

# ARES Workshop - Ansys CFD

Cas Kent & Ann Phan





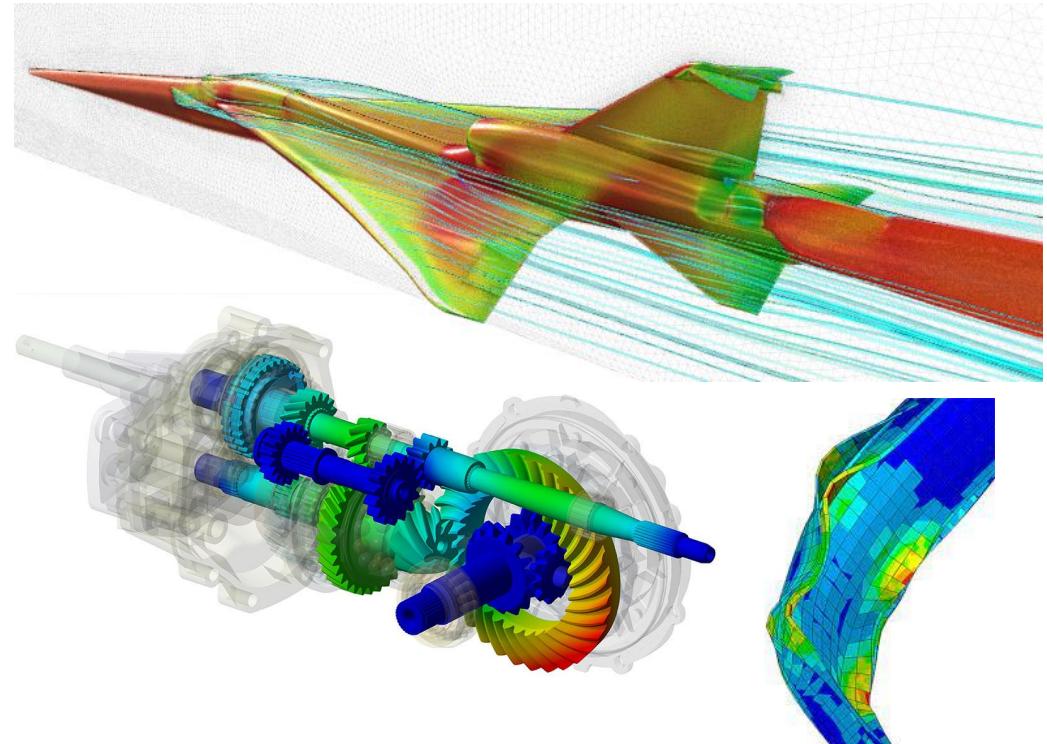
- Industry standard engineering **simulation** software
- Complete package solution
- Global leader in simulation
- Trusted partner of ARES - **LEAP Australia** provide support

**leap** **australia**  
pty ltd

A graphic element consisting of a dark blue wavy line that curves upwards and to the right, ending in a small five-pointed star.

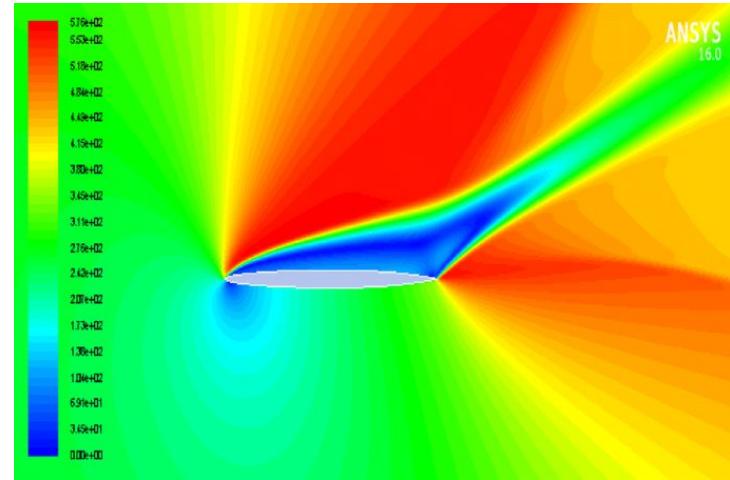
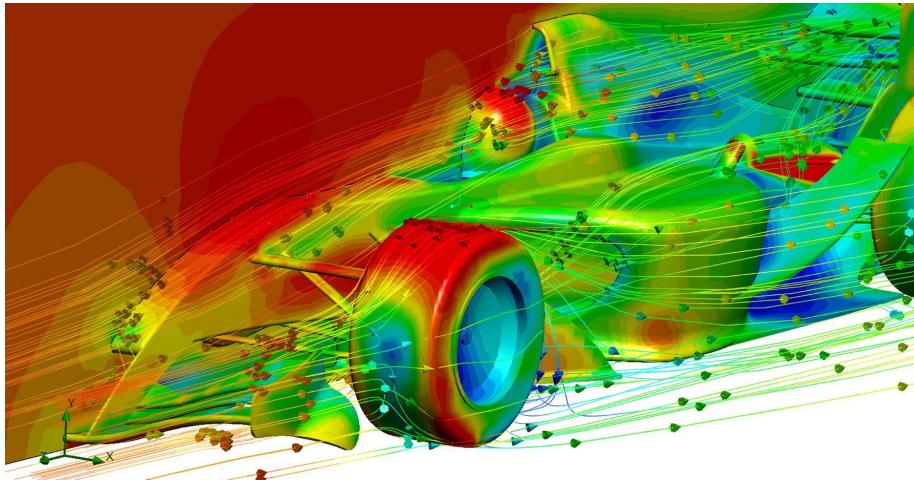


- 3D Design tools (**CAD**)
- Simulation of various physical phenomena, including:
  - Fluid dynamics (**CFD**)
  - Structural mechanics (**FEA**)
  - Thermal
  - Electromagnetic
- Multiphysics simulations
- Model composite materials (carbon fibre)



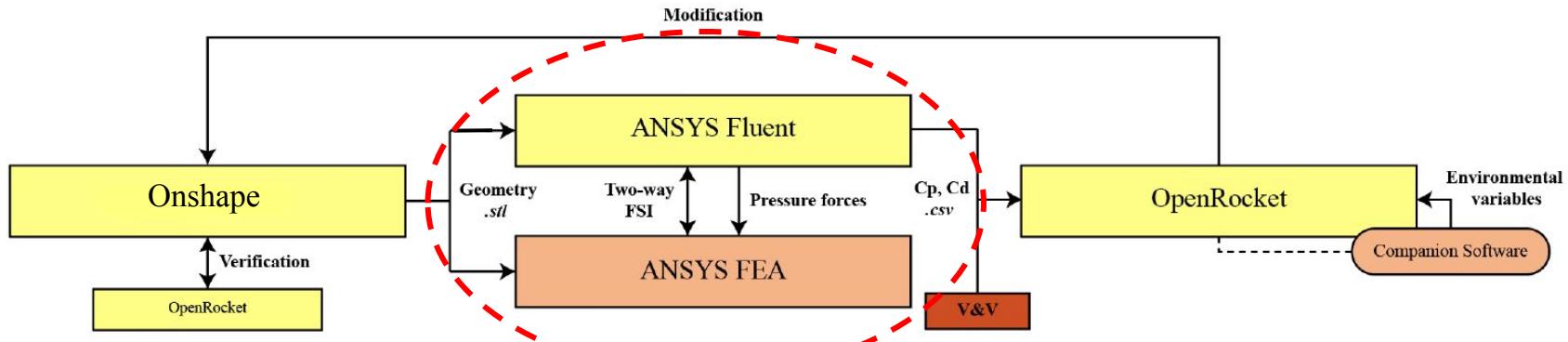
# Why CFD?

- Numerically solve fluids problems
- Detailed info about **pressure**, velocity, temperature distributions
- Approximate lift and **drag forces**



# Why CFD?

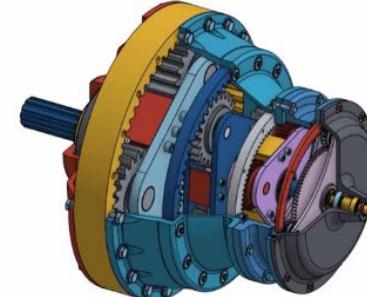
- Update our **OpenRocket** data for more accurate flight simulations
- Couple with **FEA** (stress distribution) for greater accuracy
- Reduces effort and cost by avoiding physical experiments





# Software you need

- **OpenRocket**
  - 2D model and flight simulations
- **Onshape**
  - Online 3D modelling (CAD) software
  - ARES uses this for all of our **3D design projects**



## Software you need

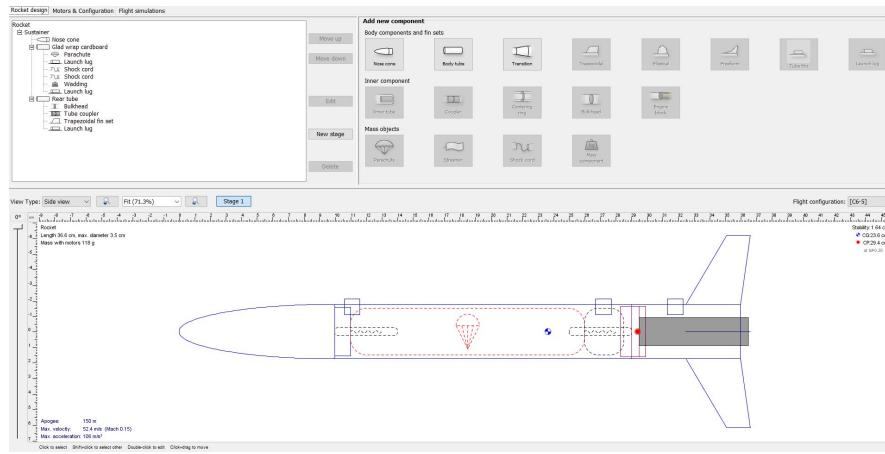
- Ansys **SpaceClaim** - CAD software
  - Don't build your model in SpaceClaim!
  - **Prepare the geometry** for CFD
- Ansys **Fluent** - CFD software
  - Build the mesh for CFD
  - Solve the CFD



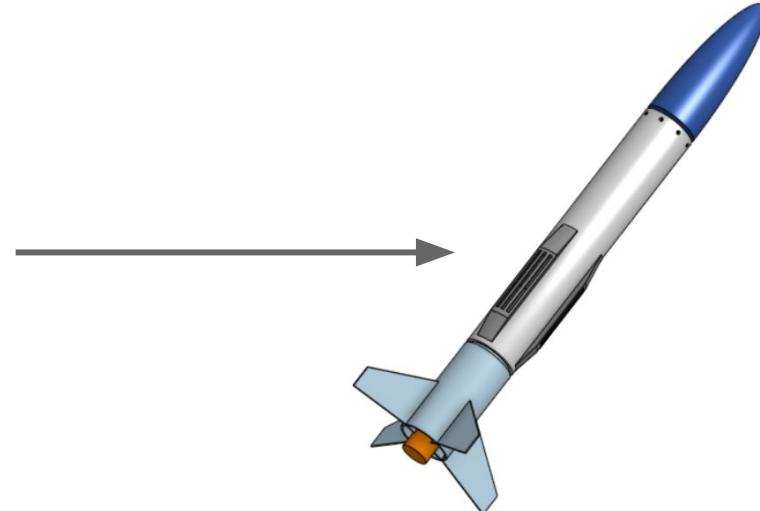


# Design Workflow

OpenRocket (2D model)  
**.ork**

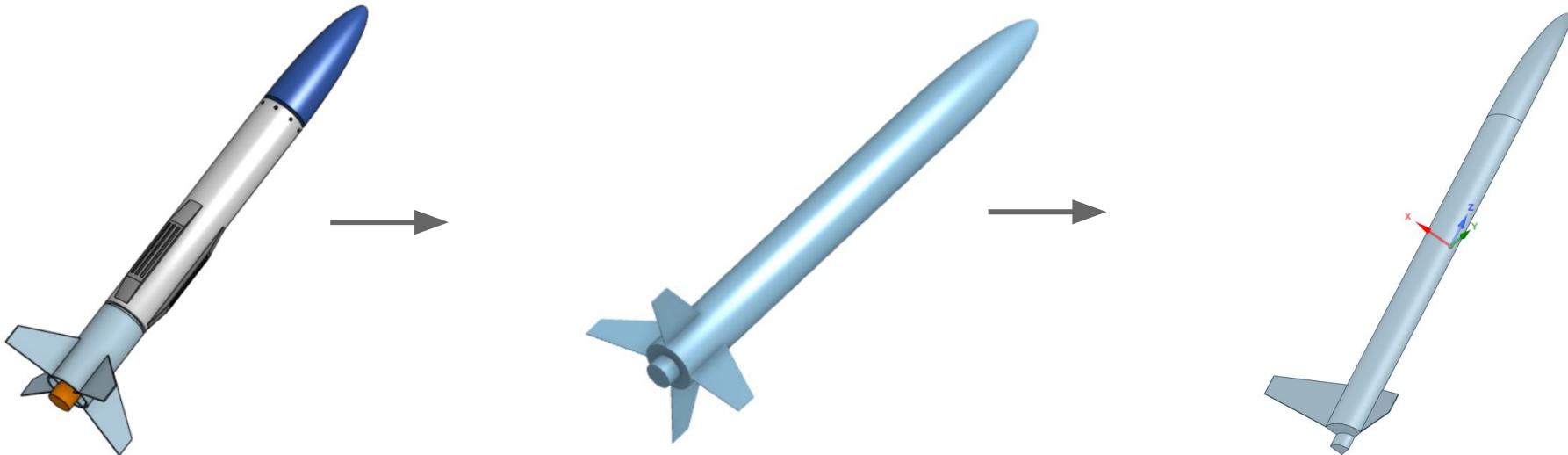
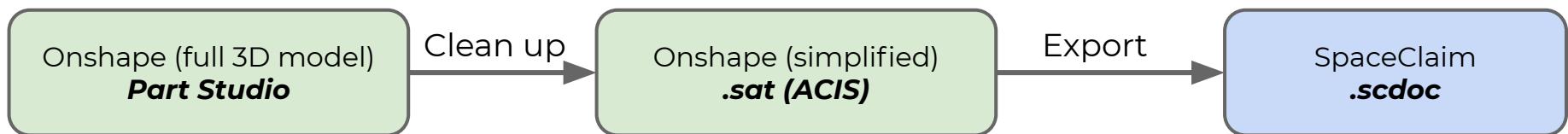


Onshape (full 3D model)  
**Part Studio**

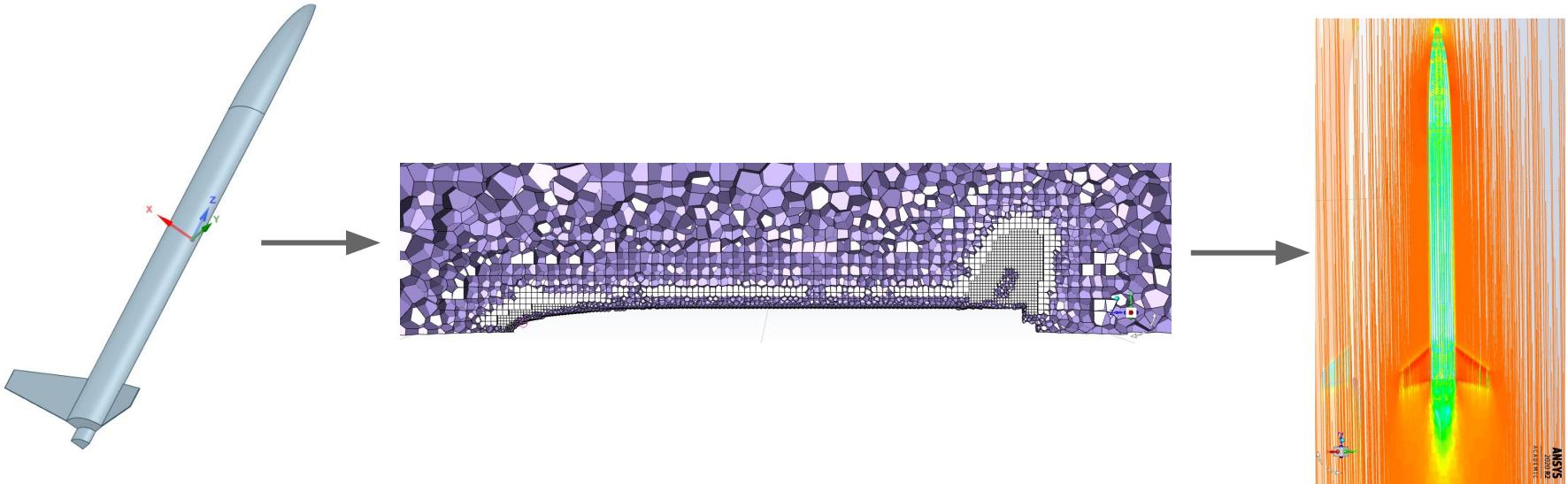




# Design Workflow

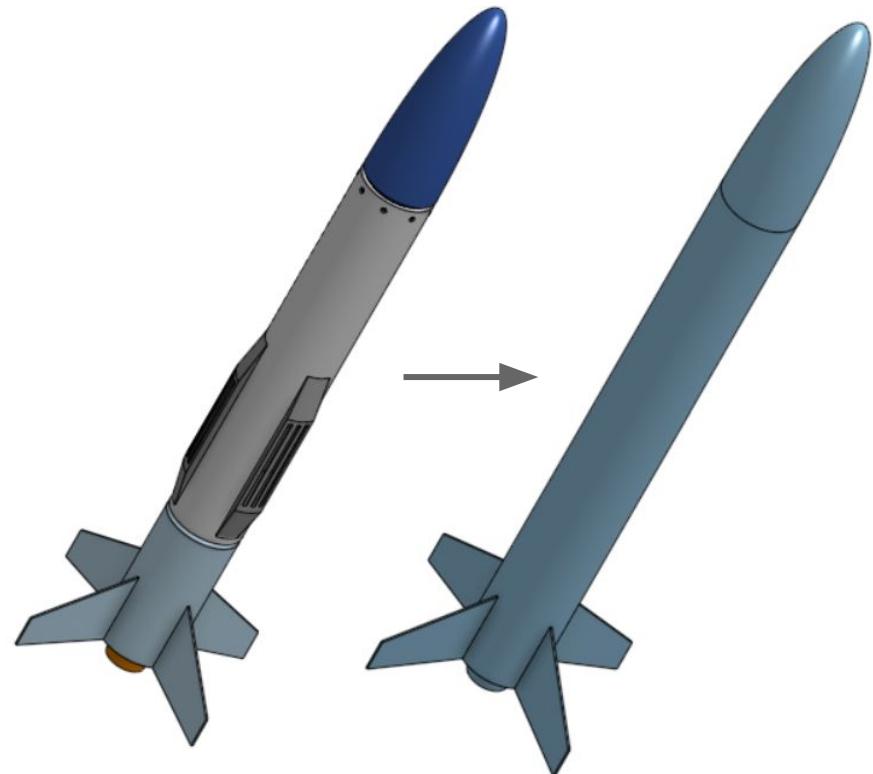


# Design Workflow



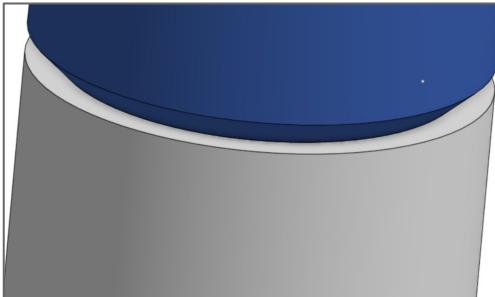
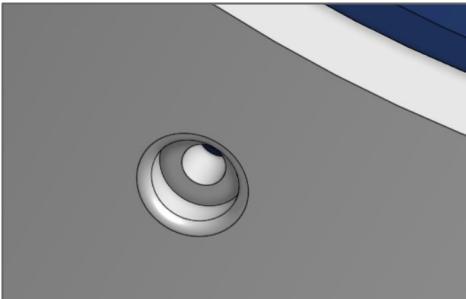
# Preparing Geometry

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

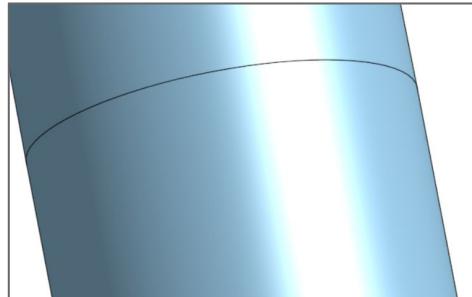
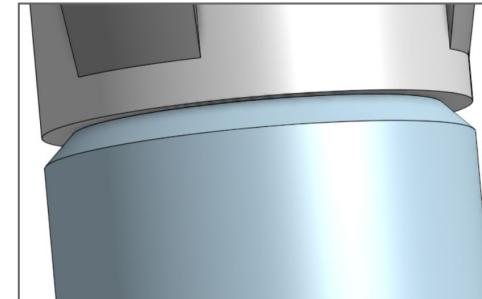


## Cleanup - Onshape

Small faces,  
manufacturing  
features

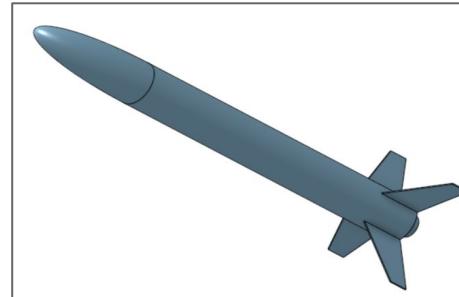
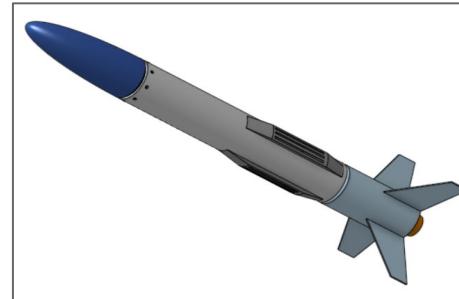


Fillets, chamfers,  
rounds



## Cleanup - Onshape

Relevant features  
(influence flow?)



# Export Onshape

The screenshot shows the Onshape interface with the following elements:

- Left Panel:** A tree view of the model structure:
  - Default geometry
    - Origin
    - Top
    - Front
    - Right
  - Sketch 1, Sketch 2, Sketch 3, Sketch 4, Sketch 5
  - Revolve 1
  - Extrude 1
  - Circular pattern 1
  - Parts (1)
    - C6-5 Rocket
- Center Viewport:** A 3D model of a C6-5 Rocket, colored blue.
- Right Context Menu:** A context menu is open over the rocket model, listing options:
  - Rename
  - Properties...
  - Assign material...
  - Edit appearance...
  - Copy...
  - Create Drawing of C6-5 Rocket...
  - Export...** (This option is highlighted with a blue background)
  - Hide
  - Isolate...
  - Make transparent...
  - Add comment
  - Zoom to selection
  - Delete...
- Bottom Navigation Bar:** Includes icons for adding a new part (+), the current part name ("Rocket model"), and two other tabs: "CFD Model Simplified" (which is highlighted with a red border) and "CFD Model Co".

# Export Onshape

The screenshot shows the Onshape interface with a blue header bar. The main area displays a 3D model of a C6-5 Rocket, which is blue with a pointed nose cone and three fins at the base. To the left of the model is a tree view of the part's history and components:

- Default geometry
  - Origin
  - Top
  - Front
  - Right
  - Sketch 1
  - Sketch 2
  - Sketch 3
  - Sketch 4
  - Sketch 5
- Revolve 1
- Extrude 1
- Circular pattern 1
- Parts (1)
  - C6-5 Rocket

A context menu is open over the rocket model, listing options: Rename, Properties..., Assign material..., Edit appearance..., Copy..., Create Drawing of C6-5 Rocket..., Export..., Hide, Isolate..., Make transparent..., Add comment, Zoom to selection, and Delete... . The "Export..." option is highlighted with a blue background.

At the bottom of the screen, there are several tabs and icons: a plus sign icon, a "Rocket model" tab, a "CFD Model Simplified" tab, and a "CFD Model Co" tab.

# Export Onshape

The screenshot shows the Onshape interface with a 3D model of a C6-5 Rocket. The model is blue and has three fins at the base. A context menu is open over the rocket, with the 'Export...' option highlighted by a red rectangle. The left sidebar lists various geometry elements like 'Origin', 'Sketch 1', and 'Revolve 1', and a 'Parts (1)' section with 'C6-5 Rocket' selected.

- Default geometry
  - Origin
  - Top
  - Front
  - Right
  - Sketch 1
  - Sketch 2
  - Sketch 3
  - Sketch 4
  - Sketch 5
  - Revolve 1
  - Extrude 1
  - Circular pattern 1
- Parts (1)
  - C6-5 Rocket

+

Rocket model

CFD Model Simplified

CFD Model Co

# Export Onshape

## Default geometry

Origin

Top

Front

Right

Sketch 1

Sketch 2

Sketch 3

Sketch 4

Sketch 5

Revolve 1

Extrude 1

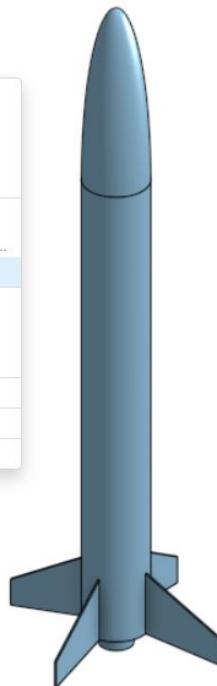
Circular pattern 1

## Parts (1)

C6-5 Rocket



- Rename
- Properties...
- Assign material...
- Edit appearance...
- Copy...
- Create Drawing of C6-5 Rocket...
- Export...**
- Hide
- Isolate...
- Make transparent...
- Add comment
- Zoom to selection
- Delete...



## Export

**File name** [View export rules](#)

CFD Model Simplified - C6-5 Rocket

### Format

ACIS

### Options

Download

OK

Cancel



Rocket model

CFD Model Simplified

CFD Model Co

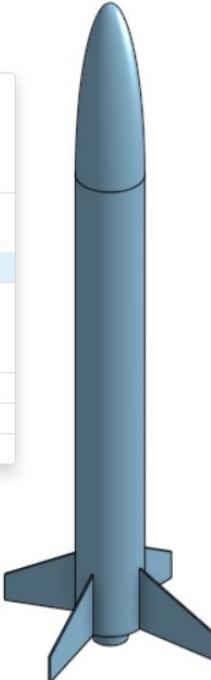
# Export Onshape

Default geometry

- Origin
- Top
- Front
- Right
- Sketch 1
- Sketch 2
- Sketch 3
- Sketch 4
- Sketch 5
- Revolve 1
- Extrude 1
- Circular pattern 1

Parts (1)

C6-5 Rocket



Rocket model CFD Model Simplified CFD Model Co

Contextual menu for C6-5 Rocket:

- Rename
- Properties...
- Assign material...
- Edit appearance...
- Copy...
- Create Drawing of C6-5 Rocket...
- Export...**
- Hide
- Isolate...
- Make transparent...
- Add comment
- Zoom to selection
- Delete...

