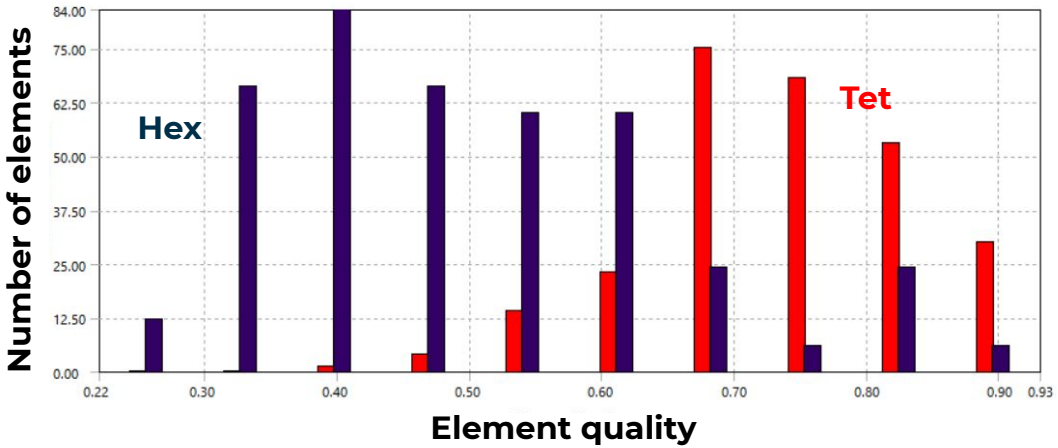


Sweep Meshing

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
Target Quality	Default (0.050000)
Smoothing	Medium
Mesh Metric	None

