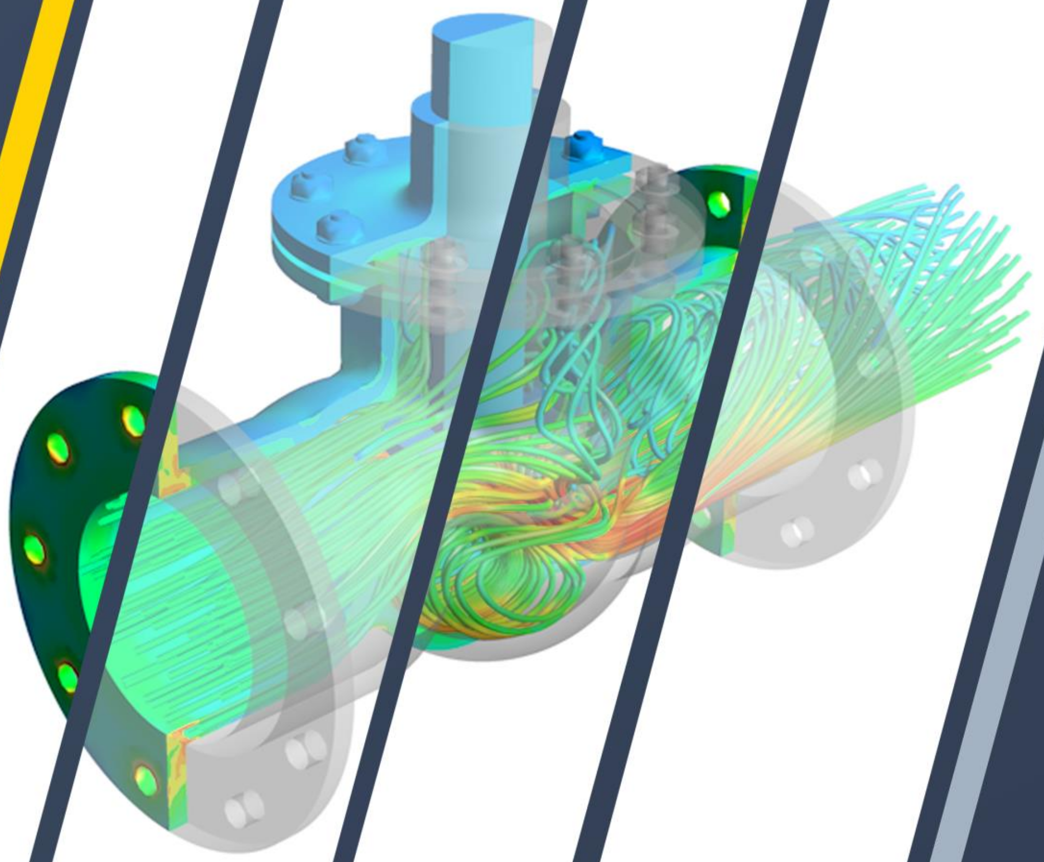
ANSYS

Workshop 03.1: Pump Analysis using CFD Post

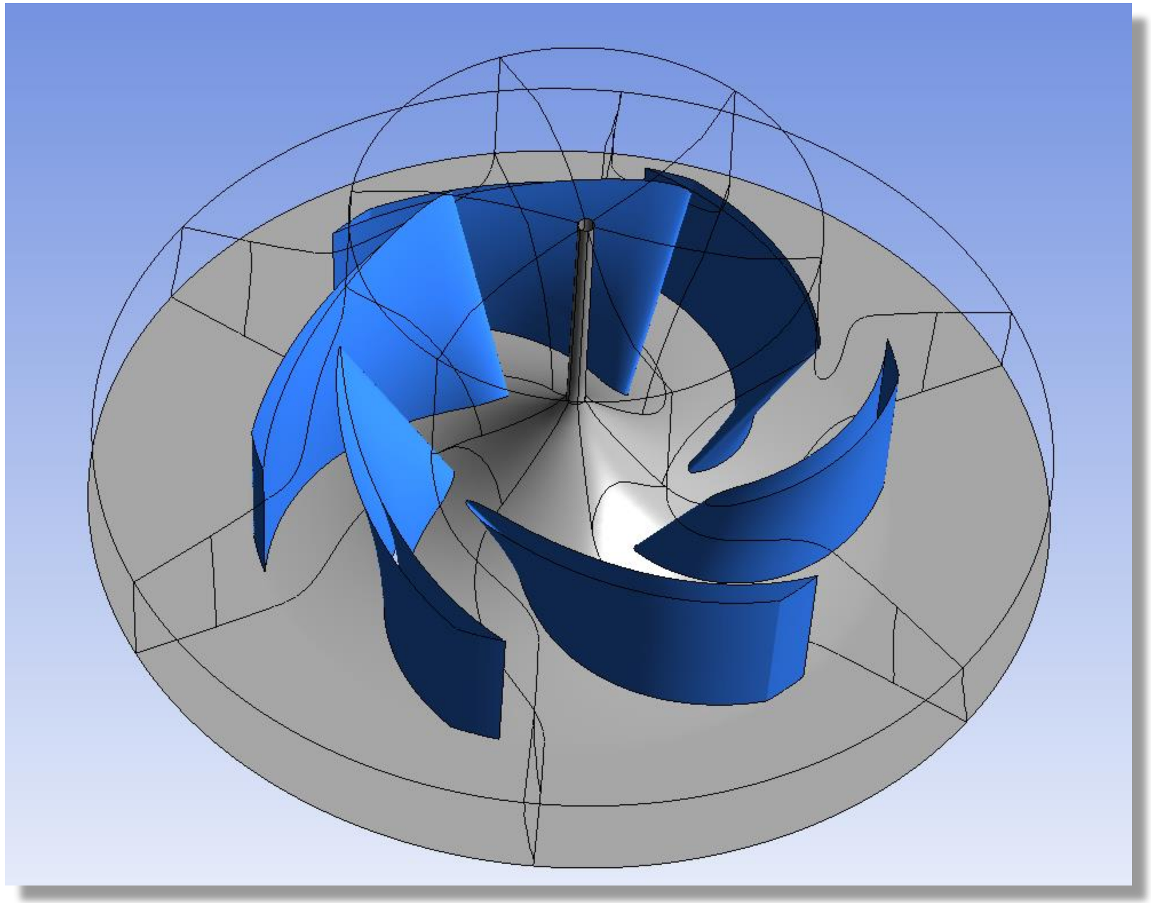
ANSYS CFX Rotating Machinery Modeling

Release 2019 R3



Introduction

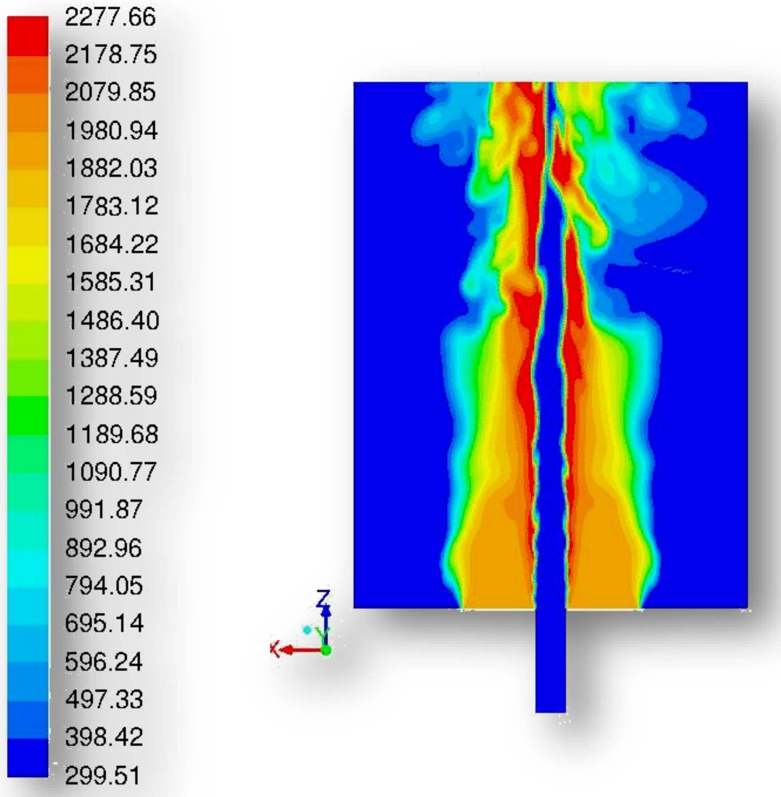
- Workshop Description:
  - This Workshop deals with post-processing aspect for the pump impeller solved in Workshop 1
- Learning Aims:
  - Turbo-specific post-processing
  - Creating CEL expressions to calculate various relevant quantities
  - Report generation



© 2019 ANSYS, Inc.