## Export

**File name** [View export rules](#)

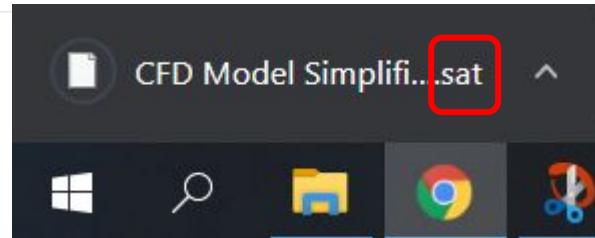
CFD Model Simplified - C6-5 Rocket

### Format

ACIS

### Options

Download

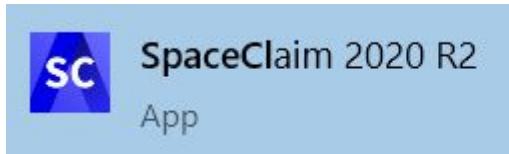


OK

Cancel

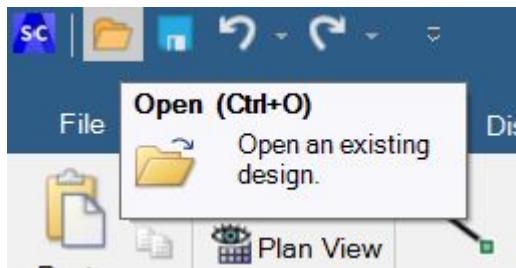
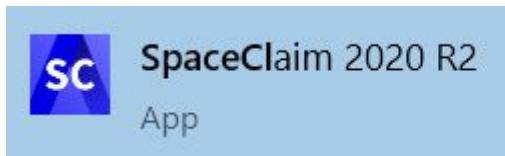


# Export Onshape to SpaceClaim



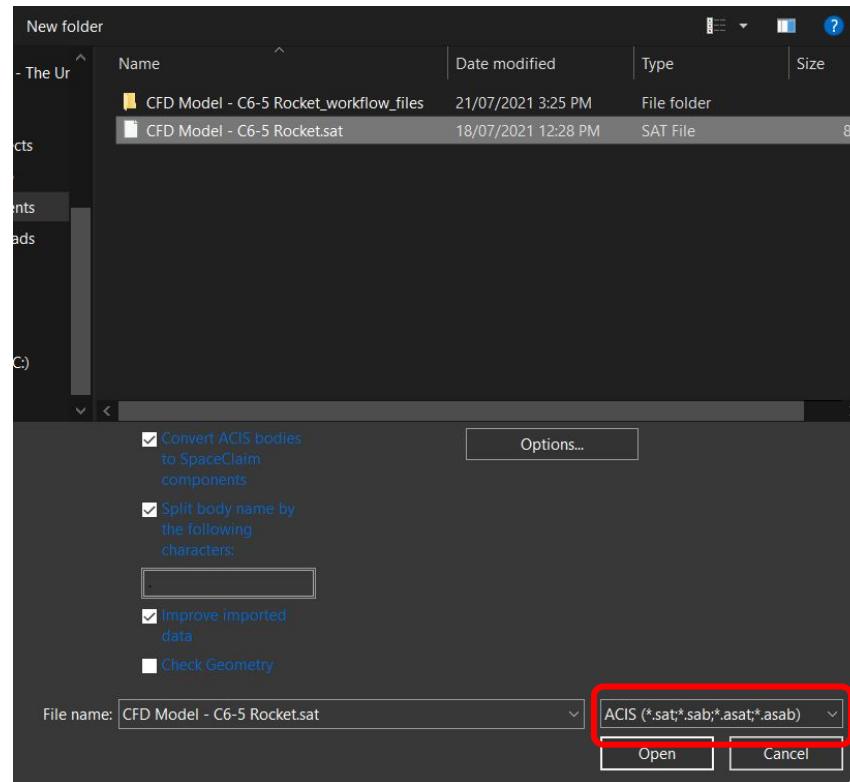
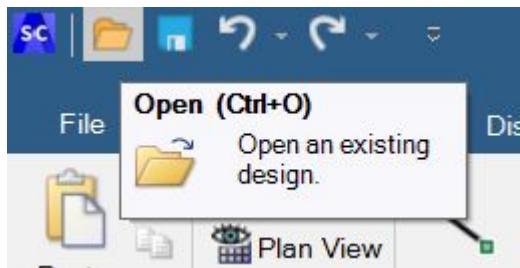
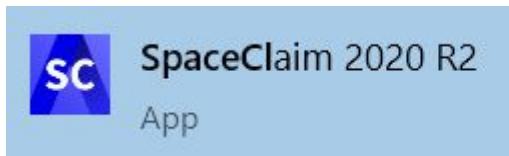


# Export Onshape to SpaceClaim



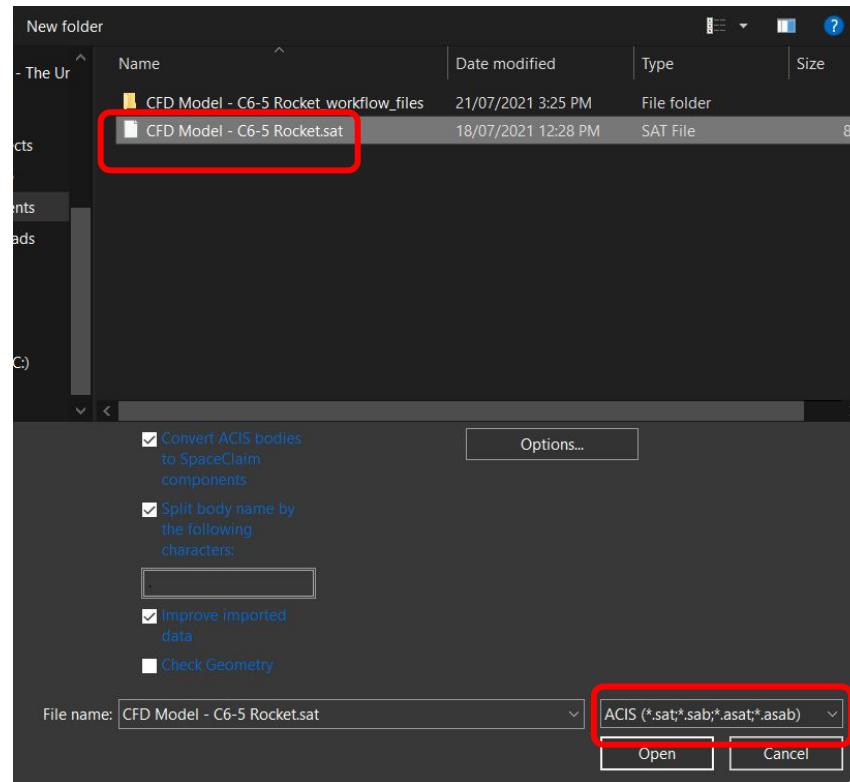
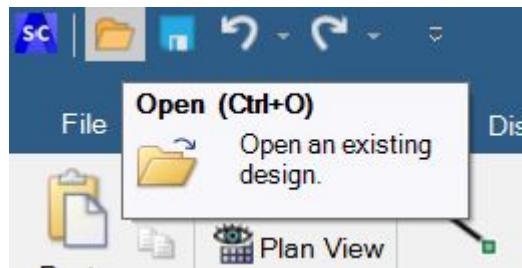
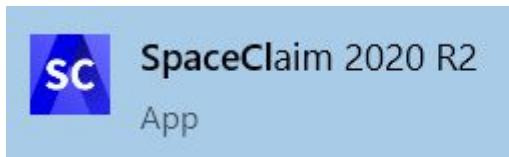


# Export Onshape to SpaceClaim



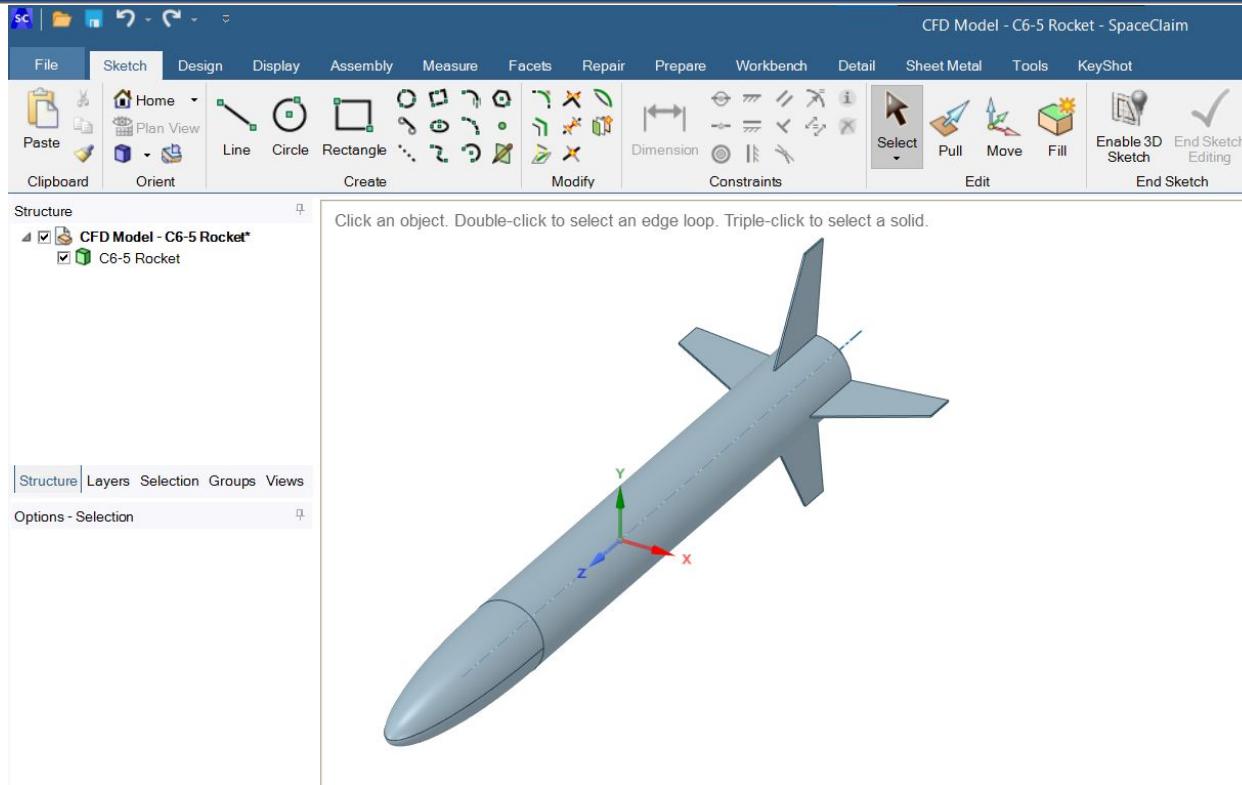


# Export Onshape to SpaceClaim



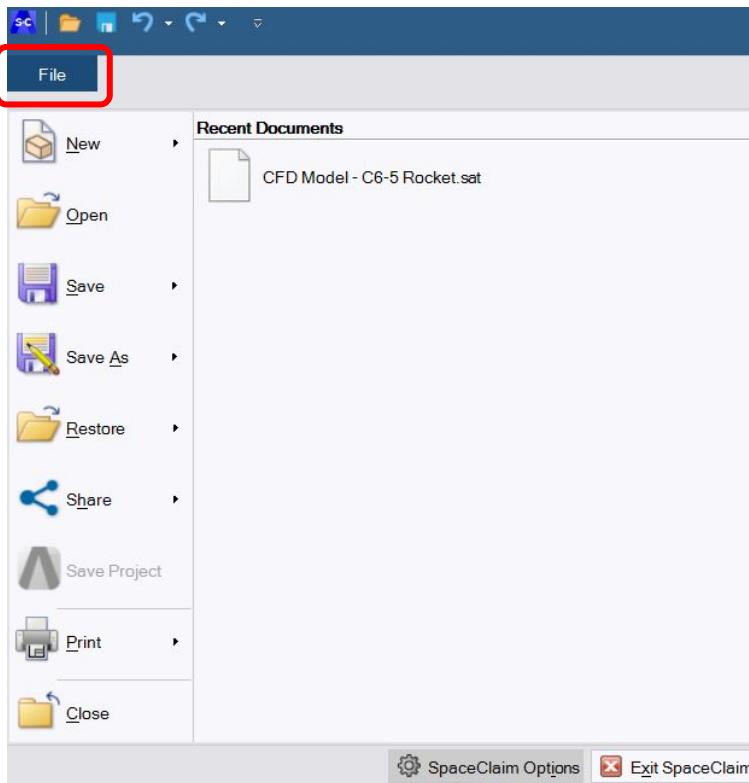


# Export Onshape to SpaceClaim



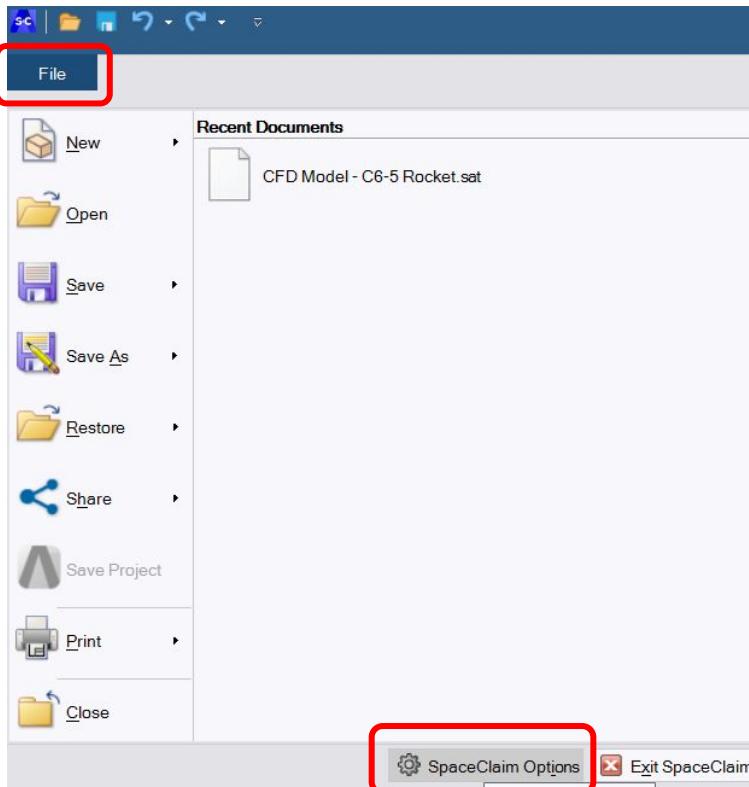


# SpaceClaim Mouse Controls



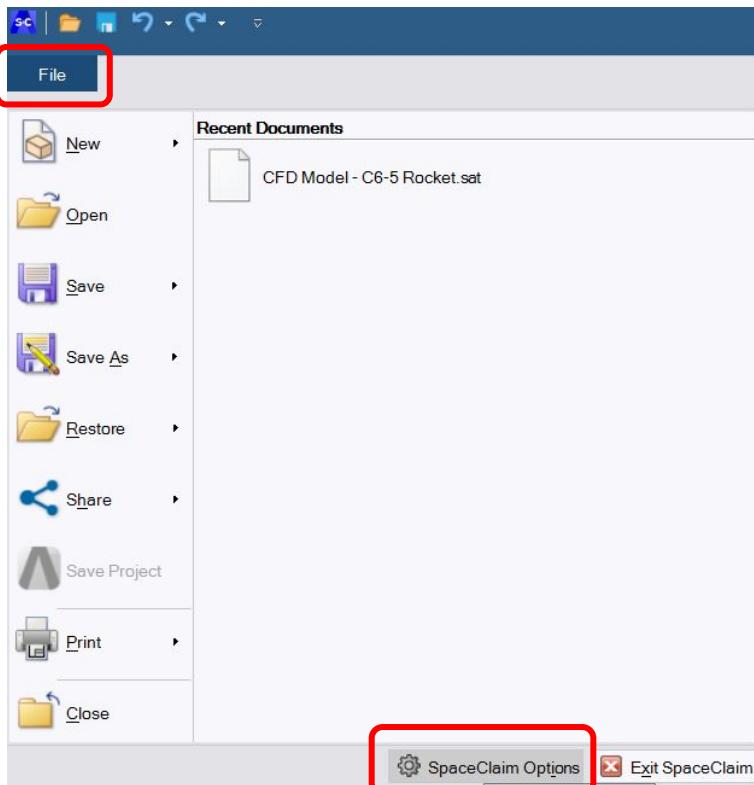


# SpaceClaim Mouse Controls





# SpaceClaim Mouse Controls



SpaceClaim 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

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

**Zoom**

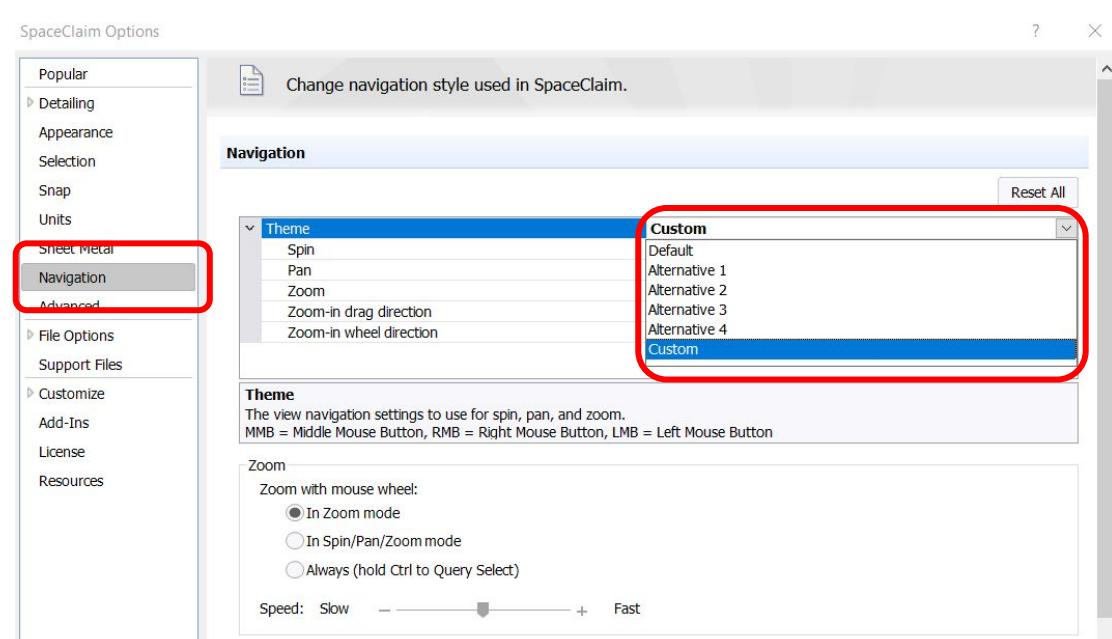
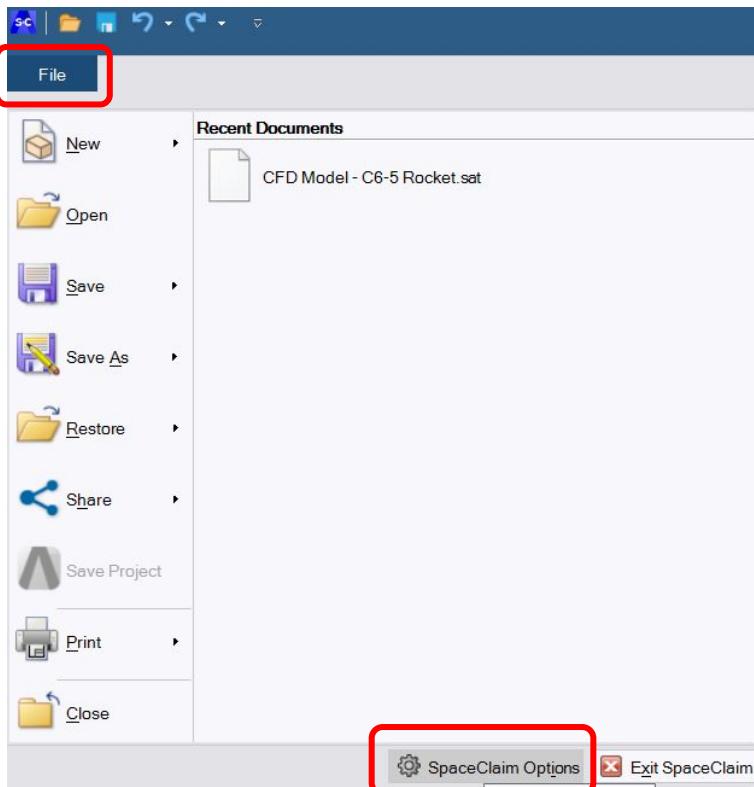
Zoom with mouse wheel:

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

Speed: Slow —  — Fast



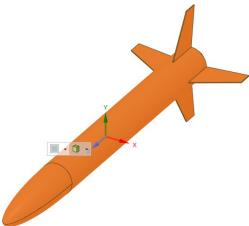
# SpaceClaim Mouse Controls



SpaceClaim Options

Exit SpaceClaim

# Fix Origin and Orientation

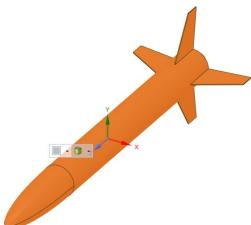


Want gravity (**z**) pointing down  
the rocket

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

Triple click to select the entire  
rocket

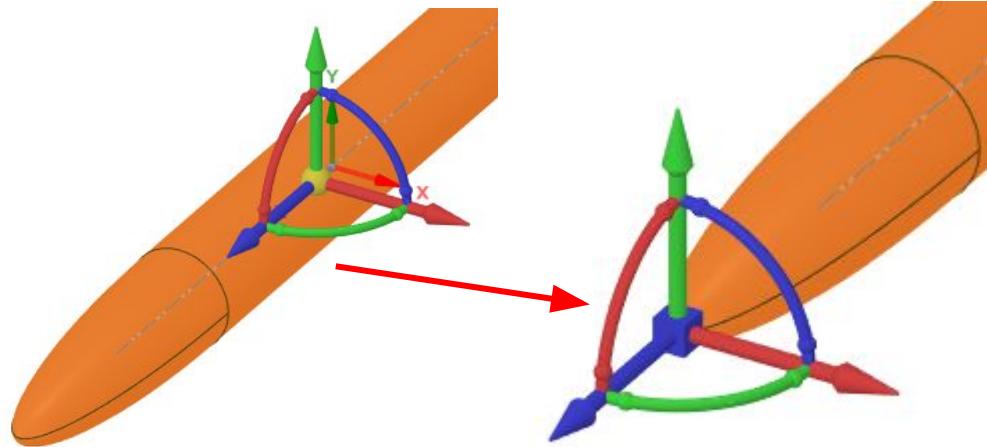
# Fix Origin and Orientation



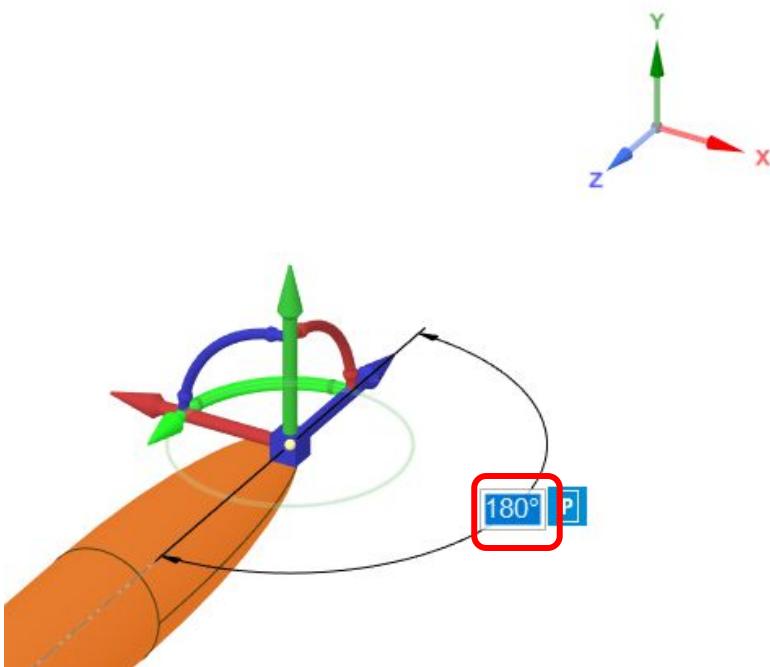
Want gravity (**z**) pointing down  
the rocket

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

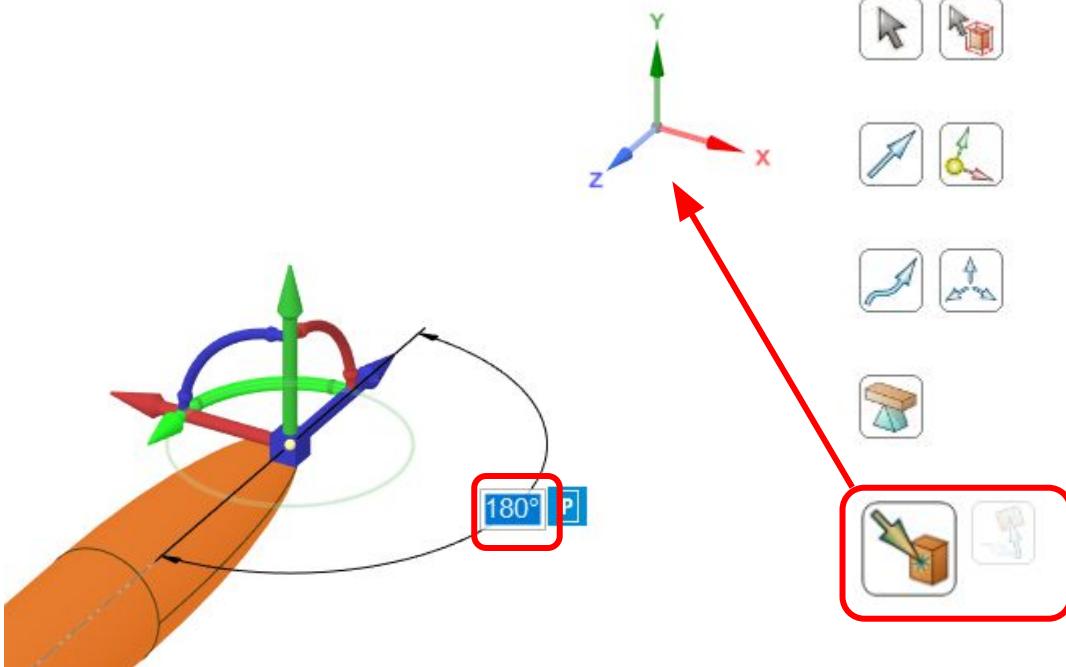
Triple click to select the entire  
rocket



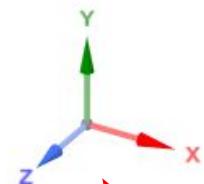
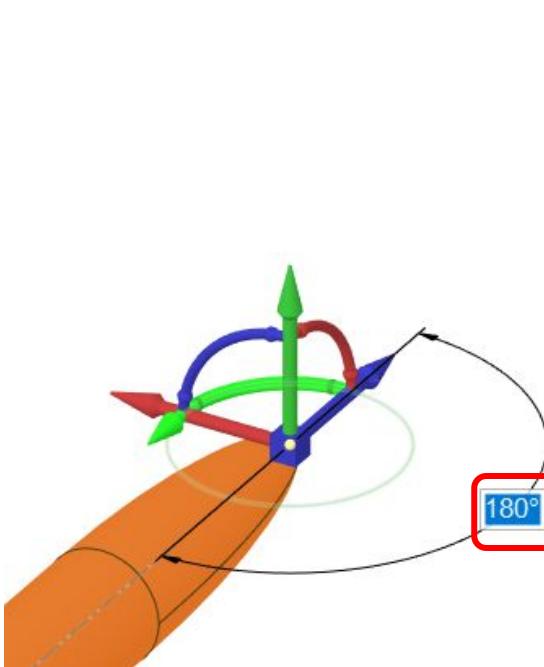
## Fix Origin and Orientation



