

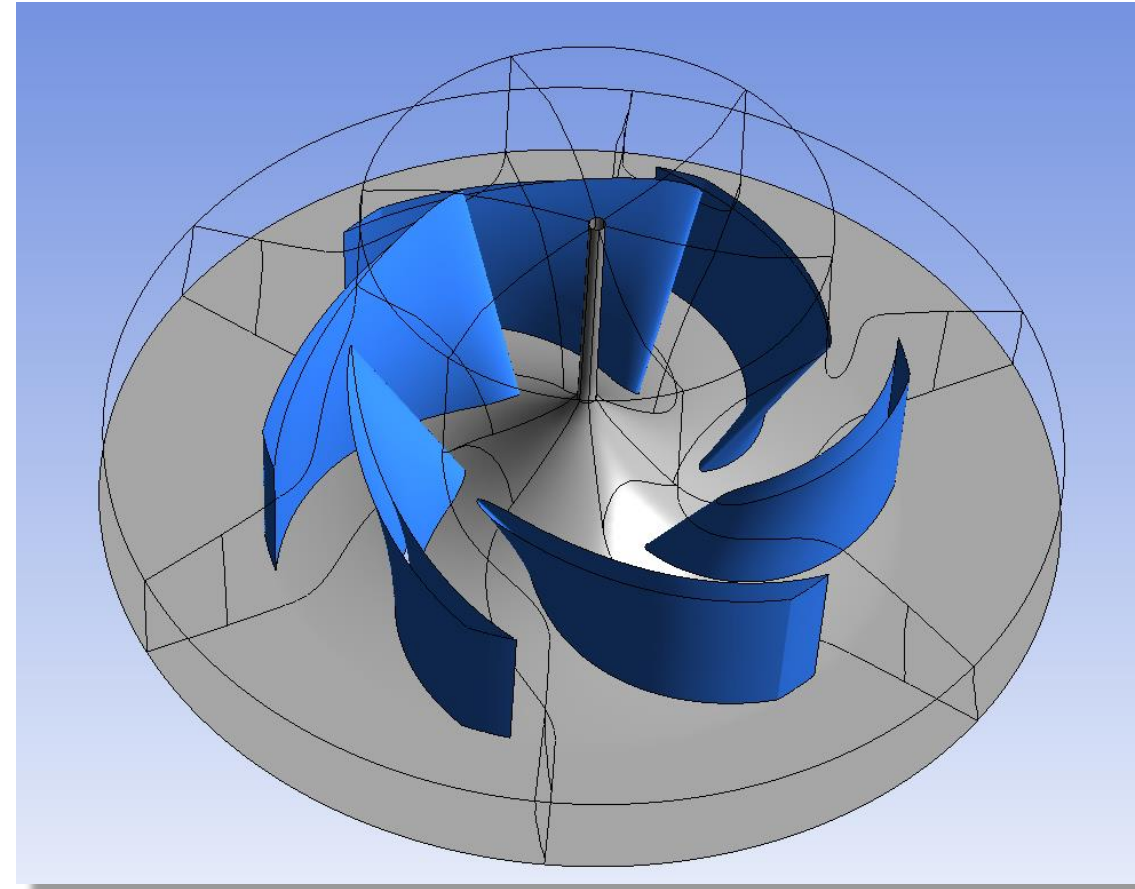
Workshop 03.1: Pump Analysis using CFD-Post

Release 2020 R2

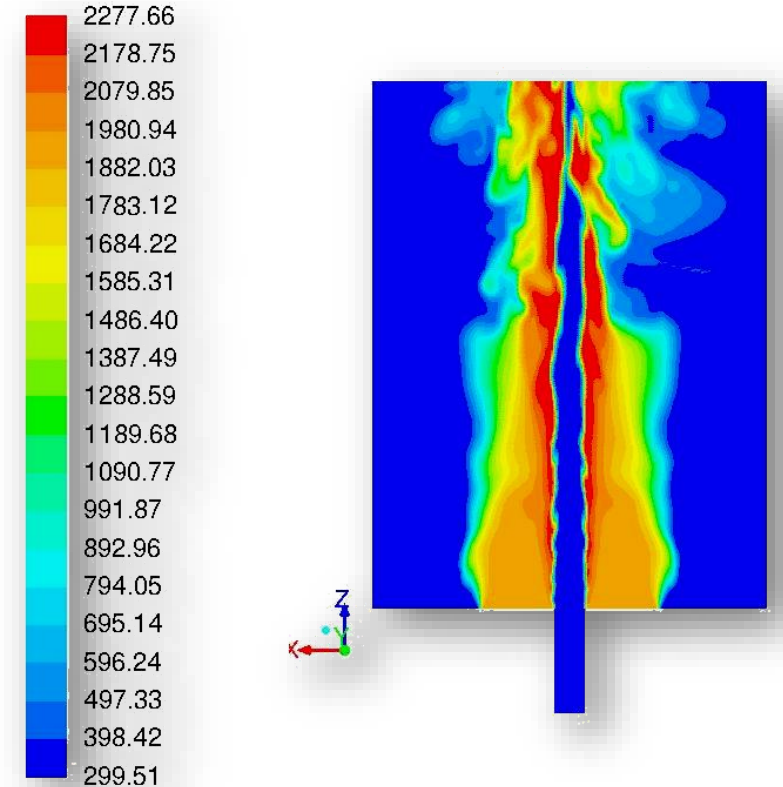


/ Introduction

- Workshop Description:
 - This Workshop deals with post-processing aspect for the pump impeller solved in Workshop 02.1
- Learning Aims:
 - Setting up turbo specific post-processing views
 - Meridional
 - Blade to Blade
 - Creating turbo specific charts
 - Blade Loading
 - Circumferential
 - Inlet to Outlet
 - Hub to Shroud
 - Creating turbo specific surfaces on which to plot vectors, contours, etc.
 - Using turbo specific variables

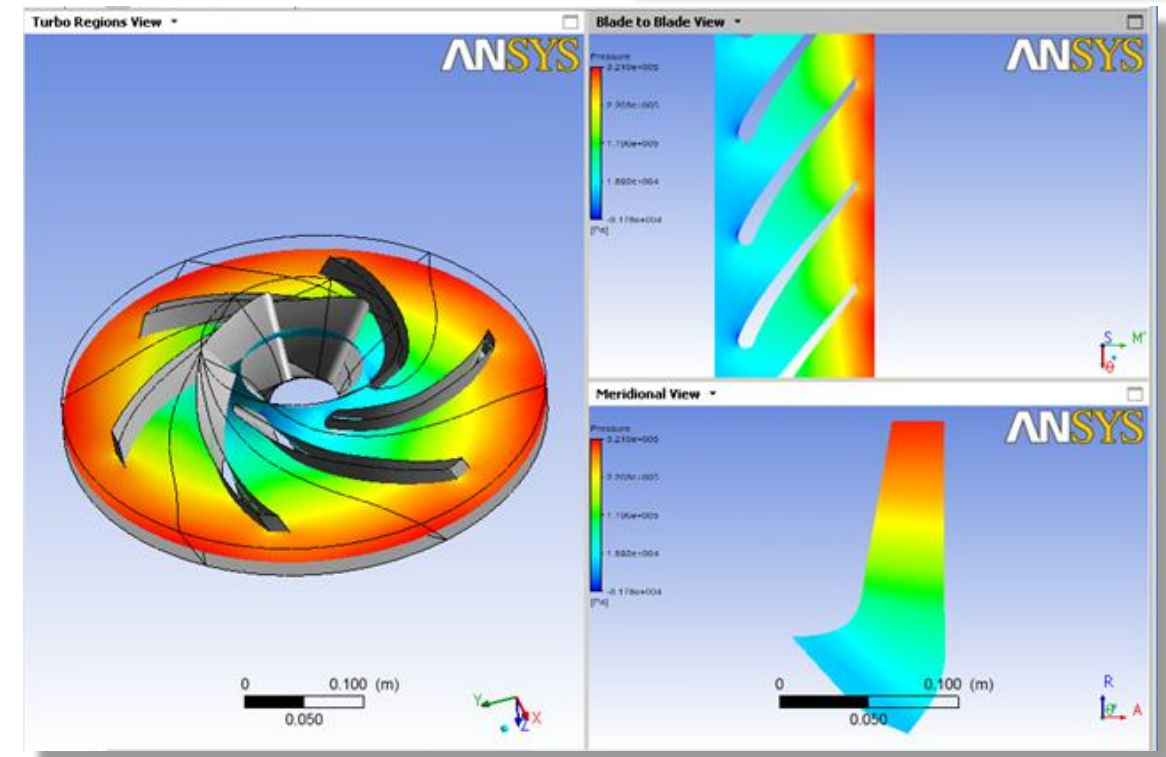
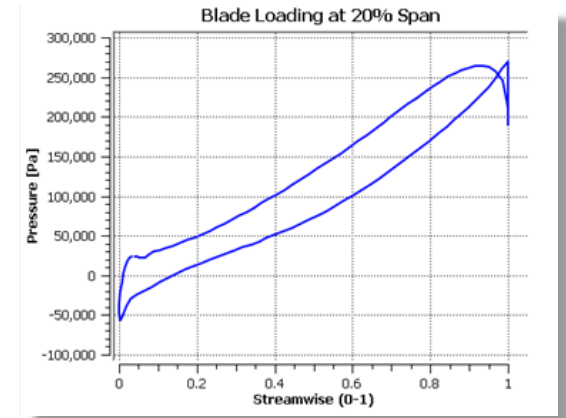


- CFD-Post includes many tools for analyzing general CFD results
 - Isosurfaces
 - Vector plots
 - Contour plots
 - Streamlines and particle tracks
 - 2D Charts
 - Animation creation



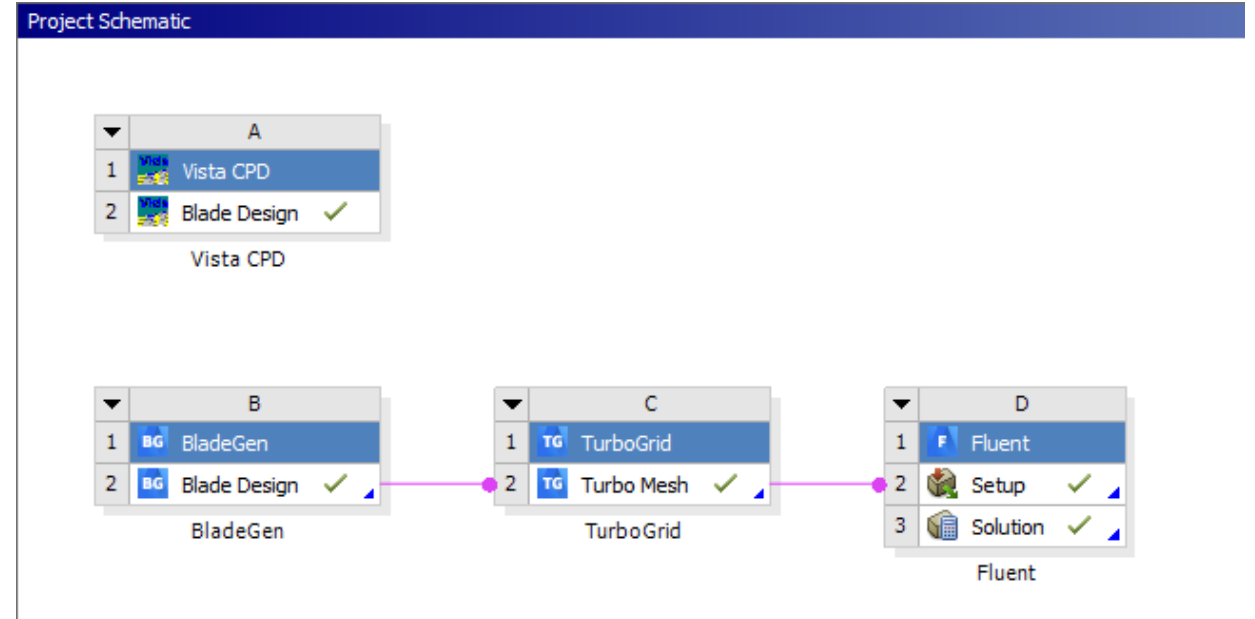
Post Processing

- CFD-Post also has turbo specific post-processing capabilities
 - Turbo Specific Post processing views
 - Meridional
 - Blade to Blade
 - Turbo Specific charts
 - Blade Loading
 - Circumferential
 - Inlet to Outlet
 - Hub to Shroud
 - Turbo Specific surfaces on which to plot vectors, contours, etc.
 - Turbo Specific variables



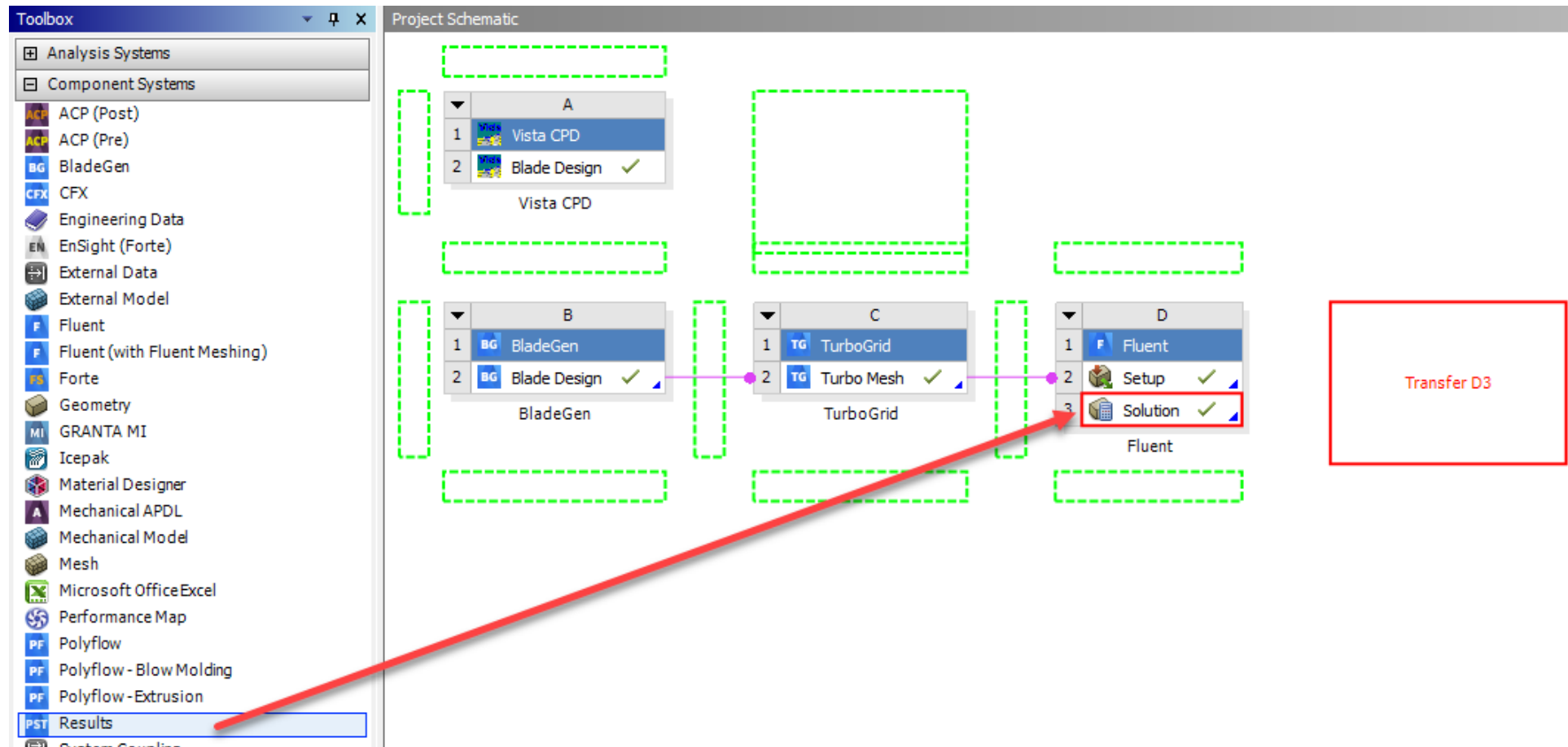
/ Starting Workbench Project

- Start *CFD-Post*
 - If you completed Workshop 02.1 “Pump Simulation” of this course, you may use your completed project from workshop 02.1 as a starting project
 - Alternatively, you may load the Workbench archive *Pump.wbpz*, provided with the inputs of this workshop:
 - In Workbench *File>Open*, browse to file *Pump.wbpz* and *Save as*, e.g., *Pump_Post.wbpj* in your working folder



