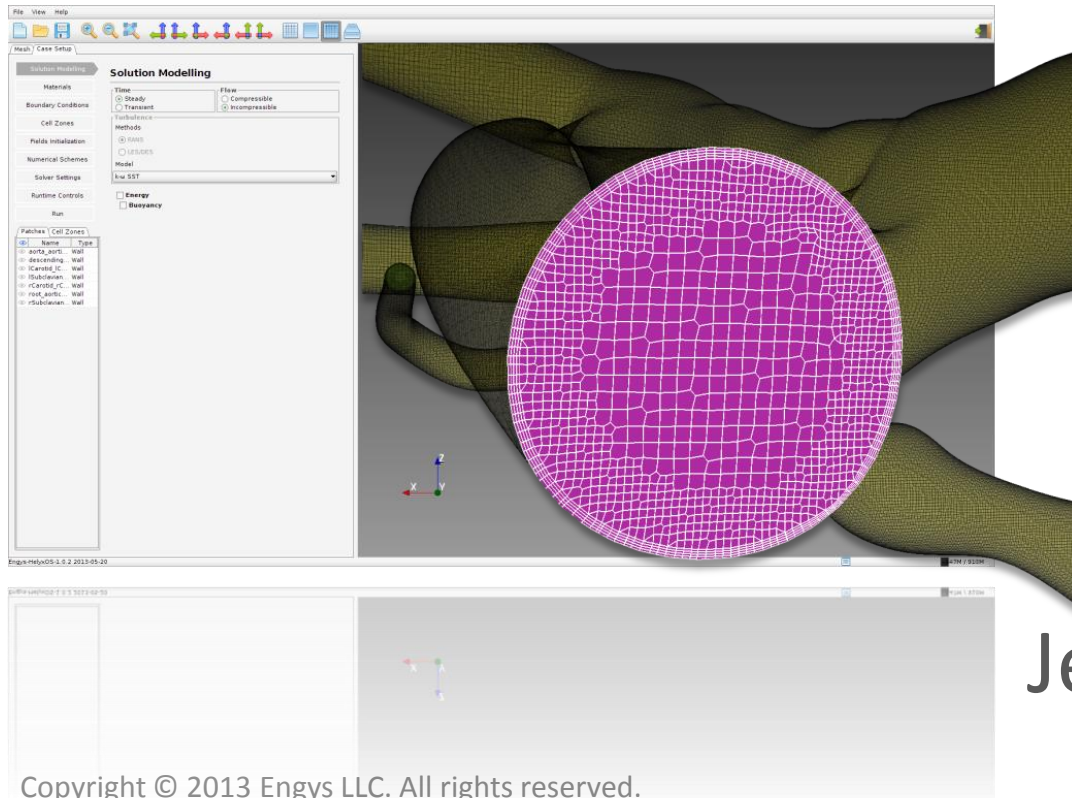# A Concise Introduction to Pre-Processing, Meshing, and Running OpenFOAM® Cases with HELYX-OS

Daniel P. Combest
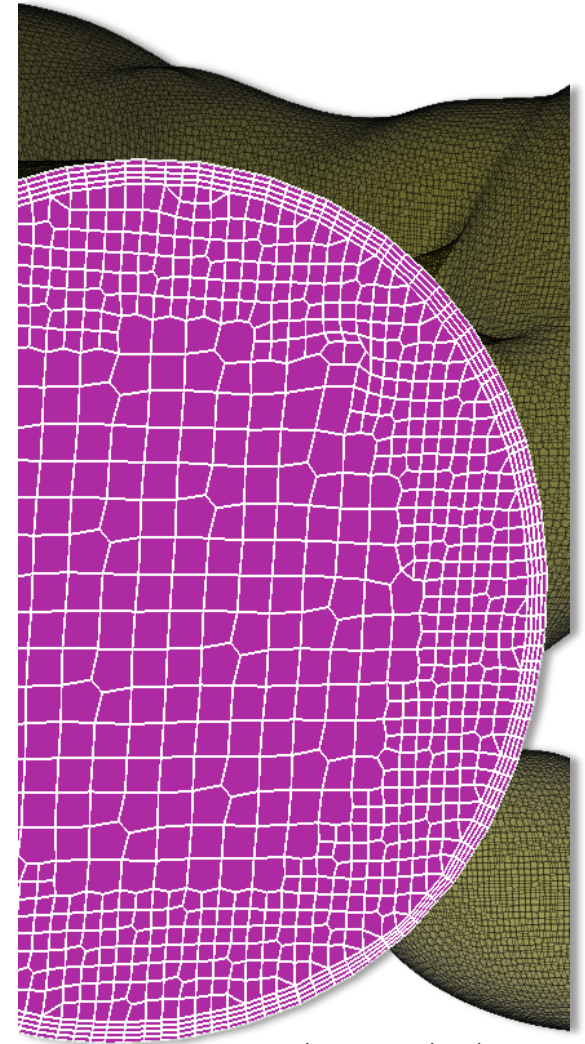
Paolo Geremia

8th OF Workshop,

Jeju Korea June 2013

Contents

- **`HELYX-OS`** (30 Minutes)
  - Capability Overview
  - Workflow Walkthrough
    - Meshing, Case Setup, and Execution
- Hands-On Examples (~1 hour)
  - Incompressible Flow
  - Turbulent Flow
- Closing Remarks (5 Minutes)

Mesh generated with HELYX™

engys™

Before we Begin

To take part in hands-on portion of training

- Must have HELYX-OS installed (Already on workshot virtual Machine)

- Download patch and training material from https://sourceforge.net/projects/helyx-os/files

    - Material includes presentations "Training_material.zip"

    - Install patch where directory "Engys/HelyxOS/v1.0.2" is located with

```
tar xvjf 20130610-Engys-HelyxOS-1.0.2-linux-x86_64-PATCH.tar.bz2
```

Enjoy!

engys

What is HELYX-OS?
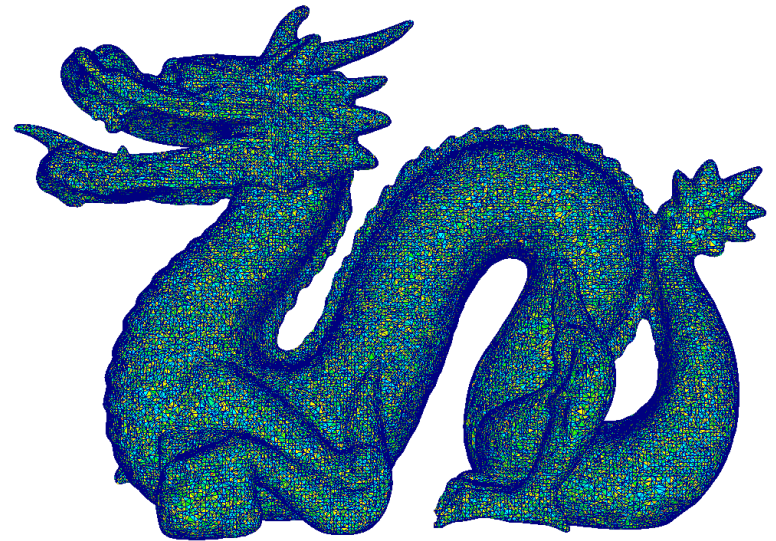
An Open Source <u>pre-processor</u> for:

Importing Geometries

Creating Meshes

Configuring Cases

Running Solvers

Geometries in stereolithography format (STL)

engys ™

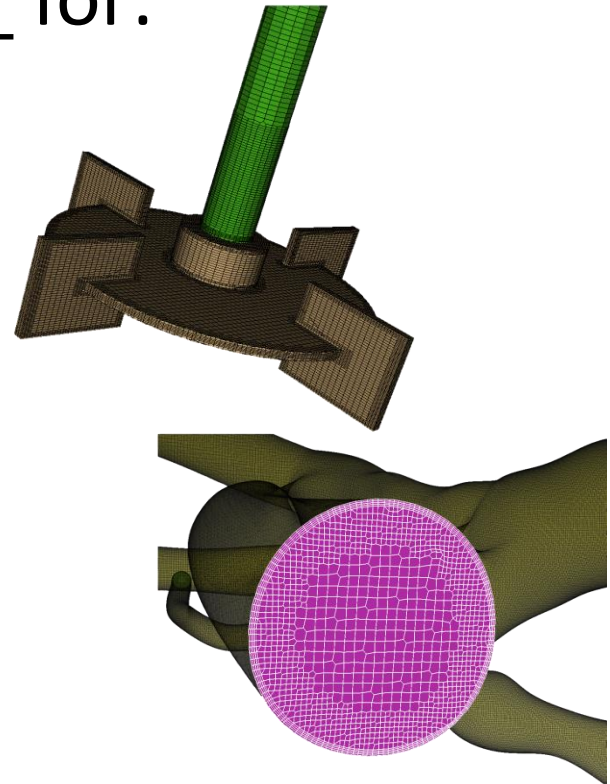# HELYX-OS | Capability Overview

What is HELYX-OS?

An Open Source <u>pre-processor</u> for:

Importing Geometries

Creating Meshes

Configuring Cases

Running Solvers

Using the `snappyHexMesh` utility

engys ™

HELYX-OS | Capability Overview

What is HELYX-OS?
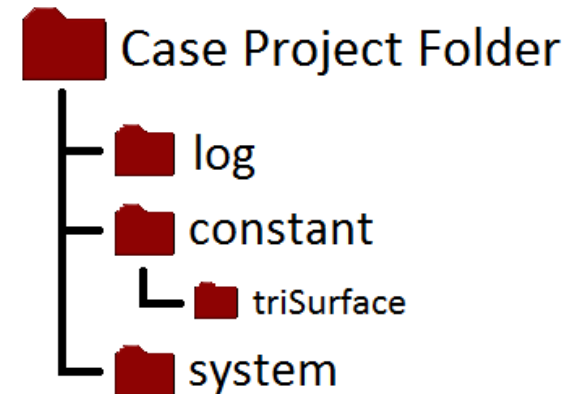
An Open Source <u>pre-processor</u> for:

Importing Geometries

Creating Meshes

Configuring Cases

Running Solvers

Setup file structure and dictionaries

Case Project Folder
- log
- constant
    - triSurface
- system

engys

What is HELYX-OS?
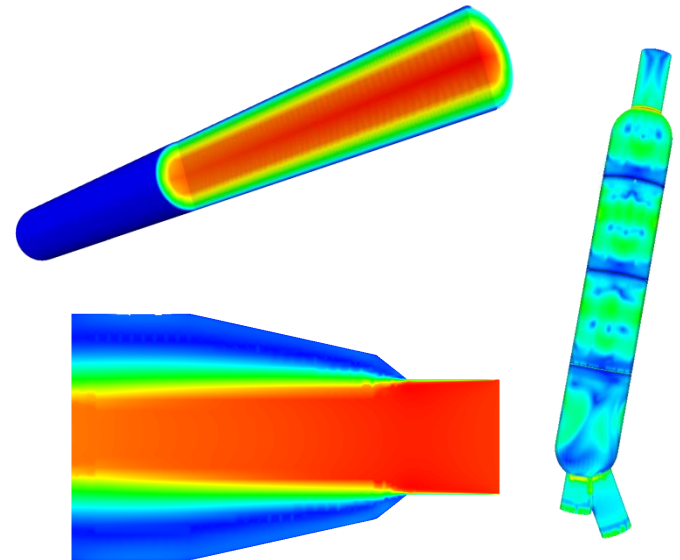
An Open Source <u>pre-processor</u> for:

Importing Geometries

Creating Meshes

Configuring Cases

Running Solvers

RANS, LES, Thermal, MRF, Porous…

engys™

HELYX-OS | Capability Overview

-Details and Compatibility-

- Written in Java, leveraging VTK

  ▪ Provided in compiled 64 bit versions

- Currently in v1.0.2

  ▪ Support for OpenFOAM® version 2.2.x

  ▪ Available for free download on Sourceforge

- Derived from HELYX™ and Related to ELEMENTS™

  ✓ Enhanced Solvers + BCs
  ✓ Enhanced Meshing
  ✓ More Functionality
  ✓ Full User Support
  ✓ Documentation

  ✓ Specifically for Automotive
  ✓ Meshing + simulation best practices
  ✓ Full Suite of tools
  ✓ Full User Support

`Solvers` supported for specific flow types

Incompressible	`simpleFoam` and `pimpleFoam` with fvOptions for MRF and porous flows, `pisoFoam`
Compressible	`rhoSimpleFoam` and `rhoPimpleFoam` with fvOptions for MRF and porous flows
Heat Transfer and Bouyancy Driven	`buoyantBoussinesqSimpleFoam`, `buoyantBoussinesqPimpleFoam`, and `buoyantSimpleFoam` or `buoyantPimpleFoam` with fvOptions for MRF and porous flows

engys™

HELYX-OS | Capability Overview - Physics

Reynolds Average Simulations

Incompressible

Standard ke

K-ω SST

Spalart-Allmaras

Realizable k-ε

RNG k-ε

Non-linear Shih k-ε

Lien Cubic k-ε

Launder-Sharma k-ε

Lam Bremhorst k-ε

Lien Cubic Low Re k-ε

Lien-Leschziner Low Re k-ε

qZeta

V2f

laminar

Compressible

Standard k-ε

K-ω SST

Spalart-Allmaras

Realizable k-ε

RNG k-epsilon

Launder-Sharma k-ε

laminar

LES and DES Simulations

Incompressible

Spalart-Allmaras (DES, DDES, IDES)

Smagorinsky (variants)

K-Equation (variants)

Compressible

Spalart-Allmaras (DES)

Smagorinsky (variants)
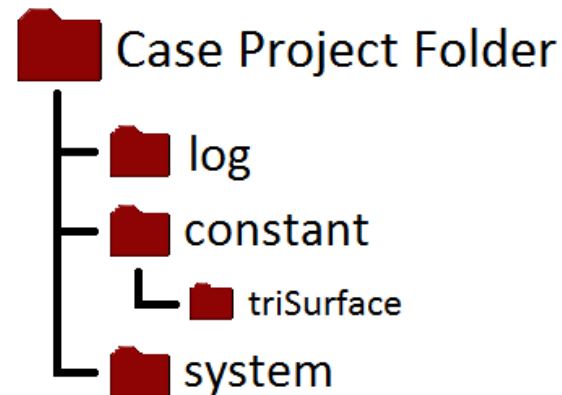
K-Equation (variants)

engys™

Contents

- **HELYX-OS** (30 Minutes)
  - Capability Overview
  - **Workflow Walkthrough**
    - Meshing, Case Setup, and Execution
- Hands-On Examples (~1 hour)
  - Incompressible Flow
  - Turbulent Flow
- Closing Remarks (5 Minutes)

engys

Create the Case

Meshing

- Parent folder definition and case directories are setup.

- Dictionaries are defined

Case Project Folder
- log
- constant
  - triSurface
- system

engys™

Create the Case

Meshing

Define a Base Mesh

- A blockMeshDict is created to provide a base mesh for `snappyHexMesh`

engys ™

HELYX-OS | Workflow Walkthrough

Meshing

Create the Case

Define a Base Mesh

Advanced Settings

- Geometries are imported

- Refinement surface and layer addition controls defined

- `snappyHexMesh` is setup and run

engys™

HELYX-OS | Workflow Walkthrough

Import the Mesh

Case Setup

- The mesh just created is imported and ready for setup

- Cannot go back to redo the mesh at this point

engys™

HELYX-OS | Workflow Walkthrough

Import the Mesh

Physics

Case Setup

- The state of the simulation is set
  - Turbulent (RANS or LES)
  - Energy
  - Steady-state or Transient

engys ™

HELYX-OS | Workflow Walkthrough

Import the Mesh

Physics

Boundary Conditions

Case Setup

- The boundary conditions are set for each patch
  - Predetermined list
  - Some are set by HELYX-OS

engys™

HELYX-OS | Workflow Walkthrough

Import the Mesh

Physics

Boundary Conditions

Numerical

Case Setup

- Divergence schemes are set
  - Time and gradient schemes are set by HELYX-OS
- Initial conditions are set

engys ™

HELYX-OS | Workflow Walkthrough

Import the Mesh

Physics

Boundary Conditions

Numerical

Control

Case Setup

- Solution control is set
  - Time steps
  - Write steps
  - Courant Number
  - Write precision

engys ™

HELYX-OS | Workflow Walkthrough

Import the Mesh

Physics

Boundary Conditions

Numerical

Control

Run

Case Setup

- The solver based on the physics selected is run automatically

engys

Contents

- **`HELYX-OS`** (30 Minutes)
  - Capability Overview
  - Workflow Walkthrough
    - Meshing, Case Setup, and Execution
- Hands-On Examples (~1 hour)
  - Incompressible Flow
  - Turbulent Flow
- Closing Remarks (5 Minutes)

engys
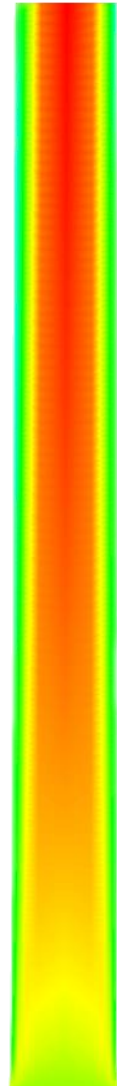
Hands-On Examples

Before you begin….

