Ansys Fluent Getting Started
(New Fluent Experience)

Workshop: Vortex Shedding

Release 2021 R1

Ansys

Capability Level

- This tutorial is supported by the Enterprise and Premium capability levels
  - These are the only capability levels which support transient flow simulations

Introduction

Workshop Description:

The purpose of this workshop is to introduce good techniques for transient flow modeling

Learning Aims:

This workshop teaches skills for using Fluent to perform time-dependent (transient) simulations. Topics covered include:

– Selecting a suitable time step   - Generating Fast Fourier Transforms (FFT)

– Auto-saving results during the simulation   - Generating animations of the solution

– Using non-iterative time advancement (NITA)

Learning Objectives:

• To show how to set up, run and post-process a transient simulation

• Additional post-processing skills in making animations and using fast Fourier transforms

Problem Description

- The case considered here is flow around a cylinder with a Reynolds number of 100

- This workshop demonstrates iterative and non-iterative time advancement schemes, Fast Fourier Transforms (FFT) and animations

- The goal of the simulation will be to visualize the transient vortex shedding behavior and to use the FFT feature to identify the vortex shedding frequency
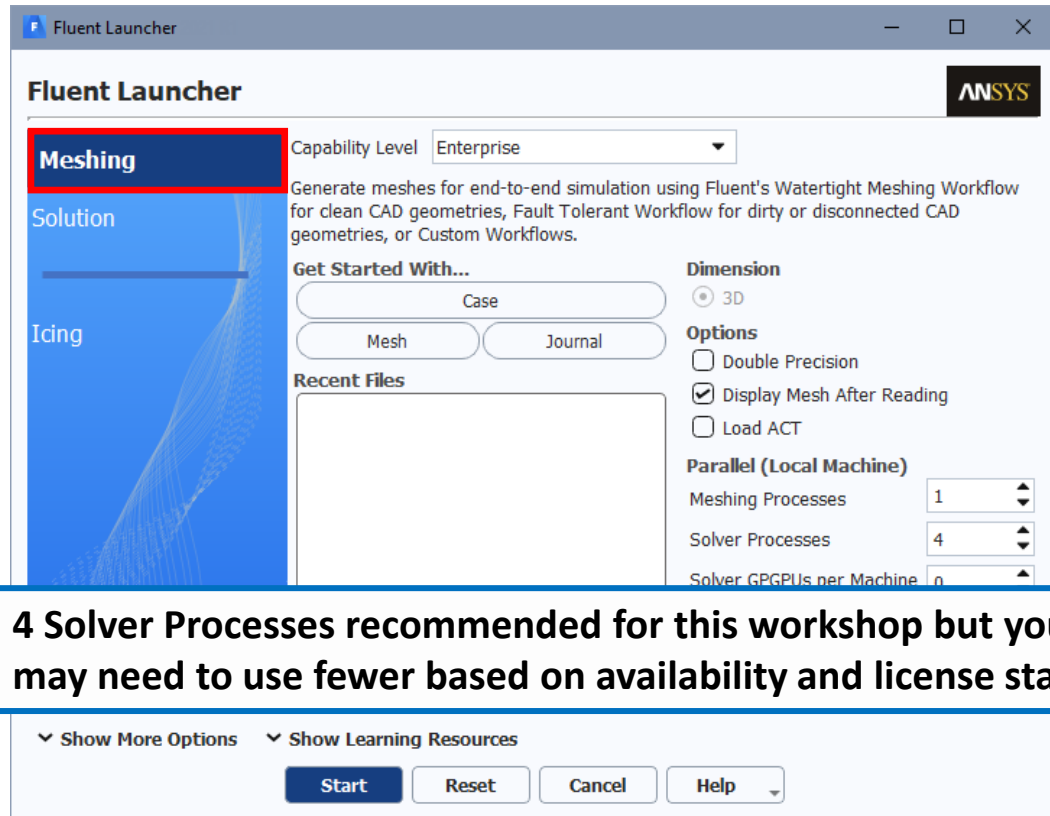
Computational Domain
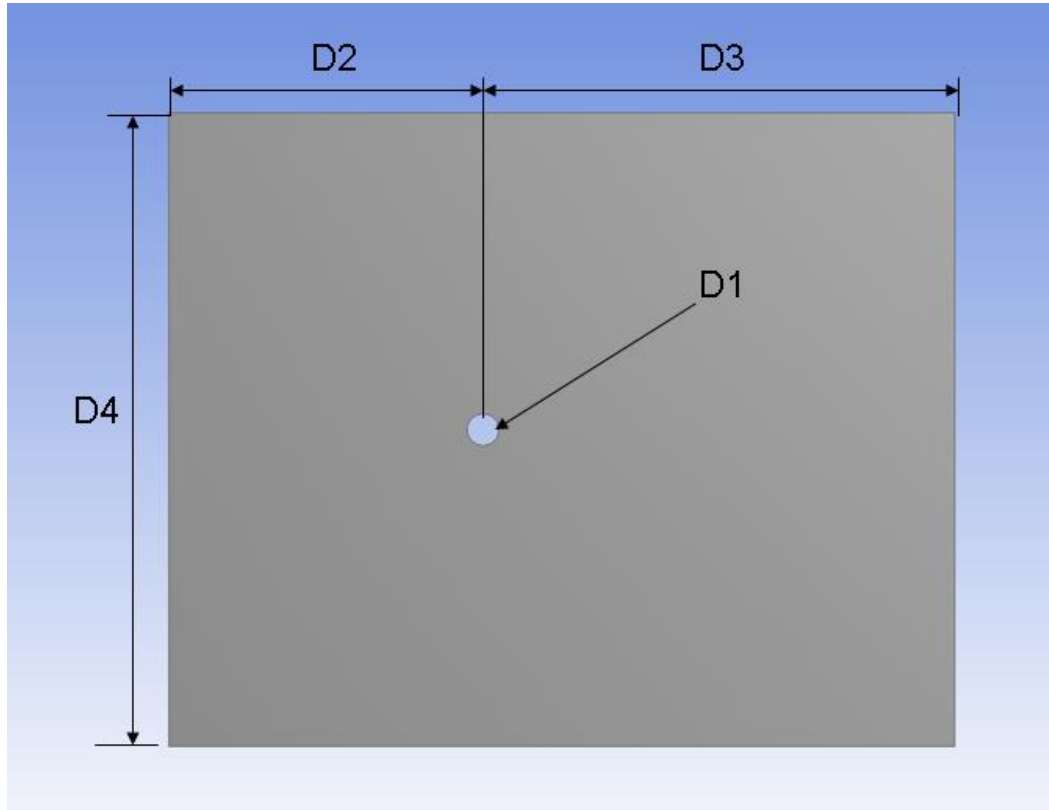
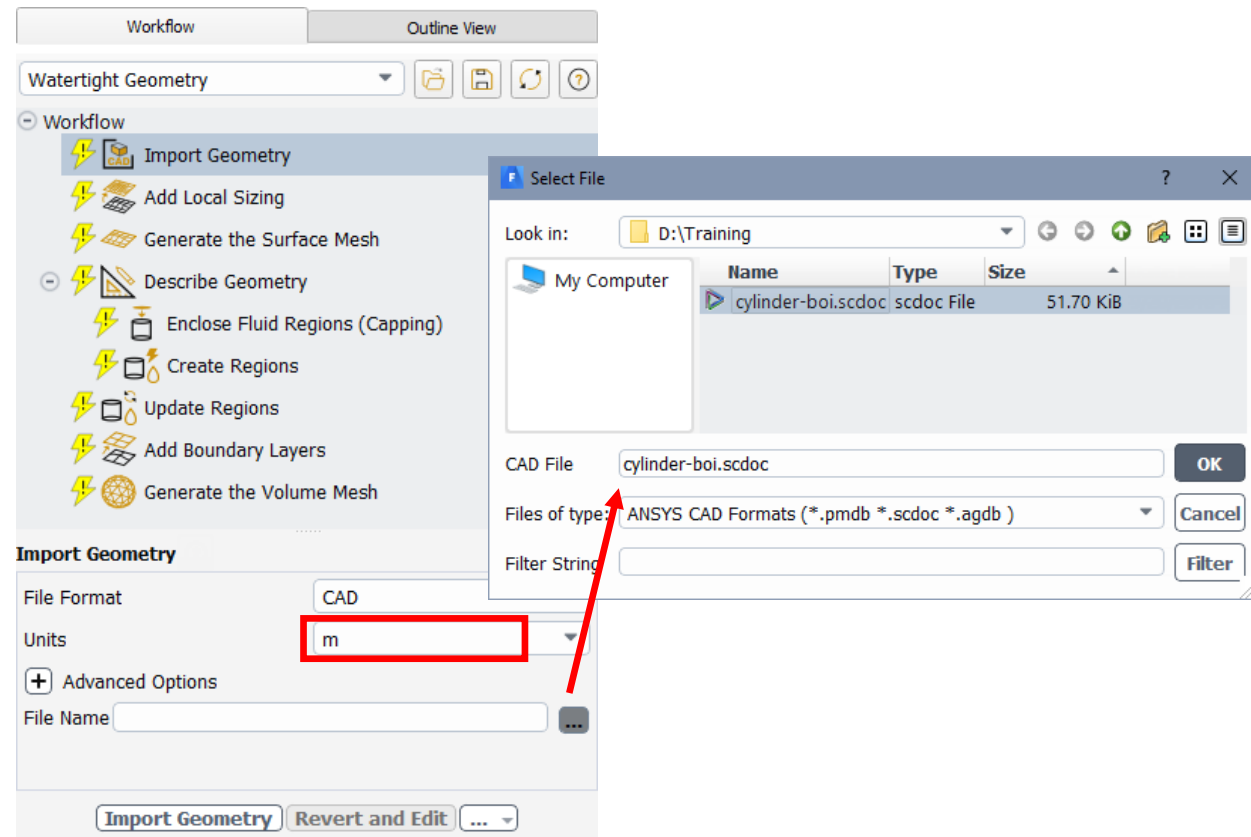| Name	Location	Dimension
Cylinder	D1	1 m (dia.)
Inlet Length	D2	10 m = 10 D
Outlet Length	D3	15 m = 15 D
Width	D4	20 m = 20 D

For use in Fluent Meshing, the geometry is extruded in the z-direction a distance of one diameter and symmetry conditions are applied to the side boundaries

Ansys

Starting Fluent

- Open the Fluent Launcher Window and ensure Meshing Mode is selected
 - If necessary, look under "Show More Options" to change the working directory

- Select the Watertight Geometry workflow, set Units to m and import "cylinder-boi.scdoc"

4 Solver Processes recommended for this workshop but you may need to use fewer based on availability and license status.

Add Local Sizing (Body of Influence)

- A BOI has been defined in the CAD file to enforce good mesh resolution in the proximity of the cylinder and the downstream wake region

BOI

Inserting a clipping plane in z may help to view BOI
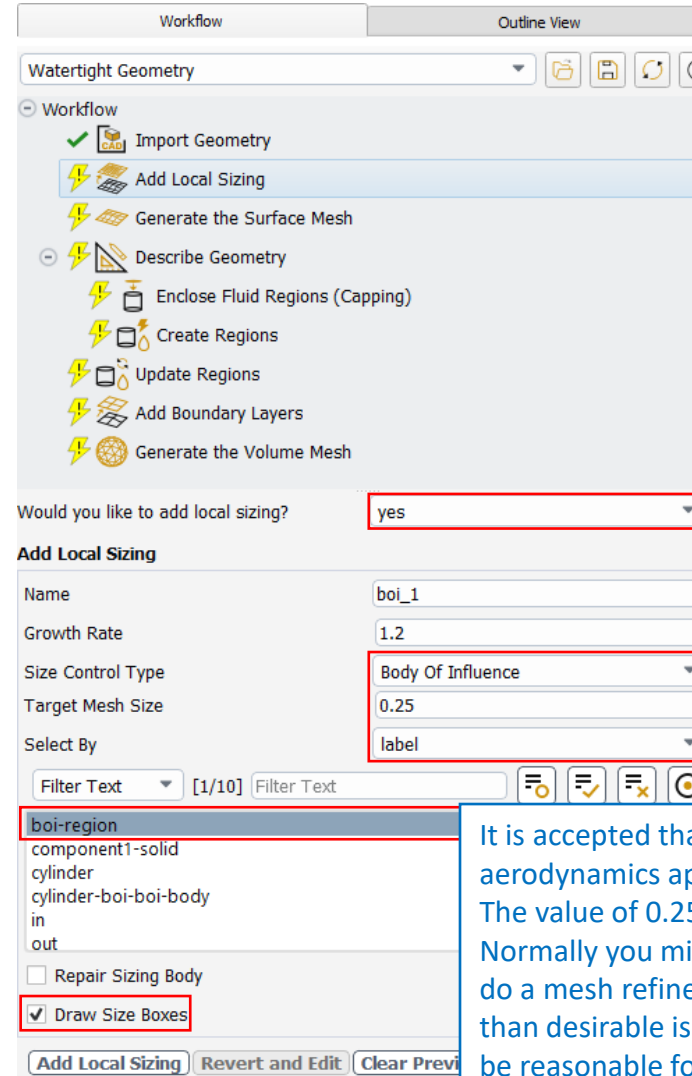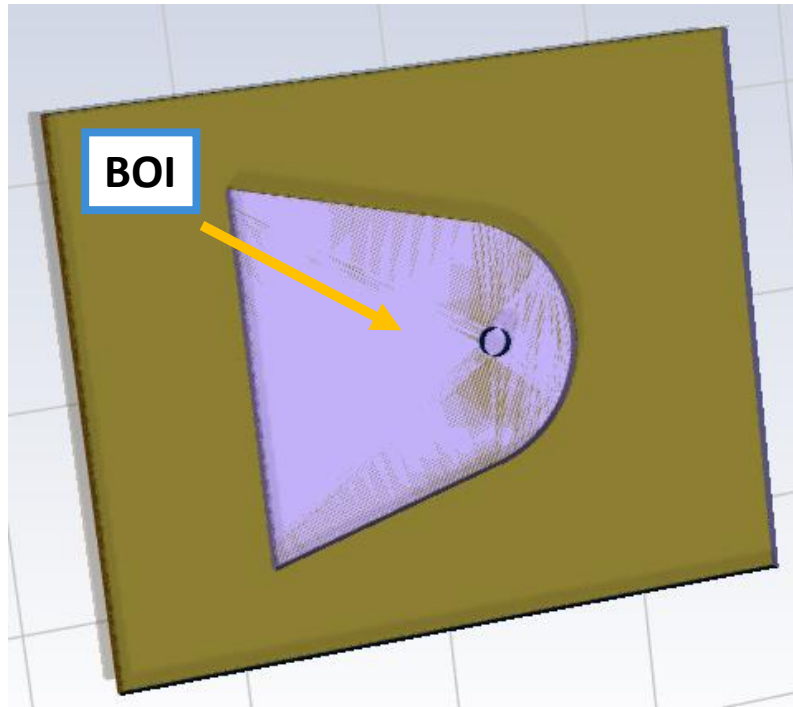
**Enter the following in the Add Local Sizing task and then click Add Local Sizing:**

Type = Body of Influence
Target Mesh Size = 0.25 m
Growth Rate = 1.2
Face Zone Label = "boi-region"

It is accepted that good resolution of the wake region is important in aerodynamics applications and the BOI allows this to be achieved. The value of 0.25 m is based on one-fourth of the cylinder diameter. Normally you might start with one-tenth of the diameter and then do a mesh refinement study, but here a value that is probably larger than desirable is used in order to keep the cell count low enough to be reasonable for a tutorial exercise.