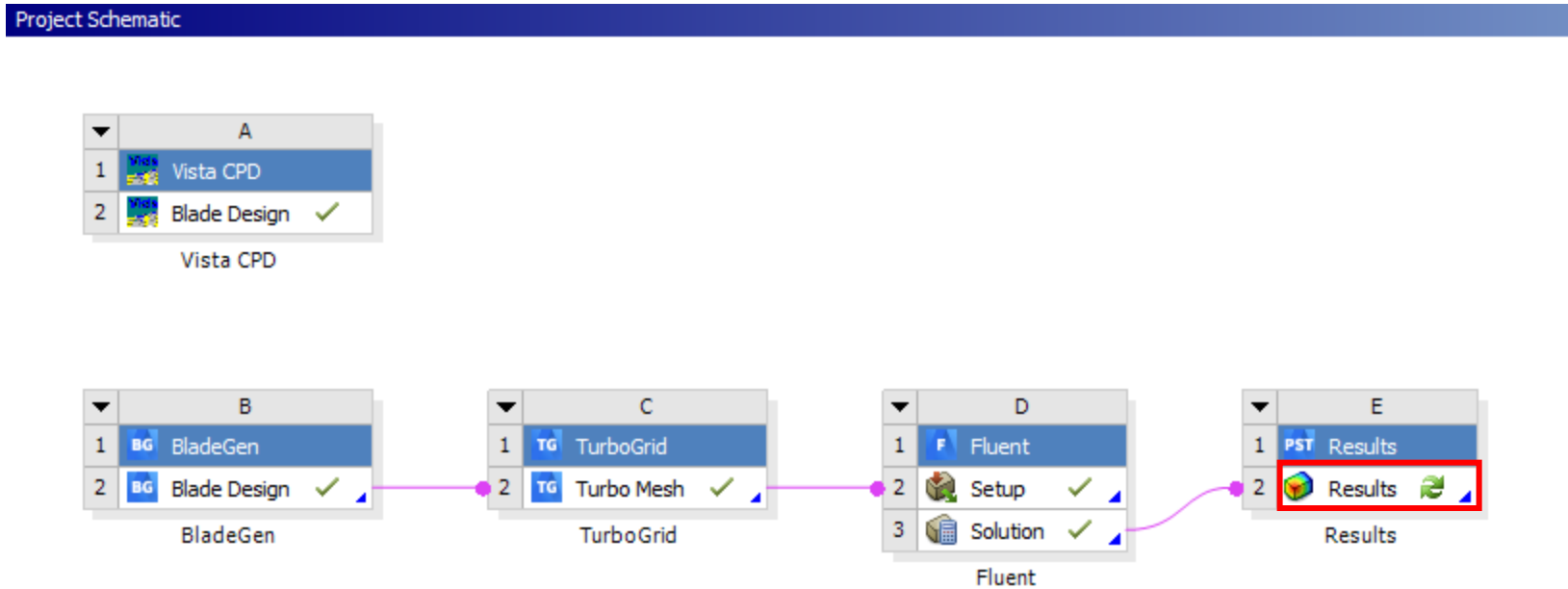
Add a Results Component System

- Add a *Results Component* to *Project Schematic*
 - Under *Component Systems* drag *Results* and drop it to cell *D3* to transfer the Fluent *Solution* to CFD-Post



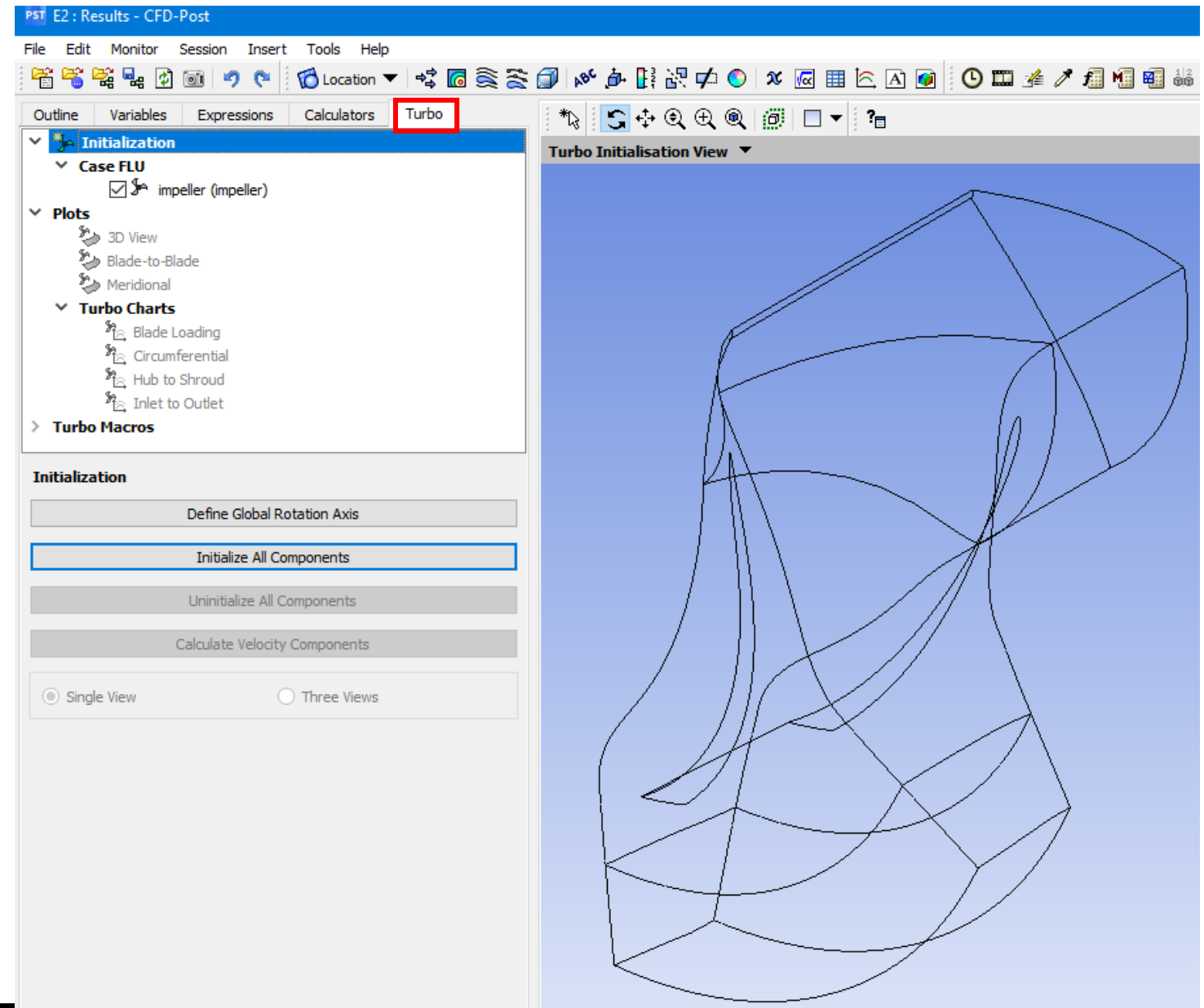
/ Launch CFD-Post

- Double click on the *Results* cell *E2* to launch *CFD-Post*



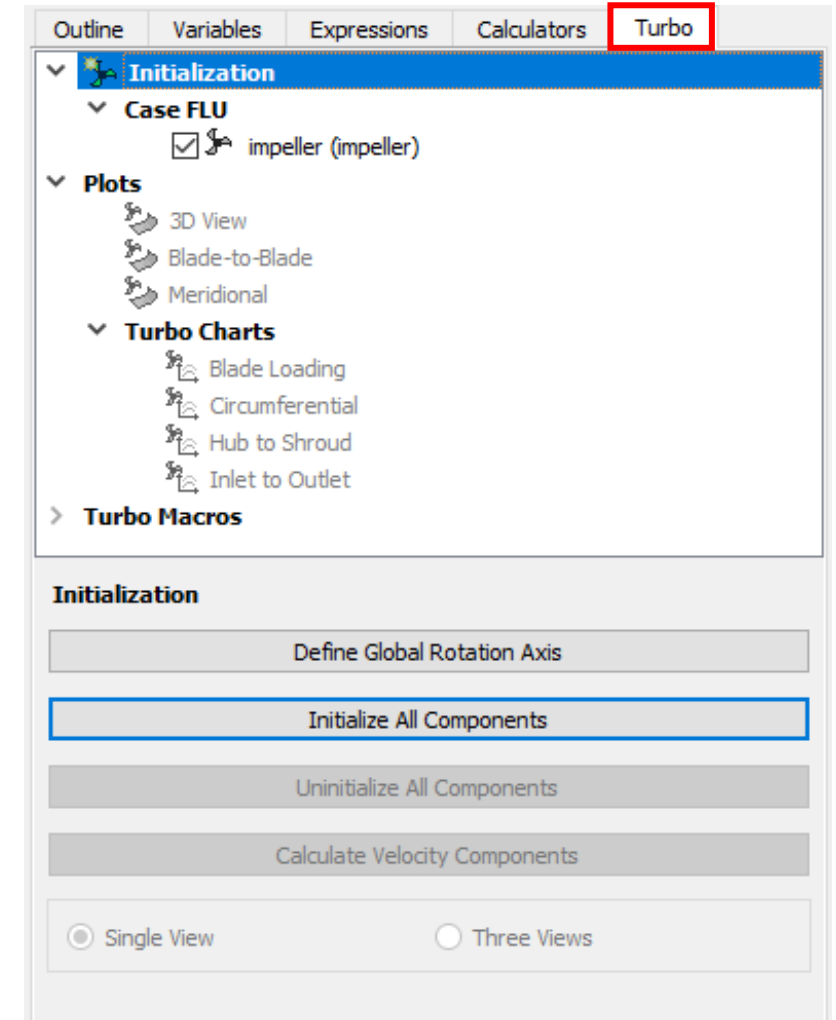
Turbo Post

- Once the case is opened in *CFD-Post*, switch to the *Turbo* Tab
- If prompted to automatically initialize the case, select *No*



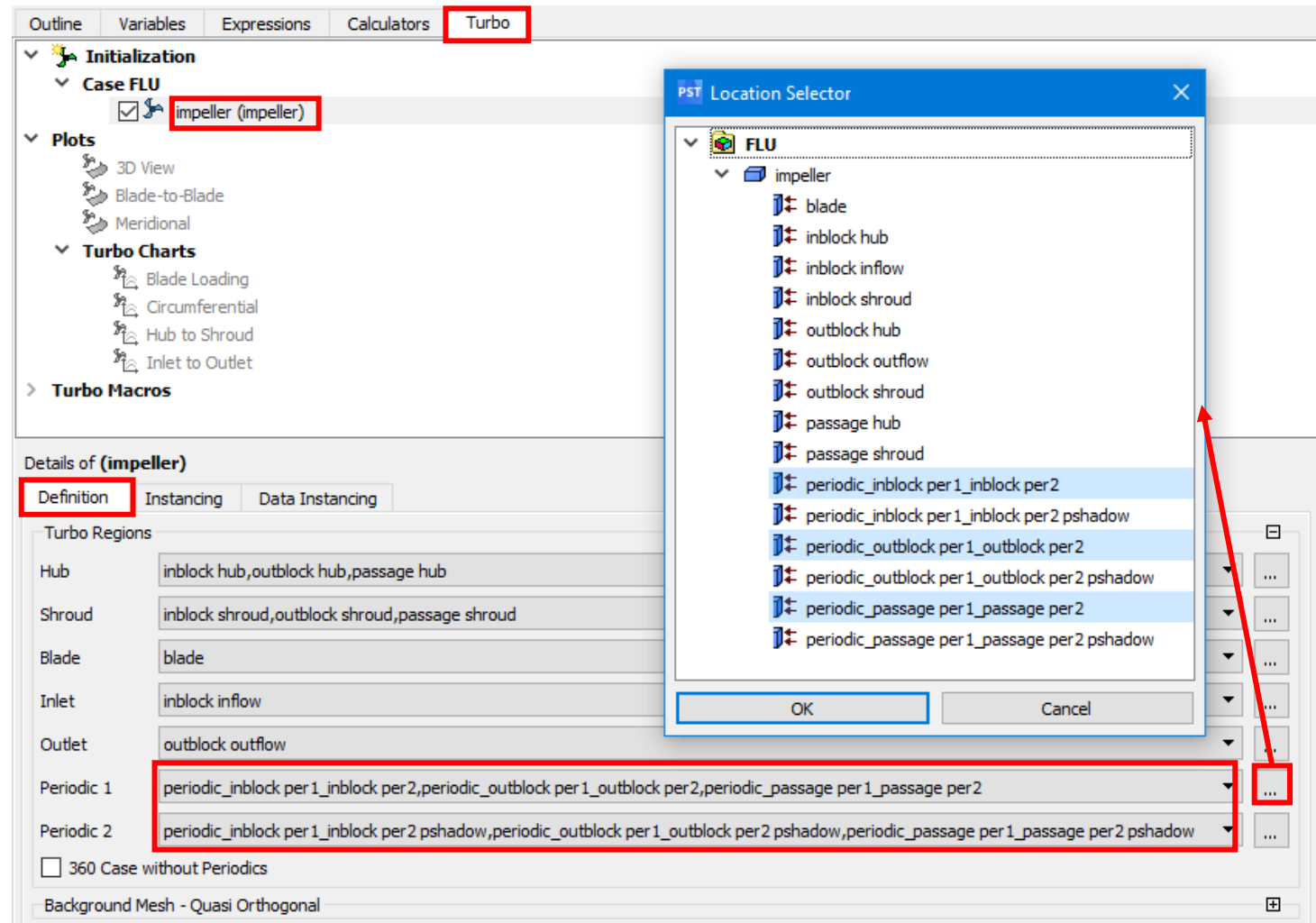
/ Initialization Principle

- Initializing the components in the Turbo tab is the first step
 - This process defines a new set of coordinates and variables that are useful for turbo post-processing
 - These coordinates and variables are based on the hub, shroud, inlet, outlet, blade, and periodic boundaries
 - By default *CFD-Post* will try to determine these boundaries, but they can always be input by the user