- Read from left to right, top to bottom
- Later cases leave out introductory material
- Meshes are left under-resolved and thus will poorly converge
- Cases are meant give you the ability to run simulations, but "inspire" you to pursue further
- We will go through the first tutorial and then you are left on your own to choose one of the other two for the duration.
- Text in **red** are meant to be "actions"
- Text in `Courier Font` are applications to be run in the terminal or names of applications

engys™

Hands-On Session | Incompressible Flow

Overall Goal

- To introduce basic HELYX-OS workflow

- Model incompressible laminar pipe flow

Skills Obtained

✓ Defining mesh primitives

✓ Configuring meshing parameters

✓ Defining properties and incompressible laminar physics

✓ Defining boundary conditions and solution control

✓ Meshing a geometry and executing a solver

engys™

Laminar Pipe Flow
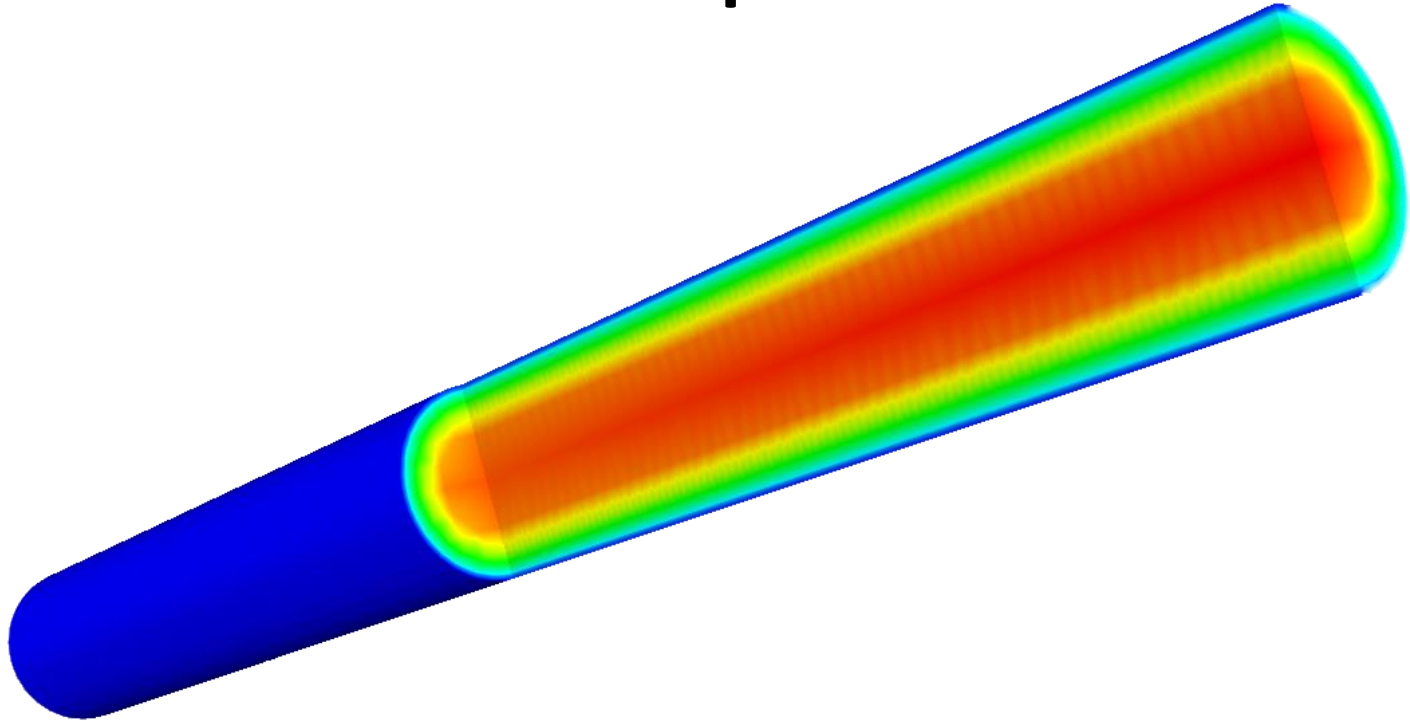
We want to create the geometry directly in HELYX-OS, mesh the tube and execute a solver

engys™

Hands-On Session | Incompressible Flow

1. **Open a terminal (control + alt + t)**
2. **Type `helyxOS.sh` in the shell command line**
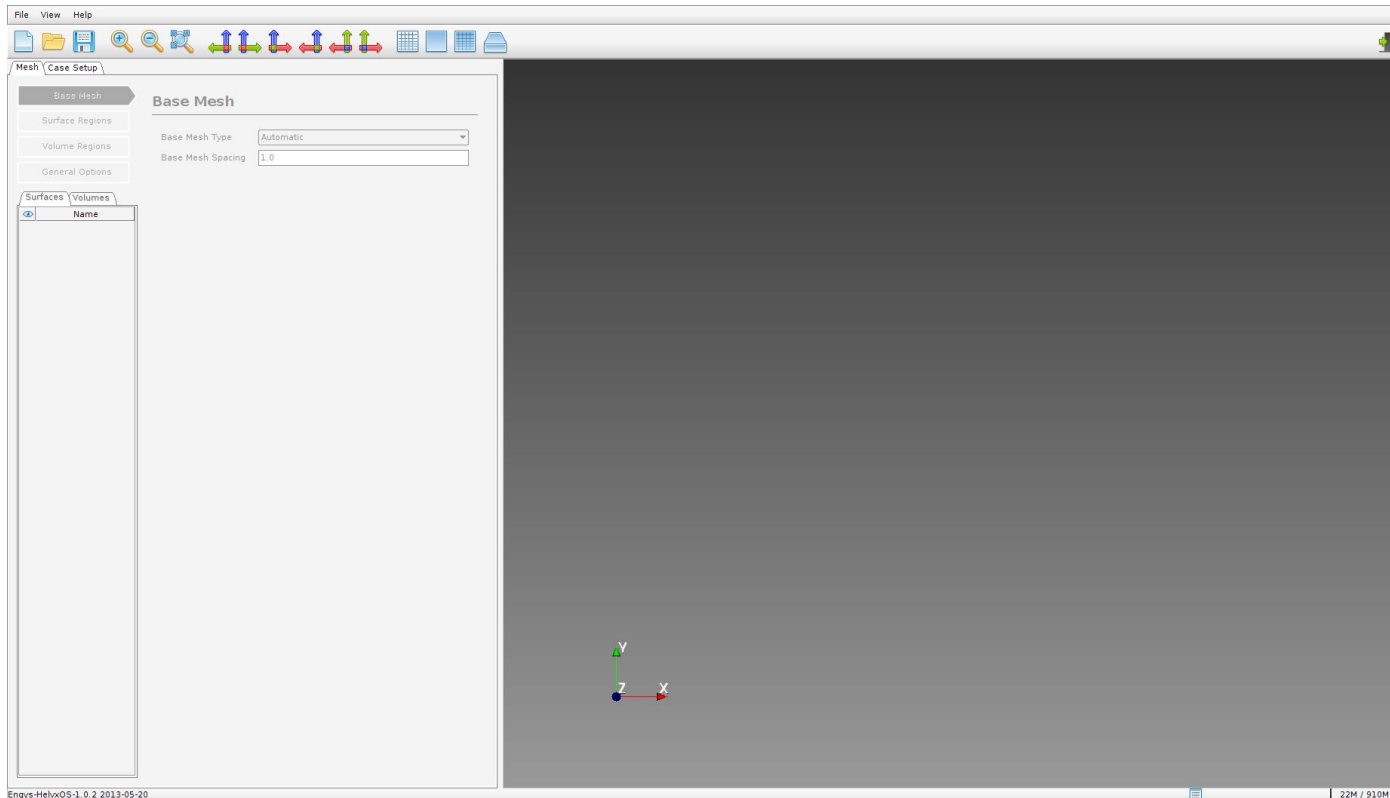
Hands-On Session │ Incompressible Flow
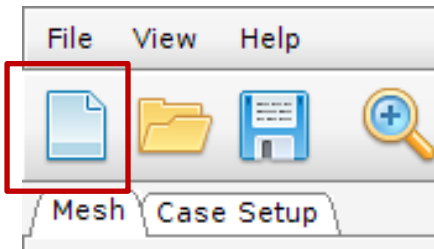
3. Click the "blank paper" icon to create a new case

Parallel Cases

Select the parent folder for the cases.

Possible to select "parallel", number of processors, and simple decomposition parameters

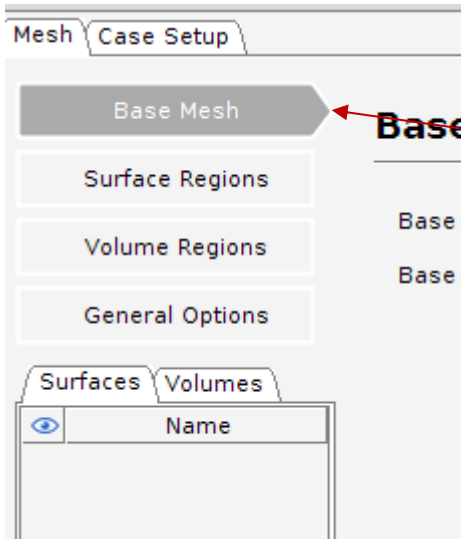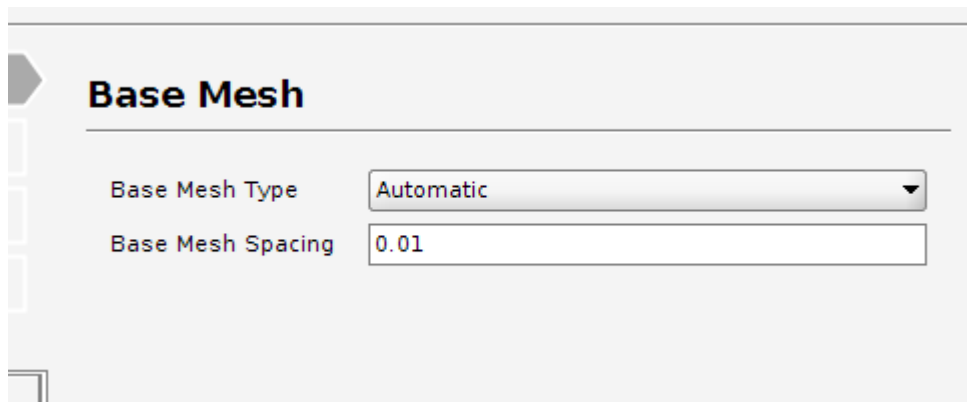**Serial Cases**

4. Call the case "pipeFlow" and select a Parent folder.

5. Leave the case as serial and Click "OK"

engys

Hands-On Session | Incompressible Flow

The "Base Mesh" tab is where we define the base mesh used by snappyHexMesh to create a geometry and is selected by default when a case is defined

By default, the method is set to automatic and HELYX-OS generates a blockMeshDict for you.

engys

Hands-On Session | Incompressible Flow

Base Mesh

Base Mesh Type	User Defined		
	X	Y	Z
Min	-0.2	0.25	-0.2
Max	0.2	2.25	0.2
Number of Elements	10	100	10

You may also define a base mesh bounding box with intervals in the x, y, and z-direction.

6. Select "User Defined" and enter these setting for this case

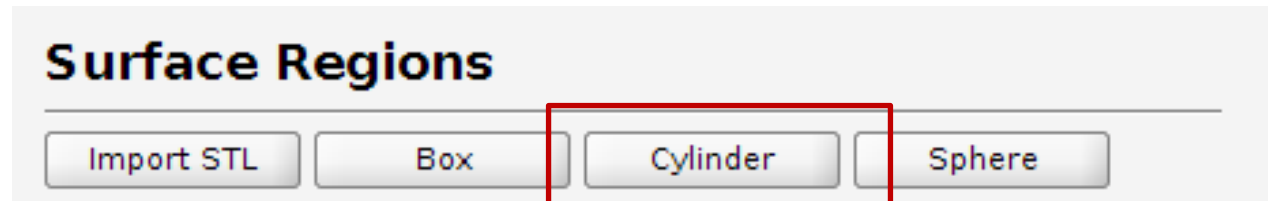- Ideally, create a mesh where cells are perfect cubes i.e.

$$\frac{|maxX - minX|}{n_x} = \frac{|maxY - minY|}{n_y} = \frac{|maxZ - minZ|}{n_z}$$

- snappyHexMesh will yield better meshes

engys

Hands-On Session | Incompressible Flow

Base Mesh

Surface Regions

Volume Regions

General Options

7. Select the "Surface Regions" tab

Surface Regions

| Import STL | Box | Cylinder | Sphere |

We are interested in creating a pipe geometry and internal mesh. **8. Select the "cylinder" primitive shape**
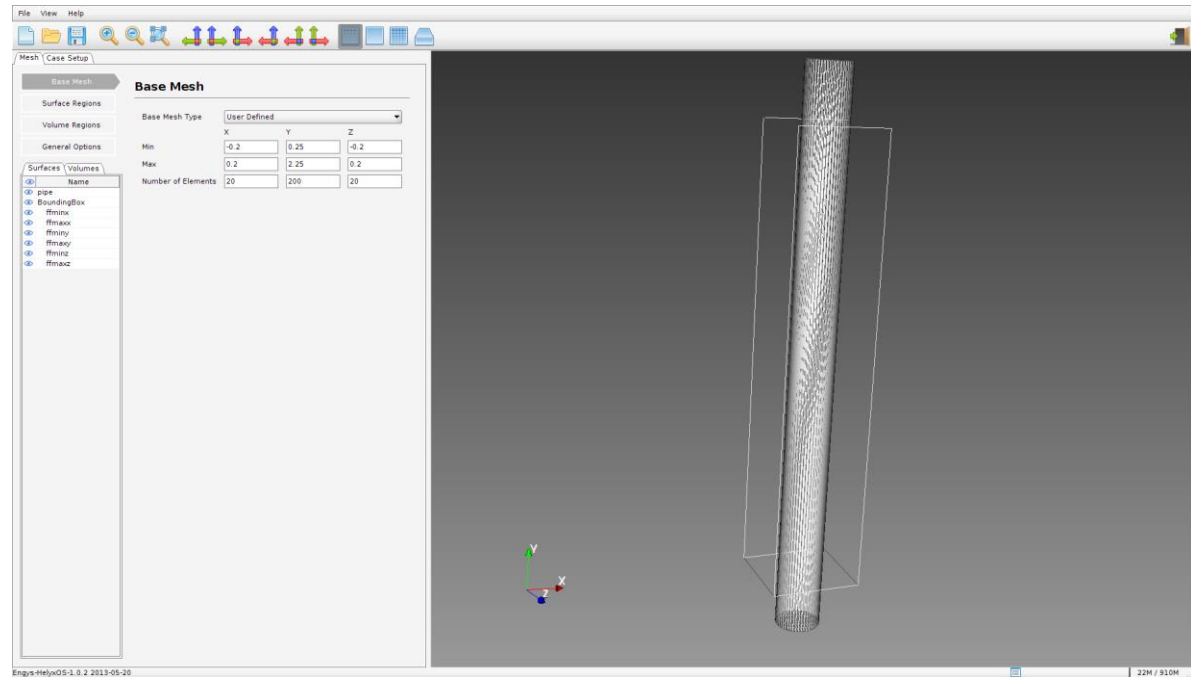
Cylinder

Cylinder Name	pipe		
	X	Y	Z
Point 1	0.0	0.0	0.0
Point 2	0.0	2.5	0.0
Radius	0.1		

9. Define the pipe geometry with a length of 2.5 m in the y-direction and a radius of 0.1 m
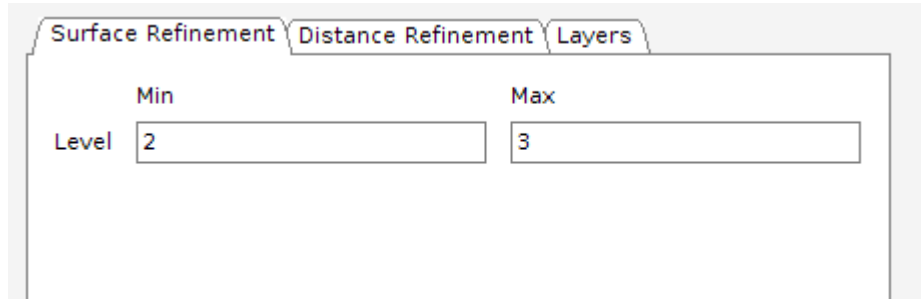
engys

Hands-On Session │ Incompressible Flow

- Both surface geometry and base mesh box should appear
- We "cut off" the geometry so snappy creates multiple BCs
- Allows use of primitive shapes

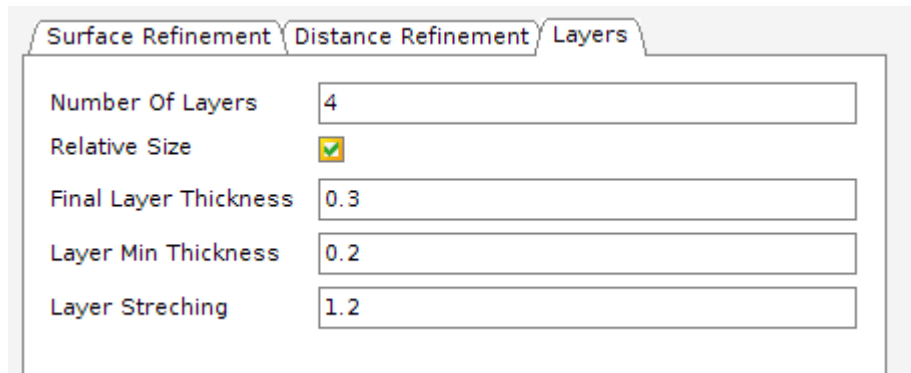**engys** ™

Hands-On Session | Incompressible Flow

Surface Refinement | Distance Refinement | Layers

	Min	Max
Level	2	3

Below the "Surface Regions" definition for the cylinder, is the "surface refinement" tab. This defines how much refinement is performed with respect to the base mesh and "feature angle". A higher number results in a finer mesh around a feature.

