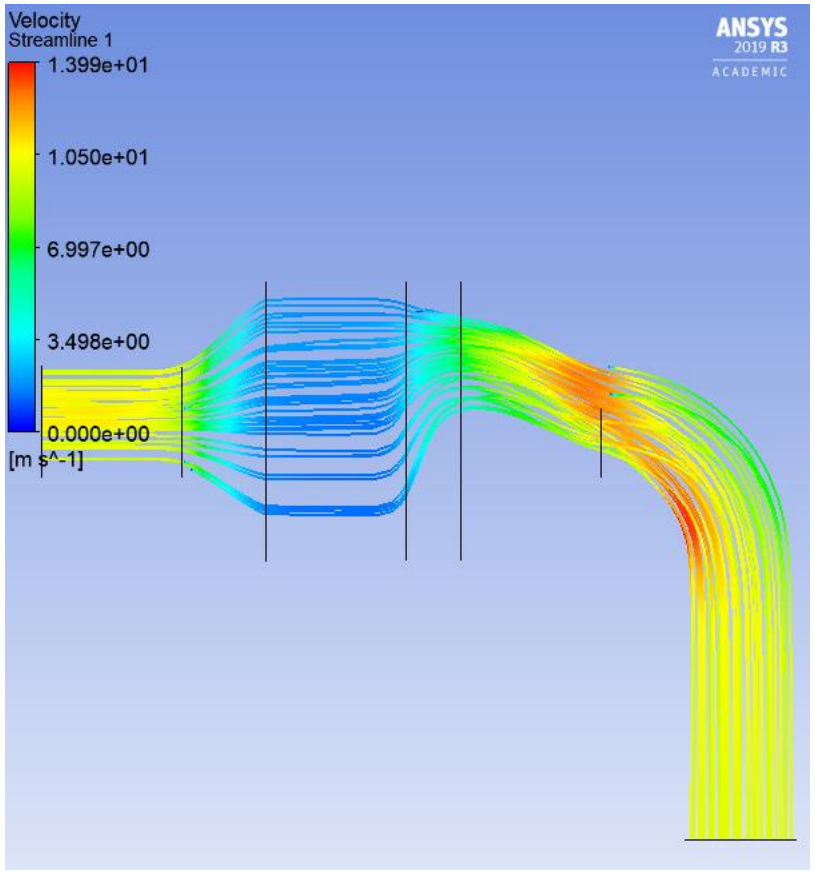
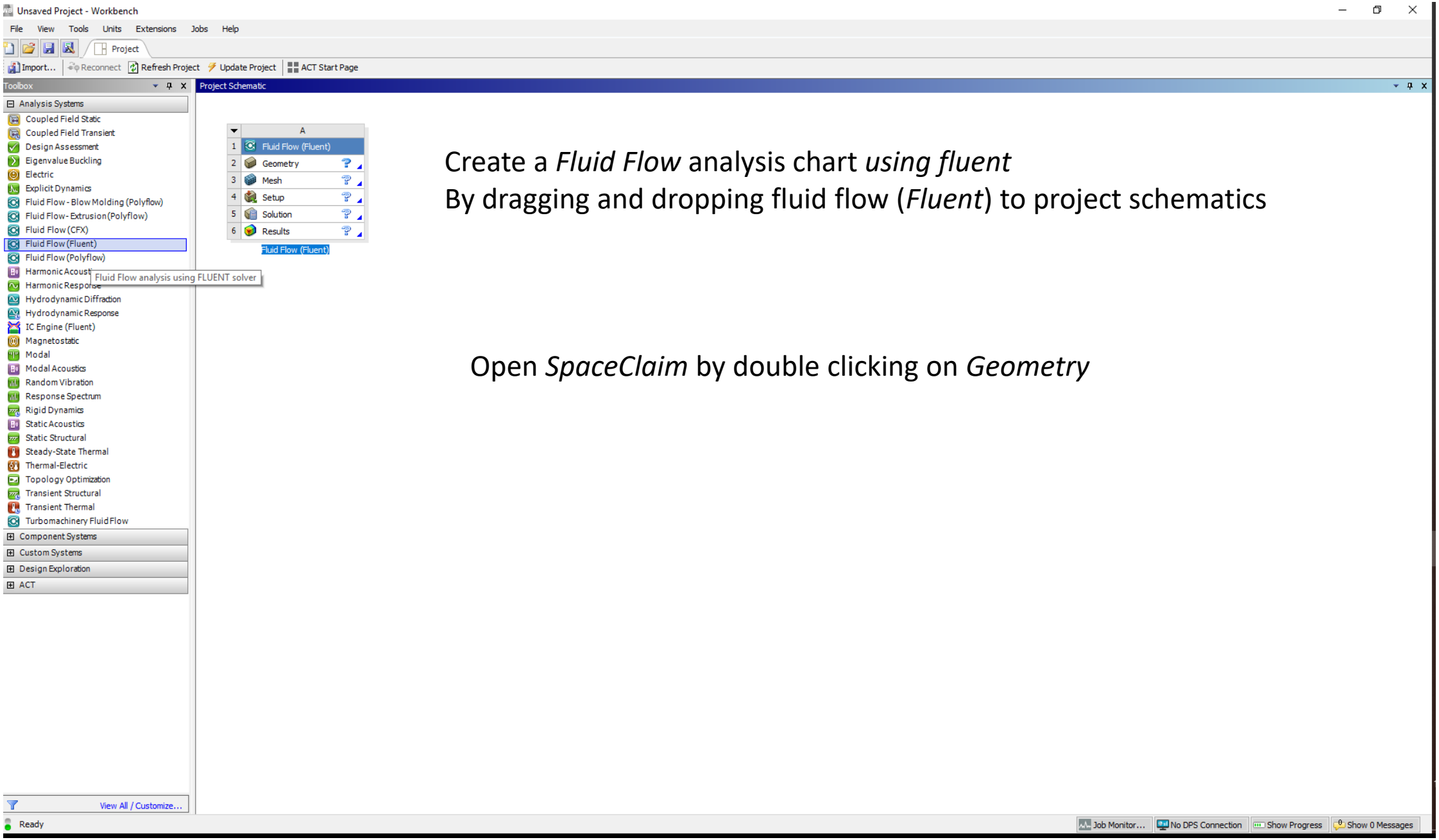
Edited by Kristóf Tokaji (tokaji@ara.bme.hu)

3rd Practice
– Catalytic converter (3D, porous zone)

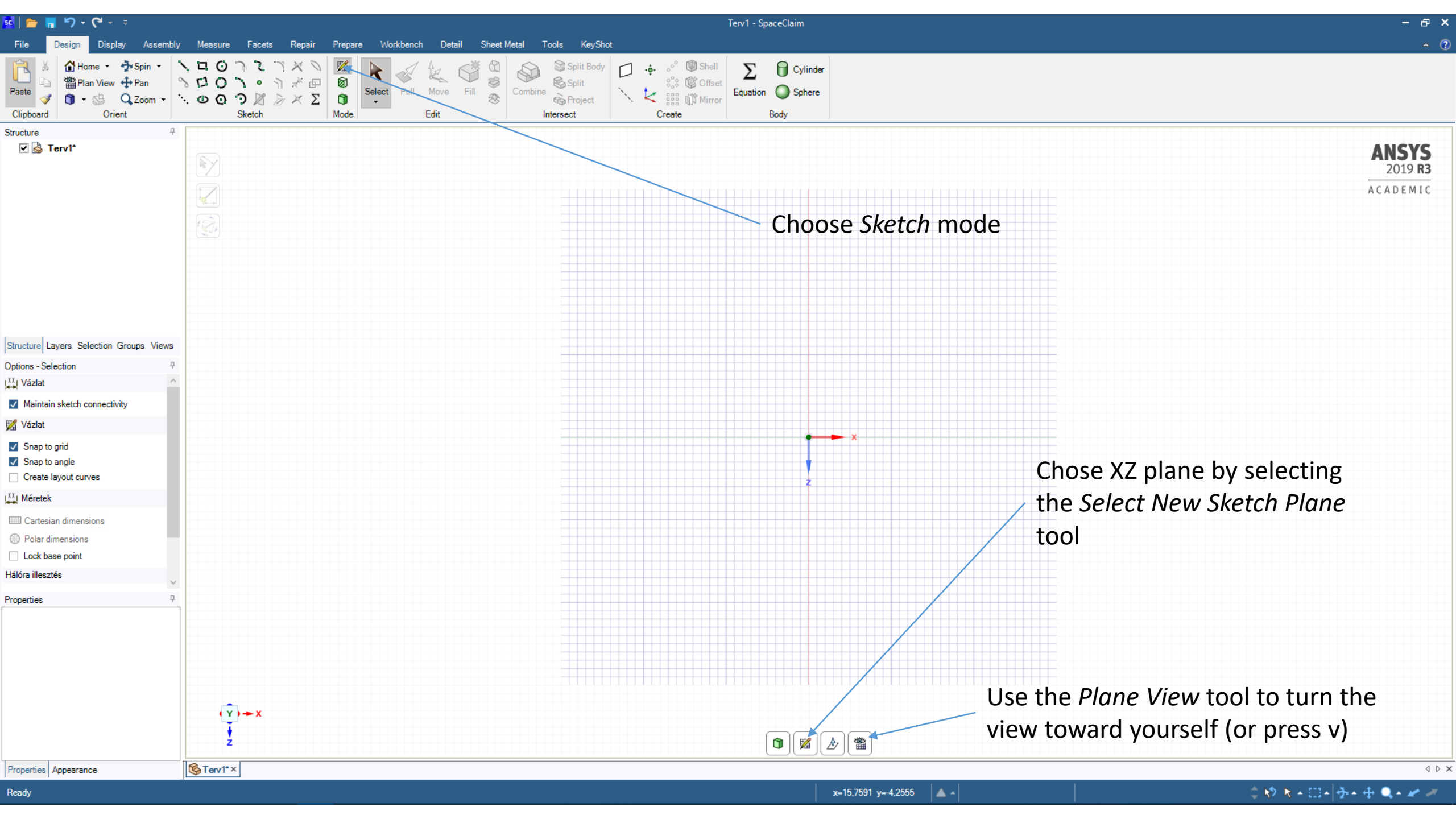


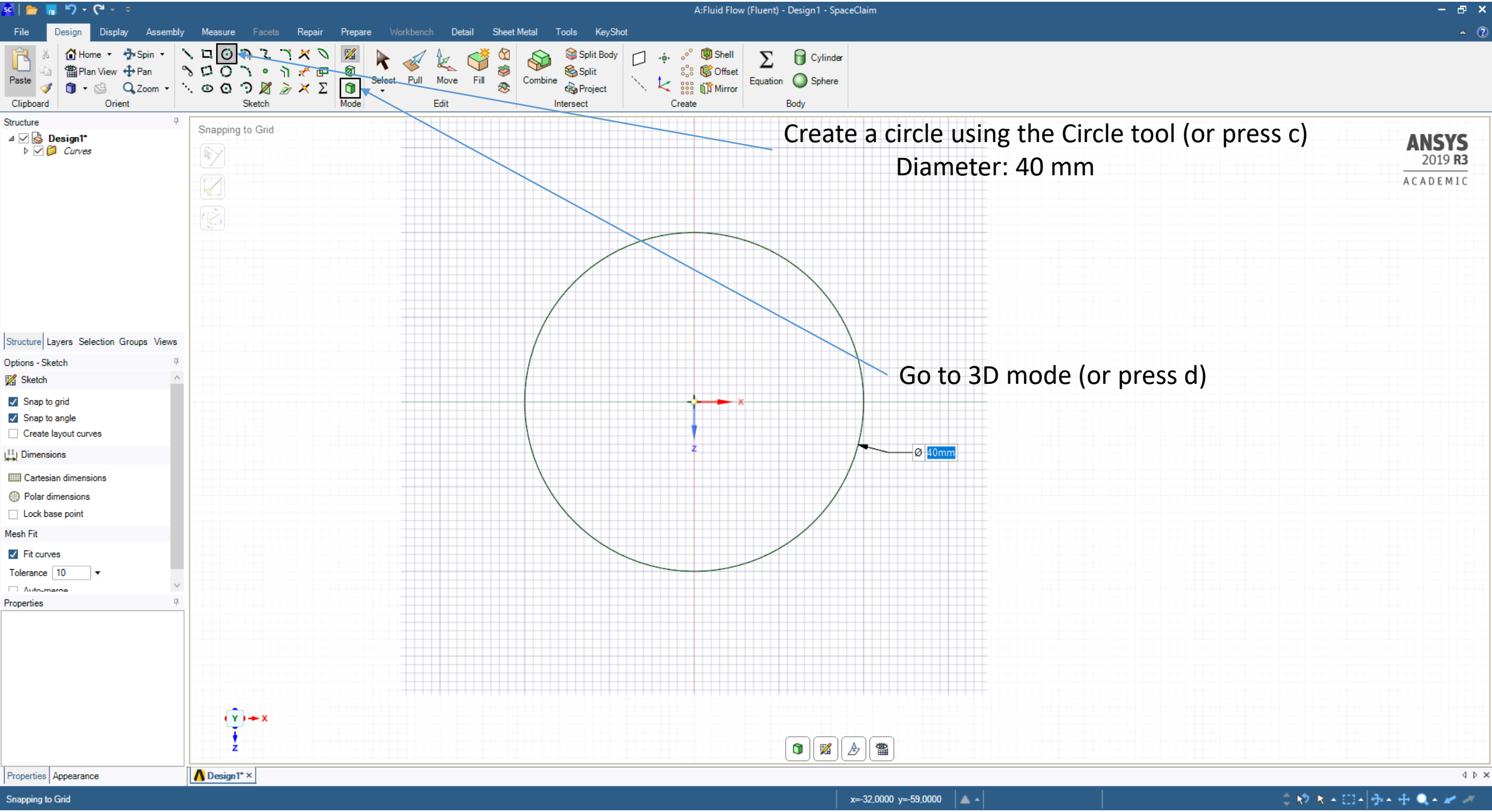
Create a fluid flow analysis



Create a *Fluid Flow* analysis chart *using fluent*
By dragging and dropping fluid flow (*Fluent*) to project schematics

Open *SpaceClaim* by double clicking on *Geometry*


Terv1 - SpaceClaim
File
Design
Display
Assembly
Measure
Facets
Repair
Prepare
Workbench
Detail
Sheet Metal
Tools
KeyShot
Paste
Clipboard
Home
Spin
Plan View
Pan
Zoom
Orient
Sketch
Mode
Select
Pull
Move
Fill
Edit
Combine
Split Body
Split
Project
Intersect
Shell
Offset
Mirror
Create
Equation
Cylinder
Sphere
Body
Structure
Terv1*
Structure
Layers
Selection
Groups
Views
Options - Selection
Vázlat
Maintain sketch connectivity
Vázlat
Snap to grid
Snap to angle
Create layout curves
Méretek
Cartesian dimensions
Polar dimensions
Lock base point
Hálóra illesztés
Properties
Properties
Appearance
ANSYS
2019 R3
ACADEMIC
Choose *Sketch* mode
Chose XZ plane by selecting the *Select New Sketch Plane* tool
Use the *Plane View* tool to turn the view toward yourself (or press v)
Terv1*
Ready
x=15,7591 y=-4,2555


A:Fluid Flow (Fluent) - Design1 - SpaceClaim
File
Design
Display
Assembly
Measure
Facets
Repair
Prepare
Workbench
Detail
Sheet Metal
Tools
KeyShot
Paste
Clipboard
Home
Spin
Plan View
Pan
Zoom
Orient
Sketch
Mode
Select
Pull
Move
Fill
Edit
Combine
Split Body
Split
Project
Intersect
Shell
Offset
Mirror
Create
Equation
Cylinder
Sphere
Body
Structure
Design1*
Curves
Structure
Layers
Selection
Groups
Views
Options - Sketch
Sketch
Snap to grid
Snap to angle
Create layout curves
Dimensions
Cartesian dimensions
Polar dimensions
Lock base point
Mesh Fit
Fit curves
Tolerance
10
Properties
Properties
Appearance
Snapping to Grid
Create a circle using the Circle tool (or press c)
Diameter: 40 mm
Go to 3D mode (or press d)
Ø 40mm
ANSYS
2019 R3
ACADEMIC
Design1*
Snapping to Grid
x=-32,0000 y=-59,0000

A:Fluid Flow (Fluent) - Design1 - SpaceClaim

File | Design | Display | Assembly | Measure | Facets | Repair | Prepare | Workbench | Detail | Sheet Metal | Tools | KeyShot

Clipboard | Orient | Sketch | Mode | Edit | Intersect | Create | Body

Select | Pull | Move | Fill | Combine | Split Body | Split | Project | Shell | Offset | Mirror | Equation | Cylinder | Sphere

Structure
Design1*
Solid

Structure | Layers | Selection | Groups | Views

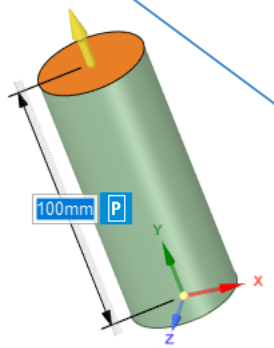
Options - Pull
General
Add | Cut | No merge

Properties
Appearance
Color ARGB: 255, 143, 175
Style By Layer, By Style
Face Type
Shape Plane
Chamfer False

Properties | Appearance

Pull 1 face

ANSYS 2019 R3 ACADEMIC



Pull the generated surface using the Pull tool (or press p)

Distance: 100 mm

Design1*

Pull 1 face | 1 Face

A:Fluid Flow (Fluent) - Design1 - SpaceClaim

File | Design | Display | Assembly | Measure | Facets | Repair | Prepare | Workbench | Detail | Sheet Metal | Tools | KeyShot

Clipboard: Paste
Orient: Home, Spin, Plan View, Pan, Zoom
Sketch
Mode
Edit: Select, Pull, Move, Fill
Intersect: Combine, Split Body, Split, Project
Create: Shell, Offset, Mirror
Body: Equation, Cylinder, Sphere

Structure
Design1*
Solid

Structure | Layers | Selection | Groups | Views

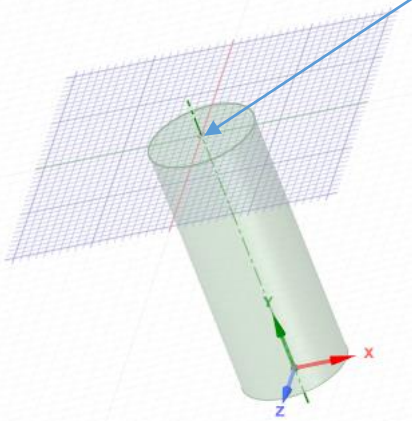
Options - Selection
Sketch
Maintain sketch connectivity
Sketch
Snap to grid
Snap to angle
Create layout curves
Dimensions
Cartesian dimensions
Polar dimensions
Lock base point
Mesh Fit

Properties

Properties | Appearance

Click to select. Double-click to select a tangent chain. Double-click again to select a closed loop. Drag to edit your sketch.

ANSYS 2019 R3 ACADEMIC



Choose Sketch mode (or press k) and select the upper surface of the cylinder

Use the *Plane View* tool to turn the view toward yourself (or press v)

Design1* ×

Click to select. Double-click to select a tangent chain. Double-click again to select a closed loop. Drag to edit your sketch.

x=20.9920 y=44.3503

A:Fluid Flow (Fluent) - Design1 - SpaceClaim

File | Design | Display | Assembly | Measure | Facets | Repair | Prepare | Workbench | Detail | Sheet Metal | Tools | KeyShot

Clipboard: Paste
Orient: Home, Spin, Plan View, Pan, Zoom
Sketch
Mode
Edit: Select, Pull, Move, Fill
Intersect: Combine, Split Body, Split, Project
Create: Shell, Offset, Mirror
Body: Equation, Cylinder, Sphere

Structure
- Design1*
  - Solid
  - Curves

Structure | Layers | Selection | Groups | Views

Options - Selection
- Sketch
  - Maintain sketch connectivity
- Sketch
  - Snap to grid
  - Snap to angle
  - Create layout curves
- Dimensions
  - Cartesian dimensions
  - Polar dimensions
  - Lock base point
- Mesh Fit

Properties

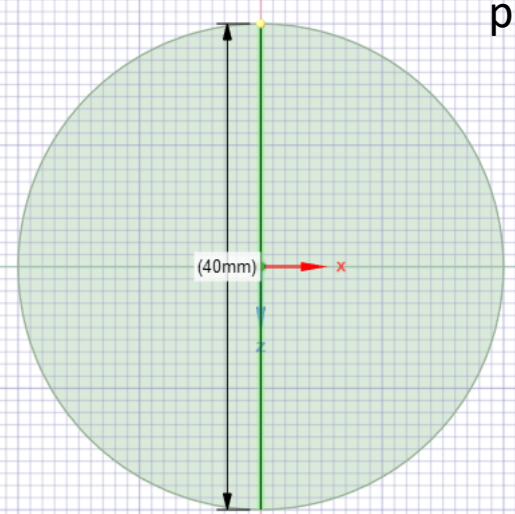
Appearance	
Color	143; 175; 143
Layout	False
Misc	
Construction	False
Mirror	False

Properties | Appearance

Click to select. Double-click to select a tangent chain. Double-click again to select a closed loop. Drag to edit your sketch.

ANSYS 2019 R3 ACADEMIC

Select the Line tool and draw the diameter parallel to z axis

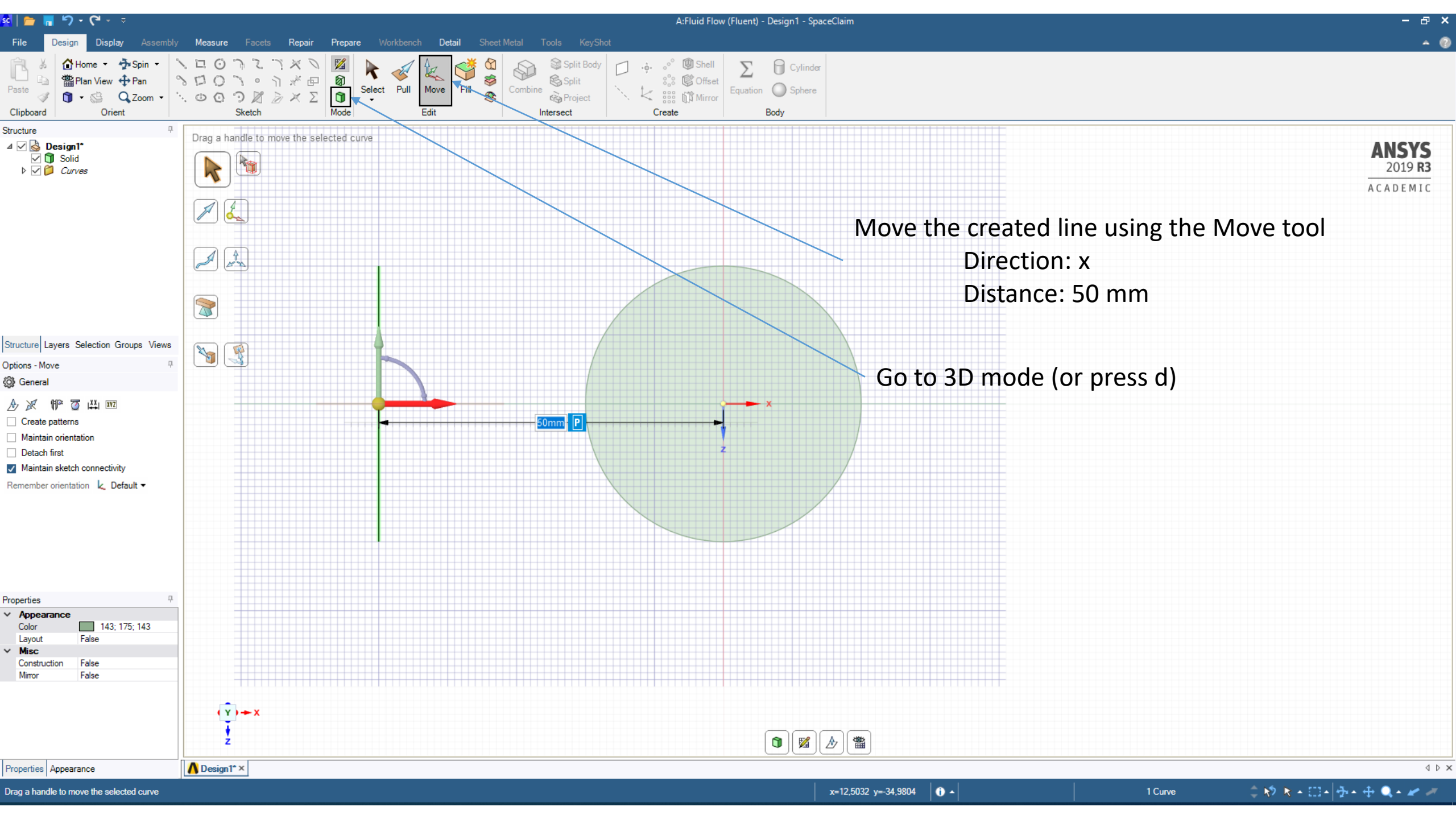


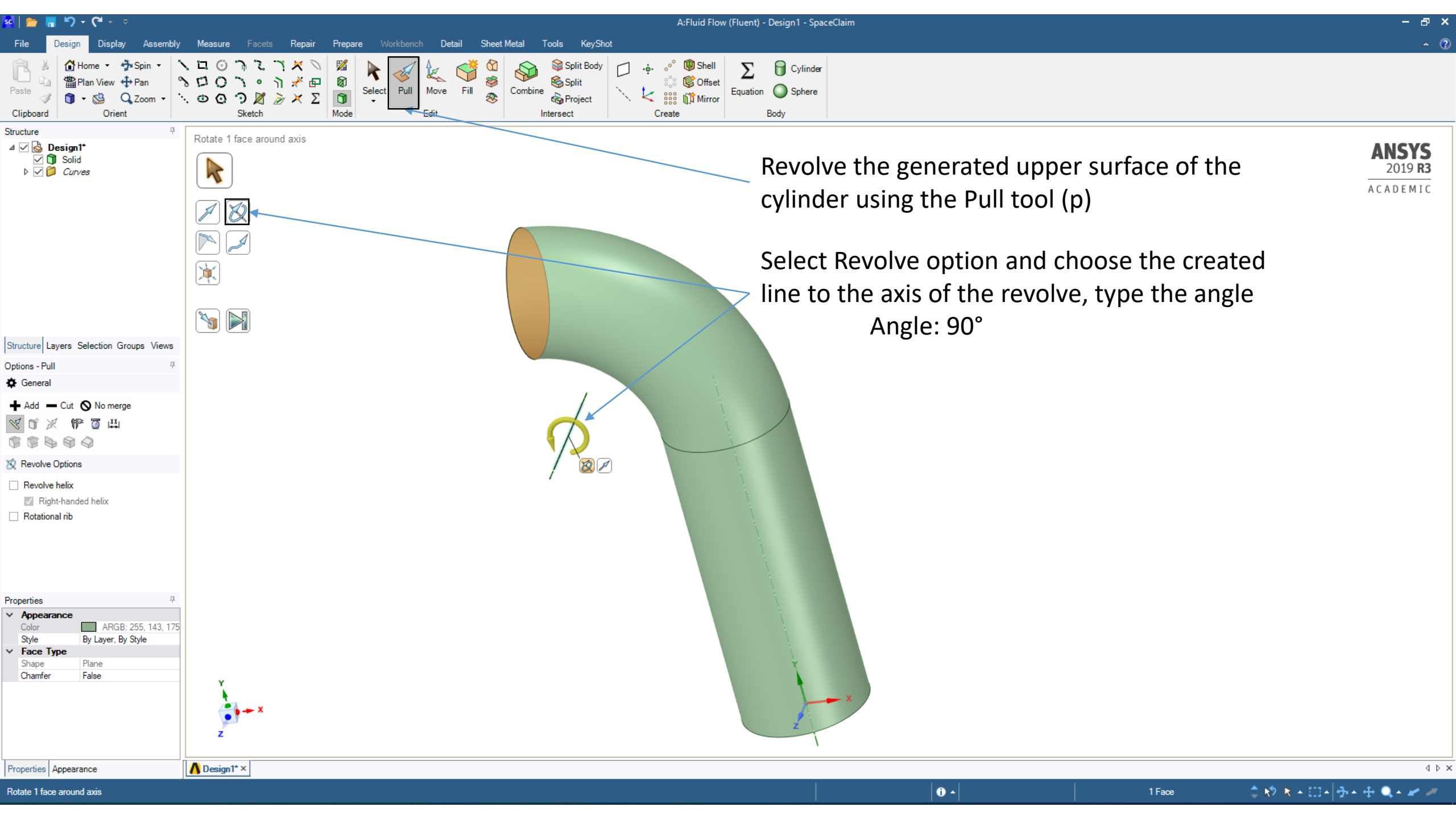
Design1*

Click to select. Double-click to select a tangent chain. Double-click again to select a closed loop. Drag to edit your sketch.

x=12.5032 y=-34.9804

1 Curve


A:Fluid Flow (Fluent) - Design1 - SpaceClaim
File
Design
Display
Assembly
Measure
Facets
Repair
Prepare
Workbench
Detail
Sheet Metal
Tools
KeyShot
Paste
Clipboard
Home
Spin
Plan View
Pan
Zoom
Orient
Sketch
Mode
Select
Pull
Move
Fill
Edit
Combine
Split Body
Split
Project
Intersect
Shell
Offset
Mirror
Create
Equation
Cylinder
Sphere
Body
Structure
Design1*
Solid
Curves
Drag a handle to move the selected curve
ANSYS
2019 R3
ACADEMIC
Move the created line using the Move tool
Direction: x
Distance: 50 mm
Go to 3D mode (or press d)
50mm
P
Structure
Layers
Selection
Groups
Views
Options - Move
General
Create patterns
Maintain orientation
Detach first
Maintain sketch connectivity
Remember orientation
Default
Properties
Appearance
Color
143; 175; 143
Layout
False
Misc
Construction
False
Mirror
False
Properties
Appearance
Design1*
Drag a handle to move the selected curve
x=12,5032 y=-34,9804
1 Curve


A:Fluid Flow (Fluent) - Design1 - SpaceClaim
File
Design
Display
Assembly
Measure
Facets
Repair
Prepare
Workbench
Detail
Sheet Metal
Tools
KeyShot
Paste
Clipboard
Home
Plan View
Zoom
Spin
Pan
Orient
Sketch
Mode
Select
Pull
Move
Fill
Edit
Combine
Split Body
Split
Project
Intersect
Shell
Offset
Mirror
Create
Equation
Cylinder
Sphere
Body
Structure
Design1*
Solid
Curves
Structure
Layers
Selection
Groups
Views
Options - Pull
General
Add
Cut
No merge
Revolve Options
Revolve helix
Right-handed helix
Rotational rib
Properties
Appearance
Color
ARGB: 255, 143, 175
Style
By Layer, By Style
Face Type
Shape
Plane
Chamfer
False
Properties
Appearance
Rotate 1 face around axis
Design1*
Rotate 1 face around axis
1 Face
ANSYS
2019 R3
ACADEMIC
Revolve the generated upper surface of the cylinder using the Pull tool (p)
Select Revolve option and choose the created line to the axis of the revolve, type the angle
Angle: 90°

A:Fluid Flow (Fluent) - Design1 - SpaceClaim

File | Design | Display | Assembly | Measure | Facets | Repair | Prepare | Workbench | Detail | Sheet Metal | Tools | KeyShot

Clipboard: Paste
Orient: Home, Spin, Plan View, Pan, Zoom
Sketch
Mode
Edit: Select, Pull, Move, Fill
Intersect: Combine, Split Body, Split, Project
Create: Shell, Offset, Mirror
Body: Equation, Cylinder, Sphere

Structure
- Design1*
  - Solid
  - Plane

Structure | Layers | Selection | Groups | Views

Options - Selection

Properties

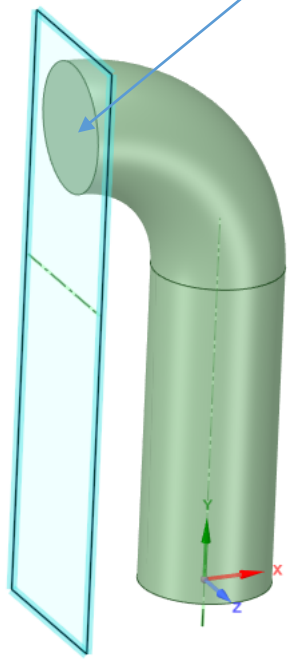
Appearance	
Clip face	
Fill	Automatic
Include Curves	True
Outline	Automatic
Misc	
Plane	Plane: Origin = (-0.05, 0.15,

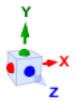
Properties | Appearance

Click an object. Double-click to select an edge loop. Triple-click to select a solid.