# Fix Origin and Orientation



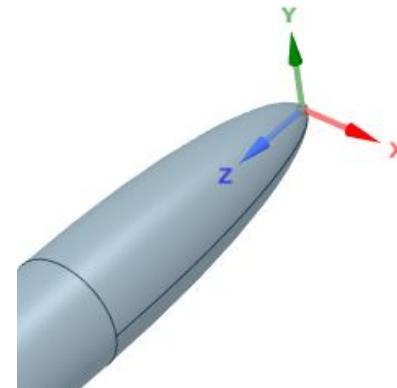
# Fix Origin and Orientation



Click a reference to translate up to



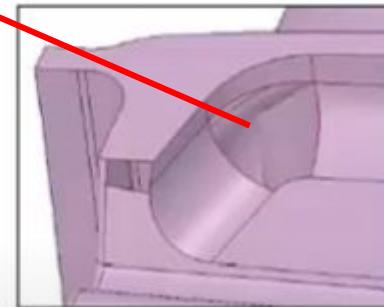
Press **esc** to leave the Move Tool



# Cleanup - SpaceClaim



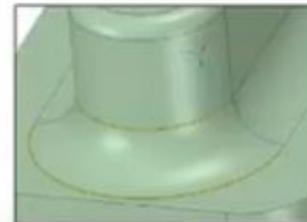
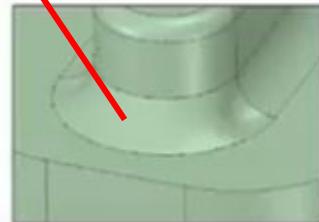
Missing faces



## Cleanup - SpaceClaim



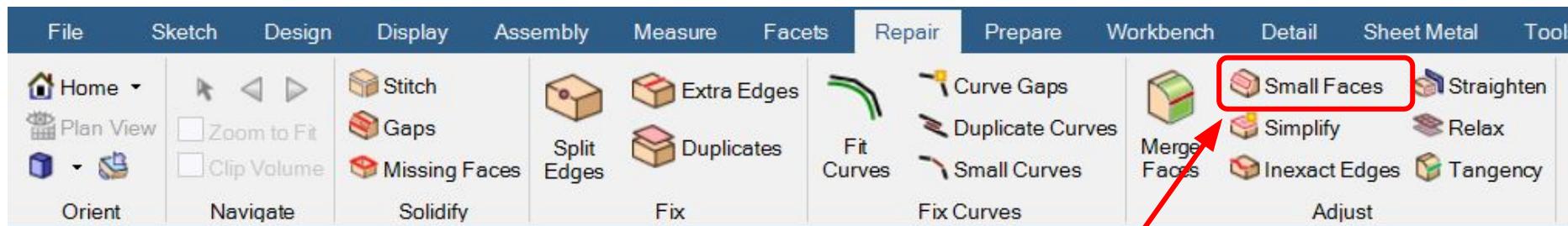
Split edges



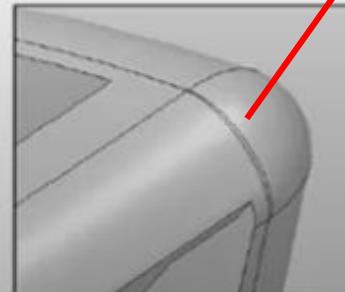
# Cleanup - SpaceClaim



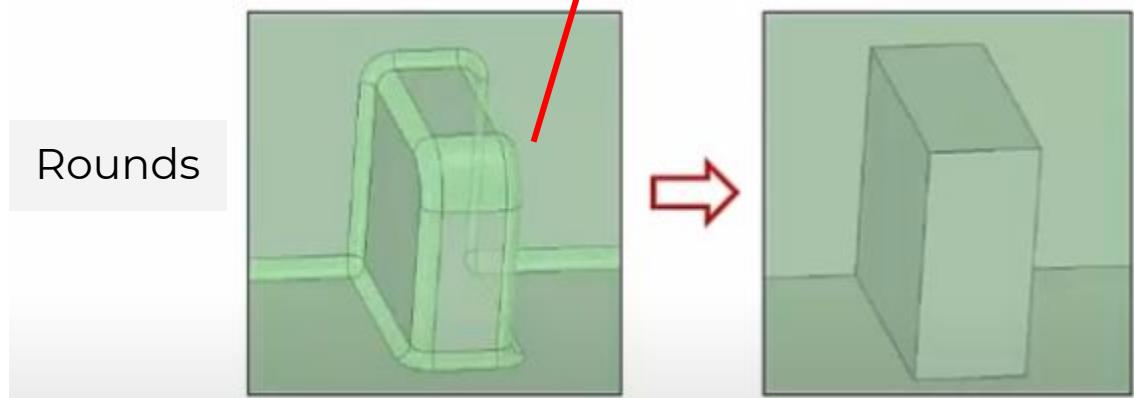
# Cleanup - SpaceClaim



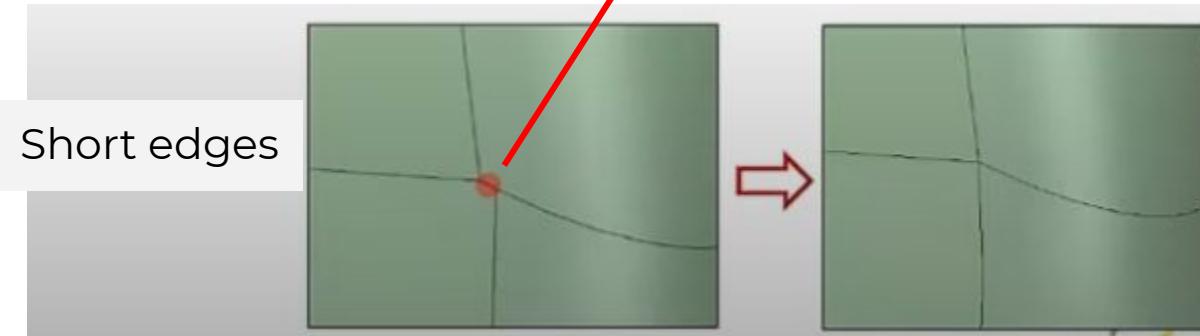
Small faces



# Cleanup - SpaceClaim

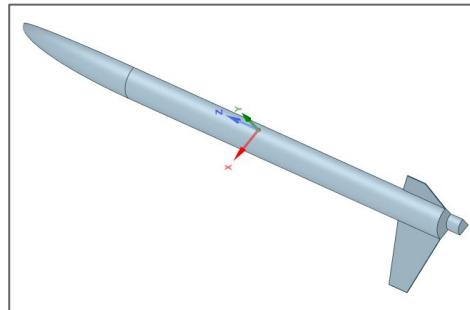
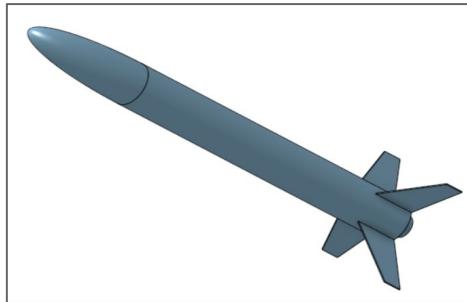


# Cleanup - SpaceClaim



## Cleanup - SpaceClaim

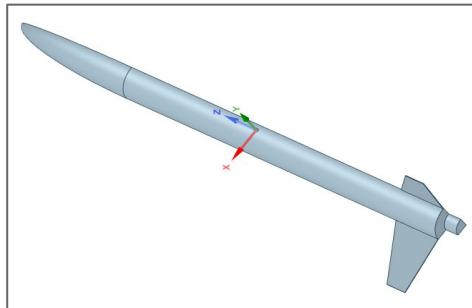
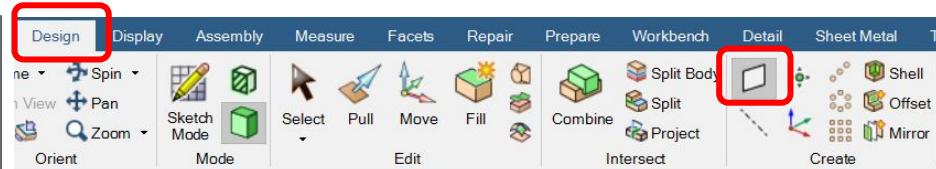
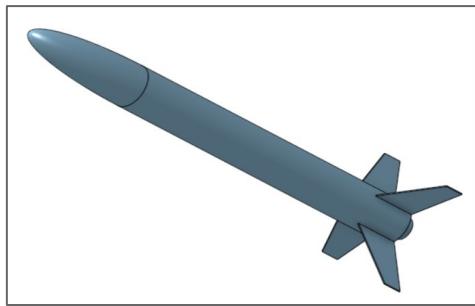
Symmetric flow?



# Cleanup - SpaceClaim

Symmetric flow?

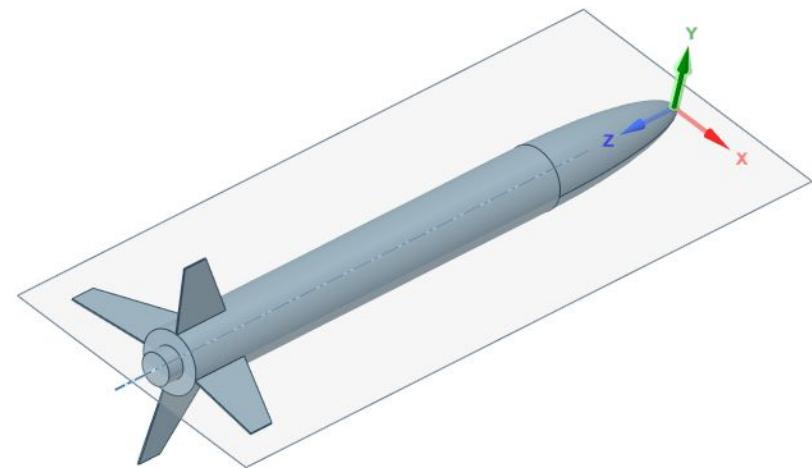
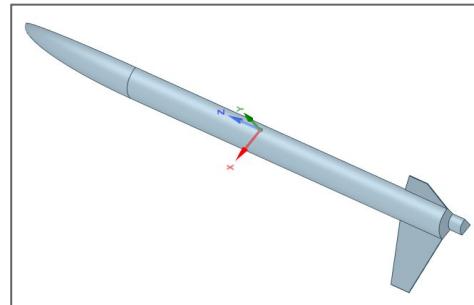
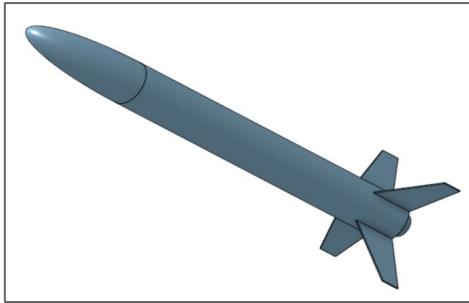
Use the **plane** tool  
to split the rocket



# Cleanup - SpaceClaim

Symmetric flow?

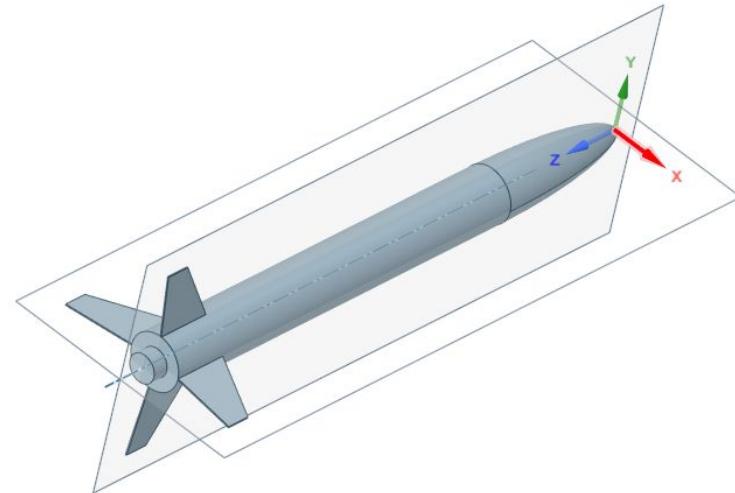
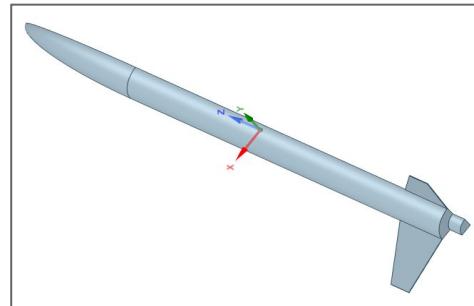
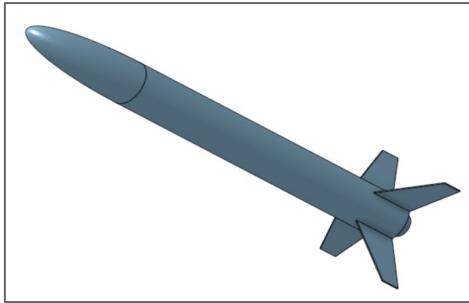
Use the plane tool  
to split the rocket



# Cleanup - SpaceClaim

Symmetric flow?

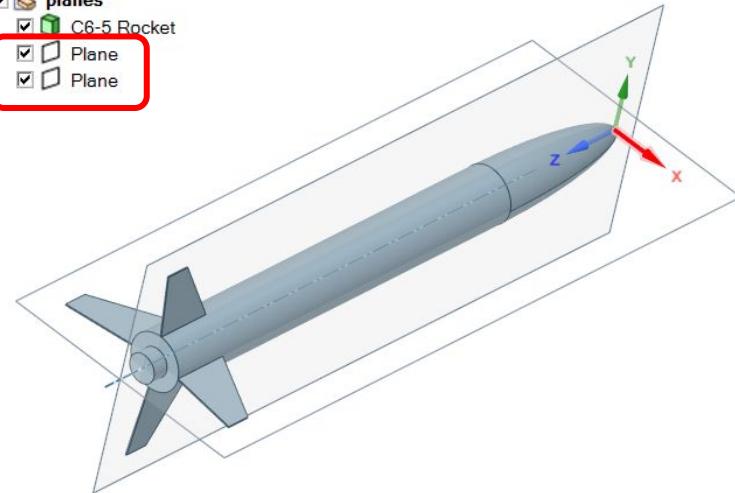
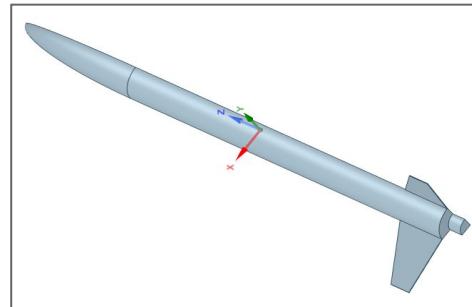
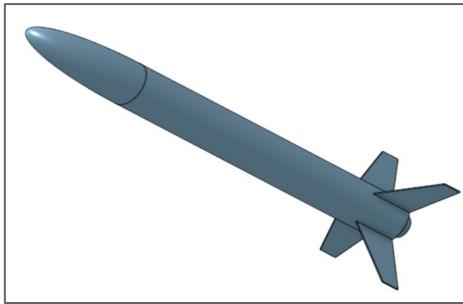
Use the plane tool  
to split the rocket



# Cleanup - SpaceClaim

Symmetric flow?

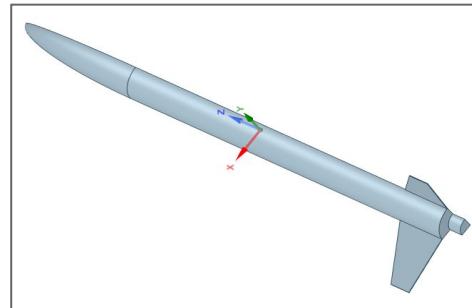
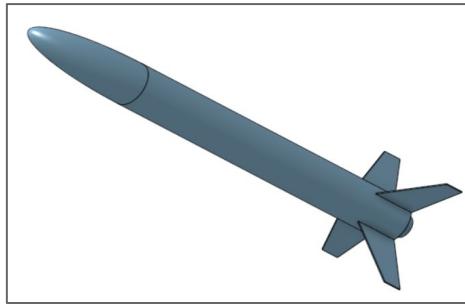
Use the plane tool  
to split the rocket



# Cleanup - SpaceClaim

Symmetric flow?

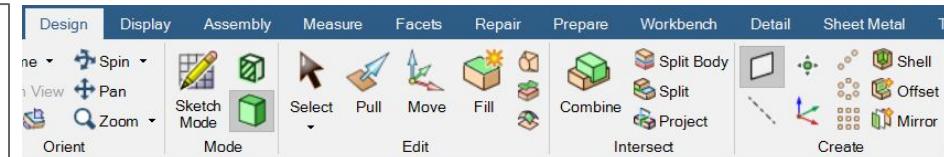
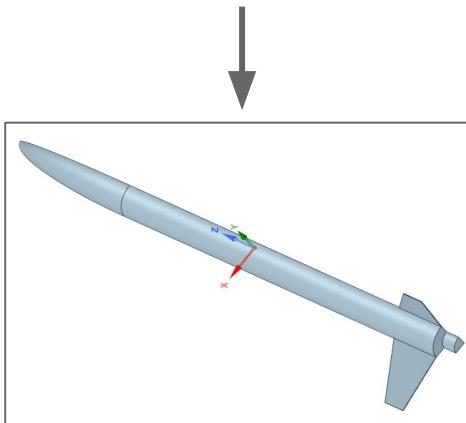
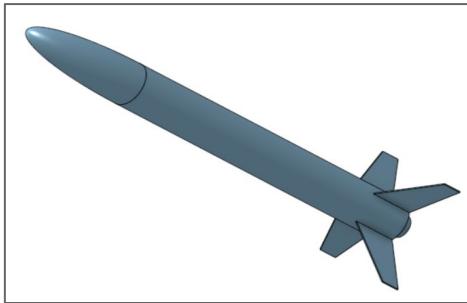
Use the **split body** tool



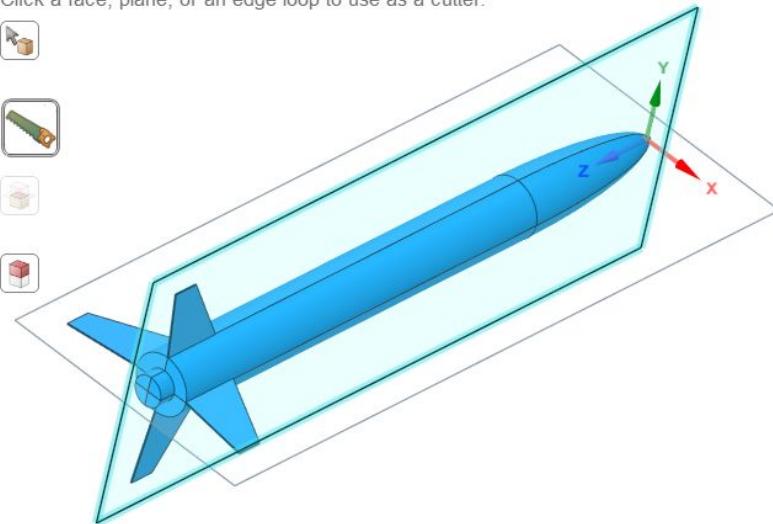
# Cleanup - SpaceClaim

Symmetric flow?

Use the **split body** tool



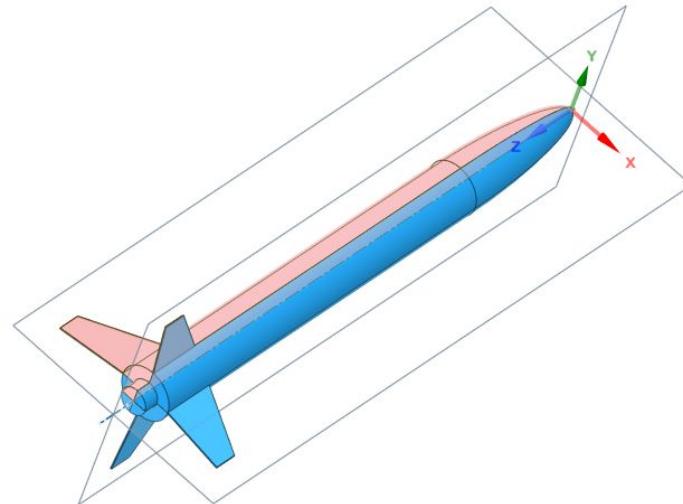
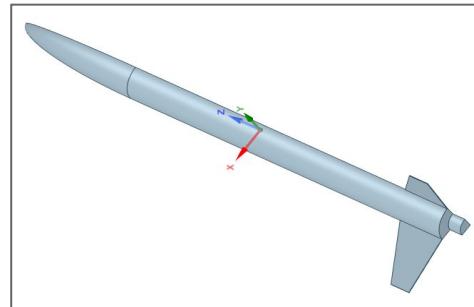
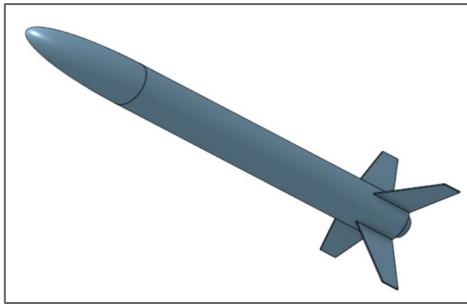
Click a face, plane, or an edge loop to use as a cutter.



# Cleanup - SpaceClaim

Symmetric flow?

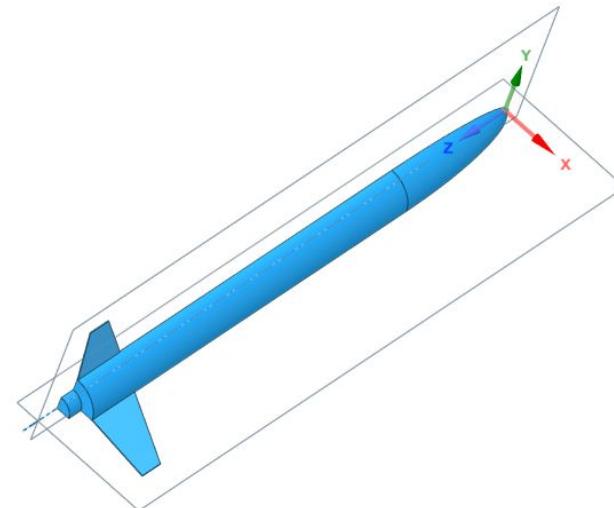
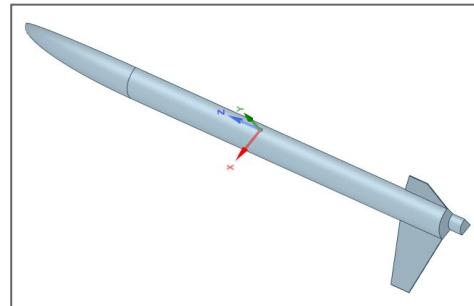
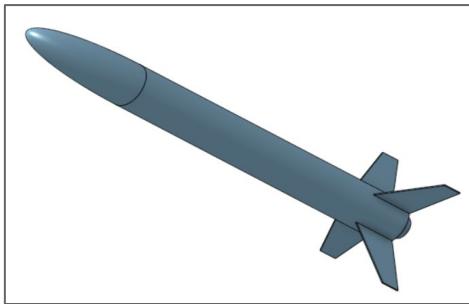
Use the **split body** tool



# Cleanup - SpaceClaim

Symmetric flow?