Generate the Surface Mesh

- Keep the default minimum, maximum sizes and growth rate
- Set Size Functions to Curvature with a Normal Angle of 12° and click Generate the Surface Mesh
  - For aerodynamics applications such as this, finer resolution is preferred so the angle is decreased from the default 18°
  - Proximity would result in over-refinement of the mesh near boundaries in this quasi-2d geometry
  - The maximum skewness is reported in the console window … values below 0.7 are considered acceptable
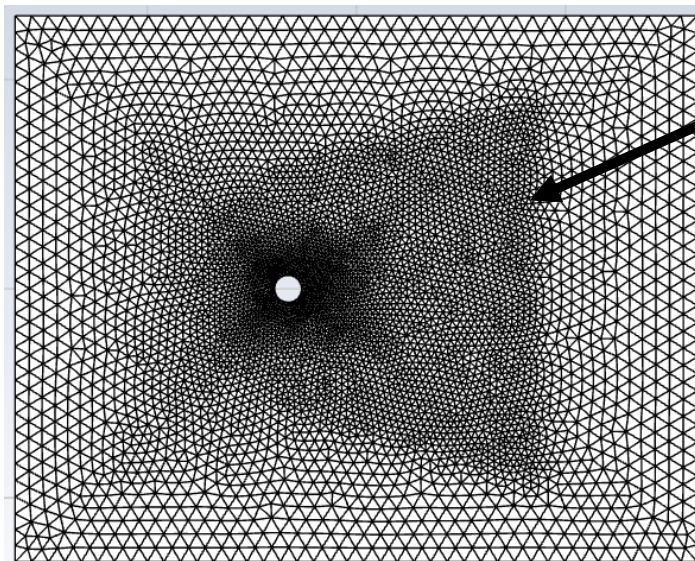
Finer mesh in BOI region
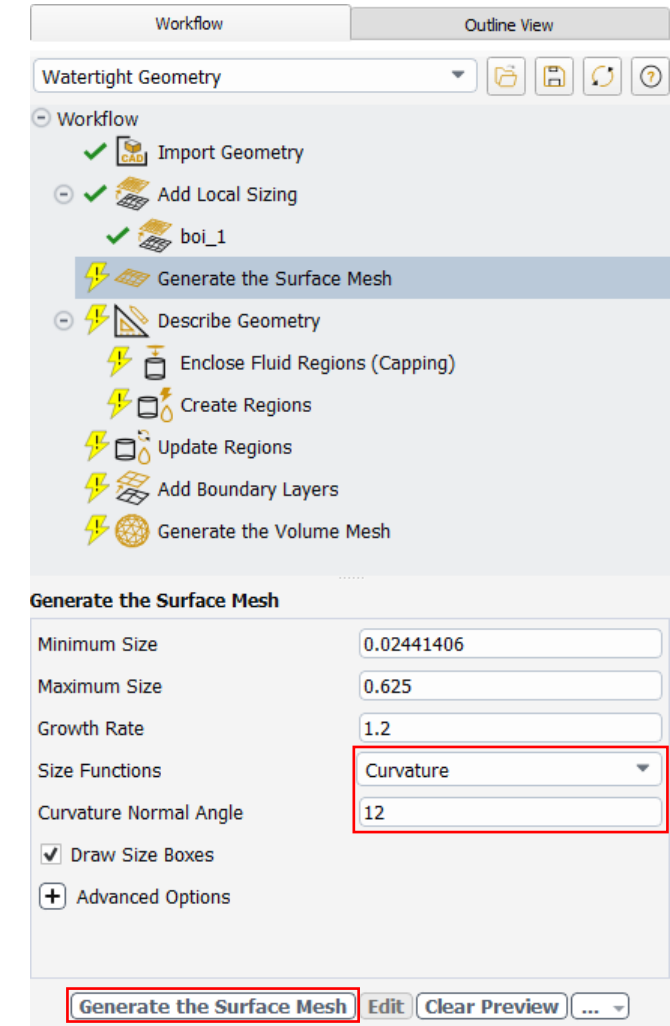
Describe Geometry

- The geometry consists of only fluid regions

- There are no fluid-fluid boundary types so there is no need to bother with that entry

- Similarly, there is no need to apply Share Topology

- Set the Geometry Type as shown to the right, click on Describe Geometry and in the next step you will assign boundary names and types to various zones
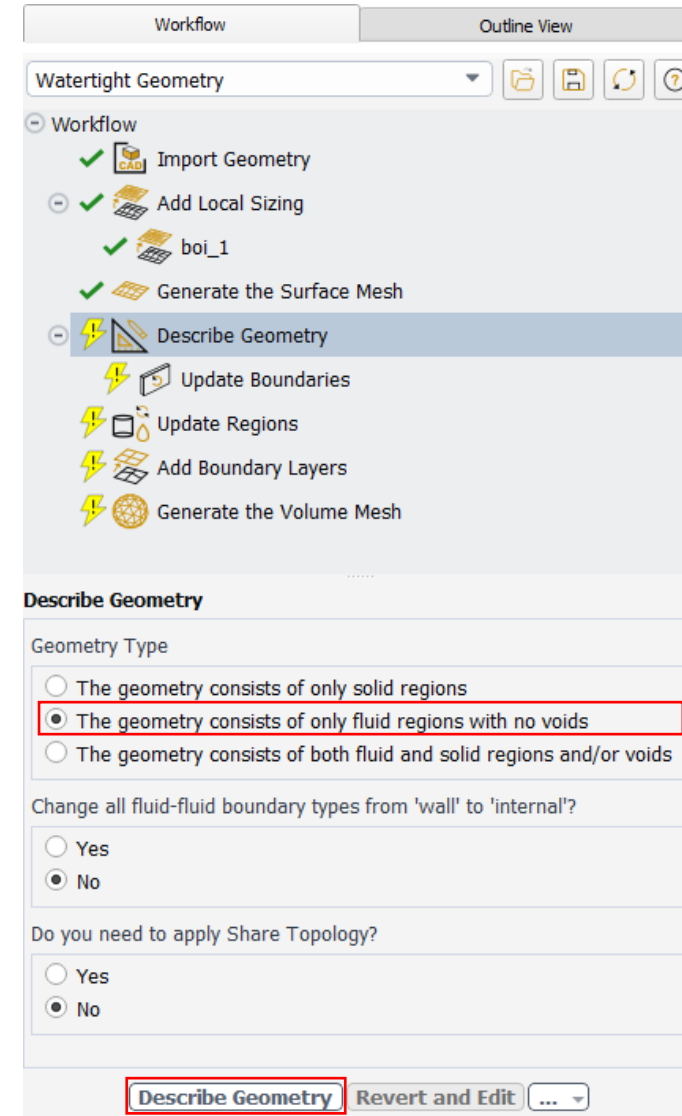
Update Boundaries

- First, change the type of "in" to velocity-inlet and the type of "out" to pressure-outlet"

- Next, select all four symmetry boundaries in the list, right click and choose Set Boundary Type

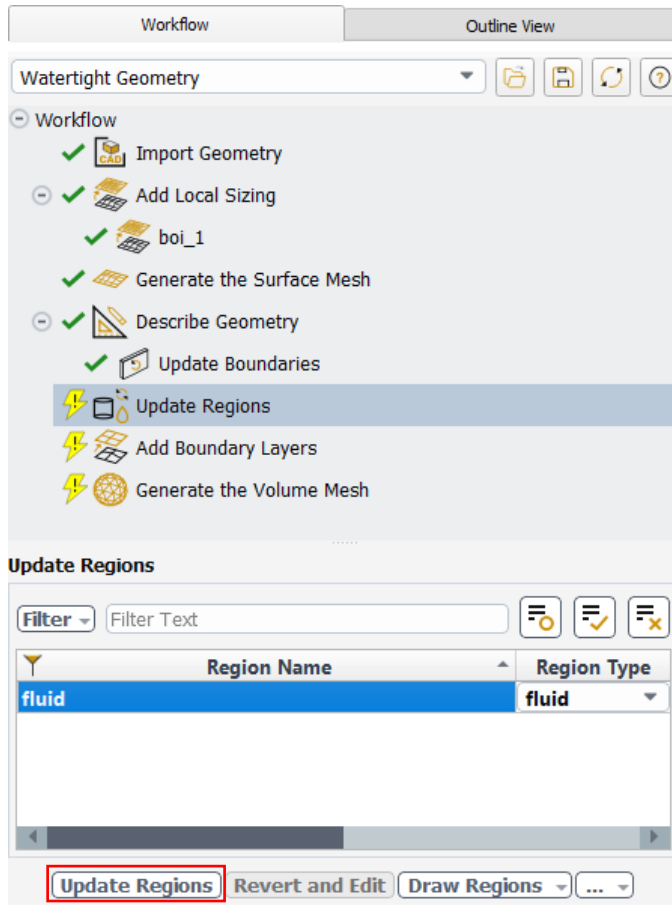- Select "symmetry" in the menu that appears
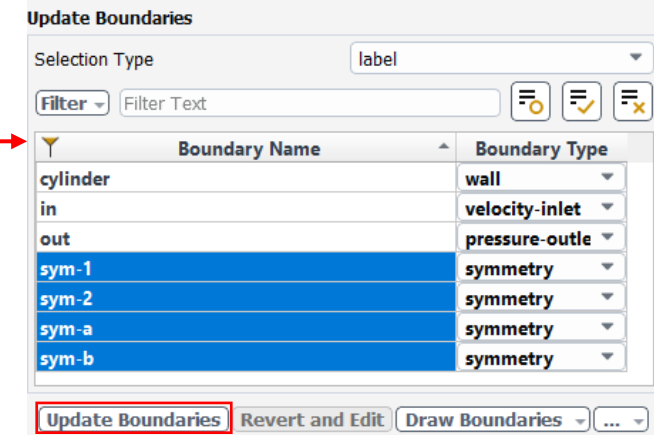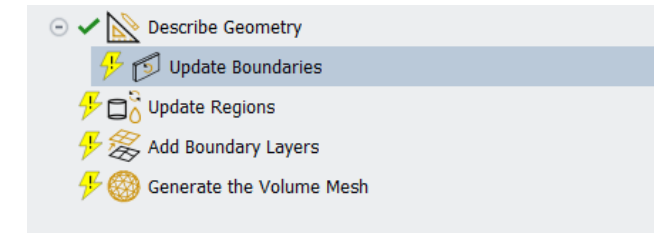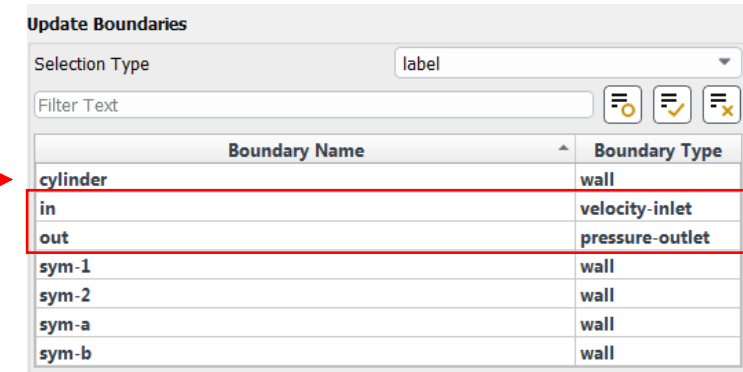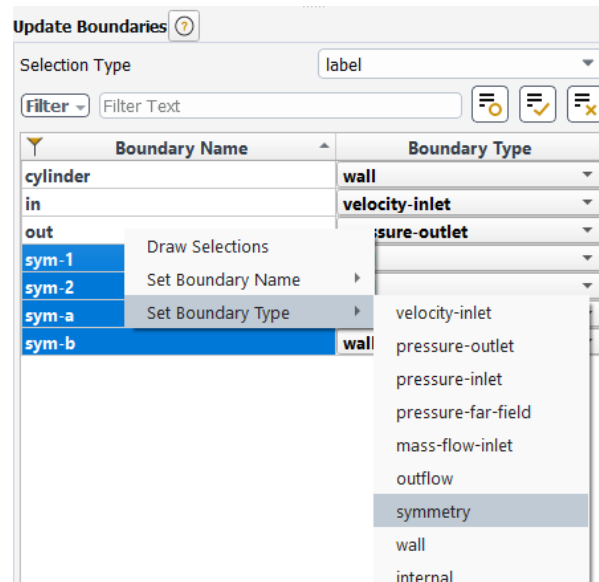
- Click on Update Boundaries

In this way, all boundaries can be changed from wall to symmetry in a single operation.

Regions

- In Update Regions, change the name of the fluid region to "fluid" and click on Update Regions

Add Boundary Layers and Generate the Volume Mesh

- In the Add Boundary Layers task, change the Offset Method Type to uniform, enter 15 layers and set the first height to 0.02 mm
  - It is intended to try to have a fine boundary layer mesh – the exact value is only an educated guess though
- In the Generate the Volume Mesh task choose polyhedra
  - If necessary, unselect Enable Parallel Meshing
  - On completion of the volume mesh task, Fluent reports the minimum orthogonal quality of the mesh in the console window – review the console output to ensure this value is 0.1 or higher
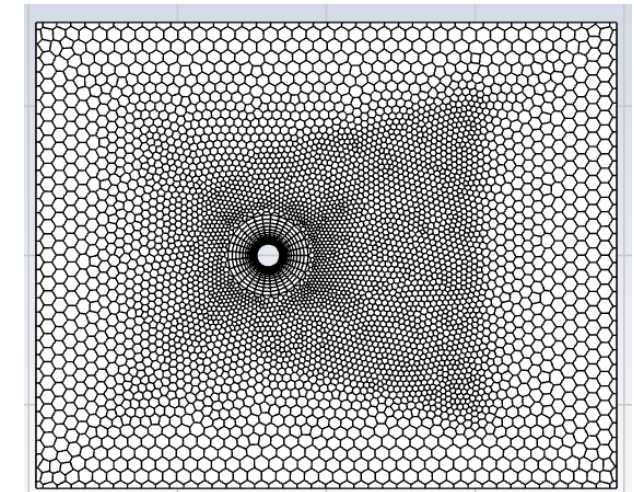
Write Mesh and Switch to Solution Mode

- Go to File > Write > Mesh and save the mesh as "cylinder-volume-mesh.msh.gz"
  - Workflow inputs are stored with the mesh so in case it is desired to make changes in the future, it is easy to do so after reading the mesh into a new Fluent Meshing session

- Click on Switch to Solution
  - Click Yes in the question panel that appears
  - Mesh information is transferred to the solver and the GUI changes from meshing mode to solution mode
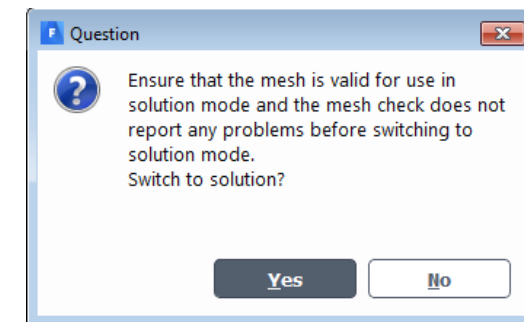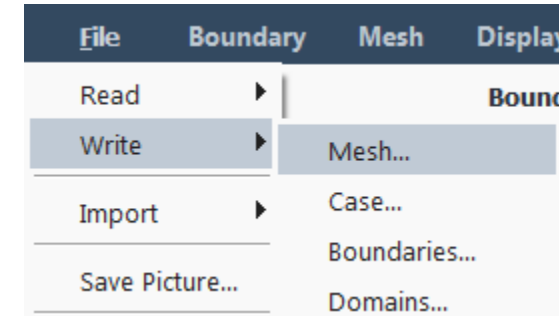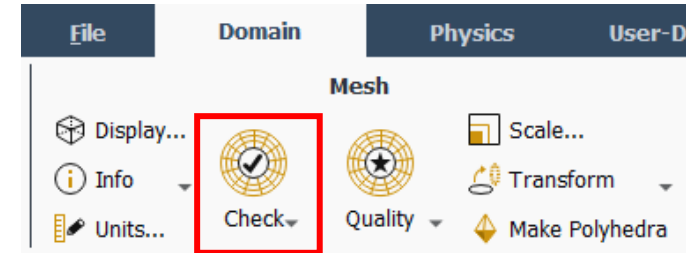
Fluent Workflow: Ribbon

- The Ribbon is used to guide the basic Fluent workflow
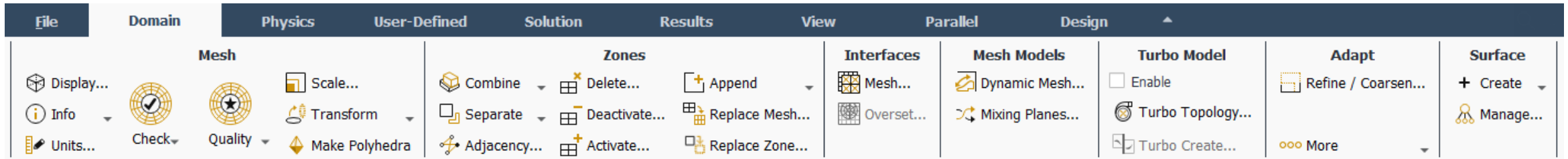
- The four primary tabs used in every simulation are
  - Domain
  - Physics
  - Solution
 - Results

Domain: Mesh Check

- In the Domain tab, in the Mesh group, click Check and and Perform Mesh Check examine the output in the Console
 - No errors are reported


```
Console
Domain Extents:
  x-coordinate: min (m) = -1.000000e+01, max (m) = 1.500000e+01
  y-coordinate: min (m) = -1.000000e+01, max (m) = 1.000000e+01
  z-coordinate: min (m) = -5.000000e-01, max (m) = 5.000000e-01
Volume statistics:
  minimum volume (m3): 9.211008e-05
  maximum volume (m3): 2.183828e-01
    total volume (m3): 4.992278e+02
Face area statistics:
  minimum face area (m2): 1.235077e-04
  maximum face area (m2): 4.399124e-01
Checking mesh.....................................
Done.
```

The mesh check returns no error messages.