CFD-Post

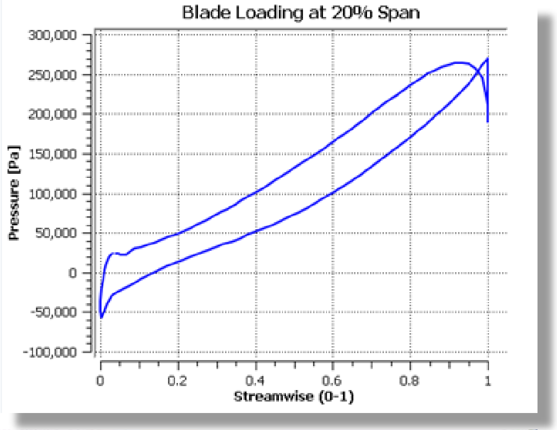
- *CFD-Post* includes many tools for analyzing general CFD results
  - Isosurfaces
  - Vector plots
  - Contour plots (shaded and graded)
  - Streamlines and particle tracks
  - XY plotting
  - Animation creation

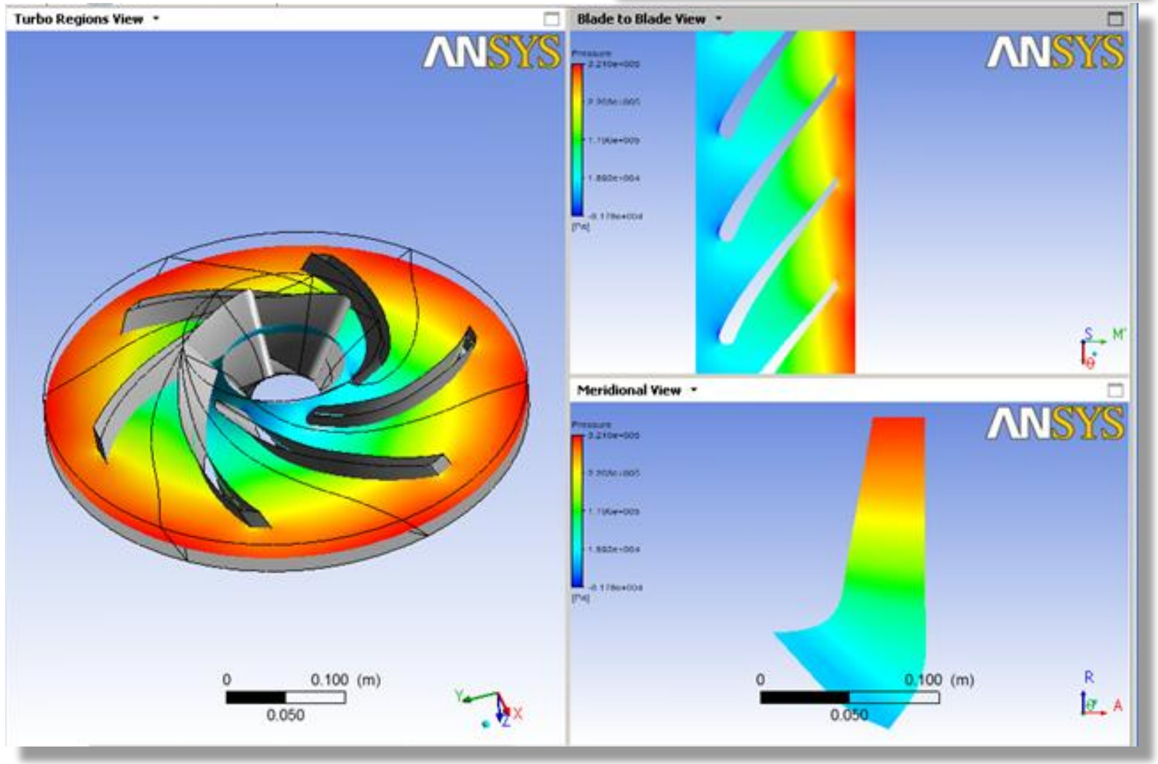


© 2019 ANSYS, Inc.

Post Processing

- *CFD-Post* also has turbo specific post-processing capabilites
  - Turbo Specific Post processing views
    - ➢ Meridional
    - ➢ Blade to Blade
  - Turbo Specifc charts
    - ➢ Blade Loading
    - ➢ Circumferential
    - ➢ Inlet to Outlet
    - ➢ Hub to Shroud
  - Turbo Specific surfaces on which to plot vectors, contours, etc.
  - Turbo Specific variables
  - Reports for different types of rotating machines





© 2019 ANSYS, Inc.

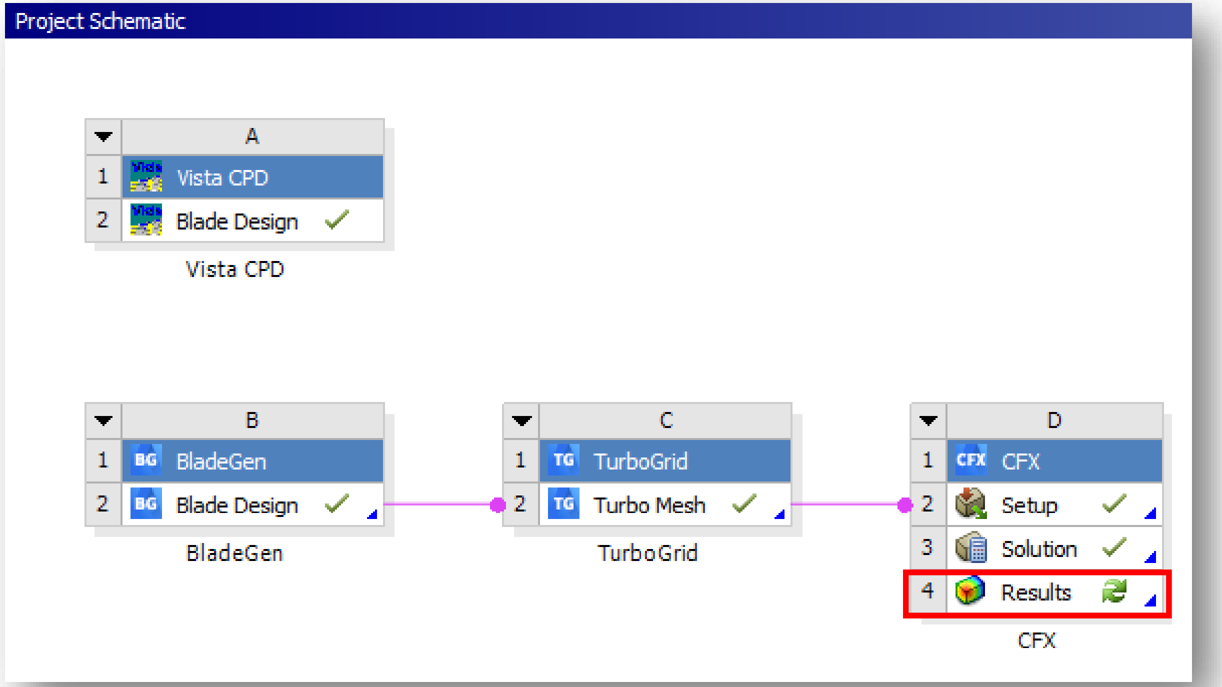
Post-processing Goals

- Configure the solution in Turbo Post
- Calculate turbo velocity components
- Create plots in blade to blade and meridional views
- Create charts for blade loading
- Use CEL to calculate some global quantitative results
- Generate a pre-defined pump report
- Add some customer plots to the report

© 2019 ANSYS, Inc.