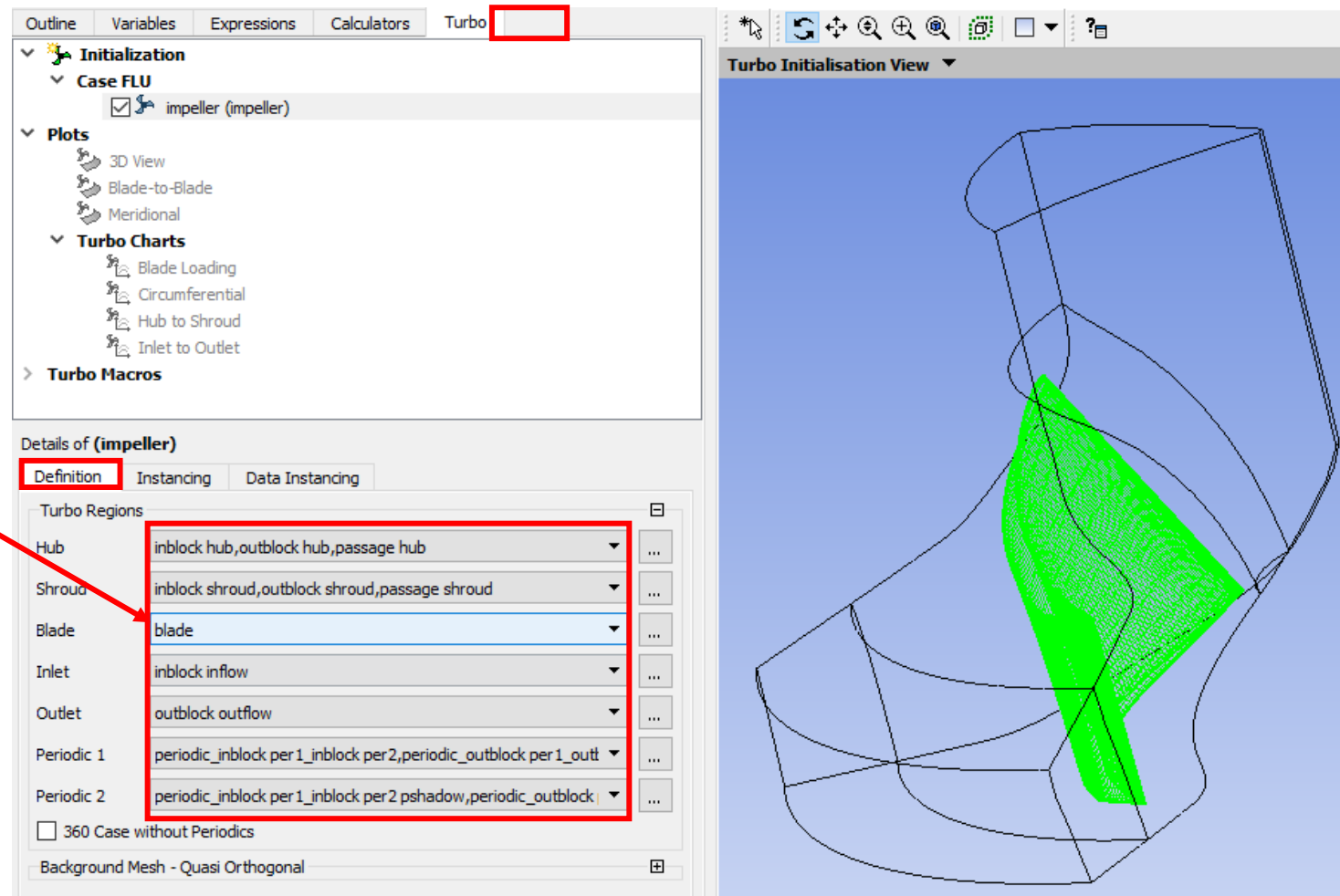
/ Initialize *impeller* (1)

- Double click on *impeller* in the tree
- In the *Details of (impeller)*, *Definition* tab, set the Turbo Regions as shown on the right
 - Some regions are detected automatically (in this case *Hub Shroud, Blade, Inlet and Outlet*)
 - Some need your intervention (in this case *Periodic 1 and Periodic 2*)
 - Use the ellipsis (...) next to a *Turbo Region* you need to set
 - In the *Location Selector* left-click to select the corresponding boundary zone and click *OK*
 - For multiple selection you need to hold down the *Ctrl* button



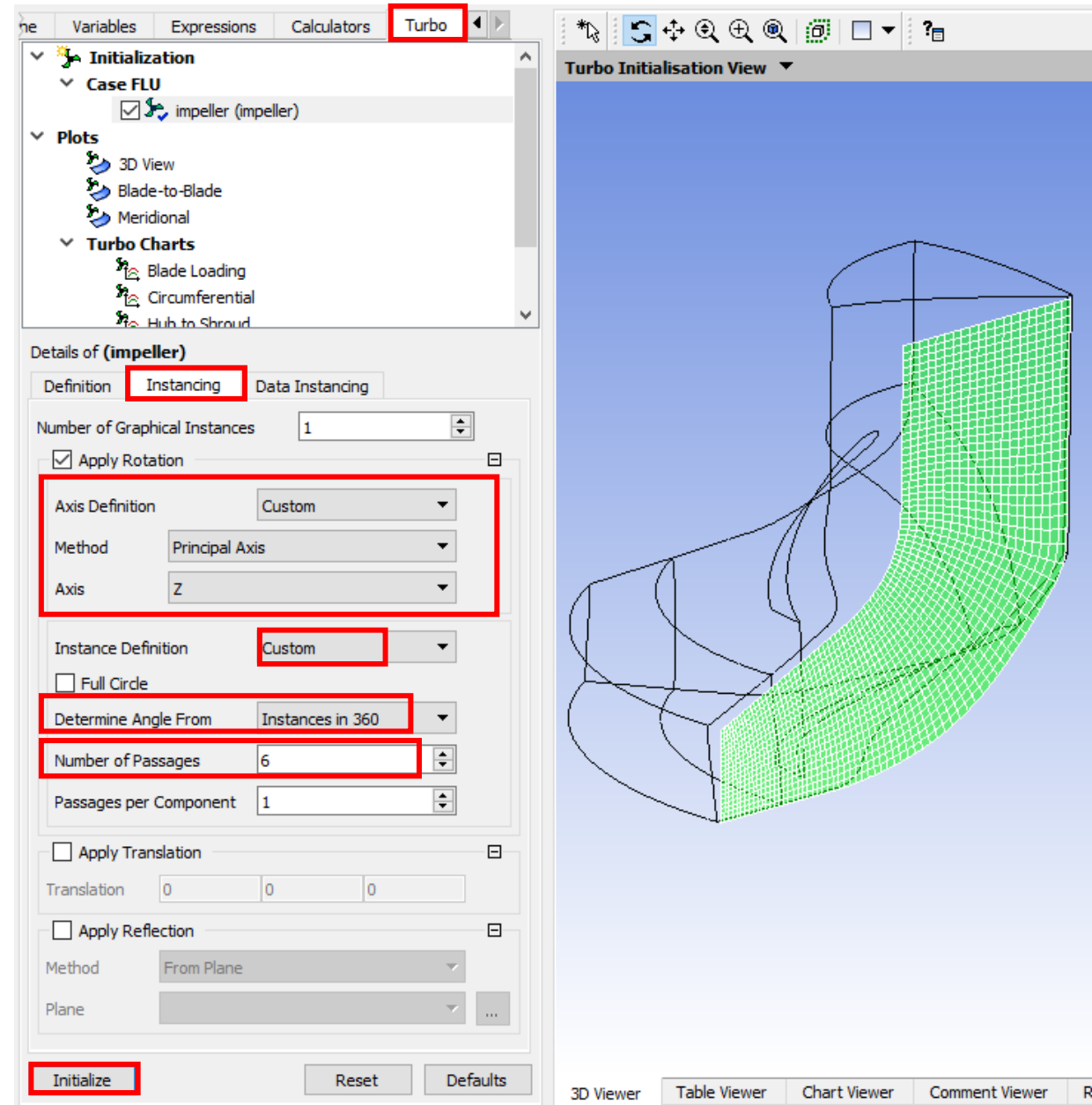
/ Initialize *impeller* (2)

- Inspect the correct selection of the Turbo regions visually
- Mouse over the various boundary zones in the area marked with the red box
- The corresponding surface is highlighted by a green color in the graphics window
 - In the example on the right, when the mouse is over the *blade* boundary zone the corresponding blade surface is highlighted



/ Initialize *impeller* (3)

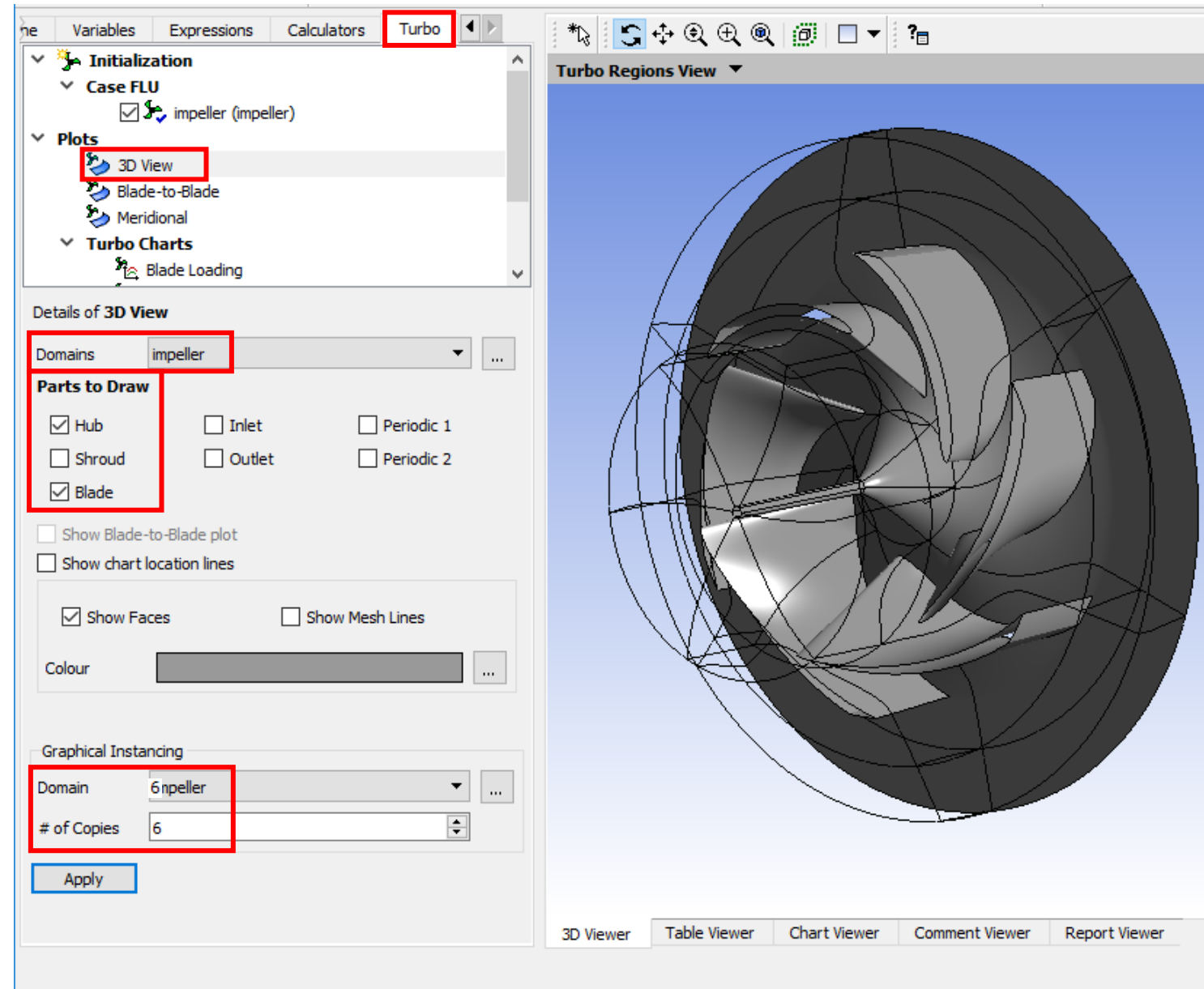
- Click the *Instanting* tab
 - In this tab you define the rotational axis and angle for the passage. This is needed since CFD-Post currently does not get turbo topology information from Fluent cases automatically
 - Leave the default *Axis Definition* (Z Axis, suitable for this case)
 - Select *Custom* from the *Instance Definition* drop-down list
 - Retain the selection of *Instances in 360* from the *Determine Angle From* drop-down list
 - Enter 6 for *Number of Passages*
 - This fixes the passage periodic angle at $360/6 = 60$ degrees
- Click *Initialize*
- This will result in a background meridional mesh being displayed for the *impeller* cell zone



Plots: 3D View

- To visualize the pump geometry (still in the *Turbo* tab) double click on *3D View* under *Plots*
 - Select *Hub* and *Blade* under *Parts to Draw*
 - Select *impeller* next to *Domains*
 - Set the number of copies to 6
 - Click *Apply*
 - All 6 blades and passages are visualized

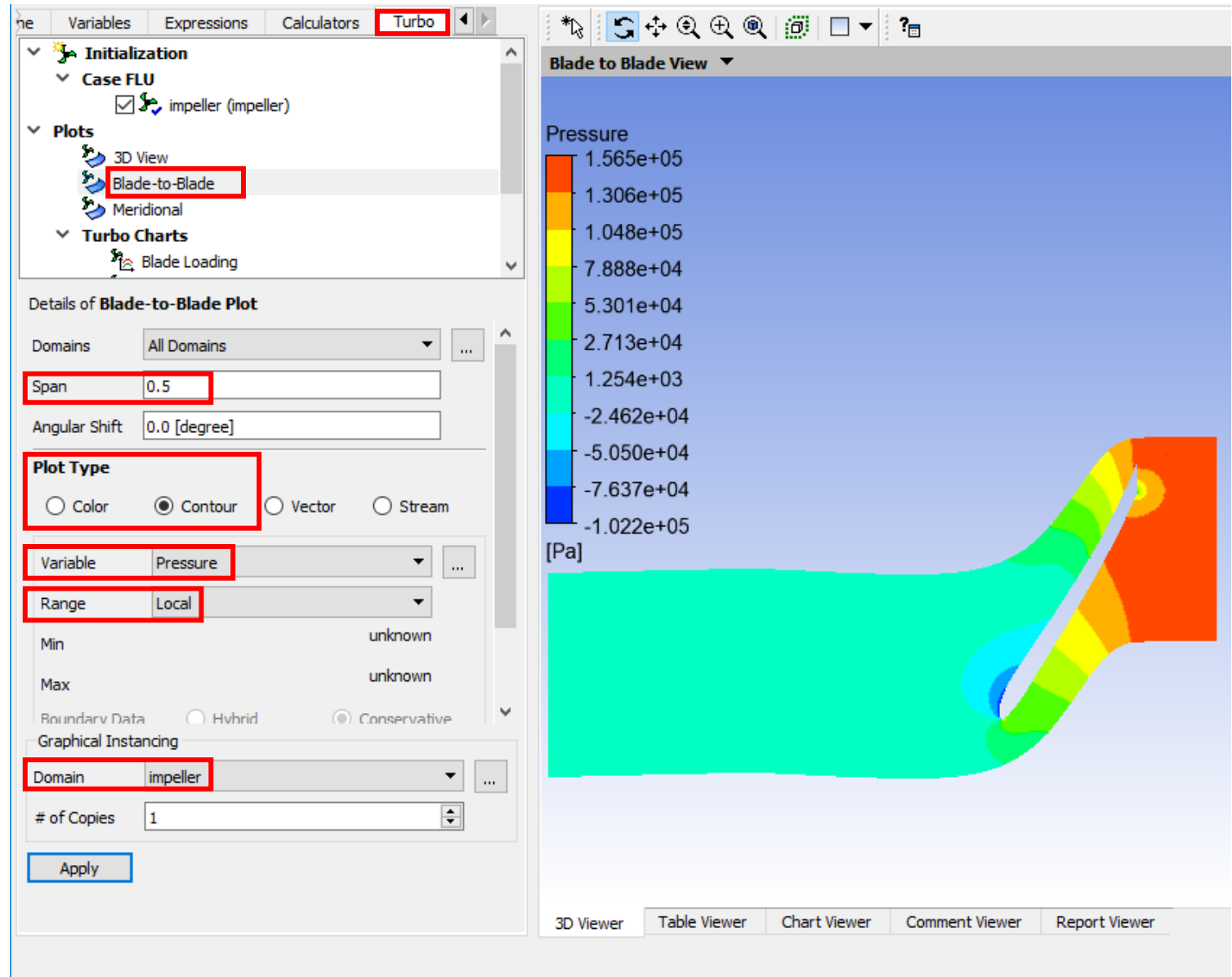
In the remaining slides numerous post-processing objects (flow visualization images, charts, tables, etc.) will be generated. Please notice that there can be some differences between the indicative images & values given in the slides and the ones created by you