Domain: Create a Monitoring Point
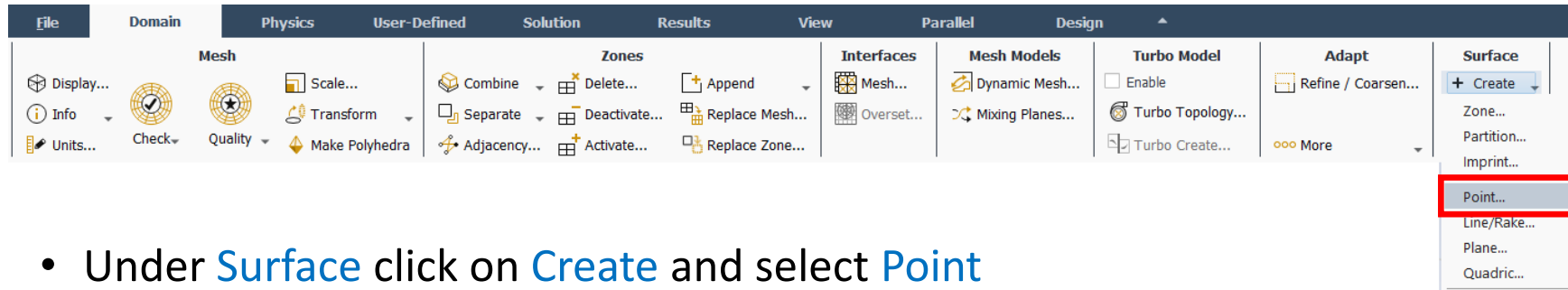
- Under Surface click on Create and select Point
  - The point will be used later as part of a Report Definition for monitoring the solution
  - It is convenient to do so now while working in the Setting Up domain tab, but points and other surfaces can also be created at any time during the setup process

- Enter coordinates (in meters) of (2,1,0) as shown in the upper right and click Create
  - The point tool is displayed in the graphics window at the location of the coordinates entered in the panel
  - The point tool's handles can be used to set or change the position in cases where the precise coordinates are not required
 - Hiding one of the symmetry planes can make it easier to view the location of the point tool
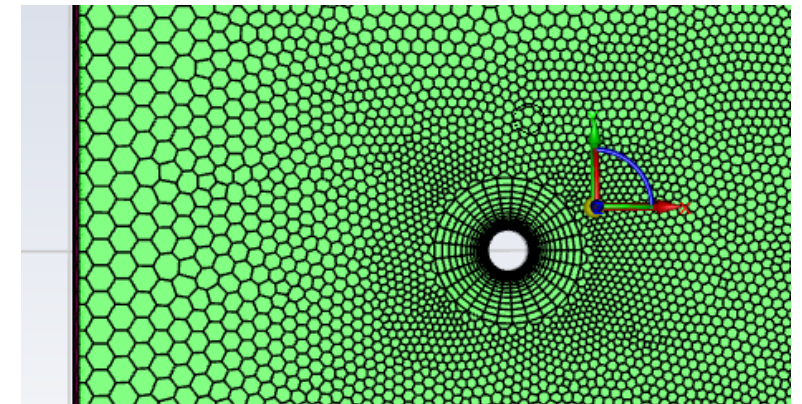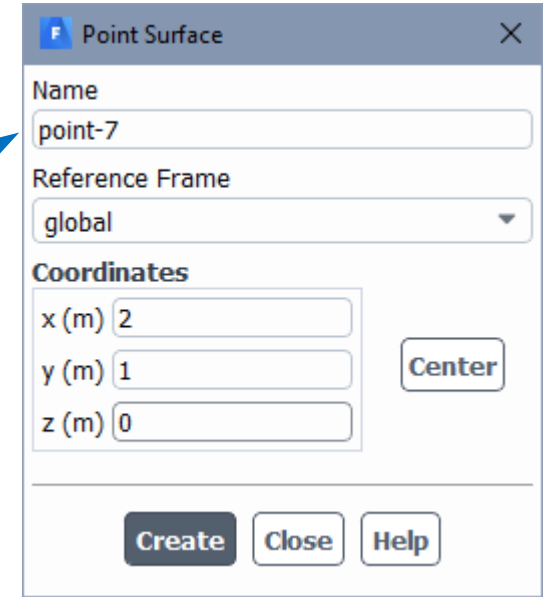
Physics: Solver and Models
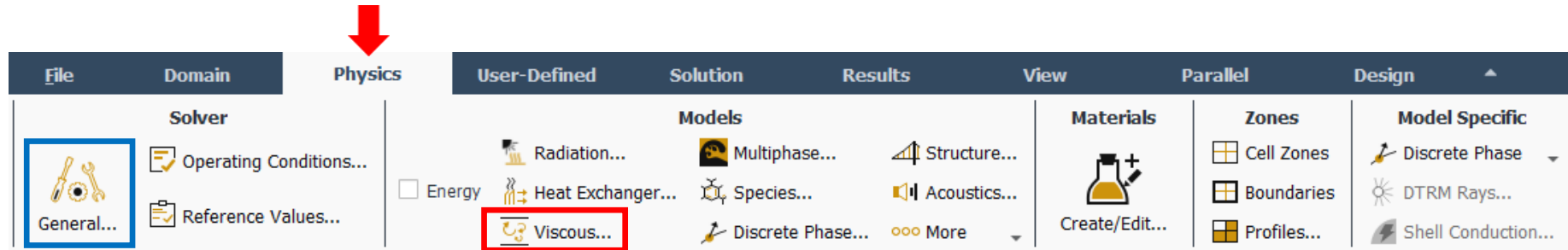
- In the Physics tab, the General task page should be displayed when Fluent opens in solution mode, if not click General

- Change from Steady to Transient

- The Reynolds number based on the cylinder diameter is 100, so the flow is laminar

  - Click Viscous under Models, select Laminar and then click OK
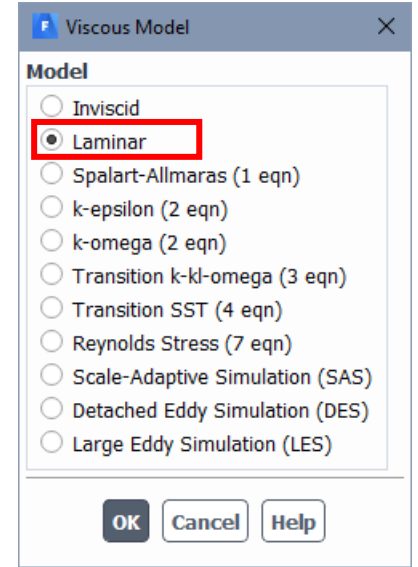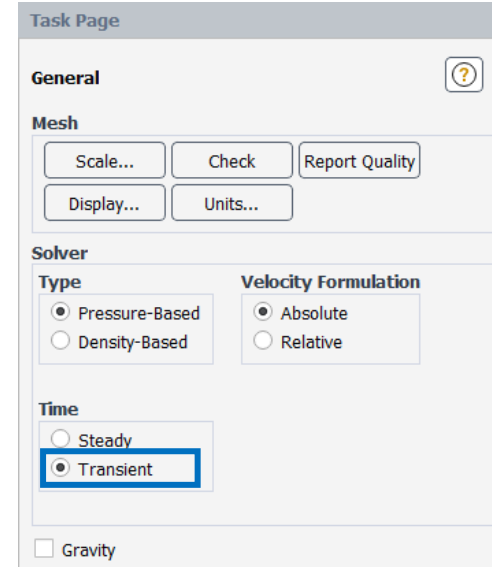
Physics: Materials
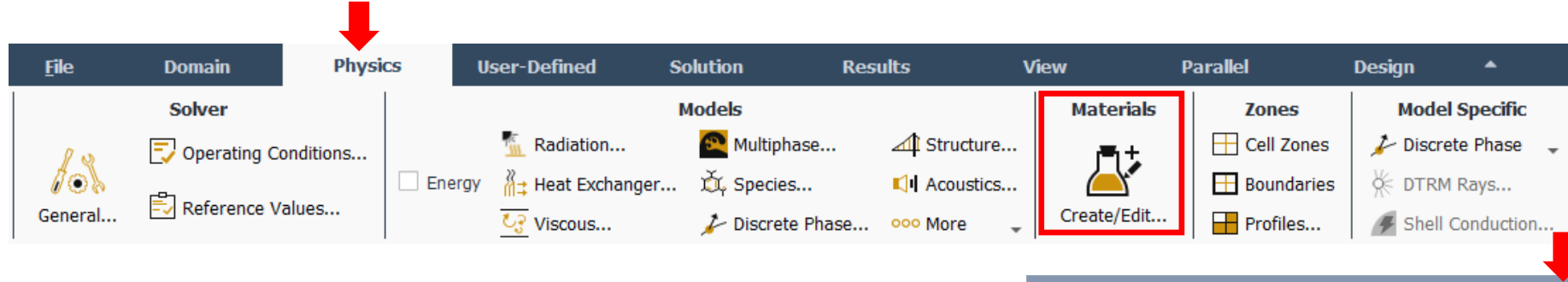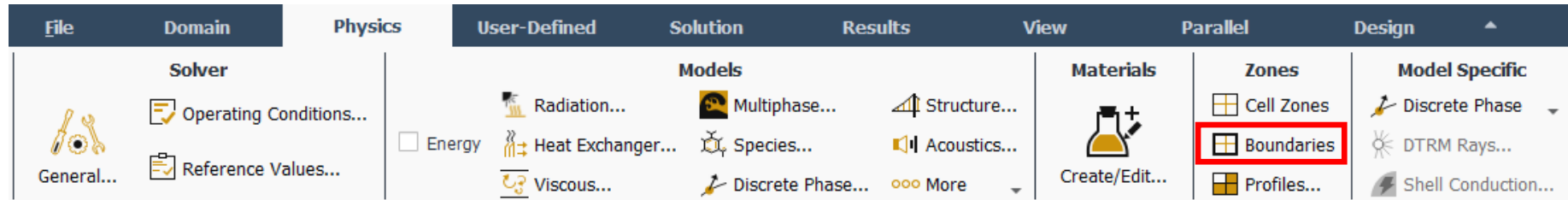

- Click Materials and set the density to 1.0 kg/m3 and the viscosity to 0.01 kg/m-s

- Click Change/Create to ensure the new values are registered
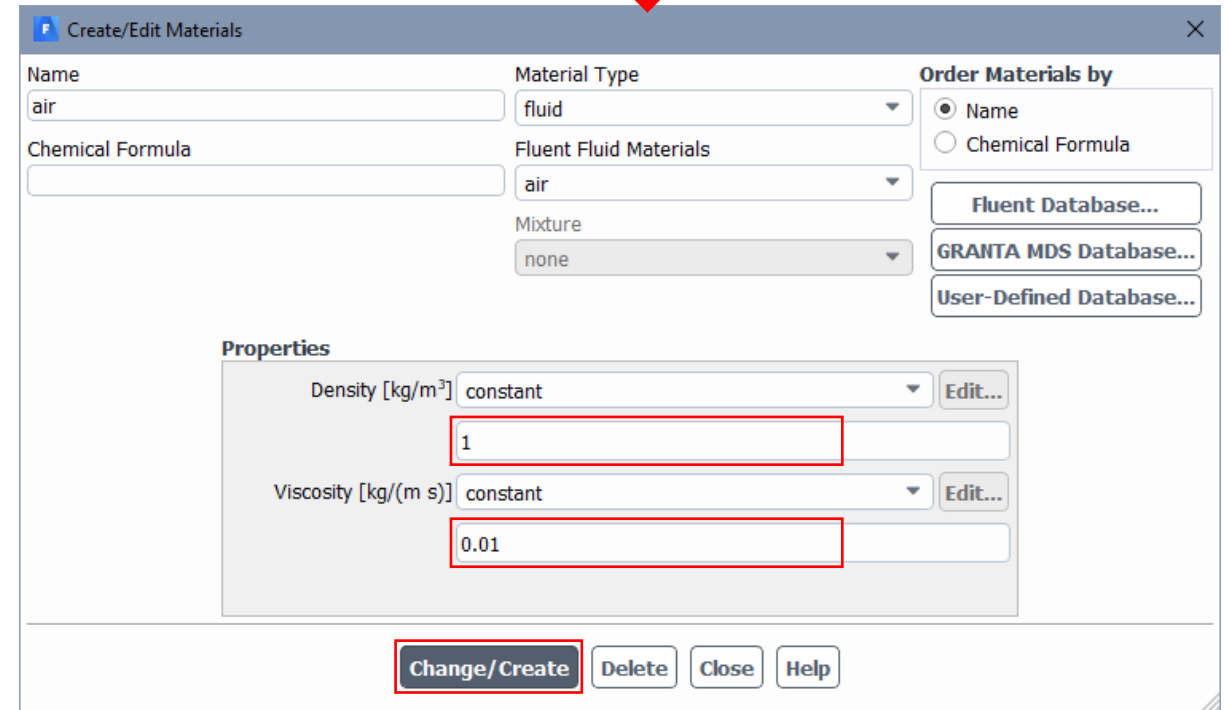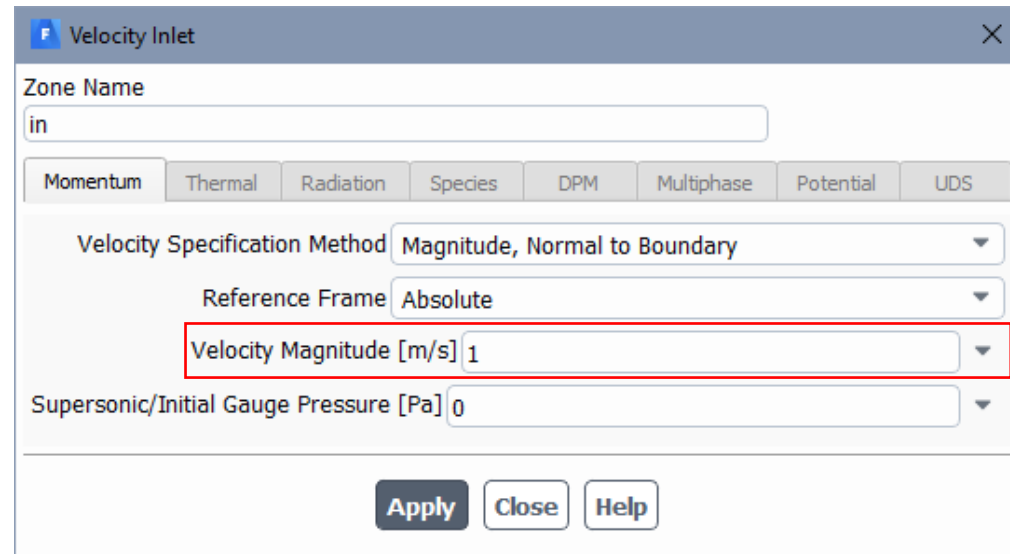
Using these values, the Reynolds number will be equal to 100.