10. Enter 2 and 3 for min and max

11. Select the "Layers" tab

Surface Refinement | Distance Refinement | Layers

Number Of Layers	4
Relative Size	☑
Final Layer Thickness	0.3
Layer Min Thickness	0.2
Layer Streching	1.2

This will allow us to define the number of surface layers to add and some characteristics of the layers.

12. Enter exactly what is shown on the left
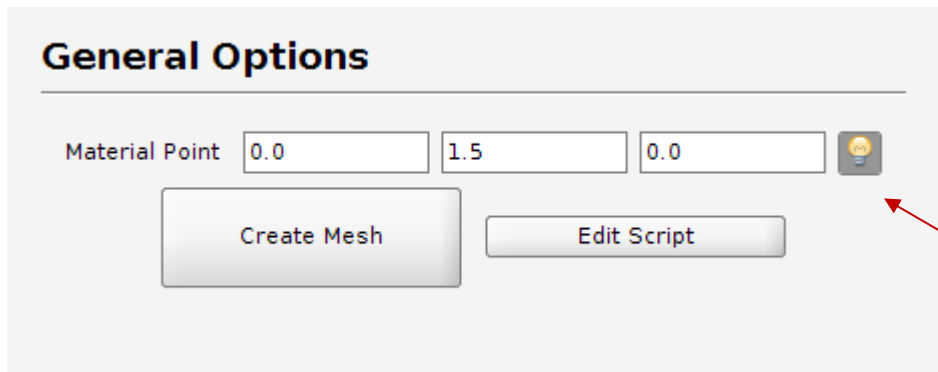
engys

Hands-On Session | Incompressible Flow

Base Mesh

Surface Regions

Volume Regions

General Options

The "Volume Regions" tab is where we can define refinement regions and cellZones

13. Nothing to do here

Base Mesh

Surface Regions

Volume Regions

General Options

14. Select the "General Options" tab

General Options

Material Point [0.0] [1.5] [0.0]
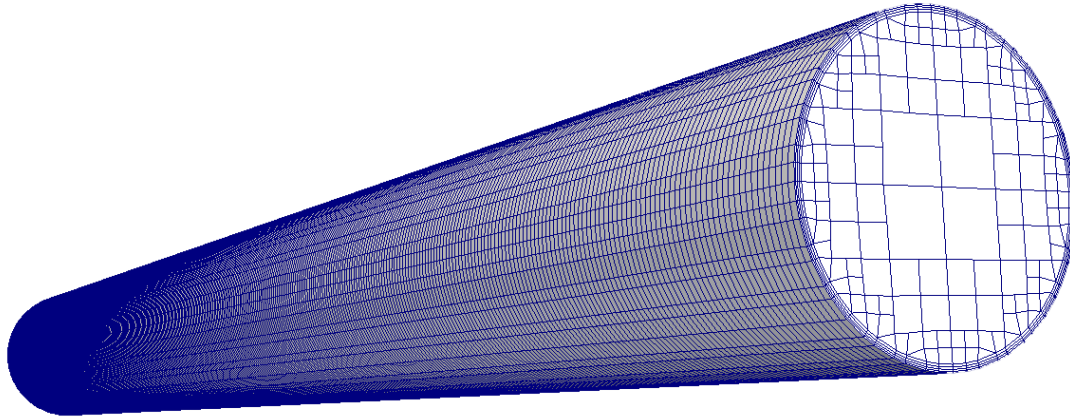
Create Mesh   Edit Script

We must define a point where a mesh cell will exist. To do this, we need to select a point in space inside the pipe.

Use the light bulb icon to visualize the current point

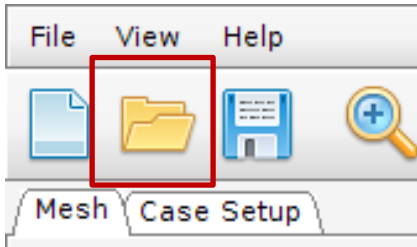**15. Enter (0, 1.5, 0) and hit "Create Mesh"**

engys

Hands-On Session | Incompressible Flow

- At this point, **HELYX-OS** will execute `blockMesh` and then `snappyHexMesh` based on values we entered, plus default values set by HELYX-OS.

- If we were to visualize this mesh, it would look similar to the one pictured.

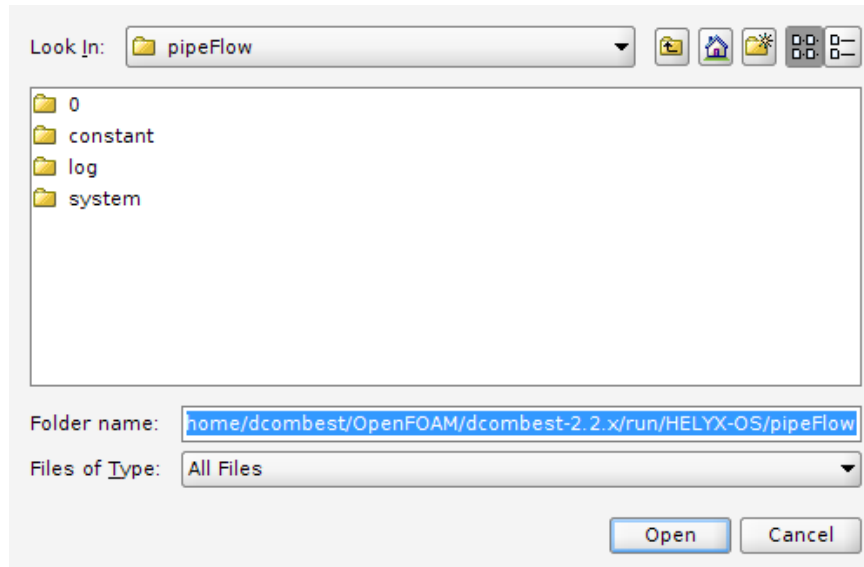- **Nice boundary layers cells!**

engys
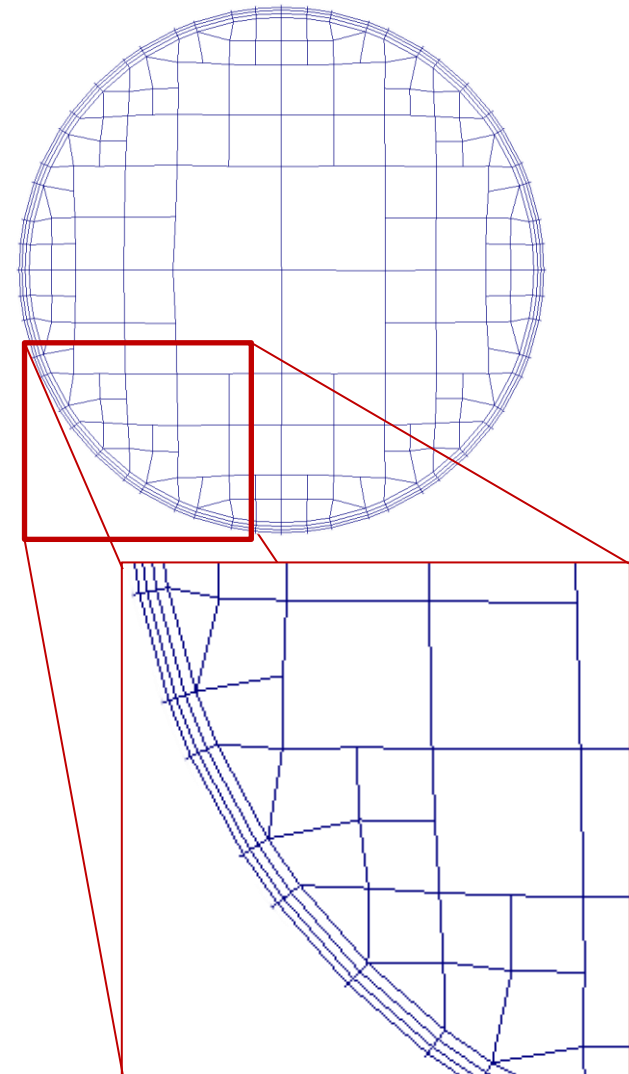
Hands-On Session | Incompressible Flow

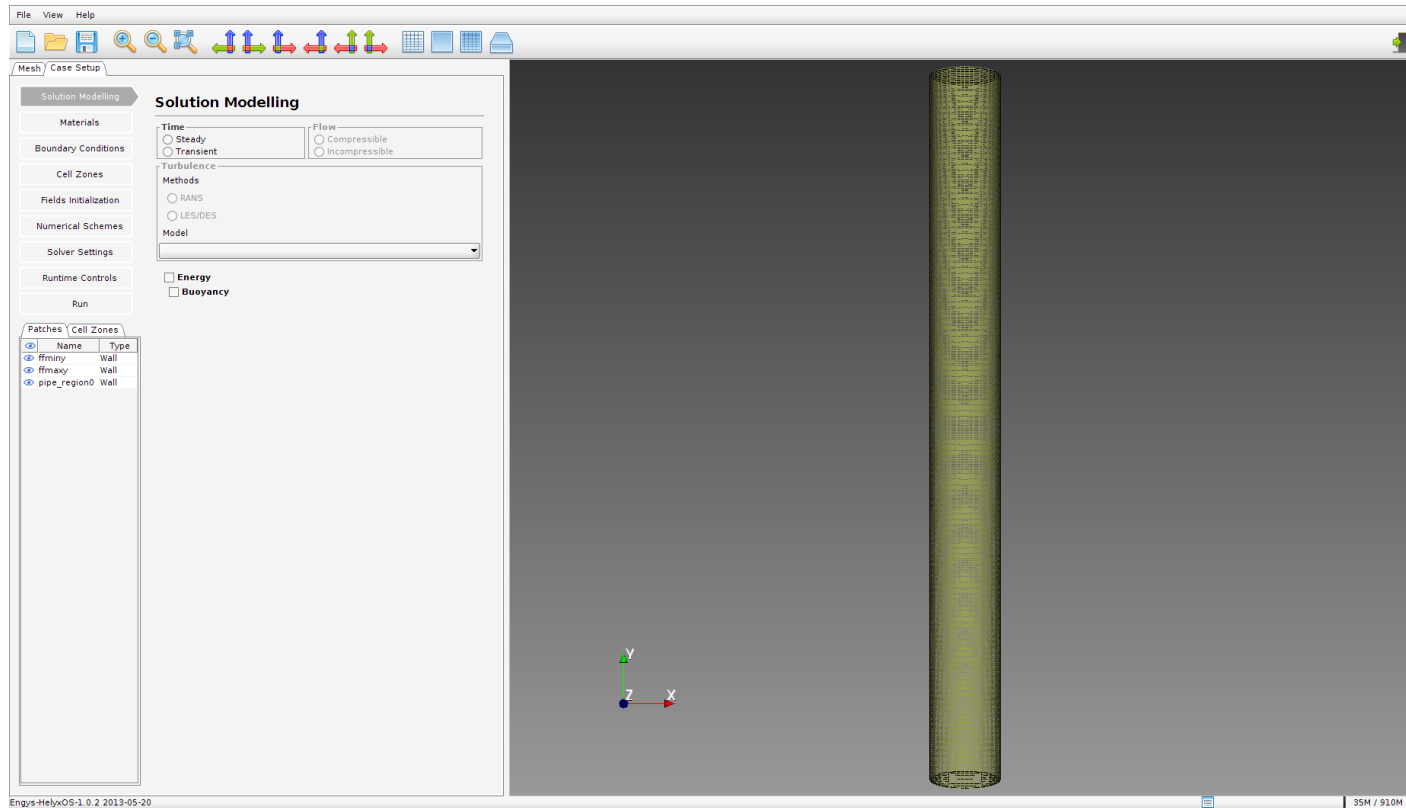We need to load our meshed case back into HELYX-OS to setup our physics and run a simulation

16. Click the "Folder" icon to open a case

The most recent case defined/run in HELYX-OS will be presented. A user can navigate to another case if desired.

17. Select "Open"

engys ™
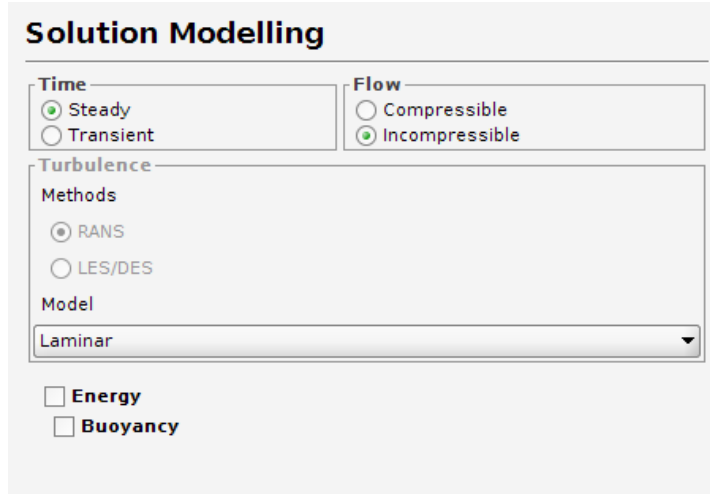
Hands-On Session | Incompressible Flow

- The meshed case should be loaded at this point
- The "mesh tab" is now greyed out and the user may **<u>not</u>** go back and mesh

engys

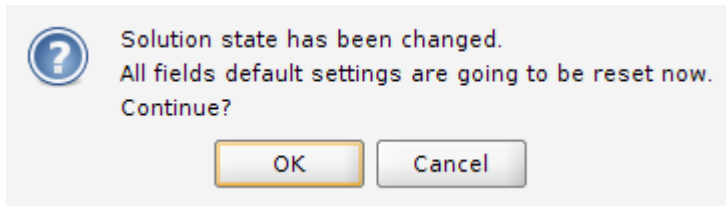# Hands-On Session | Incompressible Flow

By default the "Solution Modeling" tab is selected.

Solution modeling lets the user select the "Solution State" through selecting time frame, flow type, turbulence parameters, and additional physics

18. Select steady, incompressible, and laminar for this case

When a state has changed, a warning is issued

19a. Select "OK"

Hands-On Session | Incompressible Flow

Solution Modelling

Materials

Boundary Conditions

Cell Zones

Fields Initialization

Numerical Schemes

Solver Settings

Runtime Controls

Run

The materials tab allows for physiochemical property definitions

19. Select the "materials" tab

Materials

Database

Material Parameters

Name	air
Density [kg/m³]	1.205
Dynamic Viscosity [Pa·s]	0.000019137
Kinematic Viscosity [m²/s]	0.0000158813
Specific Heat Capacity [J/kg·K]	1,006.0
Laminar Prandtl Number	0.9
Turbulent Prandtl Number	0.85
Thermal Conductivity [W/m·K]	0.024
Reference (absolute) Pressure [Pa]	101,325.0
Thermal Expansion Coefficient [K⁻¹]	0.00333
Reference Temperature [K]	300.0

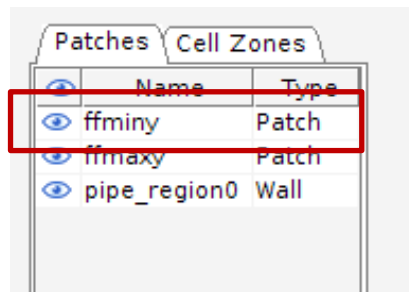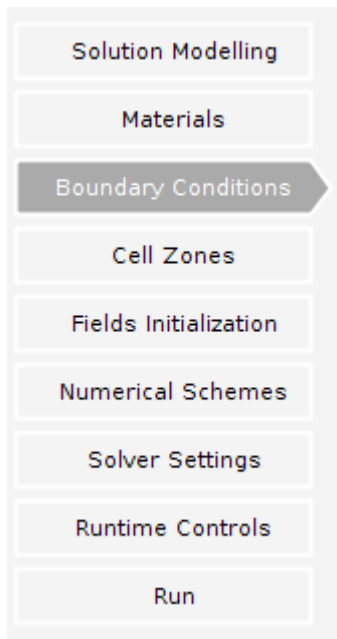Air is the default material

20. Select the "database" button

Default Library

water
liquidWater
waterVapor
airSteamMixture
air
mercury
oil

User Library

Material Parameters

Name	water
Density [kg/m³]	998.2
Dynamic Viscosity [Pa·s]	0.001002
Kinematic Viscosity [m²/s]	0.0000010038
Specific Heat Capacity [J/kg·K]	4,187.0
Laminar Prandtl Number	0.9
Turbulent Prandtl Number	0.9
Thermal Conductivity [W/m·K]	0.5985
Reference (absolute) Pressure [Pa]	0.0
Thermal Expansion Coefficient [K⁻¹]	0.0207
Reference Temperature [K]	293.0

New
Save
Copy
Remove

OK   Cancel

We can define a new Newtonian fluid in this database or select an existing component

21. Select "water" from the list and select "OK"

engys

Hands-On Session | Incompressible Flow

22. Selecting the "Boundary Conditions" tab

Boundary Conditions

Name: ffminy

Type: Patch

Momentum | Turbulence | Thermal

Velocity

Type	Fixed Value

Velocity (m/s): 0.0 | 0.0040152 | 0.0

Pressure

Type	Zero Gradient

- Solution Modelling
- Materials
- **Boundary Conditions**
- Cell Zones
- Fields Initialization
- Numerical Schemes
- Solver Settings
- Runtime Controls
- Run