Use the **split body** tool



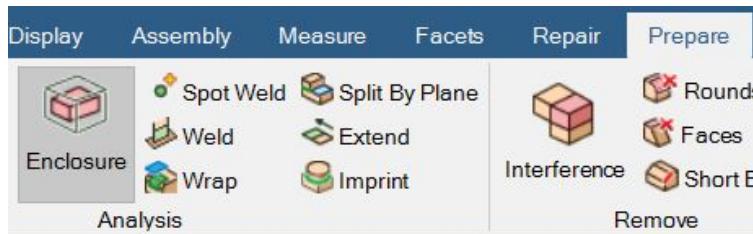
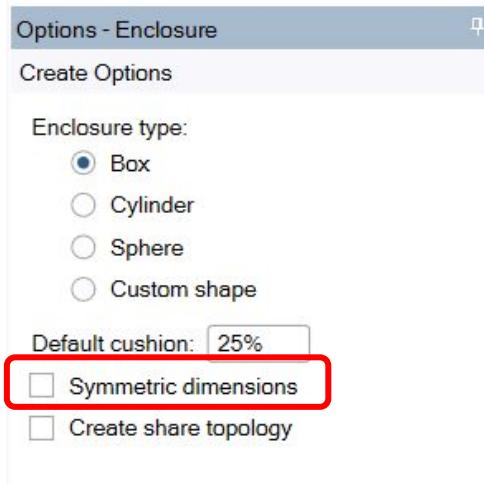
# Enclosure

## L - length of the model

Inlet - large enough to not be affected by stagnation region

Outlet - large enough to capture the wake and turbulent regions

Rule of thumb shown



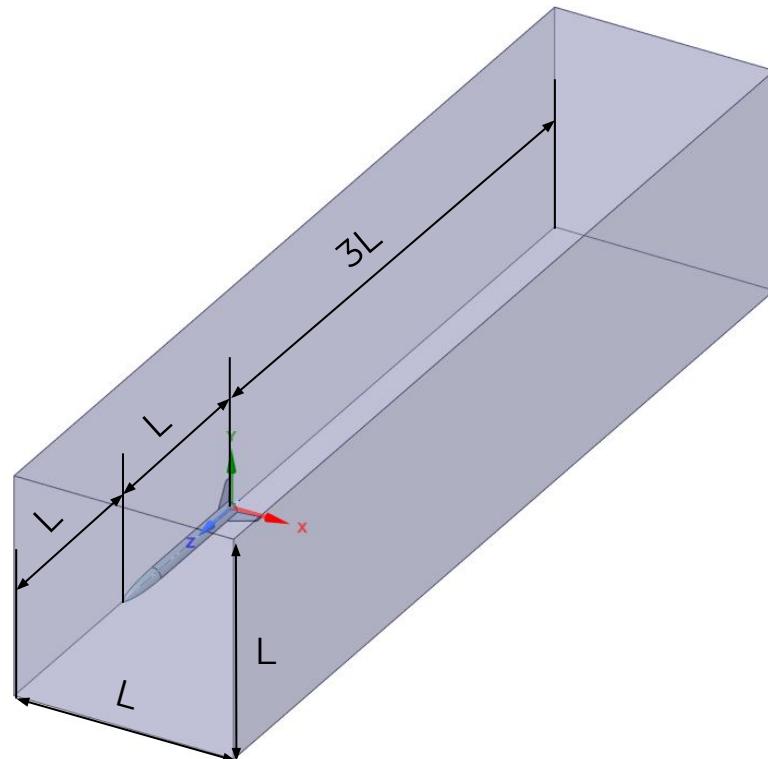
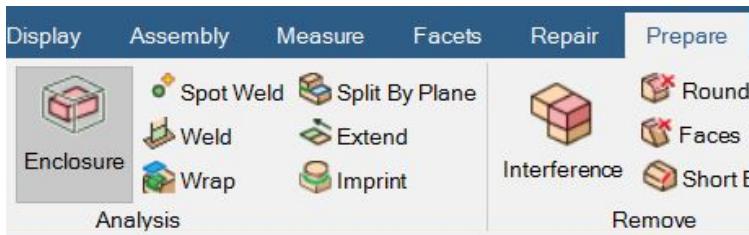
# Enclosure

## L - length of the model

Inlet - large enough to not be affected by stagnation region

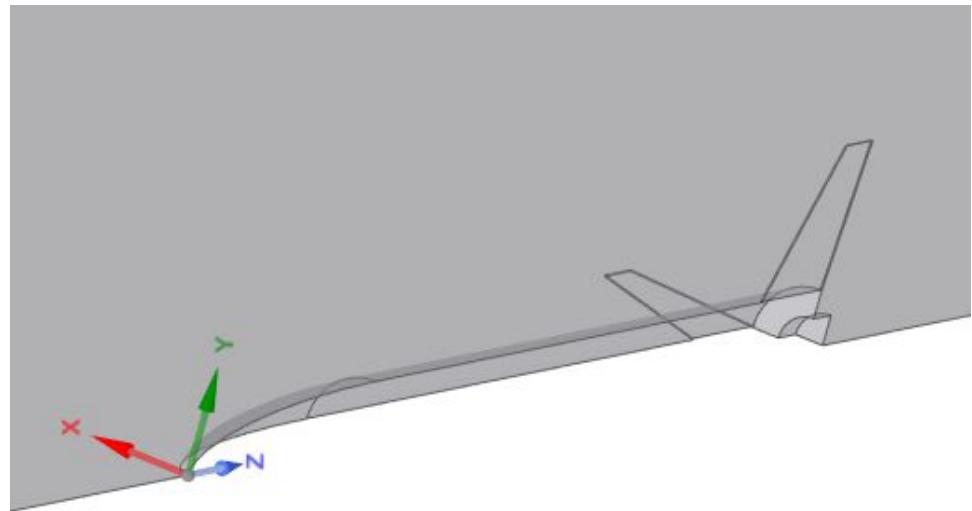
Outlet - large enough to capture the wake and turbulent regions

Rule of thumb shown



# Enclosure

Note that the enclosure  
**doesn't include the rocket** -  
only the air around it



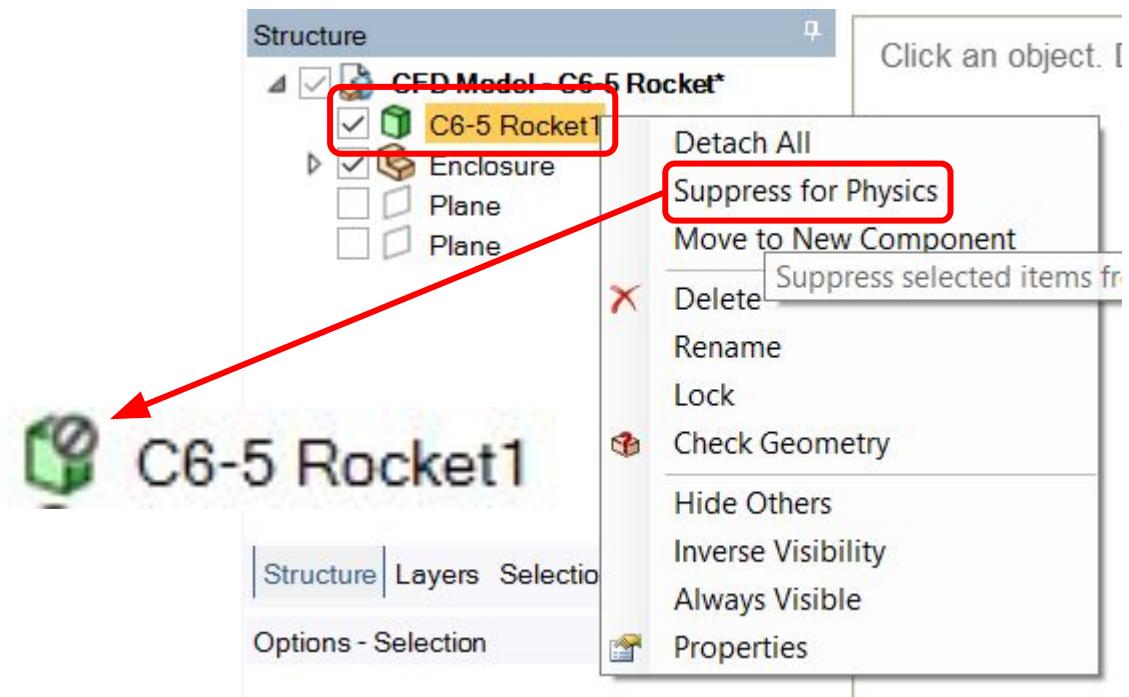
# Suppress for physics

The rocket is a **solid**

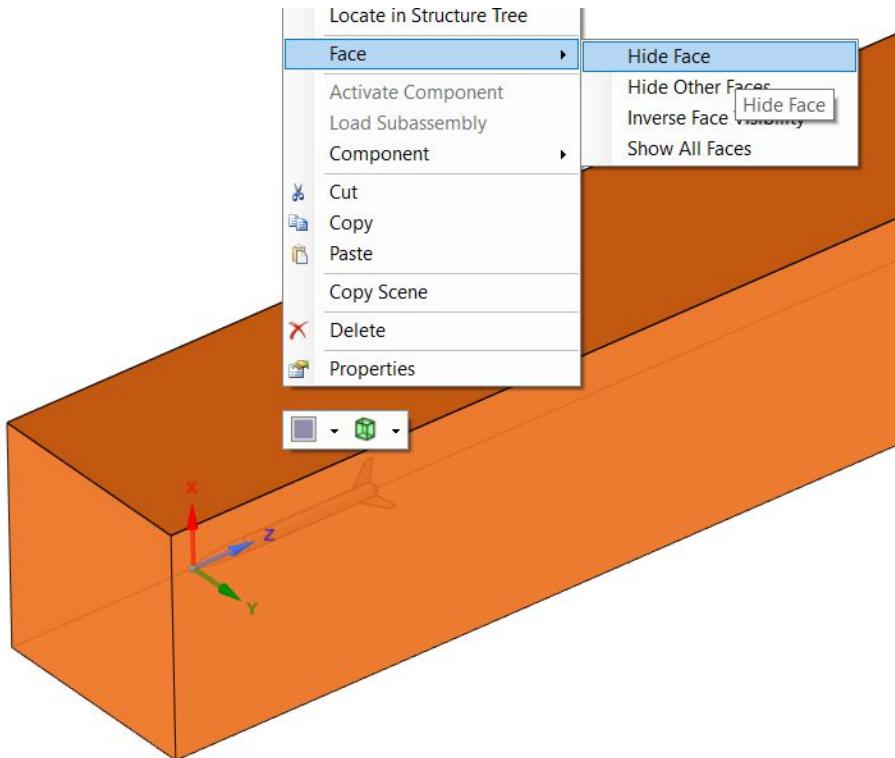
The enclosure is a **fluid** pool

**CFD only wants to model  
the fluids**

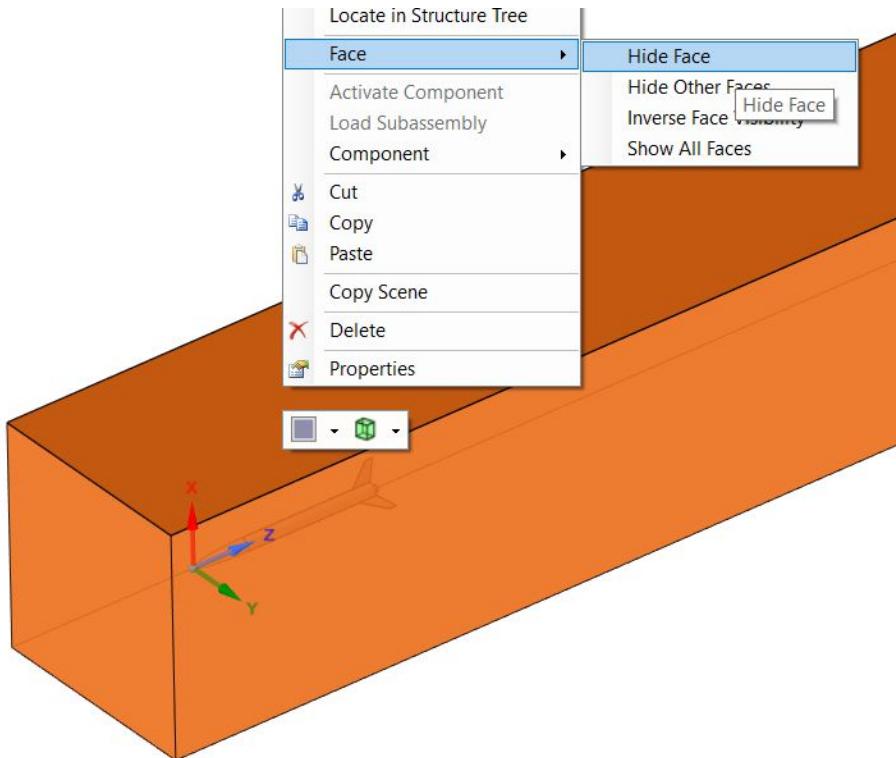
Suppress the rocket for  
physics - so it doesn't get  
passed to Fluent CFD



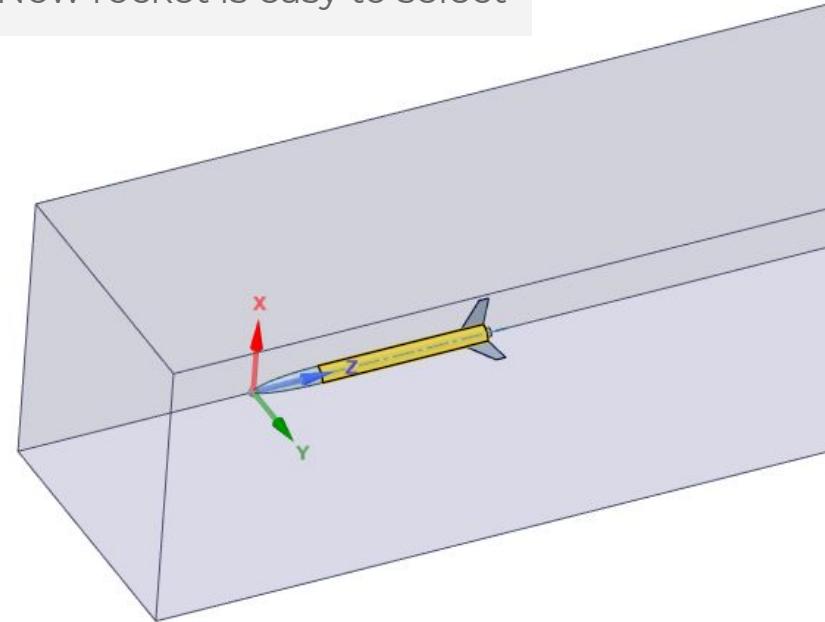
# Hide annoying faces



# Hide annoying faces



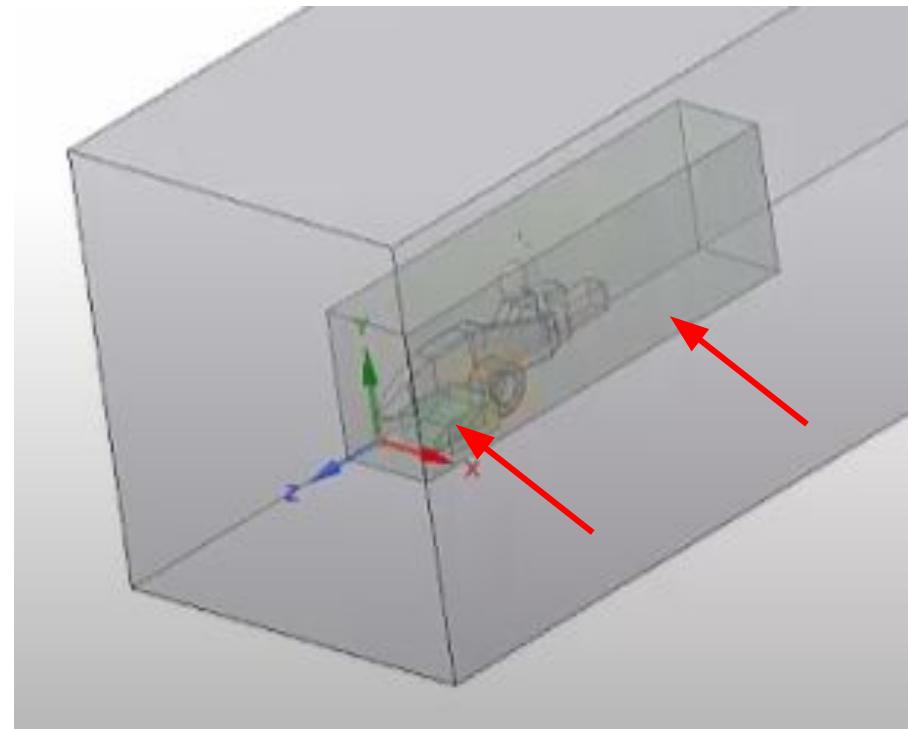
Now rocket is easy to select



# Body of Influence

Create a region of finer mesh for more important regions of the flow

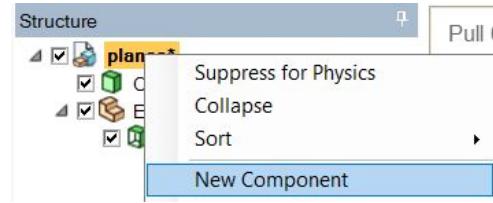
- Around the model **including the wake** region
- Around complex geometry (e.g. fins and nose)



# Body of Influence

Make a new component and name it

**boi-far** (BOI = Body Of Influence)

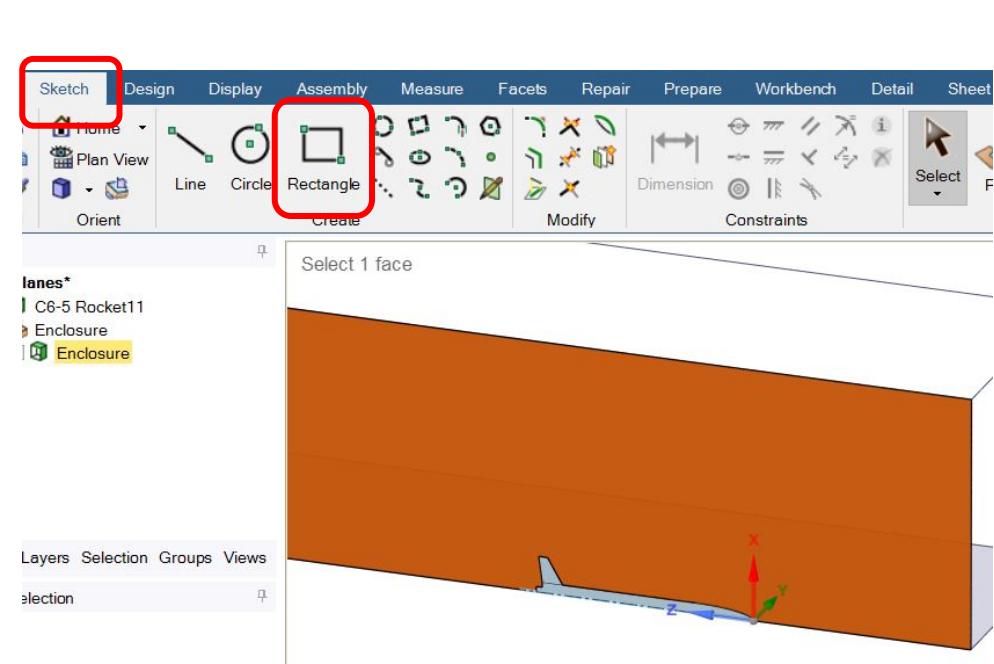


# Body of Influence

Make a new component and name it

**boi-far** (BOI = Body Of Influence)

Select one face and sketch a rectangle  
around the rocket

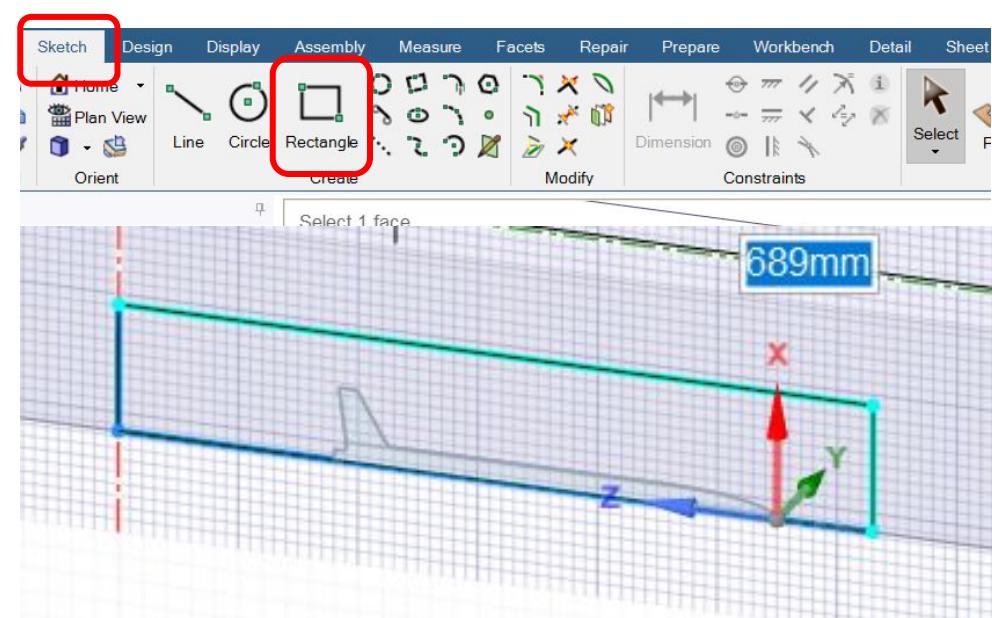


# Body of Influence

Make a new component and name it

**boi-far** (BOI = Body Of Influence)

Select one face and sketch a rectangle  
around and **behind the rocket**

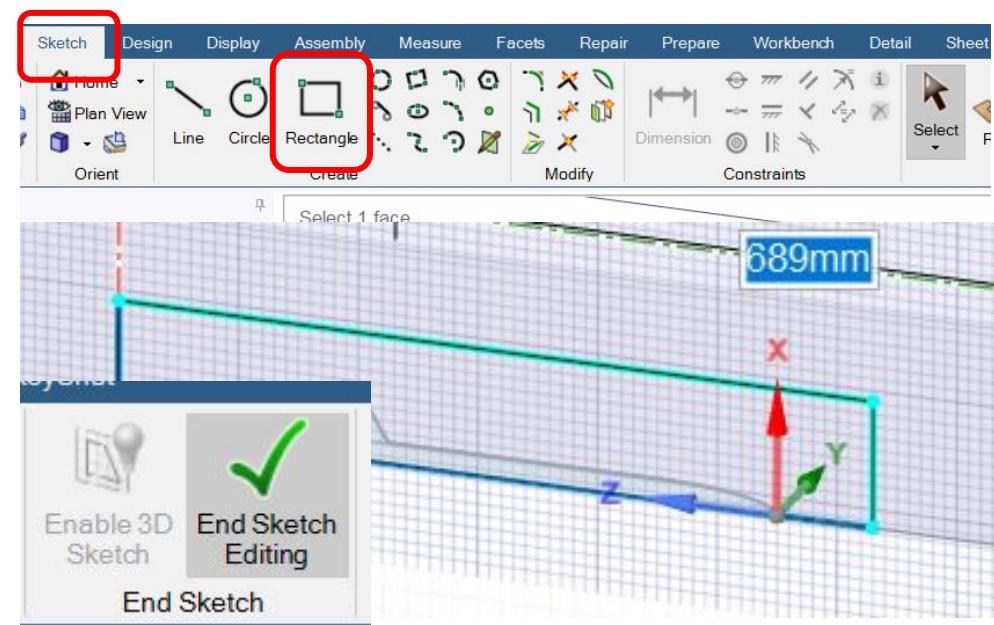


# Body of Influence

Make a new component and name it

**boi-far** (BOI = Body Of Influence)

Select one face and sketch a rectangle  
around the rocket (end sketch)



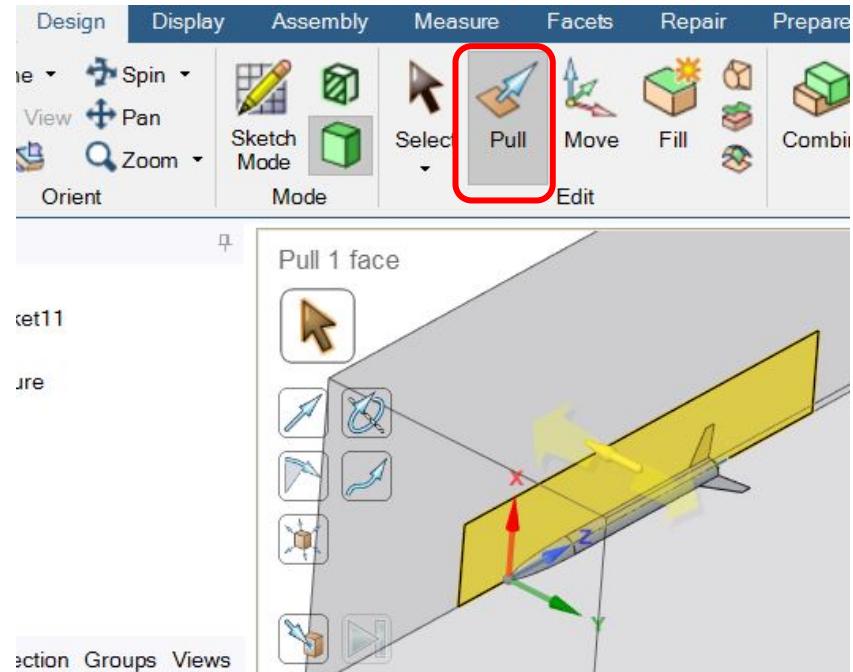
# Body of Influence

Make a new component and name it

**boi-far** (BOI = Body Of Influence)

Select one face and sketch a rectangle  
around the rocket (end sketch)

Pull the face to make a cuboid around the  
rocket



# Body of Influence

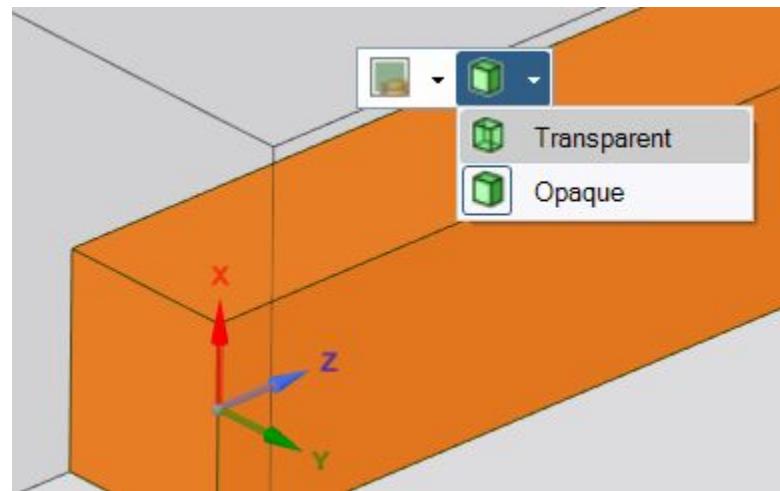
Make a new component and name it

**boi-far** (BOI = Body Of Influence)

Select one face and sketch a rectangle  
around the rocket (end sketch)

Pull the face to make a cuboid around the  
rocket

Triple click and make the BOI transparent



# Body of Influence

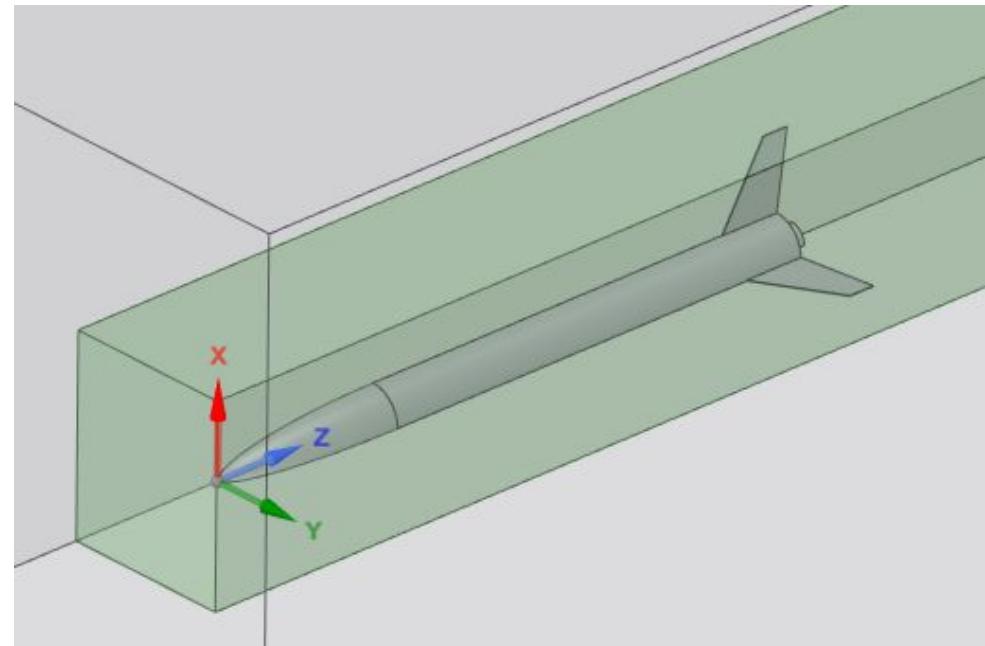
Make a new component and name it

**boi-far** (BOI = Body Of Influence)

Select one face and sketch a rectangle  
around the rocket (end sketch)

Pull the face to make a cuboid around the  
rocket

Triple click and make the BOI transparent

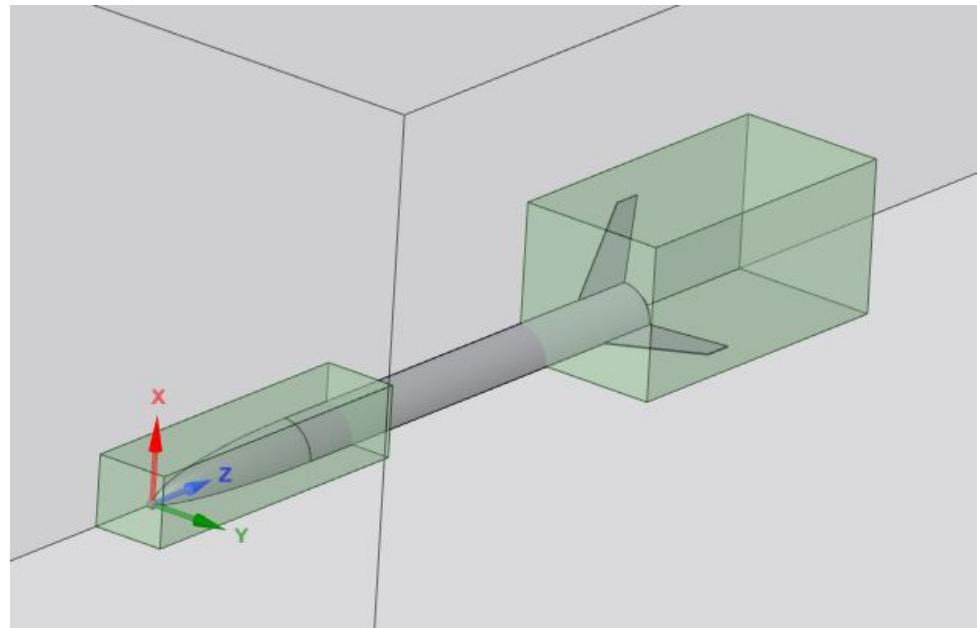


# Body of Influence

Make a new BOI component and name it  
**boi-near**

It should surround the nose and the fins

Even finer mesh around these areas later



# Named Selections

**Named selections** are very important later on in Fluent - so make them now!

- Create a named selection in SpaceClaim for **any part you might want to create special meshing conditions** for (e.g. BOIs, curved parts, small faces)
- Create a named selection for **any component you want to set boundary conditions** on (e.g. walls, surfaces, symmetry planes, inlets outlets)

# Named Selections

\*Have to include  
every face once  
and only once



**Named selections** are very important later on in Fluent - so make them now!

- Create a named selection in SpaceClaim for **any part you might want to create special meshing conditions** for (e.g. BOIs, curved parts, small faces)
- Create a named selection for **any component you want to set boundary conditions** on (e.g. walls, surfaces, symmetry planes, inlets outlets)

# Named Selections

\*Have to include  
every face once  
and only once



Let's start with **BOIs**

Structure

- planes\*
- C6-5 Rocket11
- Enclosure
- Enclosure
- boi-far
- boi-near

Groups

Create NS Create Parameter

Create NS (Ctrl+G)

Name

Select 1 face

Triple click BOI

Structure Layers Selection Groups Views

A screenshot of a CAD software interface showing a 3D model of a rectangular block. The block is orange and has a coordinate system (X, Y, Z) attached to its top surface. A callout box on the right side of the interface is labeled "Triple click BOI" and points to the top surface of the block. The interface includes a "Structure" tree view, a "Groups" panel with a "Create NS" button, and a status bar at the bottom.