Physics: Cell Zones and Boundary Conditions

- Materials definitions are complete so proceed to Zones
- The default material of air will be used so no need to set any cell zone conditions
- Click Boundaries and open the boundary conditions panel for the inlet (name = "in")
- Enter a value of 1 m/s normal to boundary

Physics: Other Boundary Conditions
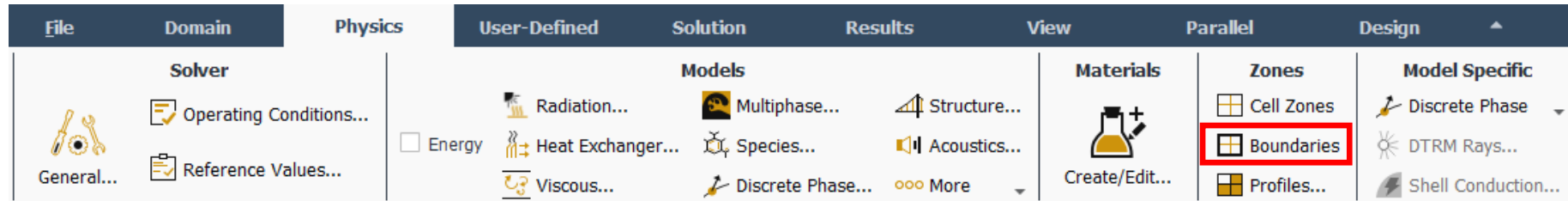
- The outlet ("out") is at zero gauge pressure
  - The default setting at a pressure outlet is gauge pressure = 0 so no need to open the panel
- "cylinder" is a wall, so no action is needed
- "sym-1", "sym-2", "sym-a", "sym-b" are symmetry boundaries
  - No input required for symmetry boundaries
- Boundary conditions are now complete and so too all entries in the Physics tab

Transient Settings: Time and Solution Methods

- Go to the Solution tab and click Methods
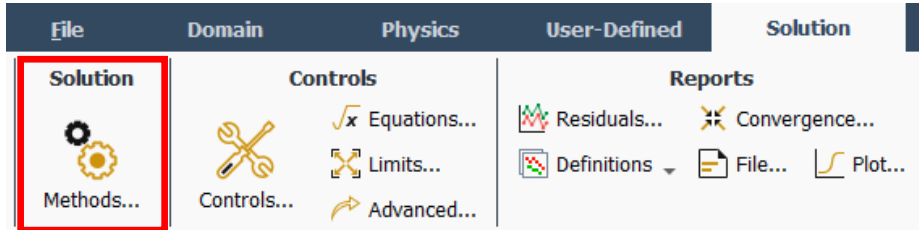
- Enter solution methods as shown to the right
  - Pressure-Velocity Coupling Scheme = *PISO*
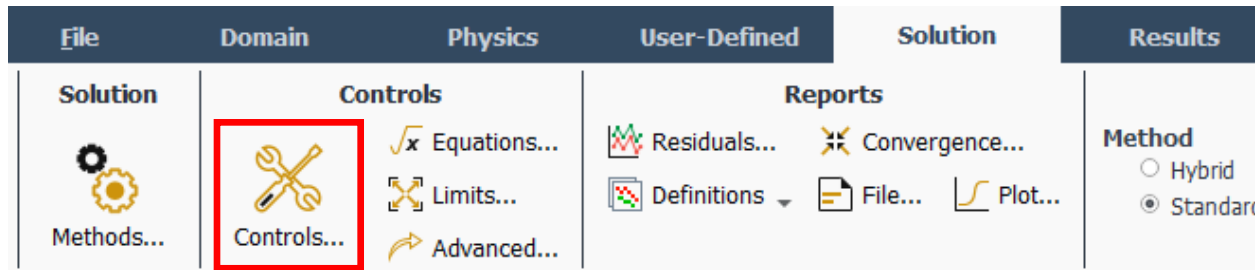 - Transient Formulation = *Bounded Second Order Implicit*
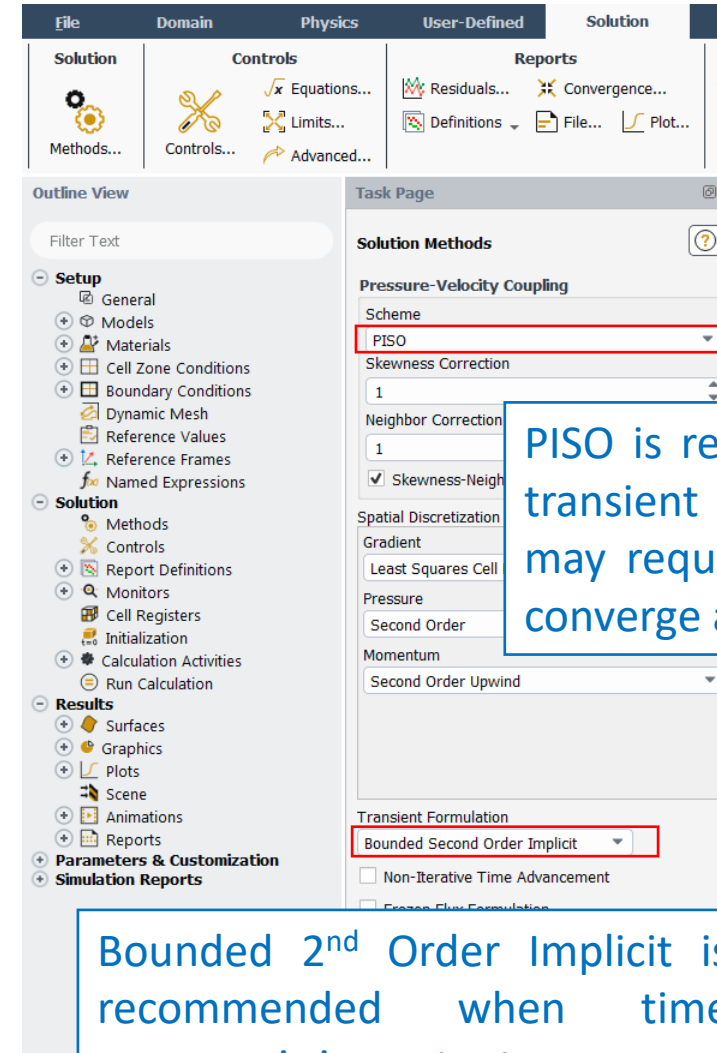
PISO is recommended for most transient cases because SIMPLE may require more iterations to converge at each time step.

Bounded 2nd Order Implicit is recommended when time accuracy is important.

Transient Settings: Solution Controls

- Next click Controls in the Solution tab


- Change the Under-Relaxation Factor for *Pressure* to 0.7

The Fluent User's Guide recommends using values of 1.0 for the pressure and momentum under-relaxation factors when using PISO. However in many cases that proves to be too high, causing the solution to diverge. In many cases, a range of 0.7 to 0.9 will often work. Here a value of 0.7 is selected because it is more likely to be stable. Trial and error can be used to find the highest possible value that will work in any given case, because in general a higher value means the solution will converge in fewer iterations per time step, but for small cases such as this, the benefits may not justify the additional upfront effort.
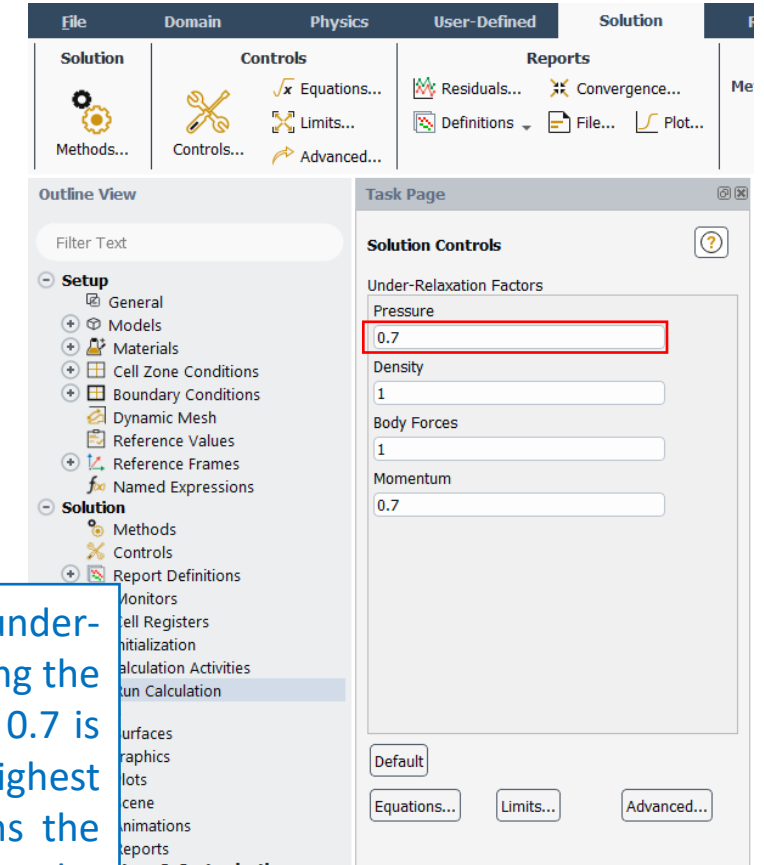
Solution: Report Definition
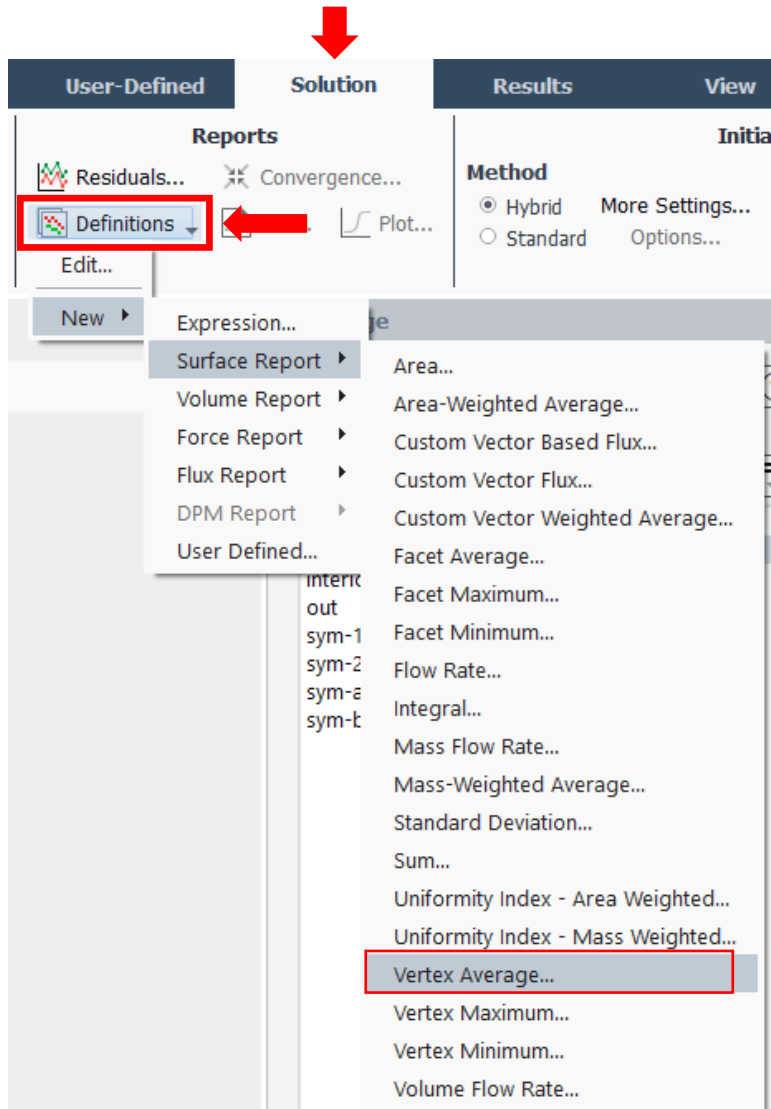
- In the Solution tab, click on Definitions in the Reports section and choose New > Surface Report > Vertex Average
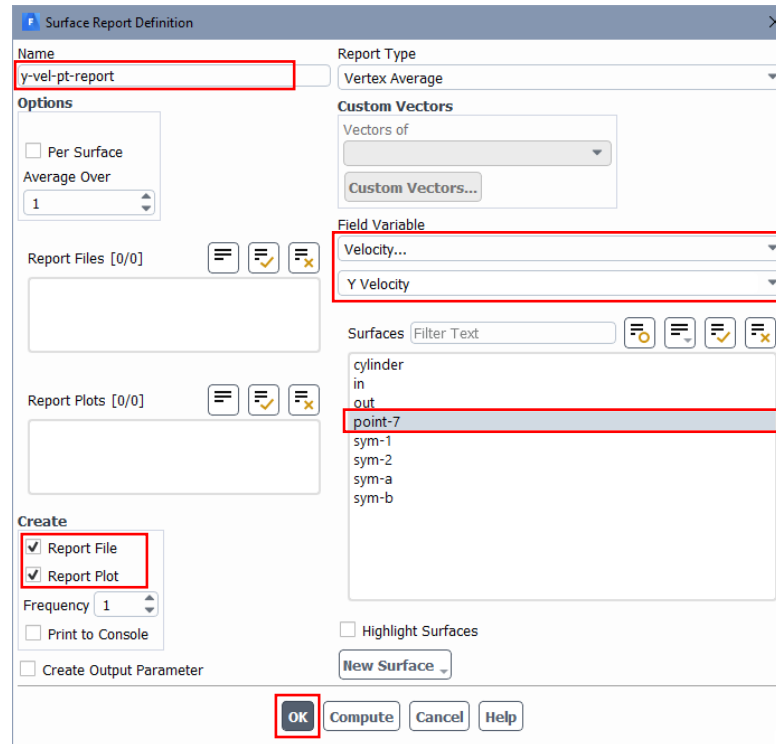
Enter the following in the definition panel, then click OK:
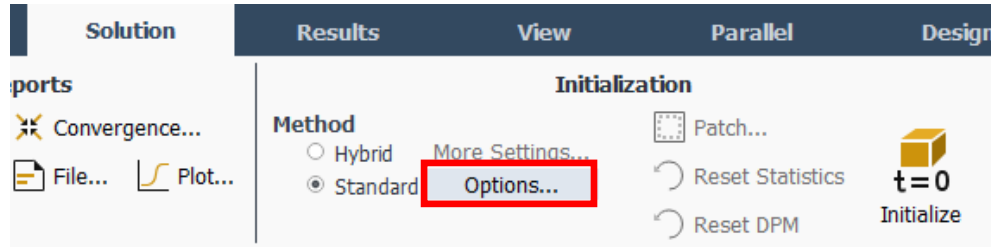
Name = **y-vel-pt-report**
Surfaces = **point-7**
**Variable =** **Y Velocity**
Report File = **check**
Report Plot = **check**

Solution: Initialization

- In the Solution tab, click Options… in the Initialization group

- In the Task Page for Solution Initialization, select the inlet under Compute From, and then click Initialize

  - Selecting the boundary can be a convenient way to automatically fill the initial values fields using information from the boundary conditions
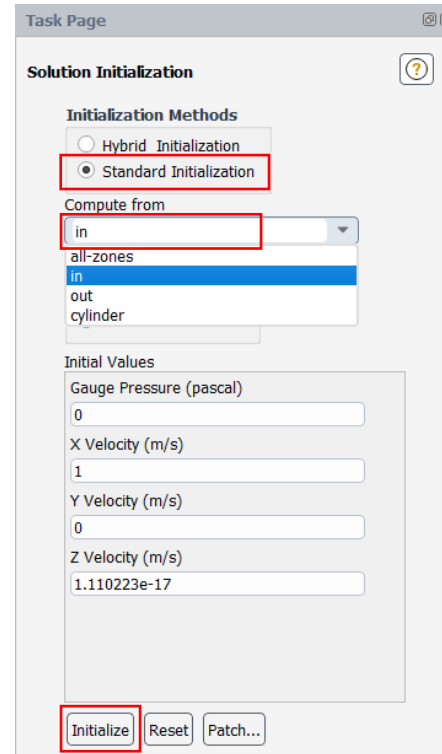

While hybrid initialization is often the best choice for steady state problems, it is not always suitable for transient problems, where the initial condition is part of the problem definition. In this case, it will be easier for the solution to achieve unsteady behaviour by using a uniform velocity field, and then patching a perturbation, which we will do in the next slides