Plots: Blade-to-Blade

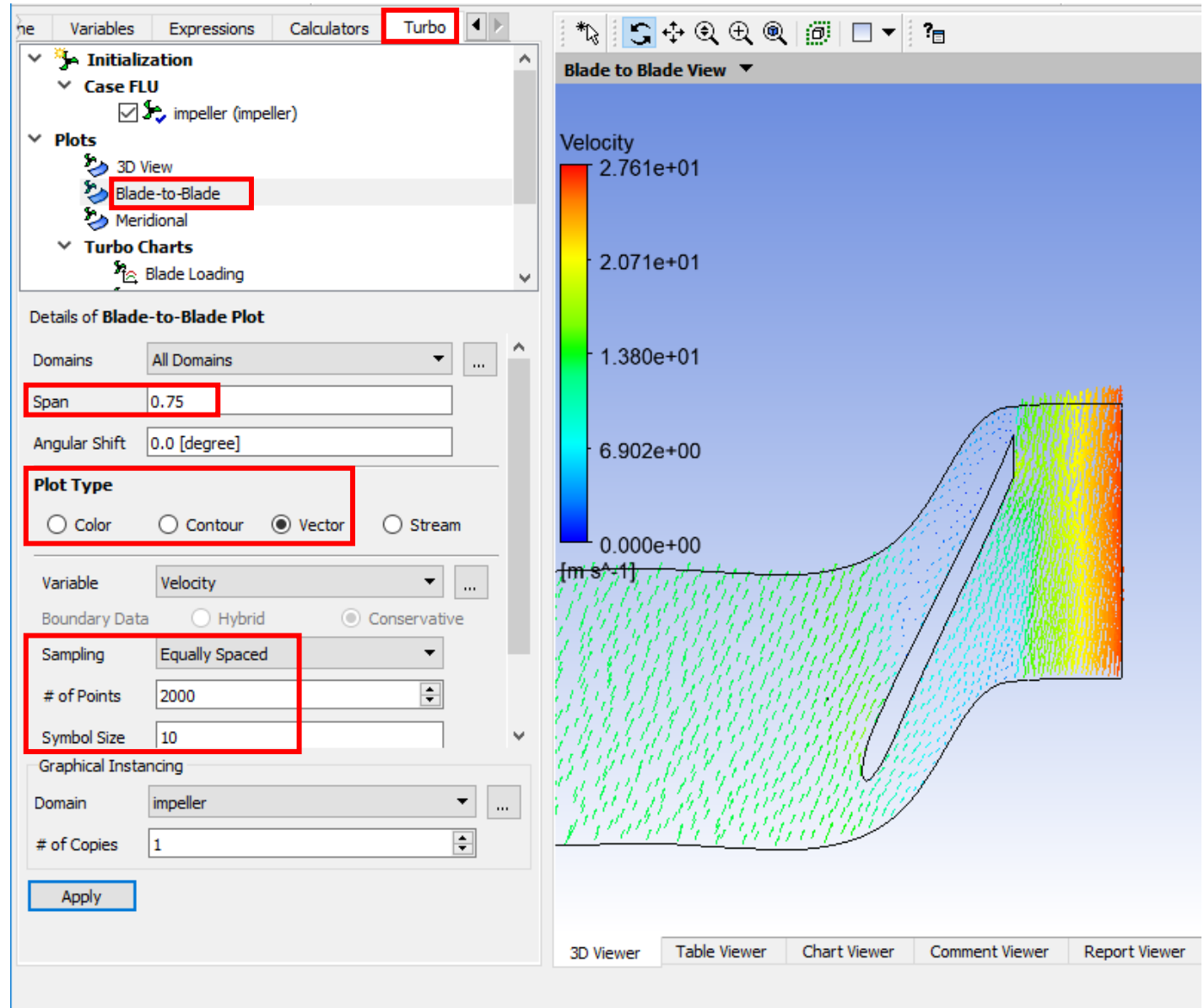
- Make an unwrapped* blade-to-blade plot of pressure at mid-span
 - Double click *Blade-to-Blade* in the outline tree, set *Contour* for *Plot Type* and do the following settings
 - *Span* = 0.5
 - *Plot Type* = *Contour*
 - *Variable* = *Pressure*
 - *Range* = *Local*
 - Set *# of Copies* back to 1 for the *impeller Domain*
 - Click *Apply*

* An “unwrapped” plot is one where the 3D surface contours are transformed so that they lie in a plane, similar to taking a curved piece of paper and unwrapping it to lie on a flat table



Blade-to-Blade

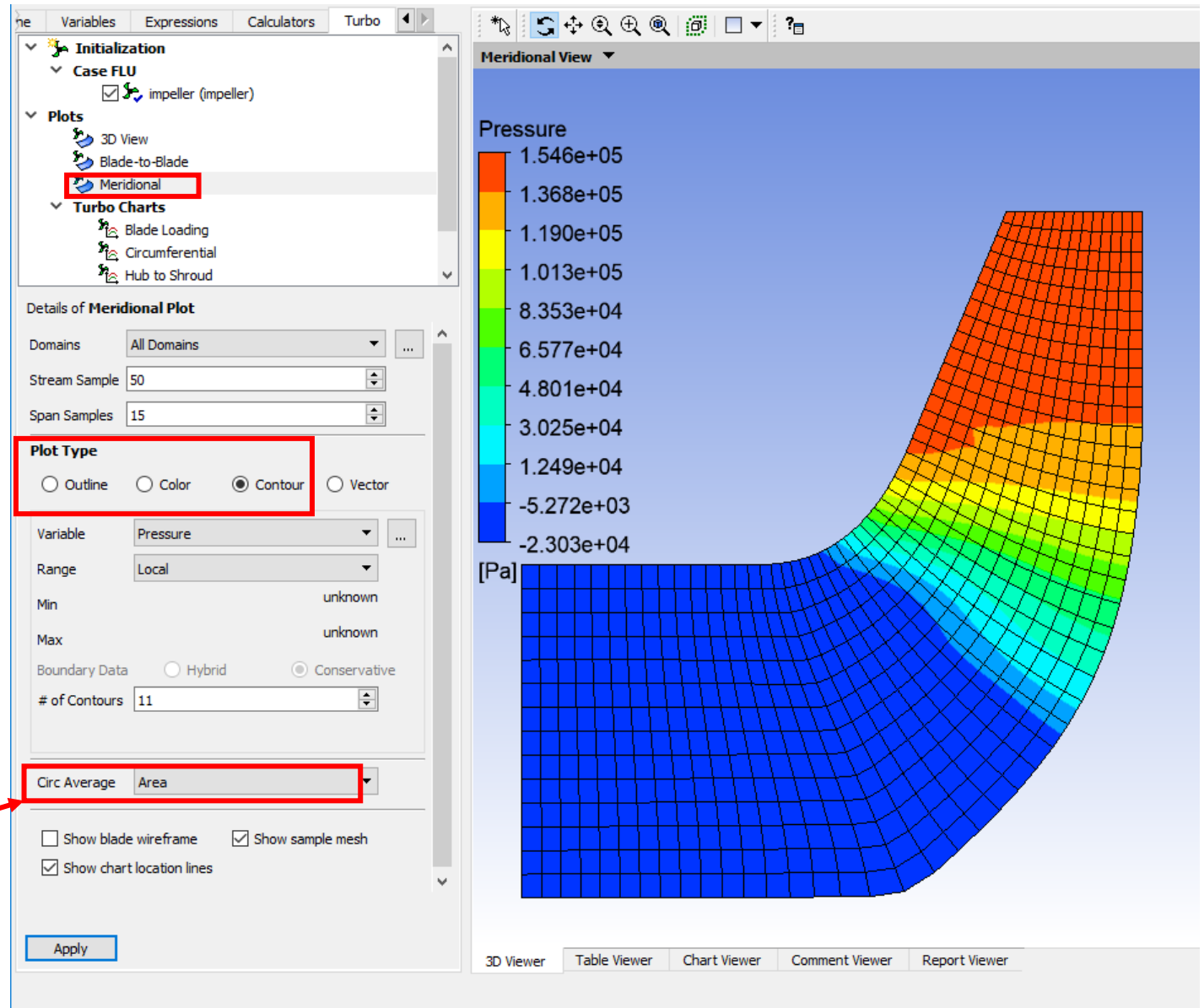
- Change this to a *Vector* plot at 75% span
 - Change *Span* to 0.75
 - Change Plot Type to *Vector*
 - Set *Sampling* to *Equally Spaced*
 - Set *# of Points* to 2000
 - Set *Symbol Size* to 10
 - Click *Apply*



Meridional Plot

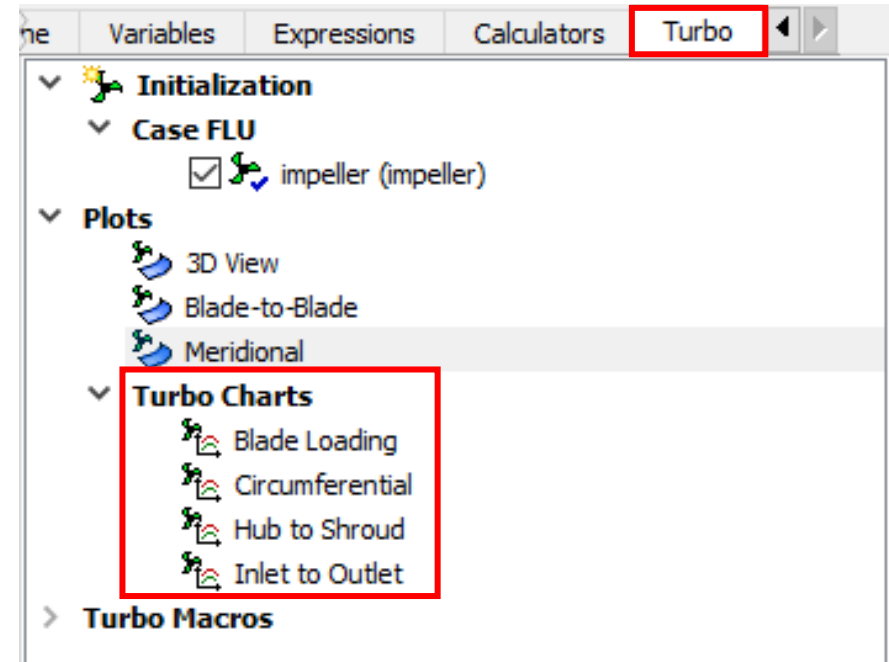
- Create a circumferentially averaged pressure contour in the meridional* view:
 - Double click on *Meridional* in the *Tree*
 - Select *Contour* for *Plot Type*
 - Select *Pressure* for the *Variable*
 - Select *Local* for *Range*
 - Click *Apply*

* A meridional view depicts the passage flow from inlet to outlet, hub to shroud using a circumferential average of the flow properties at a given position within the passage



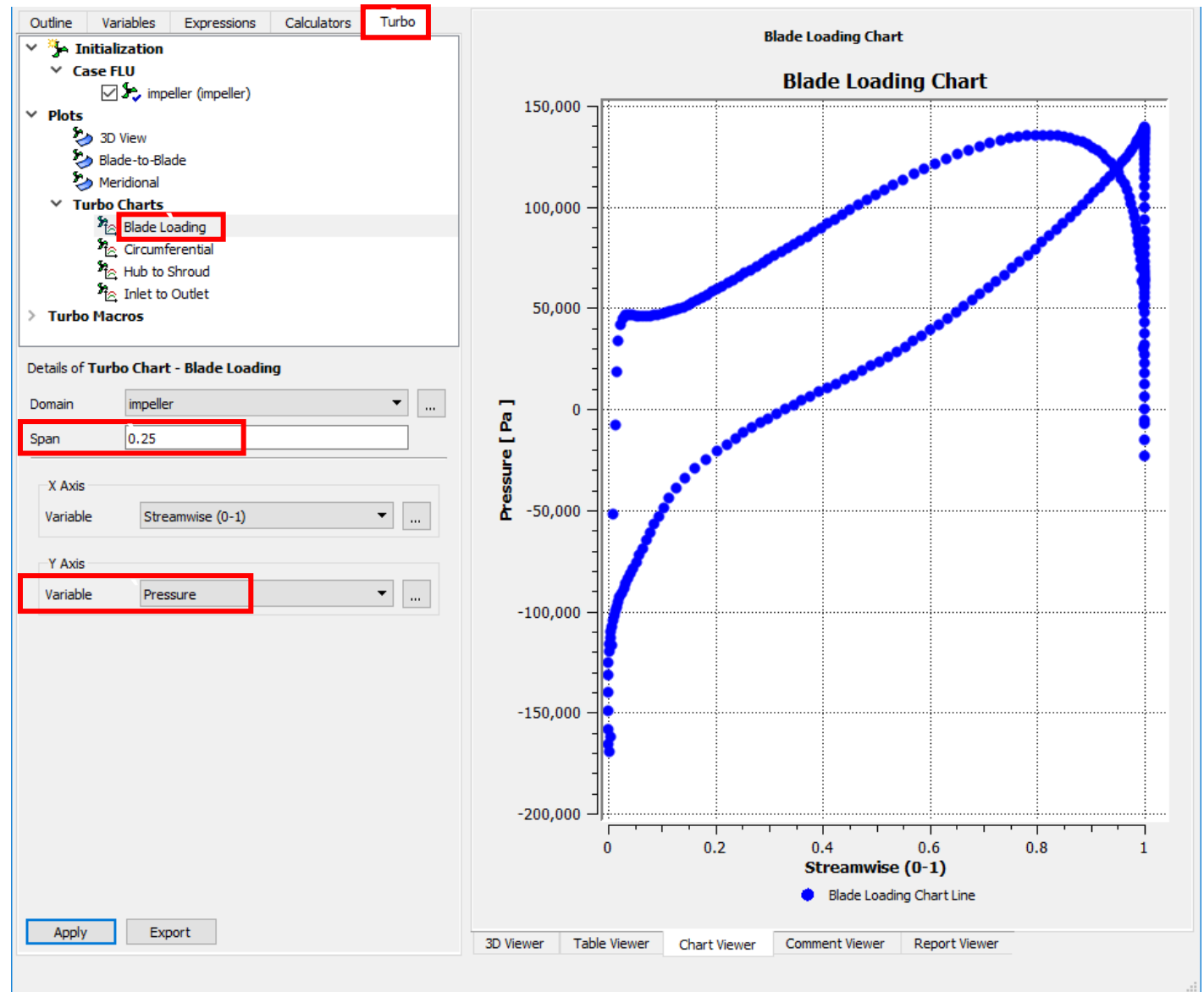
Turbo Charts

- There are four different types of 2D plots that can be created:
 - *Blade Loading*
 - *Circumferential*
 - *Hub to Shroud*
 - *Inlet to Outlet*