Launch CFD-Post

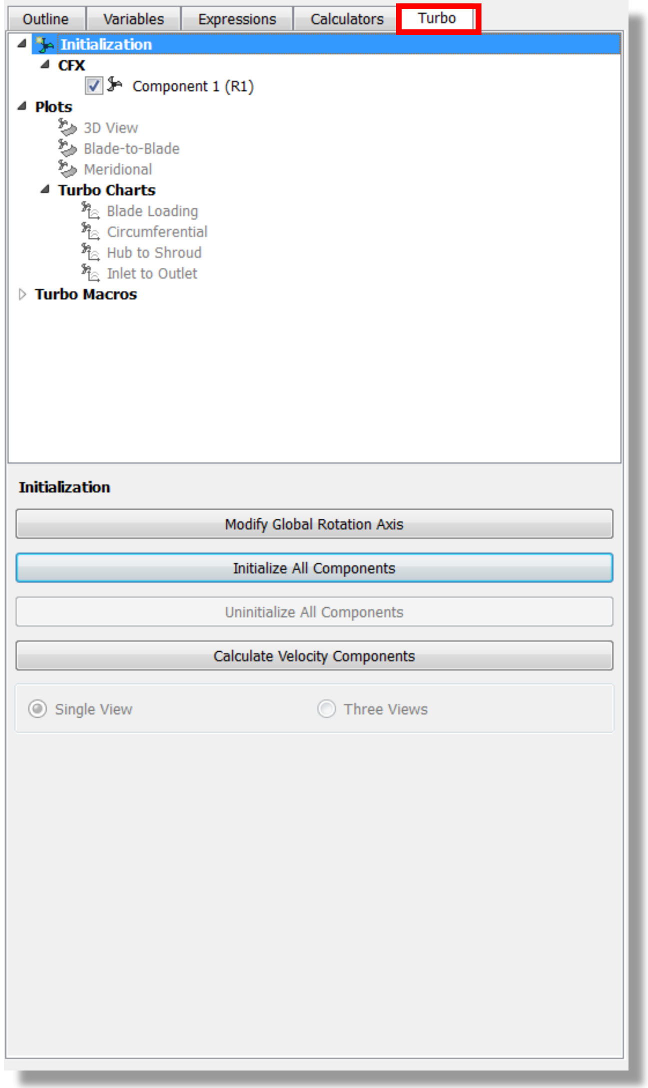
- Start *CFD-Post*
  - If you completed Workshop 1, you can double click on the Results cell to launch *CFD-Post*
  - Alternatively, you can load the completed Workbench archive *Pump_solution.wbpz* (provided with the inputs of this workshop)
    - In the Workbench main menu *File > Open…*
      - In the *Open* dialogue box *Browse to Pump_solution.wbpz* and click *Open*
      - In the *Save As* dialogue box edit the *File Name* to *Pump_results.wbpj* and click *Save*



© 2019 ANSYS, Inc.

Turbo Post

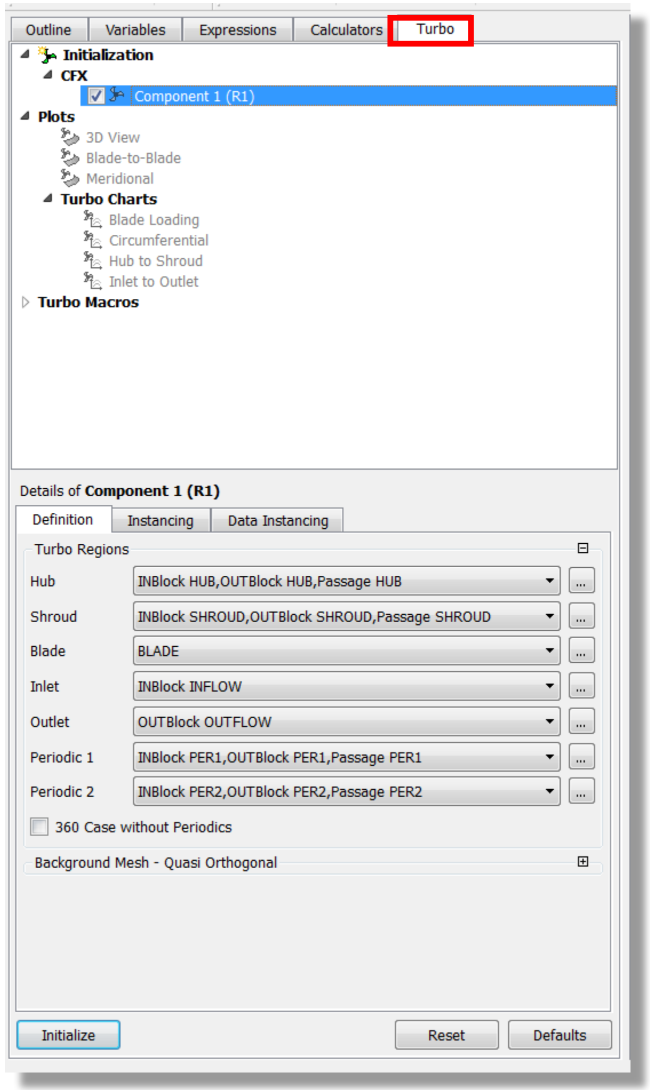
- Once the case is opened in *CFD-Post*, switch to the *Turbo* Tab
- If prompted to automatically initialize the case, select *No*



© 2019 ANSYS, Inc.

Initialization

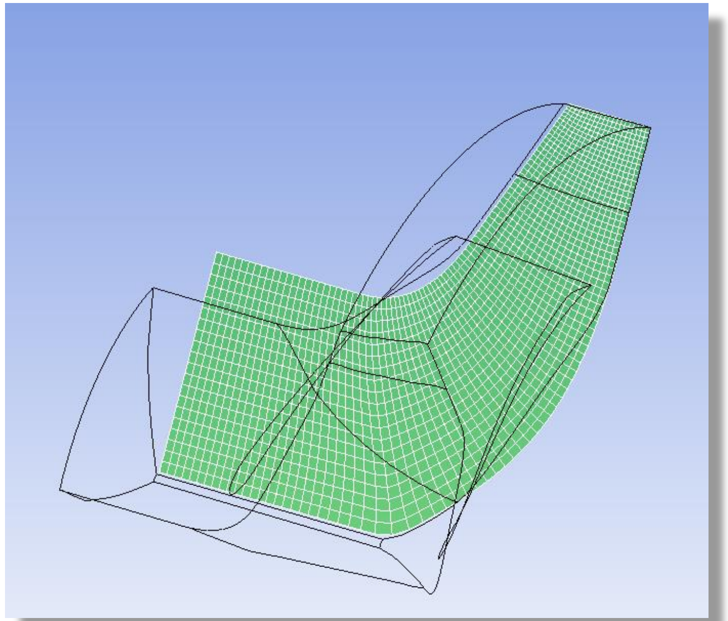
- Initializing the components is the first step
  - This process defines a new set of coordinates and variables that are useful for turbo post-processing
  - These coordinates and variables are based on the hub, shroud, inlet, outlet, blade, and periodic boundaries
  - By default *CFD-Post* will try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

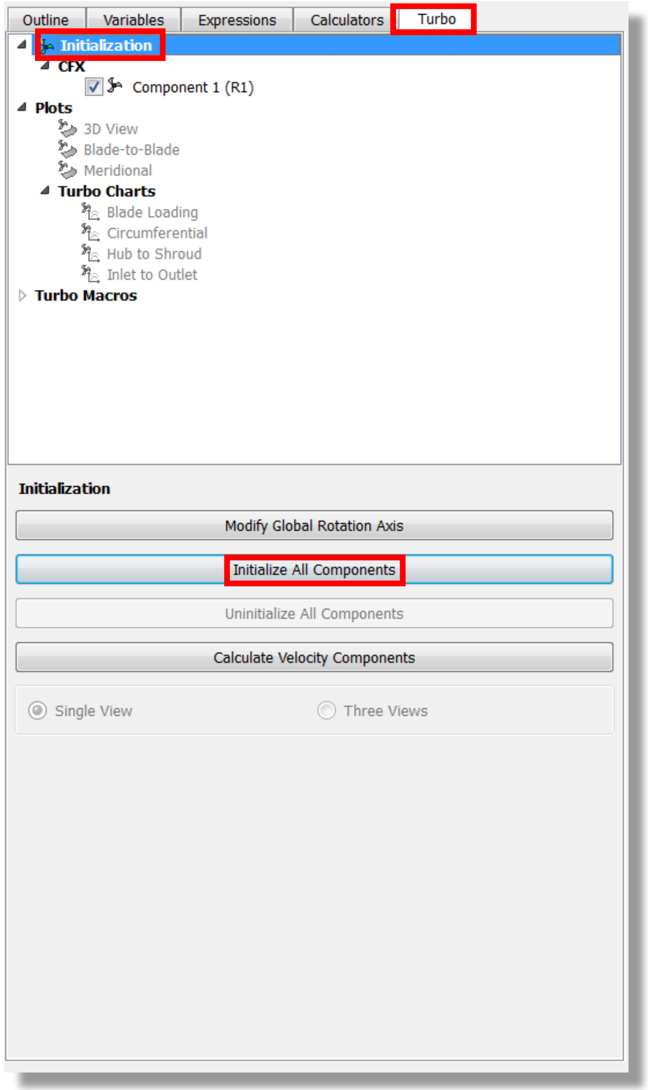


© 2019 ANSYS, Inc.