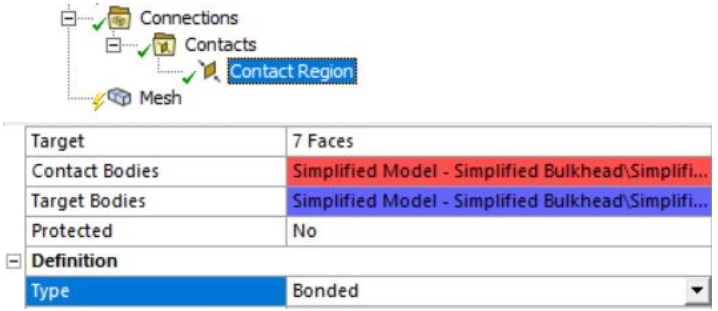


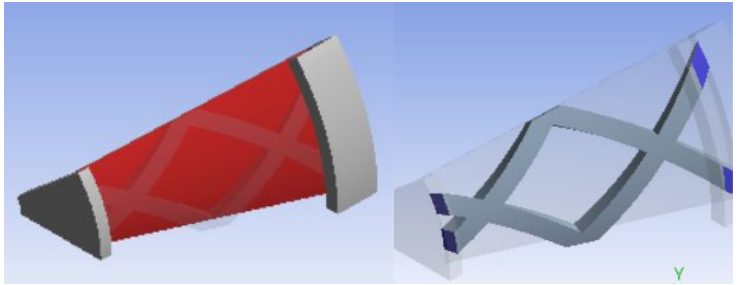
Contacts

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**



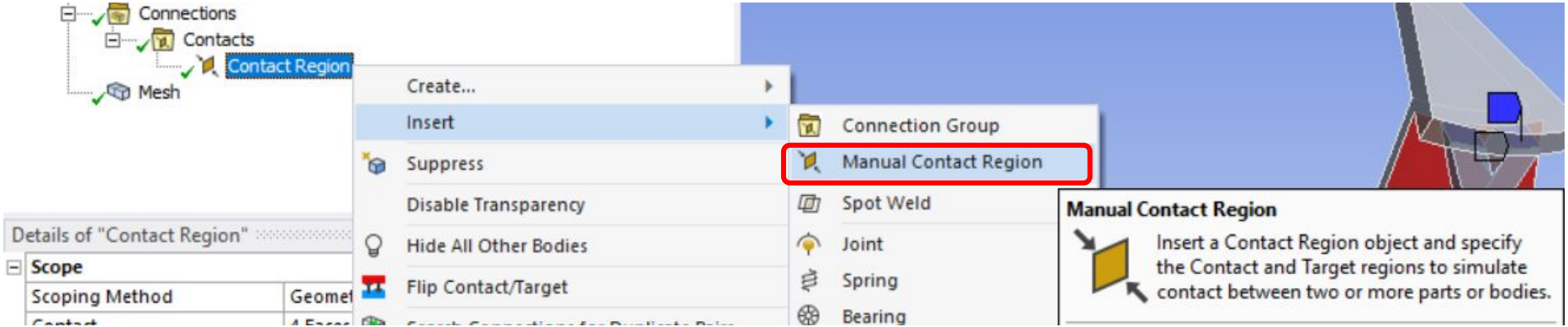


Contacts

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**

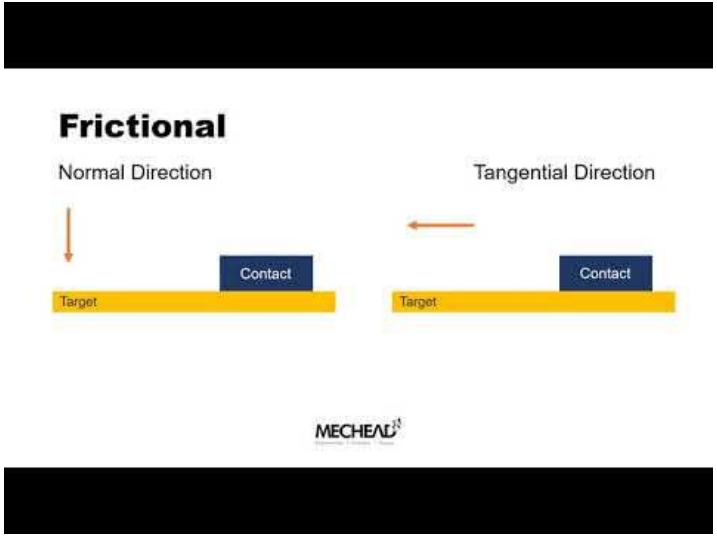
Other options include:

No separation - slide without resistance but can’t separate

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

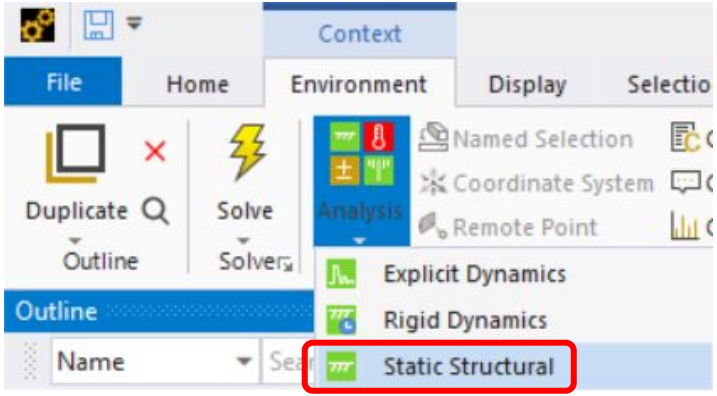
Frictional - can separate and slide with **resistance**