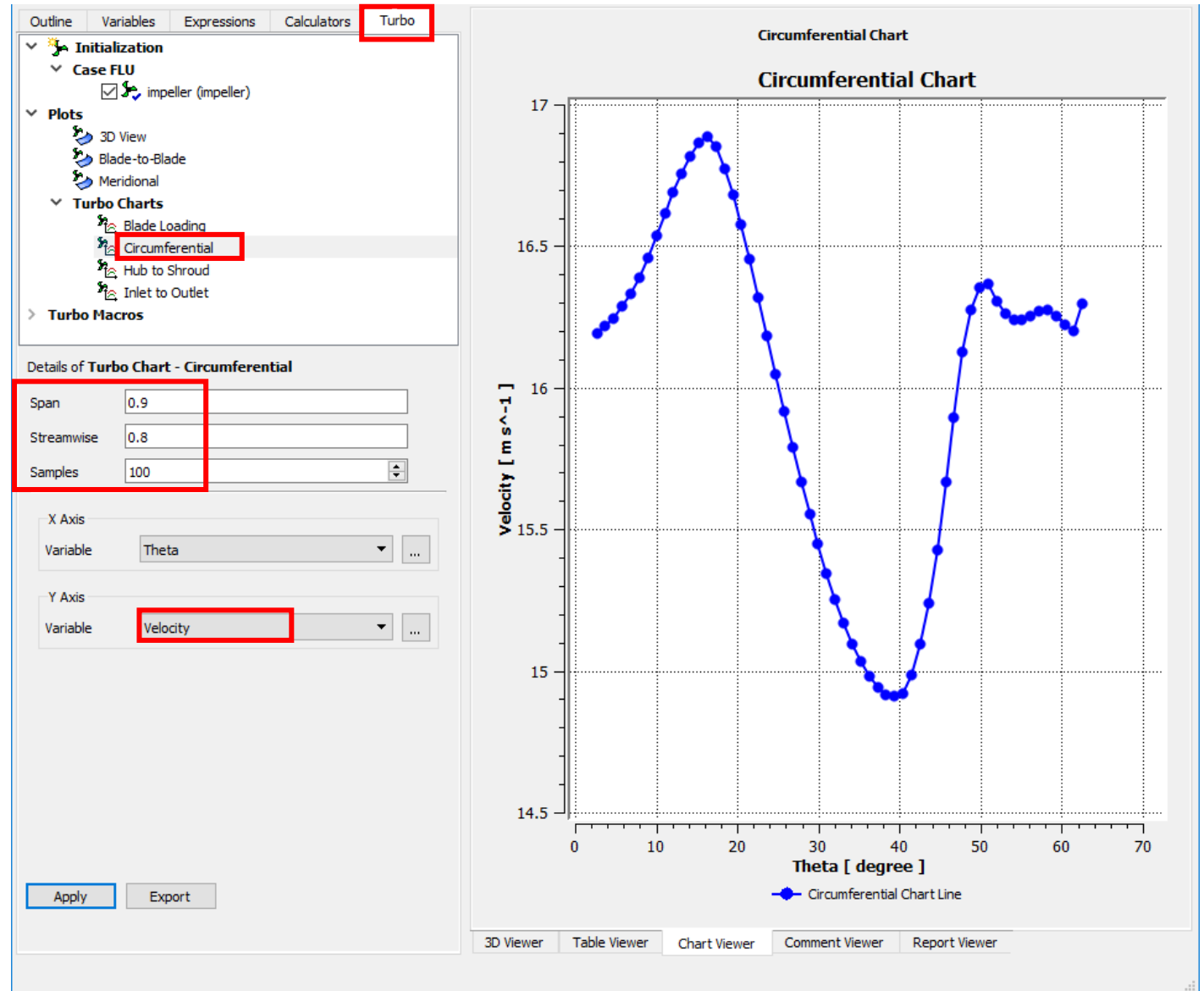
Blade Loading Chart

- Create a blade loading chart of pressure at 25% span
 - Double click *Blade Loading* under *Plots > Turbo Charts* in the *Tree*
 - Set *Span* to 0.25
 - Set *Y Axis Variable* to *Pressure*
 - Click *Apply*



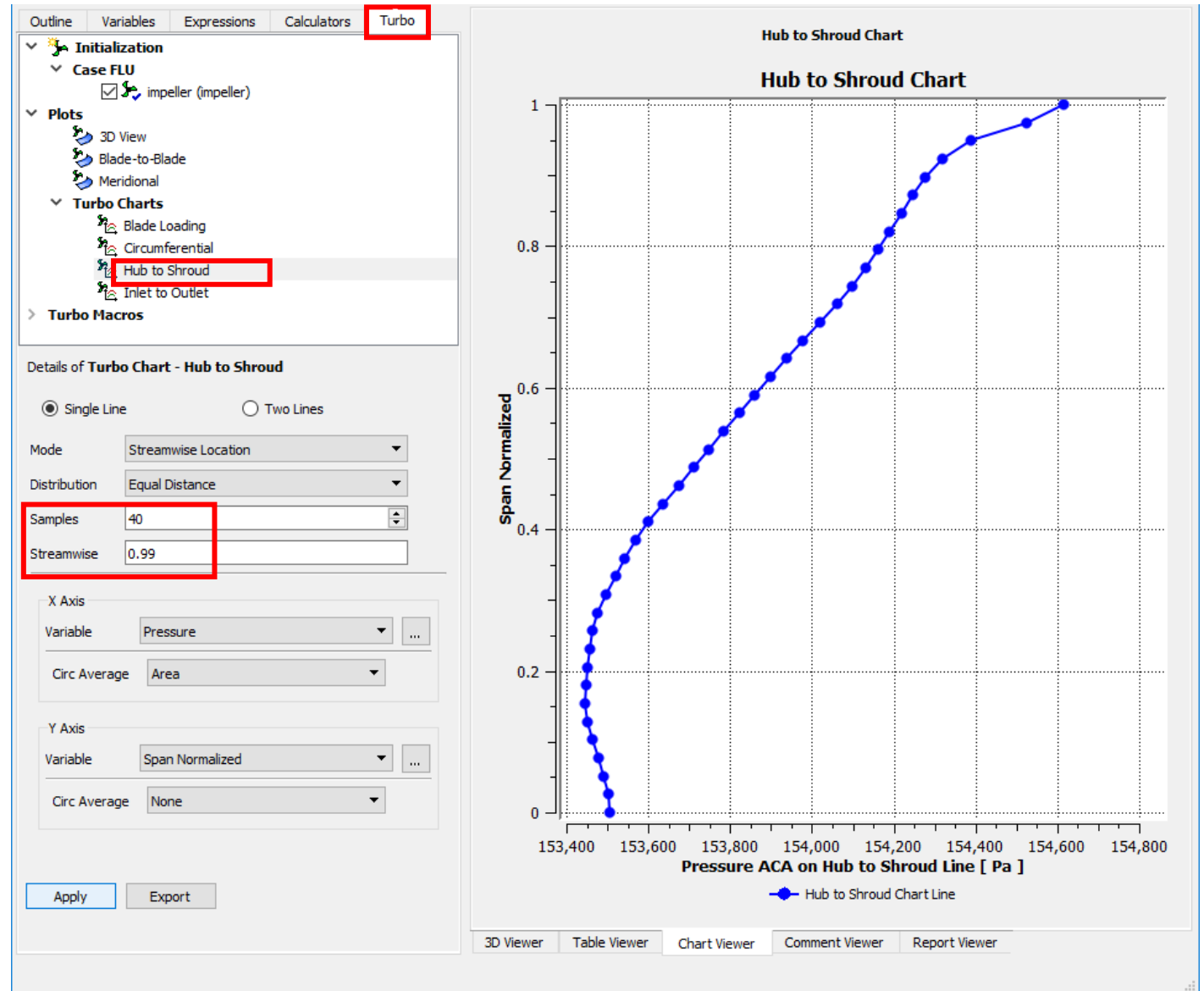
Circumferential Chart

- To examine a velocity profile in the wake of the blade, create a circumferential chart:
 - Double click *Circumferential* under *Turbo Charts*
 - Set *Span* to .9
 - Set *Streamwise* to 0.8
 - Set *Samples* to 100
 - Set *Y Axis Variable* to *Velocity*
 - Click *Apply*



Hub to Shroud

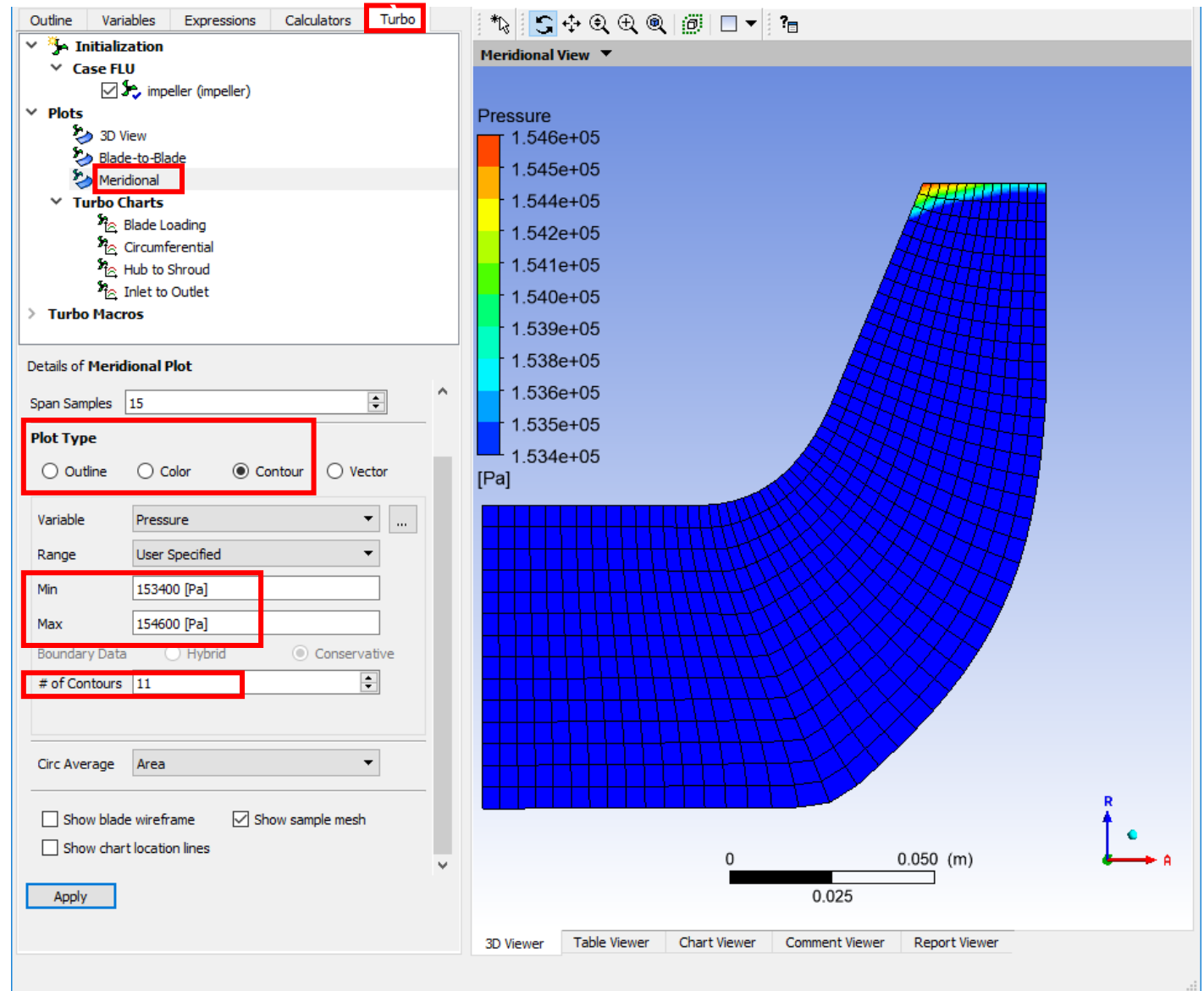
- Let's examine a circumferentially averaged pressure profile near the outlet spanwise
 - Hub to Shroud *Hub to Shroud* in the *Tree*
 - Set *Distribution* to *Equal Distance*
 - Set *Samples* to 40
 - Set *Streamwise* to 0.99
 - Click *Apply*



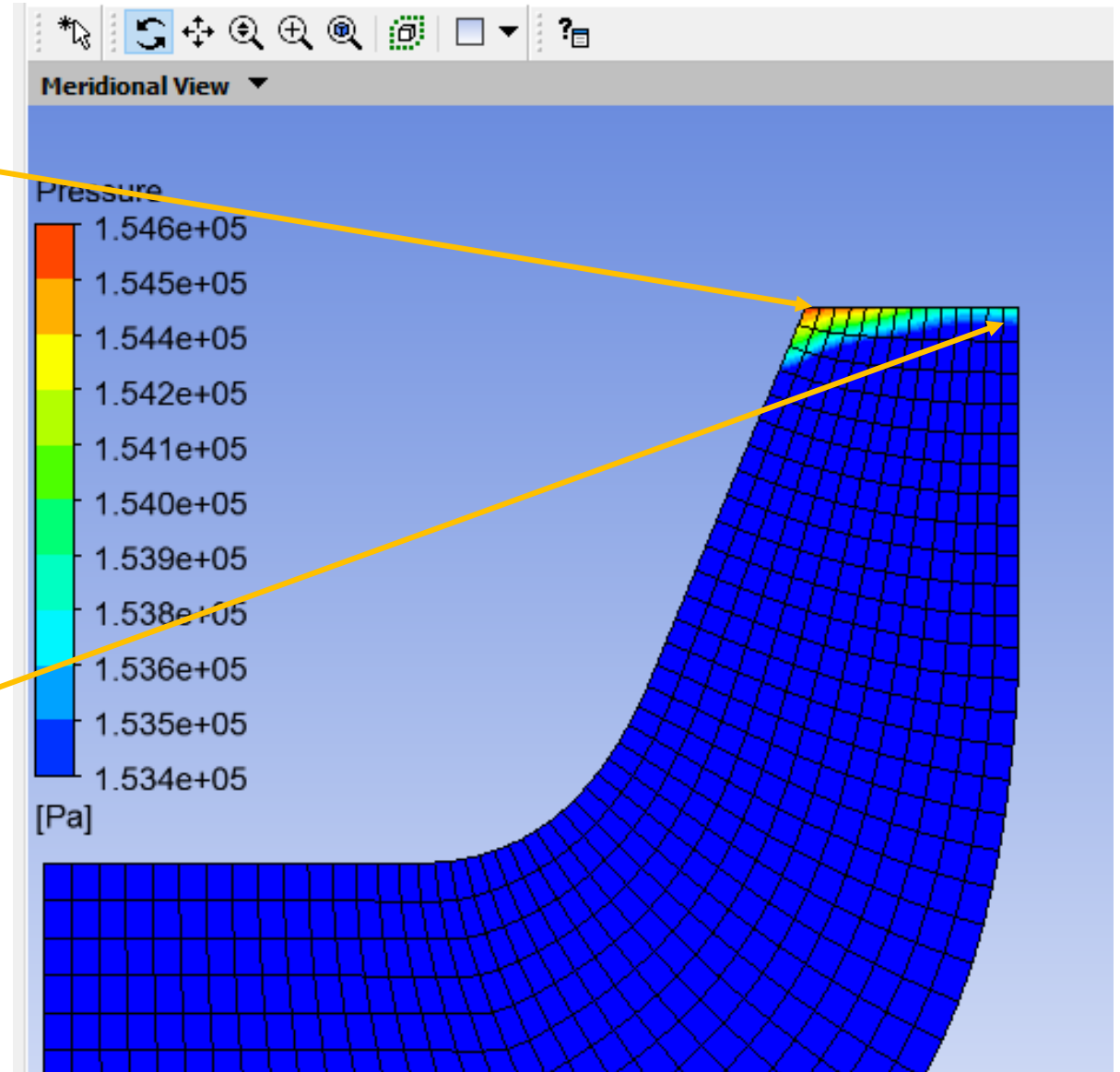
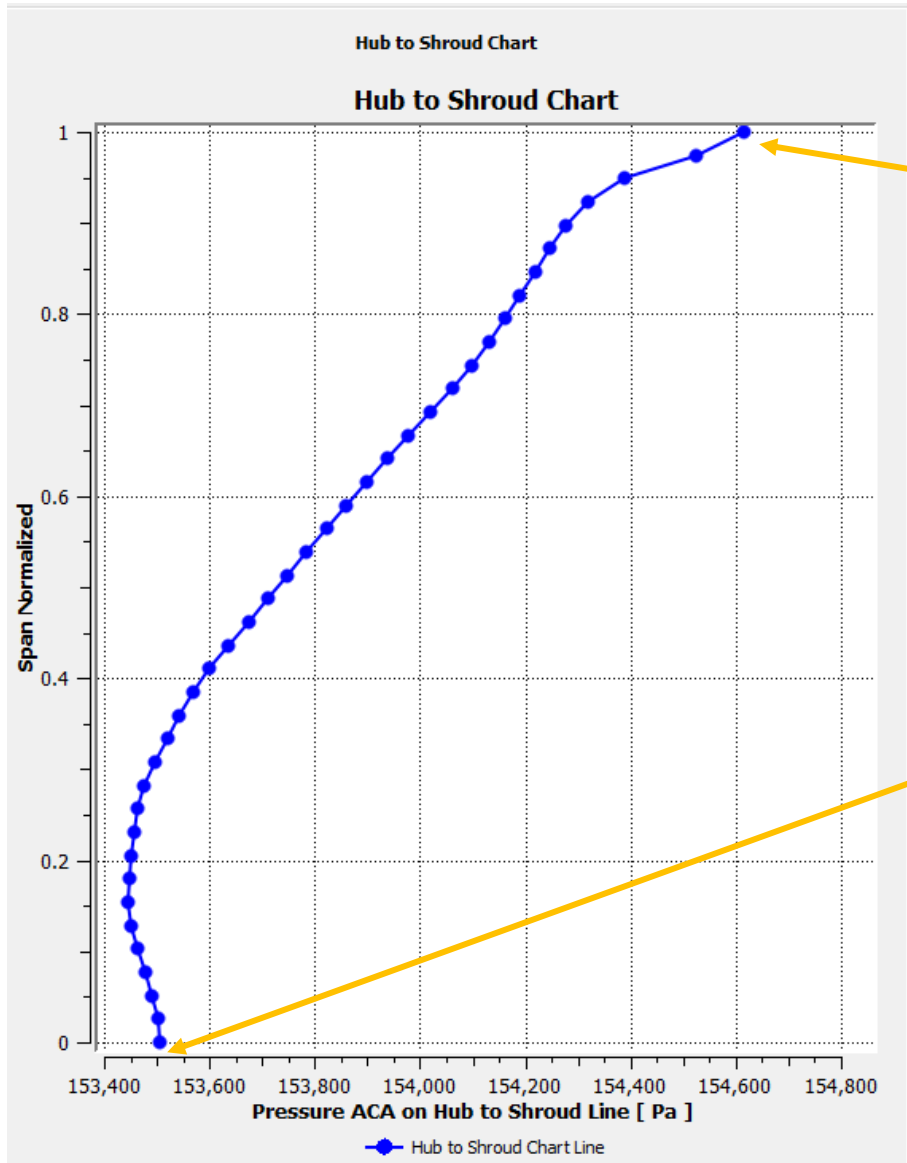
/ Compare to Meridional Plot