# Solution Initialization: Cell Register for Patching

- In order to spur the onset of unsteady flow in the numerical solution, a perturbation will be added to the initial velocity field downstream of the cylinder

- First the region where the perturbation will be applied must be defined
  - This is done using Cell Registers

- Right click Cell Registers and select New > Region

- Enter the name as "region_patch"

- Enter the Input Coordinates as seen on the right and click Save/Display

The center of the cylinder is at x=0, so 0 to 15m is everything downstream from the center. The upper and lower boundaries are at y=-10m and y=10m. A 1m buffer is given around these boundaries because it is desired for the initial flow field to remain parallel to these boundaries. The z values encompass the entire domain extents in the z-direction.
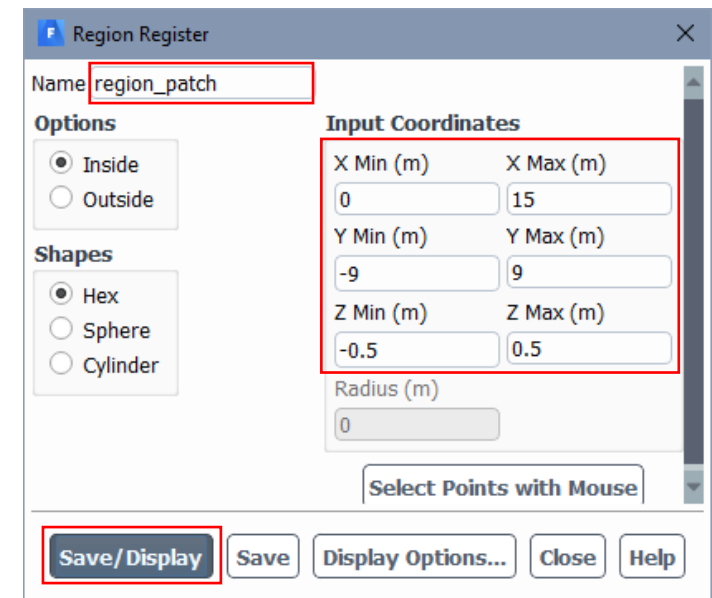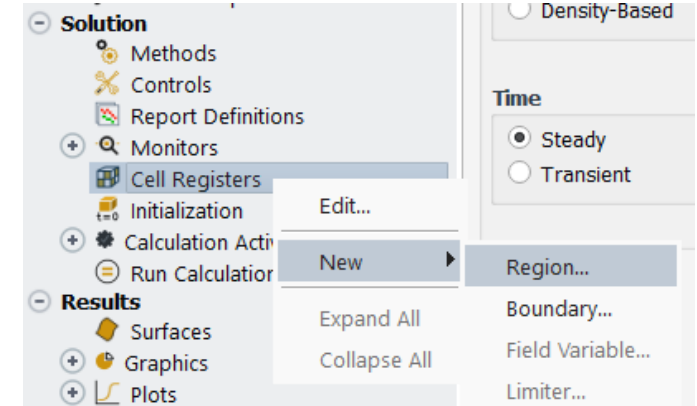
Solution: Patch

- Select Patch in the Initialization group of the Solution tab
  - The patch button in the solution initialization task page can also be used: either does the same thing
- In the Patch panel, select Y Velocity under variable and region_patch under Registers to Patch
- Enter a value of 0.18 m/s and click Patch
  - This represents a flow angle of ~10°
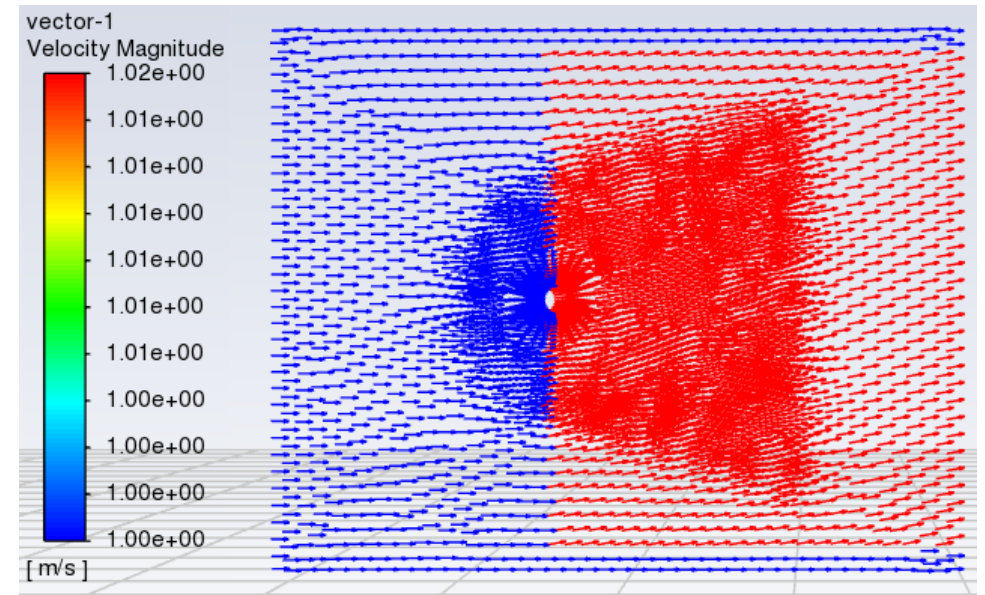 - We will visualize this on the next slide
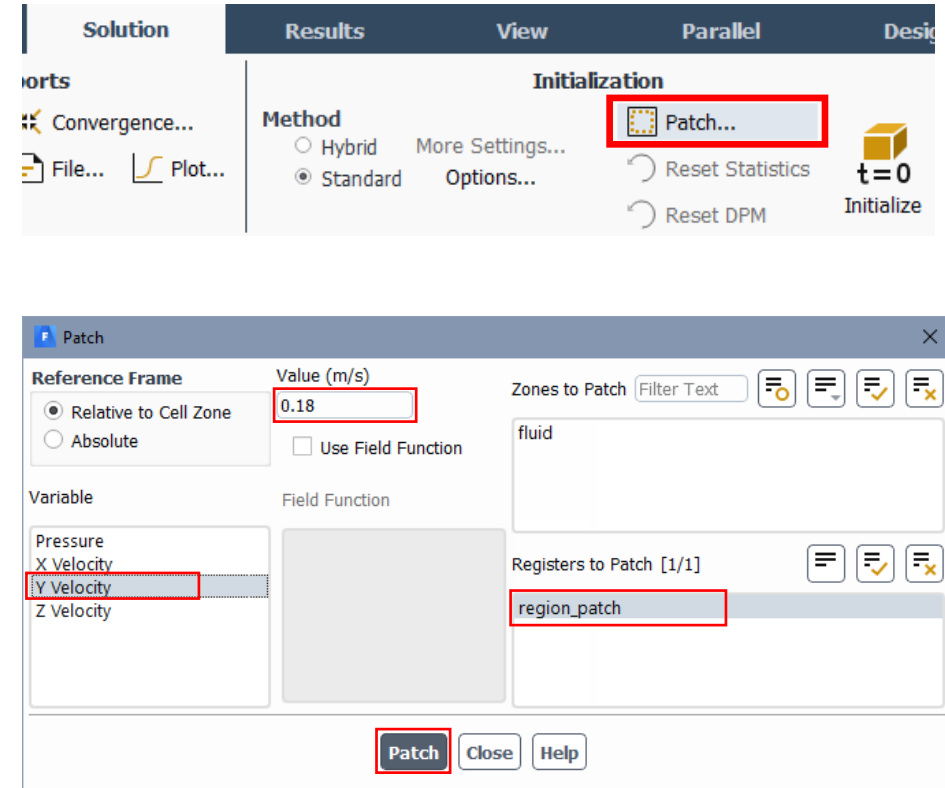
Patched Initial Condition

- In the Results tab, select Vectors > new

- Display the vectors on the "sym-a" surface
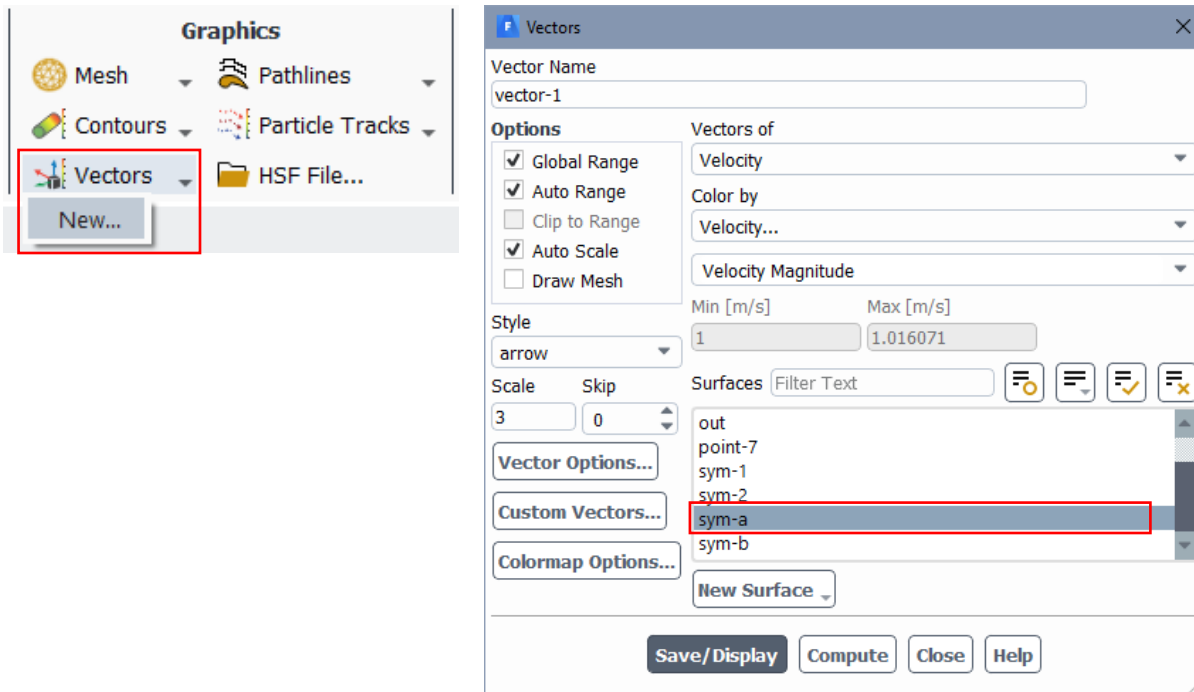
When patching an initial condition for an unsteady calculation, it is strongly recommended to use vectors and/or contour plots to verify the patch has achieved the intended result before beginning the calculation.


In this case, the artificial, asymmetric initial flow field is needed to ensure that the numerical calculation will evolve into a realistic, oscillating flow. Initialization with a symmetrical flow field would probably lead to unphysical symmetrical steady flow behaviour.
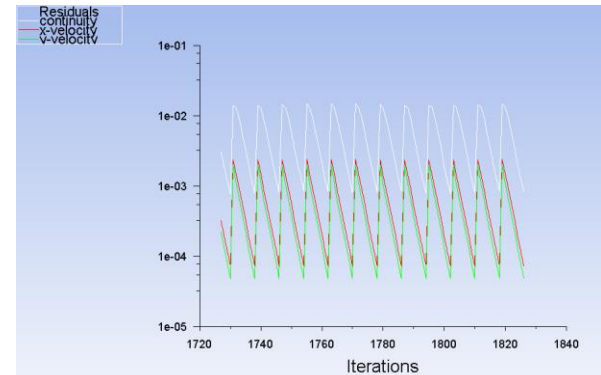
Solution: Residual Display Setting
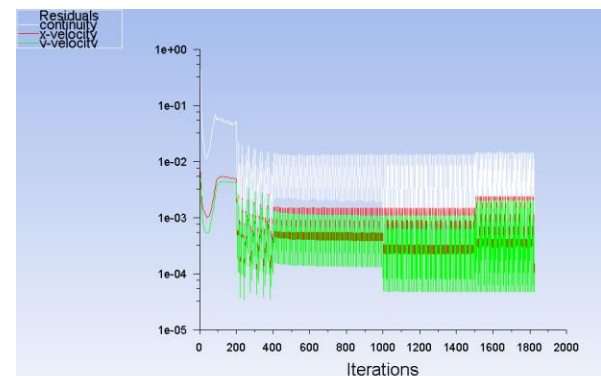
- Click Residuals in the Solution tab
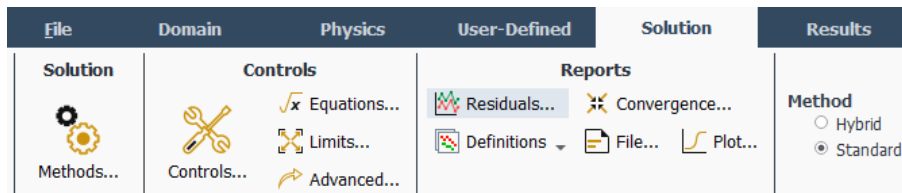
- Change Iterations to Plot to 100
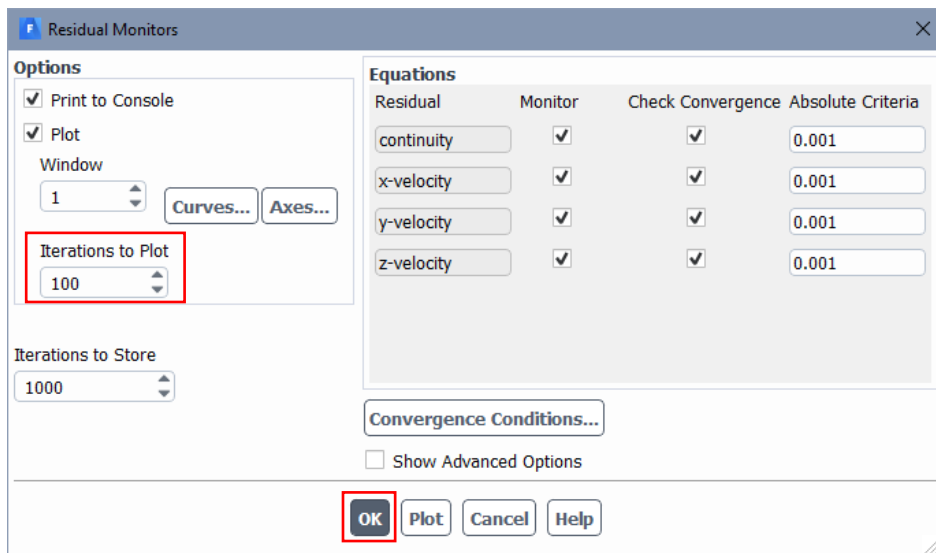
- Click OK

Iterations to Plot = 100

Iterations to Plot = 1000 (default)

It is not required to change the value of iterations to plot. Some people prefer to reduce this number in order to be able to more clearly see what is happening during each time step (at the expense of seeing fewer time steps in the plot at the same time) as in the upper figure. Others prefer the plot shown below. The purpose of doing this here is to make people aware that both ways are possible so they can choose which they prefer.

Solving: Time Step Size Discussion

- Identification of a suitable time step is an important part of every transient simulation
- One possible method would be to do a hand calculation to see how long it takes for the flow to pass through a typical grid cell. Run this, and check that convergence occurs in less than 20 iterations per time step.