Initialization

- Under *Initialization* in the *Turbo* Tree, click *Initialize All Components*
  - This will result in a background mesh being displayed (when clicking on *Component 1* in the *Turbo* Tree)

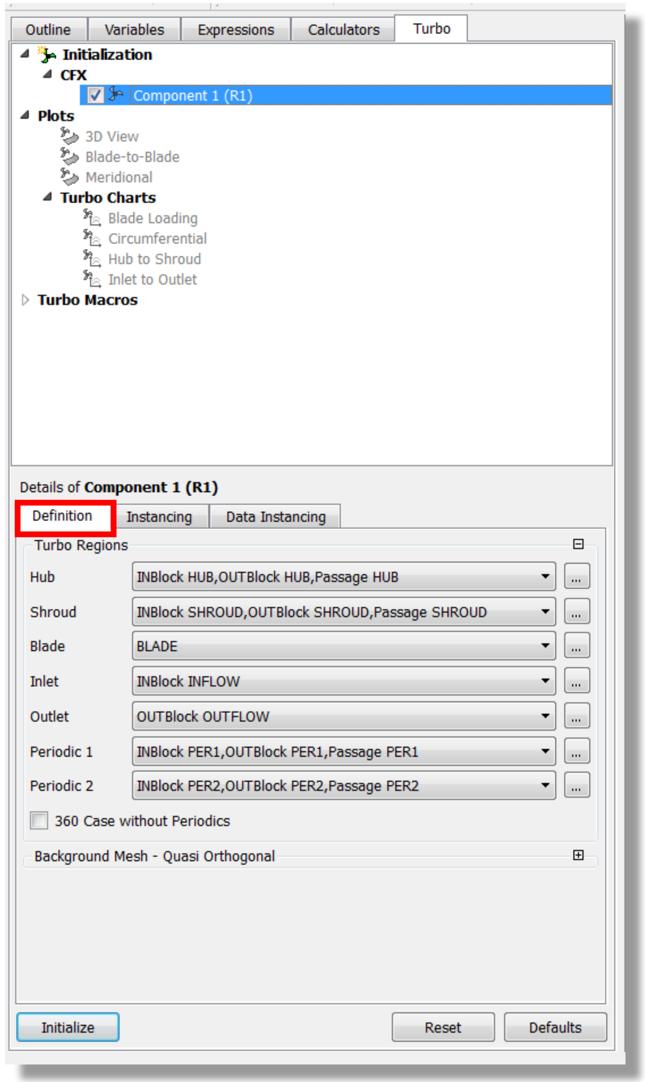




© 2019 ANSYS, Inc.

Initialization

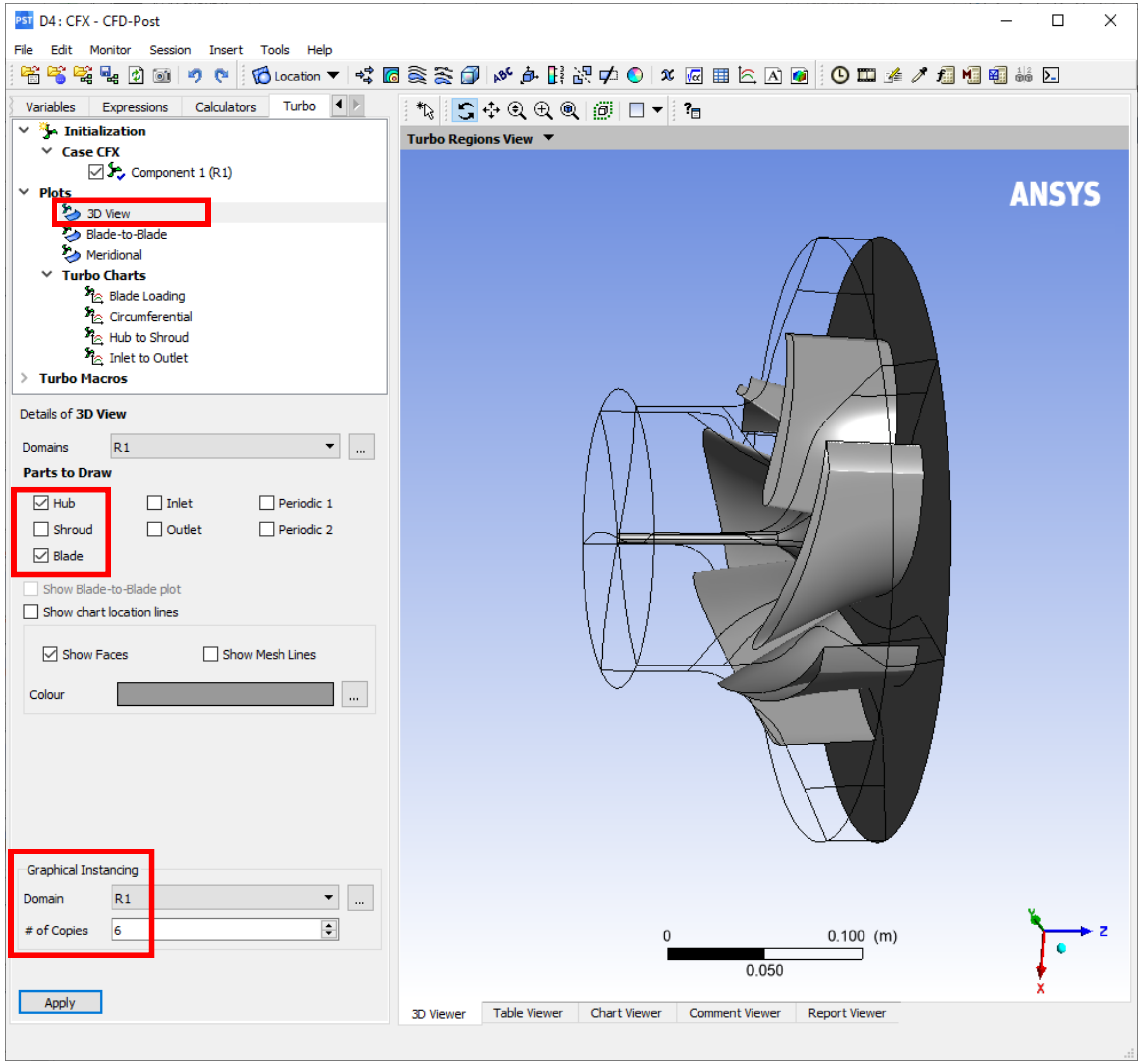
- In this case, all turbo regions have been automatically identified
- If they had not, the user could identify them in the *Definition* Tab



© 2019 ANSYS, Inc.

Plots: 3D View

- To visualize the pump geometry (still in the *Turbo* Tab) click on *3D View*
  - Select *Hub* and *Blade* under *Parts to Draw*
  - Select *R1* under *Graphical Instancing* and set the *# of Copies* to 6
  - Click *Apply*



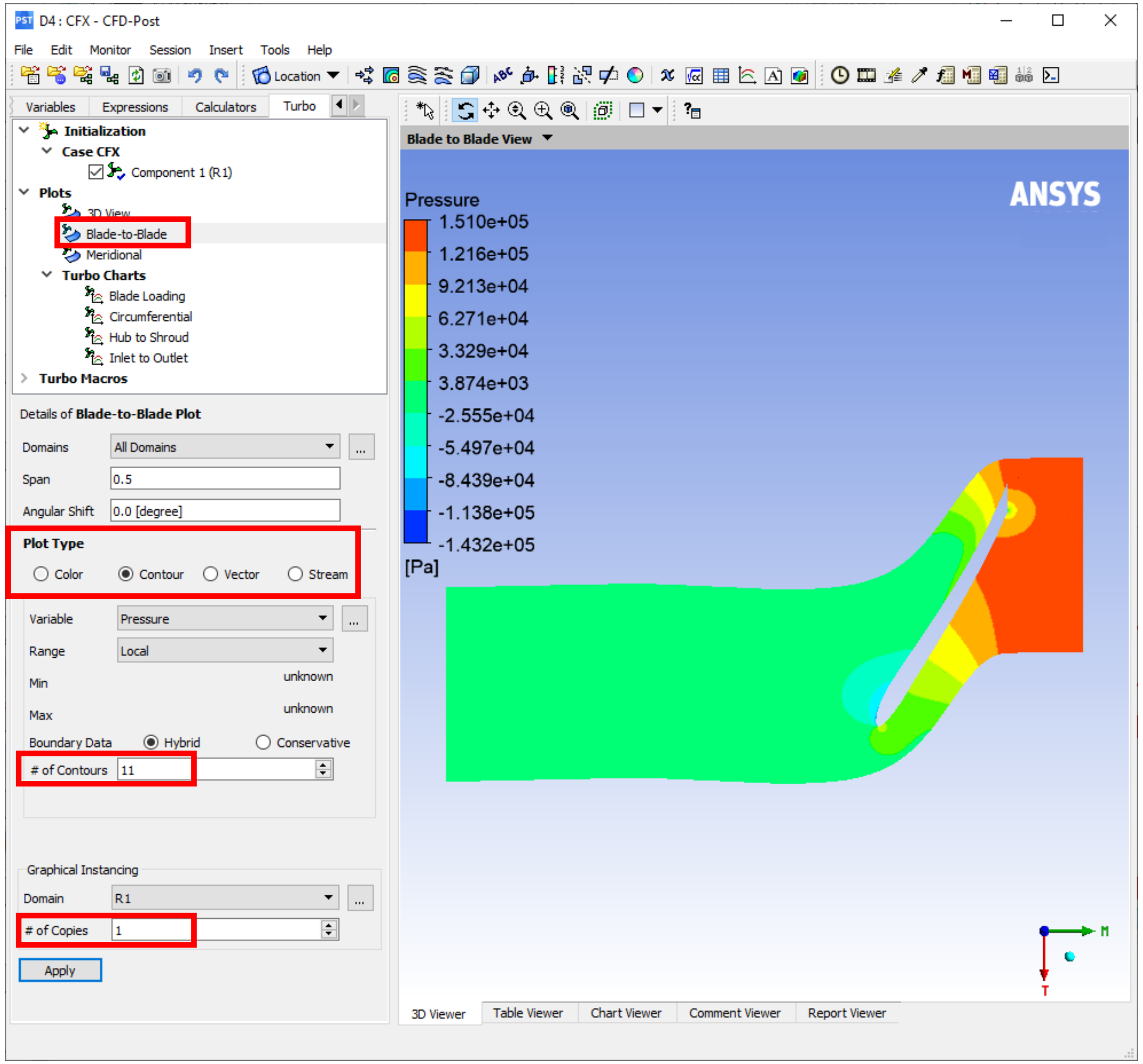
In the remaining slides numerous post-processing objects (flow visualization images, charts, tables, etc.) will be generated. Please notice that there can be differences between the indicative images & values given in the slides and the ones created by you

© 2019 ANSYS, Inc.