# Named Selections

\*Have to include  
every face once  
and only once



Let's start with **BOIs**

Structure

- planes\*
  - C6-5 Rocket11
- Enclosure
  - Enclosure
- boi-far
  - boi-far
  - boi-near

Groups

Create NS

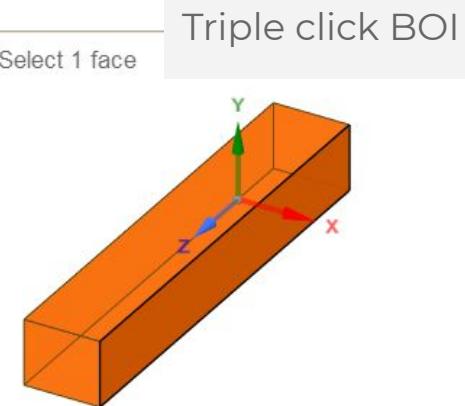
Create Parameter

Create NS (Ctrl+G)

Name

Select 1 face

Structure Layers Selection Groups Views



# Named Selections

\*Have to include  
every face once  
and only once

Let's start with **BOIs**

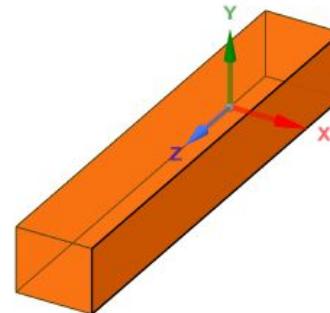
Structure



Name



Select 1 face

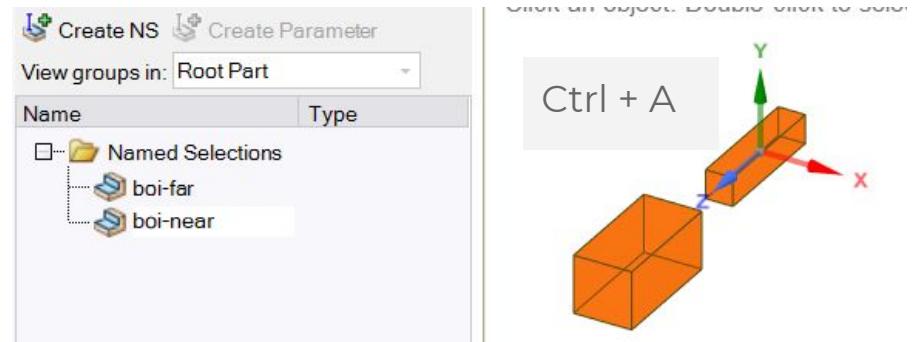


# Named Selections

\*Have to include  
every face once  
and only once



Let's start with **BOIs**



# Named Selections

\*Have to include  
every face once  
and only once



Let's start with BOIs

Now let's do **curved surfaces**



# Named Selections

\*Have to include  
every face once  
and only once



Let's start with BOIs

Now let's do **curved surfaces**

The screenshot shows the ARES CAD software interface. On the left, there is a tree view of named selections:

Name	Type
planes*	Component
C6-5 Rocket11	Component
Enclosure	Named Selection (highlighted)
boi-far	Component
boi-near	Component

A context menu is open over the "Enclosure" selection, listing options: "Suppress for Physics", "Collapse", "New Component", "Move to New Component", and "Activate Component". The "Activate Component" option is highlighted with a blue background.

Below the tree view is a table showing the structure of the named selections:

Name	Type
Named Selections	Folder
boi-far	Component
boi-near	Component
body	Component

At the bottom, tabs for "Structure", "Layers", "Selection", "Groups", and "Views" are visible, with "Selection" being the active tab. The status bar at the bottom says "Options - Selection".

On the right, a 3D view shows a yellow cylindrical component labeled "body" and several smaller gray components. A dashed blue line is drawn across the yellow cylinder.

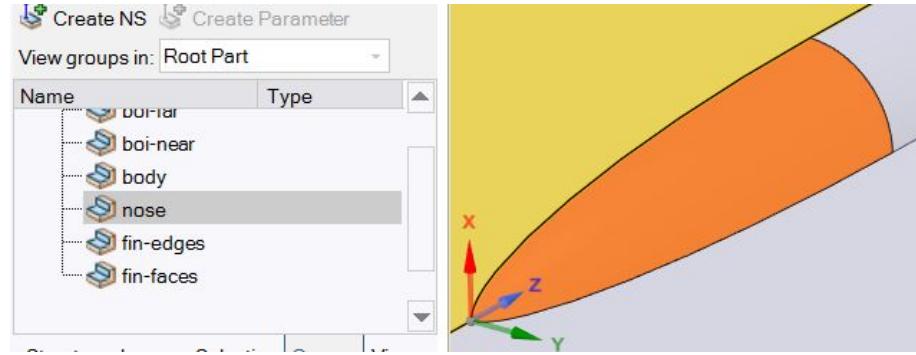
# Named Selections

\*Have to include  
every face once  
and only once



Let's start with BOIs

Now let's do **curved surfaces**



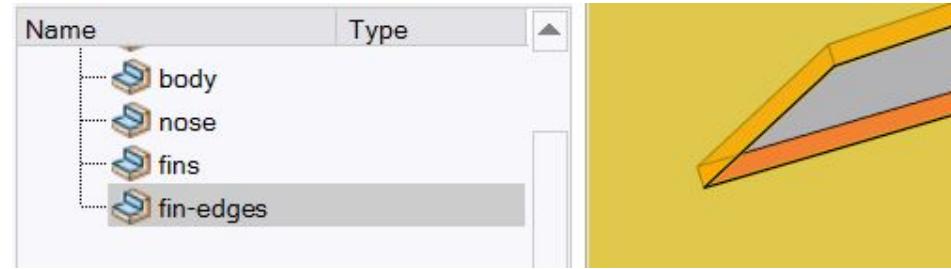
# Named Selections

\*Have to include  
every face once  
and only once



Let's start with BOIs

Now let's do curved surfaces and **small faces**



# Named Selections

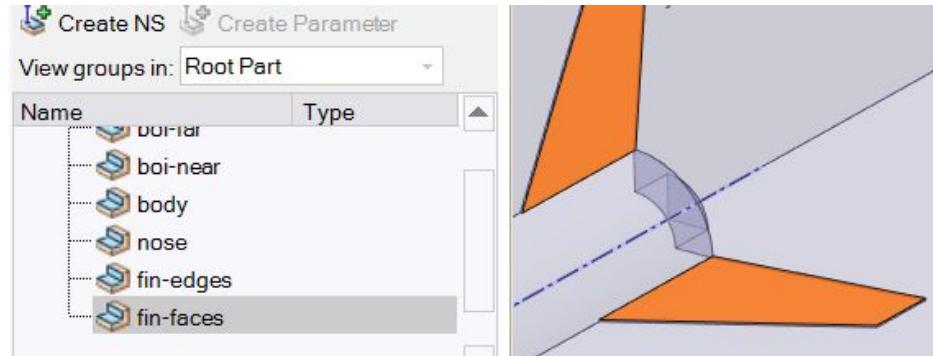
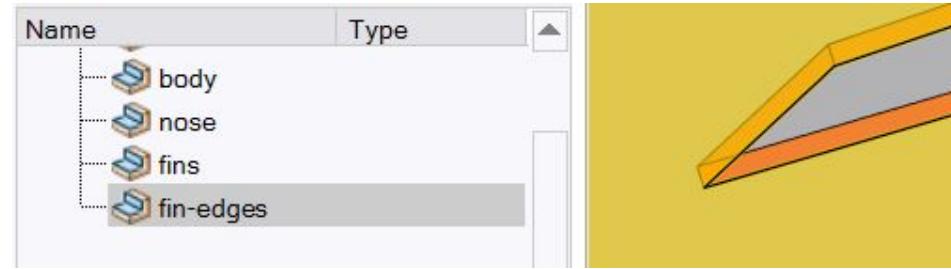
\*Have to include  
every face once  
and only once



Let's start with BOIs

Now let's do curved surfaces and small faces

Then the **remaining rocket faces**



# Named Selections

\*Have to include  
every face once  
and only once



Let's start with BOIs

Now let's do curved surfaces and small faces

Then the remaining rocket faces

Now let's do **Boundary Condition** surfaces:

# Named Selections

\*Have to include  
every face once  
and only once

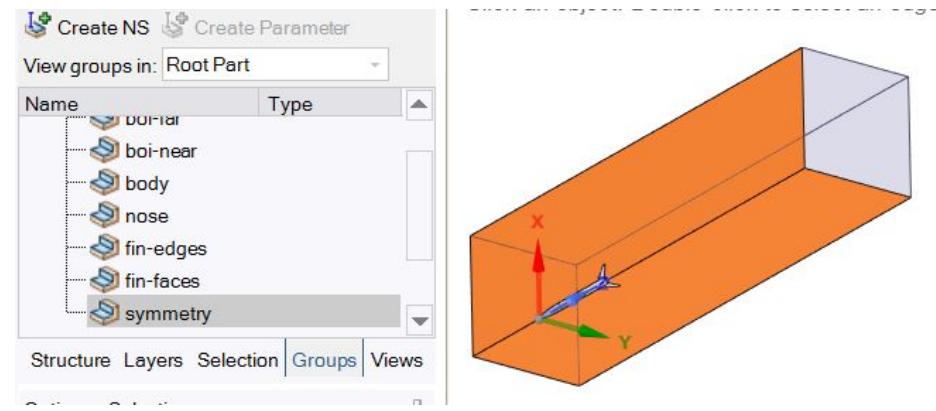


Let's start with BOIs

Now let's do curved surfaces and small faces  
Then the remaining rocket faces

Now let's do Boundary Condition surfaces:

- **Symmetry planes**



# Named Selections

\*Have to include  
every face once  
and only once



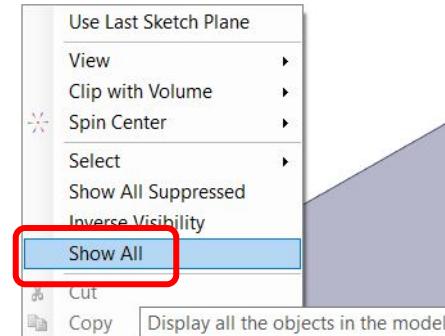
Let's start with BOIs

Now let's do curved surfaces and small faces

Then the remaining rocket faces

Now let's do Boundary Condition surfaces:

- Symmetry planes
- Inlet



# Named Selections

\*Have to include  
every face once  
and only once

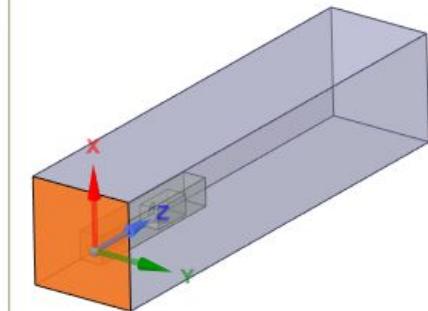
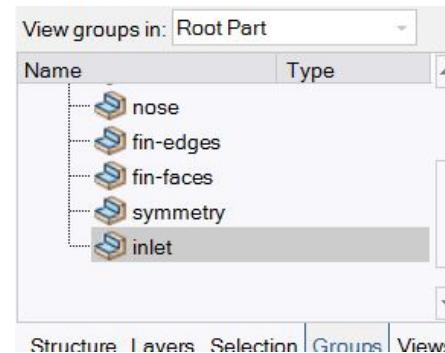
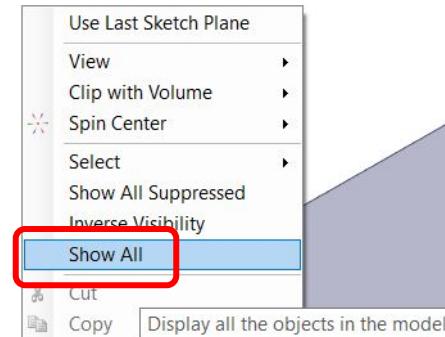


Let's start with BOIs

Now let's do curved surfaces and small faces  
Then the remaining rocket faces

Now let's do Boundary Condition surfaces:

- Symmetry planes
- Inlet



# Named Selections

\*Have to include  
every face once  
and only once



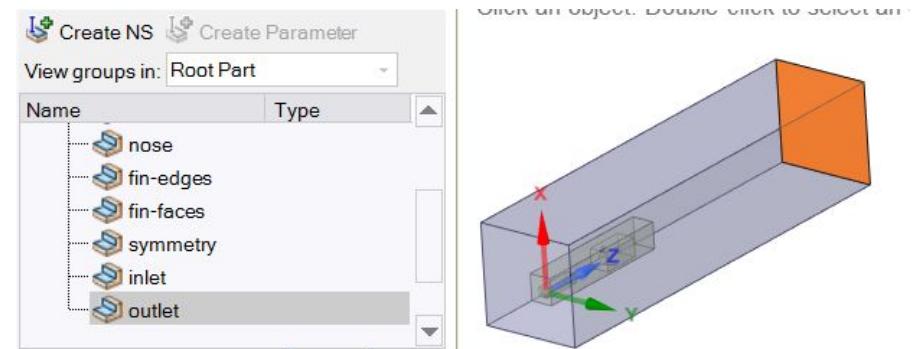
Let's start with BOIs

Now let's do curved surfaces and small faces

Then the remaining rocket faces

Now let's do Boundary Condition surfaces:

- Symmetry planes
- Inlet
- **Outlet**



# Named Selections

\*Have to include  
every face once  
and only once



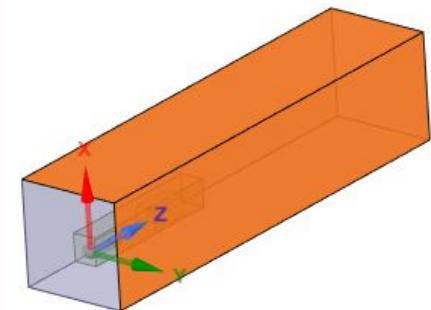
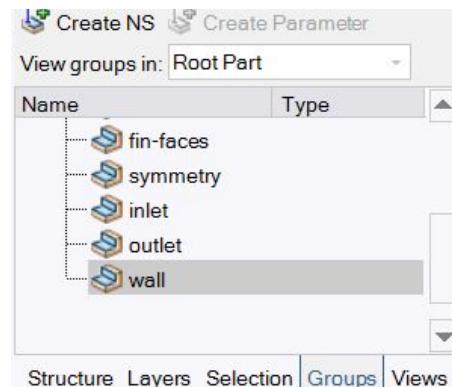
Let's start with BOIs

Now let's do curved surfaces and small faces

Then the remaining rocket faces

Now let's do Boundary Condition surfaces:

- Symmetry planes
- Inlet
- Outlet
- **Walls**



# Named Selections

\*Have to include  
every face once  
and only once



Let's start with BOIs

Now let's do curved surfaces and small faces

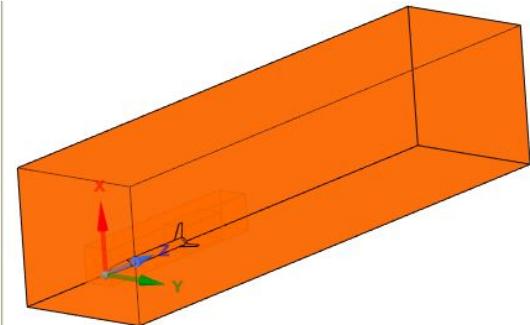
Then the remaining rocket faces

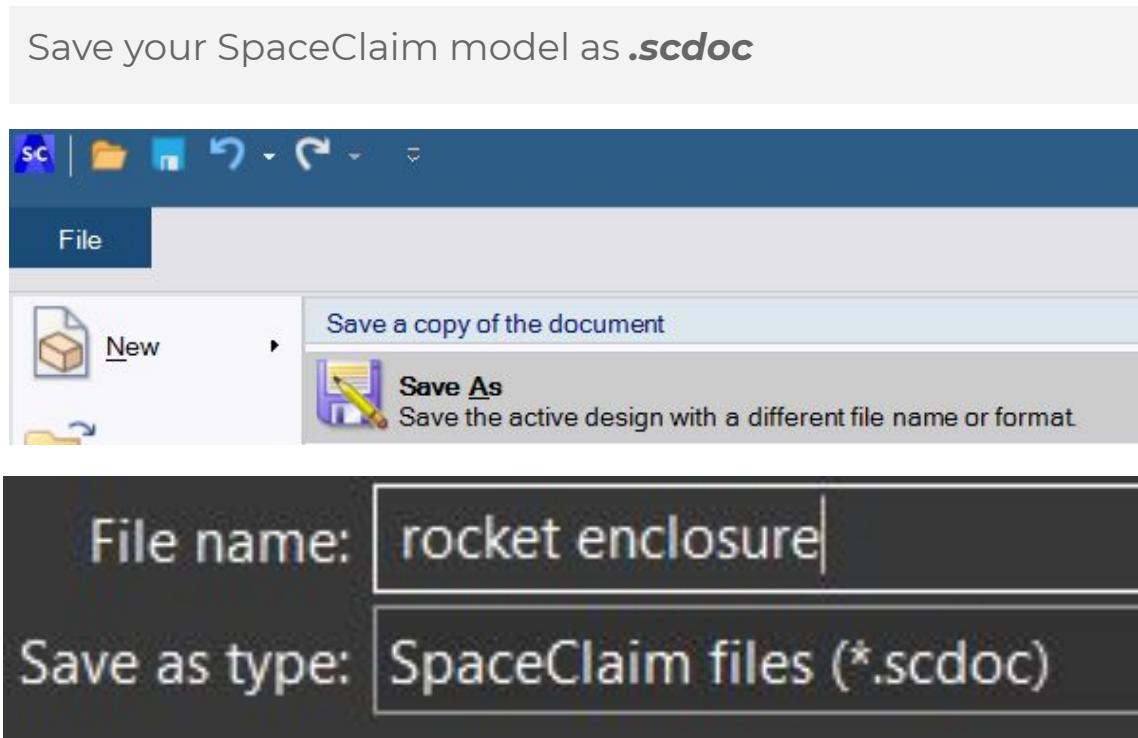
Now let's do Boundary Condition surfaces:

- Symmetry planes
- Inlet
- Outlet
- Walls

Now every face has been **selected once!**

Name	Type
Named Selections	Folder
boi-far	Face
boi-near	Face
body	Face
nose	Face
fin-edges	Edge
fin-faces	Face
symmetry	Plane
inlet	Face
outlet	Face
wall	Face





# Fluent



We're ready to open Fluent

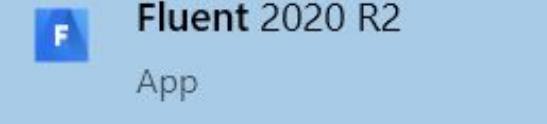


**Fluent** 2020 R2

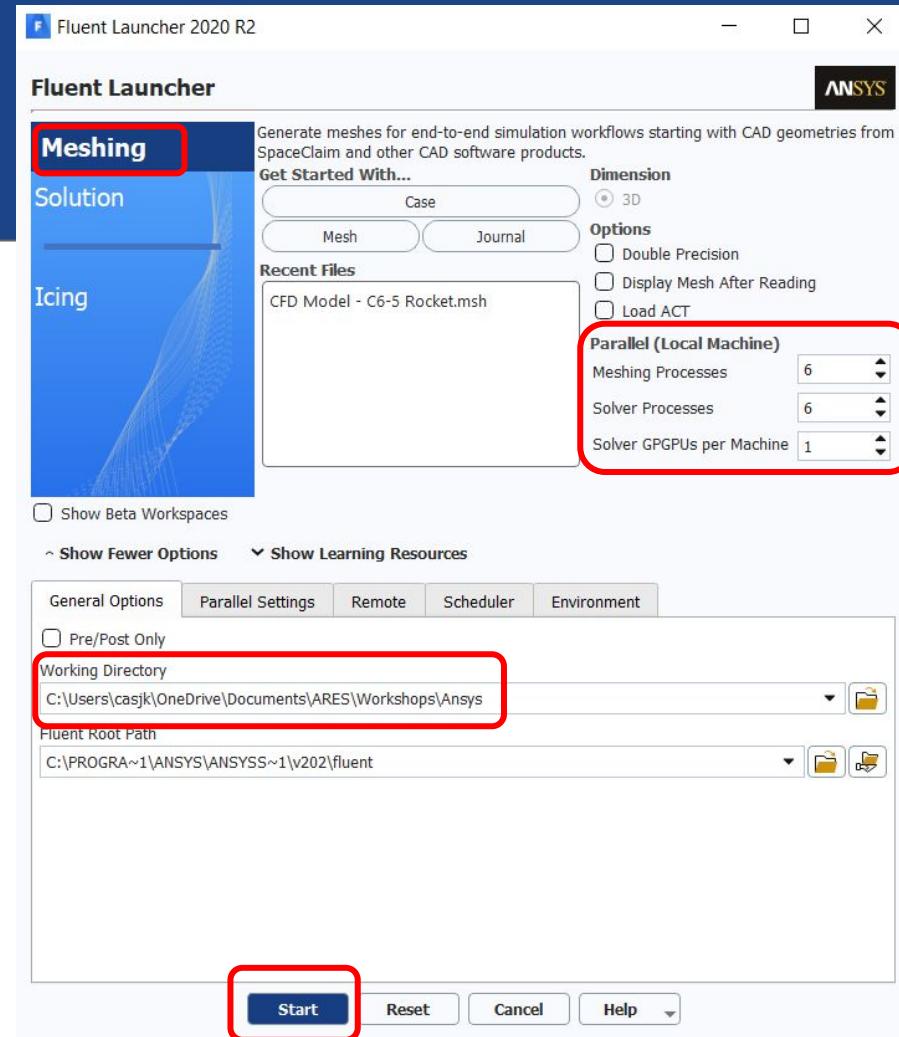
App

# Fluent

We're ready to open Fluent



- How many cores do you have
- Do you have a GPU (e.g. Nvidia)



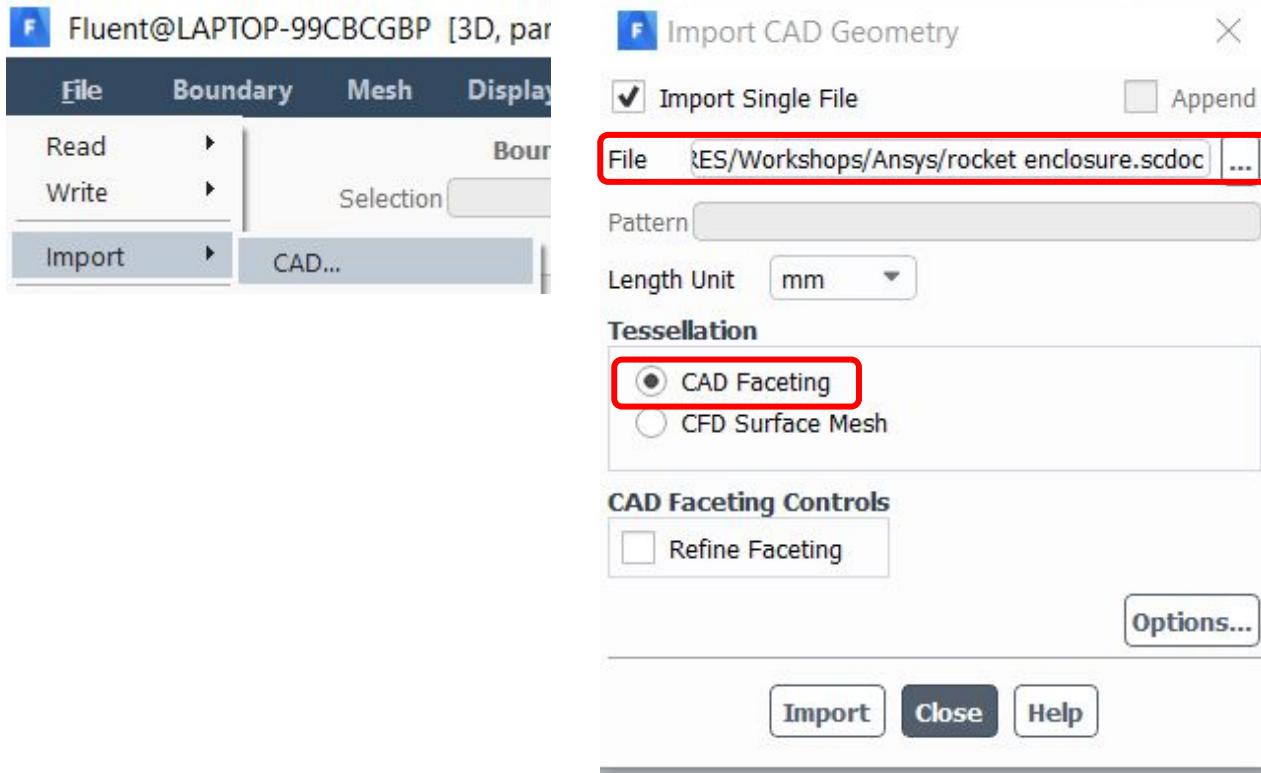


# Fluent





# Fluent





# Fluent

Fluent@LAPTOP-99CBCGBP [3D, par]

File Boundary Mesh Display

Read ▾ Write ▾ Import ▾ CAD...

Boun Selection

Length Unit mm

**Tessellation**

CAD Faceting  CFD Surface Mesh

**CAD Faceting Controls**

Refine Faceting

**Import CAD Geometry**

Import Single File  Append

File **ARES/Workshops/Ansys/rocket enclosure.scdoc** ...