ANSYS 2019 R3 ACADEMIC

Create a plane using the Plane tool

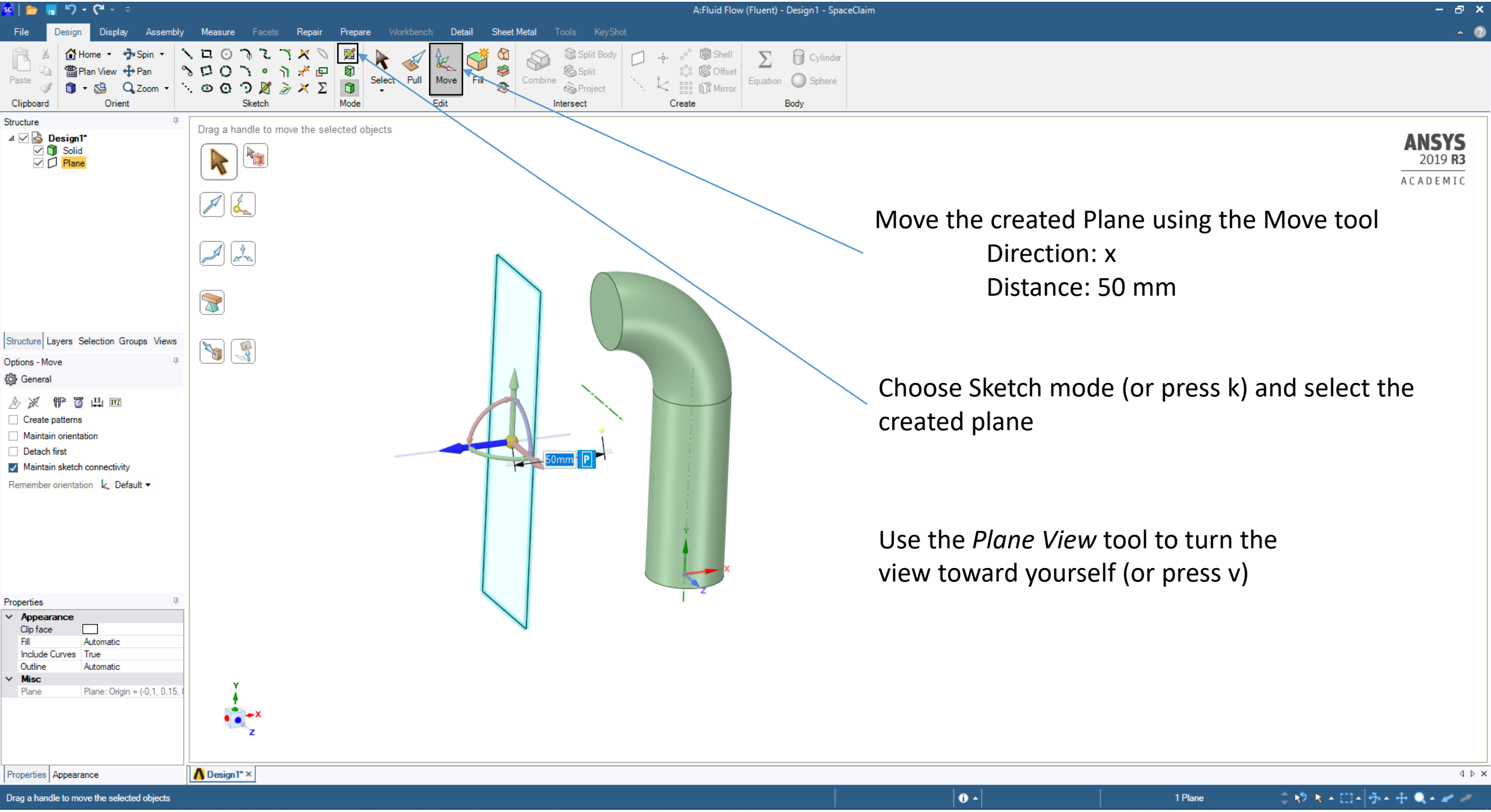




Design1* ×

Click an object. Double-click to select an edge loop. Triple-click to select a solid.

1 Plane



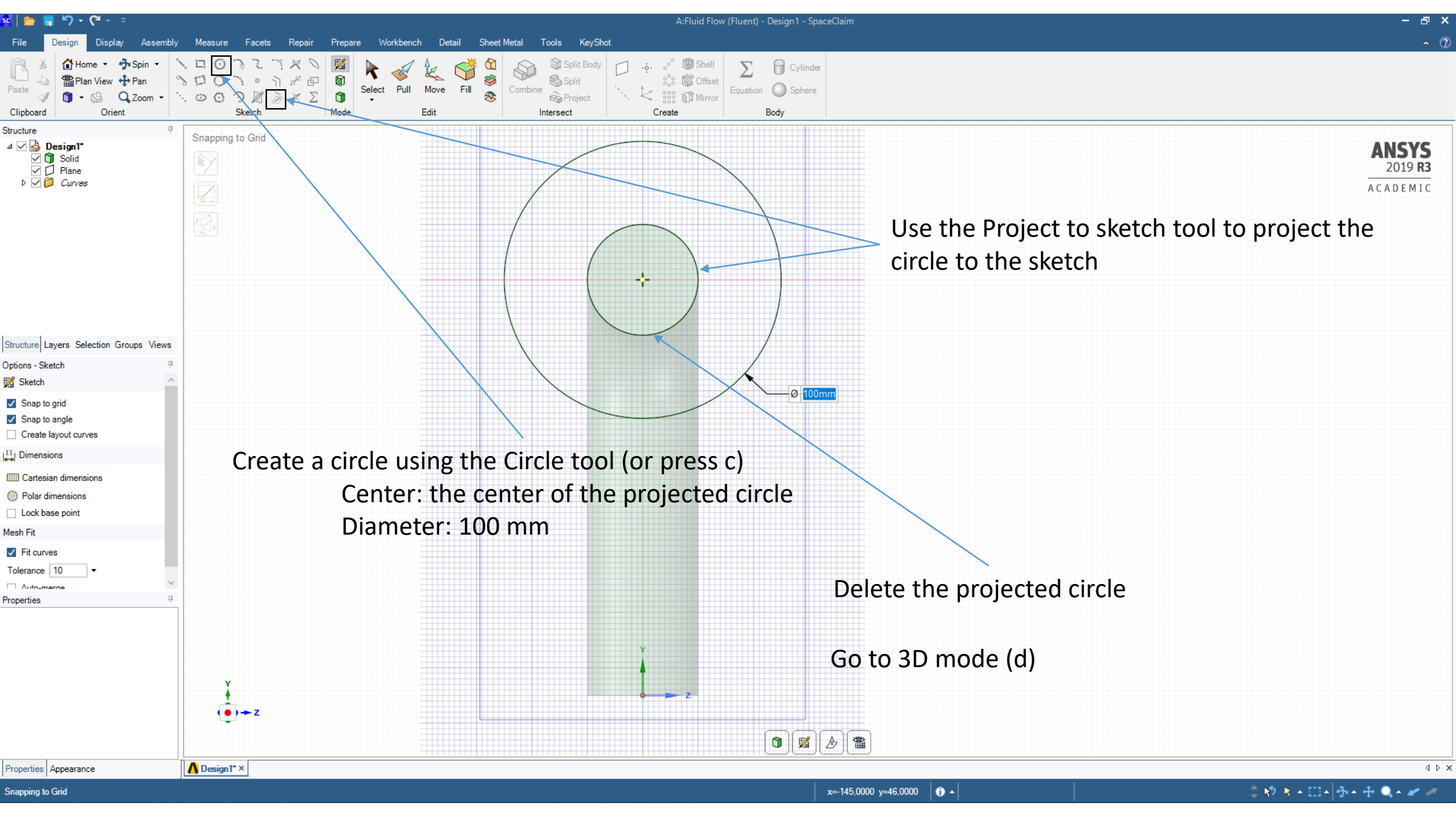
Move the created Plane using the Move tool

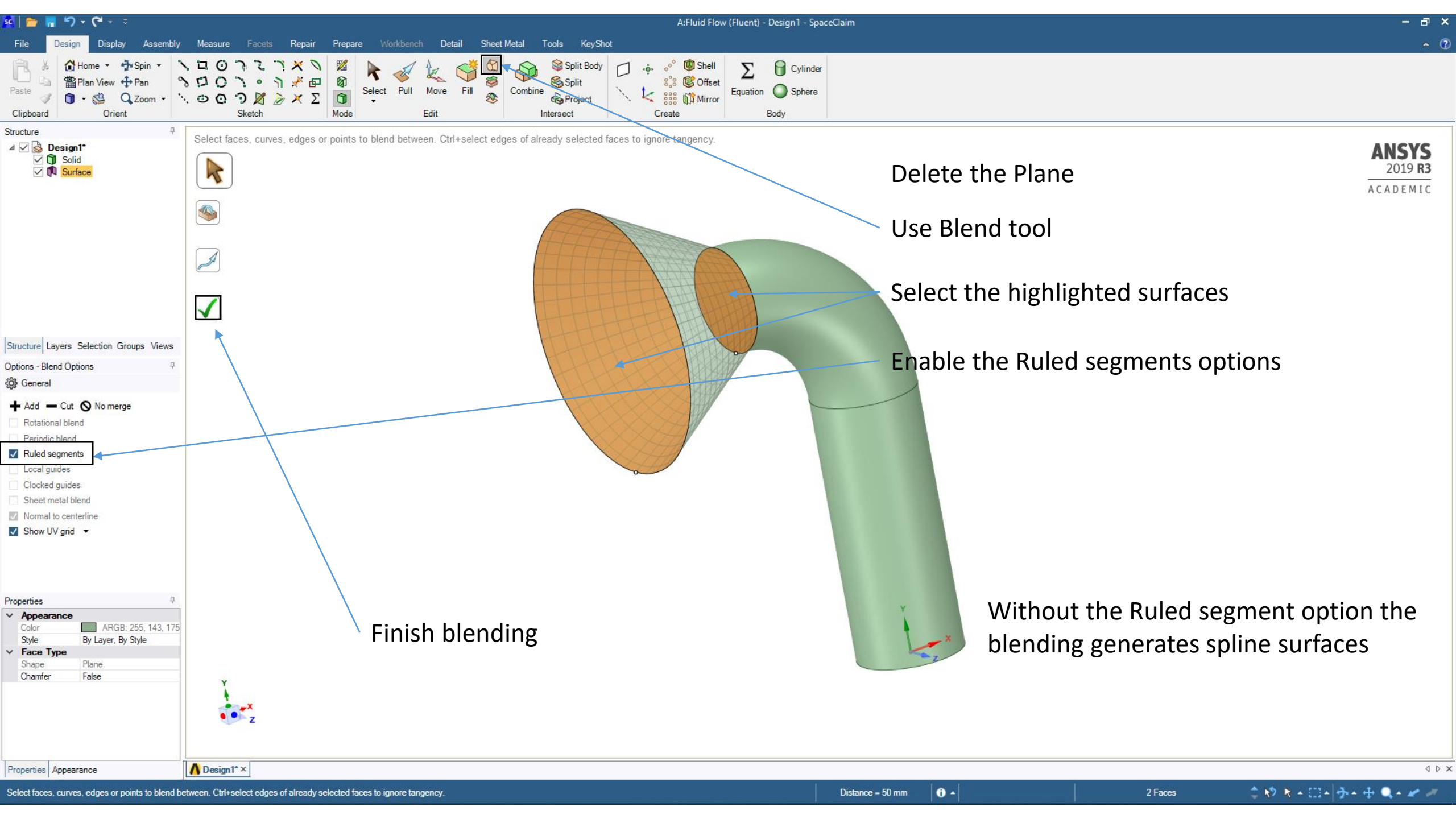
Direction: x

Distance: 50 mm

Choose Sketch mode (or press k) and select the created plane

Use the *Plane View* tool to turn the view toward yourself (or press v)


A:Fluid Flow (Fluent) - Design1 - SpaceClaim
File
Design
Display
Assembly
Measure
Facets
Repair
Prepare
Workbench
Detail
Sheet Metal
Tools
KeyShot
Clipboard
Orient
Sketch
Mode
Edit
Intersect
Create
Body
Structure
Design1*
Solid
Plane
Curves
Snapping to Grid
ANSYS
2019 R3
ACADEMIC
Use the Project to sketch tool to project the circle to the sketch
Ø 100mm
Create a circle using the Circle tool (or press c)
Center: the center of the projected circle
Diameter: 100 mm
Delete the projected circle
Go to 3D mode (d)
Options - Sketch
Snap to grid
Snap to angle
Create layout curves
Dimensions
Cartesian dimensions
Polar dimensions
Lock base point
Mesh Fit
Fit curves
Tolerance 10
Properties
Snapping to Grid
x=-145.0000 y=46.0000


A:Fluid Flow (Fluent) - Design1 - SpaceClaim
File Design Display Assembly Measure Facets Repair Prepare Workbench Detail Sheet Metal Tools KeyShot
Clipboard Orient Sketch Mode Edit Intersect Create Body
Structure
Design1*
Solid
Surface
Select faces, curves, edges or points to blend between. Ctrl+select edges of already selected faces to ignore tangency.
ANSYS 2019 R3 ACADEMIC
Structure Layers Selection Groups Views
Options - Blend Options
General
Add Cut No merge
Rotational blend
Periodic blend
Ruled segments
Local guides
Clocked guides
Sheet metal blend
Normal to centerline
Show UV grid
Properties
Appearance
Color ARGB: 255, 143, 175
Style By Layer, By Style
Face Type
Shape Plane
Chamfer False
Properties Appearance
Design1*
Select faces, curves, edges or points to blend between. Ctrl+select edges of already selected faces to ignore tangency.
Distance = 50 mm
2 Faces
Delete the Plane
Use Blend tool
Select the highlighted surfaces
Enable the Ruled segments options
Finish blending
Without the Ruled segment option the blending generates spline surfaces

A:Fluid Flow (Fluent) - Design1 - SpaceClaim

File | Design | Display | Assembly | Measure | Facets | Repair | Prepare | Workbench | Detail | Sheet Metal | Tools | KeyShot

Paste | Clipboard | Home | Plan View | Spin | Pan | Zoom | Orient | Sketch | Mode | Select | Pull | Move | Fill | Edit | Combine | Split Body | Split | Project | Intersect | Shell | Offset | Mirror | Create | Equation | Cylinder | Sphere | Body

Structure

- Design1*
  - Solid
  - Surface

Structure | Layers | Selection | Groups | Views

Options - Selection

Properties

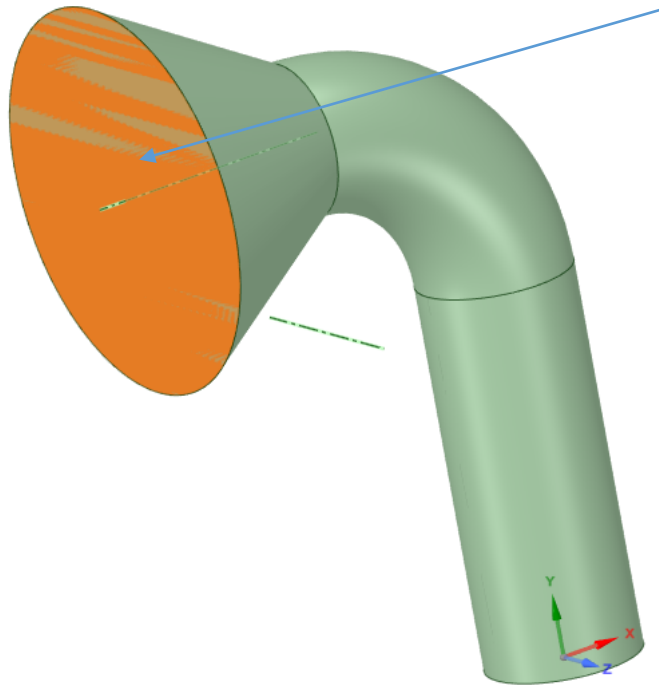
Appearance	
Color	ARGB: 255, 143, 175
Style	By Layer, By Style
Tessellation Qua	5
Material	
Material Name	Unknown Material
Fluid	False
Density	None
Midsurface	
Driving Dimensic	
Offset Type	
Thickness	

Properties | Appearance

Click an object. Double-click to select an edge loop. Triple-click to select a solid.

ANSYS 2019 R3 ACADEMIC

Copy and Paste the selected surface



Design1*

Click an object. Double-click to select an edge loop. Triple-click to select a solid.

1 Body

A:Fluid Flow (Fluent) - Design1 - SpaceClaim

File | Design | Display | Assembly | Measure | Facets | Repair | Prepare | Workbench | Detail | Sheet Metal | Tools | KeyShot

Clipboard | Orient | Sketch | Mode | Edit | Intersect | Create | Body

Select | Pull | Move | Fill | Combine | Split Body | Split | Project | Shell | Offset | Mirror | Equation | Cylinder | Sphere

Structure
Design1*
Solid

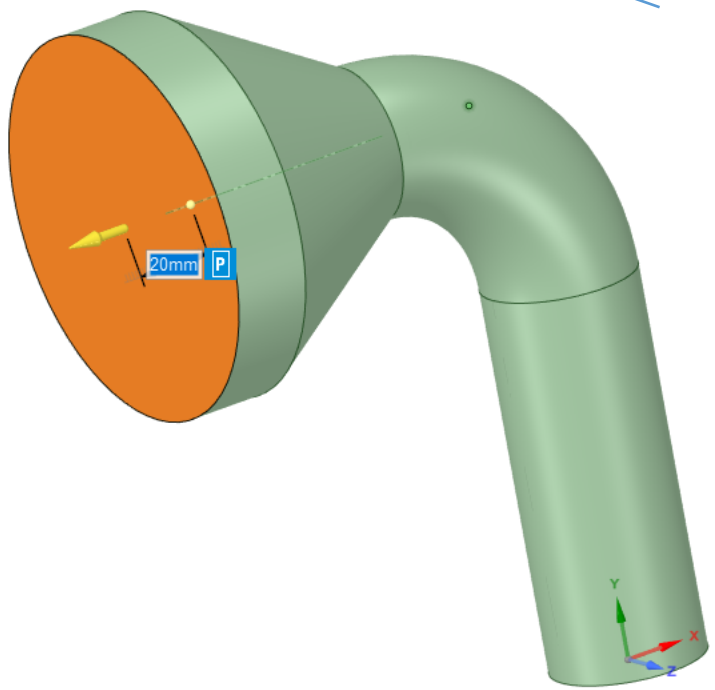
Structure | Layers | Selection | Groups | Views

Options - Pull
General
Add | Cut | No merge

Properties
Appearance
Color ARGB: 255, 143, 175
Style By Layer, By Style
Face Type
Shape Plane
Chamfer False

Properties | Appearance

Pull 1 face



ANSYS 2019 R3 ACADEMIC

Pull the new surface using the Pull tool (p)
Distance: 20 mm

Without the copy and paste of the surface the pull tool continues the cone shape

Design1*

Pull 1 face | 1 Face

A:Fluid Flow (Fluent) - Design1 - SpaceClaim

File | Design | Display | Assembly | Measure | Facets | Repair | Prepare | Workbench | Detail | Sheet Metal | Tools | KeyShot

Clipboard: Paste
Orient: Home, Spin, Plan View, Pan, Zoom
Sketch
Mode
Edit: Select, Pull, Move, Fill
Intersect: Combine, Split Body, Split, Project
Create: Shell, Offset, Mirror
Body: Equation, Cylinder, Sphere

Structure
- Design1*
  - Solid
  - Surface

Structure | Layers | Selection | Groups | Views

Options - Pull
General
Add | Cut | No merge

Properties

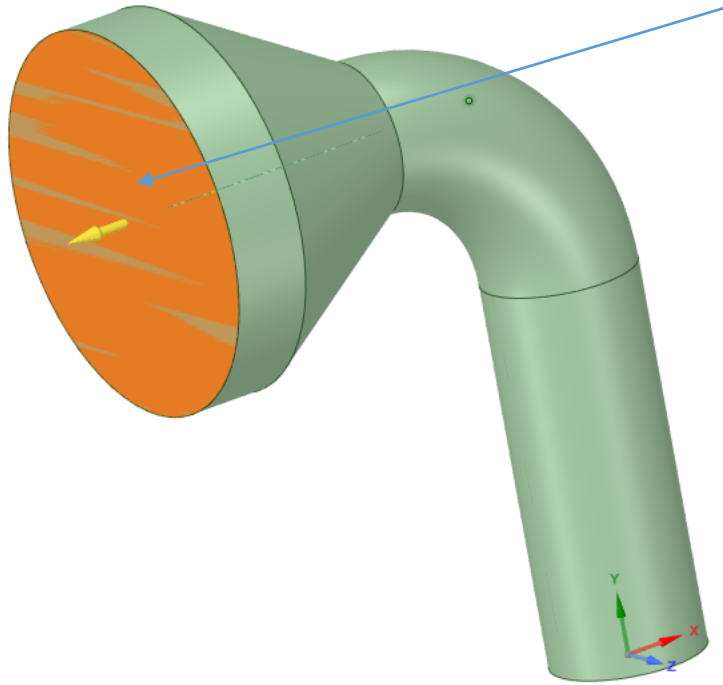
Appearance	
Color	ARGB: 255, 143, 175
Style	By Layer, By Style
Tessellation Qua	5
Material	
Material Name	Unknown Material
Fluid	False
Density	None
Midsurface	
Offset Type	
Thickness	

Properties | Appearance

Pull 1 face

ANSYS 2019 R3 ACADEMIC

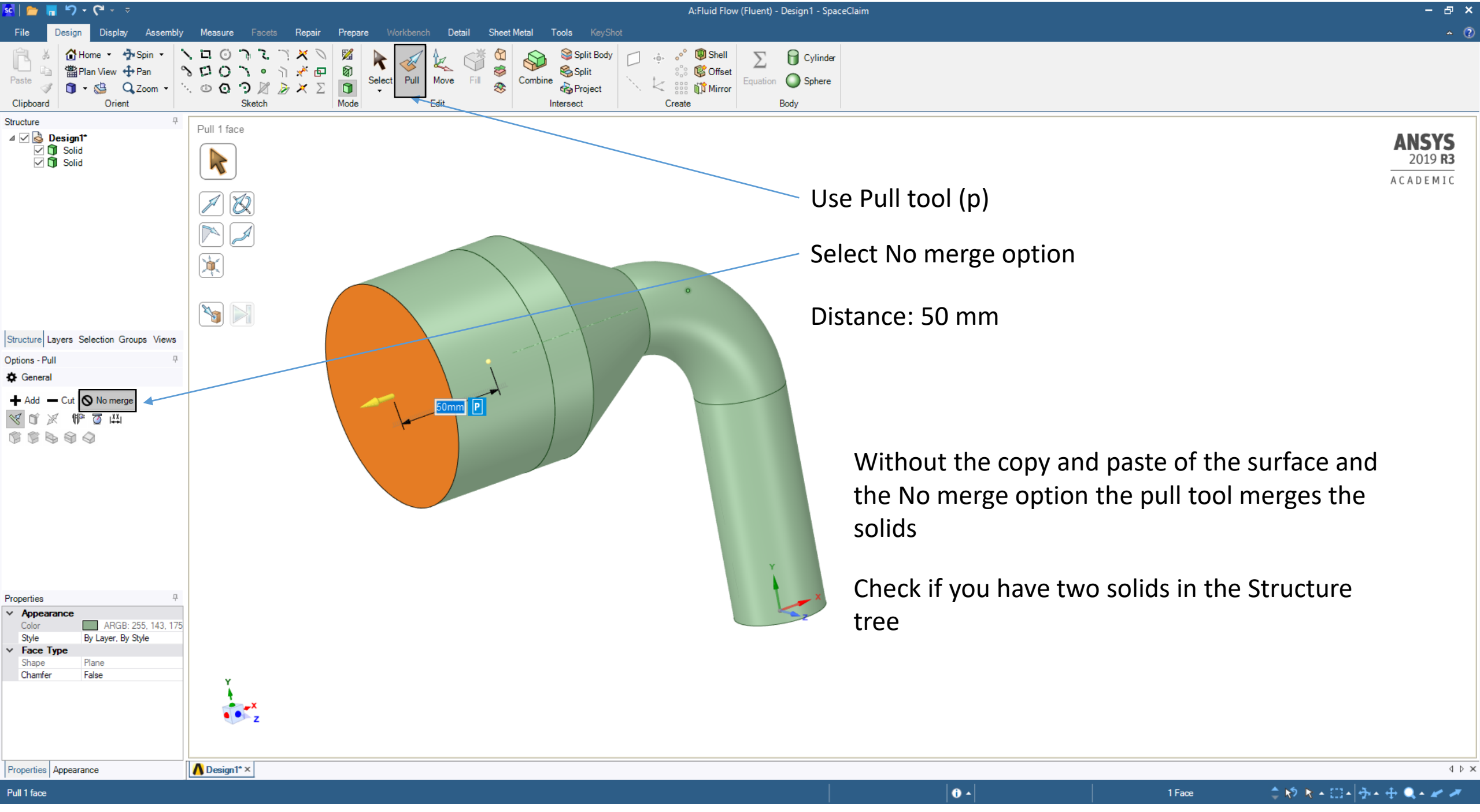
Copy and Paste the selected surface



Design1*

Pull 1 face

1 Body


A:Fluid Flow (Fluent) - Design1 - SpaceClaim
File
Design
Display
Assembly
Measure
Facets
Repair
Prepare
Workbench
Detail
Sheet Metal
Tools
KeyShot
Paste
Clipboard
Home
Spin
Plan View
Pan
Zoom
Orient
Sketch
Mode
Select
Pull
Move
Fill
Edit
Combine
Split Body
Split
Project
Intersect
Shell
Offset
Mirror
Create
Equation
Cylinder
Sphere
Body
Structure
Design1*
Solid
Solid
Structure
Layers
Selection
Groups
Views
Options - Pull
General
Add
Cut
No merge
Properties
Appearance
Color
ARGB: 255, 143, 175
Style
By Layer, By Style
Face Type
Shape
Plane
Chamfer
False
Properties
Appearance
Pull 1 face
50mm
P
ANSYS
2019 R3
ACADEMIC
Use Pull tool (p)
Select No merge option
Distance: 50 mm
Without the copy and paste of the surface and the No merge option the pull tool merges the solids
Check if you have two solids in the Structure tree
Design1*
Pull 1 face
1 Face

A:Fluid Flow (Fluent) - Design1 - SpaceClaim

File | Design | Display | Assembly | Measure | Facets | Repair | Prepare | Workbench | Detail | Sheet Metal | Tools | KeyShot

Clipboard: Paste
Orient: Home, Spin, Plan View, Pan, Zoom
Sketch
Mode
Edit: Select, Pull, Move, Fill
Intersect: Combine, Split Body, Split, Project
Create: Shell, Offset, Mirror
Body: Equation, Cylinder, Sphere

Structure
- Design1*
  - Solid
  - Solid
  - Plane

Structure | Layers | Selection | Groups | Views

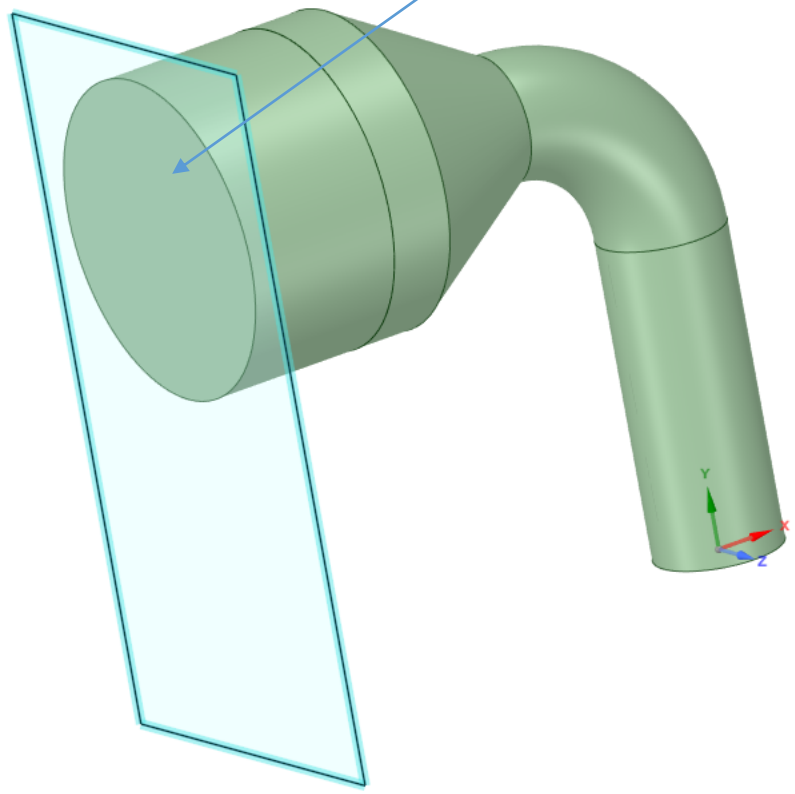
Options - Selection

Properties
- Appearance
  - Clip face
  - Fill: Automatic
  - Include Curves: True
  - Outline: Automatic
- Misc
  - Plane: Plane: Origin = (-0.17, 0.15,

Properties | Appearance

Click an object. Double-click to select an edge loop. Triple-click to select a solid.