Plots: Blade-to-Blade

- Make an unwrapped* blade-to-blade plot of pressure at mid-span
  - Under *Blade-to-Blade* in the Tree, set *Contour* for *Plot Type*
  - Set *Local* for *Range*
  - Set *# of Copies* back to 1
  - Click *Apply*

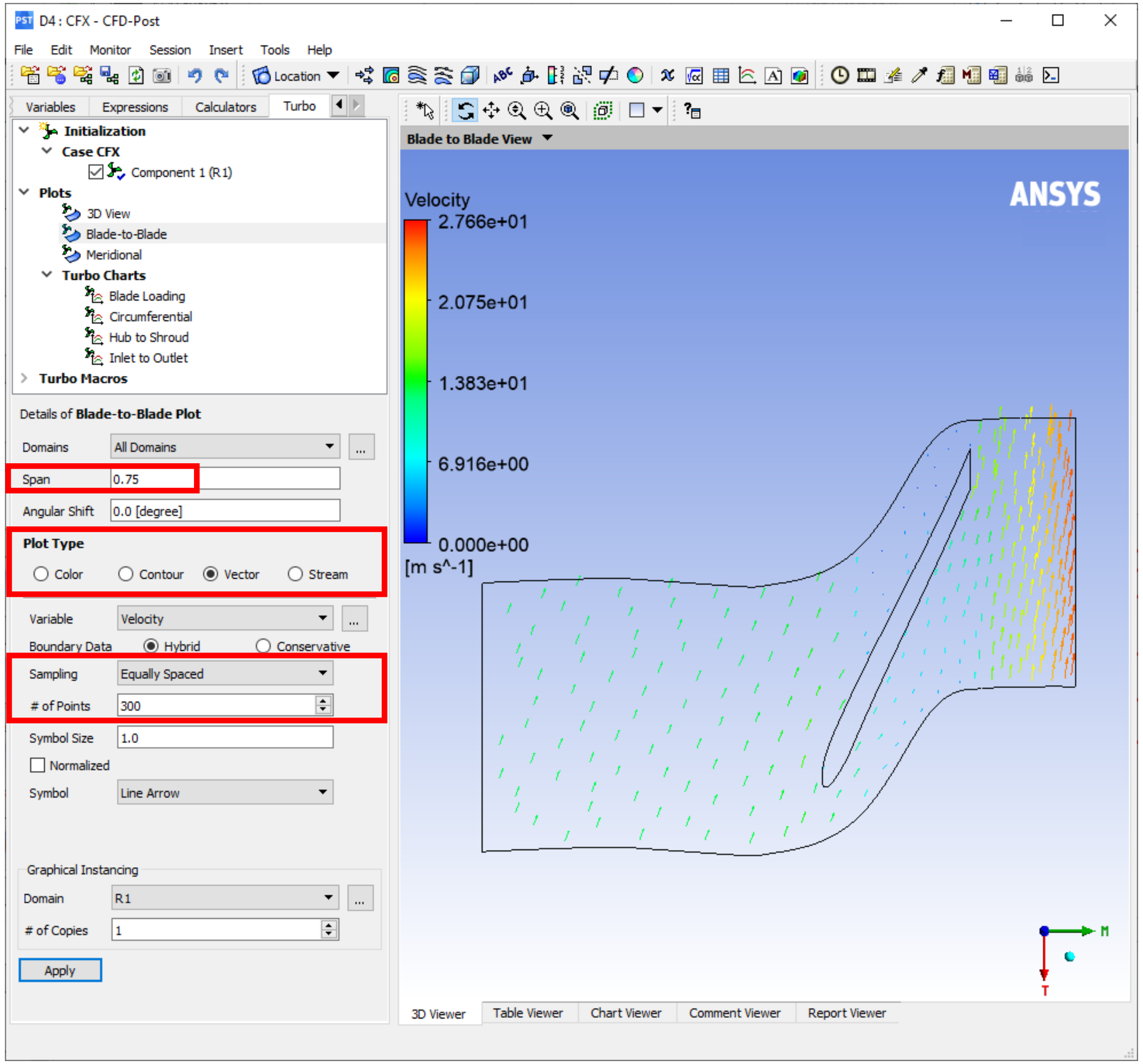
* An “unwrapped” plot is one where the 3D surface contours are transformed so that they lie in a plane, similar to taking a curved piece of paper and unwrapping it to lie on a flat table



© 2019 ANSYS, Inc.

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 300
  - Click *Apply*

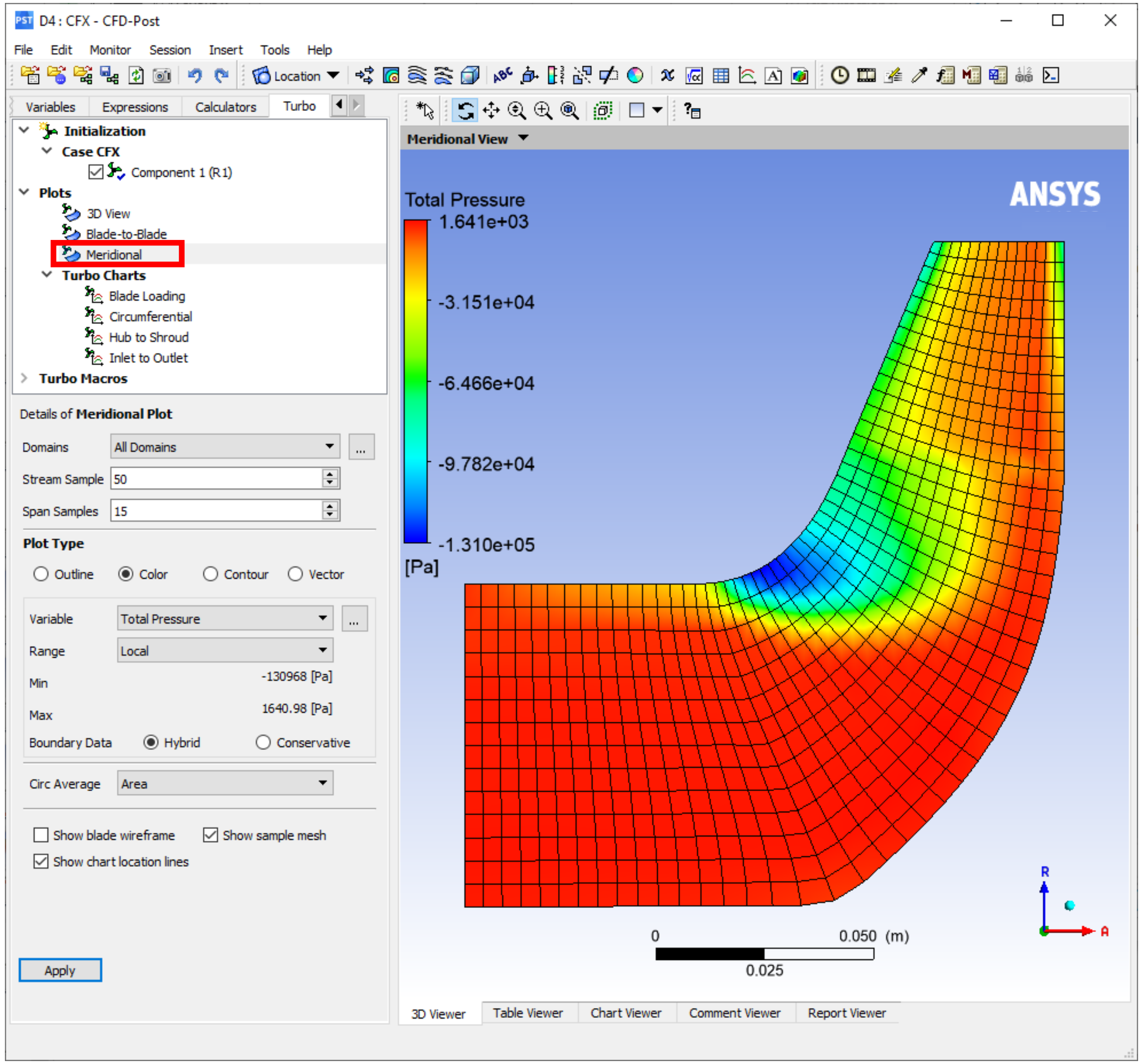


© 2019 ANSYS, Inc.