Pattern Workflow

Outline View

Filter Text

Mesh Generation

Model

- CAD Assemblies
- Geometry Objects
  - boi-far:boi-far-solid
  - boi-near:boi-near-solid
  - boi-near:boi-near-solid.1
  - enclosure:enclosure-enclosure
- Mesh Objects
- Unreferenced

Draw

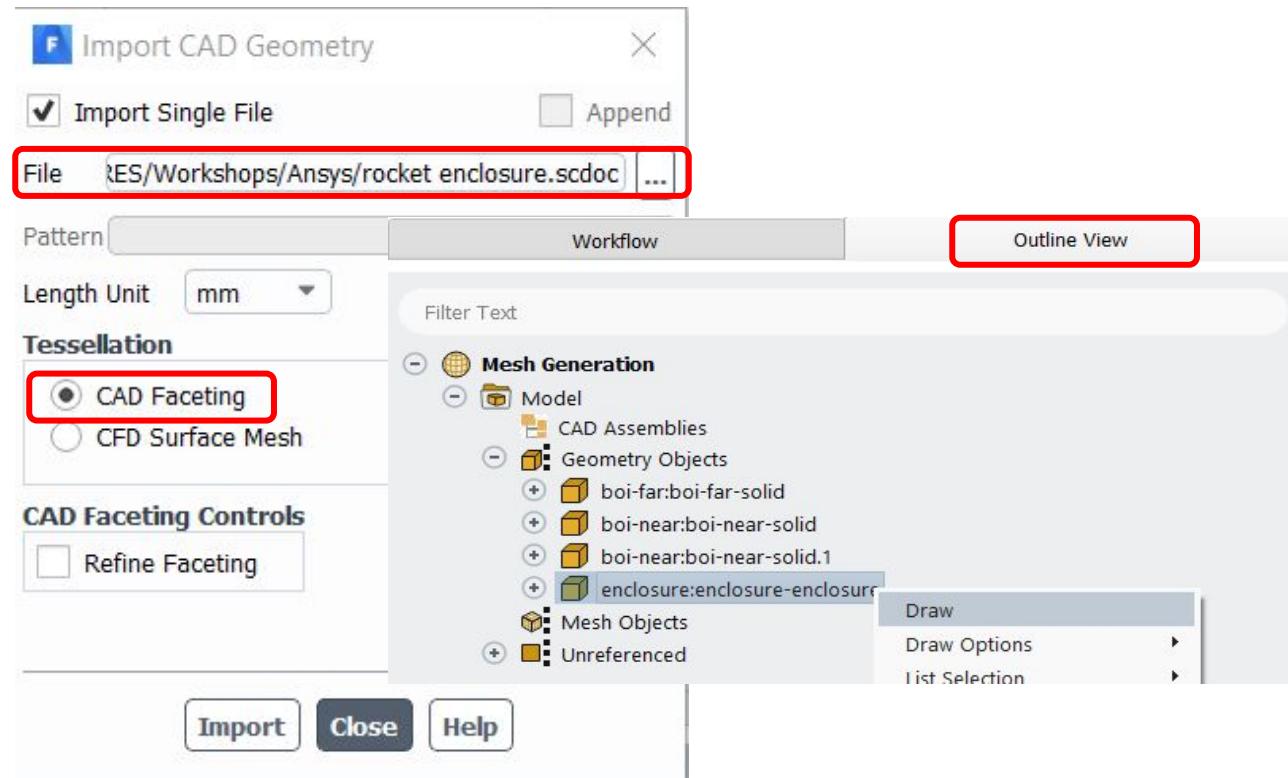
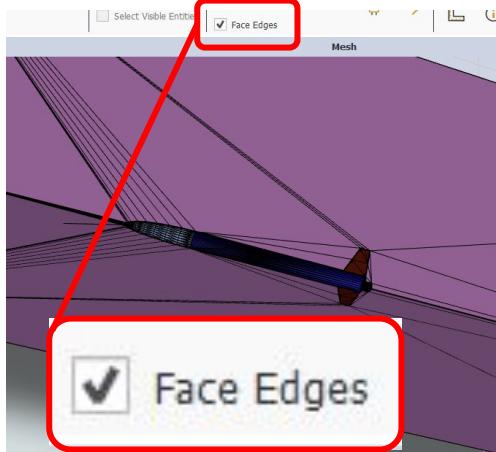
Draw Options

List Selection

Import Close Help

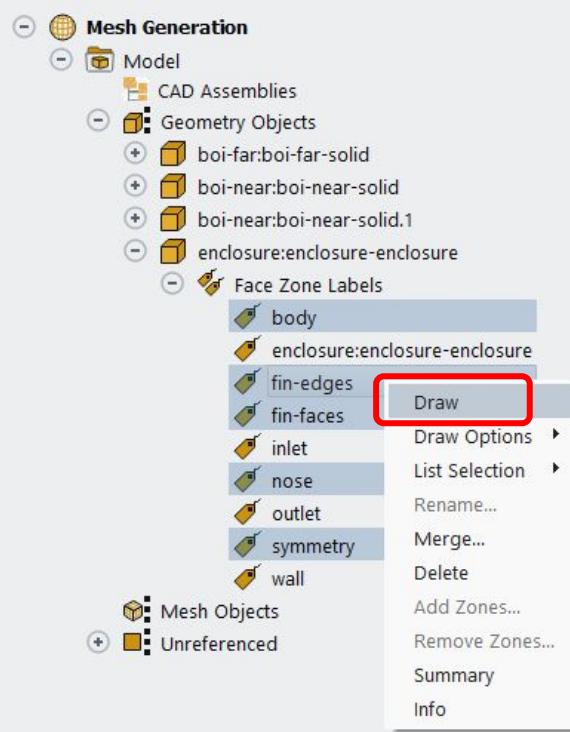


# Fluent



# Fluent

Show and hide whatever  
parts you want here



# Fluent

Show and hide whatever parts you want here

Also add a ruler here



A screenshot of the ARES Mesh Generation tree view. The tree structure includes nodes for CAD Assemblies, Geometry Objects, Face Zone Labels, Mesh Objects, and Unreferenced. A context menu is open over the "body" node under Face Zone Labels. The menu options are: Draw, Draw Options, List Selection, Rename..., Merge..., Delete, Add Zones..., Remove Zones..., Summary, and Info. The "body" node is highlighted with a blue selection bar.

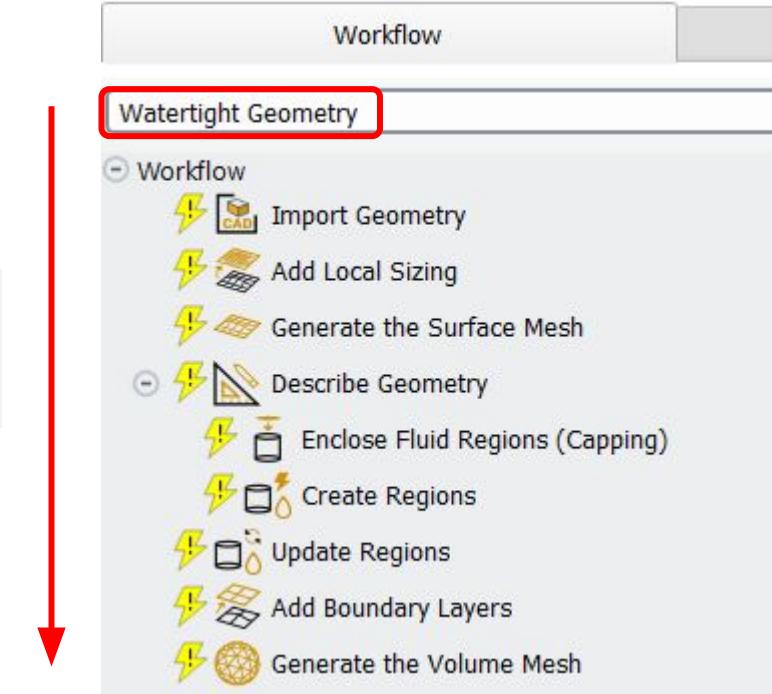
- Mesh Generation
- Model
  - CAD Assemblies
  - Geometry Objects
    - + boi-far:boi-far-solid
    - + boi-near:boi-near-solid
    - + boi-near:boi-near-solid.1
    - enclosure:enclosure-enclosure
  - Face Zone Labels
    - body
    - enclosure:enclosure-enclosure
    - fin-edges
    - fin-faces
    - inlet
    - nose
    - outlet
    - symmetry
    - wall

- Mesh Objects
- + Unreferenced

# Fluent



We will follow these steps  
one by one :)



# Fluent



We will follow these steps  
one by one :)

The screenshot shows the ARES Workflow interface. At the top, there is a tree view of tasks:

- Workflow
  - Import Geometry (selected, highlighted in blue)
  - Add Local Sizing
  - Generate the Surface Mesh
  - Describe Geometry
    - Enclose Fluid Regions (Capping)
    - Create Regions
    - Update Regions
    - Add Boundary Layers
    - Generate the Volume Mesh

Below the tree view is a detailed configuration panel for the selected "Import Geometry" step:

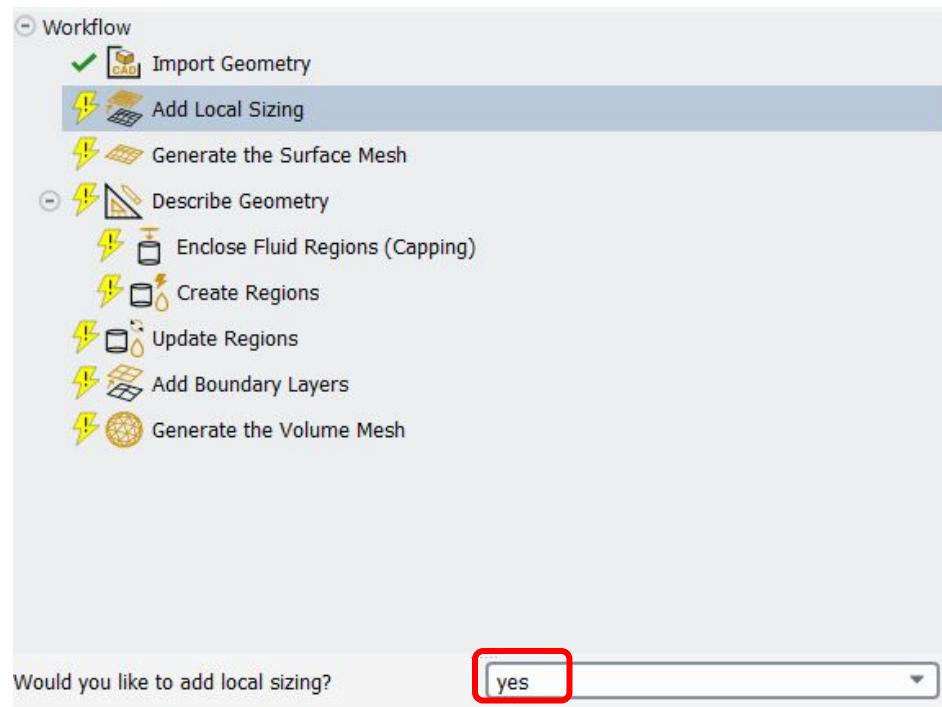
**Import Geometry**

File Format	CAD
Units	mm
<input type="checkbox"/> Advanced Options	
File Name <input type="text" value="Users/casjk/OneDrive/Documents/ARES/Workshops/Ansys/rocket enclosure.scdcc"/> <input type="button" value="..."/>	

# Choose Mesh Sizing

We can choose **local mesh sizing** for different areas of the flow:

- BOIs
- Curved surfaces
- Tiny faces



# Choose Mesh Sizing

We can choose local mesh sizing for different areas of the flow:

- **BOIs**
- Curved surfaces
- Tiny faces

Add Local Sizing

Name	boi-near
Growth Rate	1.2
Size Control Type	Body Of Influence
Target Mesh Size	20
Select By	label

Filter Text [1/11]

body  
boi-far:boi-far-solid  
**boi-near:boi-near-solid**   
enclosure:enclosure-enclosure  
fin-edges  
fin-faces  
inlet  
nose  
...

Repair Sizing Body  
 Draw Size Boxes

Add Local Sizing Revert and Edit Clear Preview Draw Bodies ...

# Choose Mesh Sizing

We can choose local mesh sizing for different areas of the flow:

- **BOIs**
- Curved surfaces
- Tiny faces

Add Local Sizing

Name	boi-near
Growth Rate	1.2
Size Control Type	Body Of Influence
Target Mesh Size	20
Select By	label

Filter Text [1/11]

body  
boi-far:boi-far-solid  
**boi-near:boi-near-solid**   
enclosure:enclosure-enclosure  
fin-edges  
fin-faces  
inlet  
nose  
...

Repair Sizing Body  
 Draw Size Boxes

Add Local Sizing Revert and Edit Clear Preview Draw Bodies ...

# Choose Mesh Sizing

We can choose local mesh sizing for different areas of the flow:

- **BOIs**
- Curved surfaces
- Tiny faces

Add Local Sizing

Name	boi-near
Growth Rate	1.2
Size Control Type	Body Of Influence
Target Mesh Size	20
Select By	label

Filter Text [1/11]

body  
boi-far:boi-far-solid  
**boi-near:boi-near-solid**   
enclosure:enclosure-enclosure  
fin-edges  
fin-faces  
inlet  
nose  
...  
 Repair Sizing Body  
 Draw Size Boxes

Add Local Sizing Revert and Edit Clear Preview Draw Bodies ...

# Choose Mesh Sizing

We can choose local mesh sizing for different areas of the flow:

- **BOIs**
- Curved surfaces
- Tiny faces

Add Local Sizing

Name	boi-near
Growth Rate	1.2
Size Control Type	Body Of Influence
Target Mesh Size	20
Select By	label

Filter Text [1/11]

body  
boi-far  
boi-far-solid  
**boi-near;boi-near-solid** Draw Selections  
enclosure  
fin-edges  
fin-faces  
inlet  
nose  
...

Repair Sizing Body  
 Draw Size Boxes

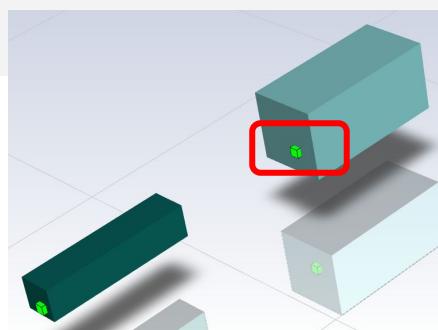
Add Local Sizing Revert and Edit Clear Preview Draw Bodies ...

# Choose Mesh Sizing

We can choose local mesh sizing for different areas of the flow:

- **BOIs**
- Curved surfaces
- Tiny faces

Green cubes indicate how big the mesh you chose



Add Local Sizing [?](#)

Name	boi-near
Growth Rate	1.2
Size Control Type	Body Of Influence
Target Mesh Size	20
Select By	label

Filter Text [1/11] Filter Text [=o](#) [=v](#) [=x](#) [=y](#)

body  
boi-far:boi-far-solid  
**boi-near:boi-near-solid** Draw Selections  
enclosure:enclosure-enclosure  
fin-edges  
fin-faces  
inlet  
nose

Repair Sizing Body

Draw Size Boxes

Add Local Sizing Revert and Edit Clear Preview Draw Bodies ...

# Choose Mesh Sizing

We can choose local mesh sizing for different areas of the flow:

- **BOIs**
- Curved surfaces
- Tiny faces

Add Local Sizing 

Name	boi-near
Growth Rate	1.2
Size Control Type	Body Of Influence
Target Mesh Size	20
Select By	label

 Filter Text [1/11]  Filter Text    

body  
boi-far:boi-far-solid  
**boi-near:boi-near-solid** Draw Selections  
enclosure:enclosure-enclosure  
fin-edges  
fin-faces  
inlet  
nose  
...  
 Repair Sizing Body  
 Draw Size Boxes

**Add Local Sizing** Revert and Edit Clear Preview Draw Bodies ...

# Choose Mesh Sizing

We can choose local mesh sizing for different areas of the flow:

- **BOIs**
- Curved surfaces
- Tiny faces

Add Local Sizing

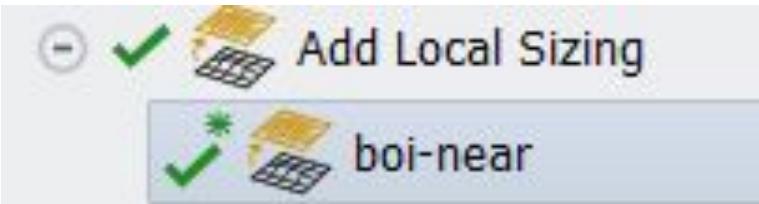
Name	boi-near
Growth Rate	1.2
Size Control Type	Body Of Influence
Target Mesh Size	20
Select By	label

Filter Text [1/11]

body  
boi-far:boi-far-solid  
**boi-near:boi-near-solid** Draw Selections  
enclosure:enclosure-enclosure  
fin-edges  
fin-faces  
inlet  
nose  
...

Repair Sizing Body  
 Draw Size Boxes

**Add Local Sizing** Revert and Edit Clear Preview Draw Bodies ...



# Choose Mesh Sizing

We can choose local mesh sizing for different areas of the flow:

- BOIs
- **Curved surfaces**
- Tiny faces

Add Local Sizing

Name	body-curv
Growth Rate	1.2
Size Control Type	Curvature
Local Min Size	0.1
Max Size	3
Curvature Normal Angle	9
Scope To	faces
Select By	label

Filter Text [1/11] Filter Text

body

boi-far:boi-far-solid  
 boi-near:boi-near-solid  
 enclosure:enclosure-enclosure  
 fin-edges

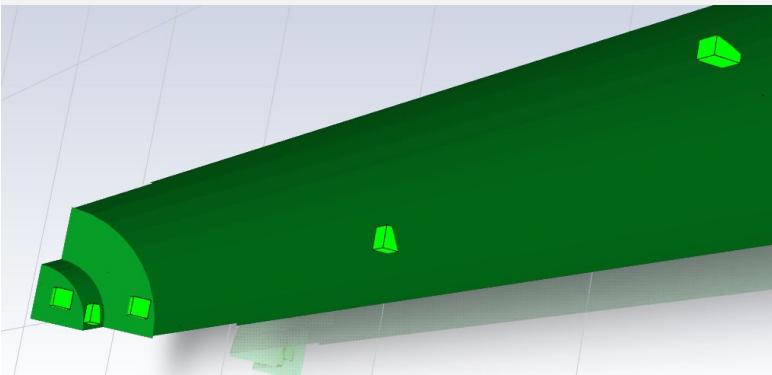
Draw Size Boxes

Add Local Sizing Revert and Edit Clear Preview Draw Bodies ...

# Choose Mesh Sizing

We can choose local mesh sizing for different areas of the flow:

- BOIs
- **Curved surfaces**
- Tiny faces



Add Local Sizing

Name	body-curv
Growth Rate	1.2
Size Control Type	Curvature
Local Min Size	0.1
Max Size	3
Curvature Normal Angle	9
Scope To	faces
Select By	label

Filter Text [1/11] Filter Text

- body
- boi-tar:boi-far-solid
- boi-near:boi-near-solid
- enclosure:enclosure-enclosure
- fin-edges

Draw Size Boxes

Add Local Sizing Revert and Edit Clear Preview Draw Bodies ...

# Choose Mesh Sizing

We can choose local mesh sizing for different areas of the flow:

- BOIs
- Curved surfaces
- **Tiny faces**

Add Local Sizing

Name	fin-edges-prox
Growth Rate	1.2
Size Control Type	Proximity
Local Min Size	0.1
Max Size	0.2
Cells Per Gap	3
Scope To	faces
Select By	label

Filter Text [1/11] Filter Text

- fin-edges
- tin-faces
- inlet
- nose
- outlet

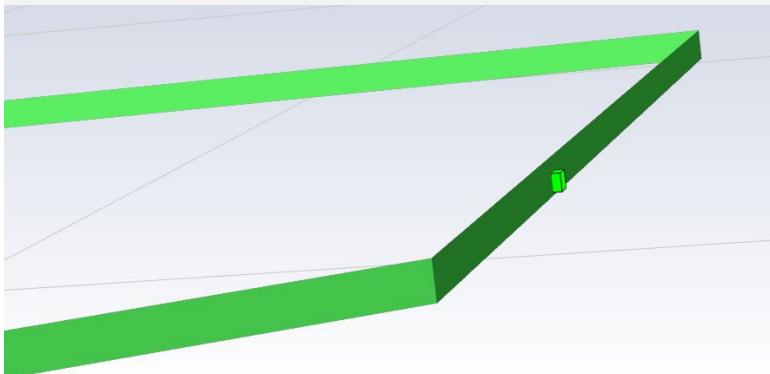
Draw Size Boxes

Add Local Sizing Revert and Edit Clear Preview Draw Bodies ...

# Choose Mesh Sizing

We can choose local mesh sizing for different areas of the flow:

- BOIs
- Curved surfaces
- **Tiny faces**



Add Local Sizing

Name	fin-edges-prox
Growth Rate	1.2
Size Control Type	Proximity
Local Min Size	0.1
Max Size	0.2
Cells Per Gap	3
Scope To	faces
Select By	label

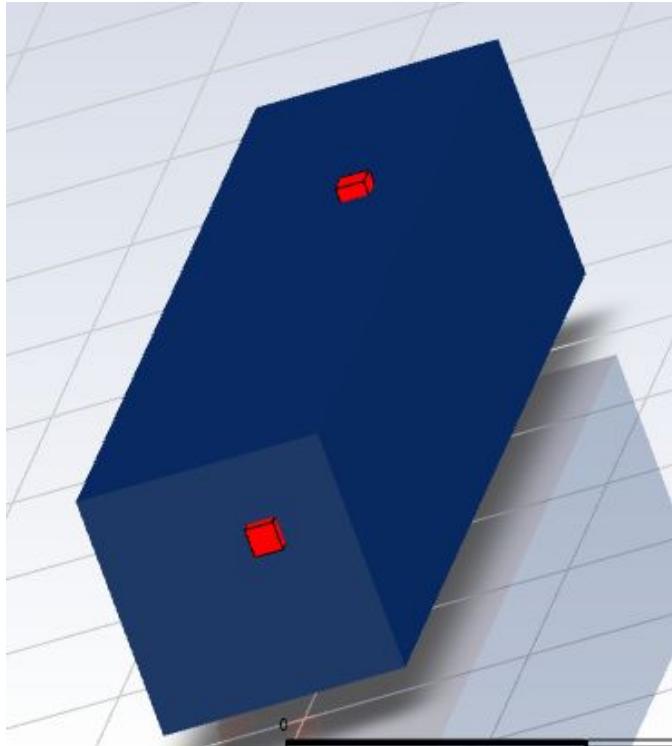
Filter Text [1/11] Filter Text

- fin-edges
- fin-faces
- inlet
- nose
- outlet

Draw Size Boxes

Add Local Sizing Revert and Edit Clear Preview Draw Bodies ...

# Make Surface Mesh



Workflow

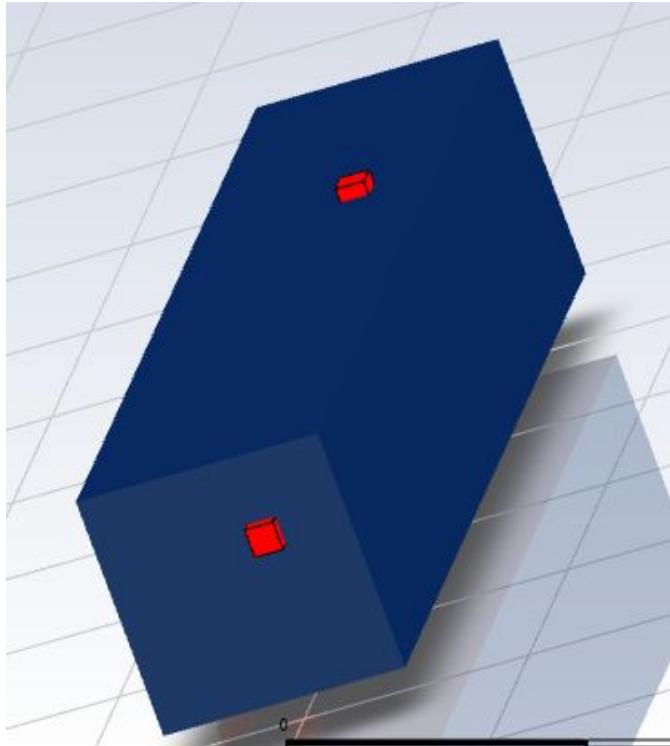
- Import Geometry
- Add Local Sizing
- Generate the Surface Mesh**
- Describe Geometry
- Enclose Fluid Regions (Capping)
- Create Regions
- Update Regions
- Add Boundary Layers
- Generate the Volume Mesh

Generate the Surface Mesh [?](#)

Minimum Size	<input type="text" value="1"/>
Maximum Size	<input type="text" value="50"/>
Growth Rate	<input type="text" value="1.2"/>
Size Functions	<input type="button" value="Curvature &amp; Proximity"/>
Curvature Normal Angle	<input type="text" value="9"/>
Cells Per Gap	<input type="text" value="3"/>
Scope Proximity To	<input type="button" value="faces-and-edges"/>
<input checked="" type="checkbox"/> Draw Size Boxes	
<input type="checkbox"/> Advanced Options	

**Generate the Surface Mesh** [Edit](#) [Clear Preview](#) ...

# Make Surface Mesh



Workflow

- Import Geometry
- Add Local Sizing
- Generate the Surface Mesh
- Describe Geometry
- Enclose Fluid Regions (Capping)
- Create Regions
- Update Regions
- Add Boundary Layers
- Generate the Volume Mesh

Generate the Surface Mesh [?](#)

Minimum Size

Maximum Size

Growth Rate

Size Functions

Curvature Normal Angle

Cells Per Gap

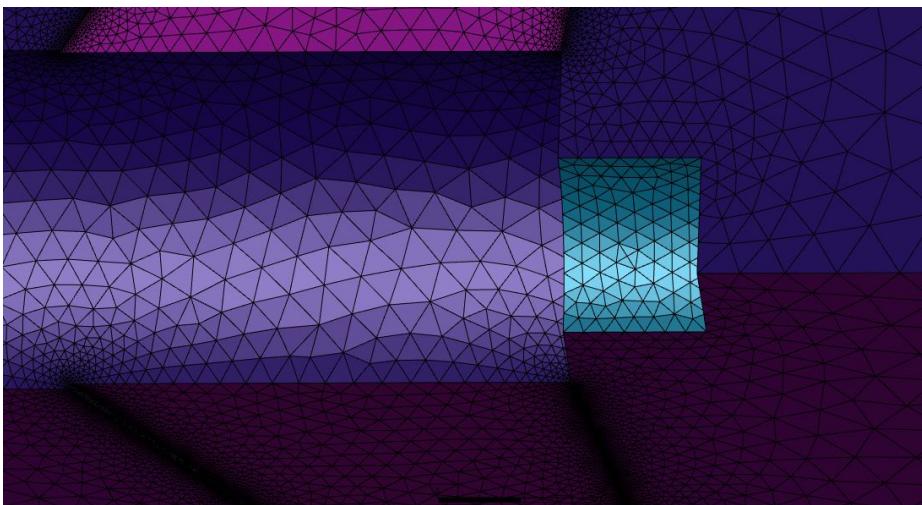
Scope Proximity To

Draw Size Boxes

Advanced Options

[Generate the Surface Mesh](#) [Edit](#) [Clear Preview](#) [...](#)