Create a plane using the Plane tool

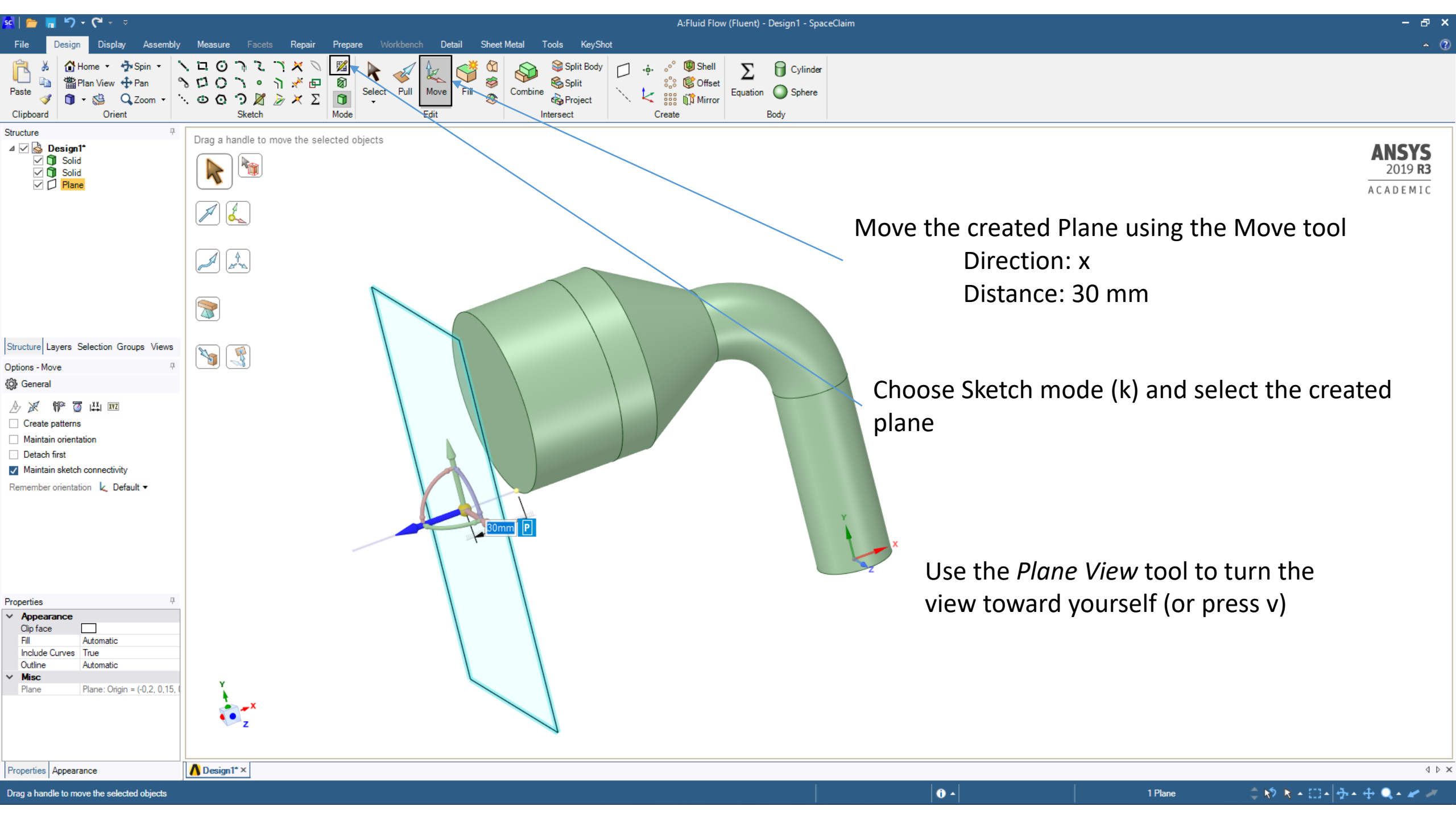


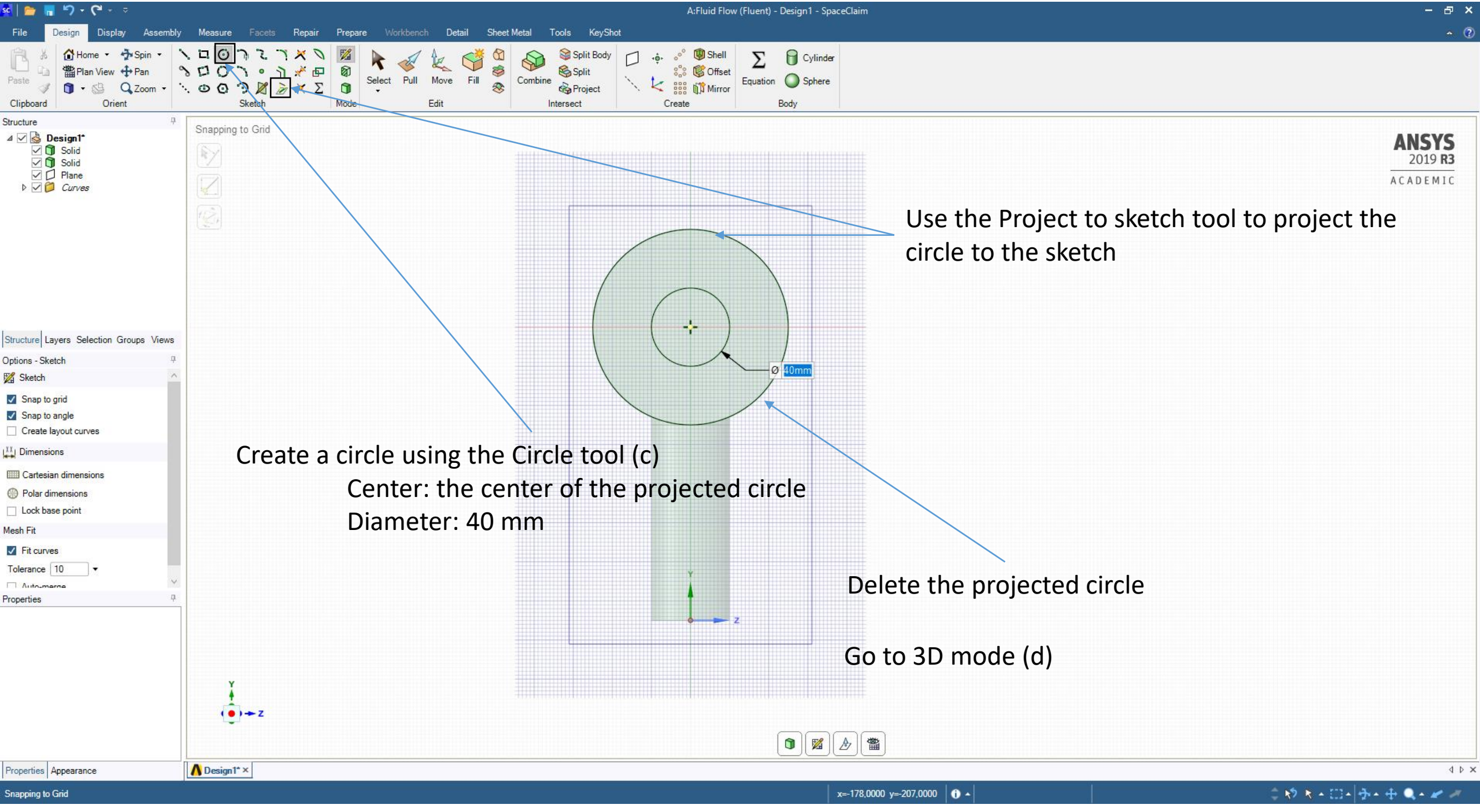
ANSYS 2019 R3 ACADEMIC

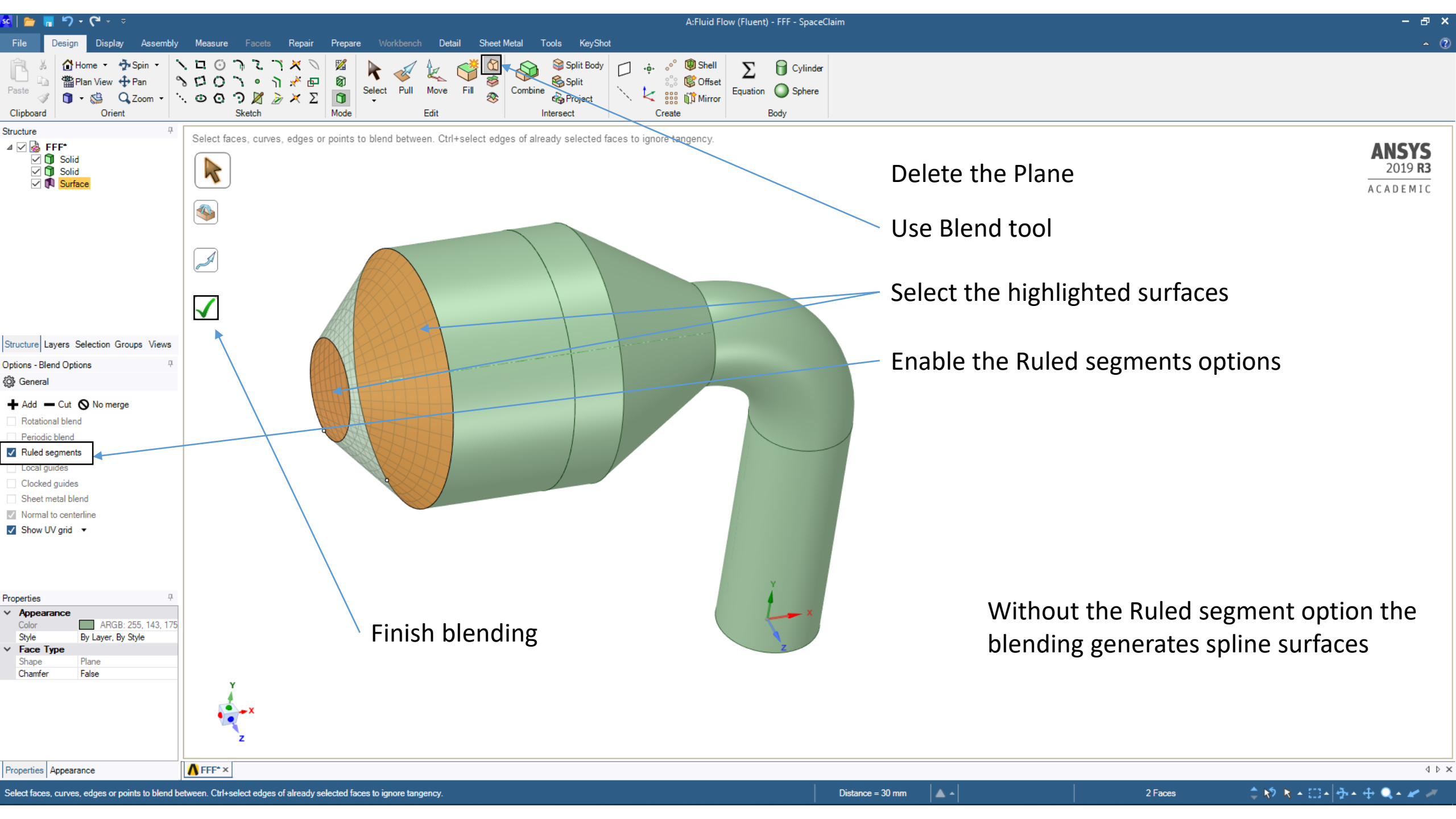
Design1*

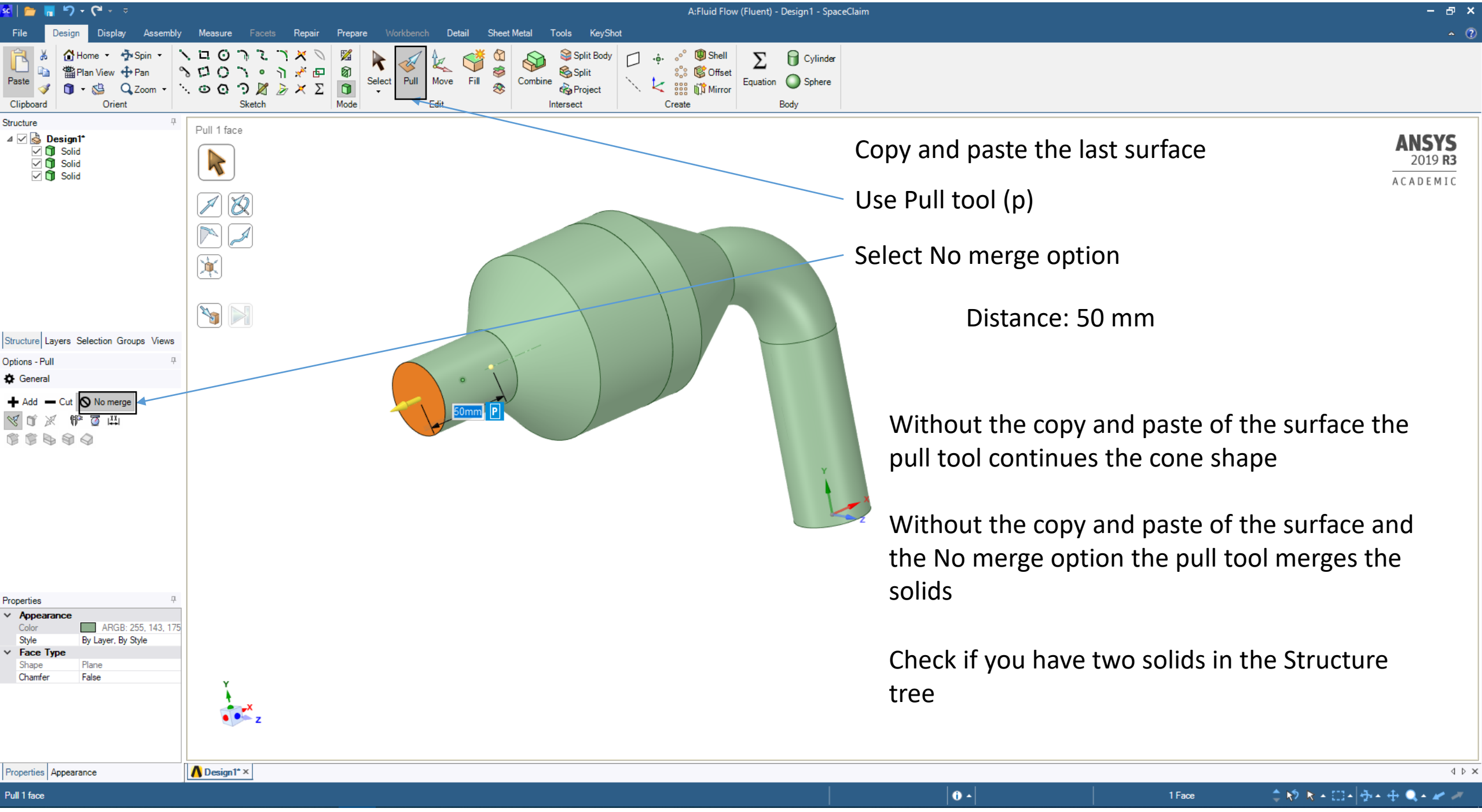
Click an object. Double-click to select an edge loop. Triple-click to select a solid.

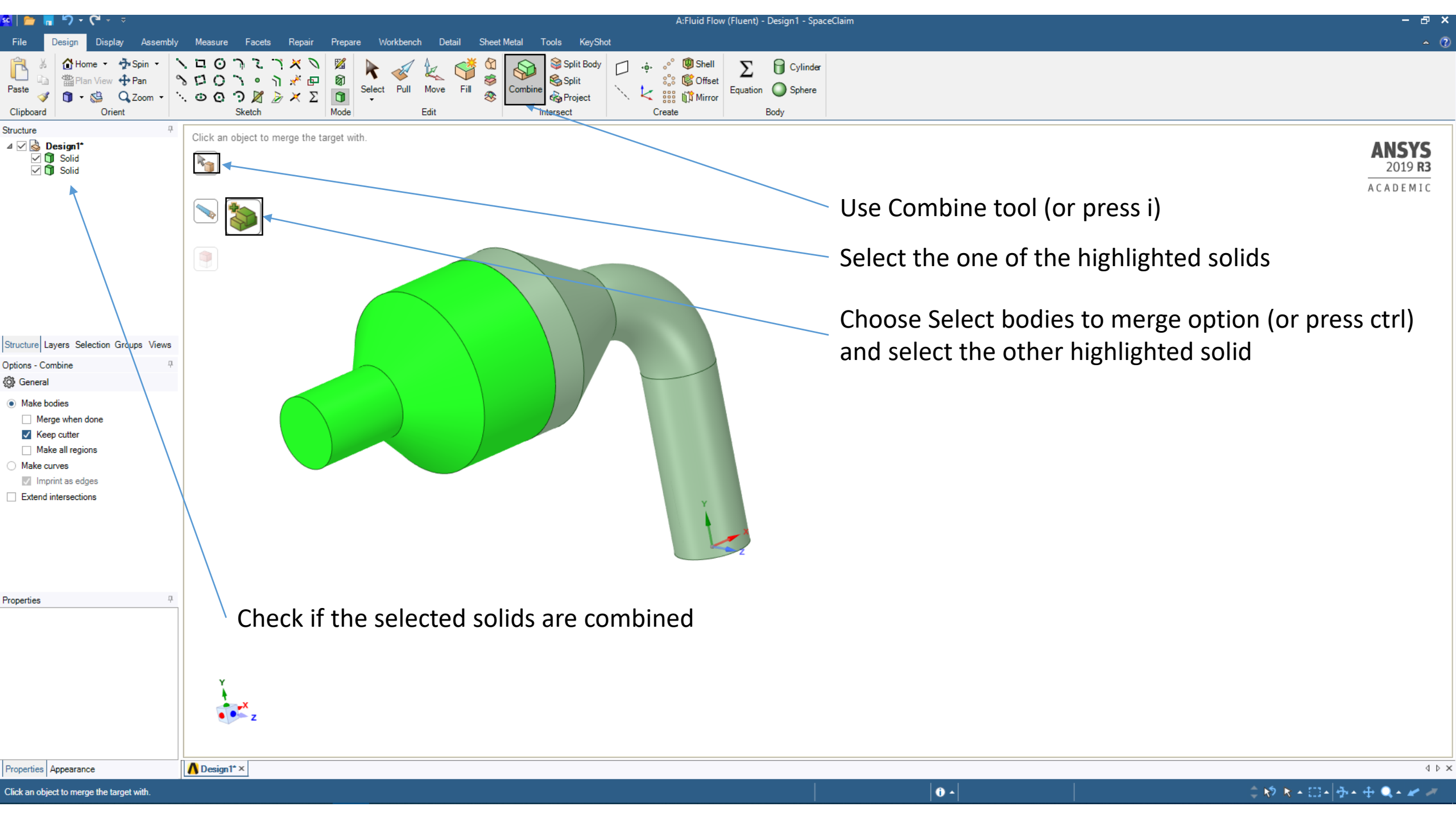
1 Plane


A:Fluid Flow (Fluent) - Design1 - SpaceClaim
File
Design
Display
Assembly
Measure
Facets
Repair
Prepare
Workbench
Detail
Sheet Metal
Tools
KeyShot
Paste
Clipboard
Home
Plan View
Zoom
Spin
Pan
Orient
Sketch
Mode
Select
Pull
Move
Fill
Edit
Combine
Split Body
Split
Project
Intersect
Shell
Offset
Mirror
Create
Equation
Cylinder
Sphere
Body
Structure
Design1*
Solid
Solid
Plane
Drag a handle to move the selected objects
ANSYS
2019 R3
ACADEMIC
Move the created Plane using the Move tool
Direction: x
Distance: 30 mm
Choose Sketch mode (k) and select the created plane
Use the Plane View tool to turn the view toward yourself (or press v)
30mm
P
Structure
Layers
Selection
Groups
Views
Options - Move
General
Create patterns
Maintain orientation
Detach first
Maintain sketch connectivity
Remember orientation
Default
Properties
Appearance
Clip face
Fill
Automatic
Include Curves
True
Outline
Automatic
Misc
Plane
Plane: Origin = (-0.2, 0.15,
Properties
Appearance
Design1*
Drag a handle to move the selected objects
1 Plane


A:Fluid Flow (Fluent) - Design1 - SpaceClaim
File
Design
Display
Assembly
Measure
Facets
Repair
Prepare
Workbench
Detail
Sheet Metal
Tools
KeyShot
Clipboard
Orient
Sketch
Mode
Edit
Intersect
Create
Body
Structure
Design1*
Solid
Solid
Plane
Curves
Snapping to Grid
ANSYS
2019 R3
ACADEMIC
Use the Project to sketch tool to project the circle to the sketch
Create a circle using the Circle tool (c)
Center: the center of the projected circle
Diameter: 40 mm
Ø 40mm
Delete the projected circle
Go to 3D mode (d)
Options - Sketch
Snap to grid
Snap to angle
Create layout curves
Dimensions
Cartesian dimensions
Polar dimensions
Lock base point
Mesh Fit
Fit curves
Tolerance 10
Properties
Snapping to Grid
x=-178,0000 y=-207,0000


A:Fluid Flow (Fluent) - FFF - SpaceClaim
File
Design
Display
Assembly
Measure
Facets
Repair
Prepare
Workbench
Detail
Sheet Metal
Tools
KeyShot
Clipboard
Orient
Sketch
Mode
Edit
Intersect
Create
Body
Structure
FFF*
Solid
Solid
Surface
Select faces, curves, edges or points to blend between. Ctrl+select edges of already selected faces to ignore tangency.
ANSYS
2019 R3
ACADEMIC
Delete the Plane
Use Blend tool
Select the highlighted surfaces
Enable the Ruled segments options
Finish blending
Without the Ruled segment option the blending generates spline surfaces
Structure
Layers
Selection
Groups
Views
Options - Blend Options
General
Add
Cut
No merge
Rotational blend
Periodic blend
Ruled segments
Local guides
Clocked guides
Sheet metal blend
Normal to centerline
Show UV grid
Properties
Appearance
Color
ARGB: 255, 143, 175
Style
By Layer, By Style
Face Type
Shape
Plane
Chamfer
False
Properties
Appearance
FFF*
Select faces, curves, edges or points to blend between. Ctrl+select edges of already selected faces to ignore tangency.
Distance = 30 mm
2 Faces


A:Fluid Flow (Fluent) - Design1 - SpaceClaim
File
Design
Display
Assembly
Measure
Facets
Repair
Prepare
Workbench
Detail
Sheet Metal
Tools
KeyShot
Paste
Clipboard
Home
Spin
Plan View
Pan
Zoom
Orient
Sketch
Mode
Select
Pull
Move
Fill
Edit
Combine
Split Body
Split
Project
Intersect
Shell
Offset
Mirror
Create
Equation
Cylinder
Sphere
Body
Structure
Design1*
Solid
Solid
Solid
Pull 1 face
50mm
P
Structure
Layers
Selection
Groups
Views
Options - Pull
General
Add
Cut
No merge
Properties
Appearance
Color
ARGB: 255, 143, 175
Style
By Layer, By Style
Face Type
Shape
Plane
Chamfer
False
Properties
Appearance
Design1*
Pull 1 face
1 Face
ANSYS
2019 R3
ACADEMIC
Copy and paste the last surface
Use Pull tool (p)
Select No merge option
Distance: 50 mm
Without the copy and paste of the surface the pull tool continues the cone shape
Without the copy and paste of the surface and the No merge option the pull tool merges the solids
Check if you have two solids in the Structure tree


A:Fluid Flow (Fluent) - Design1 - SpaceClaim
File
Design
Display
Assembly
Measure
Facets
Repair
Prepare
Workbench
Detail
Sheet Metal
Tools
KeyShot
Paste
Clipboard
Home
Spin
Plan View
Pan
Zoom
Orient
Sketch
Mode
Select
Pull
Move
Fill
Edit
Combine
Split Body
Split
Project
Intersect
Shell
Offset
Mirror
Create
Equation
Cylinder
Sphere
Body
Structure
Design1*
Solid
Solid
Structure
Layers
Selection
Groups
Views
Options - Combine
General
Make bodies
Merge when done
Keep cutter
Make all regions
Make curves
Imprint as edges
Extend intersections
Properties
Properties
Appearance
Click an object to merge the target with.
ANSYS
2019 R3
ACADEMIC
Use Combine tool (or press i)
Select the one of the highlighted solids
Choose Select bodies to merge option (or press ctrl)
and select the other highlighted solid
Check if the selected solids are combined
Design1*
Click an object to merge the target with.

A:Fluid Flow (Fluent) - Design1 - SpaceClaim

File | Design | Display | Assembly | Measure | Facets | Repair | Prepare | Workbench | Detail | Sheet Metal | Tools | KeyShot

Clipboard | Orient | Sketch | Mode | Edit | Intersect | Create | Body

Structure
- Design1*
  - Solid
  - Solid
  - Plane

Structure | Layers | Selection | Groups | Views

Options - Selection

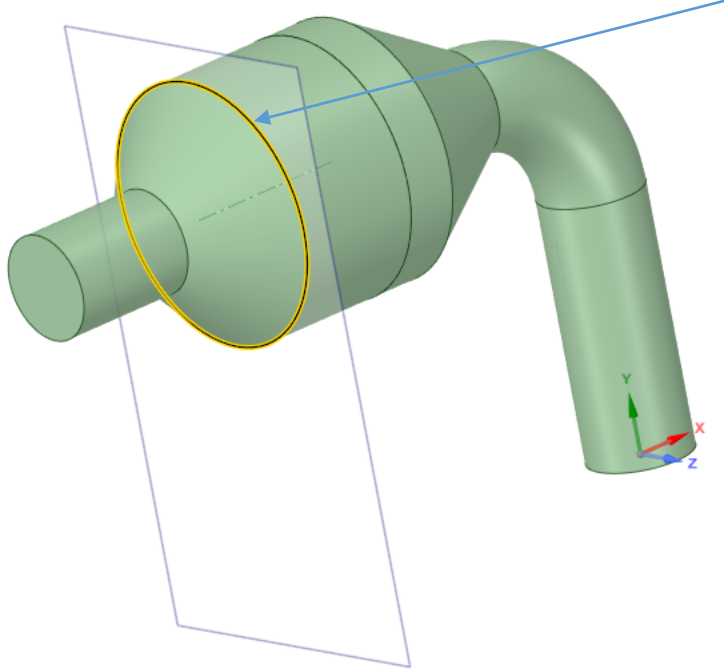
Properties

Properties | Appearance

Click an object. Double-click to select an edge loop. Triple-click to select a solid.

ANSYS 2019 R3 ACADEMIC

Select the highlighted edge and create a plane



Design1*

Click an object. Double-click to select an edge loop. Triple-click to select a solid.

Radius = 50 mm

1 Edge

A:Fluid Flow (Fluent) - Design1 - SpaceClaim

File | Design | Display | Assembly | Measure | Facets | Repair | Prepare | Workbench | Detail | Sheet Metal | Tools | KeyShot

Clipboard | Orient | Sketch | Mode | Edit | Intersect | Create | Body

Split Body | Split | Project | Shell | Offset | Mirror | Cylinder | Sphere

Structure
- Design1*
  - Solid
  - Solid
  - Solid1
  - Plane

Structure | Layers | Selection | Groups | Views

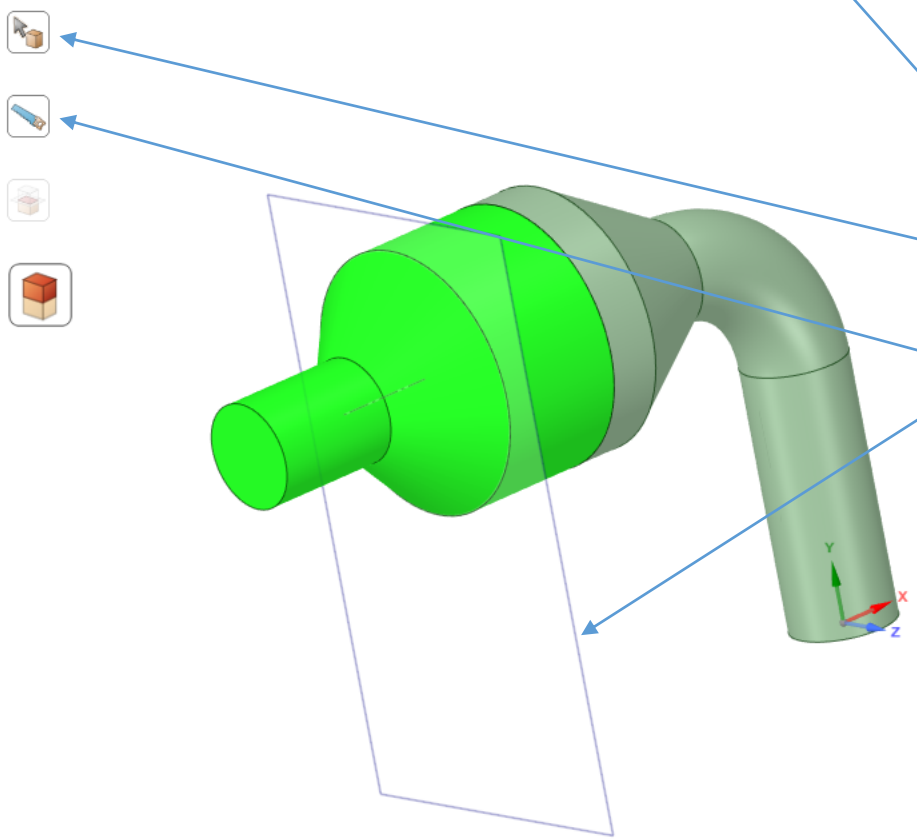
Options - Split Body
General
- [ ] Merge when done
- [x] Extend faces
- [ ] Local slicing
  - [ ] Create split surfaces

Properties

Properties | Appearance

Click regions to remove. Press Ctrl to select an additional cutter. Use Merge When Done option to merge remaining regions.

ANSYS 2019 R3 ACADEMIC



Select Split body tool

Select the highlighted body

Select the created plane to cutter

Delete the plane

Design1*

Click regions to remove. Press Ctrl to select an additional cutter. Use Merge When Done option to merge remaining regions.

A:Fluid Flow (Fluent) - Design1 - SpaceClaim

File Design Display Assembly Measure Facets Repair Prepare Workbench Detail Sheet Metal Tools KeyShot

Paste Clipboard | Home Plan View Orient | Color Layer0 Thick Style | Graphics | Camera | New Window Split Switch Windows Window | Show Sketch Grid Fade Scene Under Grid Clip Scene Above Grid Grid | Show Display

Structure

- Design1*
  - Solid
  - Solid
  - Solid1

Structure Layers Selection Groups Views

Options - Selection

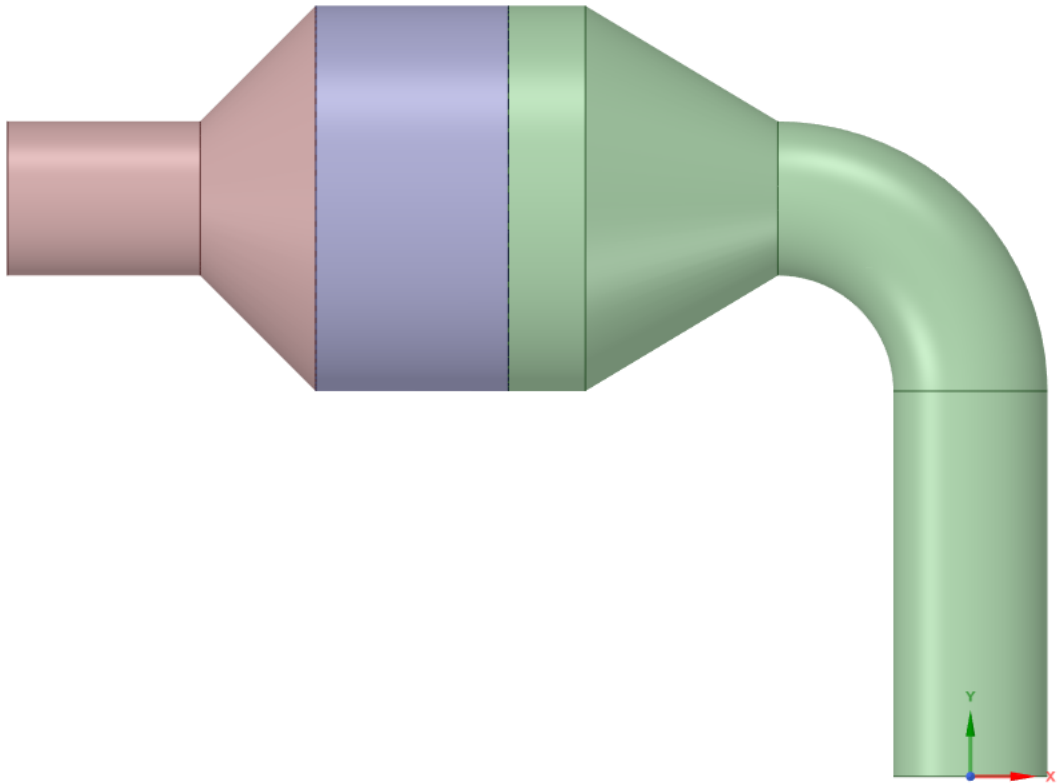
Properties

Properties Appearance

Click an object. Double-click to select an edge loop. Triple-click to select a solid.

ANSYS 2019 R3 ACADEMIC

Check if you have these three solids



Y
Z
X

Design1*

Click an object. Double-click to select an edge loop. Triple-click to select a solid.