Rough - can **separate** but cannot slide



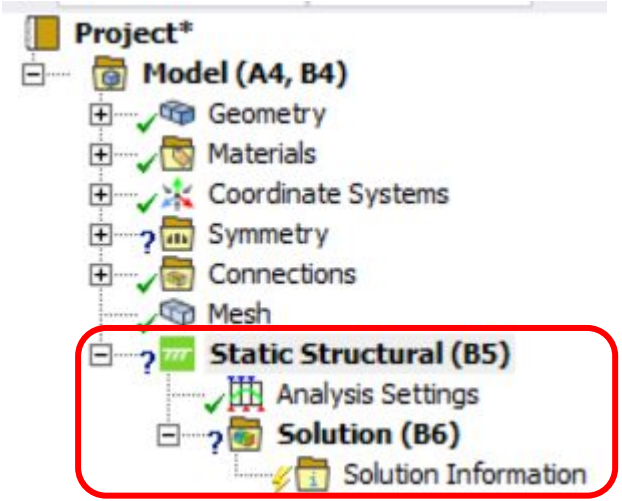
BCs - Supports

Insert a **Static Structural Analysis**



BCs - Supports

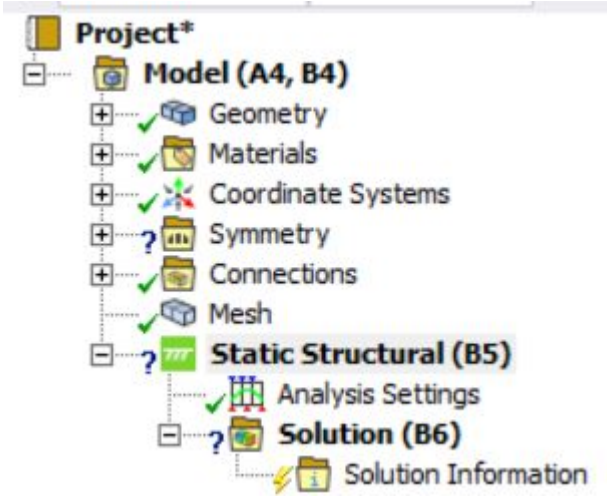
Insert a **Static Structural Analysis**

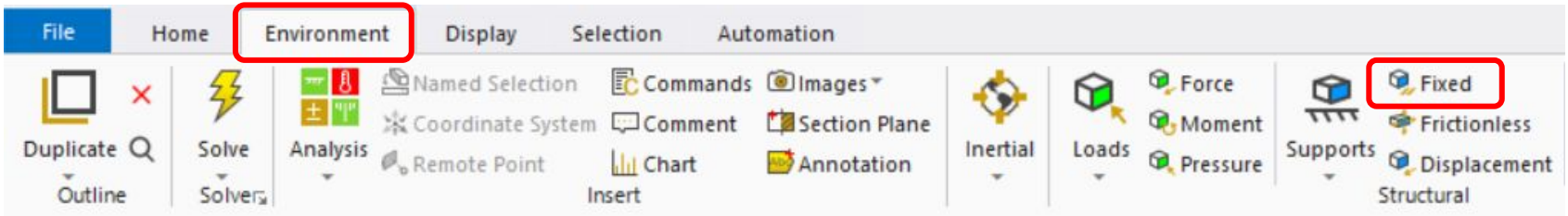


BCs - Supports

Insert a **Static Structural Analysis**

Insert a **Fixed Support** boundary condition



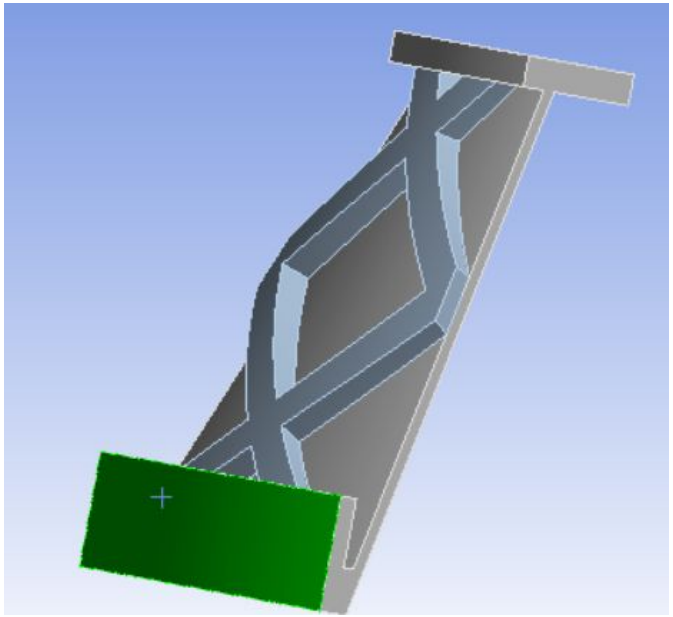


BCs - Supports

Insert a **Static Structural Analysis**

Insert a **Fixed Support** boundary condition