- Another approach is to determine the characteristic response of the system. By performing a literature search, it is identified that at this Reynolds number (Re = 100) the Strouhal number will be approximately 0.165. From this it is possible to estimate the period of the oscillations (D = 1 m, V = 1 m/s):

$$St = \frac{fD}{V} \Rightarrow period = \frac{1}{f} = \frac{D}{St\,V} = 6.06s$$

- About 60 time steps per oscillation cycle are desired, so the time step size should be 0.1 s (≈ 6.06 s / 60 time steps) and we will solve for two cycles, so the number of time steps will be 120

Solution: Time Step Size

- In the Run Calculation group of the Solution tab enter 0.1 seconds for the time step size and 120 for the number of time steps.
- Keep the default value of 20 Max Iterations/Time Step under the Run Calculation task page
- Save the case and data files, then calculate
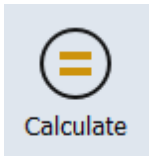
Saving the case and data files at the beginning of the transient calculation is recommended because in case any convergence problems occur, you can simply reload the data file and try running the case again with modified solver settings and controls
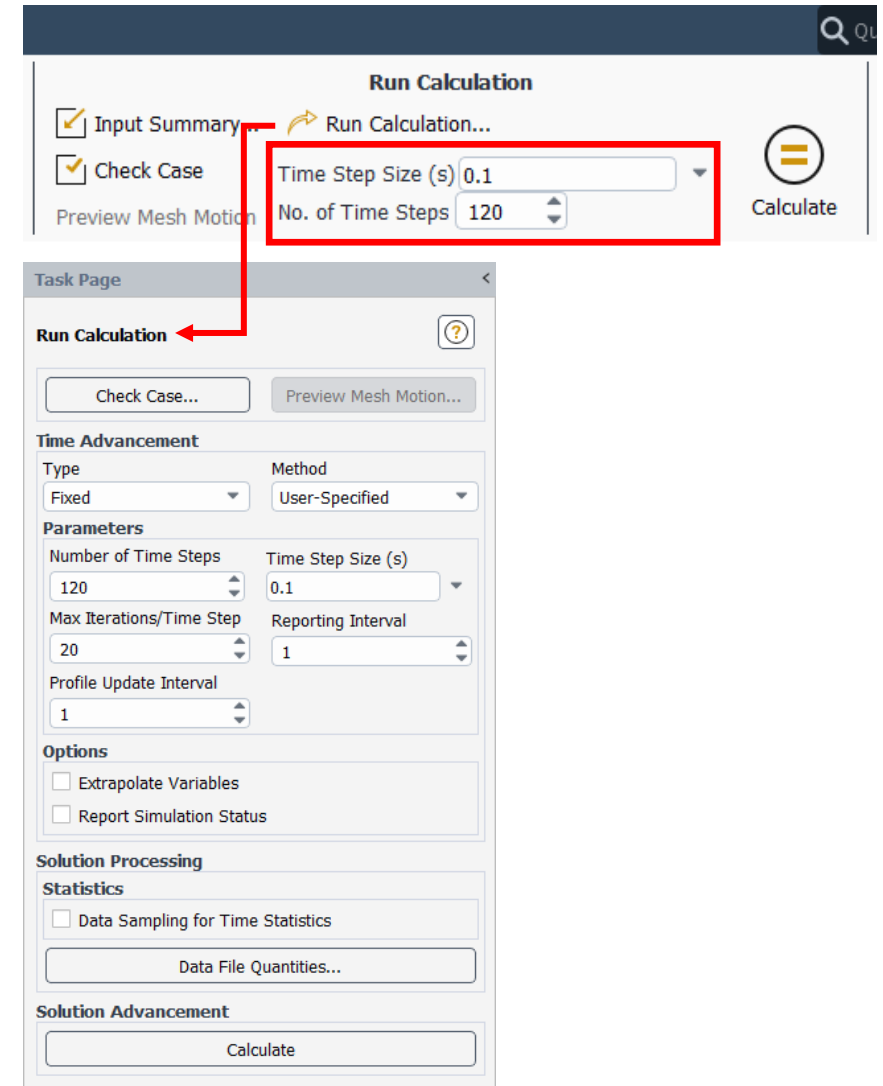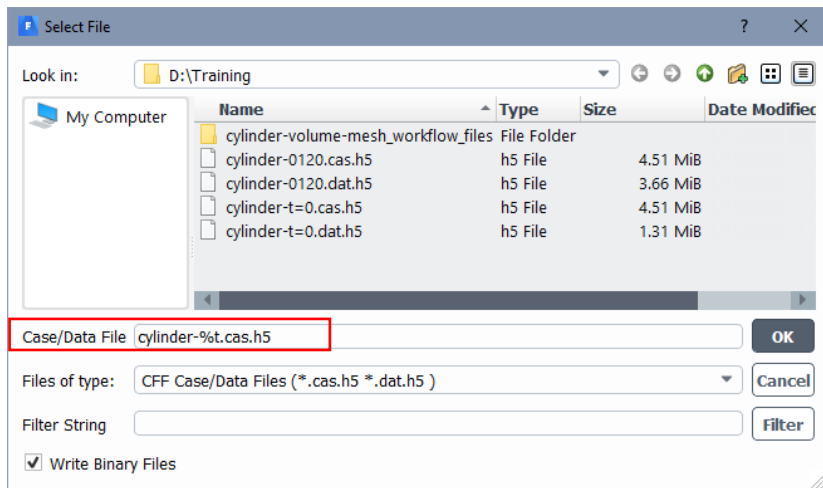
Solution: Discussion

- Save the case and data file again after the time steps have been completed
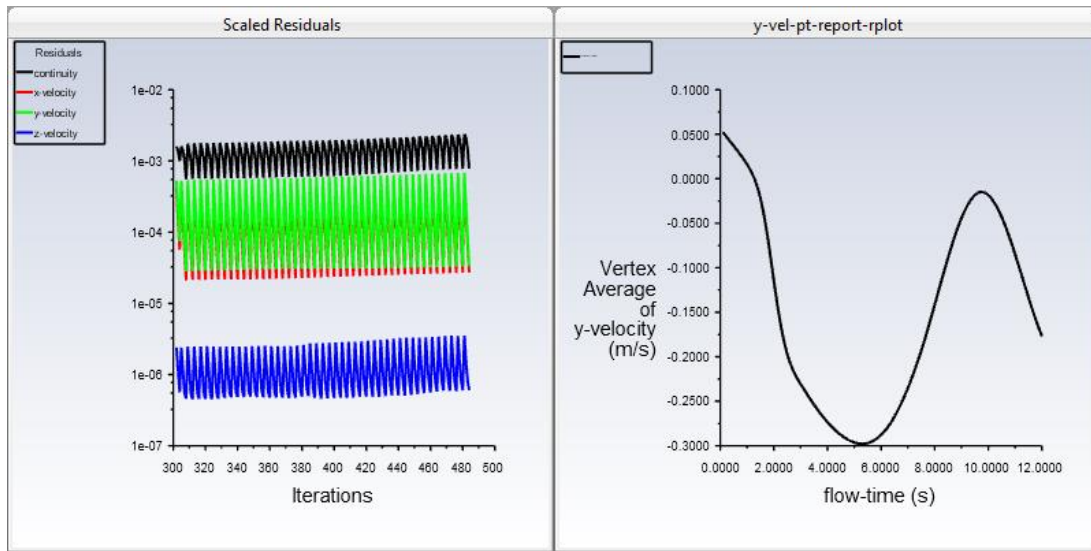 - Add "-%t" to the file name to append the number of the current time step

Including %t in the file name will cause Fluent to insert the number of the current time step, in this case 120, so the files will be written as "cylinder-unsteady-0120.cas.h5"

Solution: Discussion

- The residuals and report plots are shown here

The solution is converging in a few iterations each time step. If interested, you can check the exact number by examining the output in the console.

Of greater significance is the report plot. In transient cases, it often takes a certain amount of simulated time for the effects of the initial condition (which is generally just an educated guess) to go away. After a long enough period of simulated time, all of the cycles should be identical. The report plot here shows that this has not yet been achieved.

Solution: Additional Time Steps

- Change the number of time steps to 600 (e.g. approximately 10 additional cycles)
  - This will take around 10 minutes – if you don't want to wait, read the provided case and data files "cylinder-0720.cas.h5"
- Cyclic behavior becomes more regular after first 5 or 6 cycles
- Vector display shows the expected vortex shedding behavior
- Save the case and data files as "cylinder--%t.cas.h5", similar to the instruction on previous slide
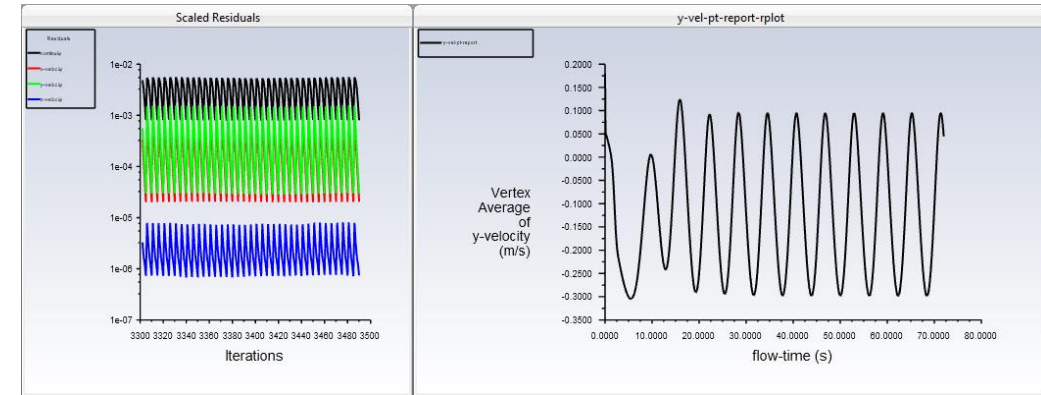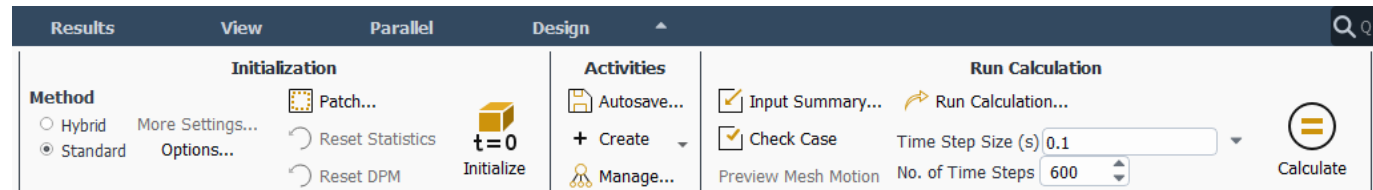
After enough time the effects of the artificial initial condition wash out of the solution and oscillations become periodic in time.
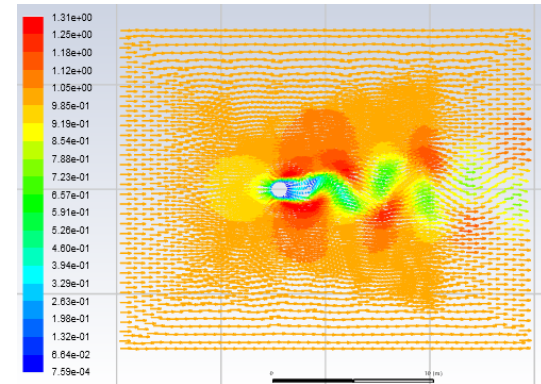
# Non-Iterative Time Advancement (NITA)

- Non-Iterative Time Advancement, or NITA schemes are algorithms used to speed up the transient solution process
  - About twice as fast as the ITA scheme
  - NITA can sometimes require smaller time steps than ITA or be harder to converge
- The goal of this step of the workshop is to expose you to NITA and get a feel for what it is and how it works

Additional Goals

- In this section of the workshop you will also learn
  - How to use scenes to create solution animations
  - How to save intermediate data files
  - How to use fast Fourier transform (FFT) processing to identify the vortex shedding frequency

Results: Vectors, Contours, and Mesh Objects

- In the Results tab, create new mesh and contour objects using the settings shown below
  - The vectors created in slide 27 will be used too
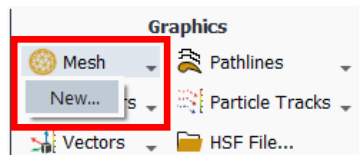
The mesh surfaces form an outline of the computational domain, which will produce nicer looking scene objects on the next slide
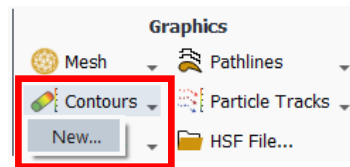
Note which boxes are ticked under Options and manual range for min and max values
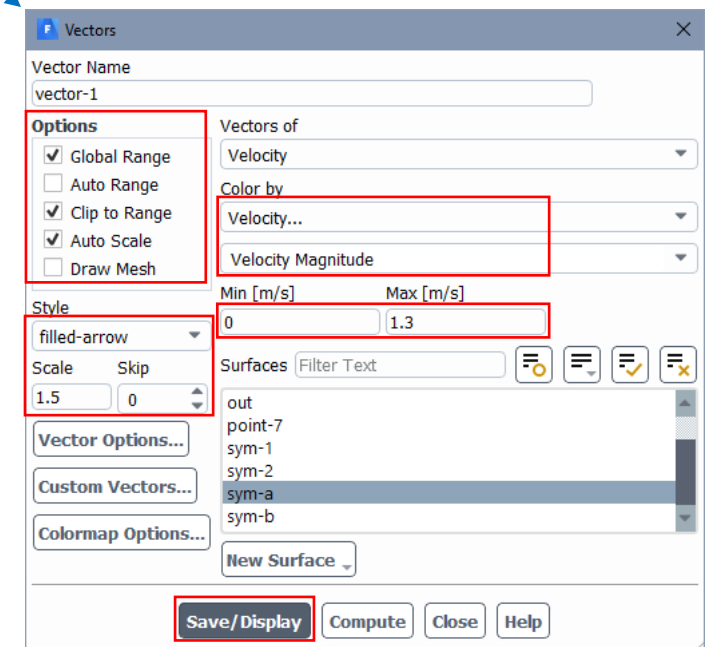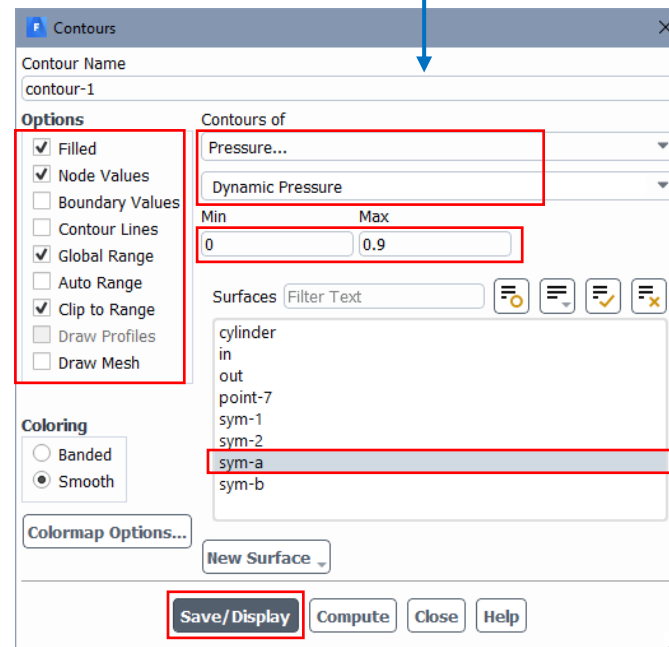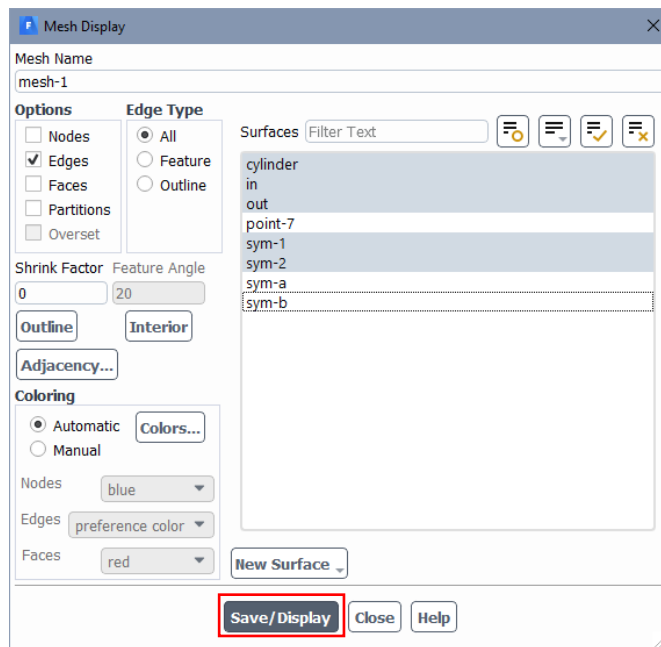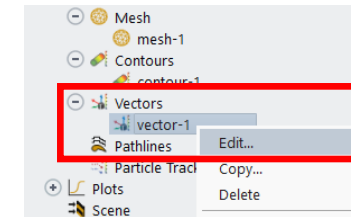
Results: Scene Objects

- After creating the objects on the previous slide, they will be visible in the Results branch of the Outline View tree
- Right click on Scene in the tree, select New, and create the scene objects shown below

Scene objects are listed in the tree after being created.