Patches | Cell Zones

Name	Type
ffminy	Patch
ffmaxy	Patch
pipe_region0	Wall

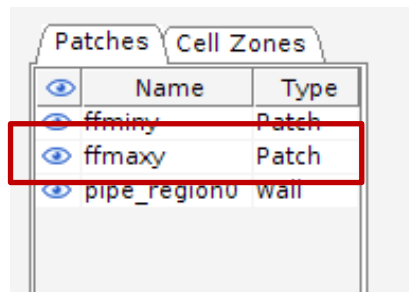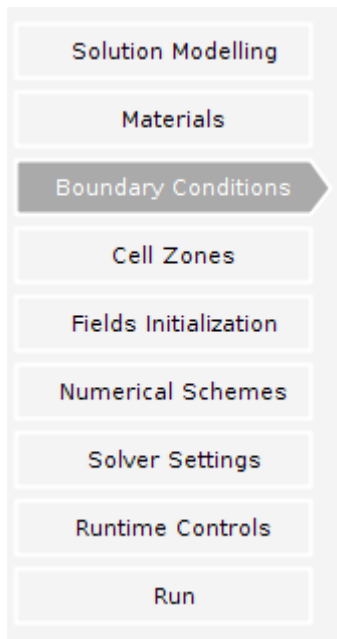**23. Select our <u>inlet</u> patch "ffminy"**

We can define the patch type and boundary condition type for our variables

24. Select type "patch"
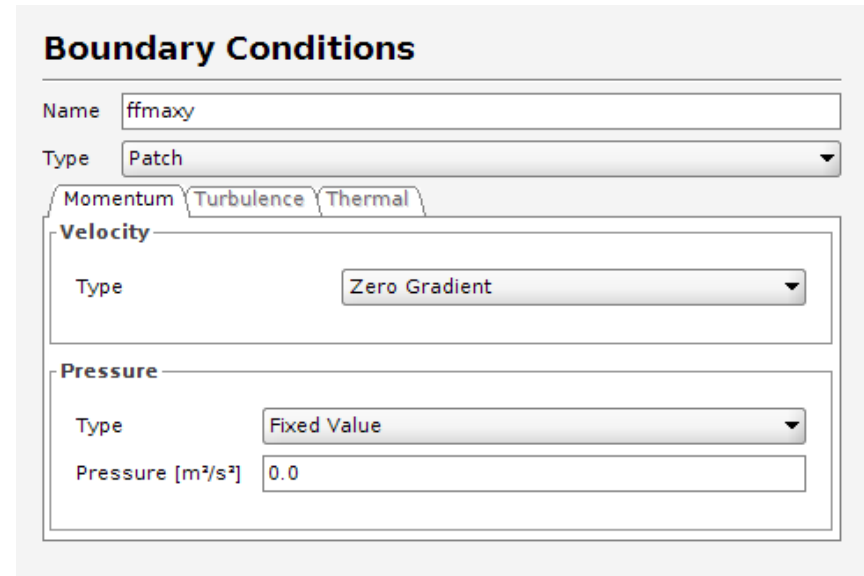**25. For velocity, make our inlet a "fixedValue" boundary condition and set it to (0,0.0040152, 0)**
26. Set pressure to "zeroGradient"

engys ™

Hands-On Session | Incompressible Flow

Boundary Conditions

Name ffmaxy

Type Patch

Momentum | Turbulence | Thermal

Velocity

Type — Zero Gradient

Pressure

Type — Fixed Value

Pressure [m²/s²] — 0.0

Solution Modelling

Materials

Boundary Conditions

Cell Zones

Fields Initialization

Numerical Schemes

Solver Settings

Runtime Controls

Run

Patches \ Cell Zones

Name	Type
ffminy	Patch
ffmaxy	Patch
pipe_region0	Wall

---

27. Select our <u>outlet</u> patch "ffmaxy"
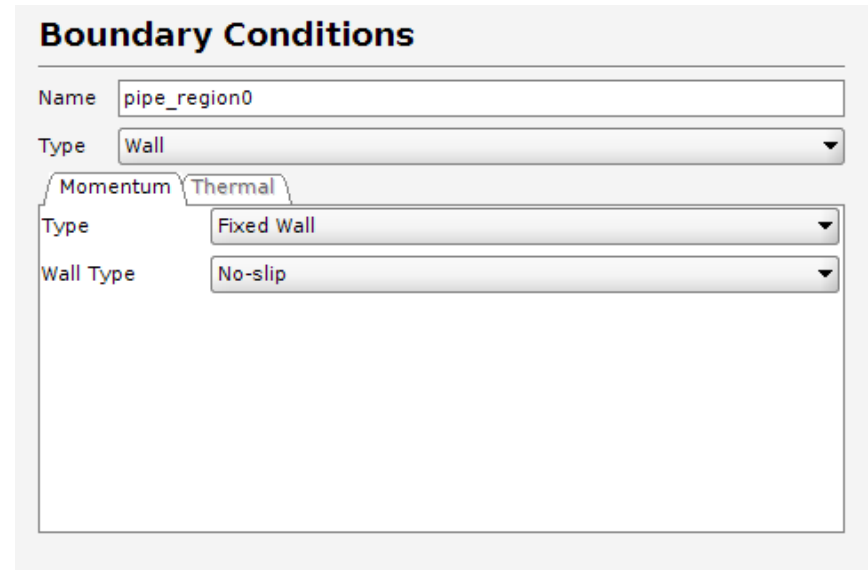
28. Select type "patch"

29. For velocity, make our outlet a "zero Gradient"

30. Set pressure to "fixed Value" with a value of 0

engys

Hands-On Session | Incompressible Flow

Boundary Conditions

Name: pipe_region0

Type: Wall

Momentum | Thermal

Type: Fixed Wall

Wall Type: No-slip

Patches | Cell Zones

Name	Type
ffminy	Patch
ffmaxy	Patch
pipe_region0	Wall

Solution Modelling

Materials

Boundary Conditions

Cell Zones

Fields Initialization

Numerical Schemes

Solver Settings

Runtime Controls

Run
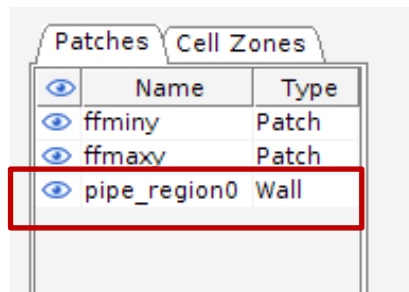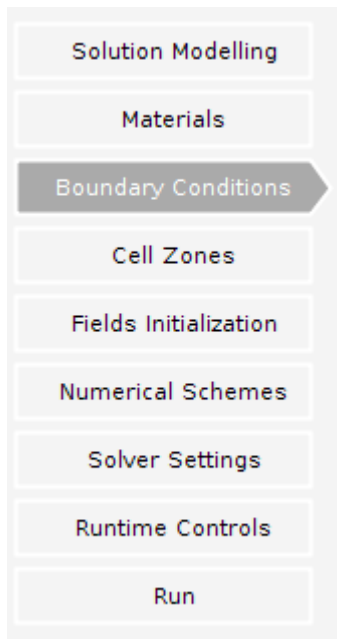
31. Select our <u>wall</u> patch "pipe_*"

32. Select type "patch"

33. For velocity, set the boundary condition type to "fixed wall"

34. Set wall type to "no-slip"

The remaining pressure BC will be set automatically to zeroGradient

engys

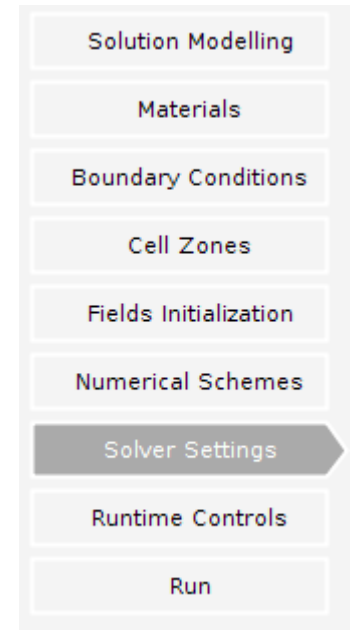# Hands-On Session | Incompressible Flow

Solution Modelling
Materials
Boundary Conditions
Cell Zones
Fields Initialization
Numerical Schemes
Solver Settings
Runtime Controls
Run

Used to define MRF or Porous Zones

Solution Modelling
Materials
Boundary Conditions
Cell Zones
Fields Initialization
Numerical Schemes
Solver Settings
Runtime Controls
Run
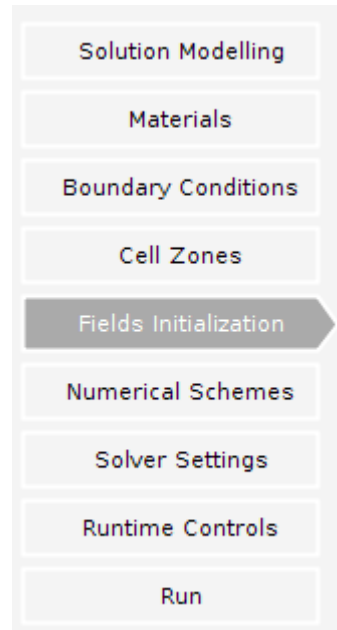
Used to set constant initial values for fields

Solution Modelling
Materials
Boundary Conditions
Cell Zones
Fields Initialization
Numerical Schemes
Solver Settings
Runtime Controls
Run
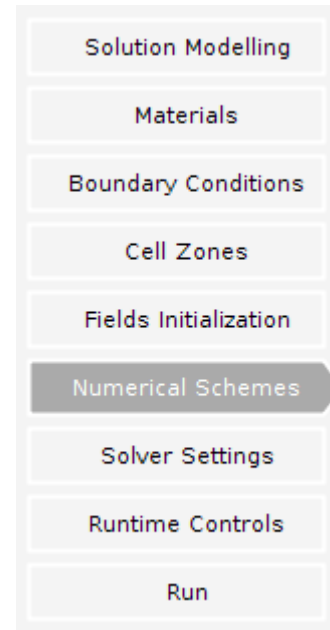
Used to set divergence discretization schemes

Solution Modelling
Materials
Boundary Conditions
Cell Zones
Fields Initialization
Numerical Schemes
Solver Settings
Runtime Controls
Run

Used to set non-orthogonal correctors and Laplacian settings

For this case, we will not adjust these defaults

engys ™

Hands-On Session | Incompressible Flow

Solution Modelling

Materials

Boundary Conditions

Cell Zones

Fields Initialization

Numerical Schemes

Solver Settings

Runtime Controls

Run

We can set the simulation duration, write steps, and other pertinent simulation settings

35. Select the "runtime controls" tab

Runtime Controls

Time Settings

Start From	Start Time	0.0
End Time	250.0	
$\Delta t(s)$	1.0	
Adjustable Time Step	☐	
Max Courant Number		
Max Time Step		

Data Writing

Write Control	Time Step	250.0
Purge Write	0	
Write Format	ASCII	
Write Precision	10	
Write Compression	Uncompressed	
Time Format	General	
Time Precision	6	
Graph Format	Raw	

Data Reading

Runtime Modifiable ☑
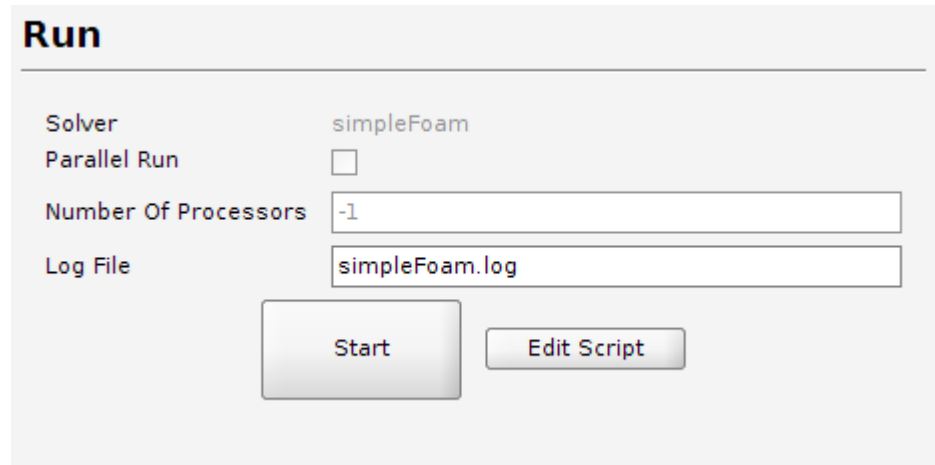
36. Set variables identical to values to the left

engys

Hands-On Session | Incompressible Flow

Solution Modelling

Materials

Boundary Conditions

Cell Zones

Fields Initialization

Numerical Schemes

Solver Settings

Runtime Controls

Run

The last step is to start the simulation

37. Select the "run" tab

Run

Solver	simpleFoam
Parallel Run	☐
Number Of Processors	-1
Log File	simpleFoam.log

Start   Edit Script

HELYX-OS will execute a run script that will execute the solver `simpleFoam`. The script may be edited using the "edit script" button, but we will not adjust anything

38. Select the "run" button
39. Open a new terminal and navigate to the case directory

engys

Hands-On Session | Incompressible Flow

HELYX-OS Has...

- ✓ Assembled file structure, dictionaries, and properties needed by OpenFOAM
  - controlDict, fvSchemes, snappyHexMeshDict, fvSolution, blockMeshDict, etc.
- ✓ Created a mesh for our pipe using **snappyHexMesh**
  - Mesh is located in polyMesh
- ✓ Allowed the user to setup an incompressible solver for a steady-state laminar flow (**simpleFoam**)
- ✓ Executed the **simpleFoam** solver for a predetermined amount of time
- ✓ When the simulation has stopped, we can post-process in Paraview

Hands-On Session | Incompressible Flow

- **If** you ran in parallel, in the case directory execute

  <code style="color:red">**reconstructParMesh –constant**</code>

- In the case directory, launch Paraview with

  <code style="color:red">**paraFoam**</code>

engys
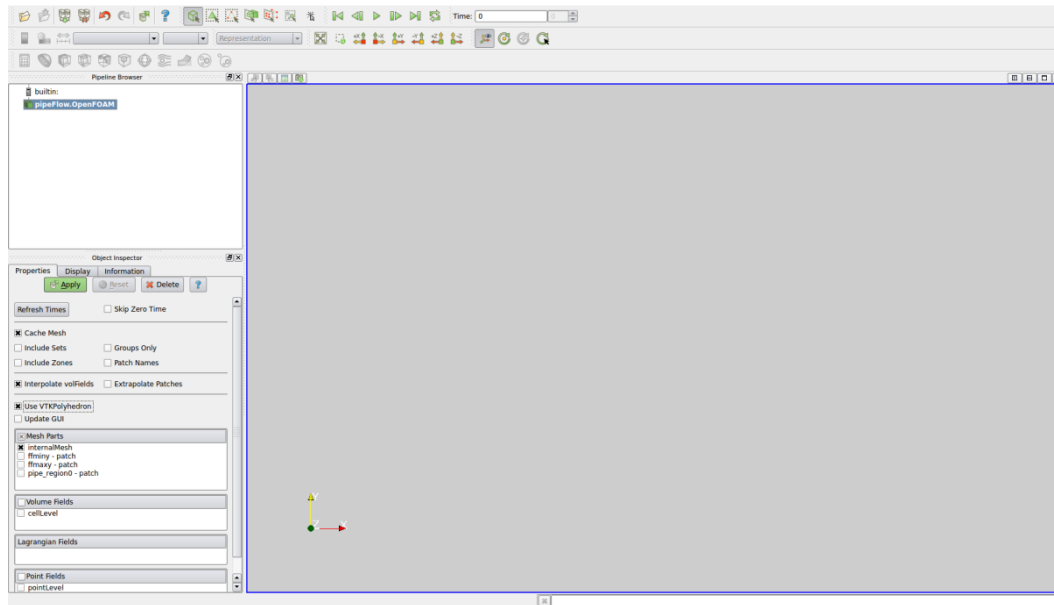
Hands-On Session | Incompressible Flow
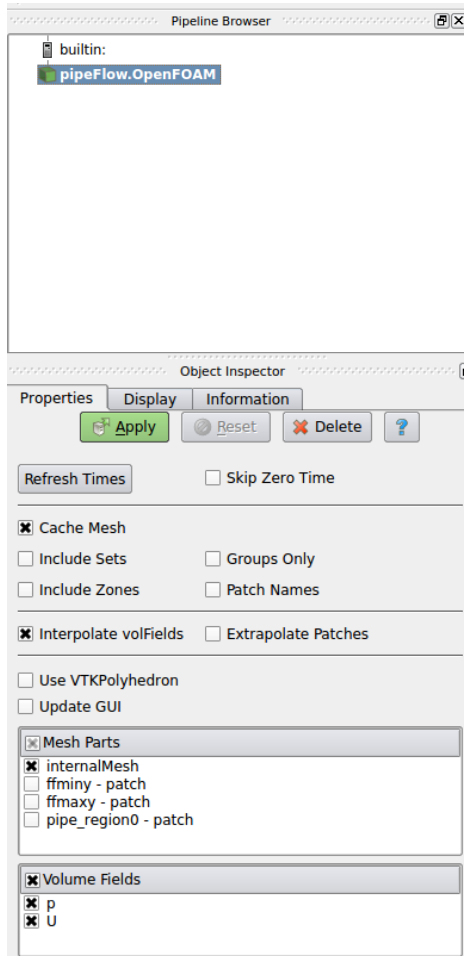
40. In the pipeline browser, make sure to have the object highlighted