Meridional Plot

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

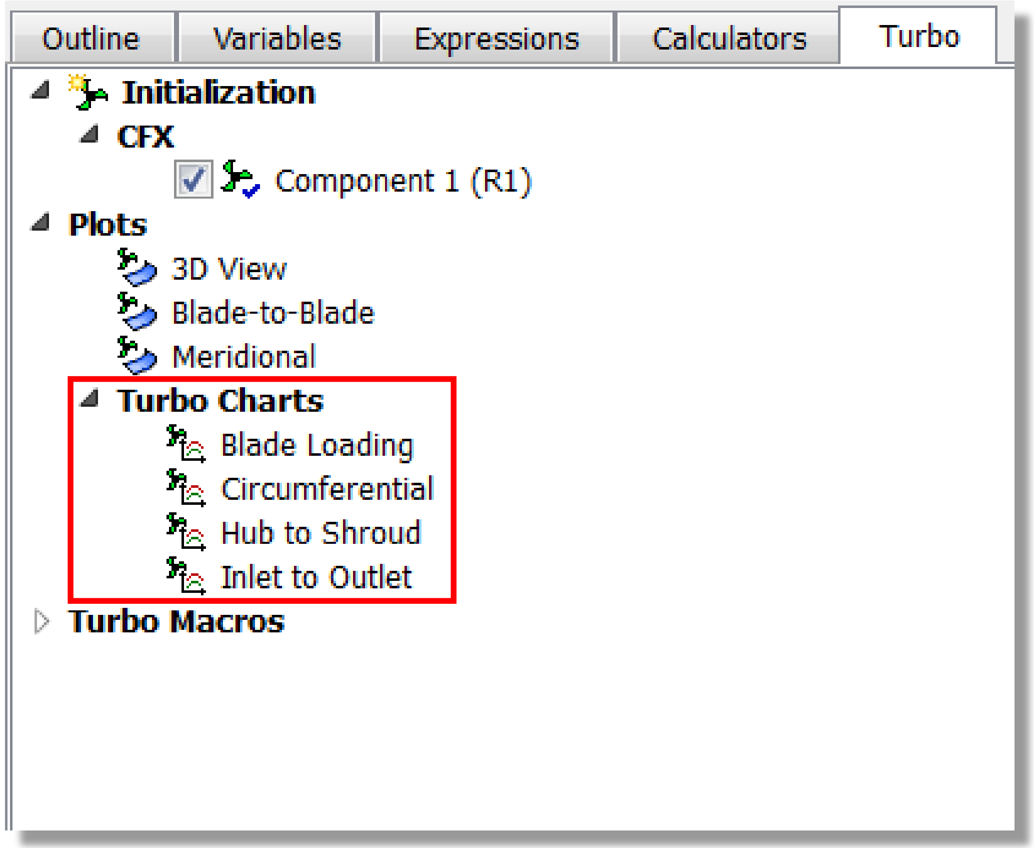
* A meridional view depicts the passage flow from inlet to outlet, hub to shroud using a circumferential average of the flow properties at a given position within the passage



© 2019 ANSYS, Inc.

Turbo Charts

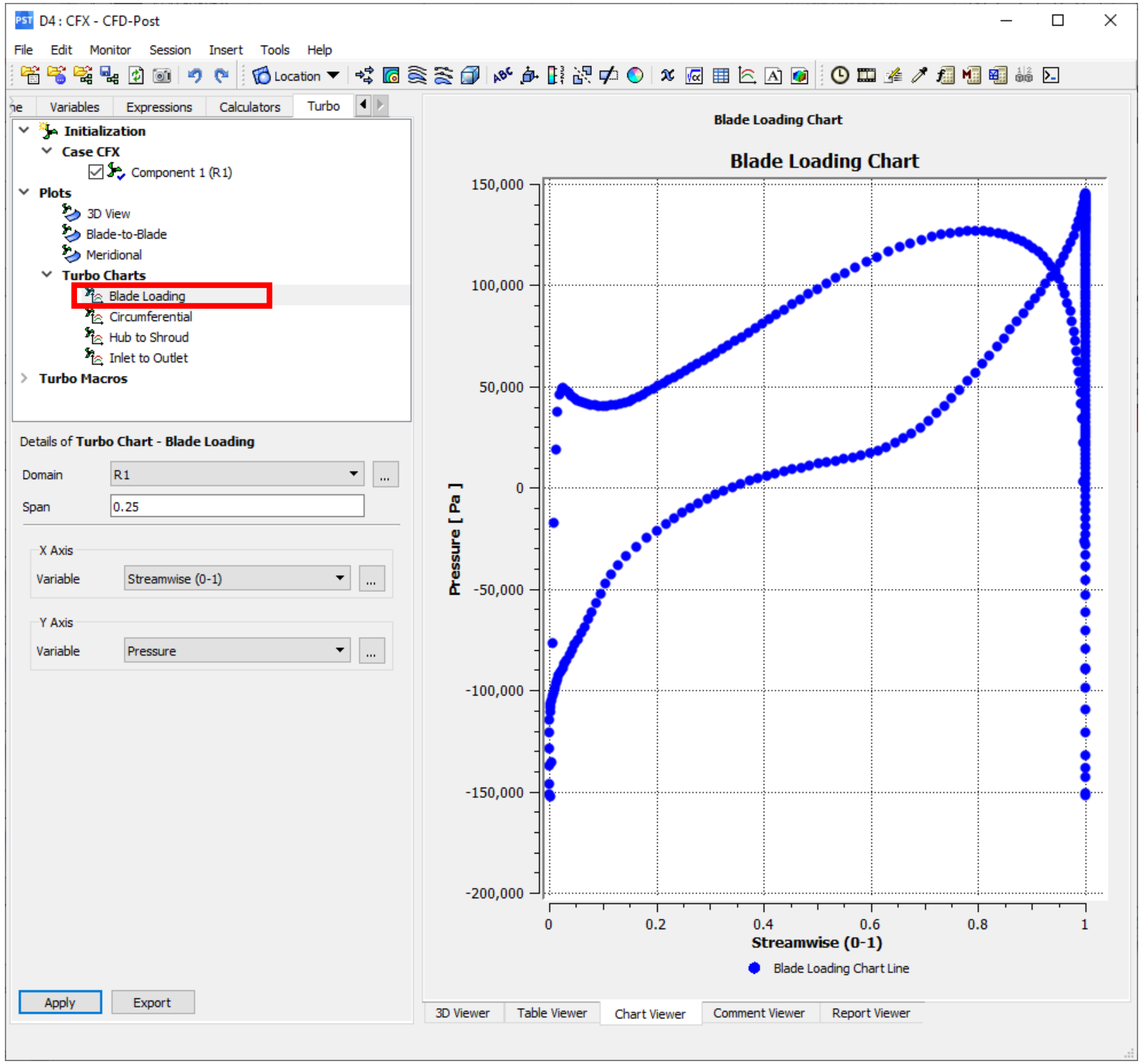
- There are four different types of 2D plots that can be created:
  - *Blade Loading*
  - *Circumferential*
  - *Hub to Shroud*
  - *Inlet to Outlet*



© 2019 ANSYS, Inc.