Solution: Define Solution Animations

- Solution Animations are defined under Calculation Activities in the Solution Branch of the Tree
- Right click, select New and define the animations shown below
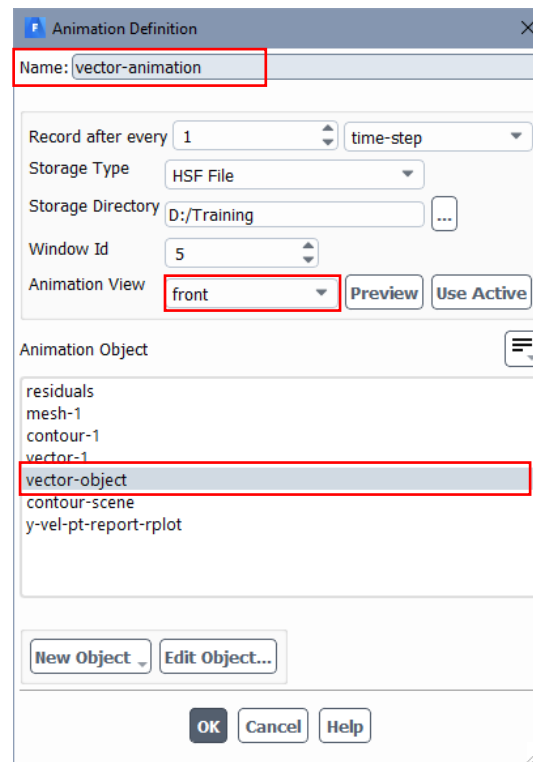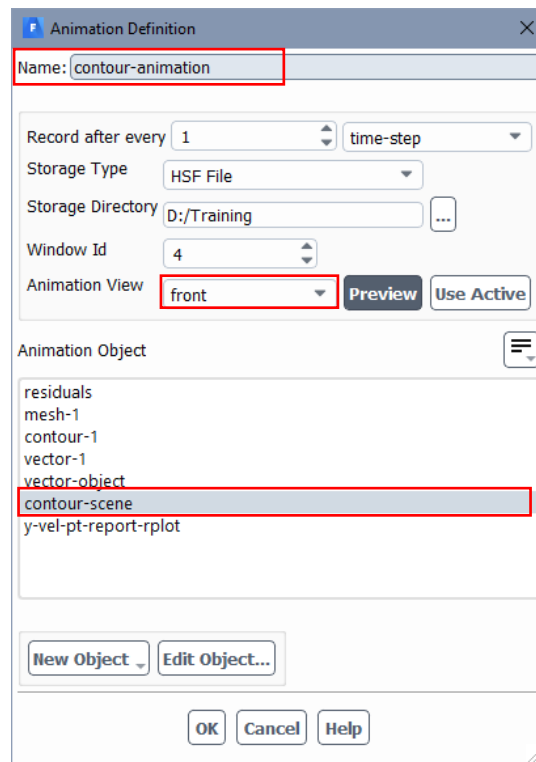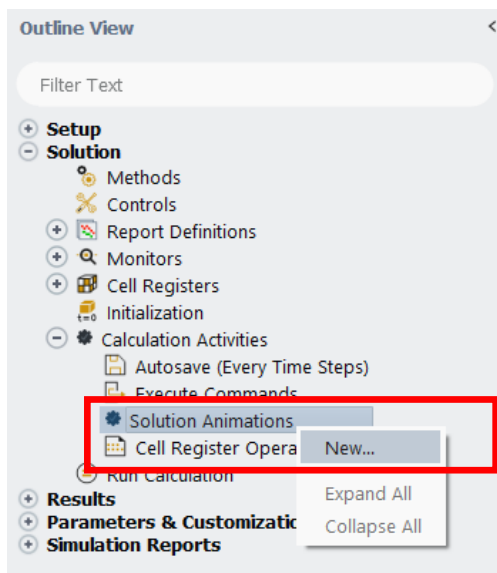
Descriptive names can be helpful, but are not required.

For transient simulations, it is very important to choose Time Step, not Iteration.

HSF File is the preferred storage type because it allows for later manipulation of the view.

It is recommended to use scenes for animation of contours or vectors (e.g. select "contour scene" instead of "contour-1") because it allows a greater level of control over any grid objects that might also be included in the animation.

Solution: Autosave

- When performing a transient simulation, Fluent does not store the entire time history of the solution
  - It only knows the solution at the current moment in time
- If it is desired to post-process data from any intermediate times, data files must be saved at those times
- Autosave allows this to be done automatically at prescribed intervals
- Click Autosave and set the interval to 5
  - This is a small case so a relatively low value (lower = more files) can be used
  - For large 3D models, higher intervals may be desired due to disk storage considerations
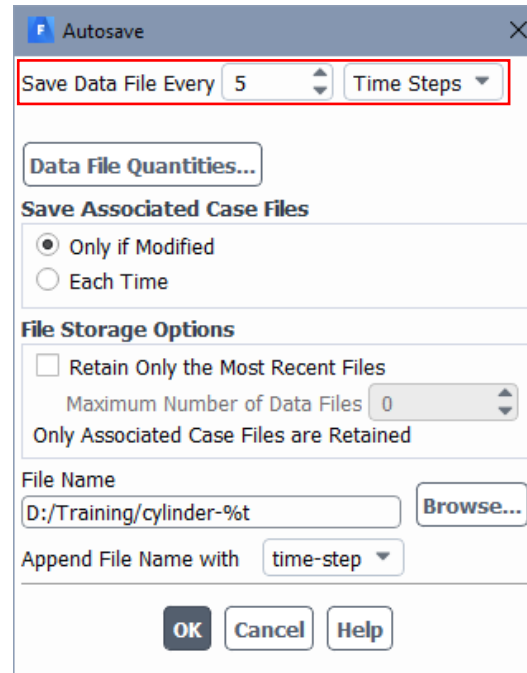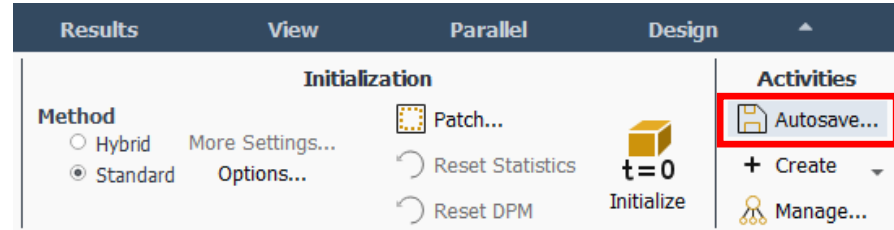
It is generally only necessary to save case files if they have been modified since the last data file.

The option to retain only the most recent files is a useful way to keep a small number of backups (thus limiting the amount of required disk space), so that if anything goes wrong, you can restart from the most recent backup instead of having to recalculate the entire history.

Solution: Activate NITA

- Click Methods in the Solution Tab
- In the Solution Methods Task Page, select Non-Iterative Time Advancement and Fractional Step

> - **NITA is an algorithm used to speed up the transient solution process**
>   - NITA runs about twice as fast as the ITA scheme
>
> - **Two flavors of NITA schemes available**
>   - PISO (NITA/PISO)
>   - Fractional-step method (NITA/FSM)
>   About 20% cheaper than NITA/PISO on a per time-step basis, hence its use here

- You are now ready to start calculating but first save case and data files as "cylinder-unsteady-nita-%t.cas.h5"
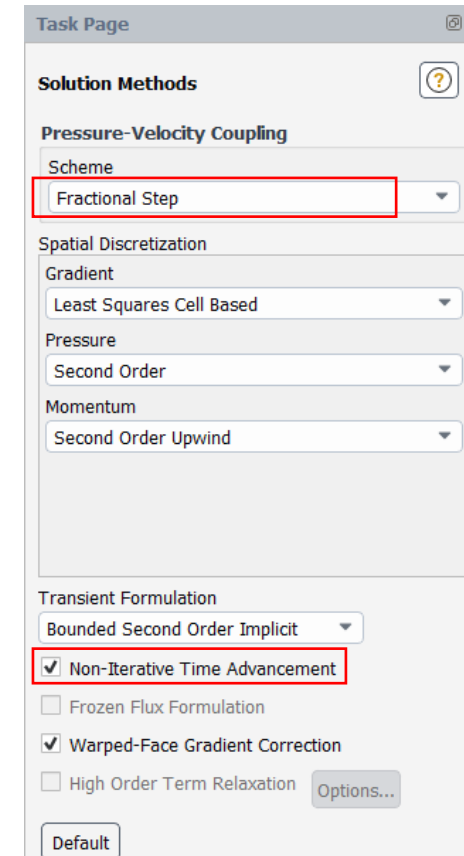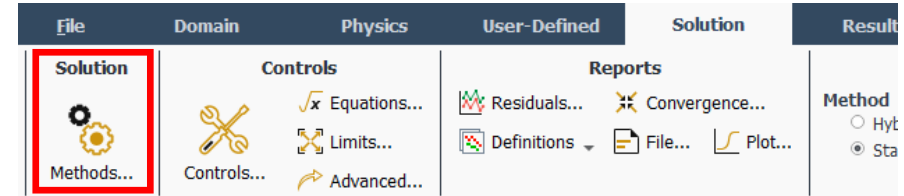
Solution: Calculate

- Keep 0.1 seconds for the time step size, change the number of time steps to 240 and click Calculate

- In addition to the usual residual and report plot output, you will see the scenes for the animations being updated each time step
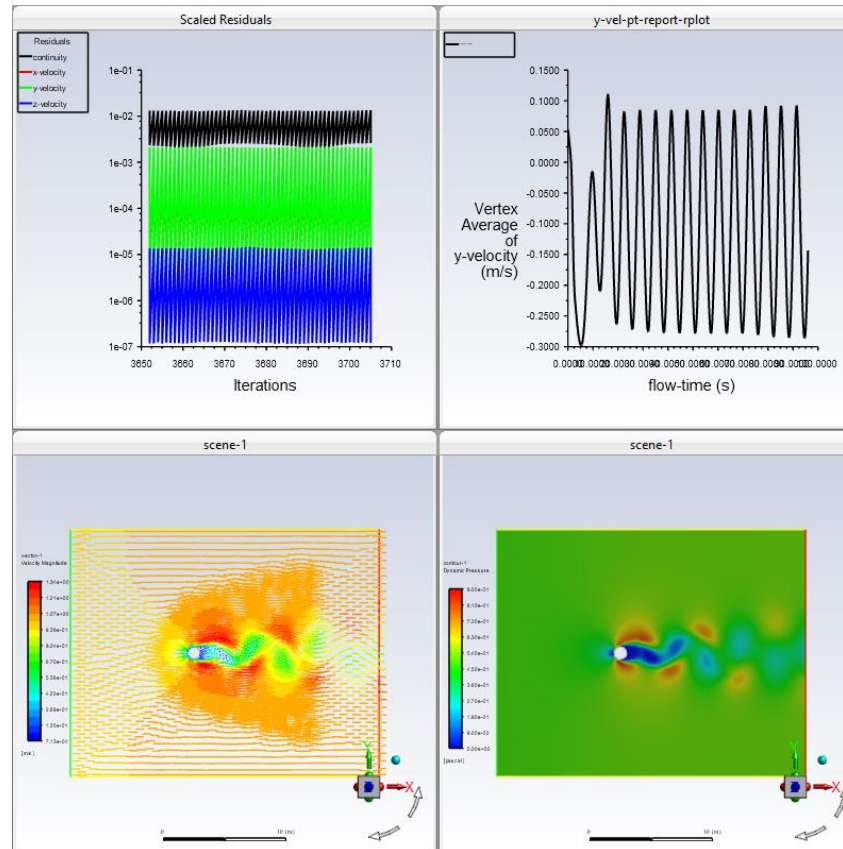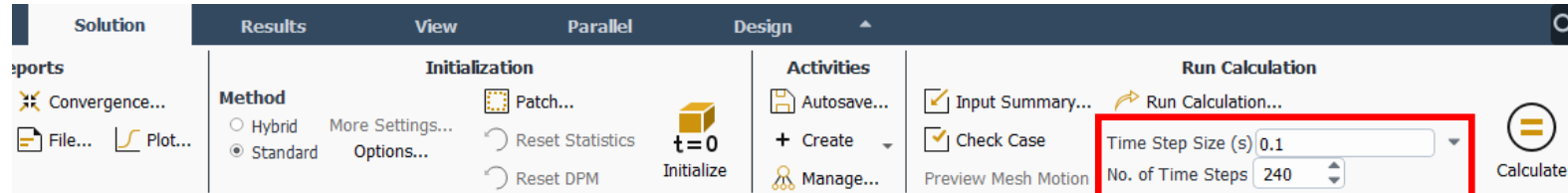
Results: Solution Playback

- Click on Solution Playback... in the Animation group
- Select the animation sequence for the contours and click the play button
- Next, unselect "Use Stored View" in the Playback panel, adjust the view in the graphics window and click the play button again
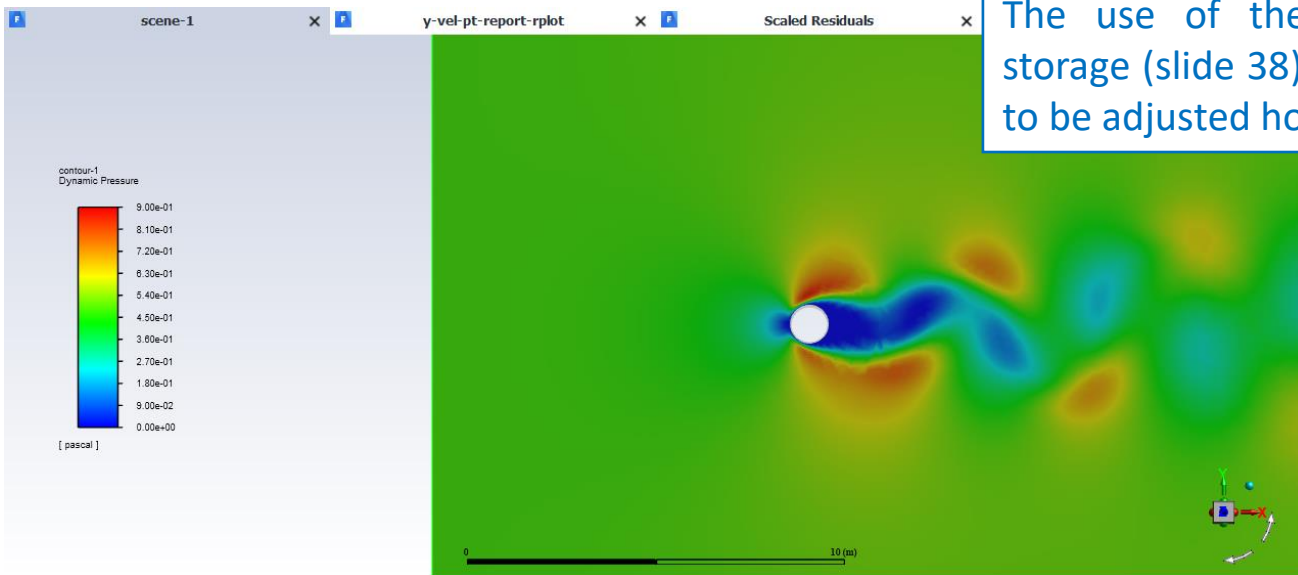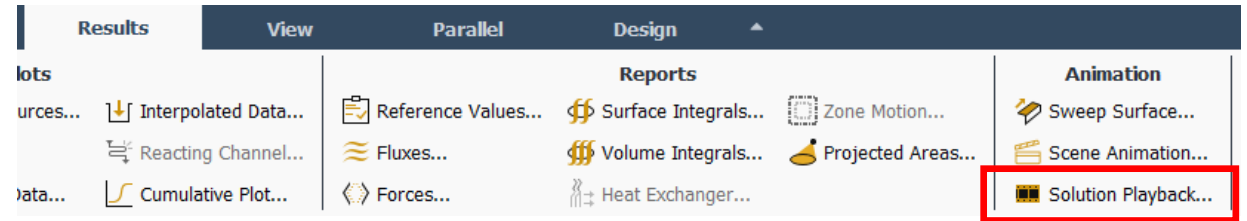