- The profile just created corresponds to data shown in a meridional plot
 - Create a meridional contour as shown on the right
 - Set the *Range* to *User Specified* and set the *Min* and *Max* values to 153400 and 154600 [Pa] respectively*
 - Notice the pressure profile near the outlet

*The *Min* and *Max* values of 153400 and 154600 are taken from the x-axis limits of the chart in the previous slide. You might have to give different values depending on own results

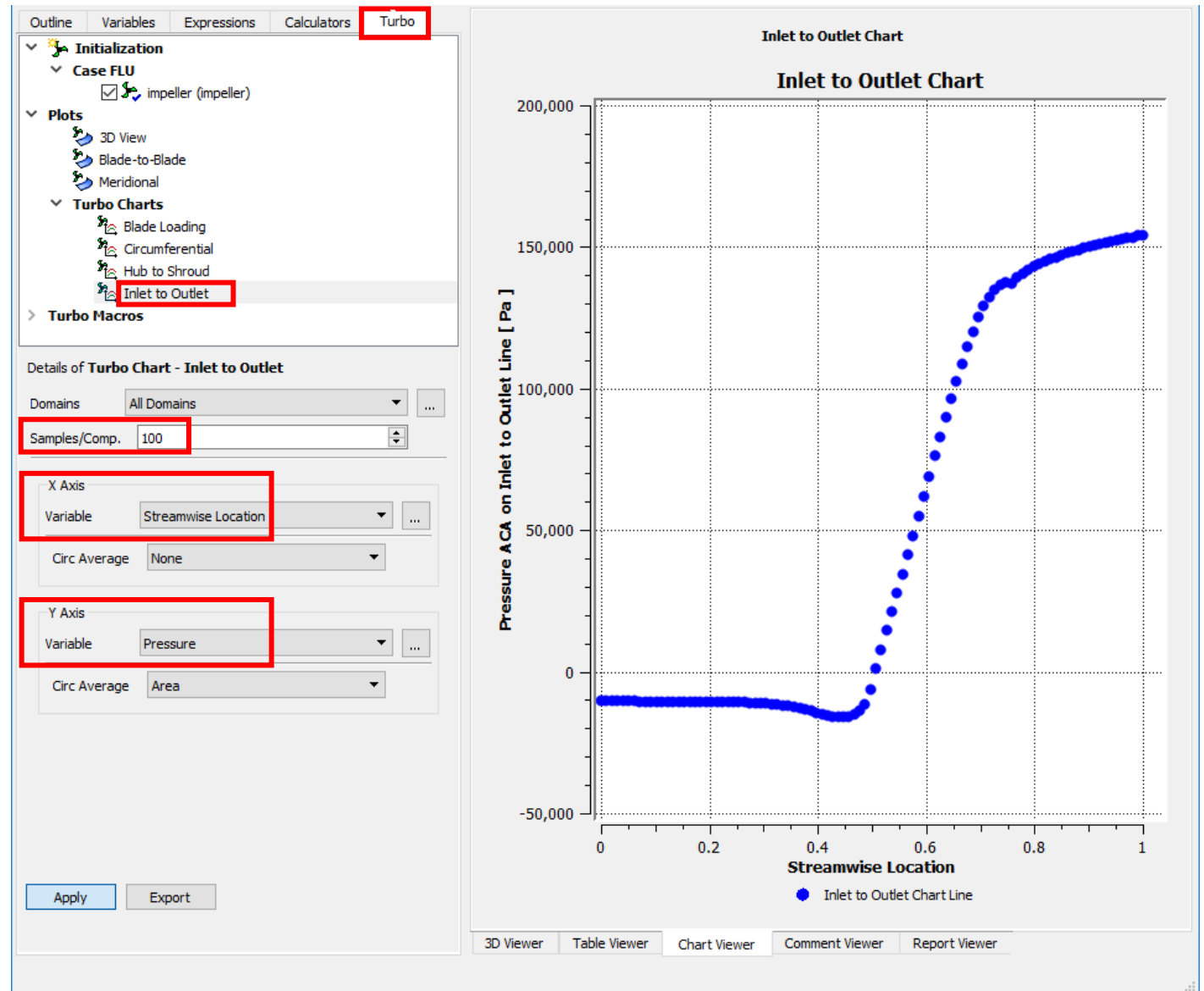


/ Compare to Meridional Plot



/ Inlet to Outlet

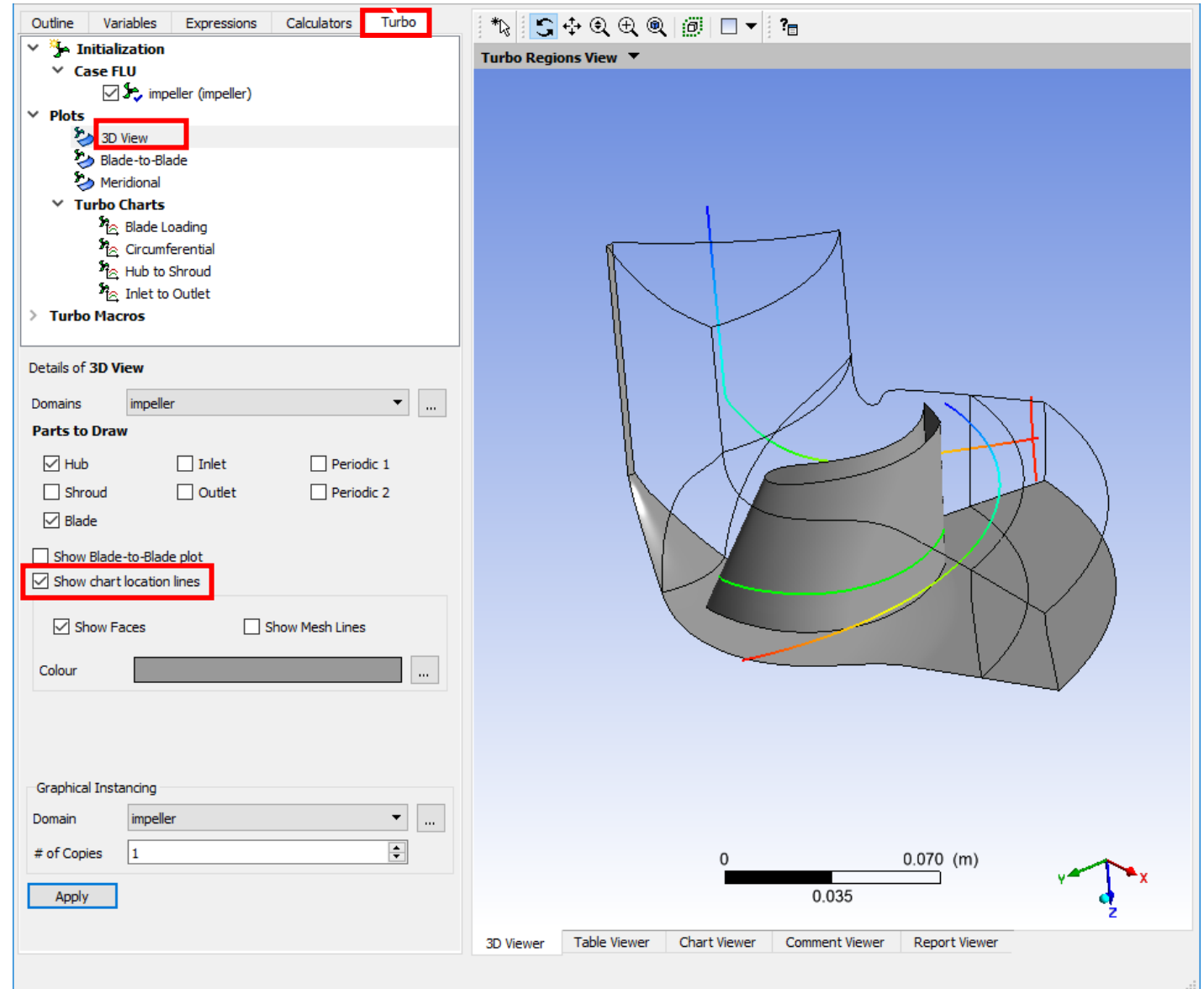
- To visualize how the pressure increases from inlet to outlet:
 - Select *Inlet to Outlet* under *Turbo Chart*
 - Set *Samples/Comp.* to 100
 - Set *X Axis Variable* to *Streamwise Location*
 - Set *Y Axis Variable* to *Pressure*
 - Click *Apply*



Visualizing the location of the chart lines

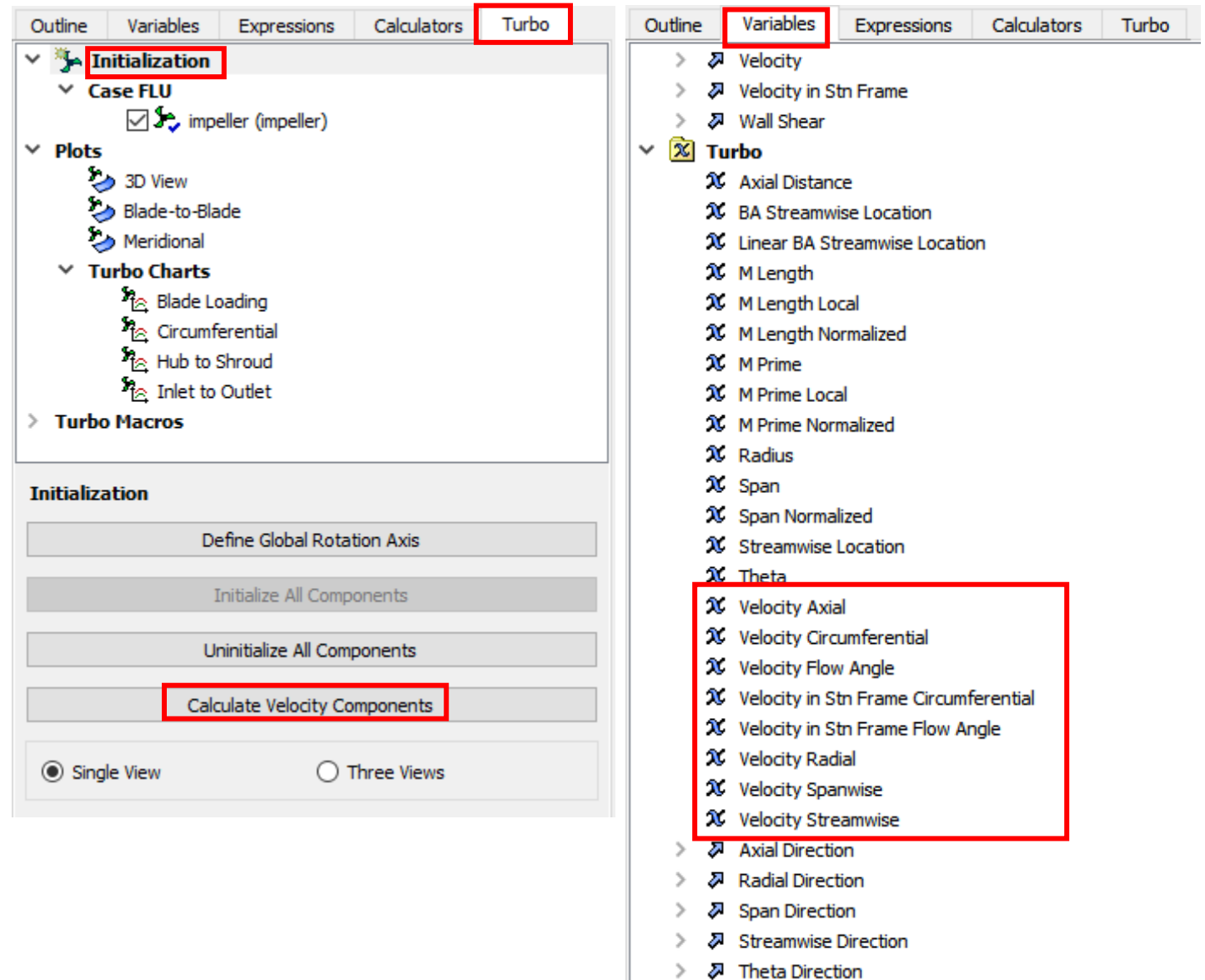
- To visualize where the previous data is being obtained:
 - Select *3D View* under *Plots* in the *Tree*
 - Check the *Show chart location lines* checkbox
 - Click *Apply*
- The lines used to create the Turbo charts are shown
 - Blade loading chart at 25% span
 - Circumferential chart at 90% span at a streamwise location of 1.9
 - Hub to Shroud chart near the outlet
 - Inlet to Outlet chart
- This enables you visualize and check where chart lines are created

To visualize one passage as shown on the right, you might have to switch to a # of Copies of 2, click *Apply* and then set it back to 1 and click *Apply* once more