A:Fluid Flow (Fluent) - Design1 - SpaceClaim

File | Design | Display | Assembly | Measure | Facets | Repair | Prepare | Workbench | Detail | Sheet Metal | Tools | KeyShot

Identify Objects | Icepak Simplify | Opening | Enclosure | Fan | Show | Grille — Icepak
Overlap Bodies | Split Edges | Extra Edges | Simplify | Small Faces | Zoom to Fit | Clip Volume — Sharing
Share | Unshare | Force Share
Faces | Edges | Vertices — Shared Display
Beams | Edges — Free Display
Update Parameters | 2019 R3 | AIM 2019 R3 | Mechanical 2019 R3 | Fluent 2019 R3 — ANSYS Transfer

Structure

- Design1*
  - Solid
  - Solid
  - Solid1

Structure | Layers | Selection | Groups | Views

Options - Selection

Properties

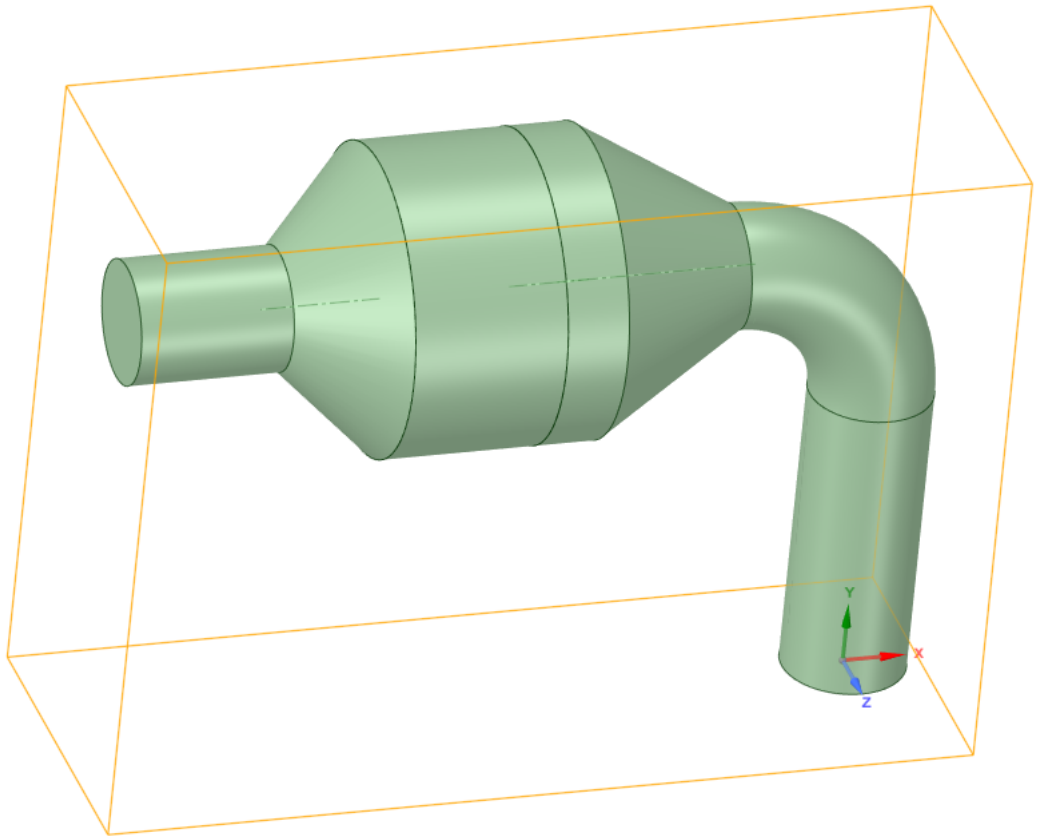
Analysis	
Share Topology	Share
Document	
Display Name	Design1
Document Path	
Locked	False
Use File Name	True
File	
Category	
Content Status	
Content Type	
Created	2020. 02. 20. 15:43

Properties | Appearance

Click an object. Double-click to select an edge loop. Triple-click to select a solid.

ANSYS 2019 R3 ACADEMIC

Select the top of the Structure tree

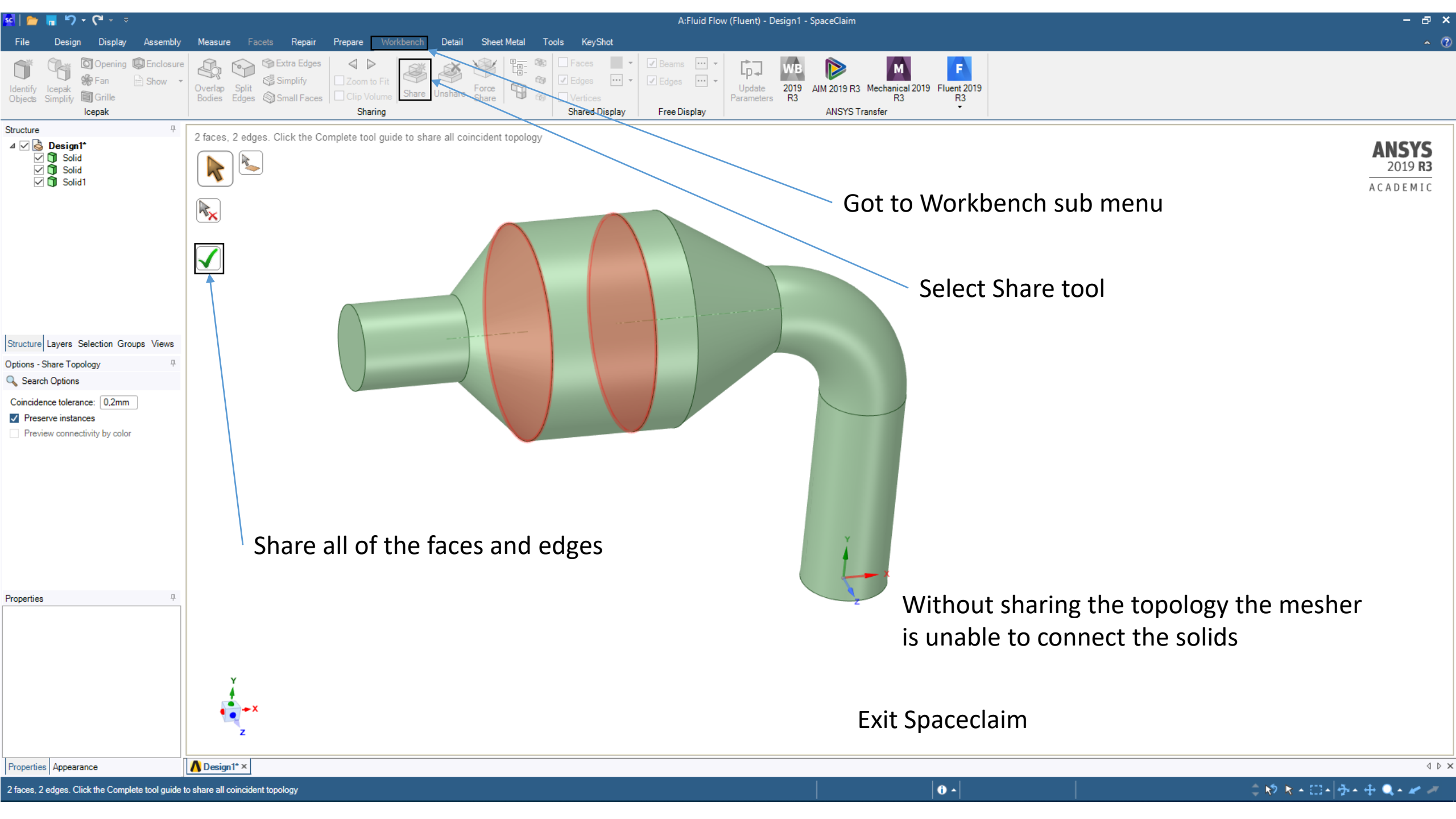


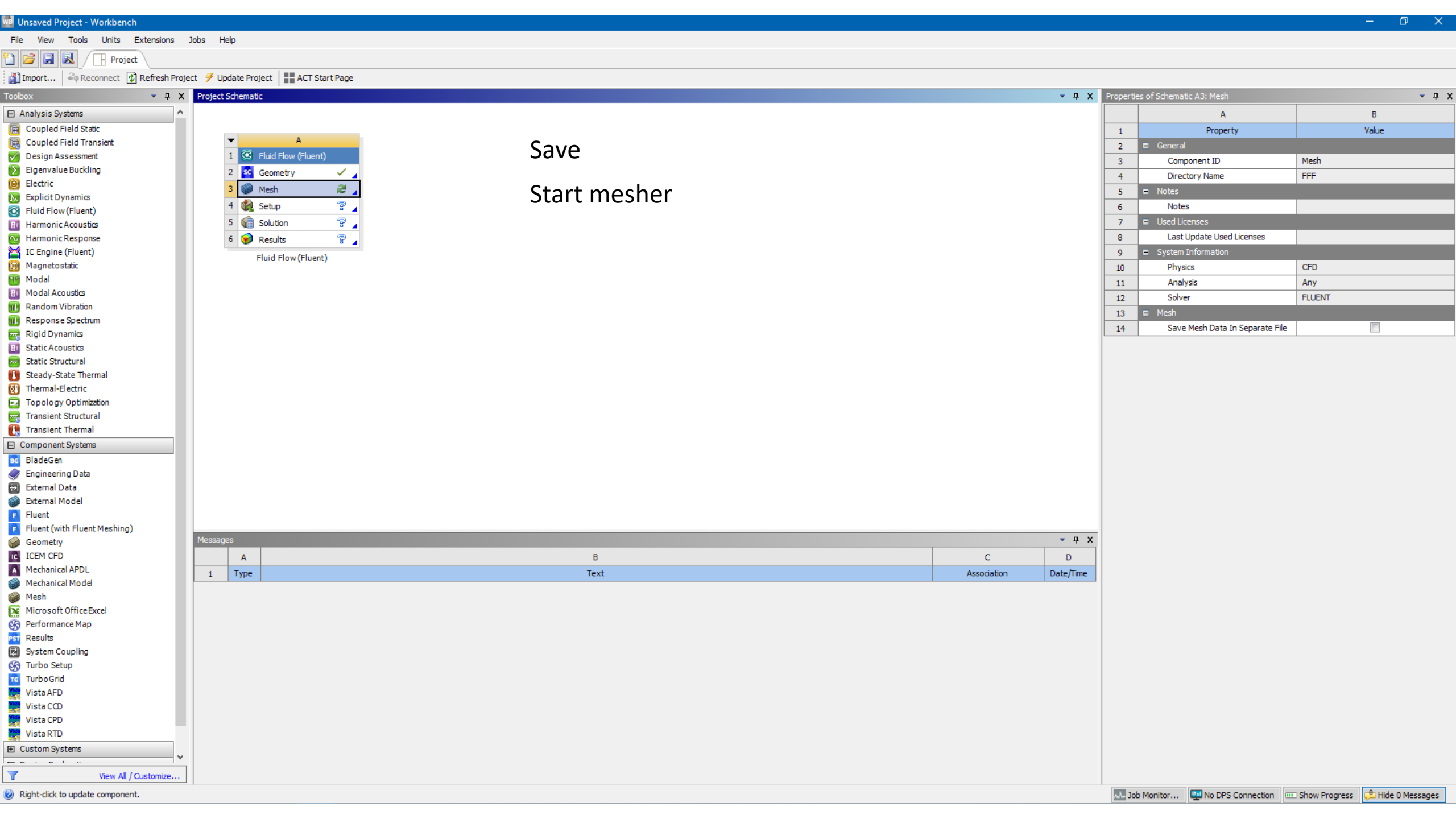
Set Analysis/Share topology to Share

Design1*

Click an object. Double-click to select an edge loop. Triple-click to select a solid.

1 Part


A:Fluid Flow (Fluent) - Design1 - SpaceClaim
File Design Display Assembly Measure Facets Repair Prepare Workbench Detail Sheet Metal Tools KeyShot
Identify Objects
Icepak Simplify
Opening
Enclosure
Fan
Show
Grille
Icepak
Overlap Bodies
Split Edges
Extra Edges
Simplify
Small Faces
Zoom to Fit
Clip Volume
Share
Unshare
Force Share
Sharing
Faces
Edges
Vertices
Shared Display
Beams
Edges
Free Display
Update Parameters
2019 R3
AIM 2019 R3
Mechanical 2019 R3
Fluent 2019 R3
ANSYS Transfer
Structure
Design1*
Solid
Solid
Solid1
2 faces, 2 edges. Click the Complete tool guide to share all coincident topology
ANSYS 2019 R3 ACADEMIC
Got to Workbench sub menu
Select Share tool
Share all of the faces and edges
Without sharing the topology the mesher is unable to connect the solids
Exit Spaceclaim
Structure Layers Selection Groups Views
Options - Share Topology
Search Options
Coincidence tolerance: 0,2mm
Preserve instances
Preview connectivity by color
Properties
Properties Appearance
Design1*
2 faces, 2 edges. Click the Complete tool guide to share all coincident topology


Unsaved Project - Workbench
File View Tools Units Extensions Jobs Help
Project
Import... Reconnect Refresh Project Update Project ACT Start Page
Toolbox
Analysis Systems
Coupled Field Static
Coupled Field Transient
Design Assessment
Eigenvalue Buckling
Electric
Explicit Dynamics
Fluid Flow (Fluent)
Harmonic Acoustics
Harmonic Response
IC Engine (Fluent)
Magnetostatic
Modal
Modal Acoustics
Random Vibration
Response Spectrum
Rigid Dynamics
Static Acoustics
Static Structural
Steady-State Thermal
Thermal-Electric
Topology Optimization
Transient Structural
Transient Thermal
Component Systems
BladeGen
Engineering Data
External Data
External Model
Fluent
Fluent (with Fluent Meshing)
Geometry
ICEM CFD
Mechanical APDL
Mechanical Model
Mesh
Microsoft Office Excel
Performance Map
Results
System Coupling
Turbo Setup
TurboGrid
Vista AFD
Vista CCD
Vista CPD
Vista RTD
Custom Systems
View All / Customize...
Project Schematic
A
1 Fluid Flow (Fluent)
2 Geometry
3 Mesh
4 Setup
5 Solution
6 Results
Fluid Flow (Fluent)
Save
Start mesher
Messages
A B C D
1 Type Text Association Date/Time
Properties of Schematic A3: Mesh
A B
1 Property Value
2 General
3 Component ID Mesh
4 Directory Name FFF
5 Notes
6 Notes
7 Used Licenses
8 Last Update Used Licenses
9 System Information
10 Physics CFD
11 Analysis Any
12 Solver FLUENT
13 Mesh
14 Save Mesh Data In Separate File
Right-click to update component.
Job Monitor... No DPS Connection Show Progress Hide 0 Messages

A : Fluid Flow (Fluent) - Meshing [ANSYS Academic Teaching Mechanical and CFD]

Context

File | Home | Named Selection | Display | Selection | Automation

Quick Launch

Duplicate | Cut | Copy | Paste | Delete | Find | Tree | Outline
Generate | Mesh
Named Selection | Coordinate System | Comment | Images | Section Plane | Annotation | Insert
Units | Worksheet | Keyframe Animation | Tags | Show Errors | Manage Views | Selection Information | Unit Converter | Print Preview | Report Preview | Key Assignments | Tools
Full Screen | Manage | User Defined | Reset Layout | Layout

Outline

Name | Search Outline

- Project*
  - Model (A3)
    - Geometry
    - Materials
    - Coordinate Systems
    - Connections
    - Mesh
    - Named Selections
      - velocity_inlet
      - pressure_outlet

Details of "Multiple Selection"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Send to Solver	Yes
Protected	Program Controlled
Visible	Yes
Program Controlled Inflation	Exclude
Statistics	
Type	Manual
Total Selection	1 Face
Surface Area	
Suppressed	0
Used by Mesh Worksheet	No

Select | Mode | Clipboard | [Empty] | Extend | Select By | Convert

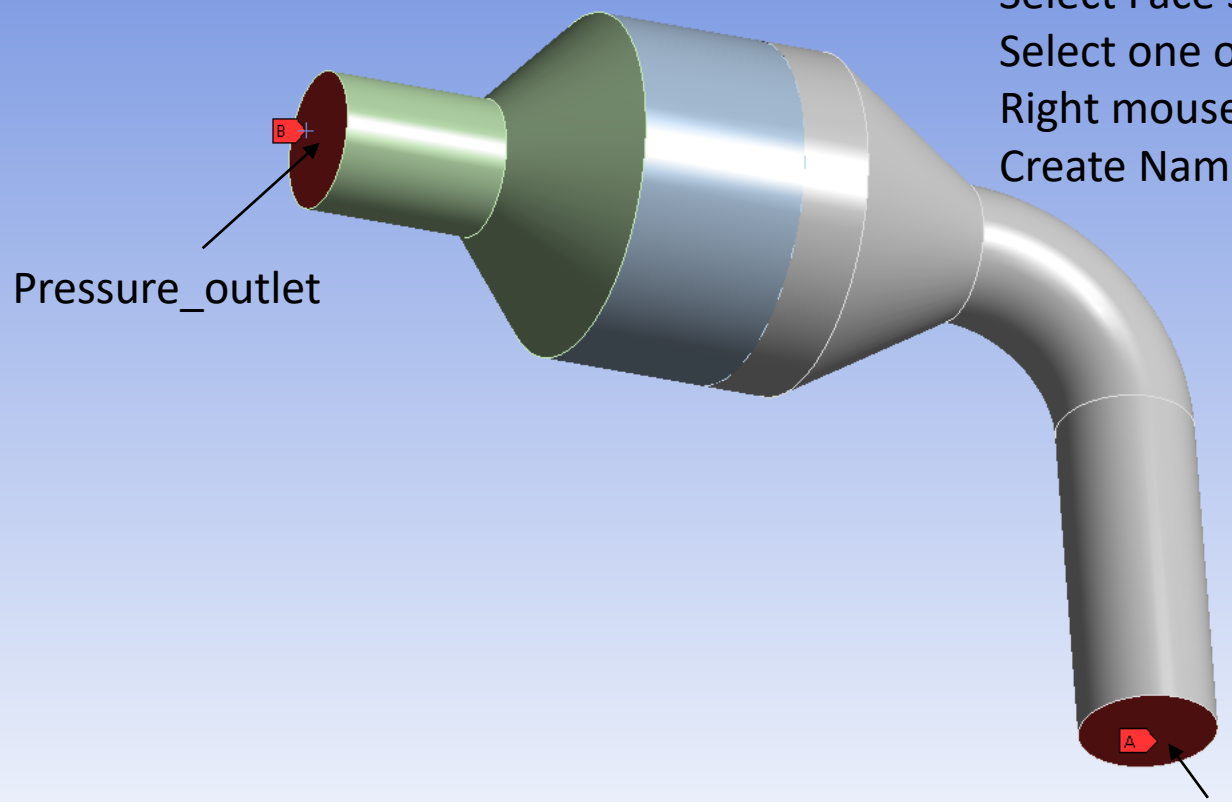
pressure_outlet
2020. 02. 20. 16:08

A velocity_inlet
B pressure_outlet

ANSYS 2019 R3 ACADEMIC

Create Named Selections

Select Mesh in the Outline
Select Face Selection
Select one of the highlighted faces
Right mouse button
Create Named Selection (or press n)



Pressure_outlet

velocity_inlet

0,000 | 0,050 | 0,100 (m)
0,025 | 0,075

Y
Z
X

Ready | Message Pane | No Selection | Metric (m, kg, N, s, V, A) | Degrees | rad/s | Celsius

A : Fluid Flow (Fluent) - Meshing [ANSYS Academic Teaching Mechanical and CFD]

Context

File | Home | Named Selection | Display | Selection | Automation

Quick Launch

Duplicate | Cut | Copy | Paste | Delete | Find | Tree | Outline
Generate | Mesh
Named Selection | Coordinate System | Comment | Images | Section Plane | Annotation | Insert
Units | Worksheet | Keyframe Animation | Tags | Show Errors | Manage Views | Selection Information | Unit Converter | Print Preview | Report Preview | Key Assignments | Tools
Full Screen | Manage | User Defined | Reset Layout | Layout

Outline

Name | Search Outline

- Project*
  - Model (A3)
    - Geometry
    - Materials
    - Coordinate Systems
    - Connections
    - Mesh
    - Named Selections
      - velocity_inlet
      - pressure_outlet
      - fluid1
      - fluid2
      - fluid3

Details of "Multiple Selection"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Send to Solver	Yes
Protected	Program Controlled
Visible	Yes
Program Controlled Inflation	Exclude
Statistics	
Type	Manual
Total Selection	1 Body
Suppressed	0
Used by Mesh Worksheet	No

Select | Mode | Clipboard | [Empty] | Extend | Select By | Convert

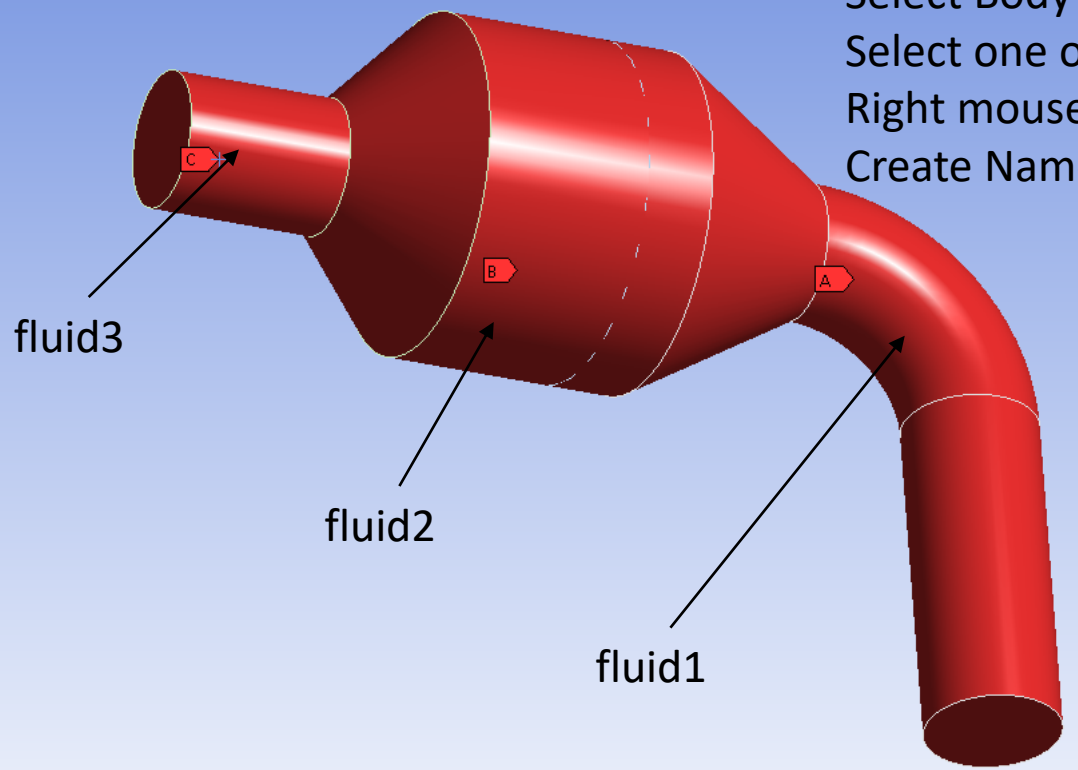
fluid3
2020. 02. 20. 16:10

A fluid1
B fluid2
C fluid3

ANSYS 2019 R3 ACADEMIC

Create Named Selections

Select Mesh in the Outline
Select Body Selection
Select one of the highlighted bodies
Right mouse button
Create Named Selection (or press n)



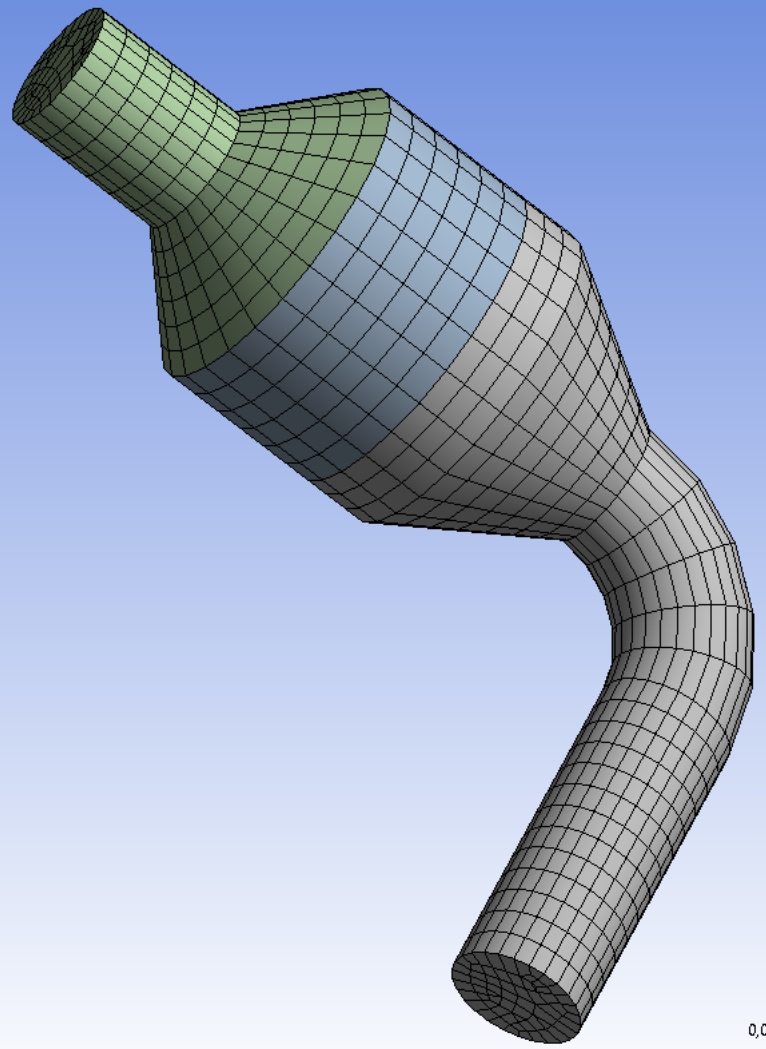
0,000 | 0,050 | 0,100 (m)
0,025 | 0,075

Y
Z
X

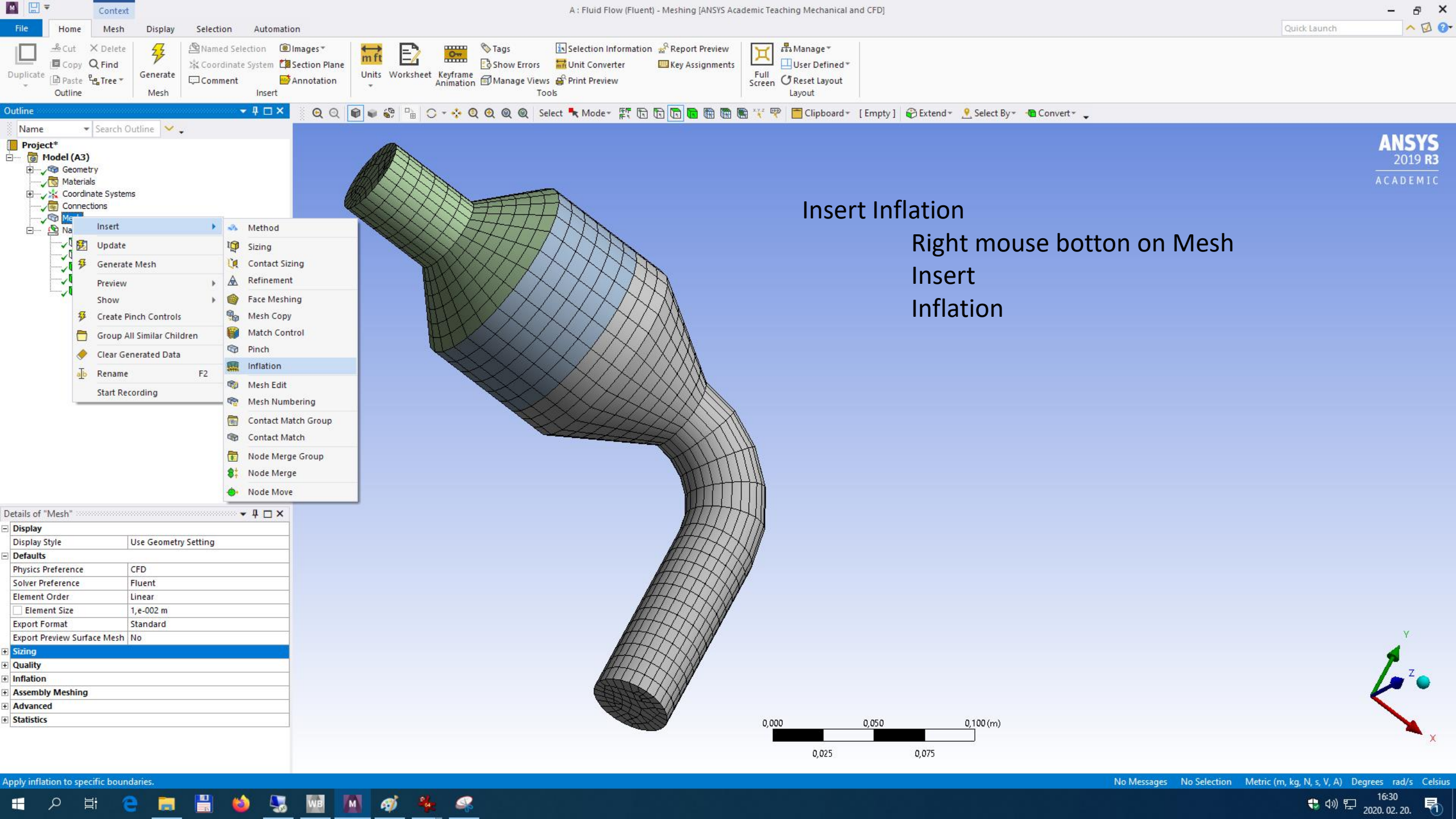
Ready | Message Pane | No Selection | Metric (m, kg, N, s, V, A) | Degrees | rad/s | Celsius

A : Fluid Flow (Fluent) - Meshing [ANSYS Academic Teaching Mechanical and CFD]

Set Element size of the Mesh to 1 cm
Generate Mesh



ANSYS 2019 R3 ACADEMIC


A : Fluid Flow (Fluent) - Meshing [ANSYS Academic Teaching Mechanical and CFD]
Insert Inflation
Right mouse botton on Mesh
Insert
Inflation
ANSYS 2019 R3 ACADEMIC
Project*
Model (A3)
Geometry
Materials
Coordinate Systems
Connections
Insert
Update
Generate Mesh
Preview
Show
Create Pinch Controls
Group All Similar Children
Clear Generated Data
Rename F2
Start Recording
Method
Sizing
Contact Sizing
Refinement
Face Meshing
Mesh Copy
Match Control
Pinch
Inflation
Mesh Edit
Mesh Numbering
Contact Match Group
Contact Match
Node Merge Group
Node Merge
Node Move
Details of "Mesh"
Display
Display Style Use Geometry Setting
Defaults
Physics Preference CFD
Solver Preference Fluent
Element Order Linear
Element Size 1,e-002 m
Export Format Standard
Export Preview Surface Mesh No
Sizing
Quality
Inflation
Assembly Meshing
Advanced
Statistics
0,000 0,050 0,100 (m)
0,025 0,075
Apply inflation to specific boundaries.
No Messages No Selection Metric (m, kg, N, s, V, A) Degrees rad/s Celsius

A : Fluid Flow (Fluent) - Meshing [ANSYS Academic Teaching Mechanical and CFD]

Context

File | Home | Mesh | Display | Selection | Automation

Quick Launch

Duplicate | Cut | Copy | Paste | Delete | Find | Tree — Outline
Generate — Mesh
Named Selection | Coordinate System | Comment | Images | Section Plane | Annotation — Insert
Units | Worksheet | Keyframe Animation | Tags | Show Errors | Manage Views | Selection Information | Unit Converter | Print Preview | Report Preview | Key Assignments — Tools
Full Screen | Manage | User Defined | Reset Layout — Layout