41. In the object inspector, make sure select
- **internal mesh**
- **Volume Fields**

42. Hit "apply"

engys
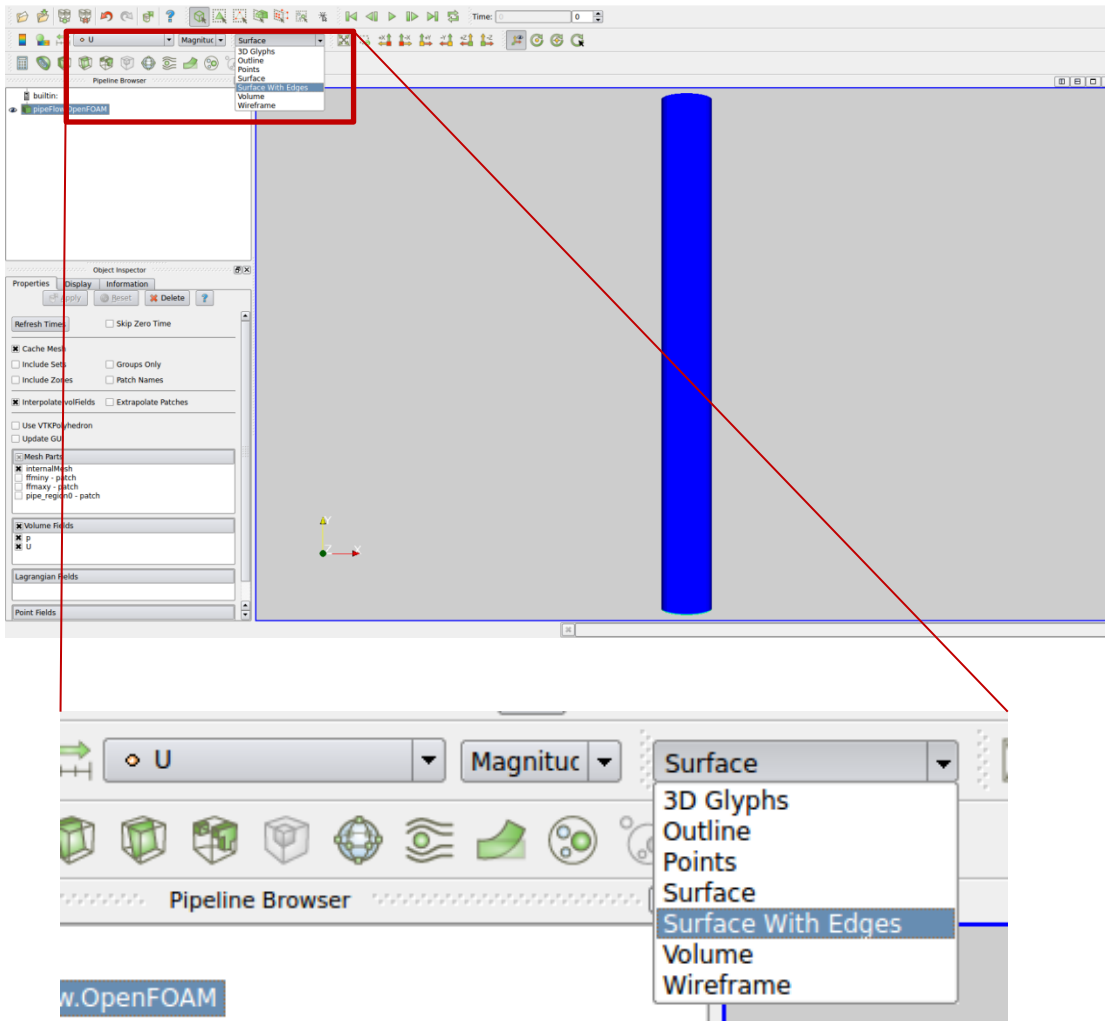
43. Select "Surface" to see a surface field or "Surface With Edges" to see the mesh
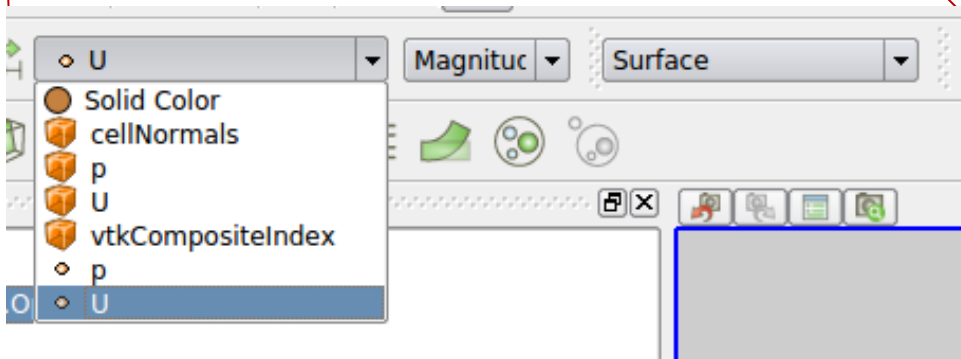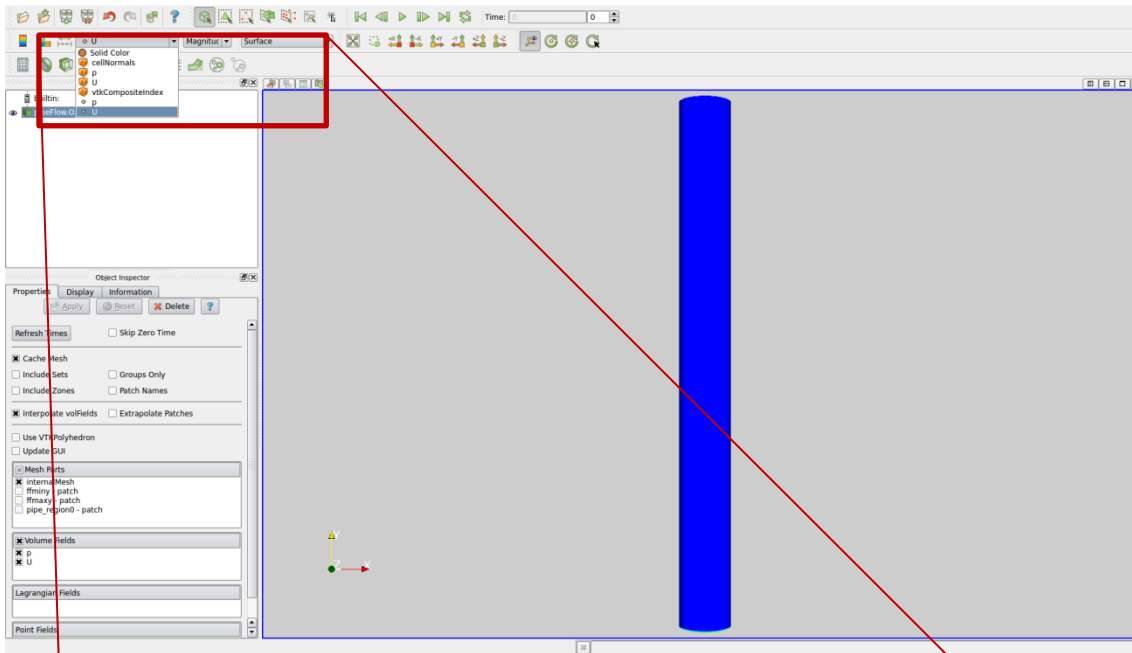
44. Select variable "U" to see a surface field of velocity

engys

Hands-On Session | Incompressible Flow
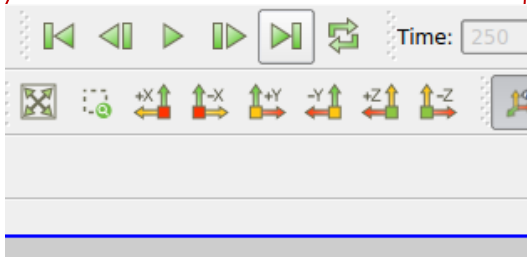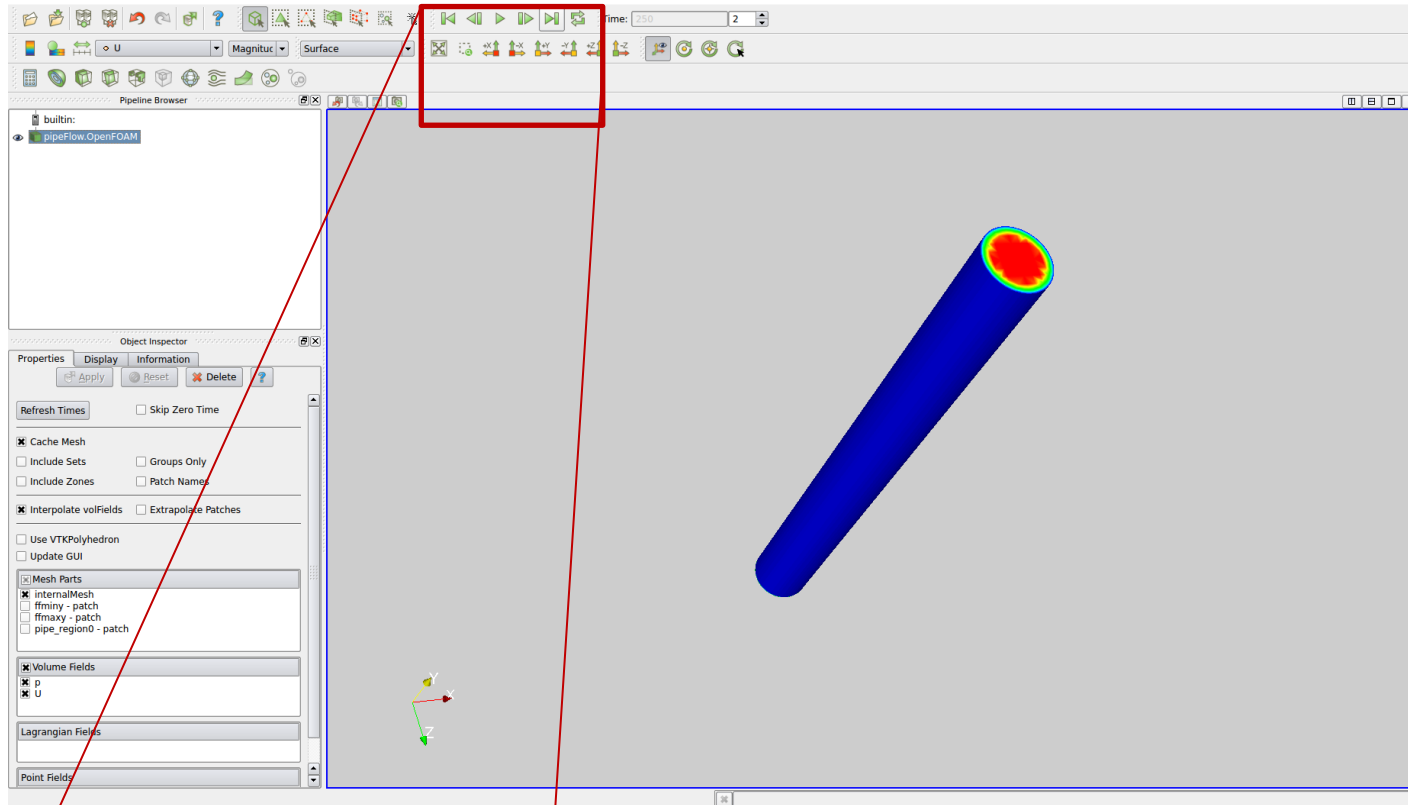
45. fast-forward to the last time step (250)

engys™

Hands-On Session | Incompressible Flow
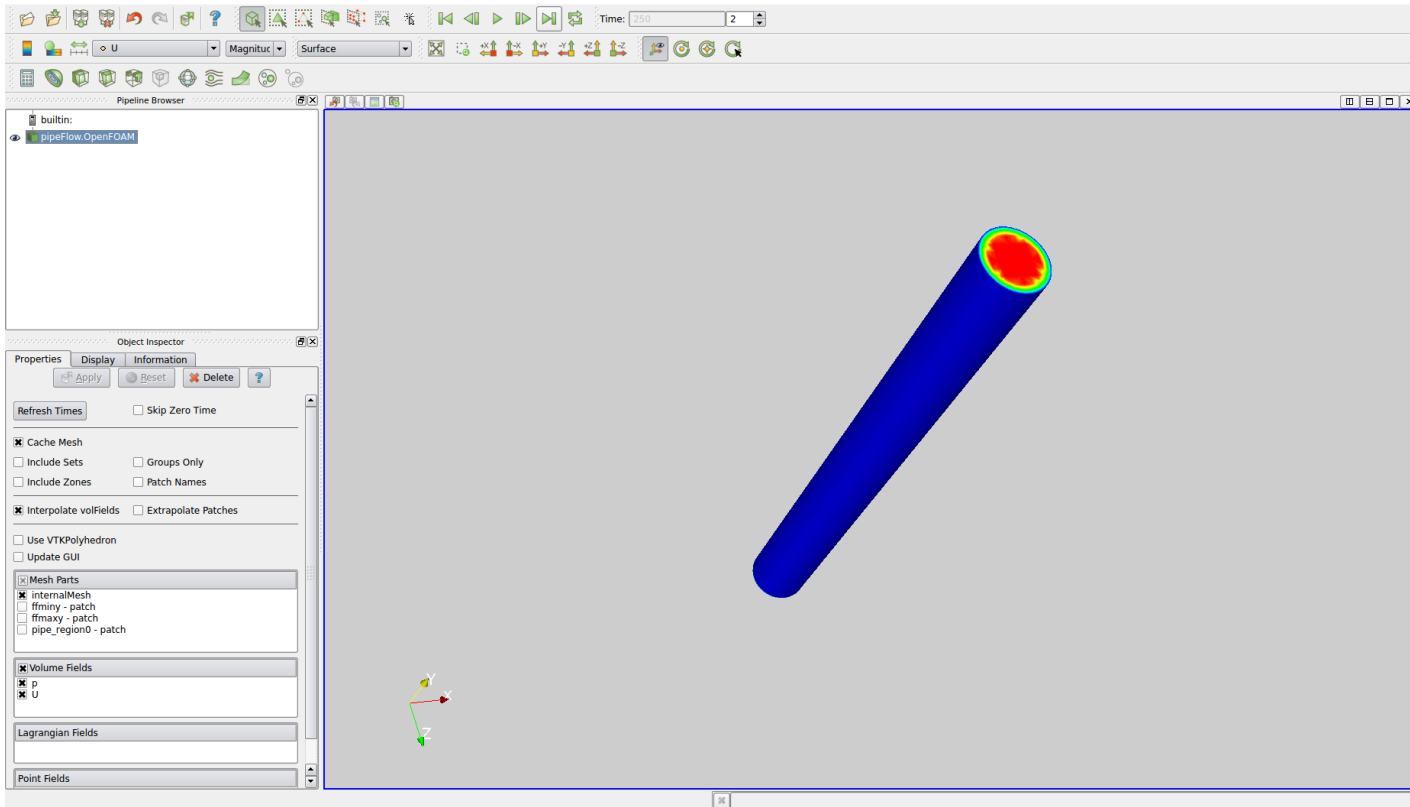
Hold left mouse button and move mouse to rotate geometry
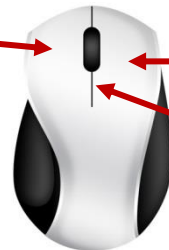
Hold right mouse button and move mouse to zoom or roll center wheel

http://www.psdgraphics.com/psd/computer-mouse-icon-psd/

engys

Hands-On Session | Incompressible Flow
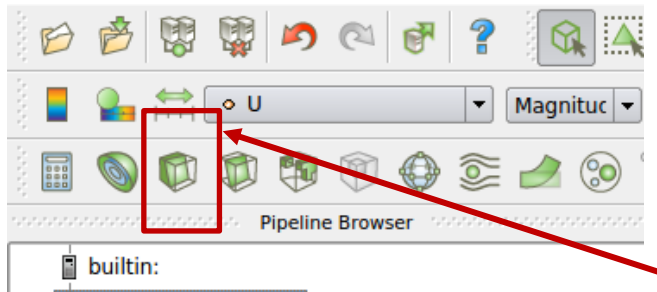
Paraview uses filters to aid in the post-processing of data from our OpenFOAM simulation

46. Select the "Clip" filter
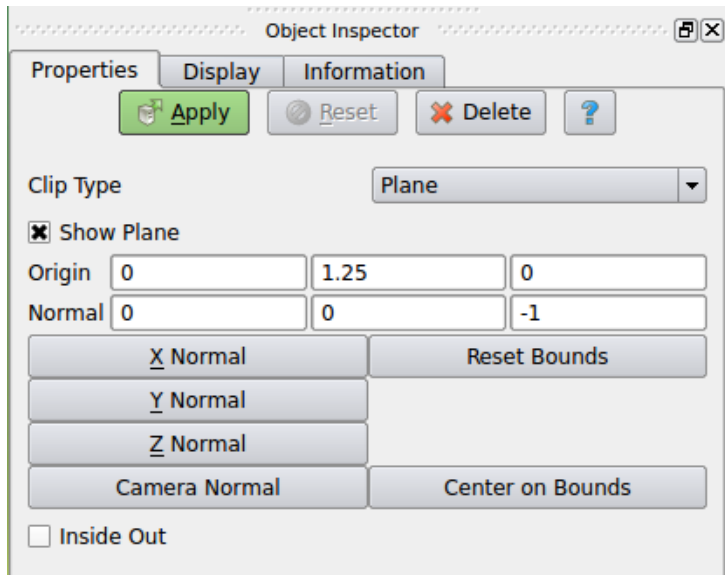
Create a clip such that half of the domain is removed so the interior of the domain can be viewed