Select the appropriate faces for the fixed support

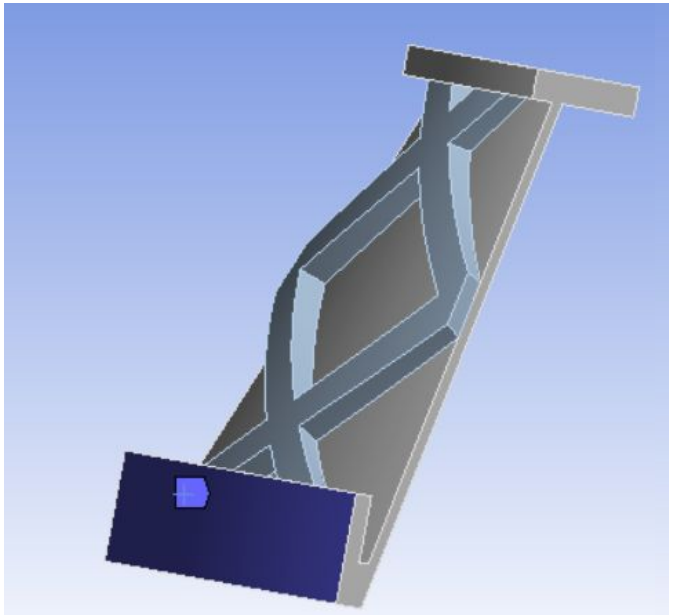


BCs - Supports

Insert a **Static Structural Analysis**

Insert a **Fixed Support** boundary condition

Select the appropriate faces for the fixed support

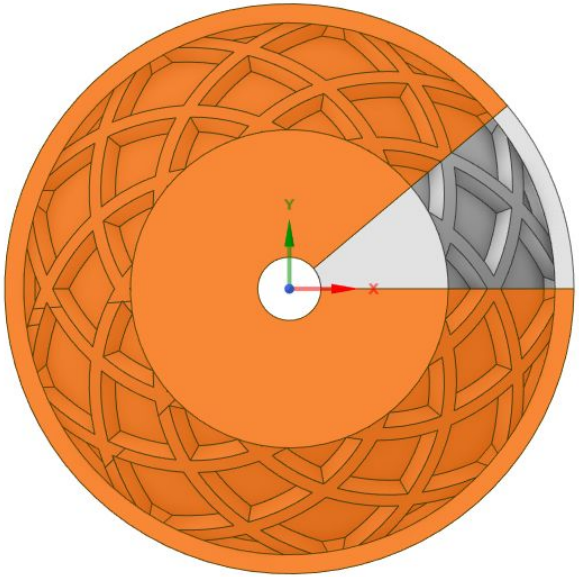


Details of "Fixed Support"

Scope		
Scoping Method	Geometry Selection	
Geometry	Apply	Cancel
Definition		
Type	Fixed Support	
Suppressed	No	

BCs - Symmetry

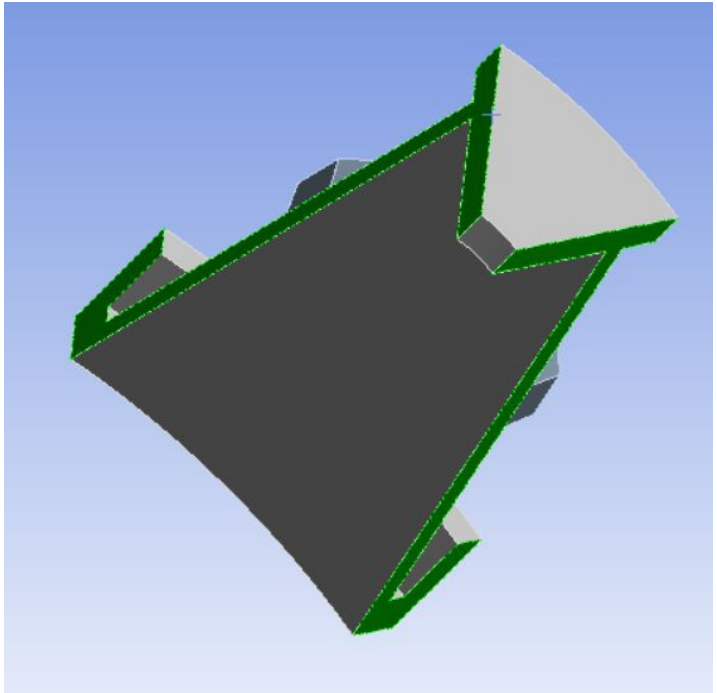
The wedge we're modelling is **constrained** by the rest of the bulkhead around it