Blade Loading Chart

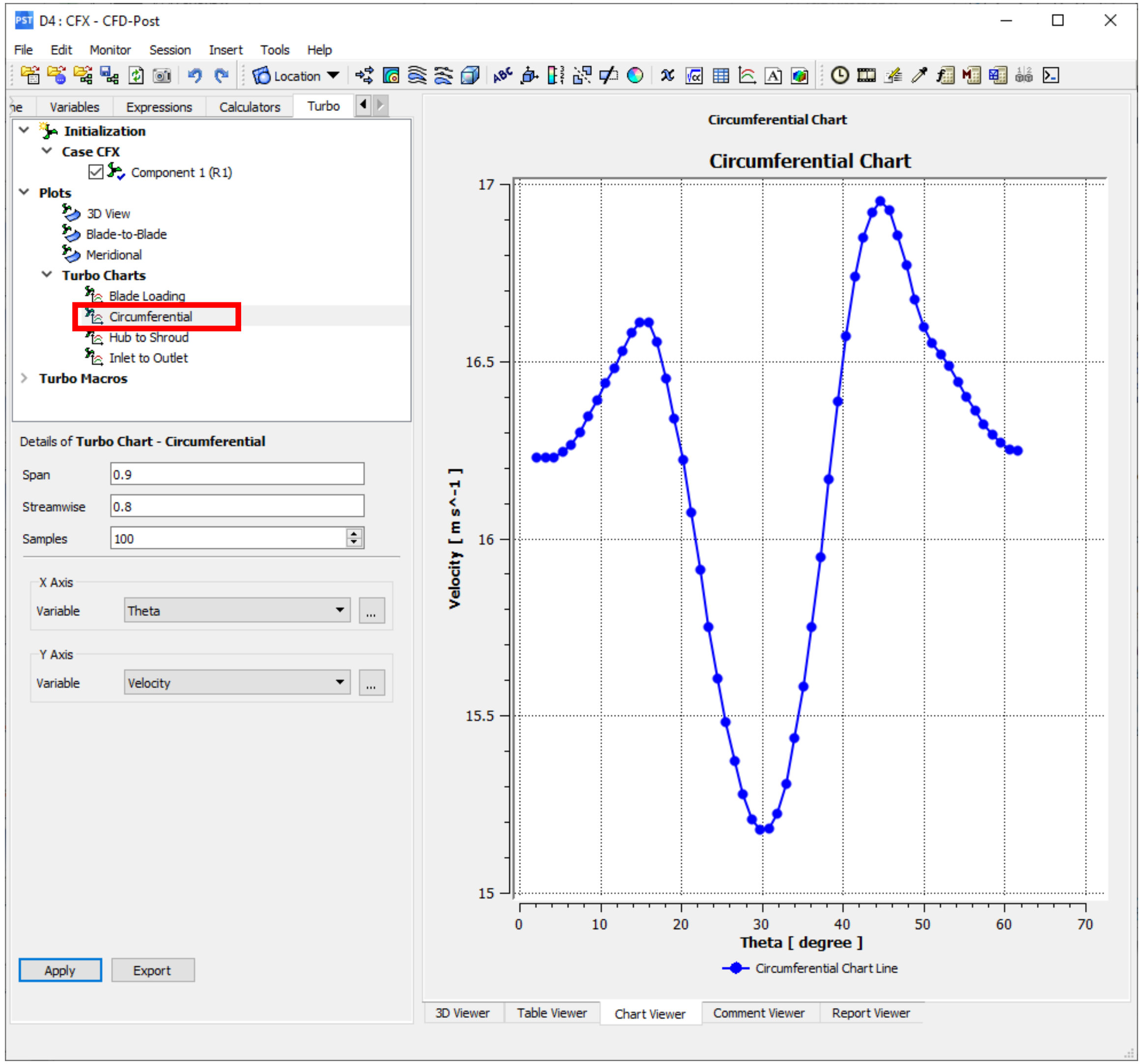
- Create a blade loading chart of pressure at 25% span
  - Click *Blade Loading* under *Plots > Turbo Charts* in the *Tree*
  - Set *Span* to 0.25
  - Set *Y Axis Variable* to *Pressure*
  - Click *Apply*



© 2019 ANSYS, Inc.

Circumferential Chart

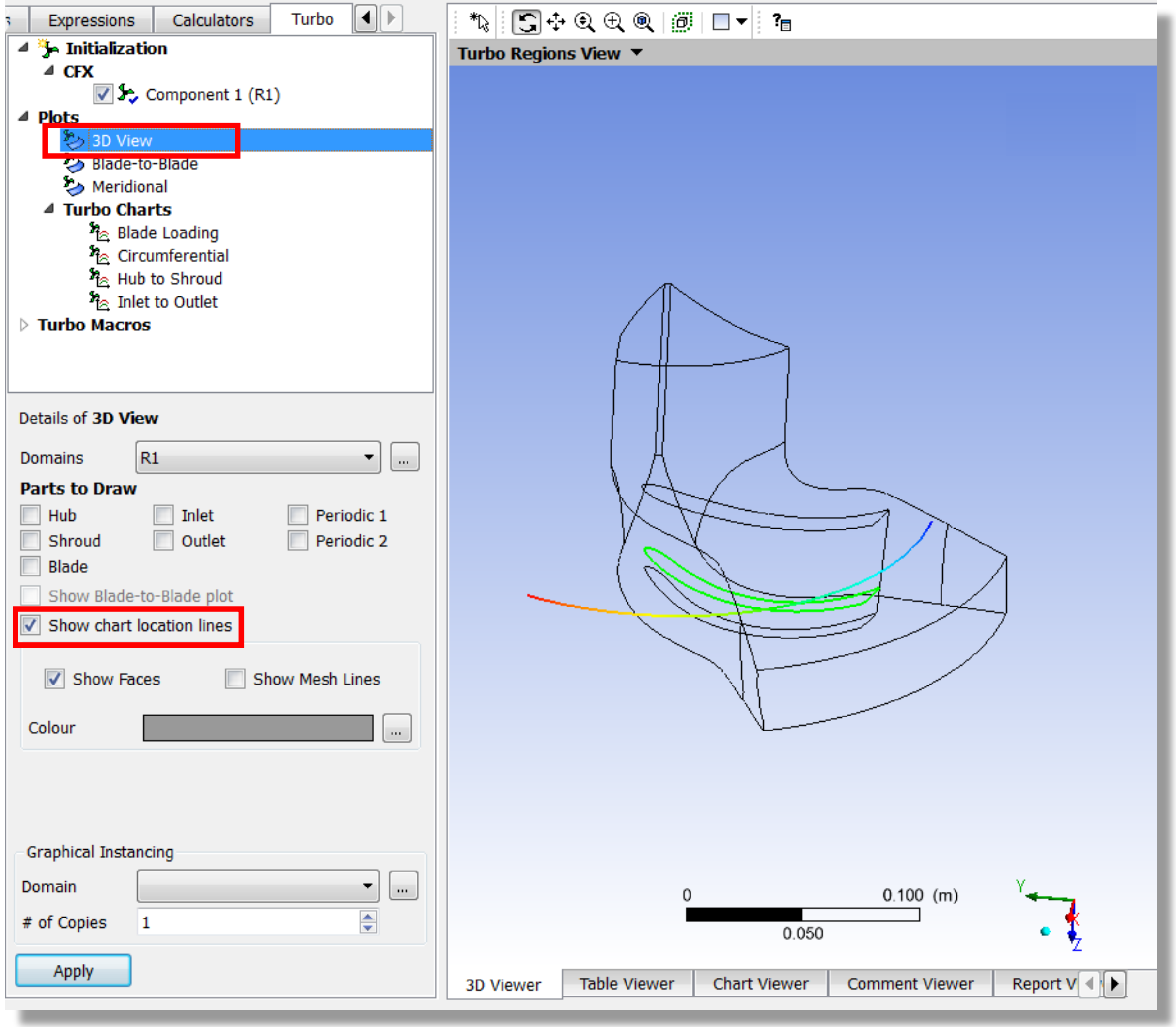
- To examine a velocity profile in the wake of the blade, create a circumferential chart:
  - Select *Circumferential* under *Turbo Charts*
  - Set *Span* to 0.9
  - Set *Streamwise* to 0.8
  - Set *Samples* to 100
  - Set *Y Axis Variable* to *Velocity*
  - Click *Apply*



© 2019 ANSYS, Inc.