Outline

Name | Search Outline

- Project*
  - Model (A3)
    - Geometry
    - Materials
    - Coordinate Systems
    - Connections
    - Mesh
      - Inflation
    - Named Selections
      - velocity_inlet
      - pressure_outlet
      - fluid1
      - fluid2
      - fluid3

Details of "Inflation" - Inflation

Scope	
Scoping Method	Geometry Selection
Geometry	Apply / Cancel
Definition	
Suppressed	No
Boundary Scoping Method	Geometry Selection
Boundary	No Selection
Inflation Option	Smooth Transition
Transition Ratio	Default (0,272)
Maximum Layers	5
Growth Rate	1,2
Inflation Algorithm	Pre

Select | Mode | Clipboard | [Empty] | Extend | Select By | Convert

Inflation
2020. 02. 20. 16:31

ANSYS 2019 R3 ACADEMIC

Inflation settings

Select all bodies to the geometry
(Body selection)
Apply

0,000 0,050 0,100 (m)
0,025 0,075

Y Z X

Ready | No Messages | 3 Bodies Selected: Volume = 1,1637e-003 m³ | Metric (m, kg, N, s, V, A) | Degrees | rad/s | Celsius

A : Fluid Flow (Fluent) - Meshing [ANSYS Academic Teaching Mechanical and CFD]

Context

File | Home | Mesh | Display | Selection | Automation

Quick Launch

Duplicate | Cut | Copy | Paste | Delete | Find | Tree | Outline

Generate | Mesh

Named Selection | Coordinate System | Comment | Images | Section Plane | Annotation | Insert

Units | Worksheet | Keyframe Animation | Tags | Show Errors | Manage Views | Selection Information | Unit Converter | Print Preview | Report Preview | Key Assignments | Tools

Full Screen | Manage | User Defined | Reset Layout | Layout

Outline

Name | Search Outline

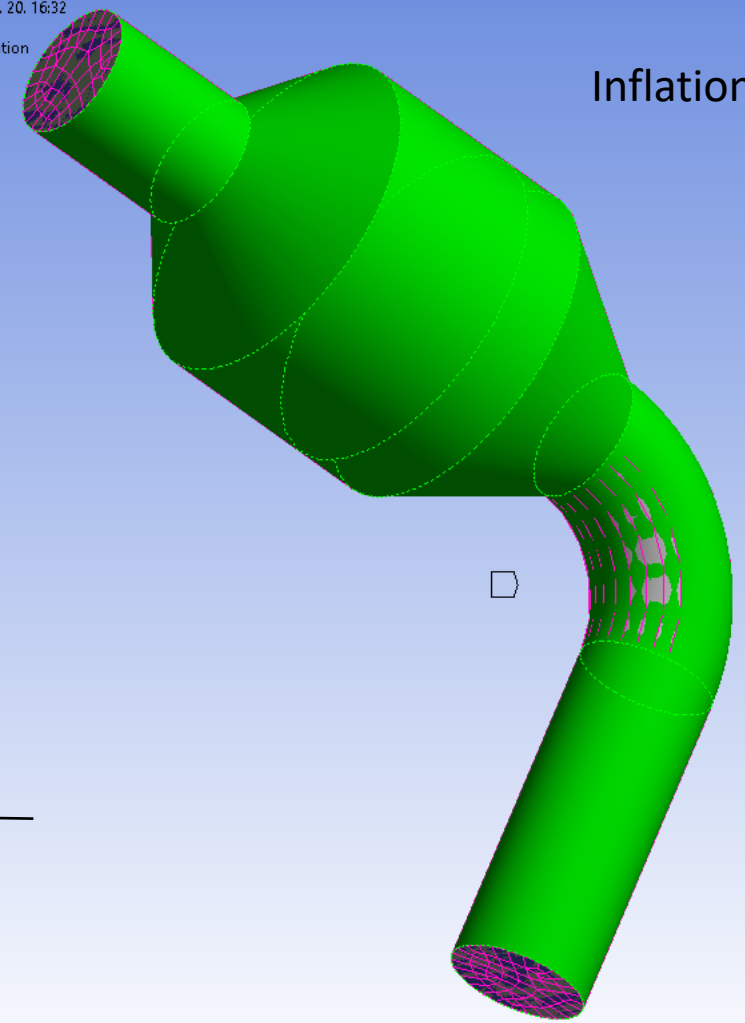
- Project*
  - Model (A3)
    - Geometry
    - Materials
    - Coordinate Systems
    - Connections
    - Mesh
      - Inflation
    - Named Selections
      - velocity_inlet
      - pressure_outlet
      - fluid1
      - fluid2
      - fluid3

Details of "Inflation" - Inflation

Scope		
Scoping Method	Geometry Selection	
Geometry	3 Bodies	
Definition		
Suppressed	No	
Boundary Scoping Method	Geometry Selection	
Boundary	Apply	Cancel
Inflation Option	Total Thickness	
Number of Layers	4	
Growth Rate	1,5	
Maximum Thickness	4, mm	
Inflation Algorithm	Pre	

Select | Mode | Clipboard | [Empty] | Extend | Select By | Convert

Inflation
2020. 02. 20. 16:32
Inflation



ANSYS 2019 R3 ACADEMIC

Inflation settings

Select the walls of the exhaust system to the Boundary (Face Selection)
Apply

Inflation option: Total Thickness
Number of layers: 4
Growth Rate: 1,5
Maximum Thickness: 4 mm

Generate Mesh

0,000 | 0,050 | 0,100 (m)
0,025 | 0,075

Y
X

Make a copy of the selected object(s) and automatically insert it into the current Outline location.

No Messages | 7 Faces Selected: Area = 7,2812e-002 m² | Metric (m, kg, N, s, V, A) | Degrees | rad/s | Celsius

A : Fluid Flow (Fluent) - Meshing [ANSYS Academic Teaching Mechanical and CFD]

Context

File | Home | Mesh | Display | Selection | Automation

Quick Launch

Duplicate | Cut | Copy | Paste | Delete | Find | Tree — Outline
Generate — Mesh
Named Selection | Coordinate System | Comment | Images | Section Plane | Annotation — Insert
Units | Worksheet | Keyframe Animation | Tags | Show Errors | Manage Views | Selection Information | Unit Converter | Print Preview | Report Preview | Key Assignments — Tools
Full Screen | Manage | User Defined | Reset Layout — Layout

Outline

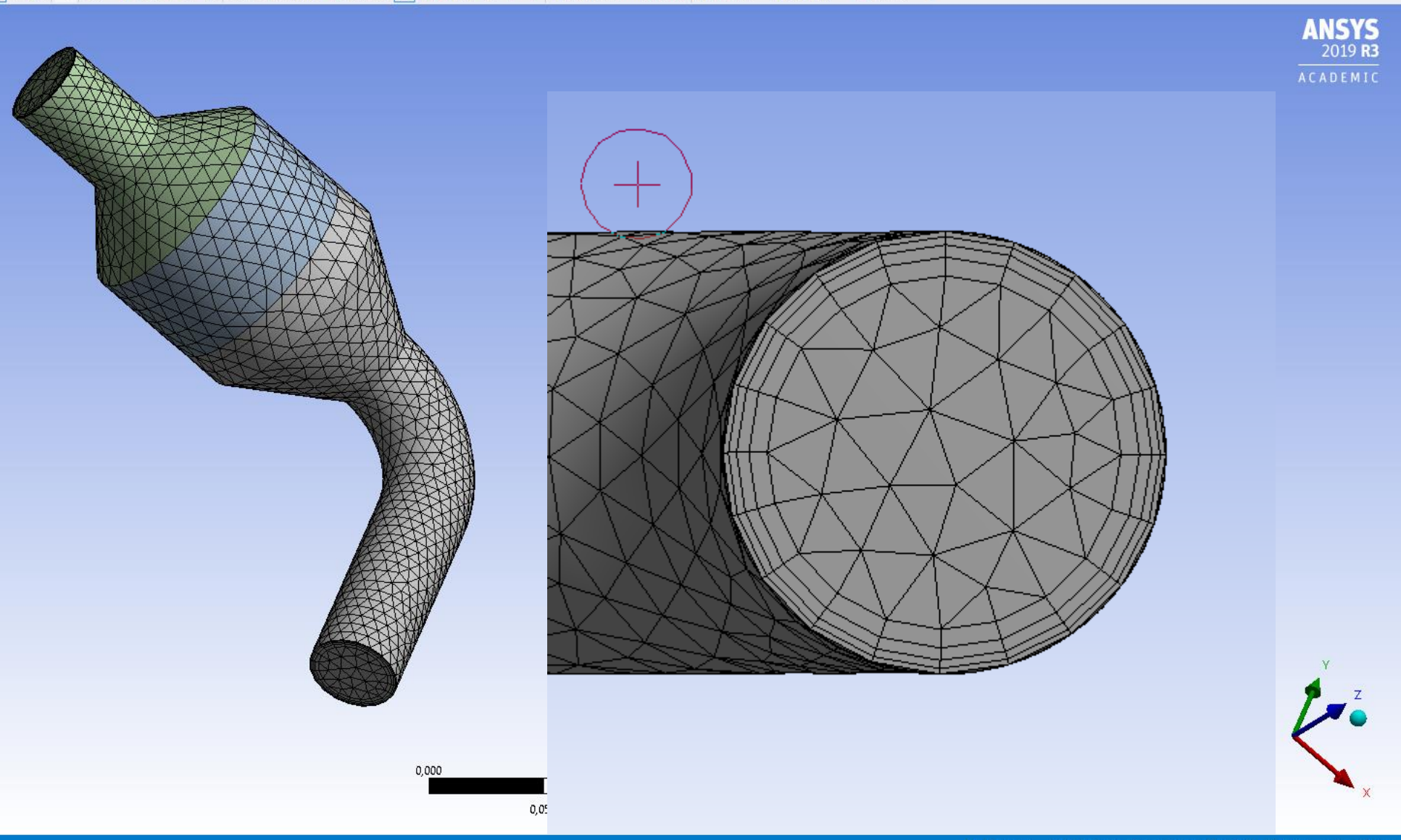
Name | Search Outline

- Project*
  - Model (A3)
    - Geometry
    - Materials
    - Coordinate Systems
    - Connections
    - Mesh
      - Inflation
    - Named Selections
      - velocity_inlet
      - pressure_outlet
      - fluid1
      - fluid2
      - fluid3

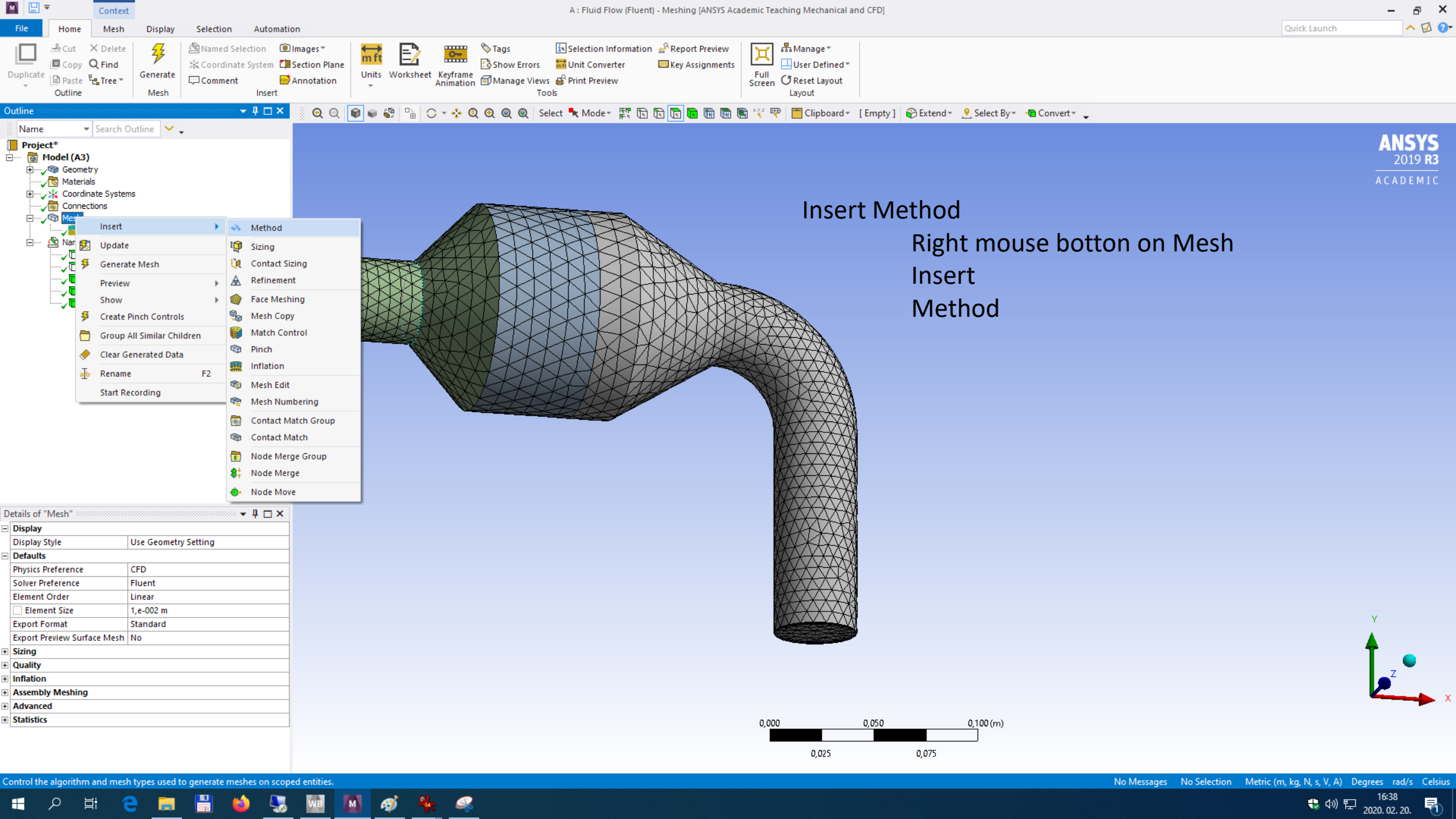
Details of "Mesh"

Display	
Display Style	Use Geometry Setting
Defaults	
Physics Preference	CFD
Solver Preference	Fluent
Element Order	Linear
Element Size	1,e-002 m
Export Format	Standard
Export Preview Surface Mesh	No
Sizing	
Quality	
Inflation	
Assembly Meshing	
Advanced	
Statistics	

Select | Mode | Clipboard | [Empty] | Extend | Select By | Convert



Ready | No Messages | No Selection | Metric (m, kg, N, s, V, A) | Degrees | rad/s | Celsius


A : Fluid Flow (Fluent) - Meshing [ANSYS Academic Teaching Mechanical and CFD]
File
Home
Mesh
Display
Selection
Automation
Context
Quick Launch
Duplicate
Cut
Copy
Paste
Delete
Find
Tree
Outline
Generate
Mesh
Named Selection
Coordinate System
Comment
Images
Section Plane
Annotation
Insert
Units
Worksheet
Keyframe Animation
Tags
Show Errors
Manage Views
Selection Information
Unit Converter
Print Preview
Report Preview
Key Assignments
Tools
Full Screen
Manage
User Defined
Reset Layout
Layout
Outline
Name
Search Outline
Project*
Model (A3)
Geometry
Materials
Coordinate Systems
Connections
Insert
Update
Generate Mesh
Preview
Show
Create Pinch Controls
Group All Similar Children
Clear Generated Data
Rename
F2
Start Recording
Method
Sizing
Contact Sizing
Refinement
Face Meshing
Mesh Copy
Match Control
Pinch
Inflation
Mesh Edit
Mesh Numbering
Contact Match Group
Contact Match
Node Merge Group
Node Merge
Node Move
Select
Mode
Clipboard
[Empty]
Extend
Select By
Convert
ANSYS
2019 R3
ACADEMIC
Insert Method
Right mouse botton on Mesh
Insert
Method
Details of "Mesh"
Display
Display Style
Use Geometry Setting
Defaults
Physics Preference
CFD
Solver Preference
Fluent
Element Order
Linear
Element Size
1,e-002 m
Export Format
Standard
Export Preview Surface Mesh
No
Sizing
Quality
Inflation
Assembly Meshing
Advanced
Statistics
0,000
0,050
0,100 (m)
0,025
0,075
Y
Z
X
Control the algorithm and mesh types used to generate meshes on scoped entities.
No Messages
No Selection
Metric (m, kg, N, s, V, A)
Degrees
rad/s
Celsius
16:38
2020. 02. 20.

A : Fluid Flow (Fluent) - Meshing [ANSYS Academic Teaching Mechanical and CFD]

Context

File | Home | Mesh | Display | Selection | Automation

Quick Launch

Duplicate | Cut | Copy | Paste | Delete | Find | Tree — Outline
Generate — Mesh
Named Selection | Coordinate System | Comment | Images | Section Plane | Annotation — Insert
Units | Worksheet | Keyframe Animation | Tags | Show Errors | Manage Views | Selection Information | Unit Converter | Print Preview | Report Preview | Key Assignments — Tools
Full Screen | Manage | User Defined | Reset Layout — Layout

Outline

Name | Search Outline

- Project*
  - Model (A3)
    - Geometry
    - Materials
    - Coordinate Systems
    - Connections
    - Mesh
      - Inflation
      - Automatic Method
    - Named Selections
      - velocity_inlet
      - pressure_outlet
      - fluid1
      - fluid2
      - fluid3

Details of "Automatic Method" - Method

Scope	
Scoping Method	Geometry Selection
Geometry	Apply / Cancel
Definition	
Suppressed	No
Method	Automatic
Element Order	Use Global Setting

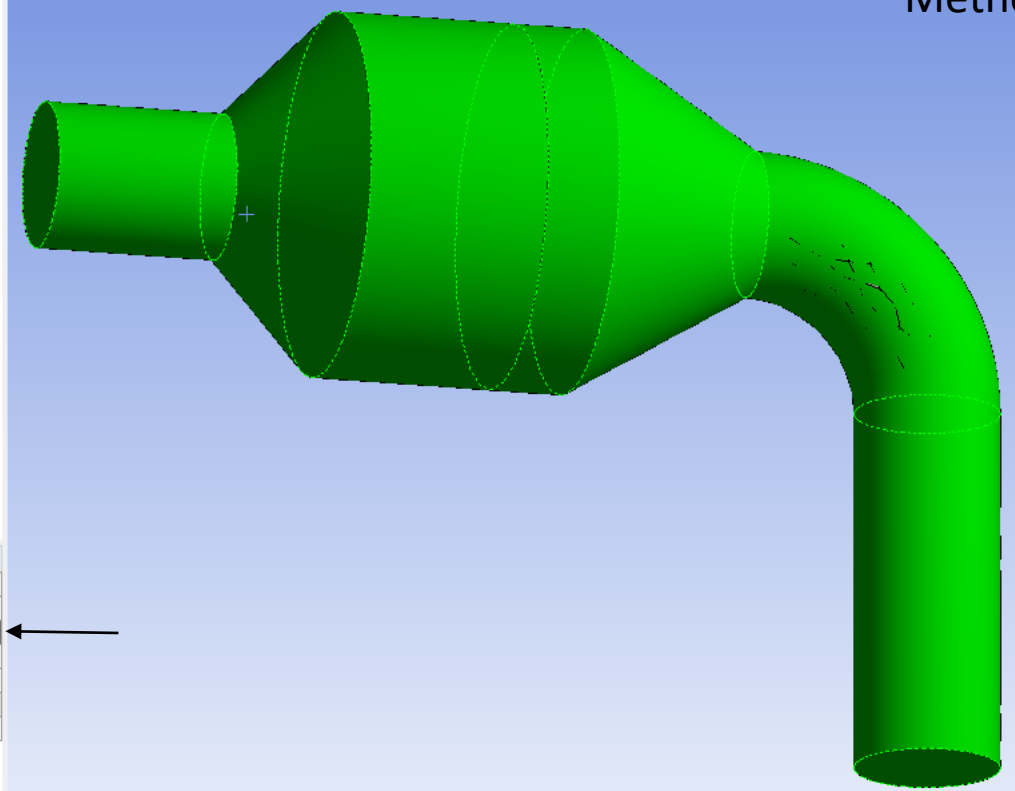
Select | Mode | Clipboard | [Empty] | Extend | Select By | Convert

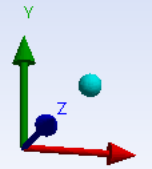
Automatic Method
2020. 02. 20. 16:38

ANSYS 2019 R3 ACADEMIC

Method settings

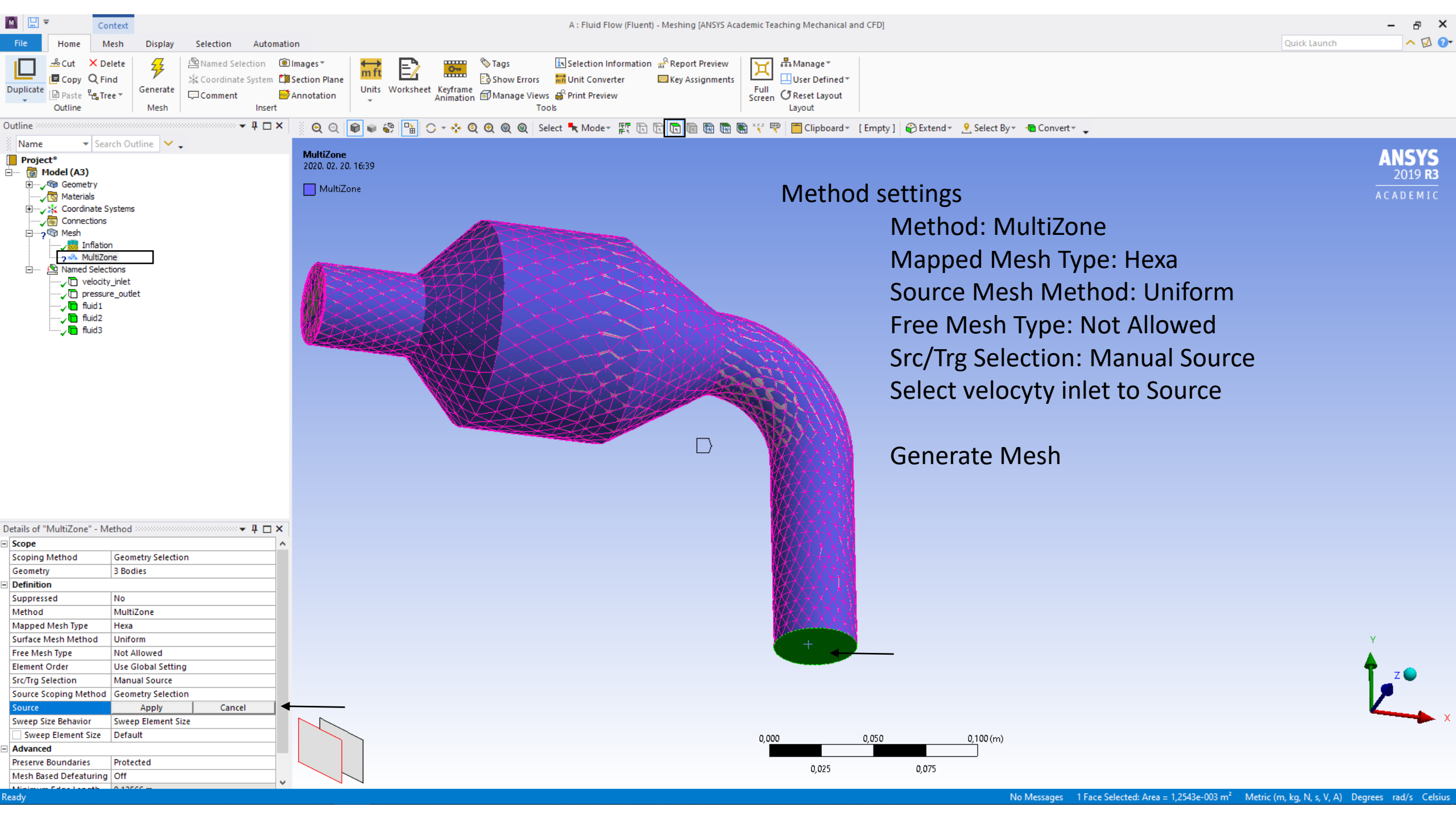
Select all bodies to the geometry
(Body selection)
Apply





0,000 | 0,050 | 0,100 (m)
0,025 | 0,075

Ready | No Messages | 3 Bodies Selected: Volume = 1,1637e-003 m³ | Metric (m, kg, N, s, V, A) | Degrees | rad/s | Celsius


A : Fluid Flow (Fluent) - Meshing [ANSYS Academic Teaching Mechanical and CFD]
Context
File
Home
Mesh
Display
Selection
Automation
Quick Launch
Duplicate
Cut
Copy
Paste
Delete
Find
Tree
Outline
Generate
Mesh
Named Selection
Coordinate System
Comment
Images
Section Plane
Annotation
Insert
Units
Worksheet
Keyframe Animation
Tags
Show Errors
Manage Views
Selection Information
Unit Converter
Print Preview
Report Preview
Key Assignments
Tools
Full Screen
Manage
User Defined
Reset Layout
Layout
Outline
Name
Search Outline
Project*
Model (A3)
Geometry
Materials
Coordinate Systems
Connections
Mesh
Inflation
MultiZone
Named Selections
velocity_inlet
pressure_outlet
fluid1
fluid2
fluid3
Select
Mode
Clipboard
[Empty]
Extend
Select By
Convert
MultiZone
2020. 02. 20. 16:39
MultiZone
ANSYS
2019 R3
ACADEMIC
Method settings
Method: MultiZone
Mapped Mesh Type: Hexa
Source Mesh Method: Uniform
Free Mesh Type: Not Allowed
Src/Trg Selection: Manual Source
Select velocyty inlet to Source
Generate Mesh
Details of "MultiZone" - Method
Scope
Scoping Method
Geometry Selection
Geometry
3 Bodies
Definition
Suppressed
No
Method
MultiZone
Mapped Mesh Type
Hexa
Surface Mesh Method
Uniform
Free Mesh Type
Not Allowed
Element Order
Use Global Setting
Src/Trg Selection
Manual Source
Source Scoping Method
Geometry Selection
Source
Apply
Cancel
Sweep Size Behavior
Sweep Element Size
Sweep Element Size
Default
Advanced
Preserve Boundaries
Protected
Mesh Based Defeaturing
Off
0,000
0,050
0,100 (m)
0,025
0,075
Y
Z
X
Ready
No Messages
1 Face Selected: Area = 1,2543e-003 m²
Metric (m, kg, N, s, V, A)
Degrees
rad/s
Celsius

A : Fluid Flow (Fluent) - Meshing [ANSYS Academic Teaching Mechanical and CFD]

Context

File | Home | Mesh | Display | Selection | Automation

Quick Launch

Duplicate | Cut | Copy | Paste | Delete | Find | Tree — Outline
Generate — Mesh
Named Selection | Coordinate System | Comment | Images | Section Plane | Annotation — Insert
Units | Worksheet | Keyframe Animation | Tags | Show Errors | Manage Views | Selection Information | Unit Converter | Print Preview | Report Preview | Key Assignments — Tools
Full Screen | Manage | User Defined | Reset Layout — Layout

Outline

Name | Search Outline

- Project*
  - Model (A3)
    - Geometry
    - Materials
    - Coordinate Systems
    - Connections
    - Mesh
      - Inflation
      - MultiZone
    - Named Selections
      - velocity_inlet
      - pressure_outlet
      - fluid1
      - fluid2
      - fluid3

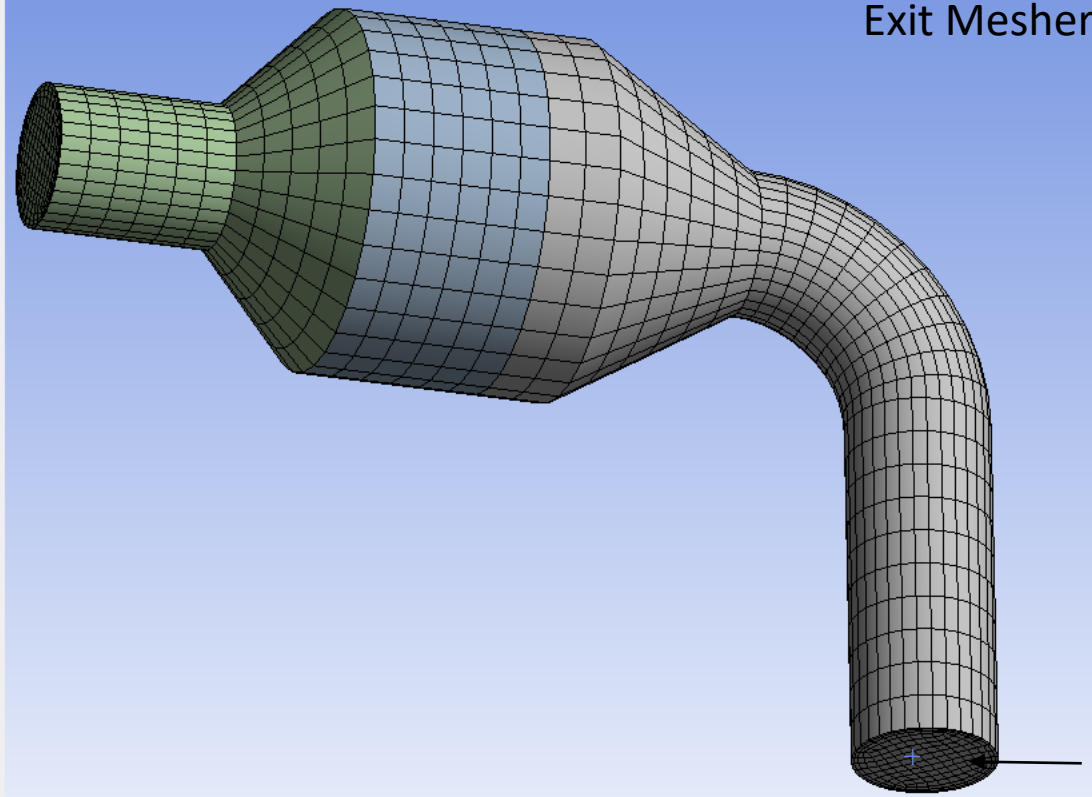
Details of "Mesh"

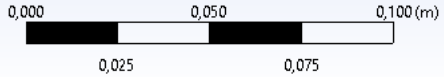
Display	
Display Style	Use Geometry Setting
Defaults	
Physics Preference	CFD
Solver Preference	Fluent
Element Order	Linear
Element Size	1,e-002 m
Export Format	Standard
Export Preview Surface Mesh	No
Sizing	
Quality	
Inflation	
Assembly Meshing	
Advanced	
Statistics	

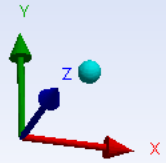
Select | Mode | Clipboard | [Empty] | Extend | Select By | Convert