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

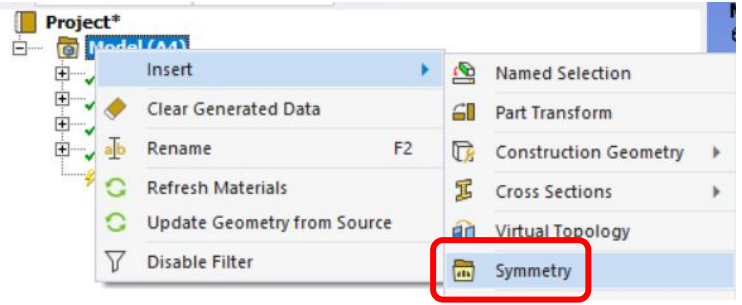


BCs - Symmetry

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

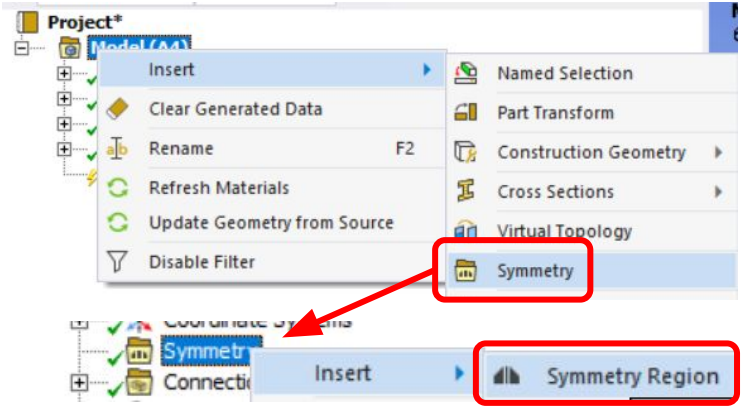


BCs - Symmetry

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

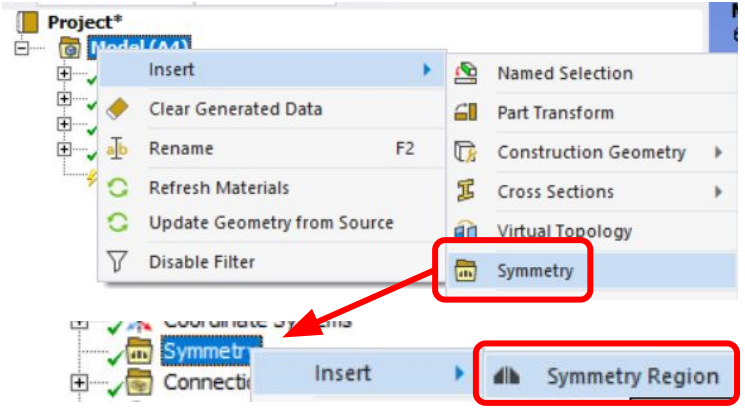
The wedge we're modelling is **constrained** by the rest of the bulkhead around it

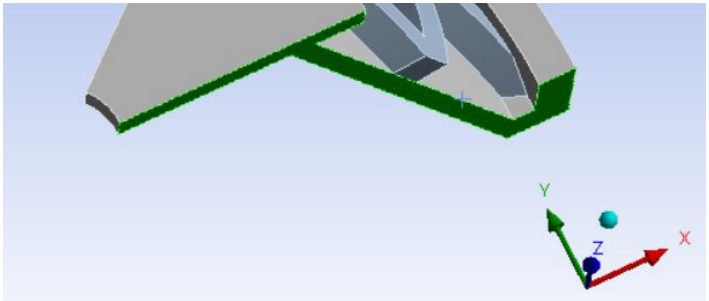
These faces can't move outside their own planes

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

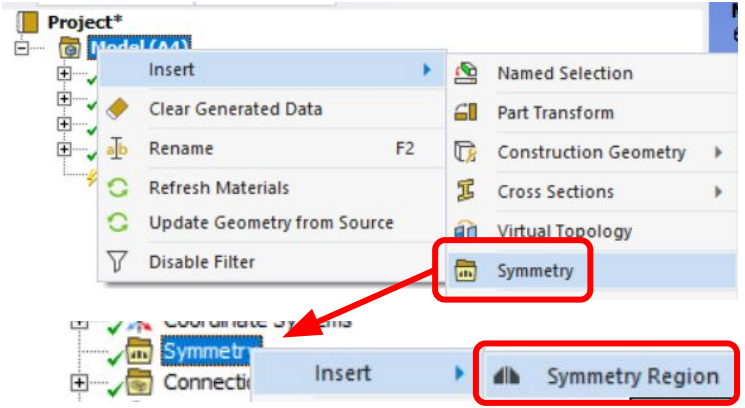
The wedge we're modelling is **constrained** by the rest of the bulkhead around it