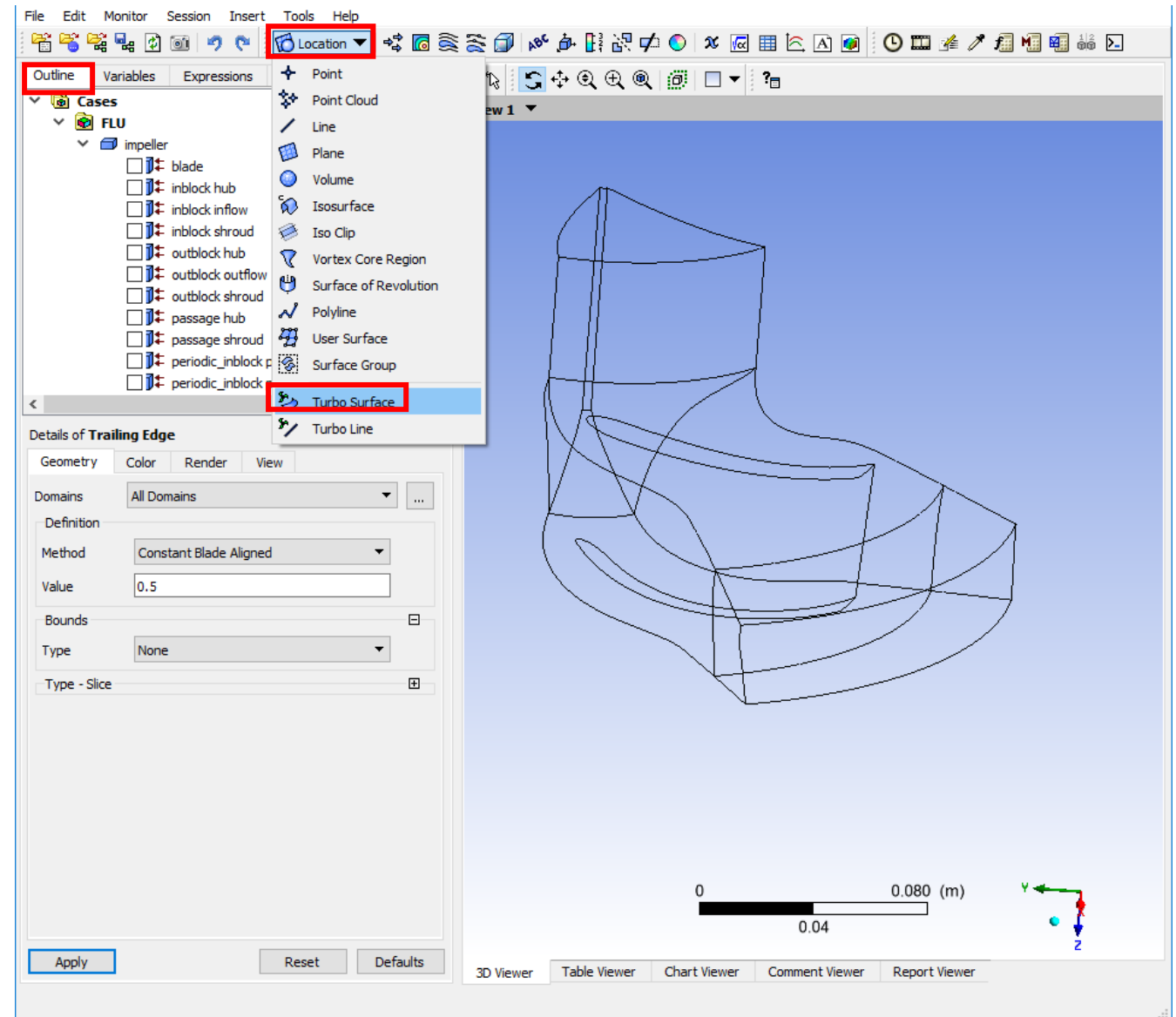
Calculate Velocity Components

- Some additional variables useful for turbo post processing can be created
 - Still in the Turbo Tab
 - Double click *Initialization*
 - Click *Calculate Velocity Components*
- This creates a number of new velocity variables, such as:
 - Velocity Axial
 - Velocity Circumferential
 - Velocity Flow Angle
 - Velocity Radial
 - Velocity Spanwise
 - Velocity Streamwise
 - ...



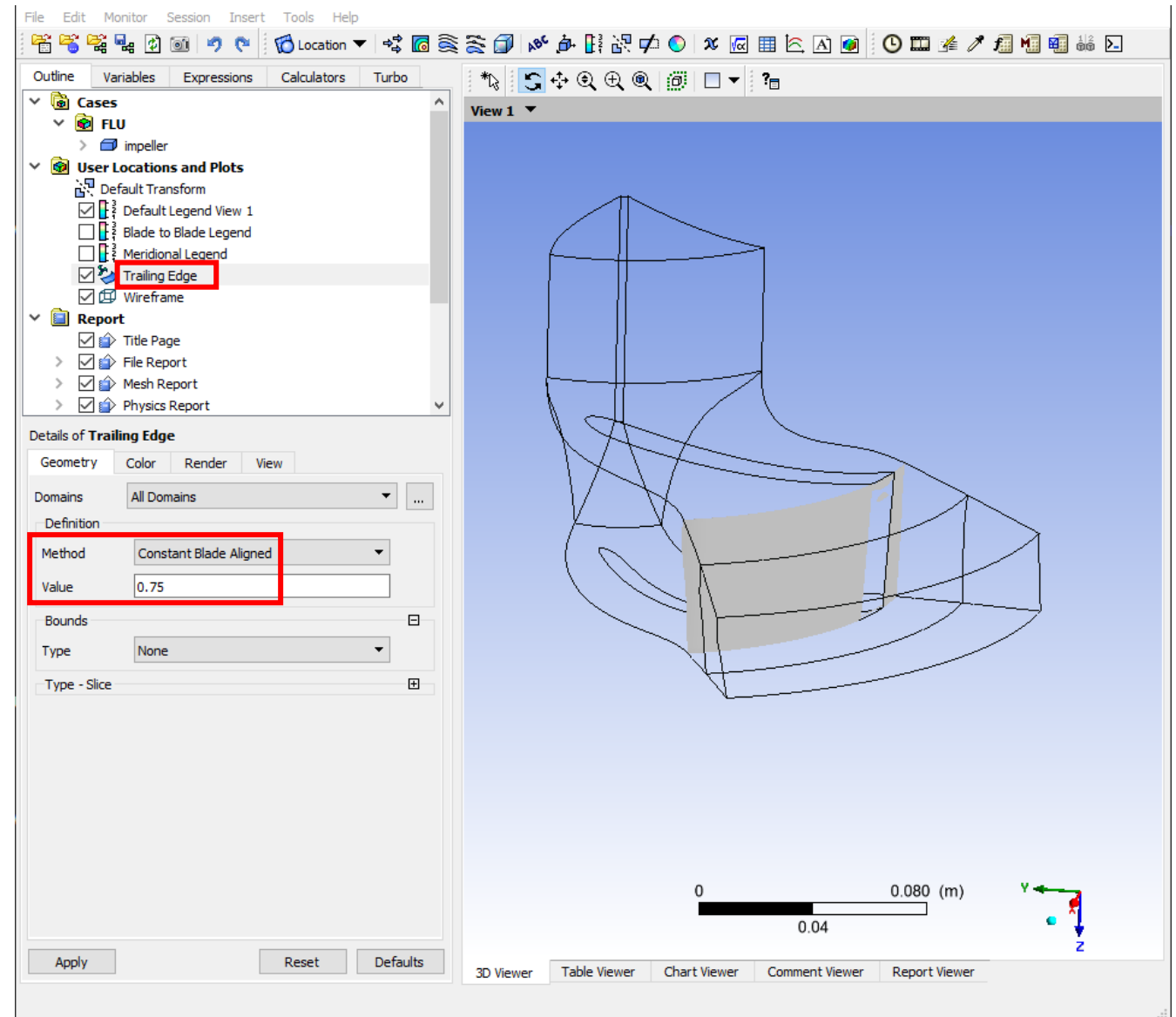
Turbo Surface (1)

- You can create Turbo Surfaces in the standard 3D Viewer
 - Switch to the *Outline* Tab at the top left
 - *Location* > *Turbo Surface*
 - Rename the *Turbo Surface* to *Trailing Edge*



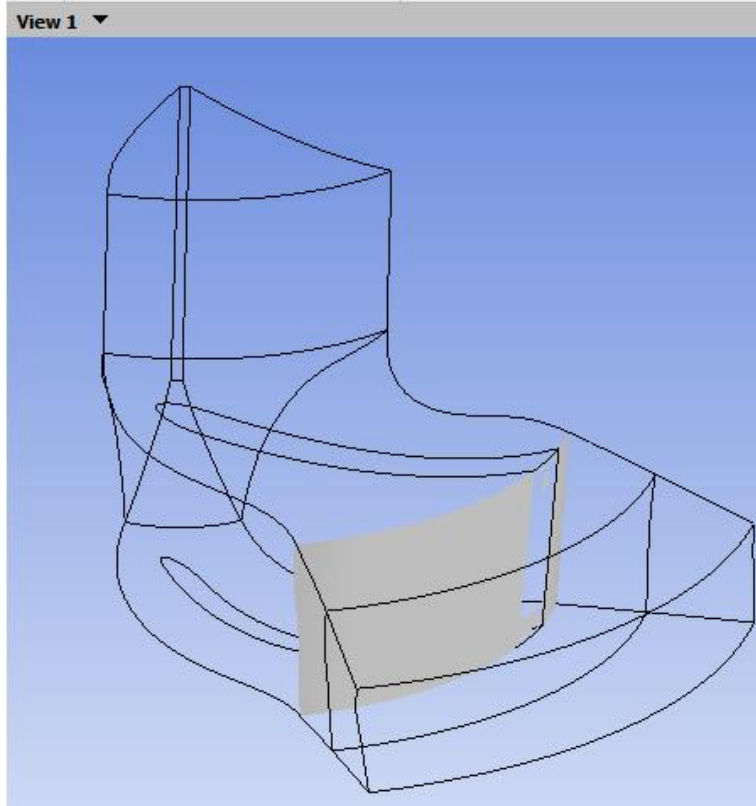
Turbo Surface (2)

- Details of *Trailing Edge Turbo Surface*
 - Set *Method* to *Constant Blade Aligned*
 - Set *Value* to 0.75
- We can see this surface intersects the trailing edge of the blade
- Move the plane slightly downstream so that it is entirely behind the trailing edge
 - Set *Value* to 0.755