47. Enter in the values from below into the Clip object

engys™

- Oscillations due to use of coarse mesh, but general trend is followed
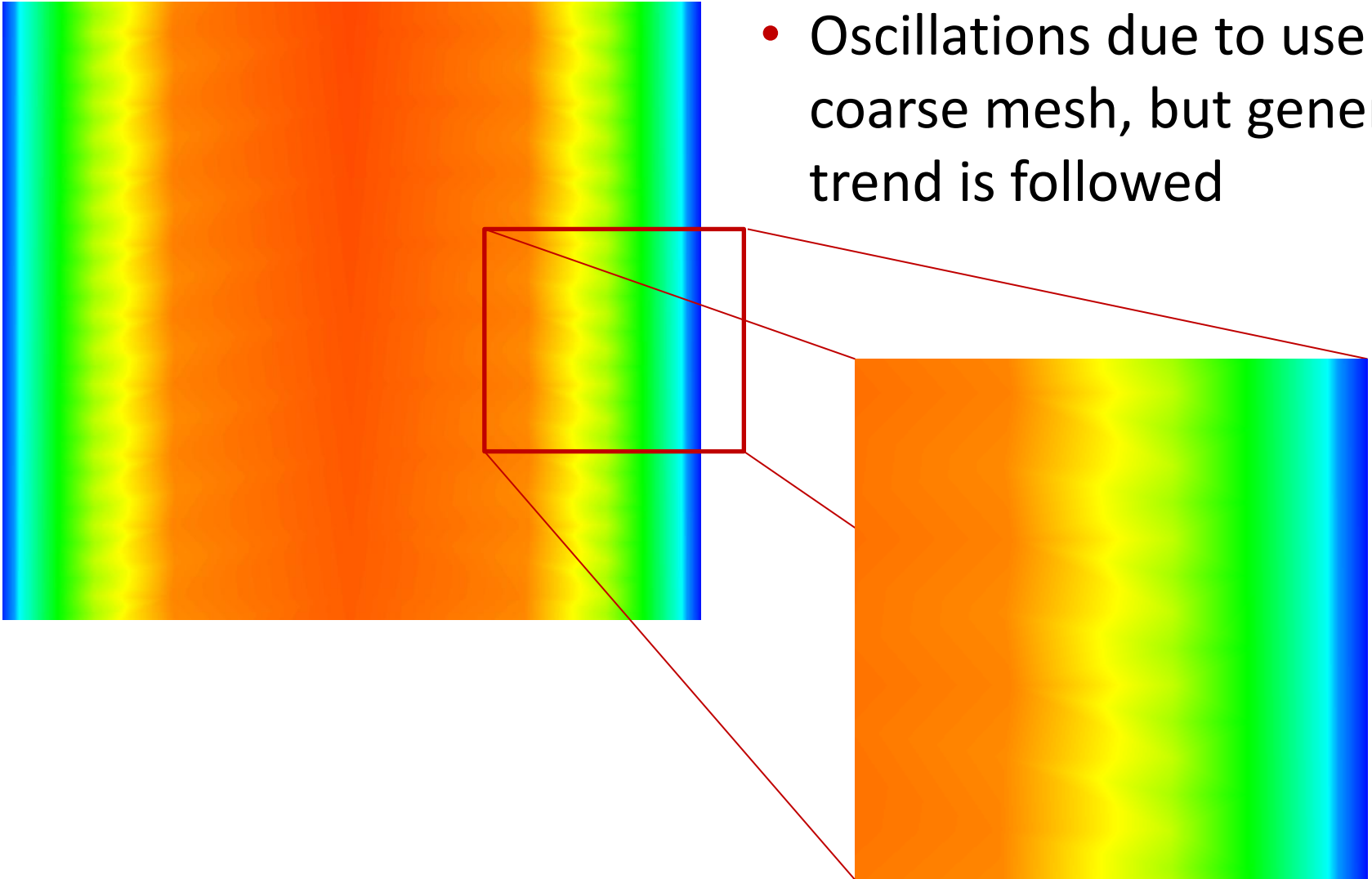
Hands-On Session | Incompressible Flow
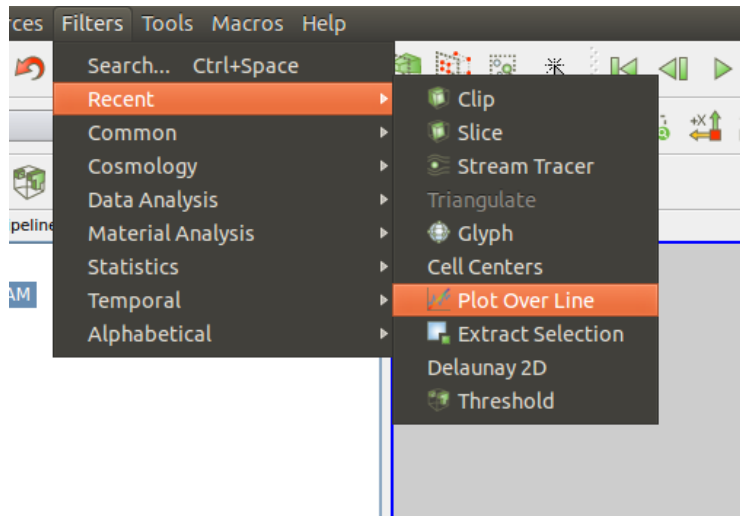
We can visualize the velocity profile by using the "plot over line" filter

48. Select plot-over-line through the Filters menu. Alphabetical > plot over line or you can select through Recent > Plot Over line
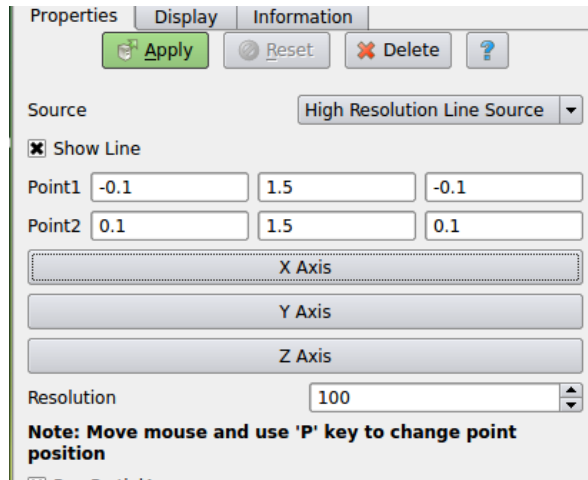
49. Enter the starting and ending points of the line, along with the resolution as show to the left.

engys

Hands-On Session | Incompressible Flow
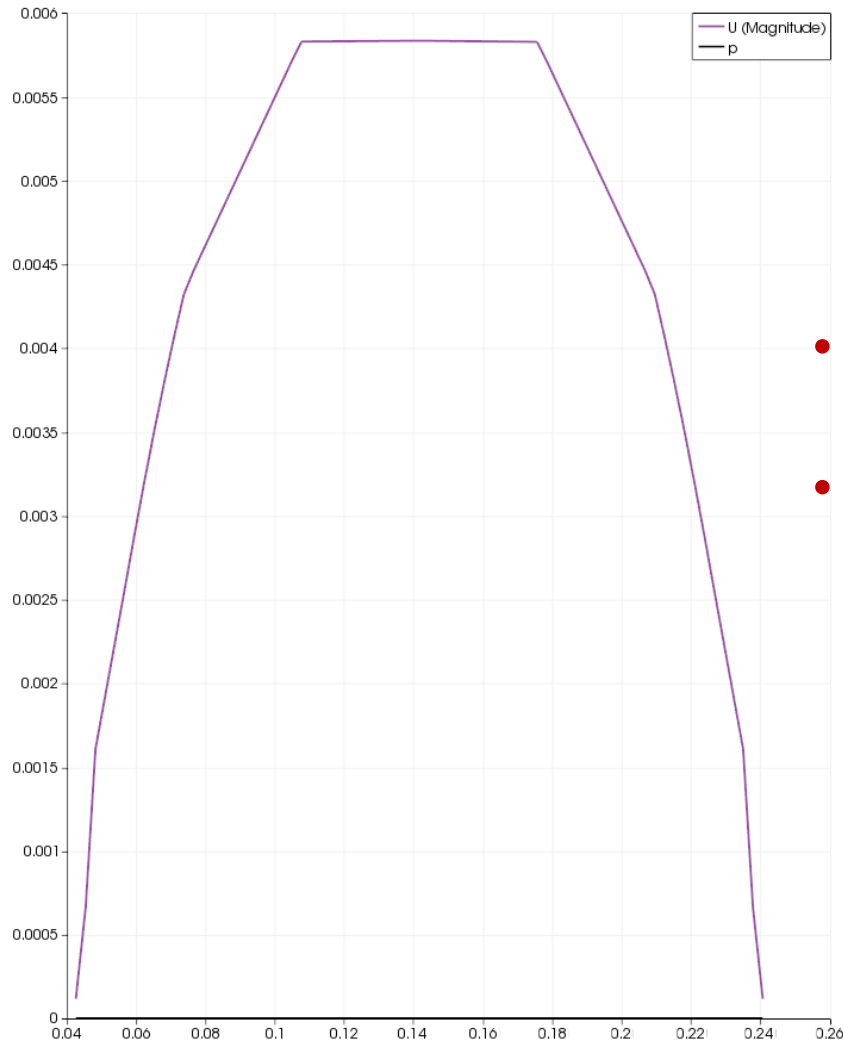
- You will see a low resolution parabola of the velocity magnitude.
- This curve should approach a parabola as our radial mesh density increases.

Hands-On Session | Incompressible Flow

Further Study

- Increase mesh density

- Increase flow rate and add turbulence model

- Add energy with temperature differences at wall and inlet

- Explore other boundary conditions

- Look at velocity profile at different y-values

- Do more complex cuts (combine multiple cuts)

engys

Hands-On Session | Turbulent Flow

Overall Goal of Session

- To familiarize users with configuring and running cases that use turbulence (RANS)

Skills Obtained

✓ Importing STL surfaces into HELYX-OS

✓ Import eMesh files to resolve edges explicitly

✓ Defining, mesh, and run solvers through HELYX-OS

✓ Visualize a result in Paraview with stream tracers and tubes

engys ™

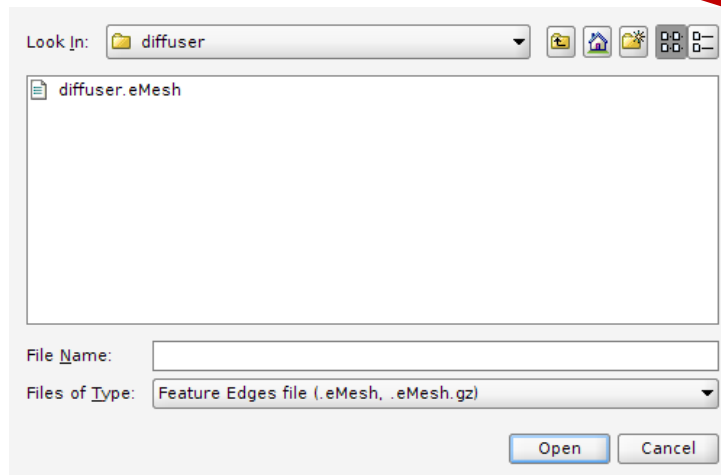## Reynolds Averaged Navier-Stokes

Modeling of Diffuser
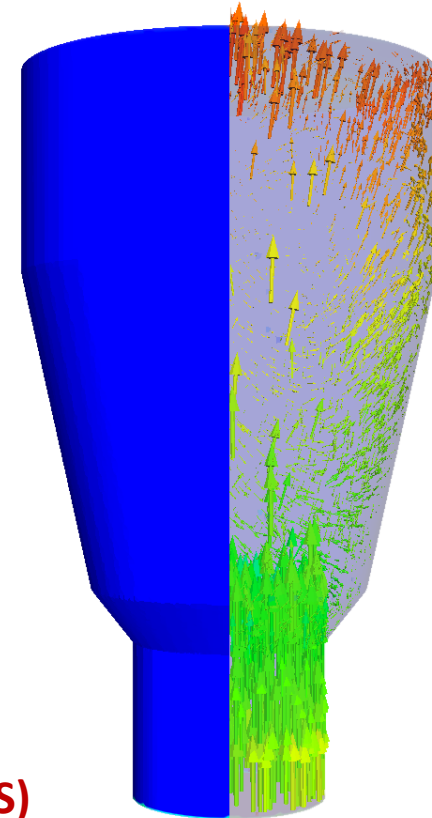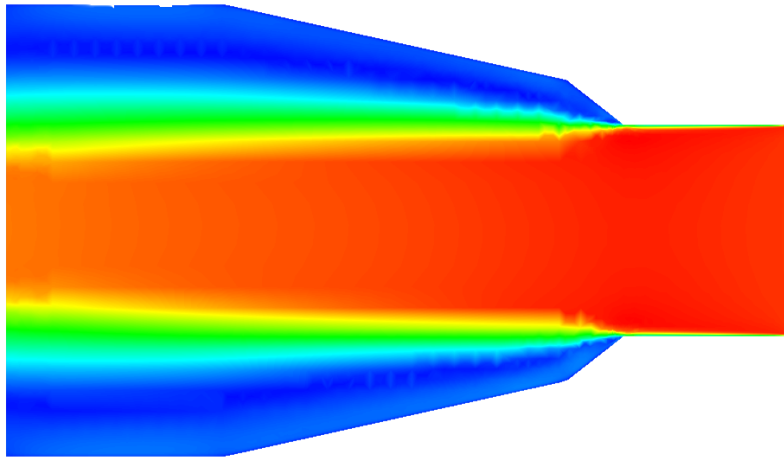
1. Create a new case and call it diffuser in the your favorite parent directory (e.g. $FOAM_RUN/HELYX-OS)

Hands-On Session | Turbulent Flow

Surface Region::Lines
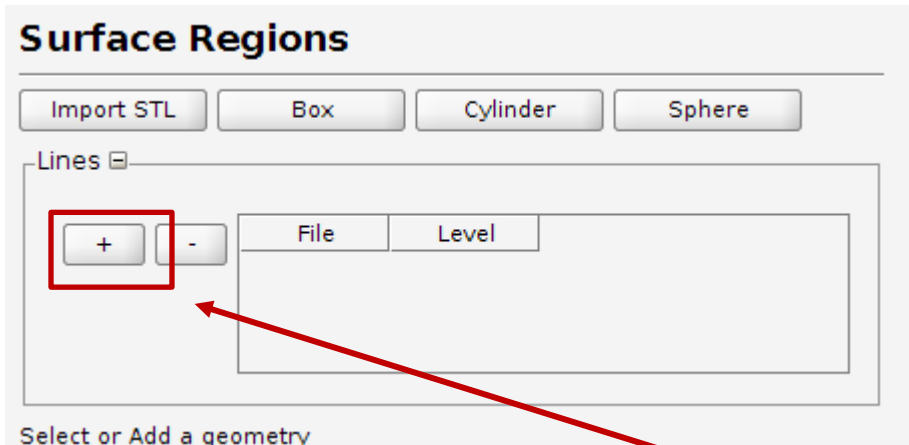
We want to further define the edges of the object using an eMesh file, created using `surfaceFeatureExtract` utility