ANSYS 2019 R3 ACADEMIC

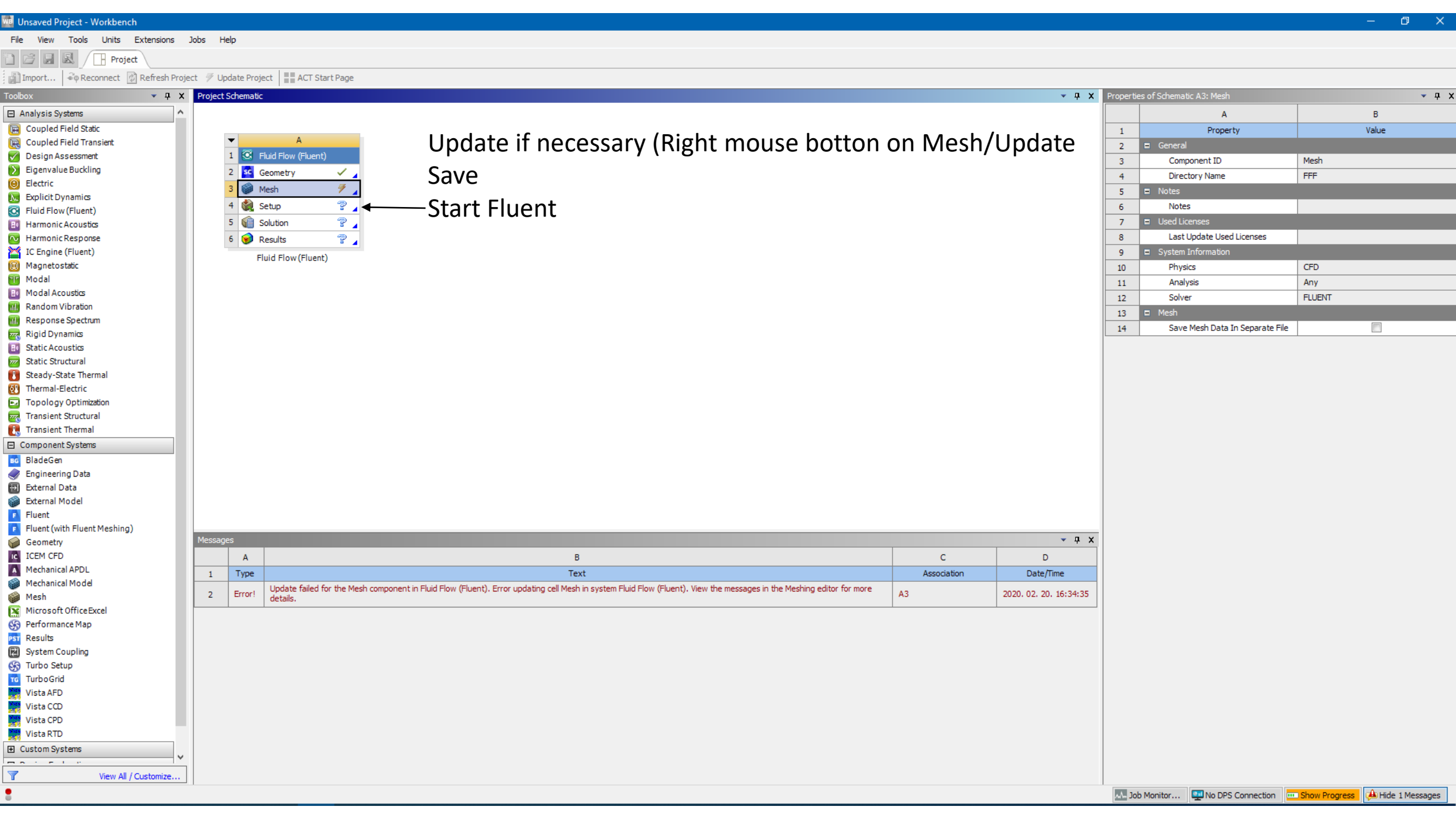
Check your new mesh and the inflation
Exit Mesher







Ready | No Messages | No Selection | Metric (m, kg, N, s, V, A) | Degrees | rad/s | Celsius


Unsaved Project - Workbench
File View Tools Units Extensions Jobs Help
Project
Import... Reconnect Refresh Project Update Project ACT Start Page
Toolbox
Analysis Systems
Coupled Field Static
Coupled Field Transient
Design Assessment
Eigenvalue Buckling
Electric
Explicit Dynamics
Fluid Flow (Fluent)
Harmonic Acoustics
Harmonic Response
IC Engine (Fluent)
Magnetostatic
Modal
Modal Acoustics
Random Vibration
Response Spectrum
Rigid Dynamics
Static Acoustics
Static Structural
Steady-State Thermal
Thermal-Electric
Topology Optimization
Transient Structural
Transient Thermal
Component Systems
BladeGen
Engineering Data
External Data
External Model
Fluent
Fluent (with Fluent Meshing)
Geometry
ICEM CFD
Mechanical APDL
Mechanical Model
Mesh
Microsoft Office Excel
Performance Map
Results
System Coupling
Turbo Setup
TurboGrid
Vista AFD
Vista CCD
Vista CPD
Vista RTD
Custom Systems
View All / Customize...
Project Schematic
A
1 Fluid Flow (Fluent)
2 Geometry
3 Mesh
4 Setup
5 Solution
6 Results
Fluid Flow (Fluent)
Update if necessary (Right mouse botton on Mesh/Update
Save
Start Fluent
Properties of Schematic A3: Mesh
A B
1 Property Value
2 General
3 Component ID Mesh
4 Directory Name FFF
5 Notes
6 Notes
7 Used Licenses
8 Last Update Used Licenses
9 System Information
10 Physics CFD
11 Analysis Any
12 Solver FLUENT
13 Mesh
14 Save Mesh Data In Separate File
Messages
A B C D
1 Type Text Association Date/Time
2 Error! Update failed for the Mesh component in Fluid Flow (Fluent). Error updating cell Mesh in system Fluid Flow (Fluent). View the messages in the Meshing editor for more details. A3 2020. 02. 20. 16:34:35
Job Monitor... No DPS Connection Show Progress Hide 1 Messages

A:Fluid Flow (Fluent) Fluent@CFDLAB-04 [3d, pbns, lam] [ANSYS Academic Teaching Mechanical and CFD]

File | Domain | Physics | User-Defined | Solution | Results | View | Parallel | Design

Quick Search (Ctrl+F)

Mesh
- Display...
- Info
- Units...
- Check
- Quality
- Scale...
- Transform
- Make Polyhedra

Zones
- Combine
- Separate
- Adjacency...
- Delete...
- Deactivate...
- Activate...
- Append
- Replace Mesh...
- Replace Zone...

Interfaces
- Mesh...
- Overset...

Mesh Models
- Dynamic Mesh...
- Mixing Planes...
- Turbo Topology...

Adapt
- Refine / Coarsen...
- More

Surface
- Create
- Manage...

Perform Mesh Check
Repair Mesh...

Outline View

Filter Text

- Setup
  - General
  - Models
  - Materials
  - Cell Zone Conditions
  - Boundary Conditions
  - Dynamic Mesh
  - Reference Values
  - Reference Frames
  - Named Expressions
- Solution
  - Methods
  - Controls
  - Report Definitions
  - Monitors
  - Cell Registers
  - Initialization
  - Calculation Activities
  - Run Calculation
- Results
  - Surfaces
  - Graphics
  - Plots
  - Animations
  - Reports
- Parameters & Customization

General

Mesh

Scale... | Check | Report Quality | Display... | Units...

Solver

Type: Pressure-Based / Density-Based

Velocity Formulation: Absolute / Relative

Time

Steady / Transient

Gravity

Perform mesh check

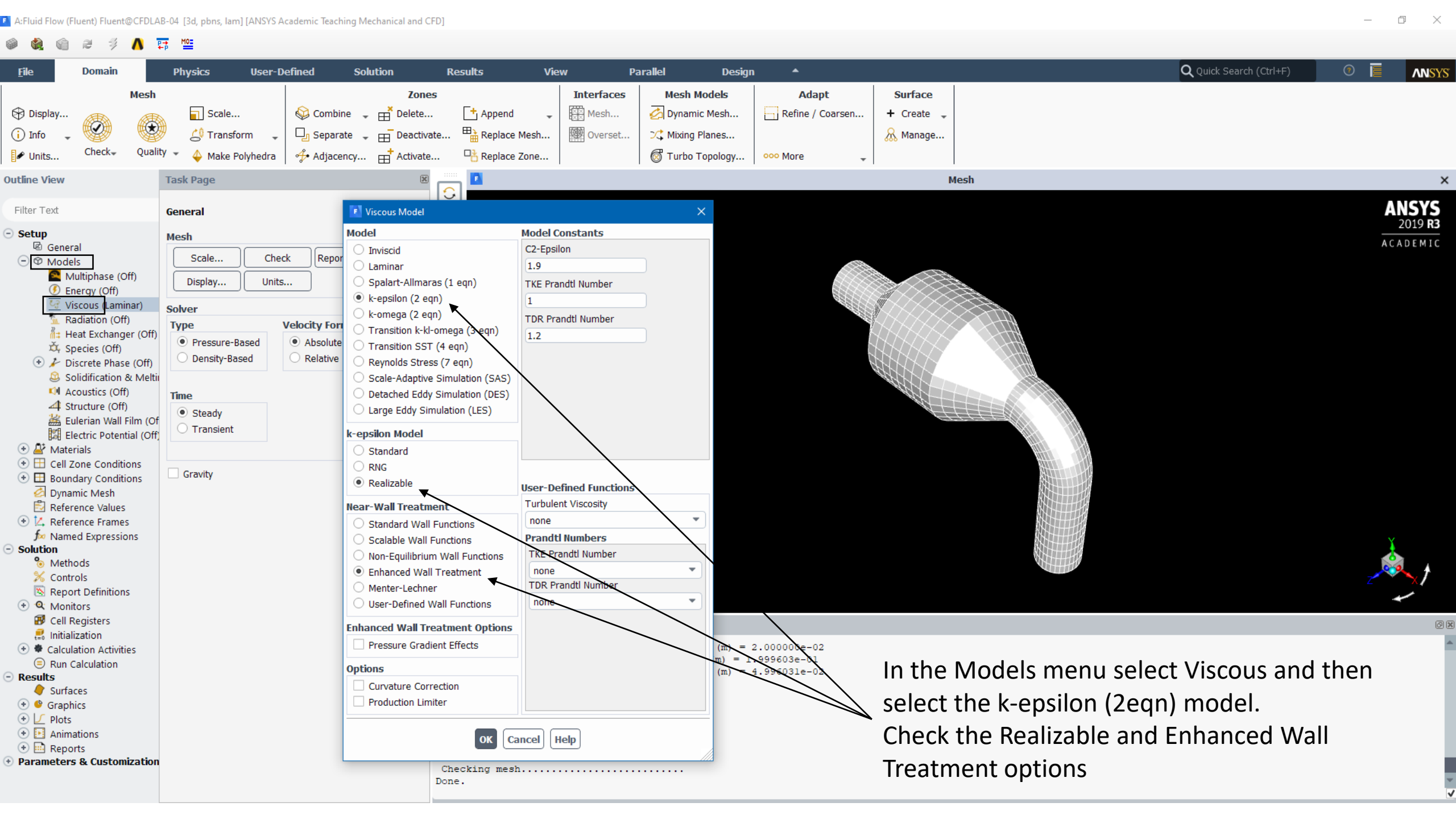
Mesh

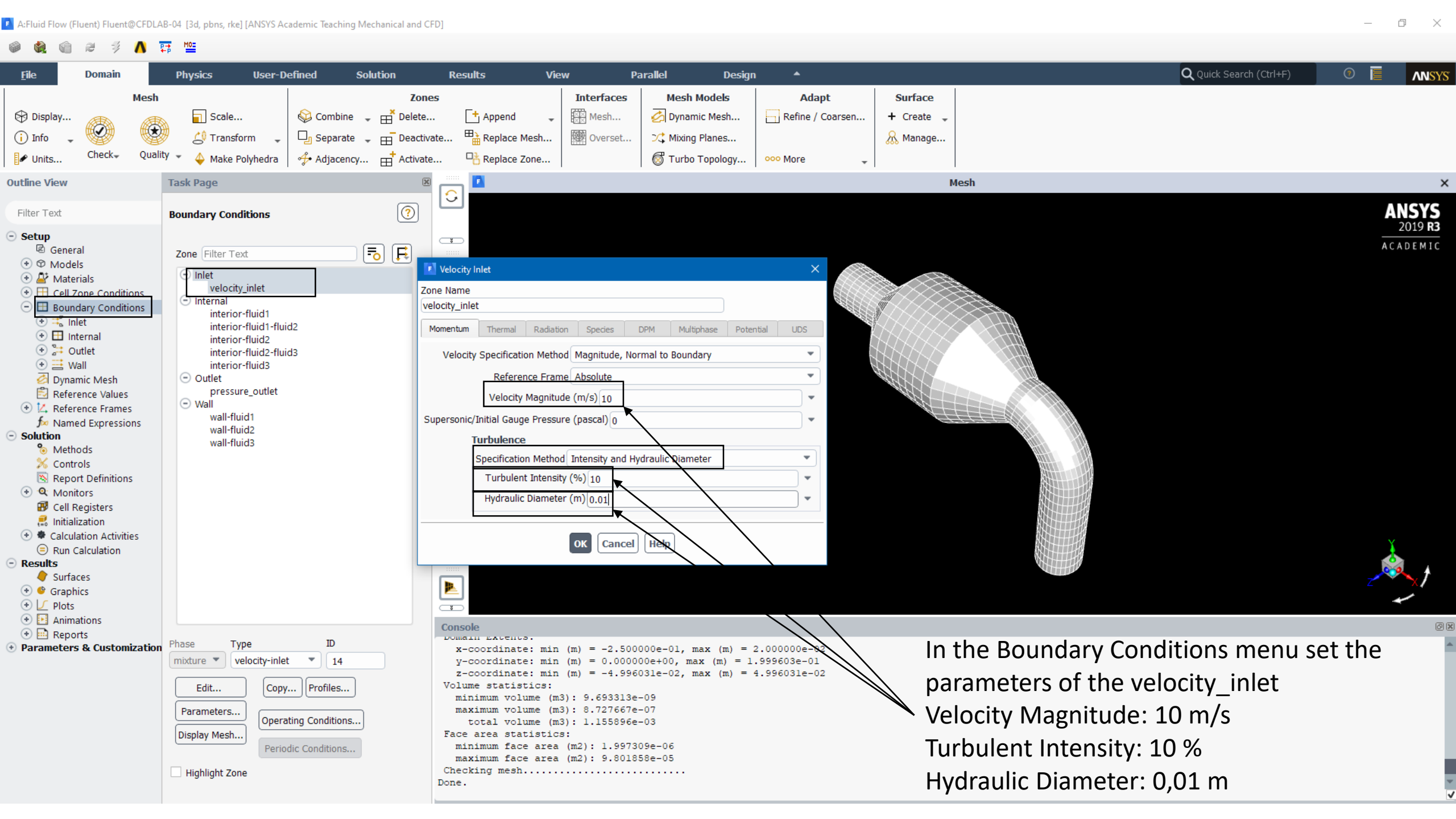
ANSYS 2019 R3 ACADEMIC

Console

```
  x-coordinate: min (m) = -2.500000e-01, max (m) = 2.000000e-02
  y-coordinate: min (m) = 0.000000e+00, max (m) = 1.999603e-01
  z-coordinate: min (m) = -4.996031e-02, max (m) = 4.996031e-02
Volume statistics:
  minimum volume (m3): 9.693313e-09
  maximum volume (m3): 8.727667e-07
    total volume (m3): 1.155896e-03
Face area statistics:
  minimum face area (m2): 1.997309e-06
  maximum face area (m2): 9.801858e-05
Checking mesh.....................
Done.
```

13:17
2020. 02. 21.


A:Fluid Flow (Fluent) Fluent@CFDLAB-04 [3d, pbns, lam] [ANSYS Academic Teaching Mechanical and CFD]
File
Domain
Physics
User-Defined
Solution
Results
View
Parallel
Design
Quick Search (Ctrl+F)
ANSYS
Mesh
Display...
Info
Units...
Check
Quality
Scale...
Transform
Make Polyhedra
Zones
Combine
Separate
Adjacency...
Delete...
Deactivate...
Activate...
Append
Replace Mesh...
Replace Zone...
Interfaces
Mesh...
Overset...
Mesh Models
Dynamic Mesh...
Mixing Planes...
Turbo Topology...
Adapt
Refine / Coarsen...
More
Surface
Create
Manage...
Outline View
Filter Text
Setup
General
Models
Multiphase (Off)
Energy (Off)
Viscous (Laminar)
Radiation (Off)
Heat Exchanger (Off)
Species (Off)
Discrete Phase (Off)
Solidification & Melti
Acoustics (Off)
Structure (Off)
Eulerian Wall Film (Of
Electric Potential (Off)
Materials
Cell Zone Conditions
Boundary Conditions
Dynamic Mesh
Reference Values
Reference Frames
Named Expressions
Solution
Methods
Controls
Report Definitions
Monitors
Cell Registers
Initialization
Calculation Activities
Run Calculation
Results
Surfaces
Graphics
Plots
Animations
Reports
Parameters & Customization
Task Page
General
Mesh
Scale...
Check
Repor
Display...
Units...
Solver
Type
Pressure-Based
Density-Based
Velocity For
Absolute
Relative
Time
Steady
Transient
Gravity
Viscous Model
Model
Inviscid
Laminar
Spalart-Allmaras (1 eqn)
k-epsilon (2 eqn)
k-omega (2 eqn)
Transition k-kl-omega (3 eqn)
Transition SST (4 eqn)
Reynolds Stress (7 eqn)
Scale-Adaptive Simulation (SAS)
Detached Eddy Simulation (DES)
Large Eddy Simulation (LES)
k-epsilon Model
Standard
RNG
Realizable
Near-Wall Treatment
Standard Wall Functions
Scalable Wall Functions
Non-Equilibrium Wall Functions
Enhanced Wall Treatment
Menter-Lechner
User-Defined Wall Functions
Enhanced Wall Treatment Options
Pressure Gradient Effects
Options
Curvature Correction
Production Limiter
Model Constants
C2-Epsilon
1.9
TKE Prandtl Number
1
TDR Prandtl Number
1.2
User-Defined Functions
Turbulent Viscosity
none
Prandtl Numbers
TKE Prandtl Number
none
TDR Prandtl Number
none
OK
Cancel
Help
Mesh
ANSYS
2019 R3
ACADEMIC
(m) = 2.000000e-02
m) = 1.999603e-01
(m) = 4.996031e-02
Checking mesh.....
Done.
In the Models menu select Viscous and then select the k-epsilon (2eqn) model.
Check the Realizable and Enhanced Wall Treatment options


A:Fluid Flow (Fluent) Fluent@CFDLAB-04 [3d, pbns, rke] [ANSYS Academic Teaching Mechanical and CFD]
File
Domain
Physics
User-Defined
Solution
Results
View
Parallel
Design
Quick Search (Ctrl+F)
ANSYS
Mesh
Display...
Info
Units...
Check
Quality
Scale...
Transform
Make Polyhedra
Zones
Combine
Separate
Adjacency...
Delete...
Deactivate...
Activate...
Append
Replace Mesh...
Replace Zone...
Interfaces
Mesh...
Overset...
Mesh Models
Dynamic Mesh...
Mixing Planes...
Turbo Topology...
Adapt
Refine / Coarsen...
More
Surface
Create
Manage...
Outline View
Filter Text
Setup
General
Models
Materials
Cell Zone Conditions
Boundary Conditions
Inlet
Internal
Outlet
Wall
Dynamic Mesh
Reference Values
Reference Frames
Named Expressions
Solution
Methods
Controls
Report Definitions
Monitors
Cell Registers
Initialization
Calculation Activities
Run Calculation
Results
Surfaces
Graphics
Plots
Animations
Reports
Parameters & Customization
Task Page
Boundary Conditions
Zone
Filter Text
Inlet
velocity_inlet
Internal
interior-fluid1
interior-fluid1-fluid2
interior-fluid2
interior-fluid2-fluid3
interior-fluid3
Outlet
pressure_outlet
Wall
wall-fluid1
wall-fluid2
wall-fluid3
Phase
Type
ID
mixture
velocity-inlet
14
Edit...
Copy...
Profiles...
Parameters...
Operating Conditions...
Display Mesh...
Periodic Conditions...
Highlight Zone
Velocity Inlet
Zone Name
velocity_inlet
Momentum
Thermal
Radiation
Species
DPM
Multiphase
Potential
UDS
Velocity Specification Method
Magnitude, Normal to Boundary
Reference Frame
Absolute
Velocity Magnitude (m/s)
10
Supersonic/Initial Gauge Pressure (pascal)
0
Turbulence
Specification Method
Intensity and Hydraulic Diameter
Turbulent Intensity (%)
10
Hydraulic Diameter (m)
0.01
OK
Cancel
Help
Mesh
ANSYS
2019 R3
ACADEMIC
Console
x-coordinate: min (m) = -2.500000e-01, max (m) = 2.000000e-02
y-coordinate: min (m) = 0.000000e+00, max (m) = 1.999603e-01
z-coordinate: min (m) = -4.996031e-02, max (m) = 4.996031e-02
Volume statistics:
minimum volume (m3): 9.693313e-09
maximum volume (m3): 8.727667e-07
total volume (m3): 1.155896e-03
Face area statistics:
minimum face area (m2): 1.997309e-06
maximum face area (m2): 9.801858e-05
Checking mesh.....
Done.
In the Boundary Conditions menu set the parameters of the velocity_inlet
Velocity Magnitude: 10 m/s
Turbulent Intensity: 10 %
Hydraulic Diameter: 0,01 m

A:Fluid Flow (Fluent) Fluent@CFDLAB-04 [3d, pbns, rke] [ANSYS Academic Teaching Mechanical and CFD]

File | Domain | Physics | User-Defined | Solution | Results | View | Parallel | Design

Quick Search (Ctrl+F)

Mesh: Display... | Info | Units... | Check | Quality | Scale... | Transform | Make Polyhedra

Zones: Combine | Separate | Adjacency... | Delete... | Deactivate... | Activate... | Append | Replace Mesh... | Replace Zone...

Interfaces: Mesh... | Overset...

Mesh Models: Dynamic Mesh... | Mixing Planes... | Turbo Topology...

Adapt: Refine / Coarsen... | More

Surface: Create | Manage...

Outline View

Filter Text

- Setup
  - General
  - Models
  - Materials
  - Cell Zone Conditions
  - Boundary Conditions
    - Inlet
    - Internal
    - Outlet
    - Wall
  - Dynamic Mesh
  - Reference Values
  - Reference Frames
  - Named Expressions
- Solution
  - Methods
  - Controls
  - Report Definitions
  - Monitors
  - Cell Registers
  - Initialization
  - Calculation Activities
  - Run Calculation
- Results
  - Surfaces
  - Graphics
  - Plots
  - Animations
  - Reports
- Parameters & Customization

Task Page

Solution Methods

Pressure-Velocity Coupling

Scheme: Coupled

Spatial Discretization

Gradient: Least Squares Cell Based

Pressure: Second Order

Momentum: Second Order Upwind

Turbulent Kinetic Energy: Second Order Upwind

Turbulent Dissipation Rate: Second Order Upwind

Transient Formulation

Non-Iterative Time Advancement

Frozen Flux Formulation

Pseudo Transient

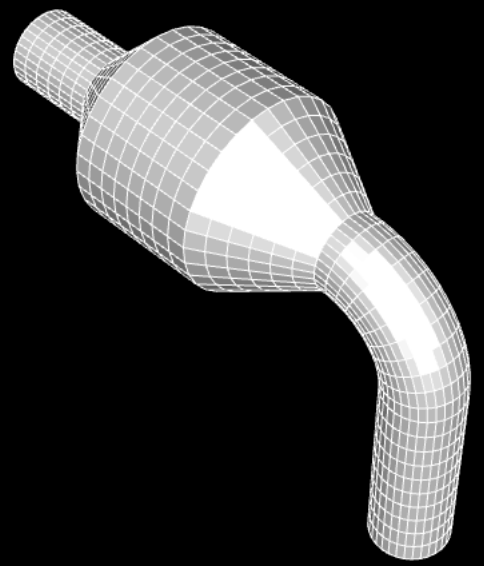
Warped-Face Gradient Correction

High Order Term Relaxation | Options...

Structure Transient Formulation

Default

Mesh

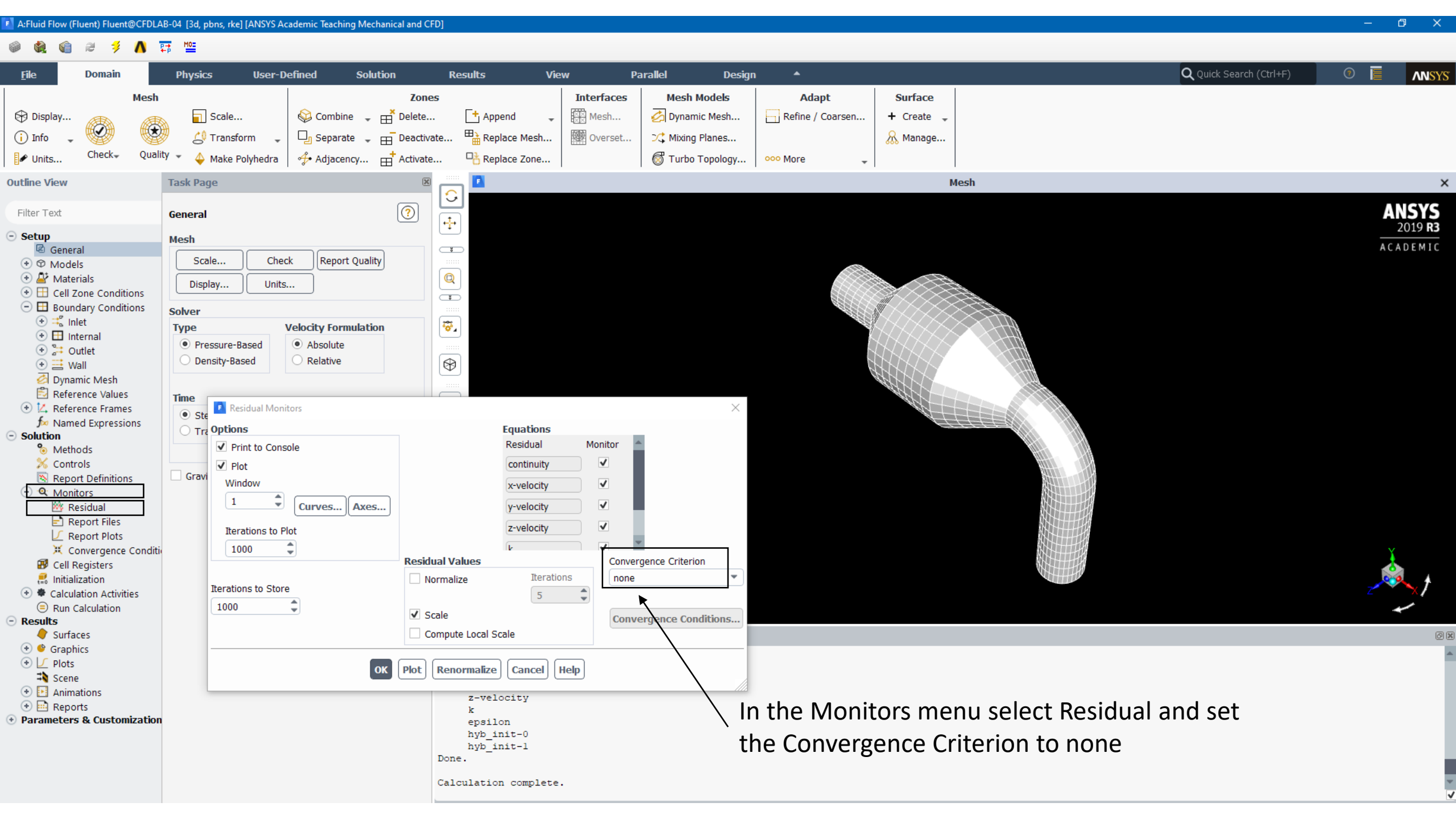


ANSYS 2019 R3 ACADEMIC

Console

```
  x-coordinate: min (m) = -2.500000e-01, max (m) = 2.000000e-02
  y-coordinate: min (m) = 0.000000e+00, max (m) = 1.999603e-01
  z-coordinate: min (m) = -4.996031e-02, max (m) = 4.996031e-02
Volume statistics:
  minimum volume (m3): 9.693313e-09
  maximum volume (m3): 8.727667e-07
    total volume (m3): 1.155896e-03
Face area statistics:
  minimum face area (m2): 1.997309e-06
  maximum face area (m2): 9.801858e-05
Checking mesh.....................
Done.
```

In the Methods menu select the higher Second Order Upwind Discretization Methods


A:Fluid Flow (Fluent) Fluent@CFDLAB-04 [3d, pbns, rke] [ANSYS Academic Teaching Mechanical and CFD]
File
Domain
Physics
User-Defined
Solution
Results
View
Parallel
Design
Quick Search (Ctrl+F)
ANSYS
Mesh
Display...
Info
Units...
Check
Quality
Scale...
Transform
Make Polyhedra
Zones
Combine
Separate
Adjacency...
Delete...
Deactivate...
Activate...
Append
Replace Mesh...
Replace Zone...
Interfaces
Mesh...
Overset...
Mesh Models
Dynamic Mesh...
Mixing Planes...
Turbo Topology...
Adapt
Refine / Coarsen...
More
Surface
Create
Manage...
Outline View
Filter Text
Setup
General
Models
Materials
Cell Zone Conditions
Boundary Conditions
Inlet
Internal
Outlet
Wall
Dynamic Mesh
Reference Values
Reference Frames
Named Expressions
Solution
Methods
Controls
Report Definitions
Monitors
Residual
Report Files
Report Plots
Convergence Conditi
Cell Registers
Initialization
Calculation Activities
Run Calculation
Results
Surfaces
Graphics
Plots
Scene
Animations
Reports
Parameters & Customization
Task Page
General
Mesh
Scale...
Check
Report Quality
Display...
Units...
Solver
Type
Pressure-Based
Density-Based
Velocity Formulation
Absolute
Relative
Time
Gravi
Residual Monitors
Options
Print to Console
Plot
Window
1
Curves...
Axes...
Iterations to Plot
1000
Iterations to Store
1000
Equations
Residual
Monitor
continuity
x-velocity
y-velocity
z-velocity
k
Residual Values
Normalize
Iterations
5
Scale
Compute Local Scale
Convergence Criterion
none
Convergence Conditions...
OK
Plot
Renormalize
Cancel
Help
Mesh
ANSYS
2019 R3
ACADEMIC
z-velocity
k
epsilon
hyb_init-0
hyb_init-1
Done.
Calculation complete.
In the Monitors menu select Residual and set
the Convergence Criterion to none

A:Fluid Flow (Fluent) Fluent@CFDLAB-04 [3d, pbns, rke] [ANSYS Academic Teaching Mechanical and CFD]

File | Domain | Physics | User-Defined | Solution | Results | View | Parallel | Design

Quick Search (Ctrl+F)

Mesh
Display... | Info | Units... | Check | Quality | Scale... | Transform | Make Polyhedra

Zones
Combine | Separate | Adjacency... | Delete... | Deactivate... | Activate... | Append | Replace Mesh... | Replace Zone...

Interfaces
Mesh... | Overset...

Mesh Models
Dynamic Mesh... | Mixing Planes... | Turbo Topology...

Adapt
Refine / Coarsen... | More

Surface
Create | Manage...