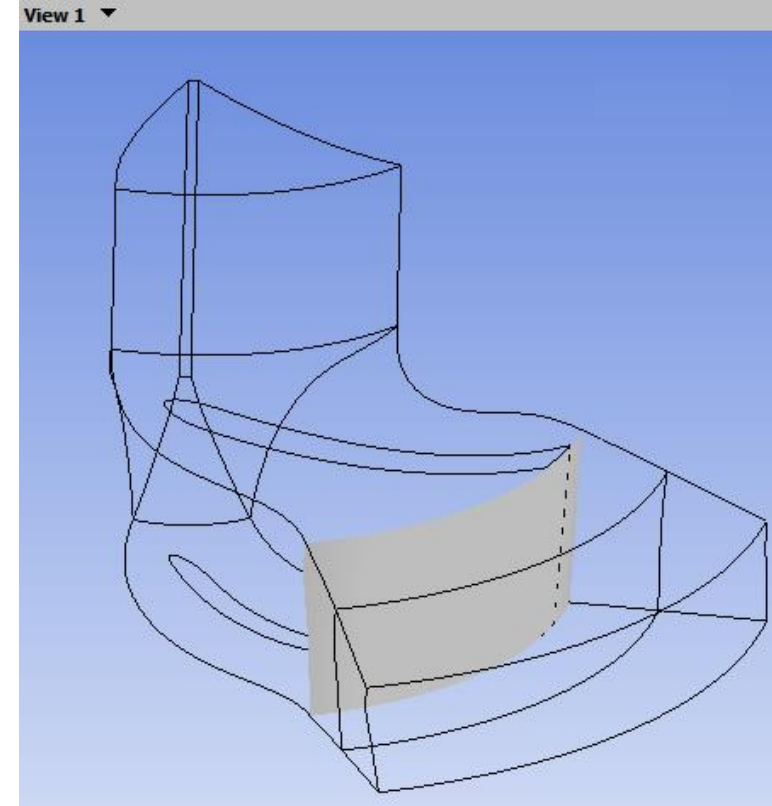


/ Constant Blade Aligned

Constant Blade Aligned = 0.75

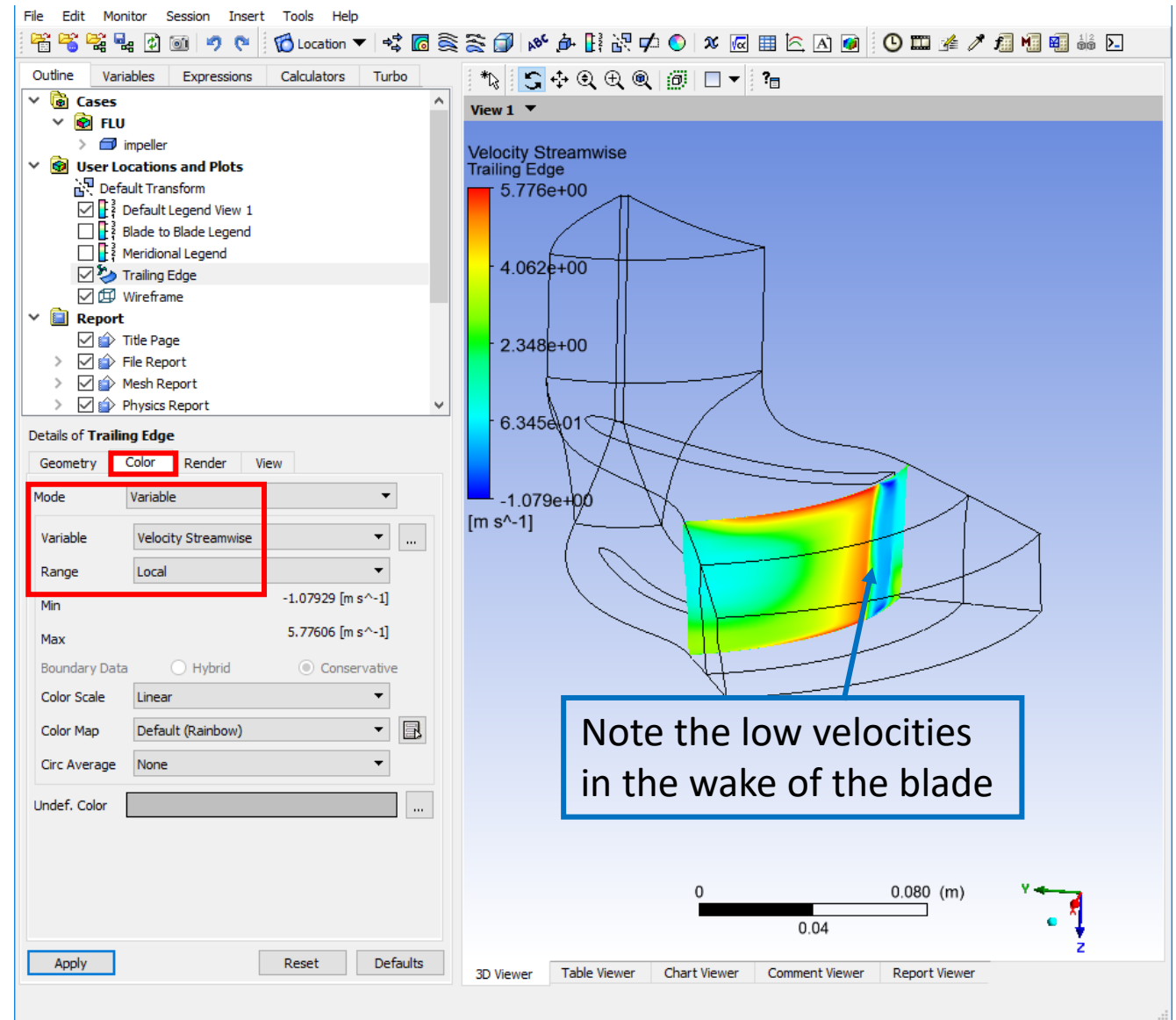


Constant Blade Aligned = 0.755



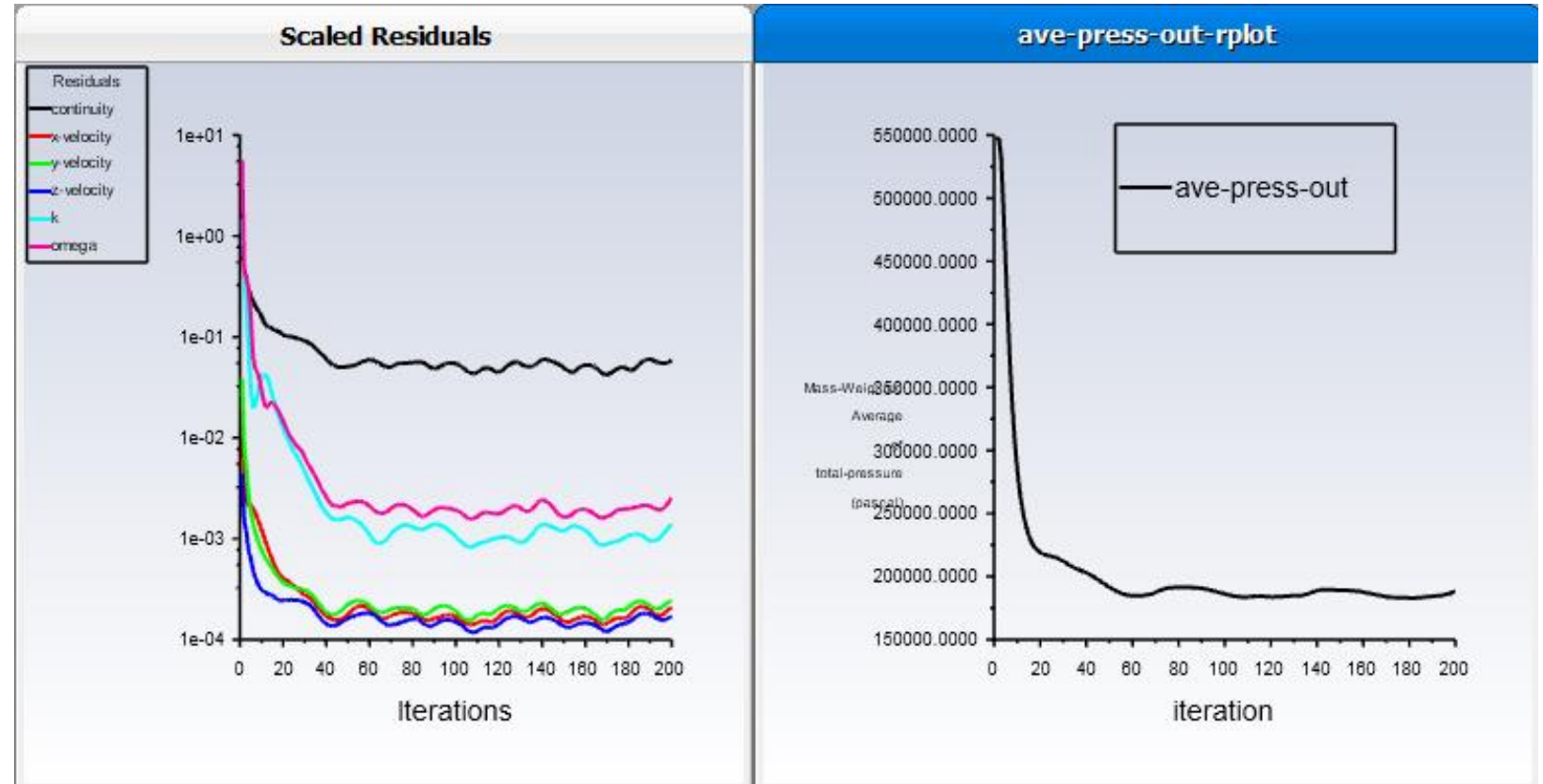
Colour Blade Aligned Surface

- Color the *Turbo Surface* by the Turbo Post variable *Velocity Streamwise*
 - Switch to the *Color* Tab under the *Details of Trailing Edge*
 - Set *Mode* to *Variable*
 - Set *Variable* to *Velocity Streamwise*
 - Set *Range* to *Local*
 - Click *Apply*



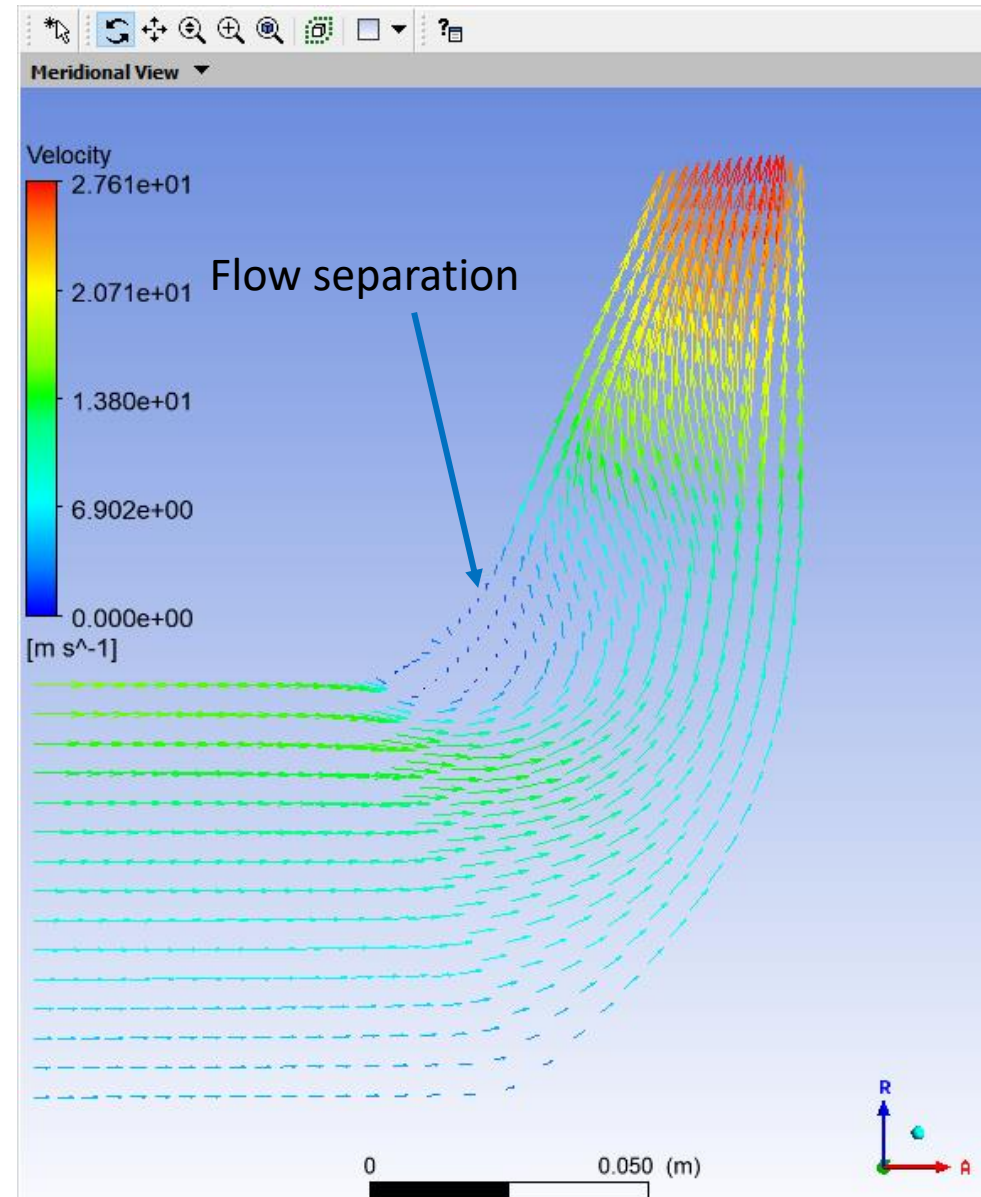
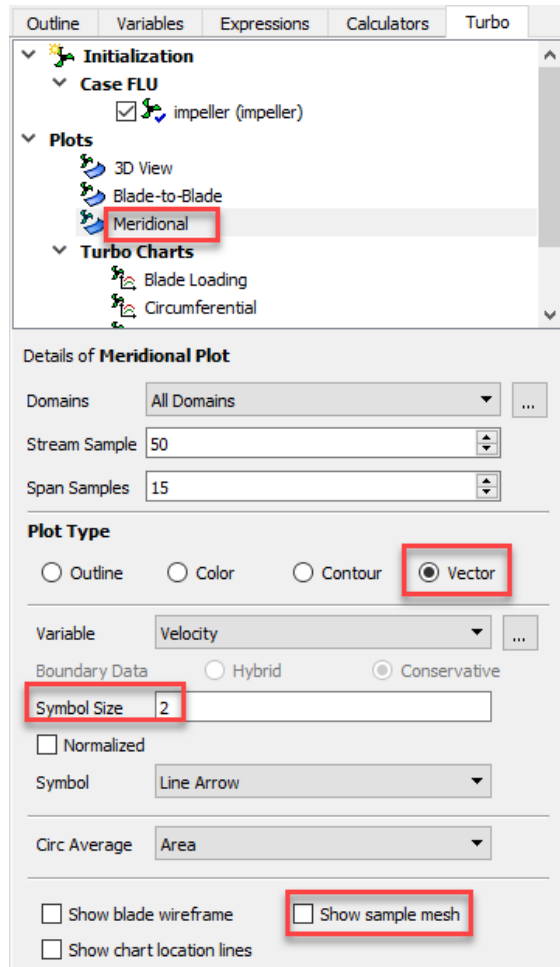
Investigation of Case-Convergence Issue

- In workshop 02.1 it was found that the case is not converging well
- Here we will show why
- As will be seen in the next 2 slides, there is significant flow separation from the shroud
- This indicates that the shroud is not designed correctly, hence the poor convergence



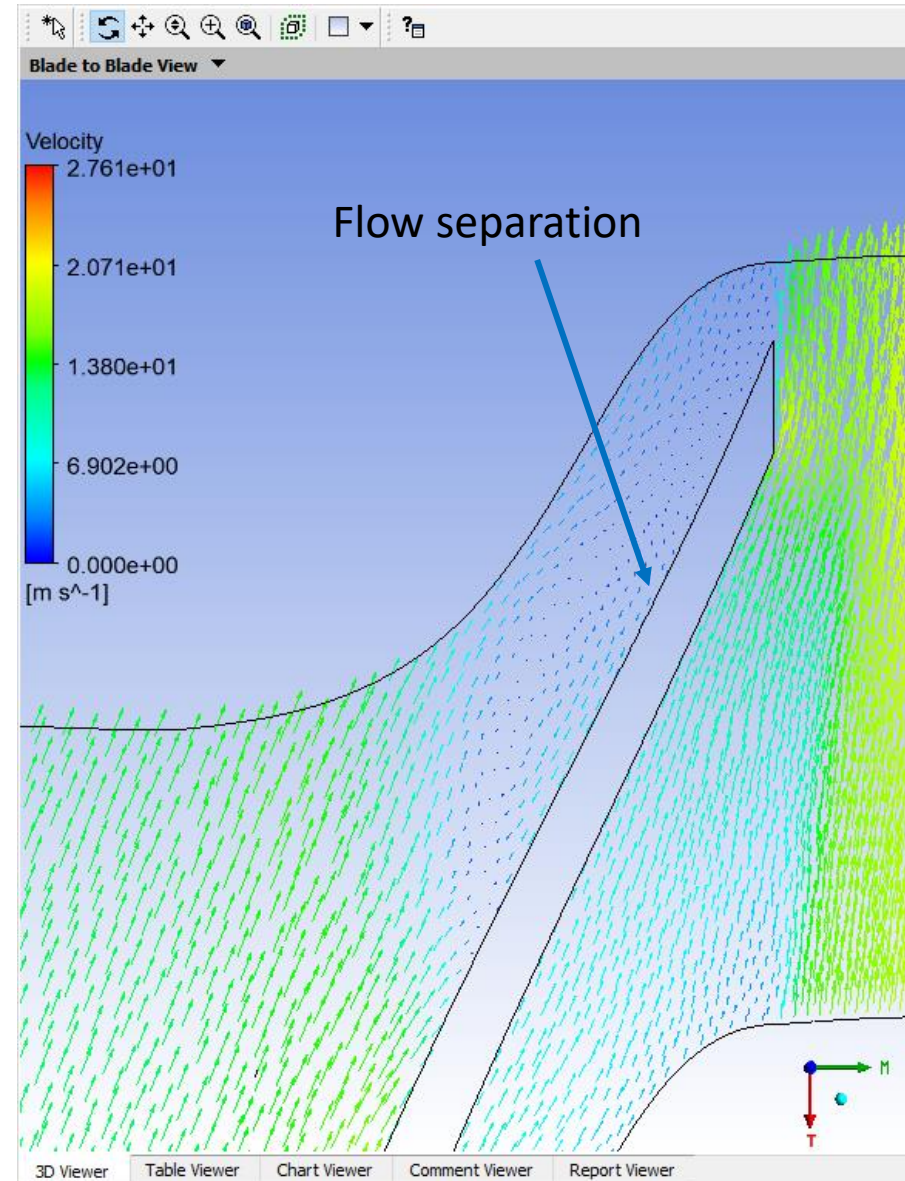
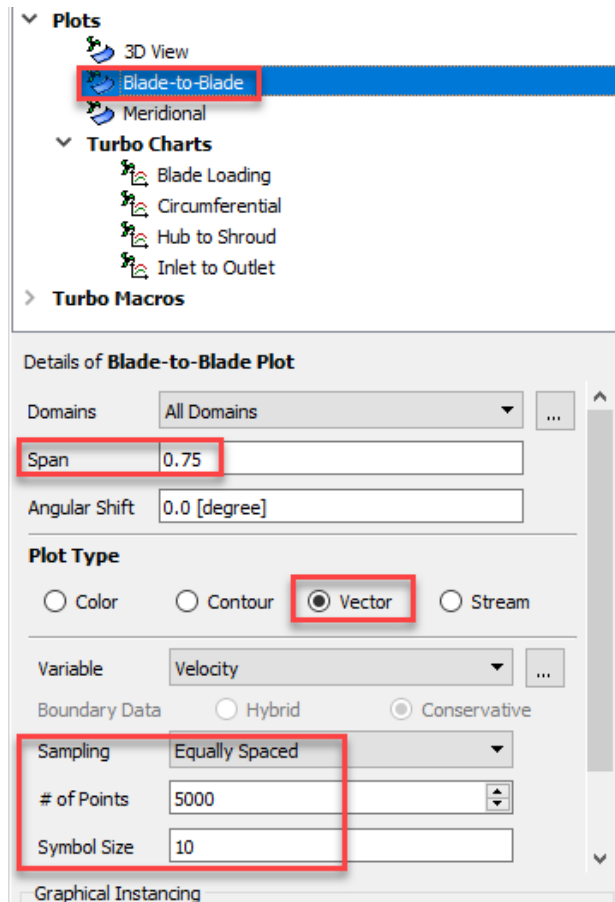
Meridional Velocity Vectors

- Create a Meridional Vector Plot with the settings shown below



Blade-to-Blade Velocity Vectors

- Create a Blade-to-Blade Vector Plot with the settings shown below



When done, save the Workbench project and exit CFD-Post

/ Summary

- This workshop demonstrated the following:
 - Setting up turbo specific post-processing views
 - Meridional
 - Blade to Blade
 - Creating turbo specific charts
 - Blade Loading
 - Circumferential
 - Inlet to Outlet
 - Hub to Shroud
 - Creating turbo specific surfaces on which to plot vectors, contours, etc.
 - Using turbo specific variables
 - Investigating the poor convergence of this pump design



End of presentation