The use of the HSF Files for storage (slide 38) allows the view to be adjusted however desired.
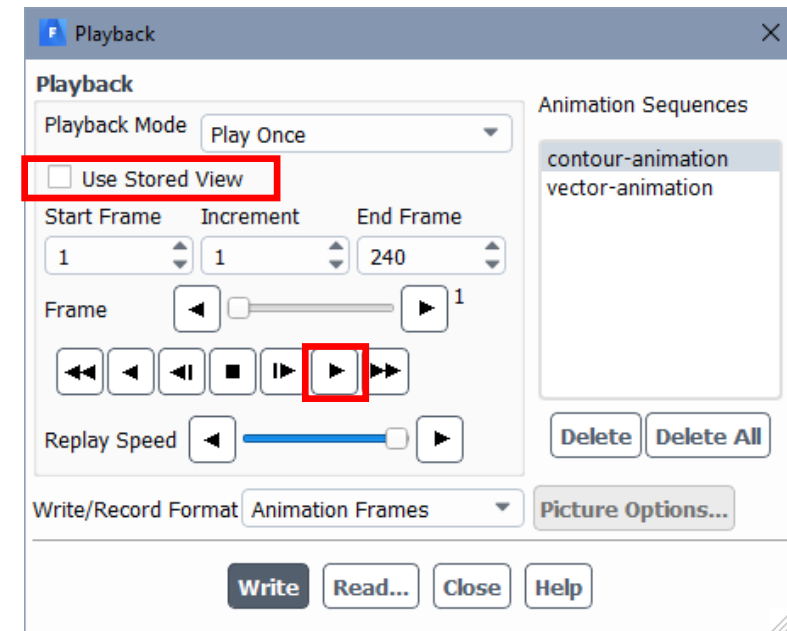
Results: Write Movie File

- After adjusting the view, change the Write/Record format to video
  - Deselect Use Stored View to capture the zoomed in region in the animation

- Click on Video Options…, select the desired video format and click Write
  - The default format is mp4 and it is only necessary to open Video Options if it is desired to use a different format

- Using the steps from this and the previous slide, also create a movie file for the vectors

- The video file will be written to the working directory, or if a different storage directory was entered in the Animation Definition panel, it will be written there

- The name of the file is taken from the animation sequence name, so here the file will be written as "contour-animation.mp4"
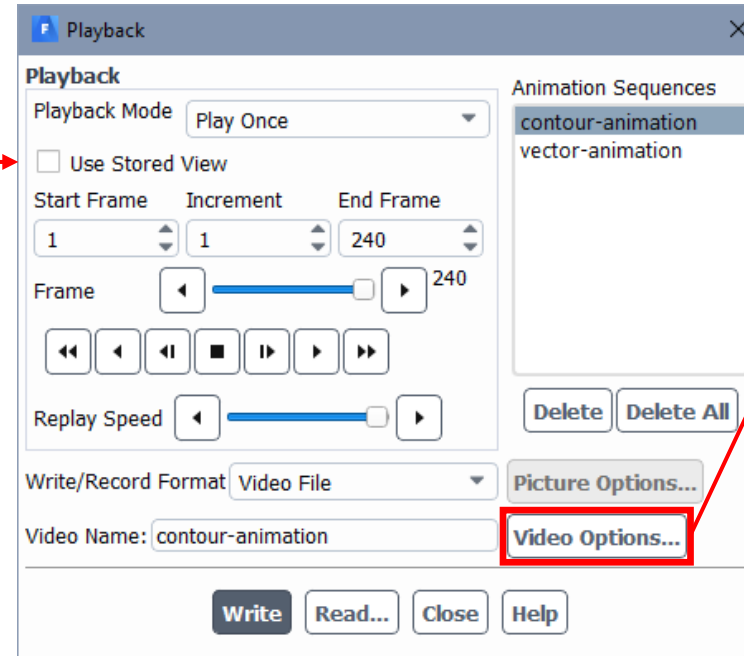
Results: FFT

- The FFT feature can be applied to the report file output to identify the vortex shedding frequency
- Click FFT, then click on Load Input File and select the file y-vel-pt-report-rfile.out
  - Or whatever name was entered when the Report File was defined
- Click the Plot/Modify Input Signal button to specify which data will be processed
  - Here we want to only use the last 240 time steps, as processing data from too close to the artificial initial condition, before a realistic vortex shedding process was established in the numerical solution, could potentially result in errors
- Select Clip to Range and set the Min value of X Axis Range to 72.1
  - 72.1 is the first time step calculated from slide 41
- Click Apply/Plot and close
  - The selected range is plotted in the graphics window
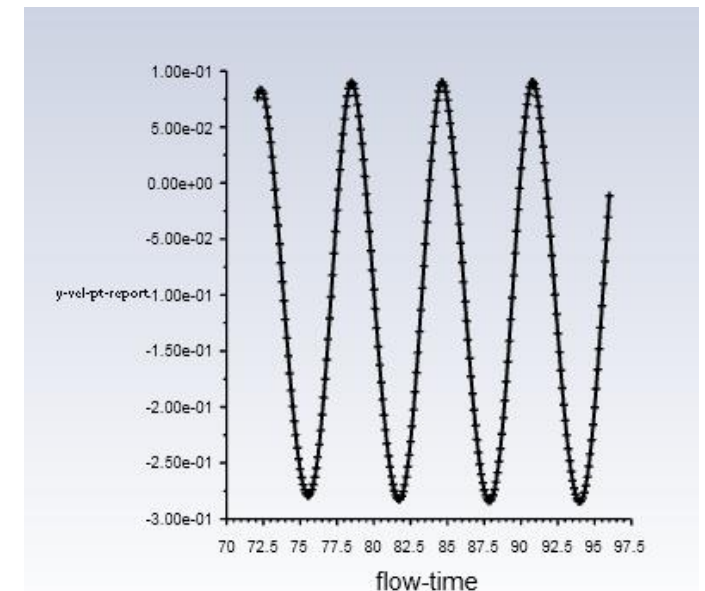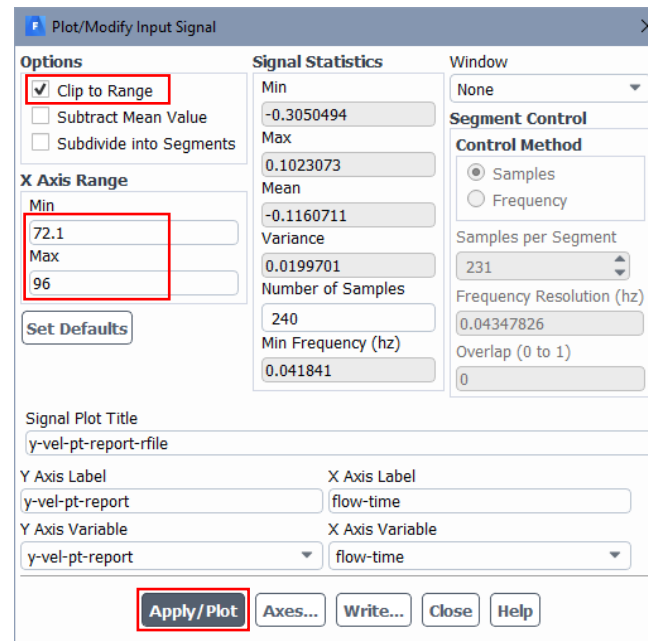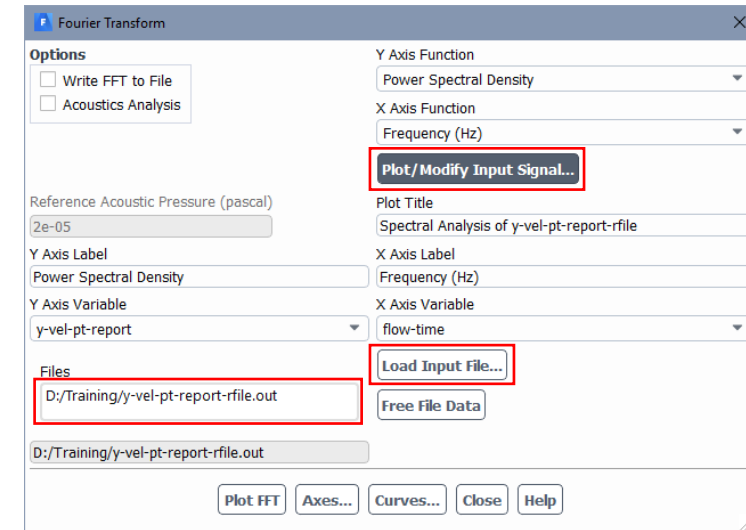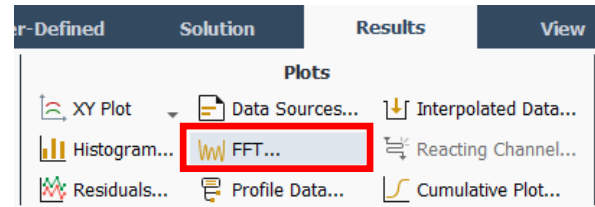
Results: FFT

- Plot the FFT
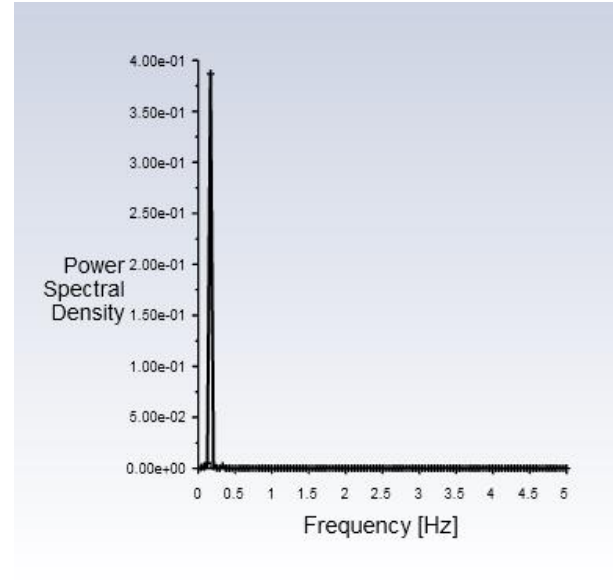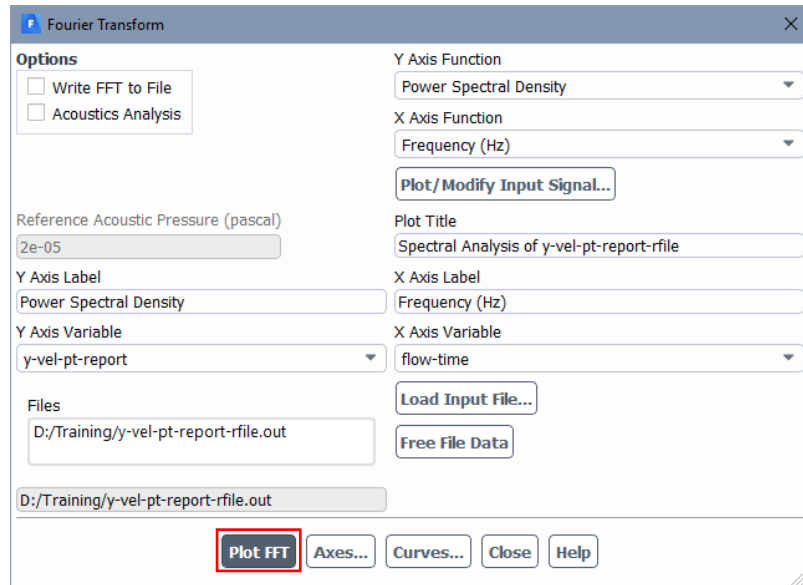
- Go to the next slide for discussion

# Results: Using FFT to Identify Vortex Shedding Frequency

- Most of the frequency range here is uninteresting, so click the Axes button, and set the range manually as shown, e.g. unselect Auto Range as shown in the lower right image
- Click Apply and then Close to return to the Fourier Transform Panel
- Click Plot FFT to plot only the specified range
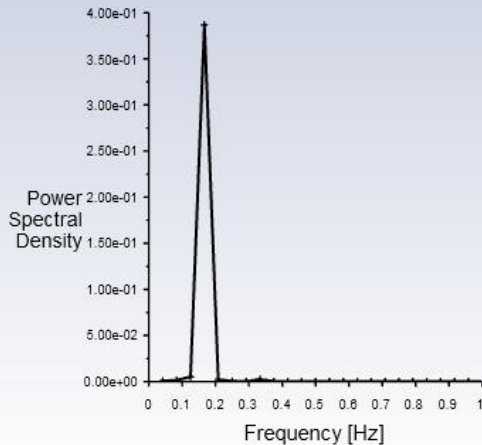
Because the sampling time (2.4 s) is relatively short, the frequency resolution shown in the plot is coarse.

The vortex shedding frequency is identified by the peak power spectral density. Using the Write FFT to File option in the Fourier Transform panel (not shown) and examining the output file, the frequency is 0.167 Hz, which agrees well with the expected value of 0.165 Hz from slide 29.

However, this should be checked by continuing the calculation for a much longer time such that the frequency resolution is improved. This is left as an optional exercise.
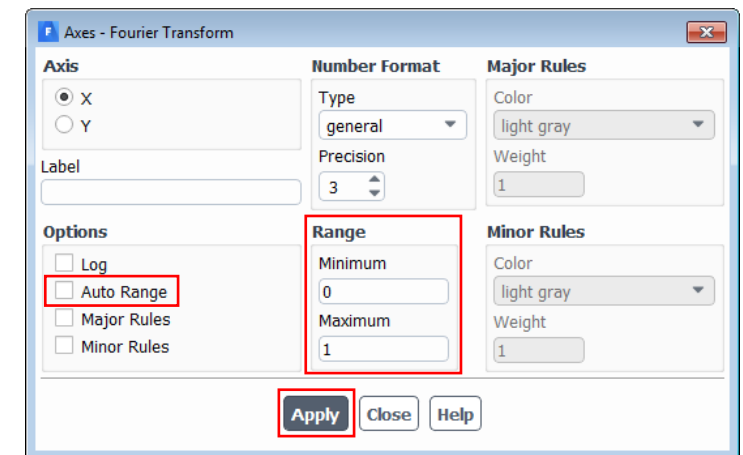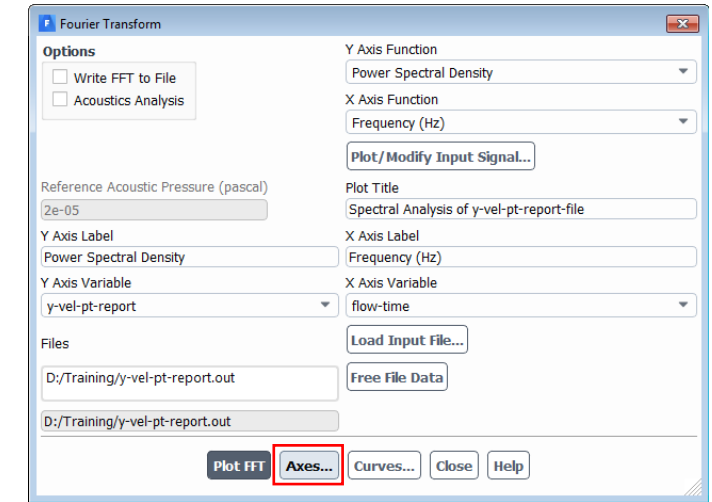
Wrap-up

- **This workshop has shown the basic steps for setting up and solving transient flows:**
  - Choosing the time step size
  - Using iterative and non-iterative time advancement
  - Patching values for the initial condition
  - Transient post-processing, including results files, FFT and animations

- **One of the most important things to remember in your own work, before even starting the ANSYS software, is to think WHY you are performing the simulation**
  - What information are you looking for?
  - What do you know about the boundary conditions?

In this case the goal was to calculate flow around a cylinder and assess the vortex shedding frequency. FFT analysis was used to identify that the predicted frequency is in good agreement with results from literature.

Knowing your aims from the start will help you make sensible decisions of how large to make the domain, the level of mesh resolution needed and which numerical schemes should be selected.

End of presentation