Outline View

Filter Text

- Setup
  - General
  - Models
  - Materials
  - Cell Zone Conditions
  - Boundary Conditions
    - Inlet
    - Internal
    - Outlet
    - Wall
  - Dynamic Mesh
  - Reference Values
  - Reference Frames
  - Named Expressions
- Solution
  - Methods
  - Controls
  - Report Definitions
  - Monitors
  - Cell Registers
  - Initialization
  - Calculation Activities
  - Run Calculation
- Results
  - Surfaces
  - Graphics
  - Plots
  - Scene
  - Animations
  - Reports
- Parameters & Customization

Task Page

Solution Initialization

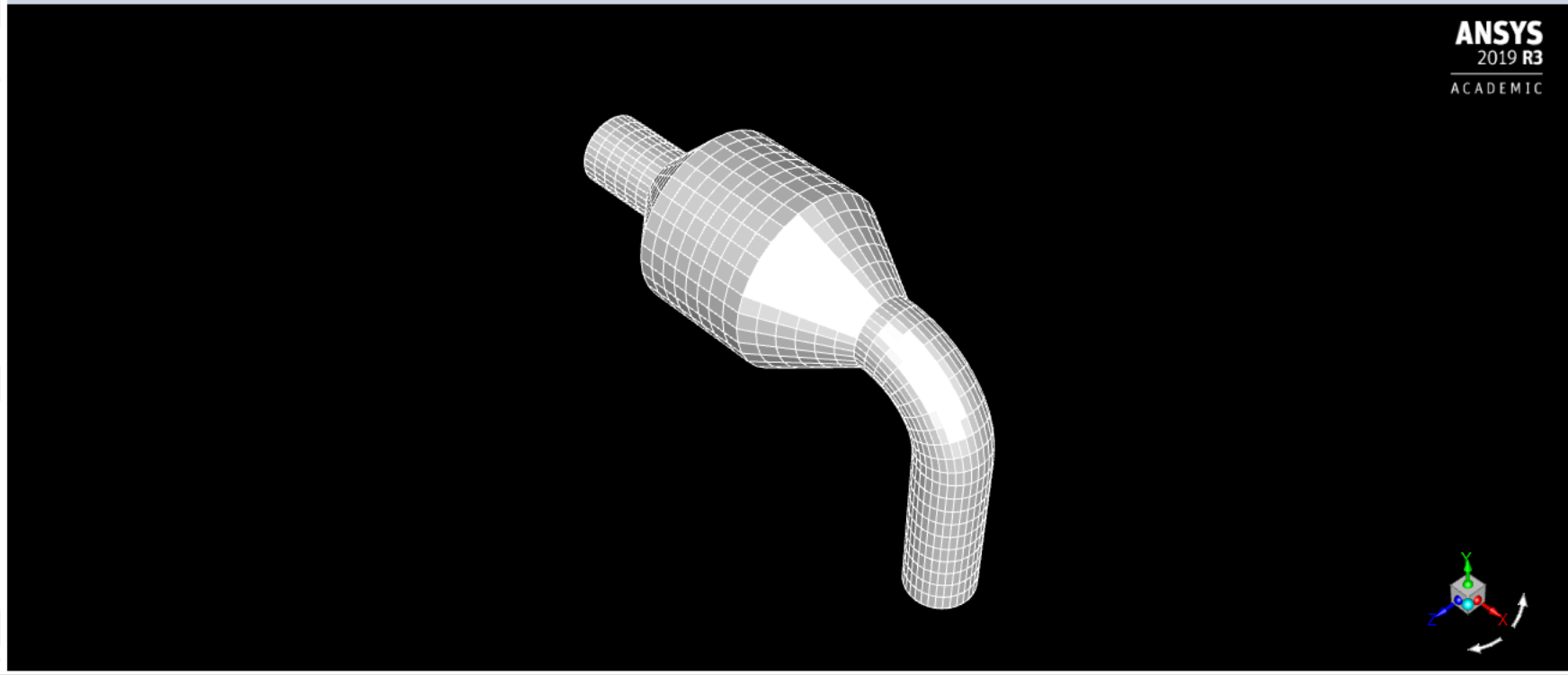
Initialization Methods

(•) Hybrid Initialization
( ) Standard Initialization

More Settings... | Initialize | Patch... | Reset DPM Sources | Reset Statistics

Initialize

Mesh



Console

```
      4          3.606091e-05
      5          1.622087e-05
      6          1.073864e-05
      7          5.899173e-06
      8          3.733213e-06
      9          2.304106e-06
      10         1.470640e-06

Hybrid initialization is done.

Warning: convergence tolerance of 1.000000e-06 not reached
during Hybrid Initialization.
```

A:Fluid Flow (Fluent) Fluent@CFDLAB-04 [3d, pbns, rke] [ANSYS Academic Teaching Mechanical and CFD]

File | Domain | Physics | User-Defined | Solution | Results | View | Parallel | Design

Quick Search (Ctrl+F)

Mesh: Display... | Info | Units... | Check | Quality | Scale... | Transform | Make Polyhedra

Zones: Combine | Separate | Adjacency... | Delete... | Deactivate... | Activate... | Append | Replace Mesh... | Replace Zone...

Interfaces: Mesh... | Overset...

Mesh Models: Dynamic Mesh... | Mixing Planes... | Turbo Topology...

Adapt: Refine / Coarsen... | More

Surface: Create | Manage...

Outline View

Filter Text

- Setup
  - General
  - Models
  - Materials
  - Cell Zone Conditions
  - Boundary Conditions
    - Inlet
    - Internal
    - Outlet
    - Wall
  - Dynamic Mesh
  - Reference Values
  - Reference Frames
  - Named Expressions
- Solution
  - Methods
  - Controls
  - Report Definitions
  - Monitors
    - Residual
    - Report Files
    - Report Plots
    - Convergence Conditi
  - Cell Registers
  - Initialization
  - Calculation Activities
  - Run Calculation
- Results
  - Surfaces
  - Graphics
  - Plots
  - Scene
  - Animations
  - Reports
- Parameters & Customization

Task Page

Run Calculation

Check Case... | Update Dynamic Mesh...

Pseudo Transient Settings

Fluid Time Scale

Time Step Method: Automatic
Time Scale Factor: 1
Length Scale Method: Conservative
Verbosity: 0

Parameters

Number of Iterations: 500
Reporting Interval: 1
Profile Update Interval: 1

Solution Processing

Statistics

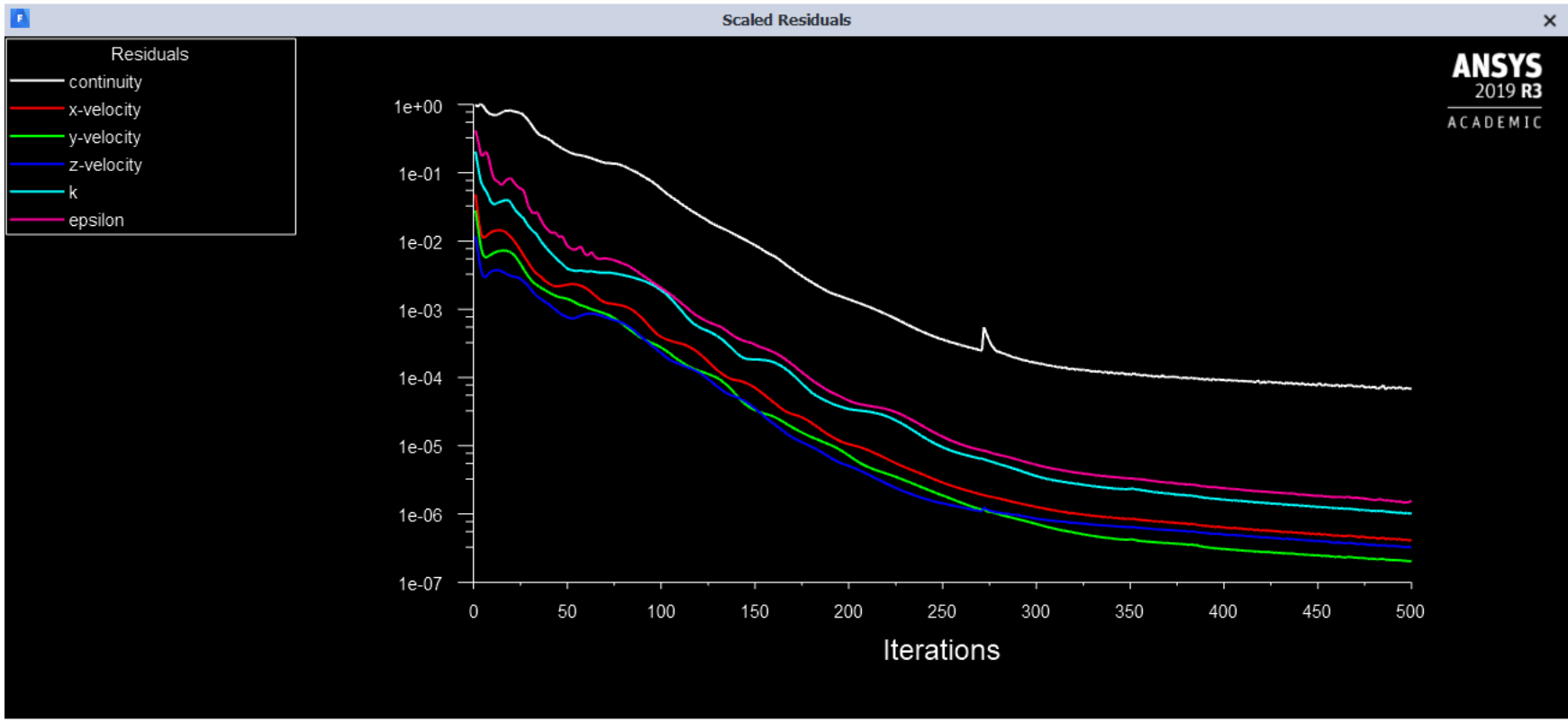
Data Sampling for Steady Statistics

Data File Quantities...

Solution Advancement

Calculate

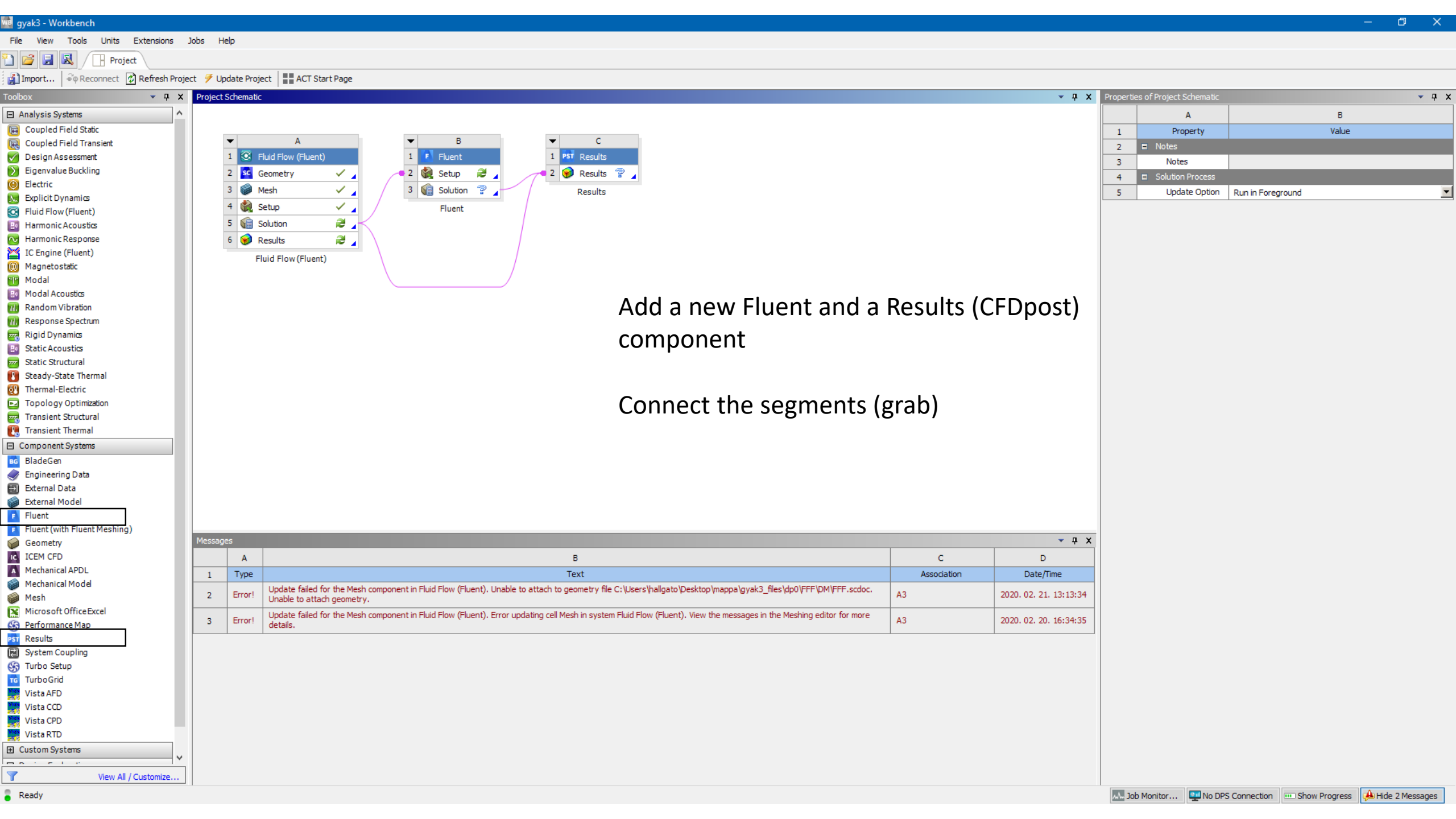
Calculate
Number of iterations: 500
Close Fluent

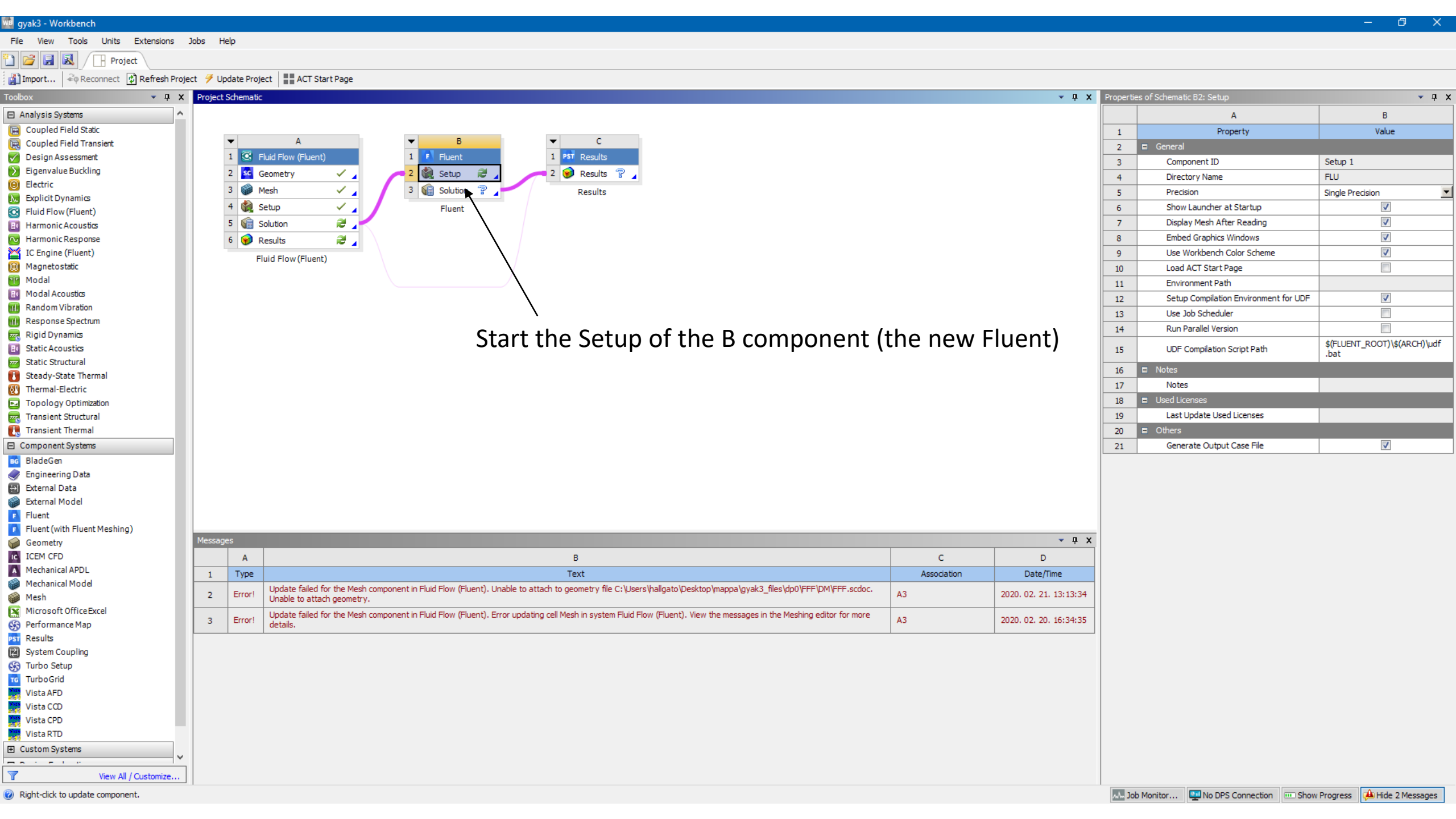


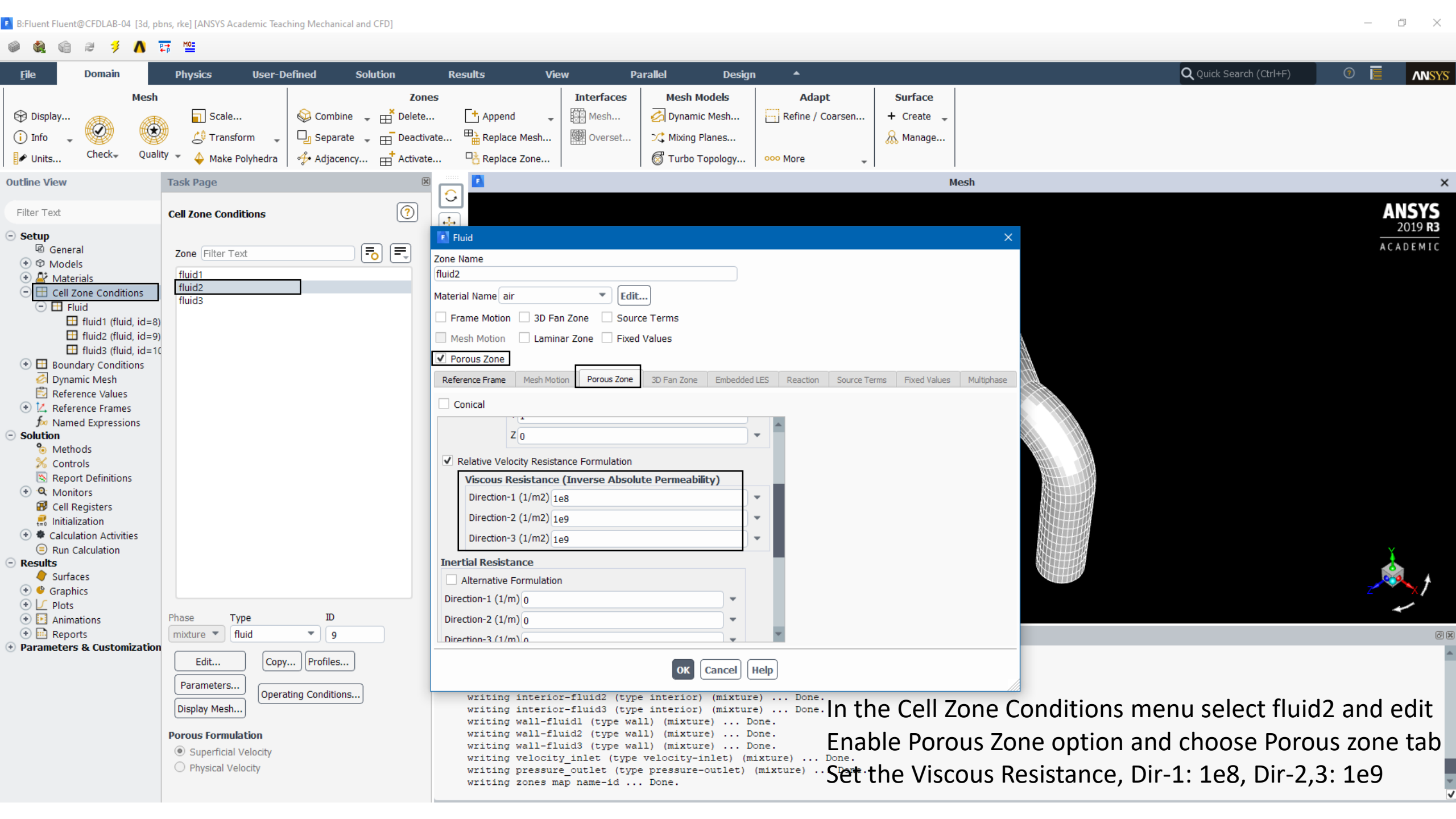
Console

```
     z-coord
     pressure
     x-velocity
     y-velocity
     z-velocity
     k
     epsilon
     hyb_init-0
     hyb_init-1
Done.

Calculation complete.
```


gyak3 - Workbench
File View Tools Units Extensions Jobs Help
Project
Import... Reconnect Refresh Project Update Project ACT Start Page
Toolbox
Analysis Systems
Coupled Field Static
Coupled Field Transient
Design Assessment
Eigenvalue Buckling
Electric
Explicit Dynamics
Fluid Flow (Fluent)
Harmonic Acoustics
Harmonic Response
IC Engine (Fluent)
Magnetostatic
Modal
Modal Acoustics
Random Vibration
Response Spectrum
Rigid Dynamics
Static Acoustics
Static Structural
Steady-State Thermal
Thermal-Electric
Topology Optimization
Transient Structural
Transient Thermal
Component Systems
BladeGen
Engineering Data
External Data
External Model
Fluent
Fluent (with Fluent Meshing)
Geometry
ICEM CFD
Mechanical APDL
Mechanical Model
Mesh
Microsoft Office Excel
Performance Map
Results
System Coupling
Turbo Setup
TurboGrid
Vista AFD
Vista CCD
Vista CPD
Vista RTD
Custom Systems
View All / Customize...
Project Schematic
A
1 Fluid Flow (Fluent)
2 Geometry
3 Mesh
4 Setup
5 Solution
6 Results
Fluid Flow (Fluent)
B
1 Fluent
2 Setup
3 Solution
Fluent
C
1 Results
2 Results
Results
Add a new Fluent and a Results (CFDpost) component
Connect the segments (grab)
Messages
A B C D
1 Type Text Association Date/Time
2 Error! Update failed for the Mesh component in Fluid Flow (Fluent). Unable to attach to geometry file C:\Users\hallgato\Desktop\mappa\gyak3_files\dp0\FFF\DM\FFF.scdoc. Unable to attach geometry. A3 2020. 02. 21. 13:13:34
3 Error! Update failed for the Mesh component in Fluid Flow (Fluent). Error updating cell Mesh in system Fluid Flow (Fluent). View the messages in the Meshing editor for more details. A3 2020. 02. 20. 16:34:35
Properties of Project Schematic
A B
1 Property Value
2 Notes
3 Notes
4 Solution Process
5 Update Option Run in Foreground
Ready
Job Monitor... No DPS Connection Show Progress Hide 2 Messages


gyak3 - Workbench
File View Tools Units Extensions Jobs Help
Project
Import... Reconnect Refresh Project Update Project ACT Start Page
Toolbox
Analysis Systems
Coupled Field Static
Coupled Field Transient
Design Assessment
Eigenvalue Buckling
Electric
Explicit Dynamics
Fluid Flow (Fluent)
Harmonic Acoustics
Harmonic Response
IC Engine (Fluent)
Magnetostatic
Modal
Modal Acoustics
Random Vibration
Response Spectrum
Rigid Dynamics
Static Acoustics
Static Structural
Steady-State Thermal
Thermal-Electric
Topology Optimization
Transient Structural
Transient Thermal
Component Systems
BladeGen
Engineering Data
External Data
External Model
Fluent
Fluent (with Fluent Meshing)
Geometry
ICEM CFD
Mechanical APDL
Mechanical Model
Mesh
Microsoft Office Excel
Performance Map
Results
System Coupling
Turbo Setup
TurboGrid
Vista AFD
Vista CCD
Vista CPD
Vista RTD
Custom Systems
View All / Customize...
Project Schematic
A
1 Fluid Flow (Fluent)
2 Geometry
3 Mesh
4 Setup
5 Solution
6 Results
Fluid Flow (Fluent)
B
1 Fluent
2 Setup
3 Solution
Fluent
C
1 Results
2 Results
Results
Start the Setup of the B component (the new Fluent)
Messages
A B C D
1 Type Text Association Date/Time
2 Error! Update failed for the Mesh component in Fluid Flow (Fluent). Unable to attach to geometry file C:\Users\hallgato\Desktop\mappa\gyak3_files\dp0\FFF\DM\FFF.scdoc. Unable to attach geometry. A3 2020. 02. 21. 13:13:34
3 Error! Update failed for the Mesh component in Fluid Flow (Fluent). Error updating cell Mesh in system Fluid Flow (Fluent). View the messages in the Meshing editor for more details. A3 2020. 02. 20. 16:34:35
Properties of Schematic B2: Setup
A B
1 Property Value
2 General
3 Component ID Setup 1
4 Directory Name FLU
5 Precision Single Precision
6 Show Launcher at Startup
7 Display Mesh After Reading
8 Embed Graphics Windows
9 Use Workbench Color Scheme
10 Load ACT Start Page
11 Environment Path
12 Setup Compilation Environment for UDF
13 Use Job Scheduler
14 Run Parallel Version
15 UDF Compilation Script Path $(FLUENT_ROOT)\$(ARCH)\udf.bat
16 Notes
17 Notes
18 Used Licenses
19 Last Update Used Licenses
20 Others
21 Generate Output Case File
Right-click to update component.
Job Monitor... No DPS Connection Show Progress Hide 2 Messages


B:Fluent Fluent@CFDLAB-04 [3d, pbns, rke] [ANSYS Academic Teaching Mechanical and CFD]
File
Domain
Physics
User-Defined
Solution
Results
View
Parallel
Design
Quick Search (Ctrl+F)
ANSYS
Mesh
Display...
Info
Units...
Check
Quality
Scale...
Transform
Make Polyhedra
Zones
Combine
Separate
Adjacency...
Delete...
Deactivate...
Activate...
Append
Replace Mesh...
Replace Zone...
Interfaces
Mesh...
Overset...
Mesh Models
Dynamic Mesh...
Mixing Planes...
Turbo Topology...
Adapt
Refine / Coarsen...
More
Surface
Create
Manage...
Outline View
Filter Text
Setup
General
Models
Materials
Cell Zone Conditions
Fluid
fluid1 (fluid, id=8)
fluid2 (fluid, id=9)
fluid3 (fluid, id=10
Boundary Conditions
Dynamic Mesh
Reference Values
Reference Frames
Named Expressions
Solution
Methods
Controls
Report Definitions
Monitors
Cell Registers
Initialization
Calculation Activities
Run Calculation
Results
Surfaces
Graphics
Plots
Animations
Reports
Parameters & Customization
Task Page
Cell Zone Conditions
Zone
Filter Text
fluid1
fluid2
fluid3
Phase
Type
ID
mixture
fluid
9
Edit...
Copy...
Profiles...
Parameters...
Operating Conditions...
Display Mesh...
Porous Formulation
Superficial Velocity
Physical Velocity
Mesh
ANSYS
2019 R3
ACADEMIC
Fluid
Zone Name
fluid2
Material Name
air
Edit...
Frame Motion
3D Fan Zone
Source Terms
Mesh Motion
Laminar Zone
Fixed Values
Porous Zone
Reference Frame
Mesh Motion
Porous Zone
3D Fan Zone
Embedded LES
Reaction
Source Terms
Fixed Values
Multiphase
Conical
Z
0
Relative Velocity Resistance Formulation
Viscous Resistance (Inverse Absolute Permeability)
Direction-1 (1/m2)
1e8
Direction-2 (1/m2)
1e9
Direction-3 (1/m2)
1e9
Inertial Resistance
Alternative Formulation
Direction-1 (1/m)
0
Direction-2 (1/m)
0
Direction-3 (1/m)
0
OK
Cancel
Help
writing interior-fluid2 (type interior) (mixture) ... Done.
writing interior-fluid3 (type interior) (mixture) ... Done.
writing wall-fluid1 (type wall) (mixture) ... Done.
writing wall-fluid2 (type wall) (mixture) ... Done.
writing wall-fluid3 (type wall) (mixture) ... Done.
writing velocity_inlet (type velocity-inlet) (mixture) ... Done.
writing pressure_outlet (type pressure-outlet) (mixture) ... Done.
writing zones map name-id ... Done.
In the Cell Zone Conditions menu select fluid2 and edit
Enable Porous Zone option and choose Porous zone tab
Set the Viscous Resistance, Dir-1: 1e8, Dir-2,3: 1e9

A:Fluid Flow (Fluent) Fluent@CFDLAB-04 [3d, pbns, rke] [ANSYS Academic Teaching Mechanical and CFD]

File | Domain | Physics | User-Defined | Solution | Results | View | Parallel | Design

Quick Search (Ctrl+F)

Mesh
Display... | Info | Units... | Check | Quality | Scale... | Transform | Make Polyhedra

Zones
Combine | Separate | Adjacency... | Delete... | Deactivate... | Activate... | Append | Replace Mesh... | Replace Zone...

Interfaces
Mesh... | Overset...

Mesh Models
Dynamic Mesh... | Mixing Planes... | Turbo Topology...

Adapt
Refine / Coarsen... | More

Surface
Create | Manage...

Outline View

Filter Text

- Setup
  - General
  - Models
  - Materials
  - Cell Zone Conditions
  - Boundary Conditions
    - Inlet
    - Internal
    - Outlet
    - Wall
  - Dynamic Mesh
  - Reference Values
  - Reference Frames
  - Named Expressions
- Solution
  - Methods
  - Controls
  - Report Definitions
  - Monitors
  - Cell Registers
  - Initialization
  - Calculation Activities
  - Run Calculation
- Results
  - Surfaces
  - Graphics
  - Plots
  - Scene
  - Animations
  - Reports
- Parameters & Customization

Task Page

Solution Initialization

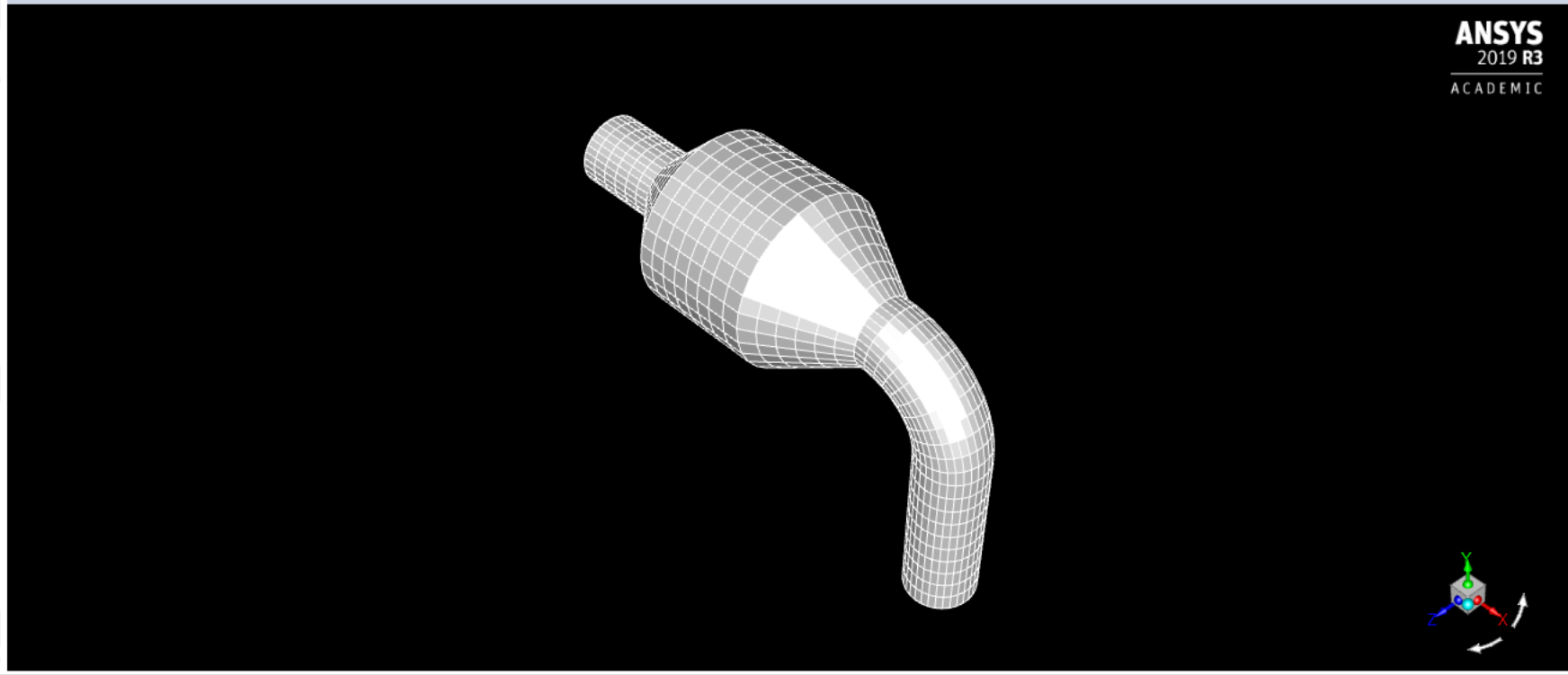
Initialization Methods

(•) Hybrid Initialization
( ) Standard Initialization

More Settings... | Initialize | Patch... | Reset DPM Sources | Reset Statistics

Initialize

Mesh



Console

```
      4          3.606091e-05
      5          1.622087e-05
      6          1.073864e-05
      7          5.899173e-06
      8          3.733213e-06
      9          2.304106e-06
      10         1.470640e-06

Hybrid initialization is done.

Warning: convergence tolerance of 1.000000e-06 not reached
during Hybrid Initialization.
```

B:Fluent Fluent@CFDLAB-04 [3d, pbns, rke] [ANSYS Academic Teaching Mechanical and CFD]

File | Domain | Physics | User-Defined | Solution | Results | View | Parallel | Design

Quick Search (Ctrl+F)

Mesh: Display... | Info | Units... | Check | Quality | Scale... | Transform | Make Polyhedra

Zones: Combine | Separate | Adjacency... | Delete... | Deactivate... | Activate... | Append | Replace Mesh... | Replace Zone...