# Make Surface Mesh



Workflow

- Import Geometry
- Add Local Sizing
- Generate the Surface Mesh**
- Describe Geometry
- Enclose Fluid Regions (Capping)
- Create Regions
- Update Regions
- Add Boundary Layers
- Generate the Volume Mesh

Generate the Surface Mesh [?](#)

Minimum Size  50

Maximum Size

Growth Rate

Size Functions Curvature & Proximity

Curvature Normal Angle

Cells Per Gap

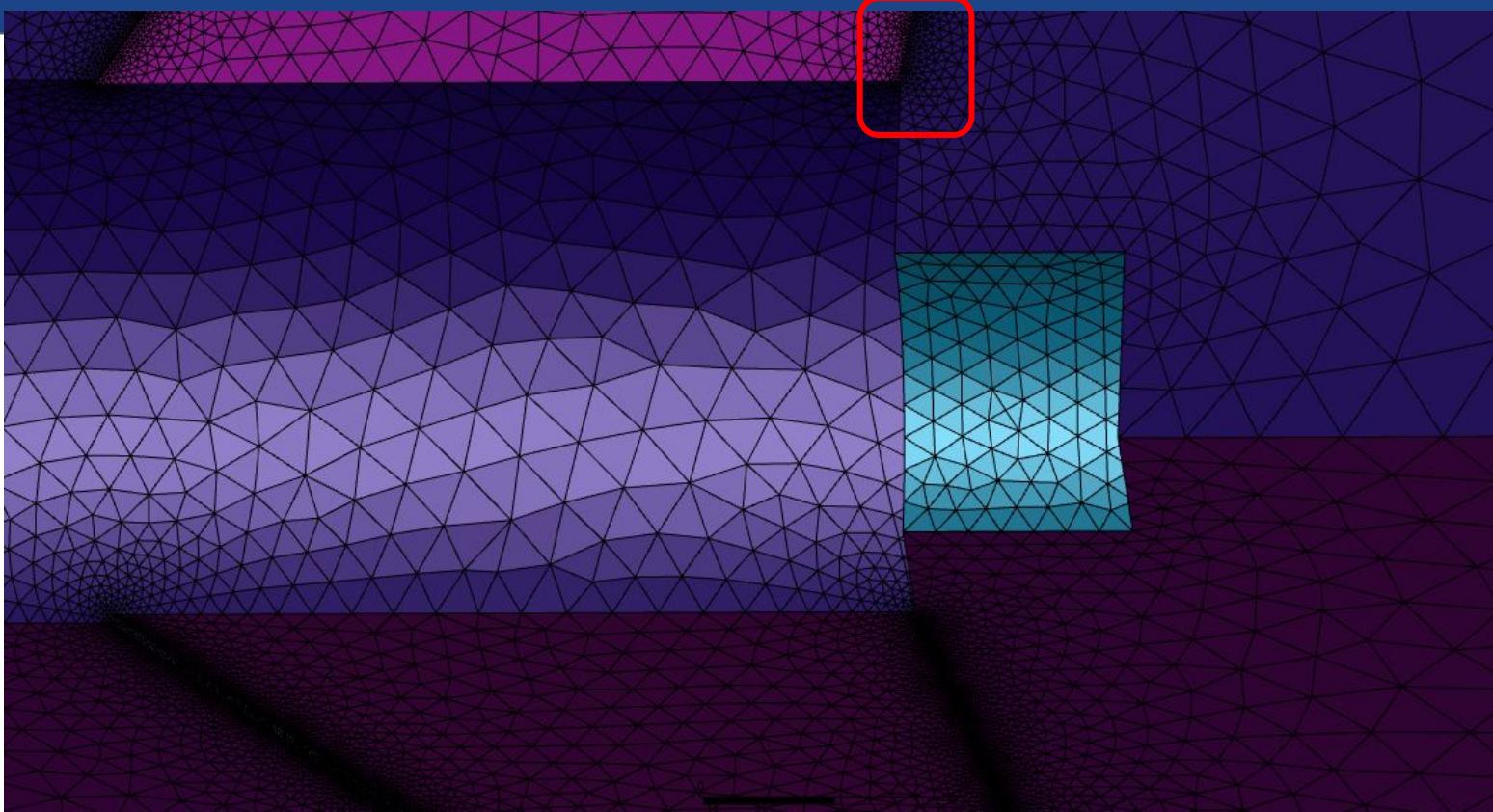
Scope Proximity To faces-and-edges

Draw Size Boxes

Advanced Options

**Generate the Surface Mesh** [Edit](#) [Clear Preview](#) [...](#)

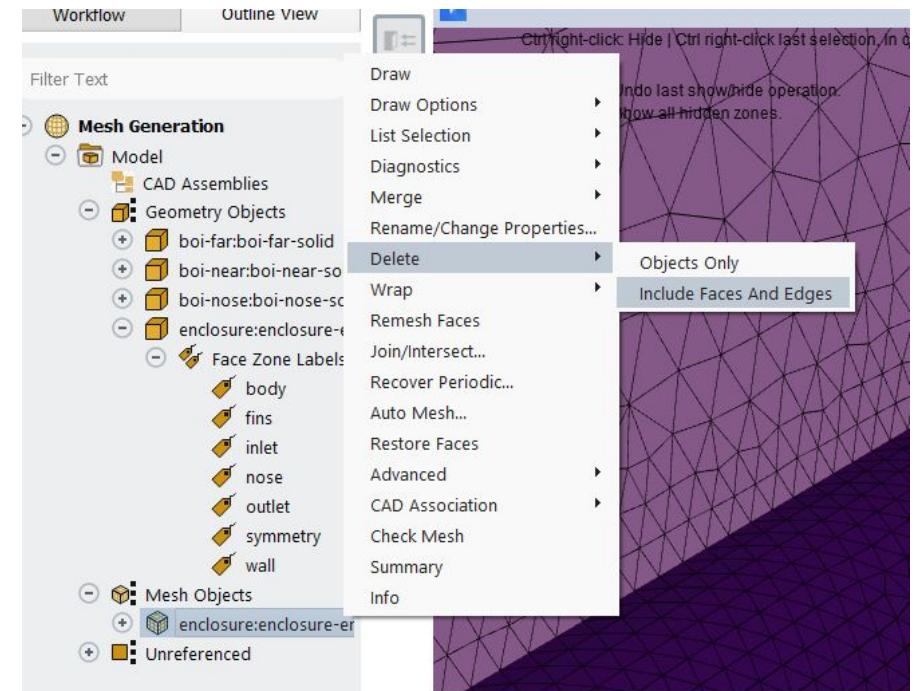
# Make Surface Mesh



# Fix Mesh Sizing

If your final mesh has issues, **you can**  
**retrospectively add more scoped sizing**

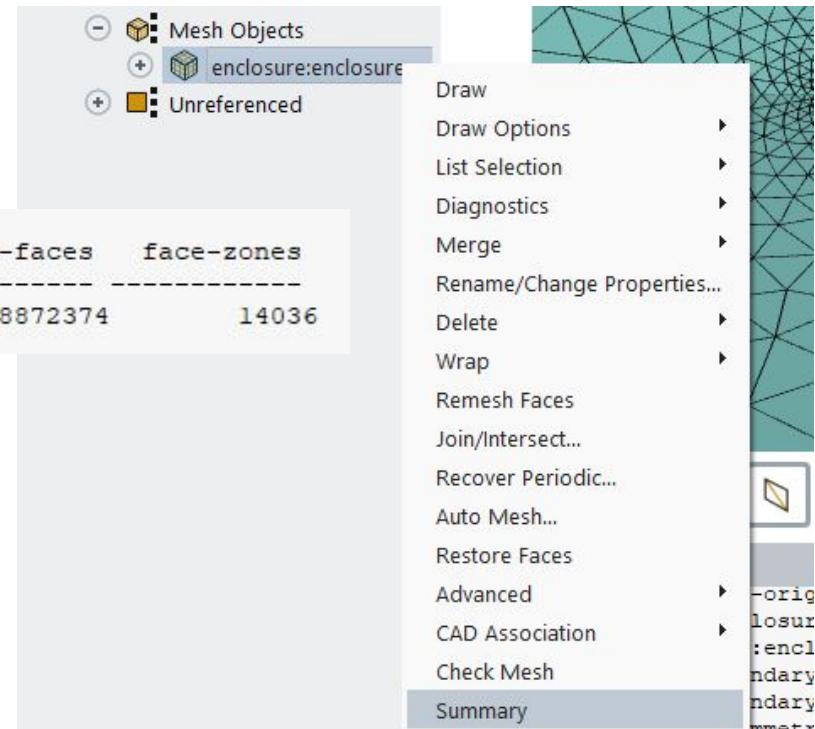
- Delete the old mesh
- Delete the size field
- Add more local scoped sizing as desired
- Recompute
- Remesh the environment object



# Check Quality

Afterwards, right click mesh -> **summary** to check how many **skewed elements** there are

skewed-faces (> 0.85)	maximum-skewness	all-faces	face-zones
0	0	0.68872374	14036



# Describe Geometry

Workflow

- Import Geometry
- Add Local Sizing
  - boi-near
  - boi-far
  - body-curv
  - nose-curv
  - fin-edges-prox
  - fin-faces-prox
- Generate the Surface Mesh
- Describe Geometry
- Update Boundaries

Describe Geometry

Geometry Type

The geometry consists of only solid regions

The geometry consists of only fluid regions with no voids

The geometry consists of both fluid and solid regions and/or voids

Change all fluid-fluid boundary types from 'wall' to 'internal'?

Yes

No

Do you need to apply Share Topology?

Yes

No

**Describe Geometry** **Revert and Edit** **...**

# Boundary Conditions

- Rocket surfaces - wall (**stationary, no slip**)
- Inlet - velocity-inlet
- Outlet - pressure-outlet
- Symmetry planes - symmetry
- Outer wall - wall (**moving at same velocity as inlet**)

The screenshot shows a software interface for mesh generation. At the top, there is a vertical list of workflow steps:

- Import Geometry (checkmark)
- Add Local Sizing (checkmark)
- Generate the Surface Mesh (checkmark)
- Describe Geometry (checkmark)
- Update Boundaries** (highlighted with a blue bar)
- Update Regions
- Add Boundary Layers
- Generate the Volume Mesh

Below this is a dialog box titled "Update Boundaries". It has a "Selection Type" dropdown set to "label" and a "Filter" section with a "Filter Text" input and three filter icons. A table lists boundary names and their types, with the "inlet" row highlighted by a red rectangle:

Boundary Name	Boundary Type
body	wall
fin-edges	wall
fin-faces	wall
<b>inlet</b>	<b>velocity-inlet</b>
nose	wall
outlet	pressure-outlet
symmetry	symmetry
wall	wall

At the bottom of the dialog are buttons for "List All Boundaries", "Update Boundaries" (highlighted with a blue bar), "Revert and Edit", "Draw Boundaries", and an ellipsis button.

# Boundary Conditions

- Rocket surfaces - wall (**stationary, no slip**)
- Inlet - velocity-inlet
- Outlet - pressure-outlet
- Symmetry planes - symmetry
- Outer wall - wall (**moving at same velocity as inlet**)

The screenshot shows a software interface for mesh generation. At the top, there is a vertical list of workflow steps:

- Import Geometry (checkmark)
- Add Local Sizing (checkmark)
- Generate the Surface Mesh (checkmark)
- Describe Geometry (checkmark)
- Update Boundaries** (highlighted with a blue bar)
- Update Regions
- Add Boundary Layers
- Generate the Volume Mesh

Below this is a dialog box titled "Update Boundaries". It has a "Selection Type" dropdown set to "label" and a "Filter" section with a "Filter Text" input and three filter icons. A red box highlights a table below:

Boundary Name	Boundary Type
body	wall
fin-edges	wall
fin-faces	wall
inlet	velocity-inlet
nose	wall
outlet	pressure-outlet
symmetry	symmetry
wall	wall

At the bottom of the dialog are buttons for "List All Boundaries", "Update Boundaries" (highlighted with a blue bar), "Revert and Edit", "Draw Boundaries", and an ellipsis button.

# Update Regions

Workflow

- Import Geometry
- Add Local Sizing
- Generate the Surface Mesh
- Describe Geometry
- Update Boundaries
- Update Regions**
- Add Boundary Layers
- Generate the Volume Mesh

Update Regions

Region Name	Region Type
enclosure-enclosure	fluid

Update Regions Revert and Edit Draw Regions ...

# Boundary Layers

Workflow

- Import Geometry
- Add Local Sizing
- Generate the Surface Mesh
- Describe Geometry
- Update Boundaries
- Update Regions
- Add Boundary Layers
- Generate the Volume Mesh

Would you like to add boundary layers?

Add Boundary Layers

Name	smooth-transition_1
Offset Method Type	smooth-transition
Number of Layers	5
Transition Ratio	0.272
Growth Rate	1.2
Add in	fluid-regions
Grow on	only-walls

Advanced Options

**Add Boundary Layers** **Revert and Edit** **Draw Regions** ...

# Compute Volumetric Mesh

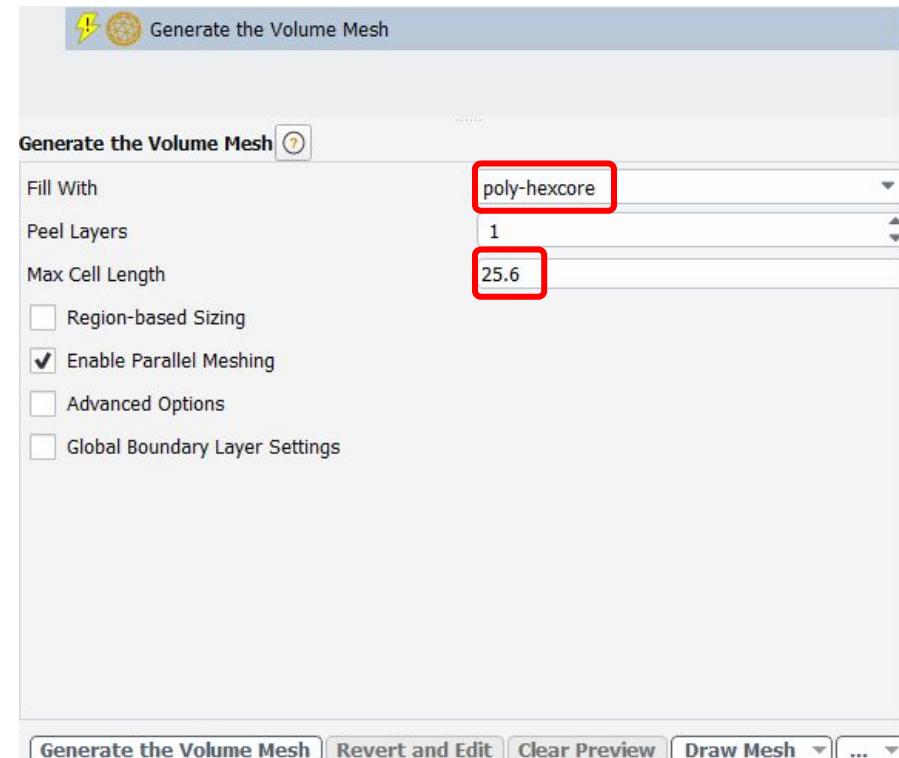
Everything up to this point was just the surface mesh

Now we'll make the **3D (volumetric) mesh**

Use **poly-hexcore** (makes a mix of polyhedra and hexahedra)

Want a **minimum Orthogonal Quality of >0.1**

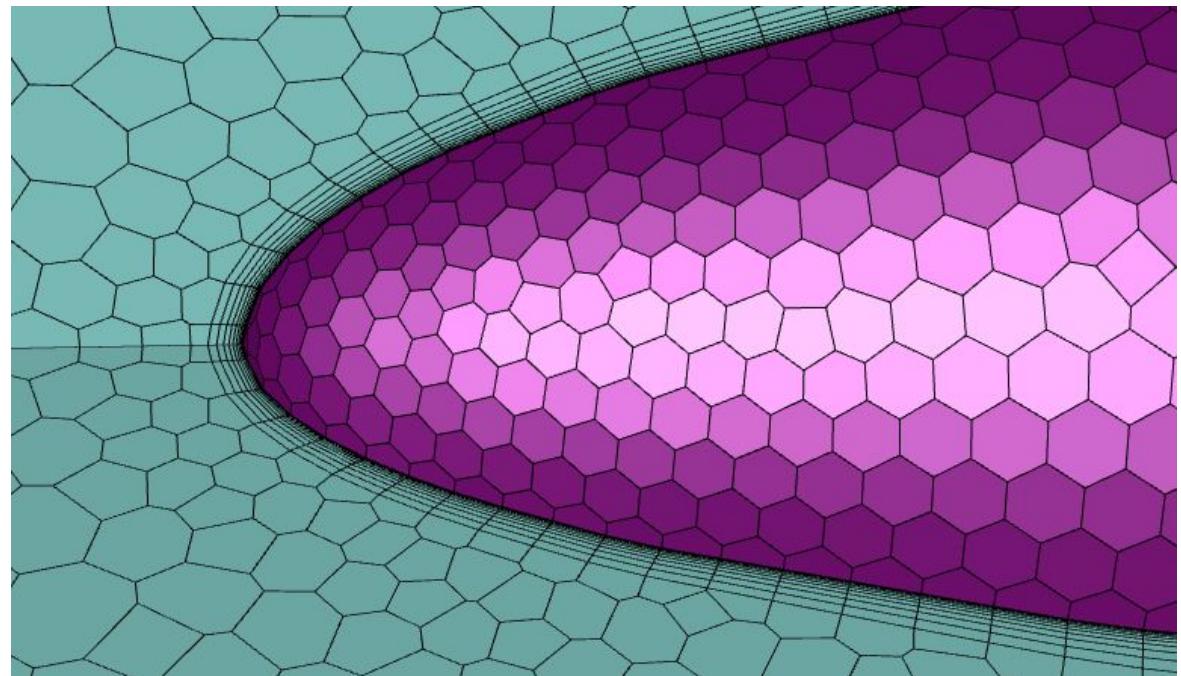
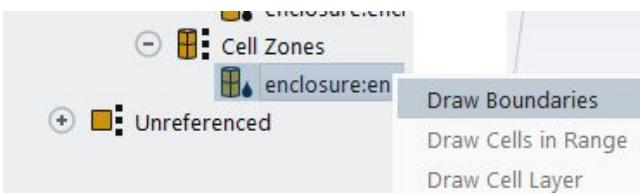
- The mesh has a minimum Orthogonal Quality of: **0.20**



# Draw Volumetric Mesh

Let's draw the 3D mesh and see how it looks

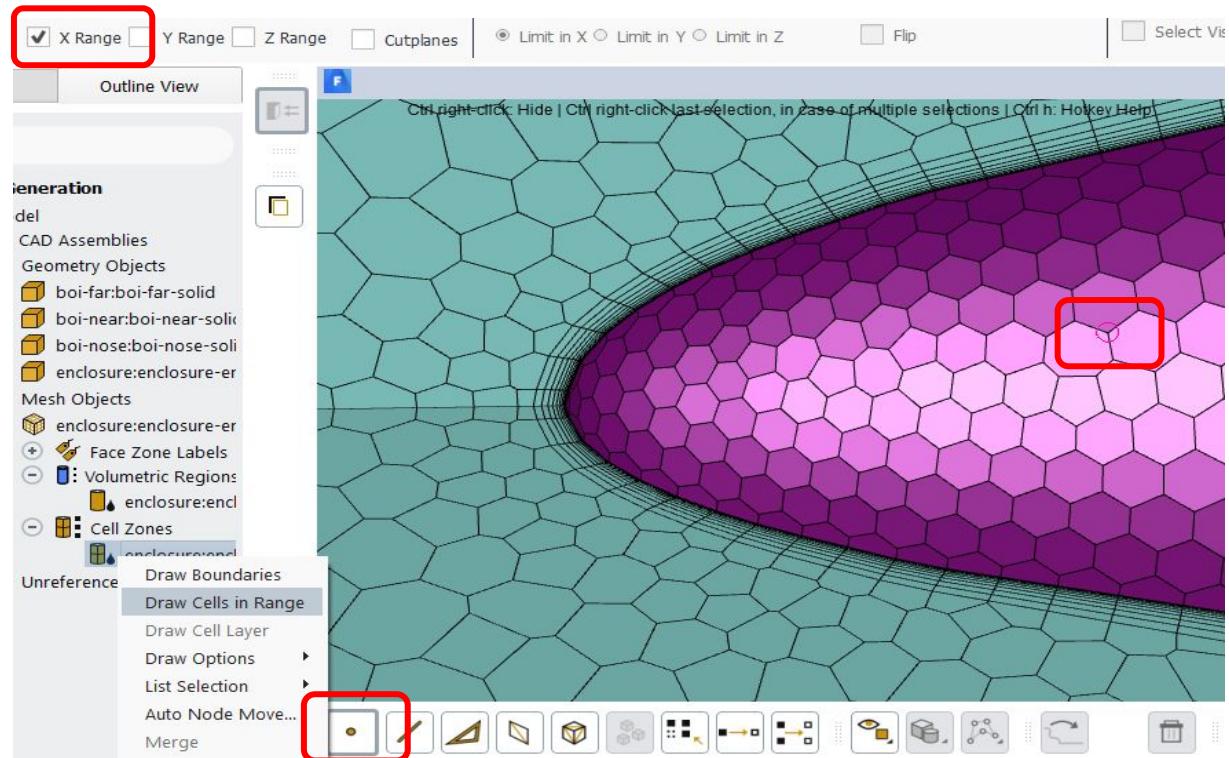
We can see the **boundary layer prisms**



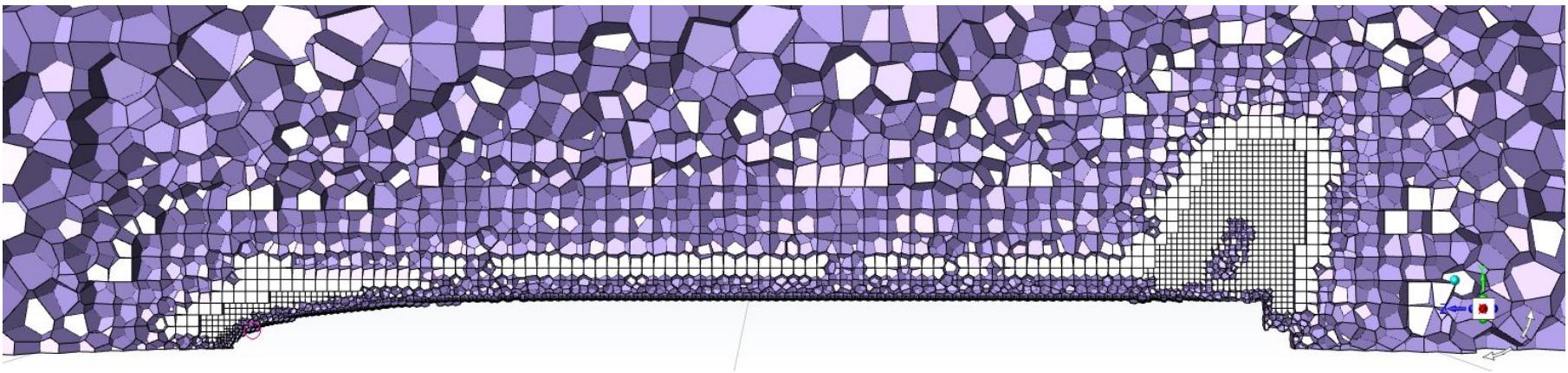
# Draw Volumetric Mesh

Let's make a section view

- Select a single node
- Select X range
- Select draw cells in range



# Draw Volumetric Mesh



Cool!

Spot the different types of mesh

- **Prism**
- **Polygon**
- **Hex**



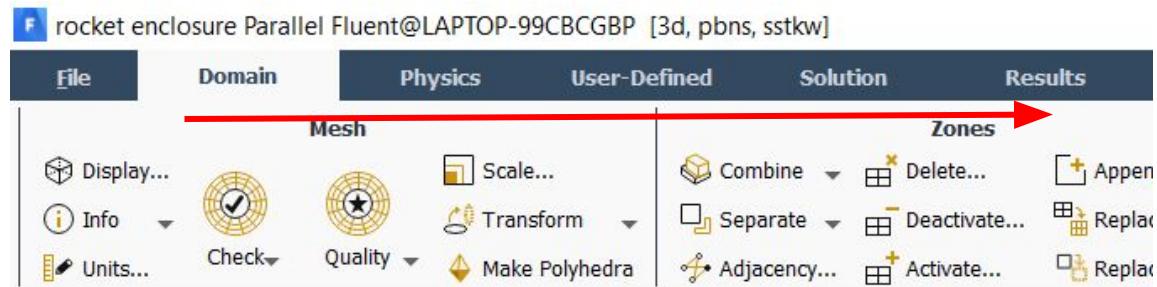
# Fluent Solver



# Fluent Solver

We will follow  
these **easy** steps  
one by one :)

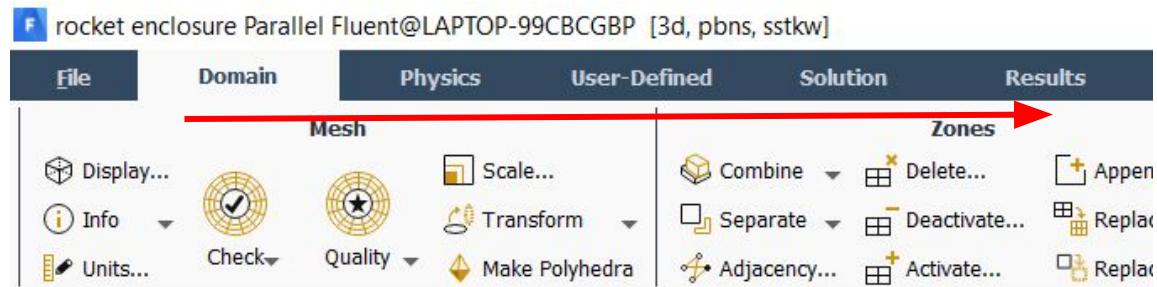
- **Domain** tab
- **Physics** tab
- **Solution** tab
- **Results** tab



# Domain Tab

We will follow  
these steps one by  
one :)

- **Domain** tab
- **Physics** tab
- **Solution** tab
- **Results** tab



Should give

no errors.

Can also check  
quality here if  
you want.



# Physics Tab

The screenshot shows the ARES software interface with the "Physics" tab selected. The "General" button in the "Solver" section is highlighted with a red box.

**Remember z axis  
is pointing down  
the rocket to the  
ground**

**General**

**Mesh**

- Scale...
- Check
- Report Quality

- Display...
- Units...