These faces can't move outside their own planes

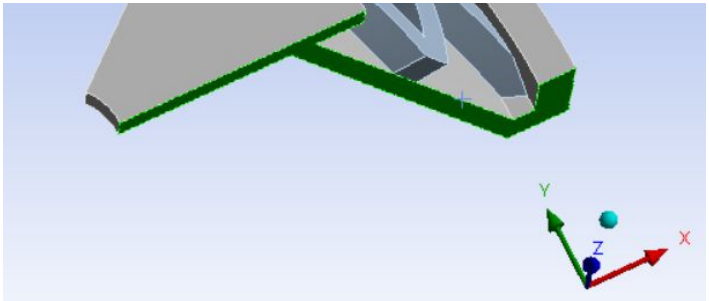
Symmetry BCs limit the motion of a face to a single plane

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

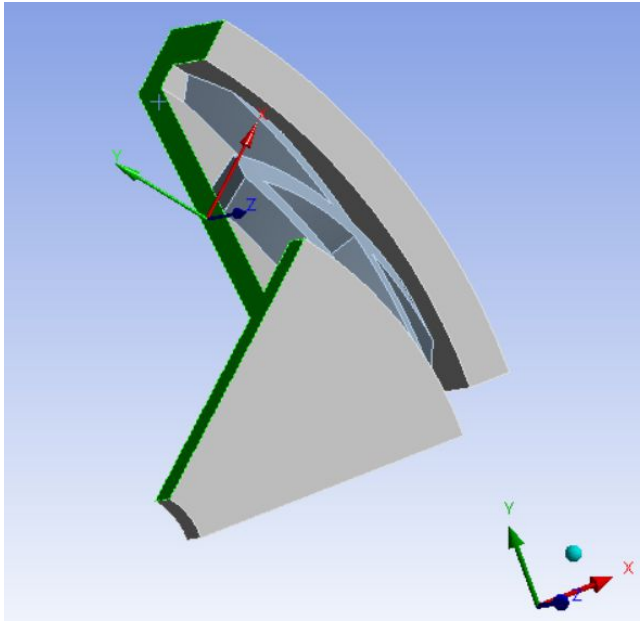
Need a **coordinate system** normal to the other face

BCs - Symmetry

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

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

Principal Axis		
Axis	Y	
Define By	Geometry Selection	
Geometry	Apply	Cancel

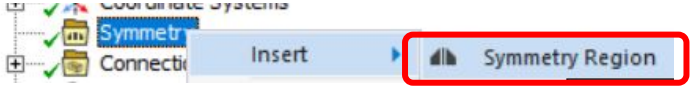


BCs - Symmetry

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

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

Insert another **symmetry region**



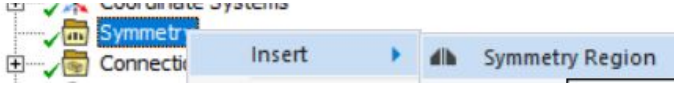
BCs - Symmetry

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

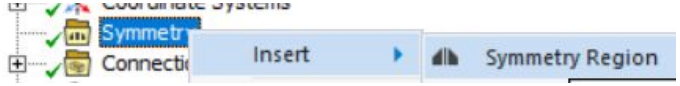
Need a **coordinate system** normal to the other face

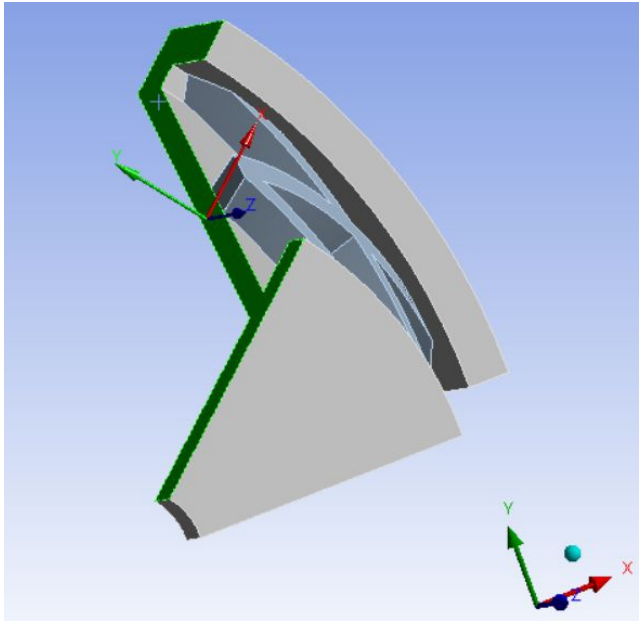
Want principal ***y*** axis normal to the **face** shown