Interfaces: Mesh... | Overset...

Mesh Models: Dynamic Mesh... | Mixing Planes... | Turbo Topology...

Adapt: Refine / Coarsen... | More

Surface: Create | Manage...

Outline View

Filter Text

- Setup
  - General
  - Models
  - Materials
  - Cell Zone Conditions
    - Fluid
      - fluid1 (fluid, id=8)
      - fluid2 (fluid, id=9)
      - fluid3 (fluid, id=10
  - Boundary Conditions
  - Dynamic Mesh
  - Reference Values
  - Reference Frames
  - Named Expressions
- Solution
  - Methods
  - Controls
  - Report Definitions
  - Monitors
  - Cell Registers
  - Initialization
  - Calculation Activities
  - Run Calculation
- Results
  - Surfaces
  - Graphics
  - Plots
  - Scene
  - Animations
  - Reports
- Parameters & Customization

Task Page

Run Calculation

Check Case... | Update Dynamic Mesh...

Pseudo Transient Settings

Fluid Time Scale

Time Step Method: Automatic
Time Scale Factor: 1
Length Scale Method: Conservative
Verbosity: 0

Parameters

Number of Iterations: 300
Reporting Interval: 1
Profile Update Interval: 1

Solution Processing

Statistics

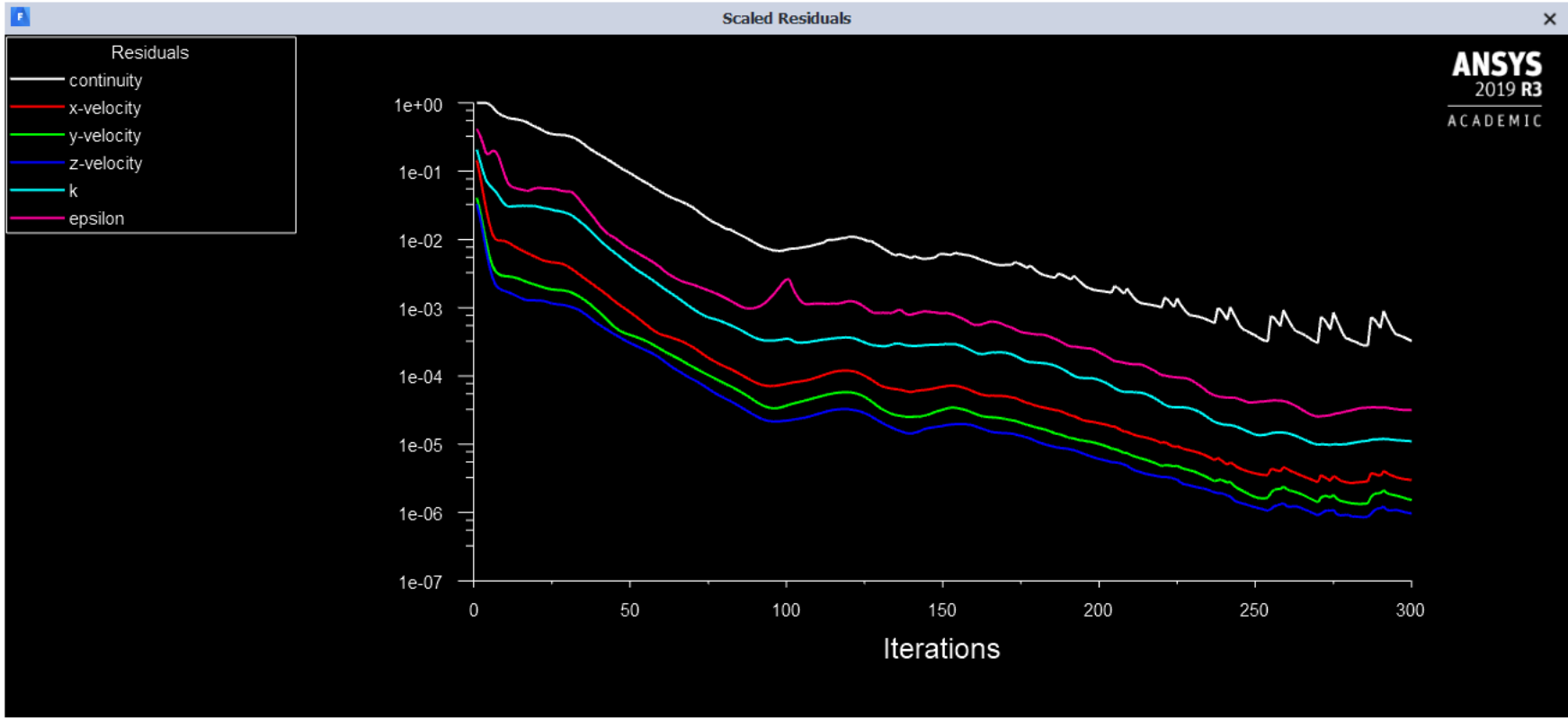
Data Sampling for Steady Statistics

Data File Quantities...

Solution Advancement

Calculate

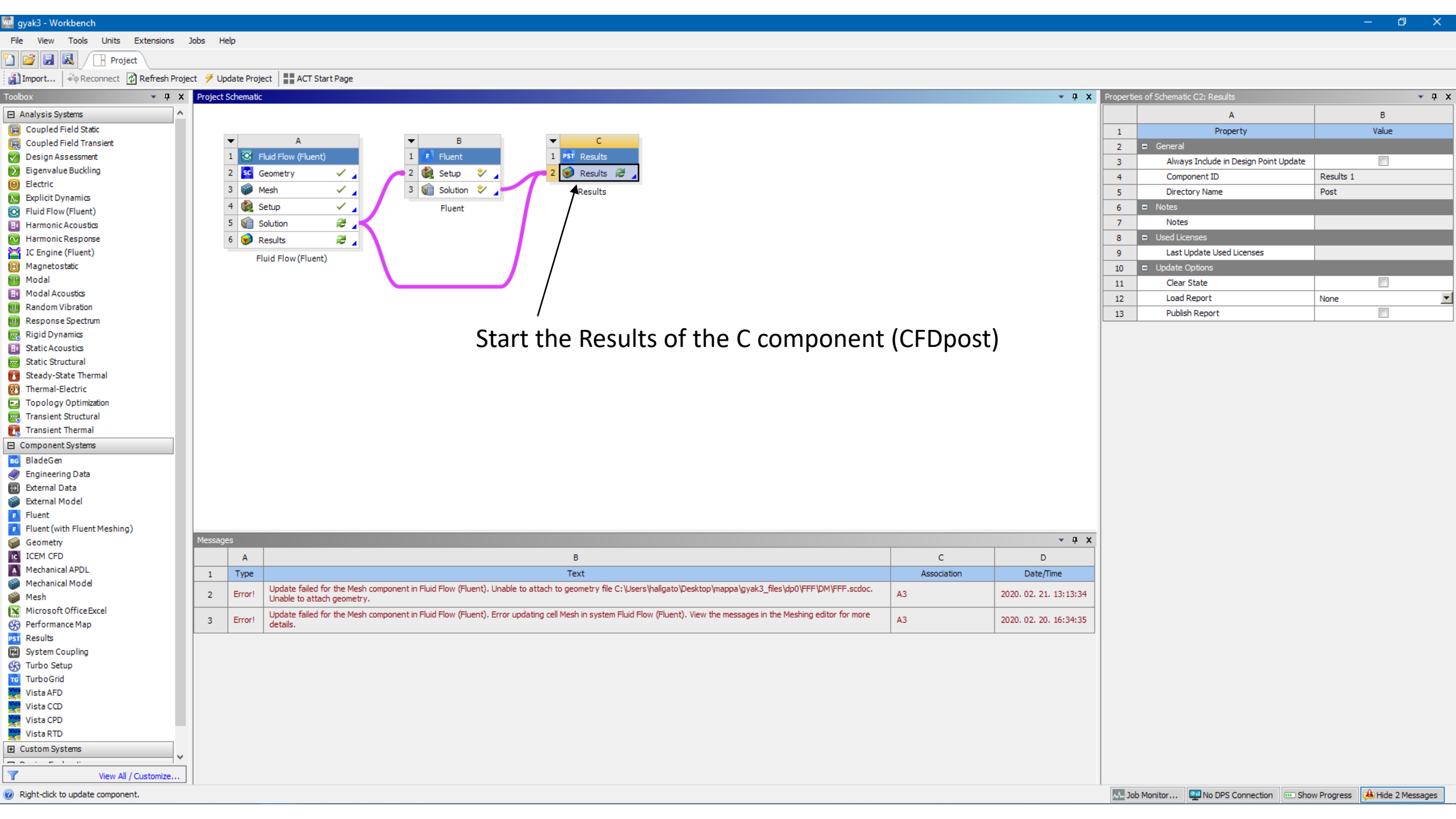
Calculate
Number of iterations: 300
Close Fluent

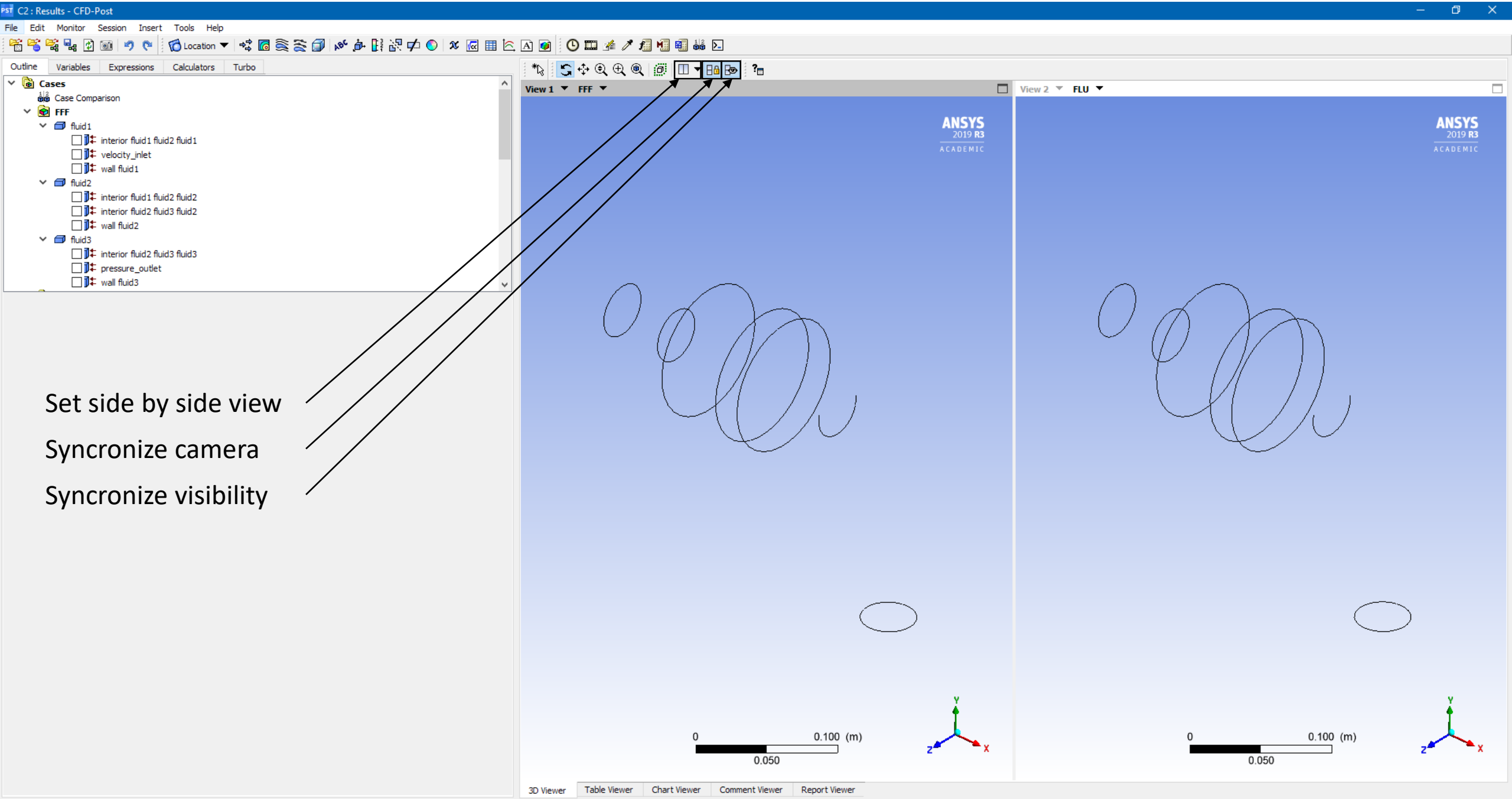


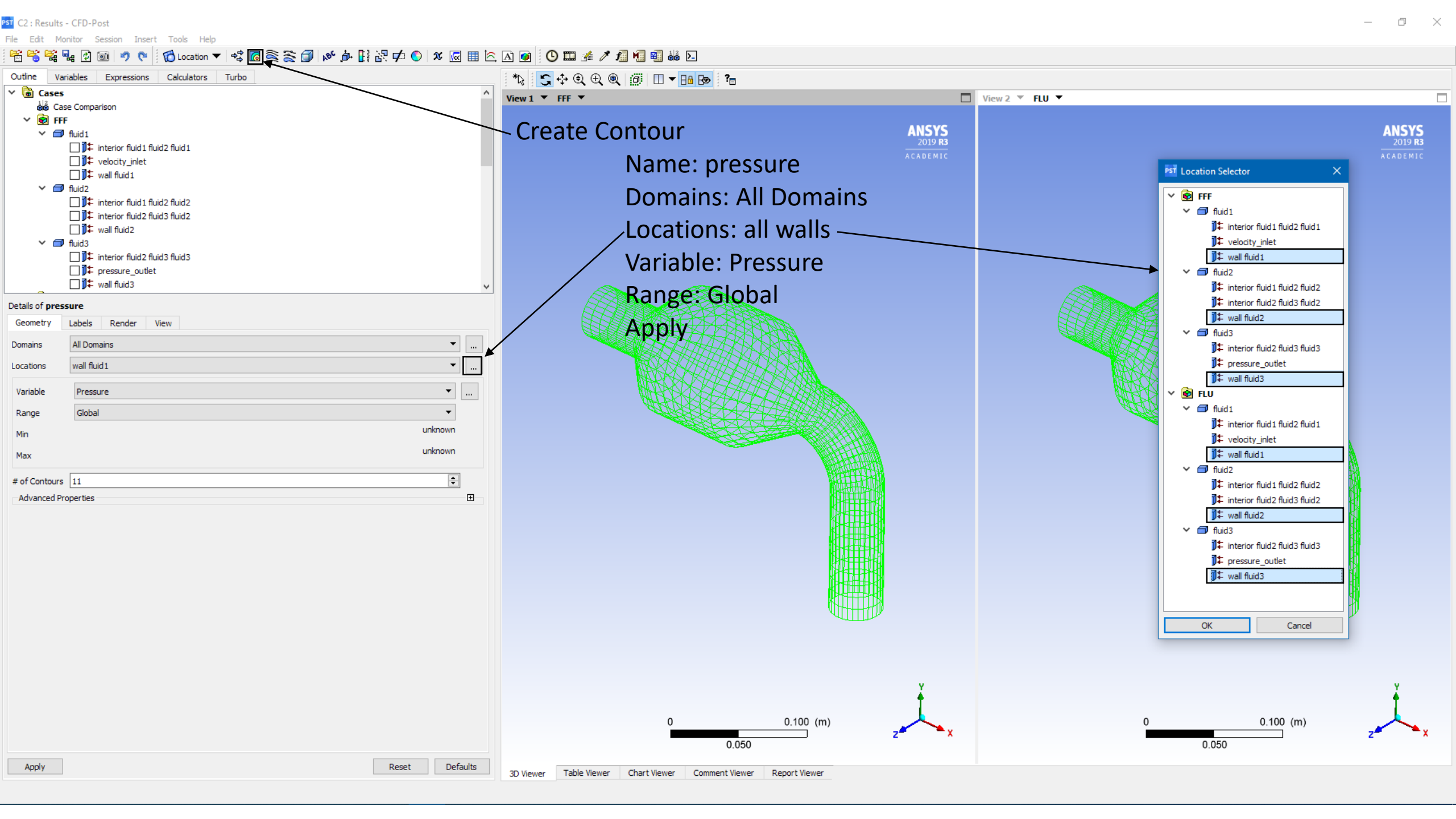
Console

```
    z-coord
    pressure
    x-velocity
    y-velocity
    z-velocity
    k
    epsilon
    hyb_init-0
    hyb_init-1
Done.

Calculation complete.
```


gyak3 - Workbench
File View Tools Units Extensions Jobs Help
Project
Import... Reconnect Refresh Project Update Project ACT Start Page
Toolbox
Analysis Systems
Coupled Field Static
Coupled Field Transient
Design Assessment
Eigenvalue Buckling
Electric
Explicit Dynamics
Fluid Flow (Fluent)
Harmonic Acoustics
Harmonic Response
IC Engine (Fluent)
Magnetostatic
Modal
Modal Acoustics
Random Vibration
Response Spectrum
Rigid Dynamics
Static Acoustics
Static Structural
Steady-State Thermal
Thermal-Electric
Topology Optimization
Transient Structural
Transient Thermal
Component Systems
BladeGen
Engineering Data
External Data
External Model
Fluent
Fluent (with Fluent Meshing)
Geometry
ICEM CFD
Mechanical APDL
Mechanical Model
Mesh
Microsoft Office Excel
Performance Map
Results
System Coupling
Turbo Setup
TurboGrid
Vista AFD
Vista CCD
Vista CPD
Vista RTD
Custom Systems
View All / Customize...
Project Schematic
A
1 Fluid Flow (Fluent)
2 Geometry
3 Mesh
4 Setup
5 Solution
6 Results
Fluid Flow (Fluent)
B
1 Fluent
2 Setup
3 Solution
Fluent
C
1 Results
2 Results
Results
Start the Results of the C component (CFDpost)
Properties of Schematic C2: Results
A B
1 Property Value
2 General
3 Always Include in Design Point Update
4 Component ID Results 1
5 Directory Name Post
6 Notes
7 Notes
8 Used Licenses
9 Last Update Used Licenses
10 Update Options
11 Clear State
12 Load Report None
13 Publish Report
Messages
A B C D
1 Type Text Association Date/Time
2 Error! Update failed for the Mesh component in Fluid Flow (Fluent). Unable to attach to geometry file C:\Users\hallgato\Desktop\mappa\gyak3_files\dp0\FFF\DM\FFF.scdoc. Unable to attach geometry. A3 2020. 02. 21. 13:13:34
3 Error! Update failed for the Mesh component in Fluid Flow (Fluent). Error updating cell Mesh in system Fluid Flow (Fluent). View the messages in the Meshing editor for more details. A3 2020. 02. 20. 16:34:35
Right-click to update component.
Job Monitor... No DPS Connection Show Progress Hide 2 Messages


C2 : Results - CFD-Post
File Edit Monitor Session Insert Tools Help
Location
Outline Variables Expressions Calculators Turbo
Cases
Case Comparison
FFF
fluid1
interior fluid1 fluid2 fluid1
velocity_inlet
wall fluid1
fluid2
interior fluid1 fluid2 fluid2
interior fluid2 fluid3 fluid2
wall fluid2
fluid3
interior fluid2 fluid3 fluid3
pressure_outlet
wall fluid3
View 1 FFF
View 2 FLU
ANSYS 2019 R3 ACADEMIC
0 0.100 (m) 0.050
Y Z X
3D Viewer Table Viewer Chart Viewer Comment Viewer Report Viewer
Set side by side view
Syncronize camera
Syncronize visibility


C2 : Results - CFD-Post
File Edit Monitor Session Insert Tools Help
Location
Outline Variables Expressions Calculators Turbo
Cases
Case Comparison
FFF
fluid1
interior fluid1 fluid2 fluid1
velocity_inlet
wall fluid1
fluid2
interior fluid1 fluid2 fluid2
interior fluid2 fluid3 fluid2
wall fluid2
fluid3
interior fluid2 fluid3 fluid3
pressure_outlet
wall fluid3
Details of pressure
Geometry Labels Render View
Domains All Domains
Locations wall fluid1
Variable Pressure
Range Global
Min unknown
Max unknown
# of Contours 11
Advanced Properties
Apply Reset Defaults
Create Contour
Name: pressure
Domains: All Domains
Locations: all walls
Variable: Pressure
Range: Global
Apply
View 1 FFF
ANSYS 2019 R3 ACADEMIC
0 0.100 (m) 0.050
Y Z X
3D Viewer Table Viewer Chart Viewer Comment Viewer Report Viewer
View 2 FLU
Location Selector
FFF
fluid1
interior fluid1 fluid2 fluid1
velocity_inlet
wall fluid1
fluid2
interior fluid1 fluid2 fluid2
interior fluid2 fluid3 fluid2
wall fluid2
fluid3
interior fluid2 fluid3 fluid3
pressure_outlet
wall fluid3
FLU
fluid1
interior fluid1 fluid2 fluid1
velocity_inlet
wall fluid1
fluid2
interior fluid1 fluid2 fluid2
interior fluid2 fluid3 fluid2
wall fluid2
fluid3
interior fluid2 fluid3 fluid3
pressure_outlet
wall fluid3
OK Cancel

C2 : Results - CFD-Post

File Edit Monitor Session Insert Tools Help

Location

Outline | Variables | Expressions | Calculators | Turbo

- Cases
  - Case Comparison
  - FFF
  - FLU
- User Locations and Plots
  - pressure
  - Default Transform
  - Default Legend View 1
  - Wireframe
- Report
  - Title Page
  - File Report
  - Mesh Report
  - Physics Report
  - Solution Report

Details of **pressure**

Geometry | Labels | Render | View

Domains: All Domains

Locations: wall fluid1,wall fluid2,wall fluid3

Variable: Pressure

Range: Global

Min: -72.5641 [Pa]

Max: 266.225 [Pa]

# of Contours: 11

Advanced Properties

Apply | Reset | Defaults

View 1 FFF

Pressure
pressure
2.662e+02
2.323e+02
1.985e+02
1.646e+02
1.307e+02
9.683e+01
6.295e+01
2.907e+01
-4.806e+00
-3.869e+01
-7.256e+01
[Pa]

ANSYS 2019 R3 ACADEMIC

0 0.090 (m)
0.045

View 2 FLU

Pressure
pressure
2.662e+02
2.323e+02
1.985e+02
1.646e+02
1.307e+02
9.683e+01
6.295e+01
2.907e+01
-4.806e+00
-3.869e+01
-7.256e+01
[Pa]

ANSYS 2019 R3 ACADEMIC

0 0.090 (m)
0.045

3D Viewer | Table Viewer | Chart Viewer | Comment Viewer | Report Viewer

C2 : Results - CFD-Post

File Edit Monitor Session Insert Tools Help

Outline | Variables | Expressions | Calculators | Turbo

- Cases
  - Case Comparison
  - FFF
  - FLU
- User Locations and Plots
  - pressure
  - Default Transform
  - Default Legend View 1
  - Wireframe
- Report
  - Title Page
  - File Report
  - Mesh Report
  - Physics Report
  - Solution Report

Details of **pressure**

Geometry | Labels | Render | View

Domains: All Domains

Locations: wall fluid1,wall fluid2,wall fluid3

Variable: Pressure

Range: User Specified

Min: -60 [Pa]

Max: 100 [Pa]

# of Contours: 11

Advanced Properties

Change Range to User Specified
min: -60 Pa
max: 100 Pa

Apply | Reset | Defaults

View 1 FFF

Pressure
pressure
1.000e+02
8.400e+01
6.800e+01
5.200e+01
3.600e+01
2.000e+01
4.000e+00
-1.200e+01
-2.800e+01
-4.400e+01
-6.000e+01
[Pa]

ANSYS 2019 R3 ACADEMIC

0 0.090 (m)
0.045

View 2 FLU

Pressure
pressure
1.000e+02
8.400e+01
6.800e+01
5.200e+01
3.600e+01
2.000e+01
4.000e+00
-1.200e+01
-2.800e+01
-4.400e+01
-6.000e+01
[Pa]

ANSYS 2019 R3 ACADEMIC

0 0.090 (m)
0.045

3D Viewer | Table Viewer | Chart Viewer | Comment Viewer | Report Viewer

C2 : Results - CFD-Post

File Edit Monitor Session Insert Tools Help

Location

Outline | Variables | Expressions | Calculators | Turbo

- Cases
  - Case Comparison
  - FFF
  - FLU
- User Locations and Plots
  - pressure
  - Default Transform
  - Default Legend View 1
  - Wireframe
- Report
  - Title Page
  - File Report
  - Mesh Report
  - Physics Report
  - Solution Report

Turn of the visibility of the contour

Details of **pressure**

Geometry | Labels | Render | View

Domains: All Domains

Locations: wall fluid1,wall fluid2,wall fluid3

Variable: Pressure

Range: User Specified

Min: -60 [Pa]

Max: 100 [Pa]

# of Contours: 11

Advanced Properties

Apply | Reset | Defaults

View 1 FFF

Pressure
pressure
1.000e+02
8.400e+01
6.800e+01
5.200e+01
3.600e+01
2.000e+01
4.000e+00
-1.200e+01
-2.800e+01
-4.400e+01
-6.000e+01
[Pa]

ANSYS 2019 R3 ACADEMIC

0 0.090 (m)
0.045

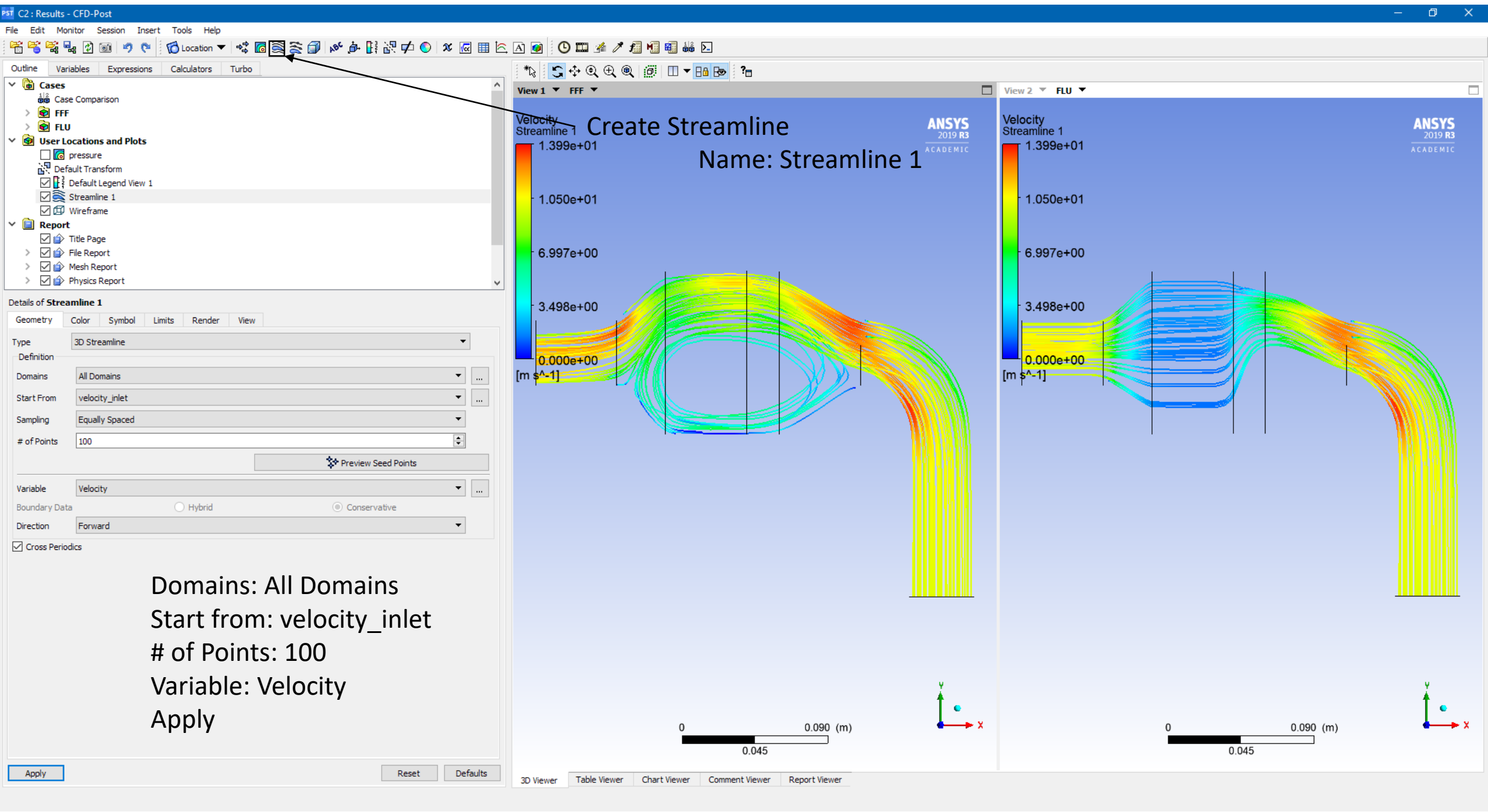
View 2 FLU

Pressure
pressure
1.000e+02
8.400e+01
6.800e+01
5.200e+01
3.600e+01
2.000e+01
4.000e+00
-1.200e+01
-2.800e+01
-4.400e+01
-6.000e+01
[Pa]

ANSYS 2019 R3 ACADEMIC

0 0.090 (m)
0.045

3D Viewer | Table Viewer | Chart Viewer | Comment Viewer | Report Viewer


C2 : Results - CFD-Post
File Edit Monitor Session Insert Tools Help
Location
Outline Variables Expressions Calculators Turbo
Cases
Case Comparison
FFF
FLU
User Locations and Plots
pressure
Default Transform
Default Legend View 1
Streamline 1
Wireframe
Report
Title Page
File Report
Mesh Report
Physics Report
Details of Streamline 1
Geometry Color Symbol Limits Render View
Type 3D Streamline
Definition
Domains All Domains
Start From velocity_inlet
Sampling Equally Spaced
# of Points 100
Preview Seed Points
Variable Velocity
Boundary Data Hybrid Conservative
Direction Forward
Cross Periodics
Apply Reset Defaults
Create Streamline
Name: Streamline 1
Domains: All Domains
Start from: velocity_inlet
# of Points: 100
Variable: Velocity
Apply
View 1 FFF
View 2 FLU
Velocity
Streamline 1
ANSYS
2019 R3
ACADEMIC
1.399e+01
1.050e+01
6.997e+00
3.498e+00
0.000e+00
[m s^-1]
0
0.090 (m)
0.045
3D Viewer Table Viewer Chart Viewer Comment Viewer Report Viewer


END