**Solver**

**Type**

- Pressure-Based
- Density-Based

**Velocity Formulation**

- Absolute
- Relative

**Time**

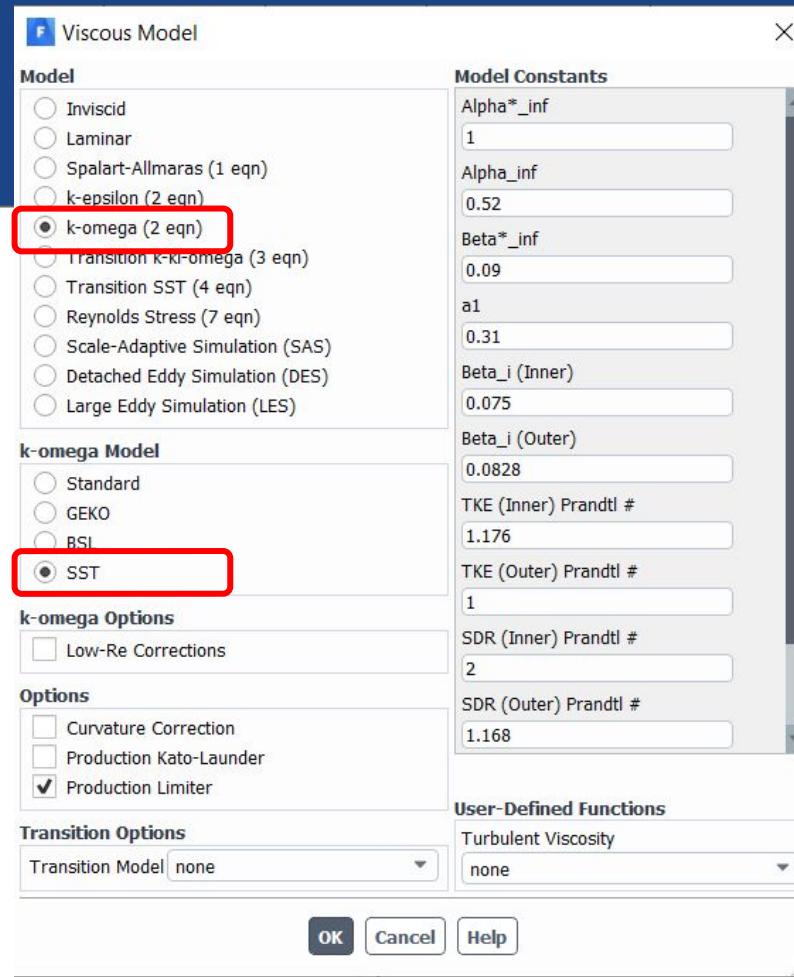
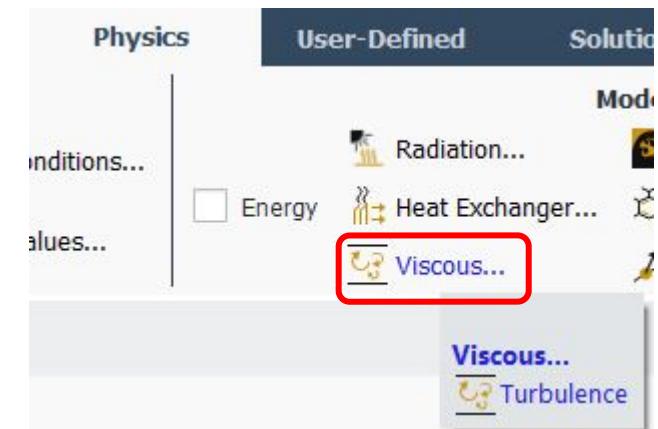
- Steady
- Transient

Gravity

**Gravitational Acceleration**

X (m/s <sup>2</sup> )	0
Y (m/s <sup>2</sup> )	0
Z (m/s <sup>2</sup> )	9.81

# Physics Tab





# Physics Tab

Time to give a bit more info about our boundary conditions.

The screenshot shows the ARES software interface with the "Physics" tab selected. The menu bar includes tabs for Physics, User-Defined, Solution, Results, View, Parallel, and Des. Under the Physics tab, there are sections for Models, Materials, and Zones. The Models section contains icons for Radiation..., Multiphase..., Structure..., Species..., Acoustics..., Discrete Phase..., and More. The Materials section has a "Create/Edit..." button with a flask icon. The Zones section includes icons for Cell Zones, Boundaries (which is highlighted with a red box), and Profiles... . On the left side of the interface, there are dropdown menus for "Boundary conditions..." and "Values...".



# Physics Tab

Time to give a bit more info about our boundary conditions.

## Stationary walls

### Surface

### roughness

Inlet velocity

Wall velocity

(same as inlet)

**Boundary Conditions**

Zone  Filter Text

body  
fin-edges  
fin-faces  
inlet  
interior--enclosure-enclosure  
nose  
outlet  
symmetry  
wall

Phase	Type	ID
mixture	wall	68

Edit... Copy... Profiles...

# Physics Tab

Time to give a bit more info about our boundary conditions.

## Stationary walls

### Surface

### roughness

Inlet velocity

Wall velocity

(same as inlet)

**Boundary Conditions**

Zone  Filter Text

body  
fin-edges  
fin-faces  
inlet  
interior--enclosure-enclosure  
nose  
outlet  
symmetry  
wall

**F Wall**

Zone Name  body

Adjacent Cell Zone  enclosure-enclosure

Momentum Thermal Radiation Species DPM Multiphase UDS Potential Structure

**Wall Motion** **Motion**

Stationary Wall  Relative to Adjacent Cell Zone  
 Moving Wall

**Shear Condition**

No Slip  
 Specified Shear  
 Specularity Coefficient  
 Marangoni Stress

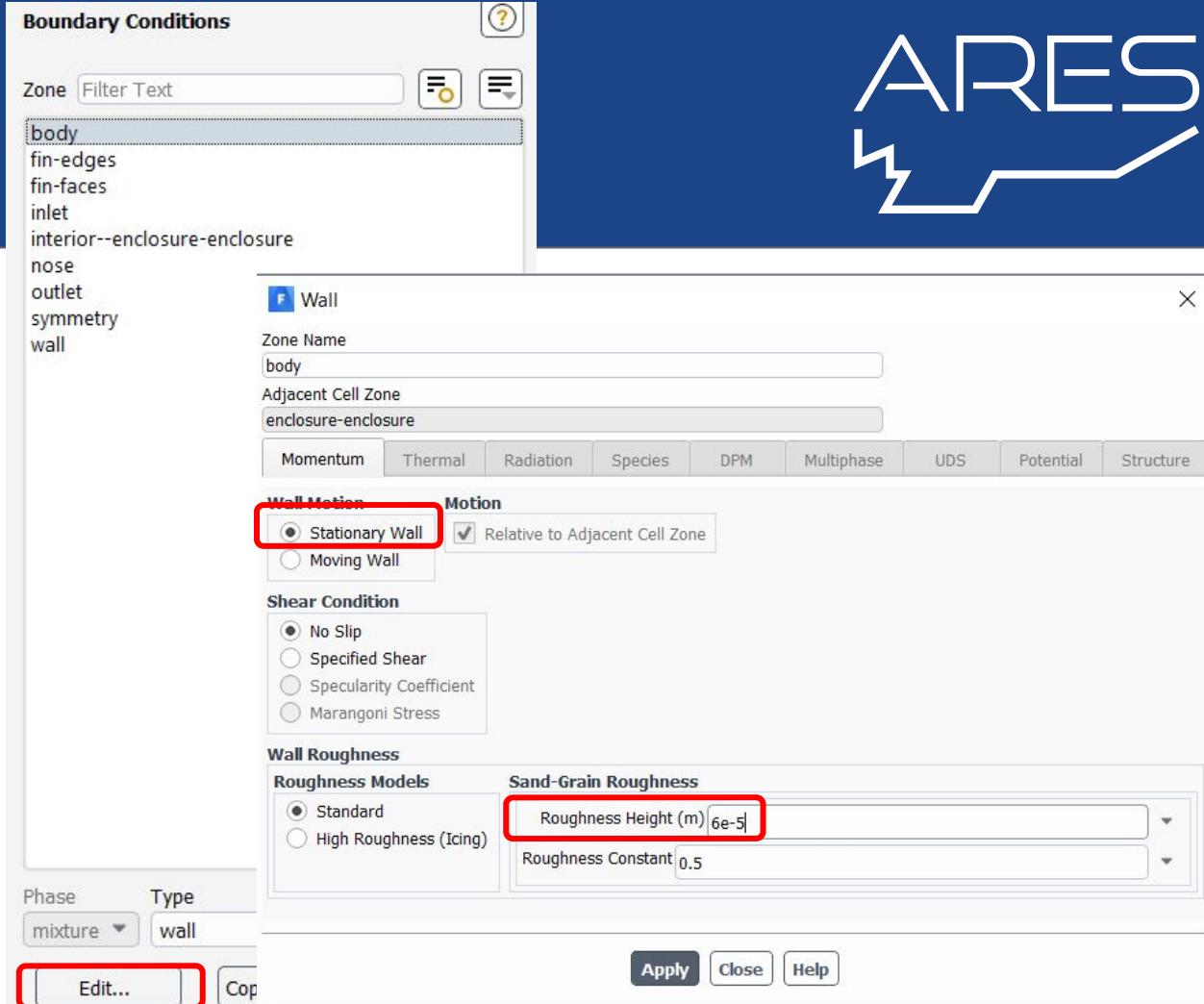
**Wall Roughness**

**Roughness Models** **Sand-Grain Roughness**

Standard  High Roughness (Icing) Roughness Height (m)  6e-5  
 High Roughness (Icing) Roughness Constant  0.5

Phase  Type  wall

Edit...  Cop  Apply  Close  Help



# Physics Tab

Time to give a bit more info about our boundary conditions.

Stationary walls  
Surface roughness

## Inlet velocity

Wall velocity  
(same as inlet)

Boundary Conditions

Zone Filter Text

body  
fin-edges  
fin-faces  
**inlet**   ?

interior--enclosure-enclosure  
nose  
outlet  
symmetry  
wall

Phase	Type	ID
mixture	velocity-inlet	73

Edit... Copy... Profiles...

# Physics Tab

Time to give a bit more info about our boundary conditions.

Stationary walls  
Surface roughness

## Inlet velocity

Wall velocity  
(same as inlet)

The screenshot shows the 'Boundary Conditions' dialog box in ARES. The 'Zone' dropdown is set to 'inlet'. The 'Velocity Inlet' tab is selected. The 'Zone Name' is 'inlet'. Under 'Momentum', the 'Velocity Specification Method' is 'Magnitude, Normal to Boundary', the 'Reference Frame' is 'Absolute', and the 'Velocity Magnitude (m/s)' is '10'. Under 'Turbulence', the 'Specification Method' is 'Intensity and Viscosity Ratio', 'Turbulent Intensity (%)' is '5', and 'Turbulent Viscosity Ratio' is '10'. At the bottom, there are buttons for 'Apply', 'Close', and 'Help', and dropdowns for 'Phase' (set to 'mixture') and 'Type' (set to 'velocity-inlet').



# Physics Tab

Time to give a bit more info about our boundary conditions.

Stationary walls

Surface

roughness

Inlet velocity

**Wall velocity**

(same as inlet)

**Boundary Conditions**

Zone Filter Text

- body
- fin-edges
- fin-faces
- inlet
- interior--enclosure-enclosure
- nose
- outlet
- symmetry
- wall

Phase Type ID

mixture wall 75

Edit... Copy... Profiles...

# Physics Tab

Time to give a bit more info about our boundary conditions.

Stationary walls  
Surface roughness  
Inlet velocity

**Wall velocity**  
**(same as inlet)**

Boundary Conditions

Zone Filter Text

body  
fin-edges  
fin-faces  
inlet  
interior--enclosure-enclosure  
nose  
outlet  
symmetry  
wall

F Wall

Zone Name: wall

Adjacent Cell Zone: enclosure-enclosure

Momentum Thermal Radiation Species DPM Multiph

**Wall Motion**

Stationary Wall  Moving Wall

**Motion**

Relative to Adjacent Cell Zone  Absolute

Translational  Rotational  Components

Speed (m/s): 10

**Direction**

X: 0  
Y: 0  
Z: 1

**Shear Condition**

No Slip  Specified Shear  Specularity Coefficient  Marangoni Stress

Phase: mixture Type: wall ID: 7

Edit... Copy... Profiles...

Remember z axis is pointing down the rocket to the ground

# Physics Tab

Time to give a bit  
more info about  
our boundary  
conditions.

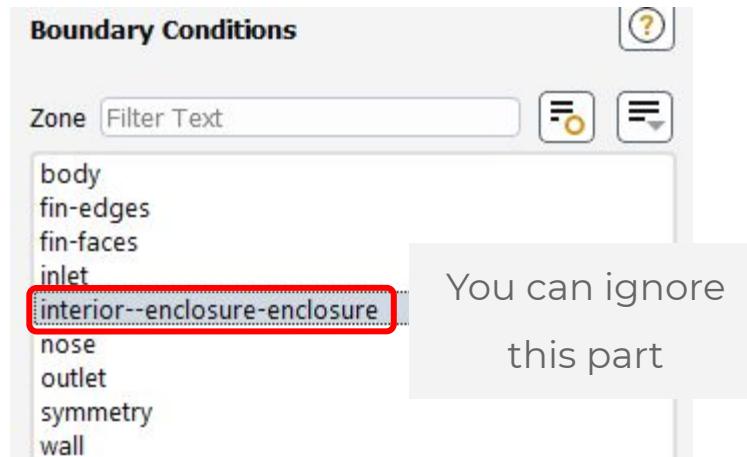
Stationary walls  
Surface  
roughness  
Inlet velocity  
Wall velocity  
(same as inlet)

**Boundary Conditions**

Zone Filter Text

body  
fin-edges  
fin-faces  
inlet  
**interior--enclosure-enclosure**  
nose  
outlet  
symmetry  
wall

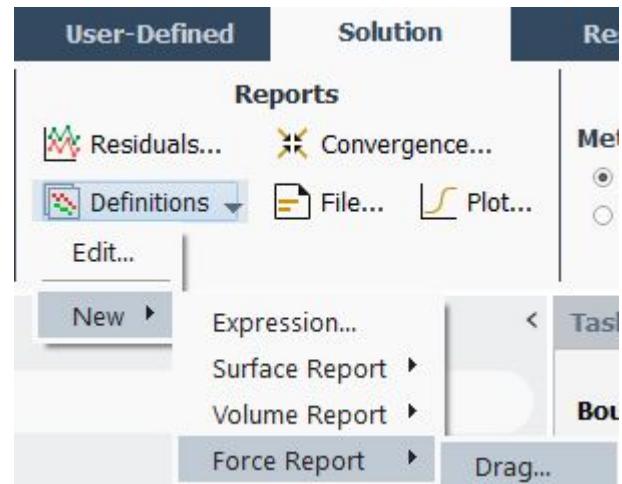
You can ignore  
this part



# Solution Tab

Let's decide what we want to solve for!

Let's solve for the drag coefficient on the rocket ( $C_d$ )



# Solution Tab

Let's decide what we want to solve for!

Let's solve for the drag coefficient on the rocket (Cd)

**F Drag Report Definition**

Name: cd-report

Options:

Per Zone

Average Over(Iterations): 1

Report Output Type:

Drag Coefficient

Drag Force

Force Vector:

X	Y	Z
0	0	1

User-Defined      Solution      Reports

Residuals...      Convergence...

Definitions      File...      Plot...

Edit...

New ▾

Expression...      Surface Report ▾

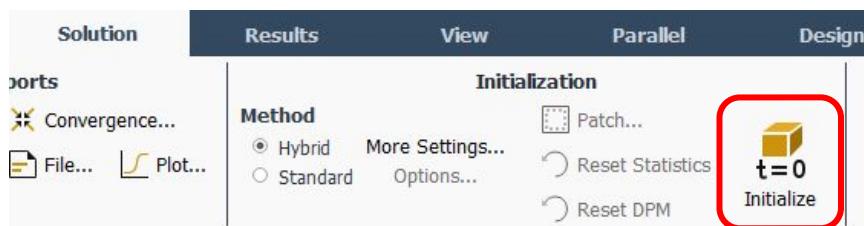
Volume Report ▾      Force Report ▾

Drag...

Wall Zones: body, fin-edges, fin-faces, nose, wall

# Solution Tab

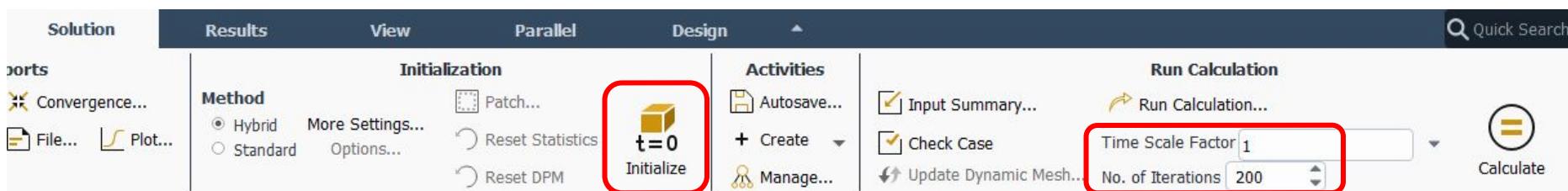
Always **initialise first**



# Solution Tab

Always **initialise first**

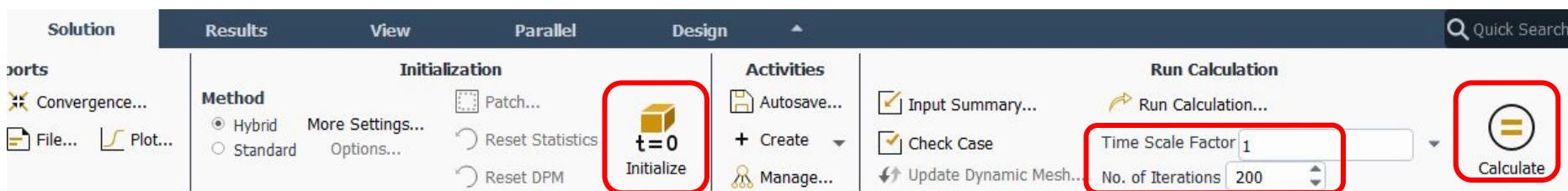
Then set iterations to 200



# Solution Tab

Always **initialise first**

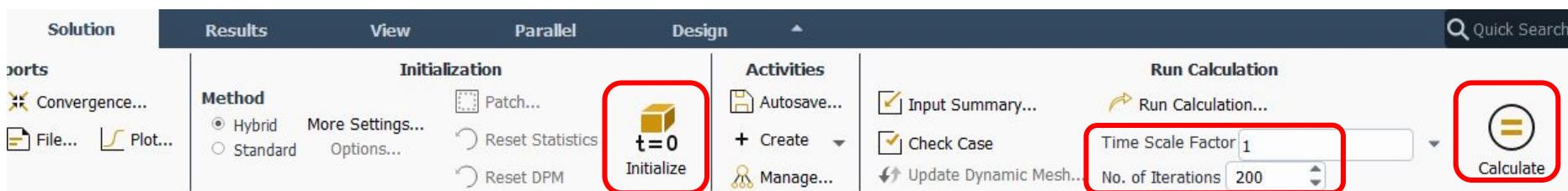
Then set iterations to 200 and hit Calculate



# Solution Tab

Always **initialise first**

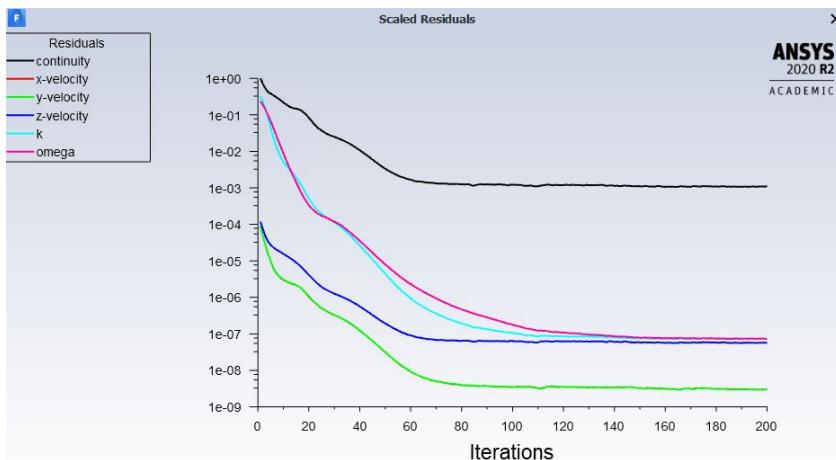
Then set iterations to 200 and hit Calculate



# Residuals

Fluent will automatically plot residuals

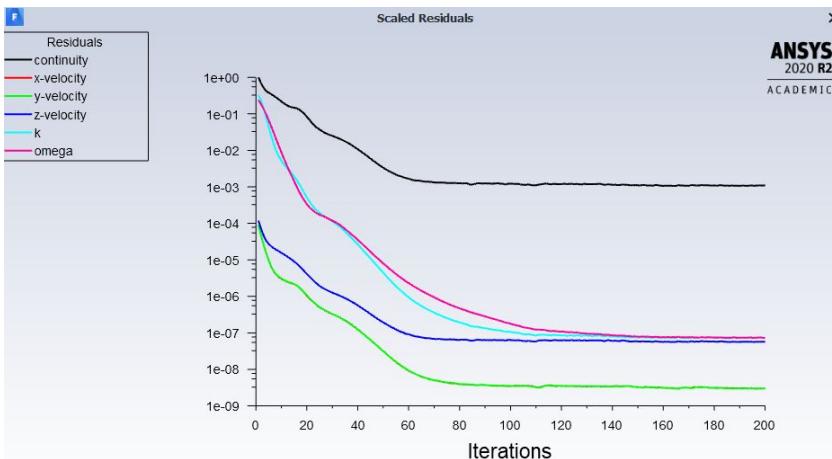
You want the **residuals to decrease** by a factor of  $10^3$  and **stabilise** for convergent solution



# Residuals

Fluent will automatically plot residuals

You want the **residuals to decrease** by a factor of  $10^3$  and **stabilise** for convergent solution



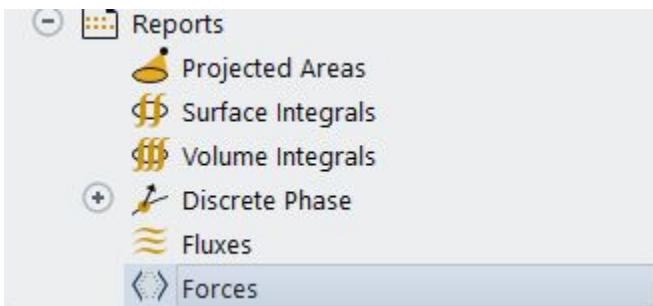
Nice low drag coefficient (is this realistic??)

Calculation complete.

Cd  
-----  
report-def-0      0.013132033

# Solve for Forces/Moments/COP

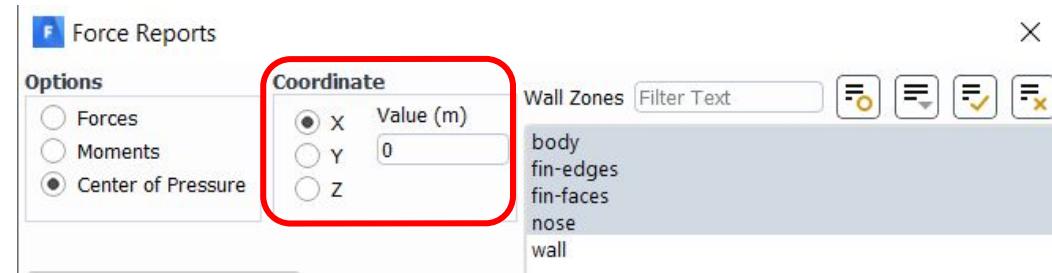
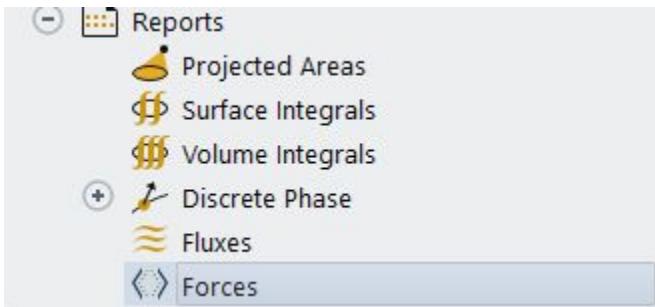
Let's solve for COP and compare  
to OpenRocket's value of **29.4cm**  
from the nose



# Solve for Forces/Moments/COP

Let's solve for COP and compare to OpenRocket's value of **29.4cm** from the nose

**x=0** - middle plane of the rocket



Center of Pressure - Set Coordinate x = 0 (m)			
Zone	y	z	
body	-1.7175273e-08	0.27724426	
fin-edges	6.9188825e-07	0.43522874	
fin-faces	-6.2408116e-05	0.34670232	
nose	-1.1326556e-08	0.07121665	
Net	-2.840451e-05	0.25710413	

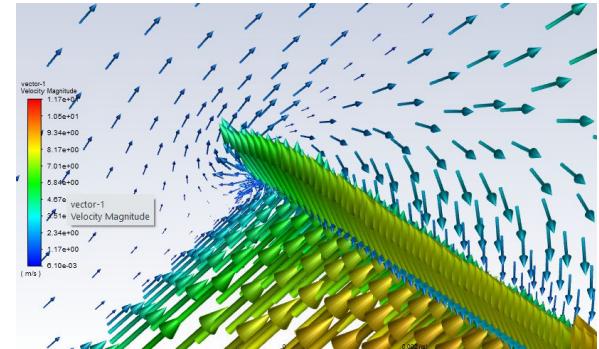
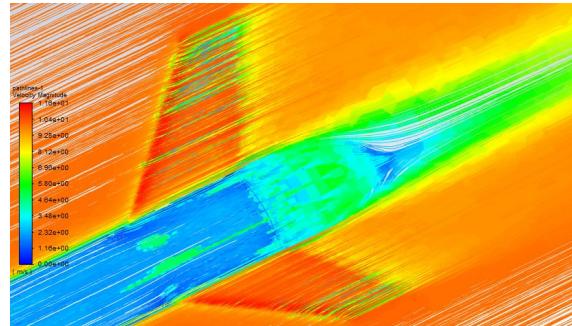
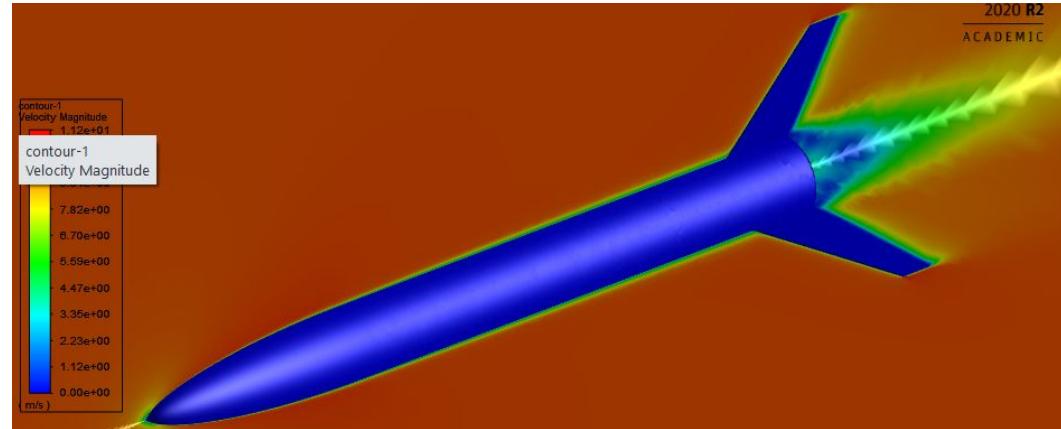
**25.7cm** - Not bad!!!

# The Fun Part!!!

Now you can play around with cool visuals

Try contours, pathlines, vectors around various bodies

- Do your contours have **sharp gradients?**
- Is your **mesh fine enough** to capture these?



# Next workshop - Week 7 - Ansys FEA

See you next week! :)