Insert another **symmetry region**

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

Select the other **face**



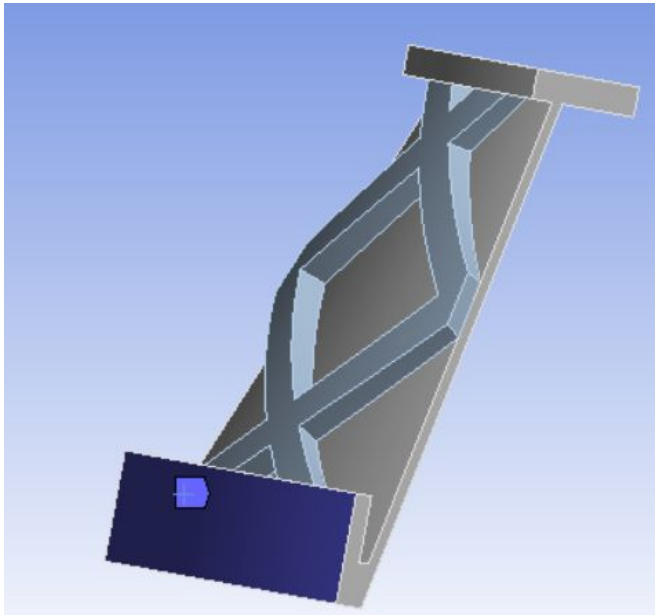


Definition	
Scope Mode	Manual
Type	Symmetric
Coordinate System	Coordinate System
Symmetry Normal	Y Axis
Suppressed	No

Scope	
Scoping Method	Geometry Selection
Geometry	Apply

Forces

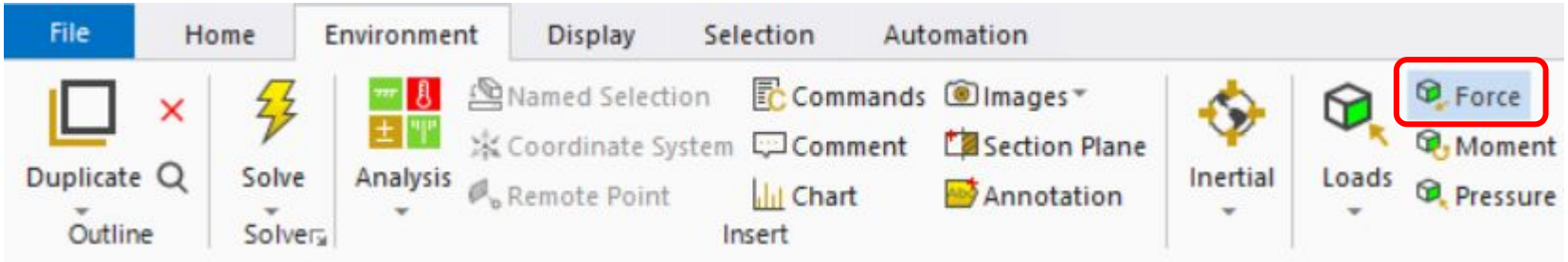
Ensure the body is **constrained** by enough **supports**



Forces

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

Insert a **force**



Forces

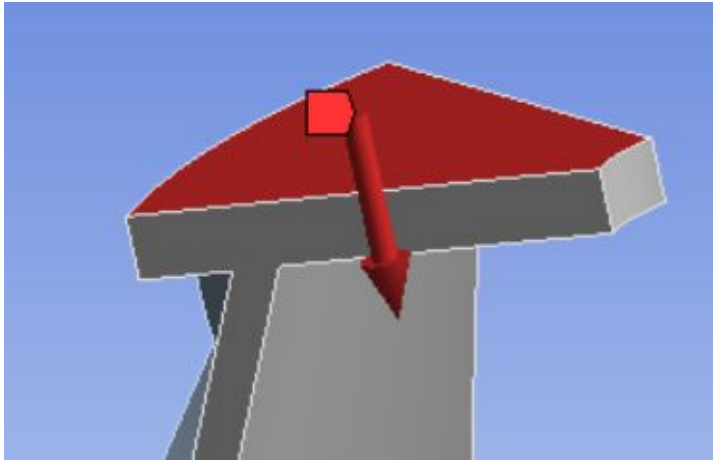
Ensure the body is **constrained** by enough **supports**

Insert a **force**

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

Details of "Force"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Type	Force
Define By	Vector
Applied By	Surface Effect
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! :)