2. **Click the "+" button**
3. **Navigate to the diffuser.eMesh file in the geometry folder and click "Open"**
4. **Set the refinement level to 2 for edge**

engys

Hands-On Session | Turbulent Flow

Base Mesh

Base Mesh

Base Mesh Type — Automatic

Base Mesh Spacing — 0.004

5. Create a base mesh using the automatic functionality with a spacing of 0.004

Surface Region::Import STL

Surface Regions

| Import STL | Box | Cylinder | Sphere |
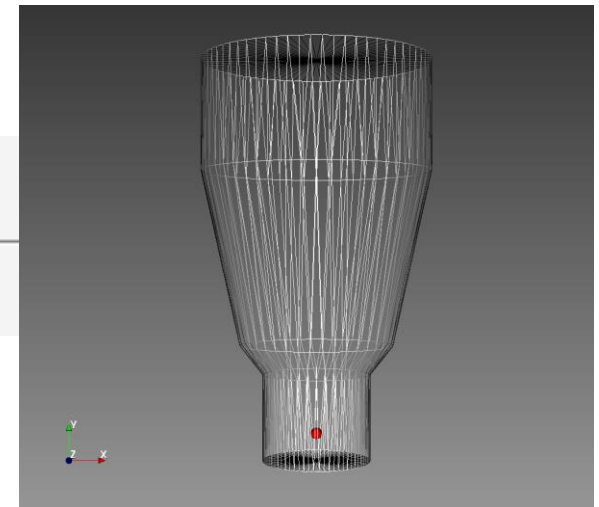
5. Navigate to the "Surface Regions" tab
6. Import the diffuser.stl geometry
7. Go through and set the refinement and layer control for each surface according to table on net slide

engys

Hands-On Session | Turbulent Flow

Surface Regions :: Surface Refinement

Region	Min	Max
Inlet	1	2
Outlet	1	2
Walls	2	3

Surface Regions :: Layers
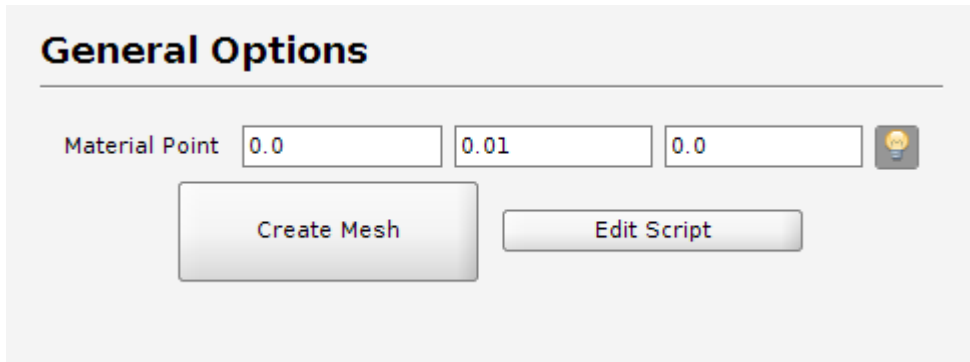
Region	Number of Layers	Relative Size	Final Thickness	Layer Min Thickness	Layer Stretching
Inlet	n/a	n/a	n/a	n/a	n/a
Outlet	n/a	n/a	n/a	n/a	n/a
Walls	3	Yes	0.33	0.22	0.101

engys™

Hands-On Session | Turbulent Flow

Material Point
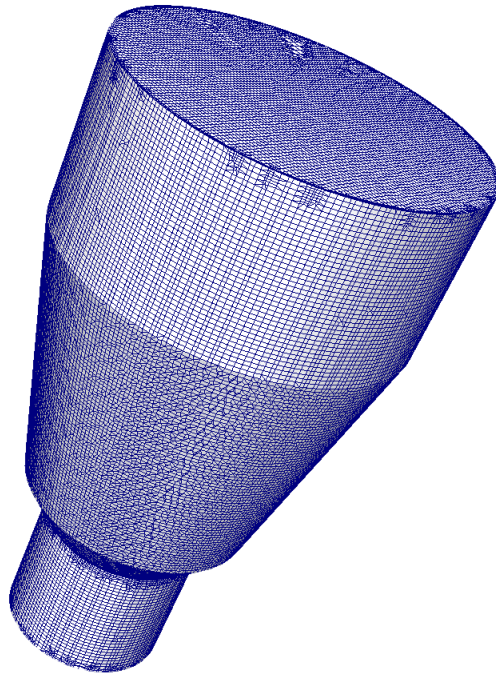
8. Navigate to the "General Options" tab
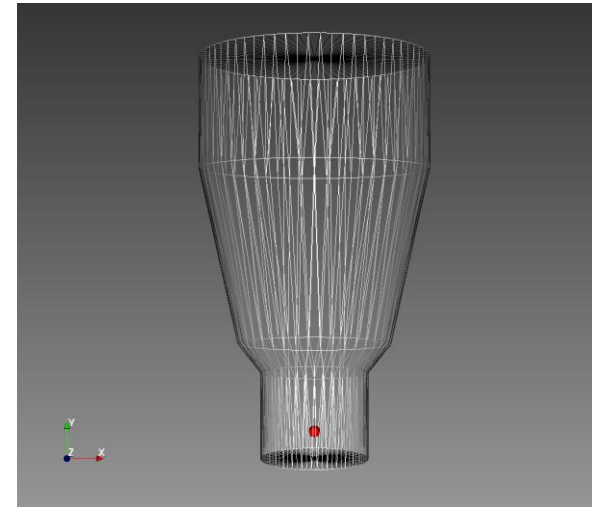
We must define a point where a mesh cell will exist. To do this, we need to select a point in space inside the pipe.
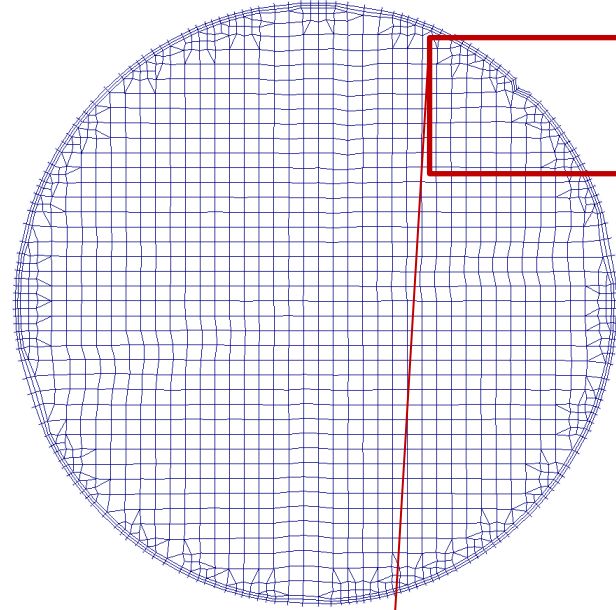
Use the light bulb icon to visualize the current point as a red dot
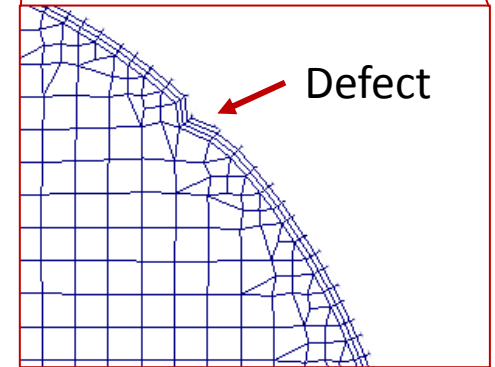
9. Enter (0, 0.01, 0) and hit "Create Mesh"
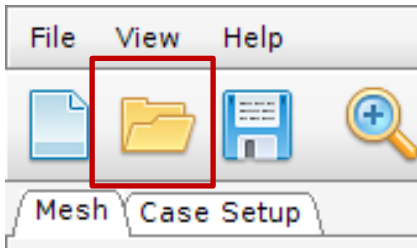
Hands-On Session | Turbulent Flow

Inlet Patch

Defect

- At this point, **HELYX-OS** will execute `blockMesh` and then `snappyHexMesh` based on values we entered, plus default values set by HELYX-OS.

- If we were to visualize this mesh, it would look similar to the one pictured.

- **A smaller base mesh would reduce mesh defects**

engys ™

Hands-On Session | Turbulent Flow

We need to load our meshed case back into HELYX-OS to setup our physics and run a simulation

10. Click the "Folder" icon to open a case

The most recent case defined/run in HELYX-OS will be presented. A user can navigate to another case if desired.

11. Select "Open"

engys

Solution Modeling

We are interested in a **RANS** simulation of the diffuser.

12. **Select "Steady"**
13. **Select "Incompressible"**
14. **Select the "Standard high-Re k-epsilon" model**

Materials

Materials

Database

Material Parameters

Name	air
Density [kg/m³]	1.205
Dynamic Viscosity [Pa·s]	0.000019137
Kinematic Viscosity [m²/s]	0.0000158813
Specific Heat Capacity [J/kg·K]	1,006.0
Laminar Prandtl Number	0.9
Turbulent Prandtl Number	0.85
Thermal Conductivity [W/m·K]	0.024
Reference (absolute) Pressure [Pa]	101,325.0
Thermal Expansion Coefficient [K⁻¹]	0.00333
Reference Temperature [K]	300.0

We want to take it easy and model "Air". Air is the default material

15. **Click on the "materials tab"**
16. **Check that "air" is the material for the simulation**
17. **Move to the "Boundary Conditions" tab**

engys

Hands-On Session | Turbulent Flow

Boundary Conditions

- **<u>Remember:</u>** Both Momentum and Turbulent variables must be defined at each boundary

18. **Click on the diffuser_inlet patch**
19. **Set the type to "patch"**
20. **In the momentum tab, set velocity to fixed value (0, 10, 0) and pressure to zeroGradient**
21. **In the Turbulence tab, set the BC type to "By turbulent intensity and mixing length"**
22. **Set k = 0.01 and the epsilon mixing length to 0.002**

engys

Hands-On Session | Turbulent Flow

Boundary Conditions

- **<u>Remember:</u>** Both Momentum and Turbulent variables must be defined at each boundary

23. **Click on the diffuser_outlet patch**
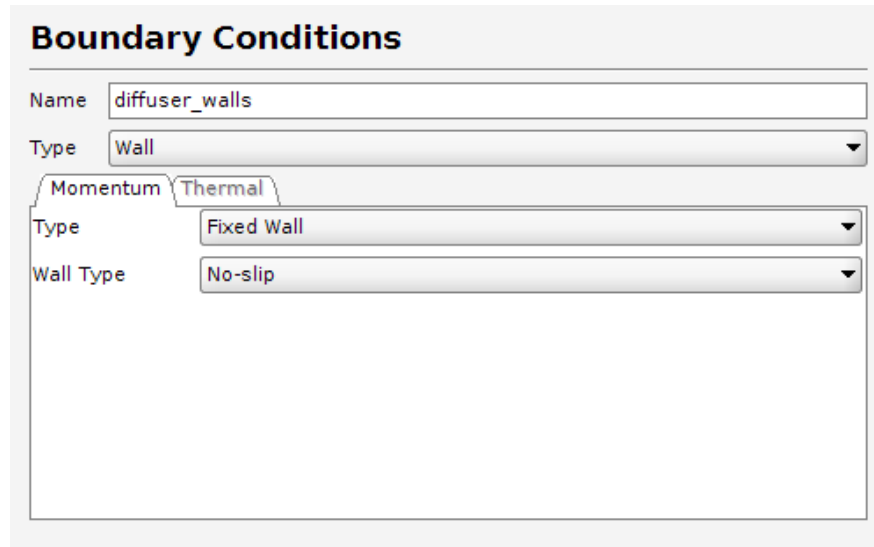24. **Set the type to "patch"**
25. **In the momentum tab, set velocity to "zeroGradient" and pressure to "total pressure" 0**
26. **In the Turbulence tab, set the BC type to "Zero Gradient"**

engys

Hands-On Session | Turbulent Flow

Boundary Conditions

- **Remember:** Both Momentum and Turbulent variables must be defined at each boundary

Boundary Conditions

Name: diffuser_walls

Type: Wall

| Momentum | Thermal |

Type: Fixed Wall

Wall Type: No-slip

27. Click on the diffuser_walls
28. Set to patch type "wall"
29. Set the momentum boundary condition type to "fixed wall" and "no-slip"

engys

Hands-On Session | Turbulent Flow

Initialization

- We can define initial conditions or leave them as defaults

Fields Initialization

U [m/s]	Fixed Value ▼	0.0	10.0	0.0
p [m²/s²]	Default ▼			
k [m²/s²]	Default ▼			
epsilon [m²/s²]	Default ▼			

30. Set the velocity field U to (0, 10, 0) and leave the rest as "default"

HELYX-OS will set the remaining variables to a sensible default value

engys™

Numerical Schemes

- We can also change the default divergence schemes using a drop-down menu

31. Set U to a 2nd order linear upwind scheme and change k and epsilon to 1st order schemes

engys ™

Hands-On Session | Turbulent Flow

Solver Settings

Runtime Controls

Time Settings

Start From	Start Time ▾	0.0
End Time	500.0	
Δt(s)	1.0	
Adjustable Time Step	☐	
Max Courant Number		
Max Time Step		

Data Writing

Write Control	Time Step ▾	250.0
Purge Write	0	
Write Format	ASCII ▾	
Write Precision	10	
Write Compression	Uncompressed ▾	
Time Format	General ▾	
Time Precision	6	
Graph Format	Raw ▾	

Data Reading

Runtime Modifiable	☑

We should set the simulation time controls so that we write out a solution periodically and provide enough time for the solution to converge relatively well.
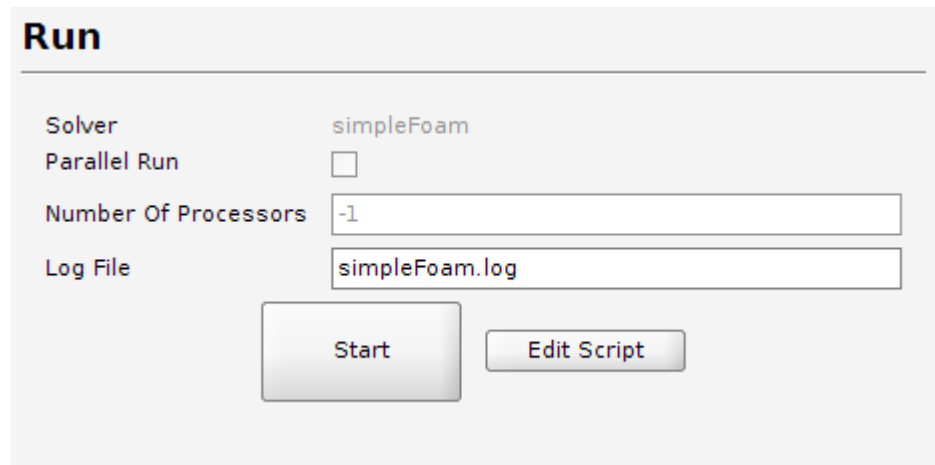
32. Set the application to run for a total of 500 iterations while writing every 250 steps.

engys™

Hands-On Session | Turbulent Flow

Solution Modelling

Materials

Boundary Conditions

Cell Zones

Fields Initialization

Numerical Schemes

Solver Settings

Runtime Controls

Run

The last step is to start the simulation

33. Select the "run" tab

Run

Solver	simpleFoam
Parallel Run	☐
Number Of Processors	-1
Log File	simpleFoam.log

Start   Edit Script

HELYX-OS will execute a run script that will execute the solver `simpleFoam`. The script may be edited using the "edit script" button, but we will not adjust anything

34. Select the "run" button
35. Open a new terminal and navigate to the case directory

engys

Hands-On Session | Turbulent Flow

HELYX-OS Has…

✓ Assembled file structure, dictionaries, and properties needed by OpenFOAM

- controlDict, fvSchemes, snappyHexMeshDict, fvSolution, blockMeshDict, etc.

✓ Created a mesh for our pipe using **`snappyHexMesh`**

- Mesh is located in polyMesh

✓ Allowed the user to setup an incompressible solver for a steady-state turbulent flow (**`simpleFoam`**)

✓ Executed the **`simpleFoam`** solver for a predetermined amount of time

✓ When the simulation has stopped, we can post-process in Paraview

engys

Hands-On Session | Turbulent Flow

- **If** you ran in parallel, in the case directory execute

  reconstructParMesh –constant

- In the case directory, launch Paraview with

  paraFoam

engys
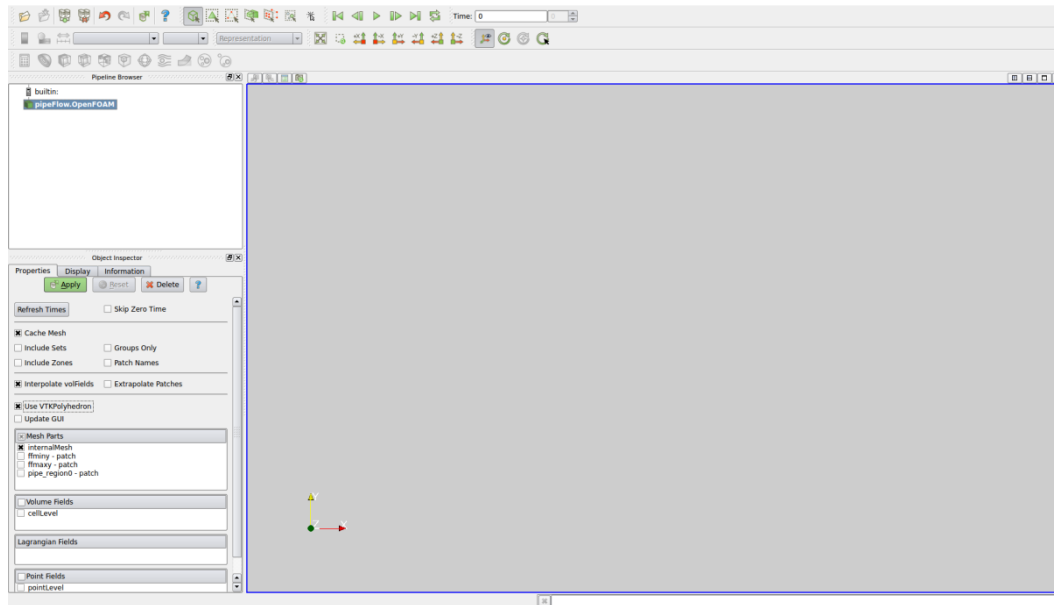
Hands-On Session | Turbulent Flow

Using what we learned from the first tutorial
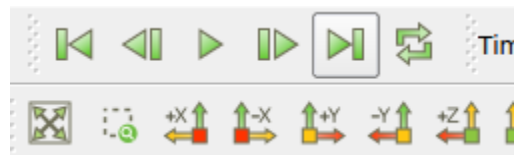
37. Load the mesh in Paraview

Hint
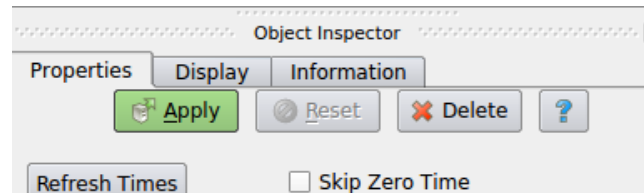
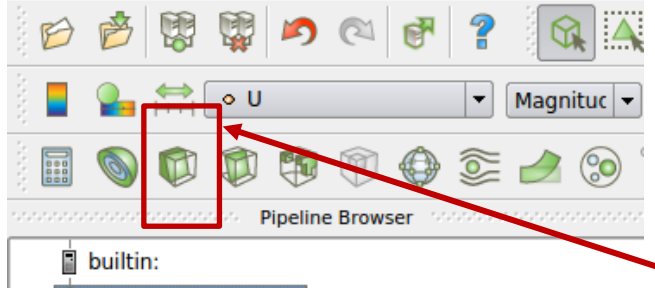

38. Select the surface view and variable U
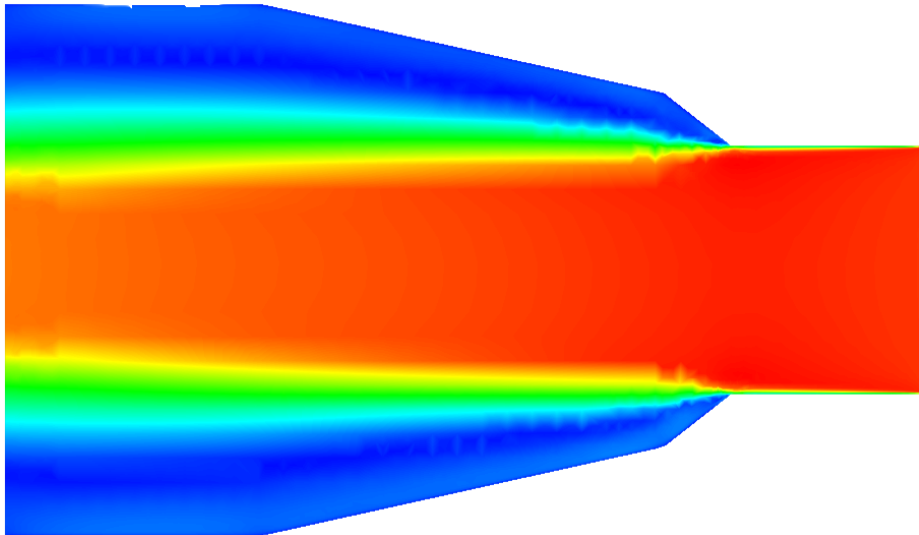
Hint
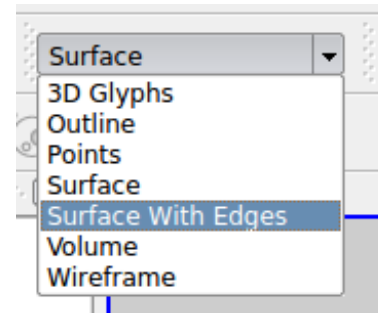
39. Skip to the last time (500)

Hint

Using a cut plane

Paraview uses filters to aid in the post-processing of data from our OpenFOAM simulation

40. Select the "Clip" filter and accept the default settings

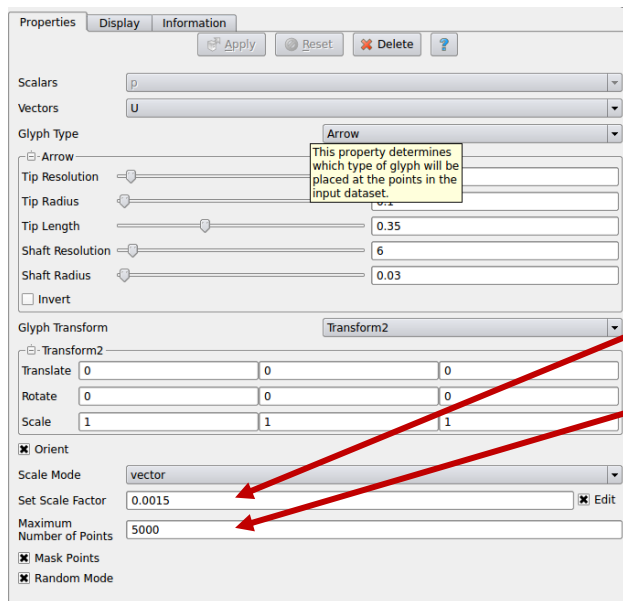# Hands-On Session | Turbulent Flow

Using Glyphs

- To use glyphs, we must first use the "cell-centers" filter

41. Un-hide the original dataset in the pipeline browser
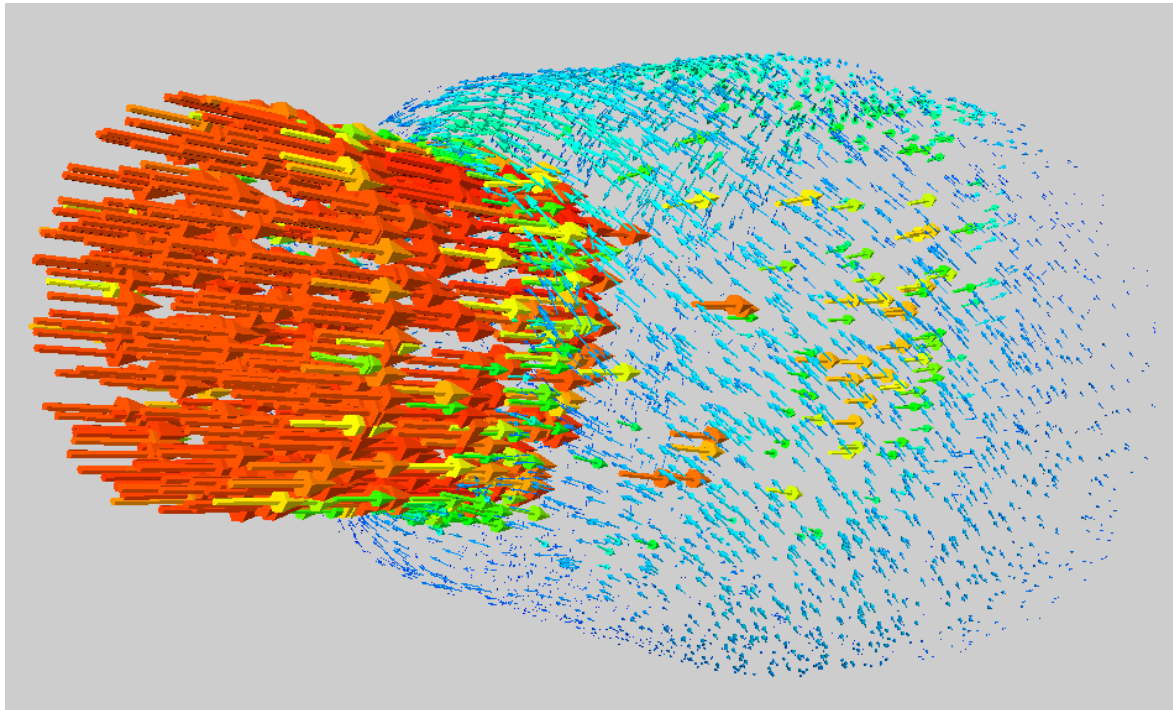
42. Run the "filters>alphabetical>cell-centers" filter

43. Run the "filters>alphabetical>glyphs" filter

Feel free to adjust the "scale factor" and "maximum number" of points to adjust the size and number of the vectors
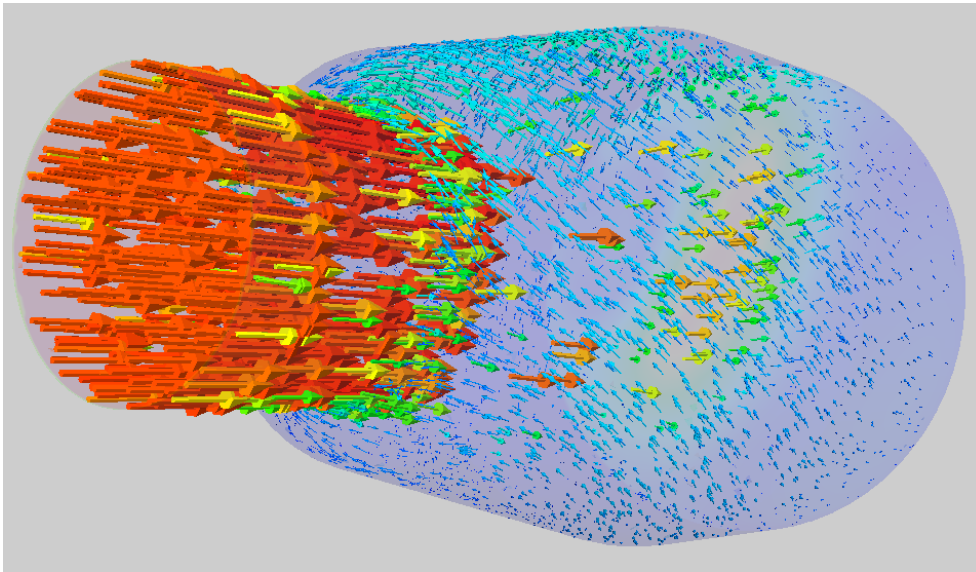
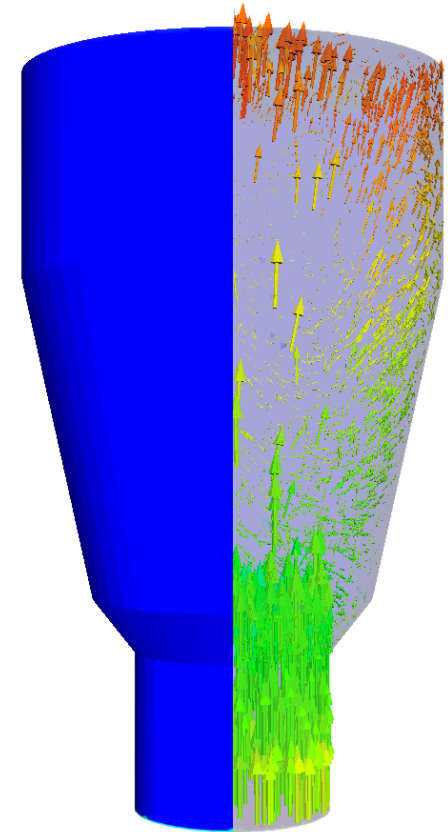engys

Hands-On Session | Turbulent Flow

Using Glyphs

Scaling the vectors and adjusting the count will change how the vector field appears and will bring out finer flow structure

engys

Combine Multiple filters and adjust view settings

Glyphs + Opacity adjustment (0.1)

Cut plane <u>after</u> glyphs have been created

engys

Hands-On Session | Incompressible Flow

Further Study

- Increase mesh density

- Add energy with temperature differences at wall and inlet

- Try different turbulence models to see any influence of model on the flow solution

- Try the "stream-tracer" filter and test out the point or line source options
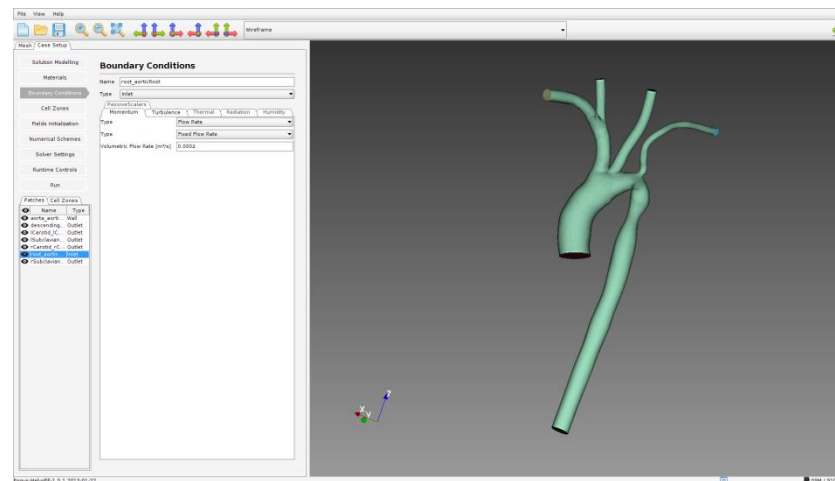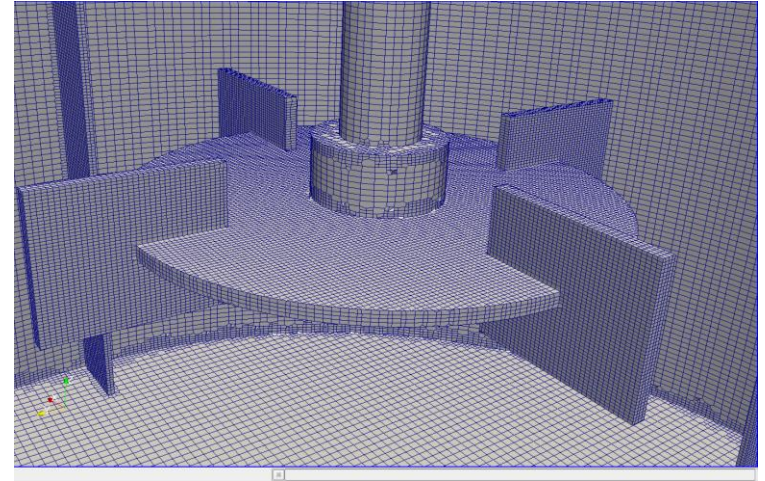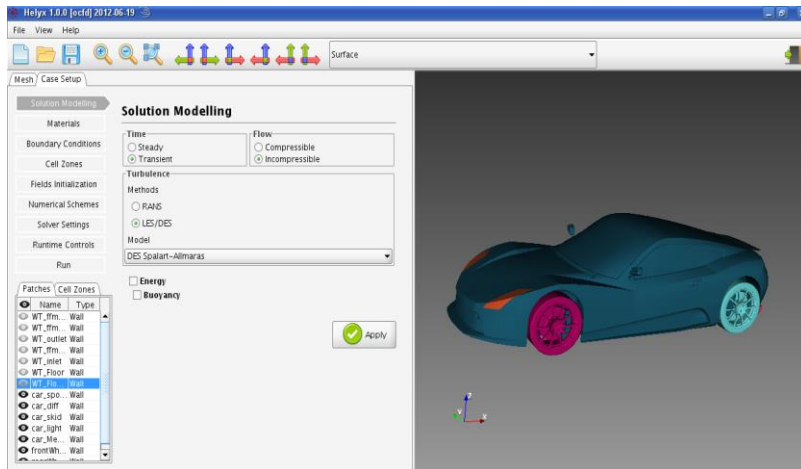
engys

Contents

- **HELYX-OS** (30 Minutes)
  - Capability Overview
  - Workflow Walkthrough
    - Meshing, Case Setup, and Execution
- Hands-On Examples (~1 hour)
  - Incompressible Flow
  - Turbulent Flow
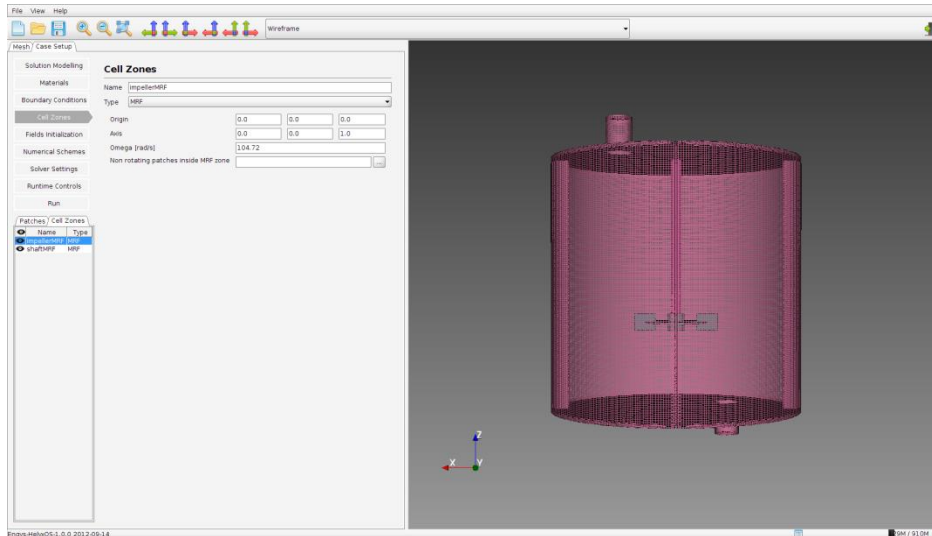- Closing Remarks (5 Minutes)

Generated with HELYX™

engys

Closing Remarks

- HELYX-OS has been used for aerospace, automotive, biomedical, chemical, HVAC, applications.

engys

Closing Remarks

- HELYX-OS has been used for aerospace, automotive, biomedical, chemical, HVAC, applications.

- More functionality that wasn't covered in this tutorial

  - Multiple Reference Frame Modeling

  - Porous Media Flow Modeling

  - Advanced Scripting within HELYX-OS

Closing Remarks

- HELYX-OS has been used for aerospace, automotive, biomedical, chemical, HVAC, applications.

- More functionality that wasn't covered in this tutorial
  - Multiple Reference Frame Modeling
  - Porous Media Flow Modeling
  - Advanced Scripting within HELYX-OS

- HELYX-OS functionality is developing quickly
  - Testing and feedback from users is key
  - Use cfd-online and sourceforge for discussion

engys ™

Closing Remarks

- HELYX-OS has been used for aerospace, automotive, biomedical, chemical, HVAC, applications.

- More functionality that wasn't covered in this tutorial
    - Multiple Reference Frame Modeling
    - Porous Media Flow Modeling
    - Advanced Scripting within HELYX-OS

- HELYX-OS functionality is developing quickly
    - Testing and feedback from users is key
    - Use cfd-online and sourceforge for discussion

- Go to www.engys.com/products

- Download at http://sourceforge.net/projects/helyx-os/

- Come to the next session on `snappyHexMesh`

engys ™

Questions?

Thank You
감사합니다

www.engys.com

engys™