Visualizing the location of the chart lines

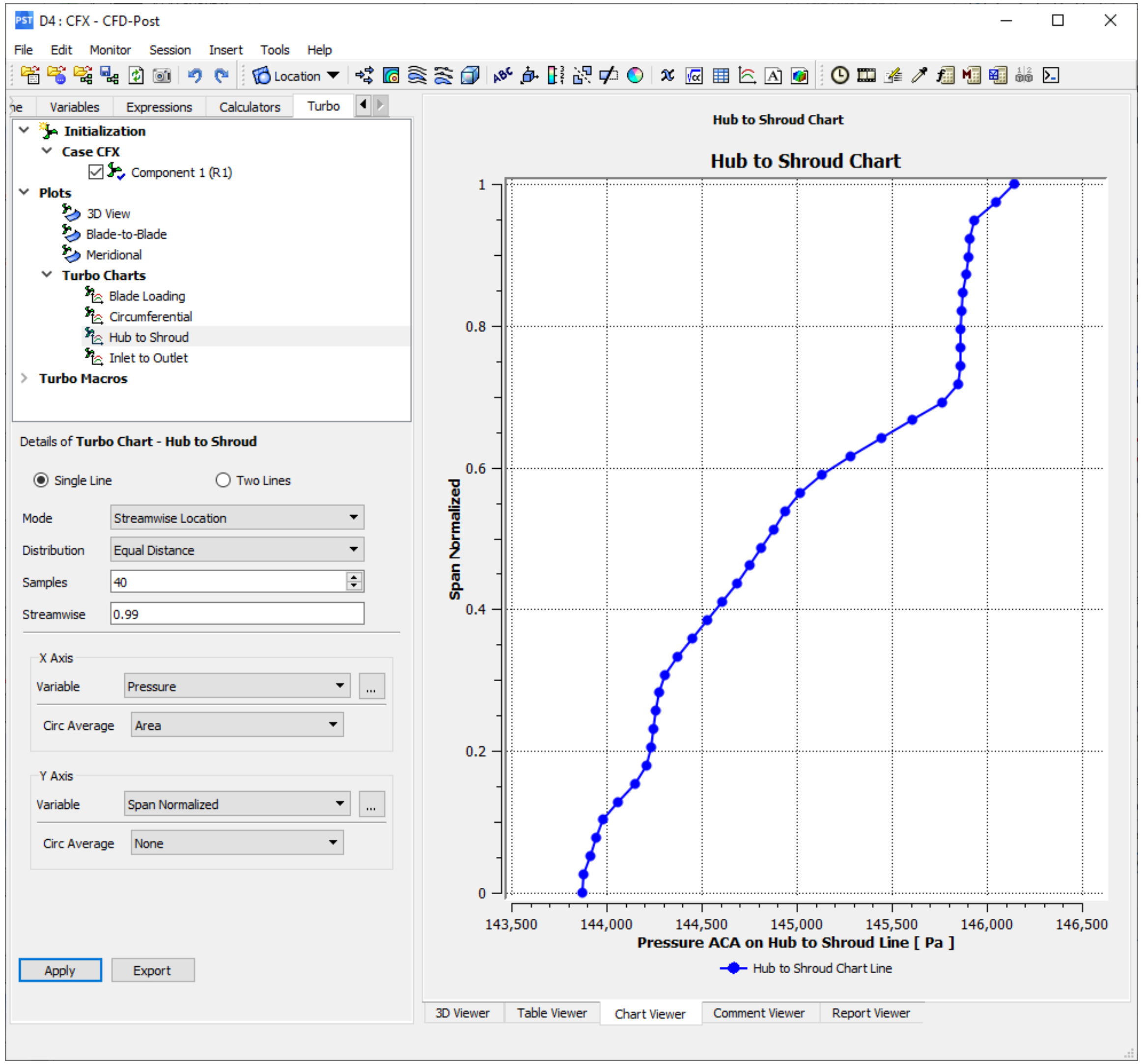
- To visualize where the previous data is being obtained:
  - Select *3D View* under *Plots* in the *Tree*
  - Check the *Show chart location lines* checkbox
  - Click *Apply*
- The lines used to create the blade loading chart at 25% span and the circumferential chart at 90% span at a streamwise location of 0.8 are shown
- This enables you visualize and check where chart lines are created



© 2019 ANSYS, Inc.

Hub to Shroud

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

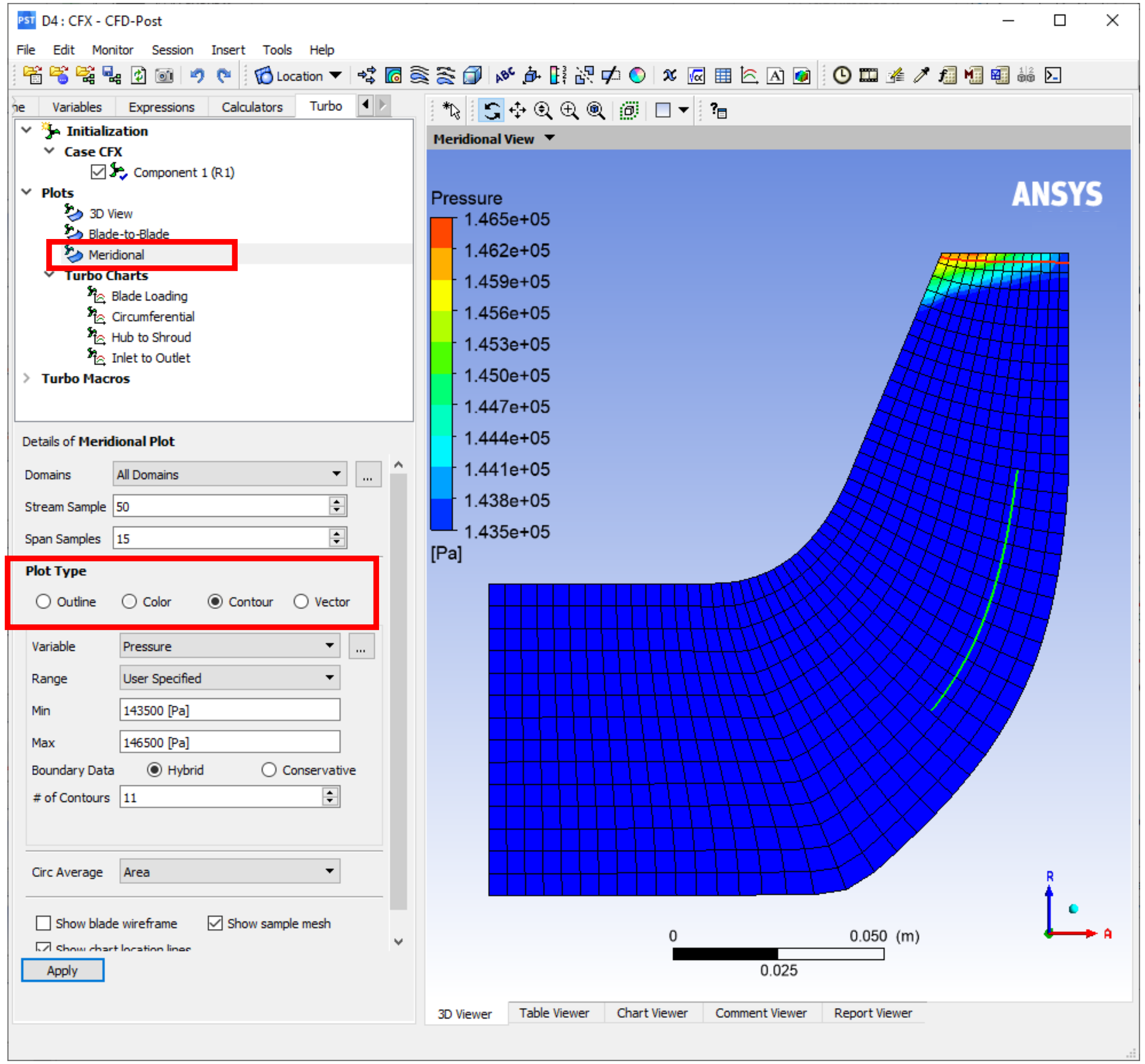


© 2019 ANSYS, Inc.

Compare to Meridional Plot

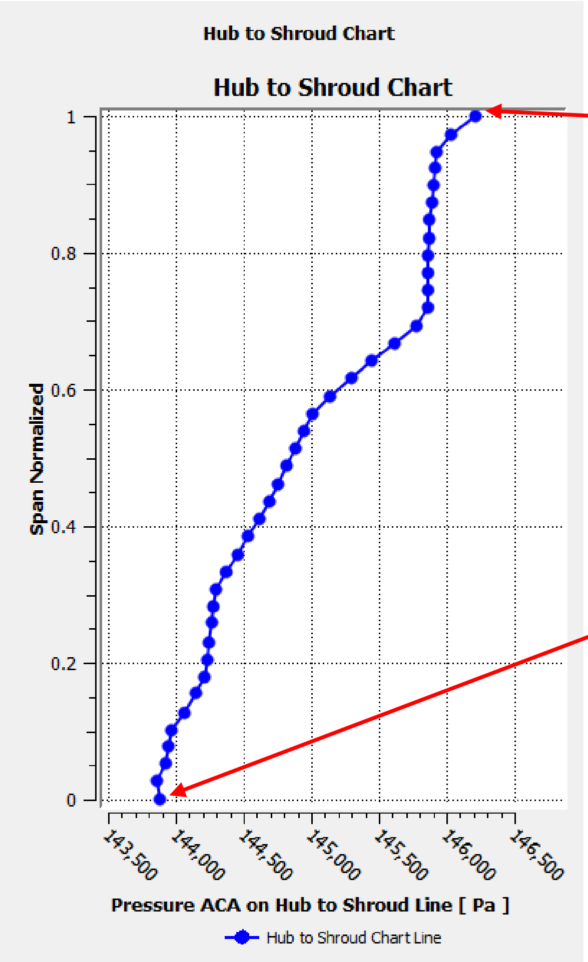
- The profile just created corresponds to data shown in a meridional plot
  - Create a meridional contour as shown to the right
  - Set the *Range* to *User Specified* and set the *Min* and *Max* values to 143500 and 146500 [Pa] respectively*
  - Notice the pressure profile near the outlet

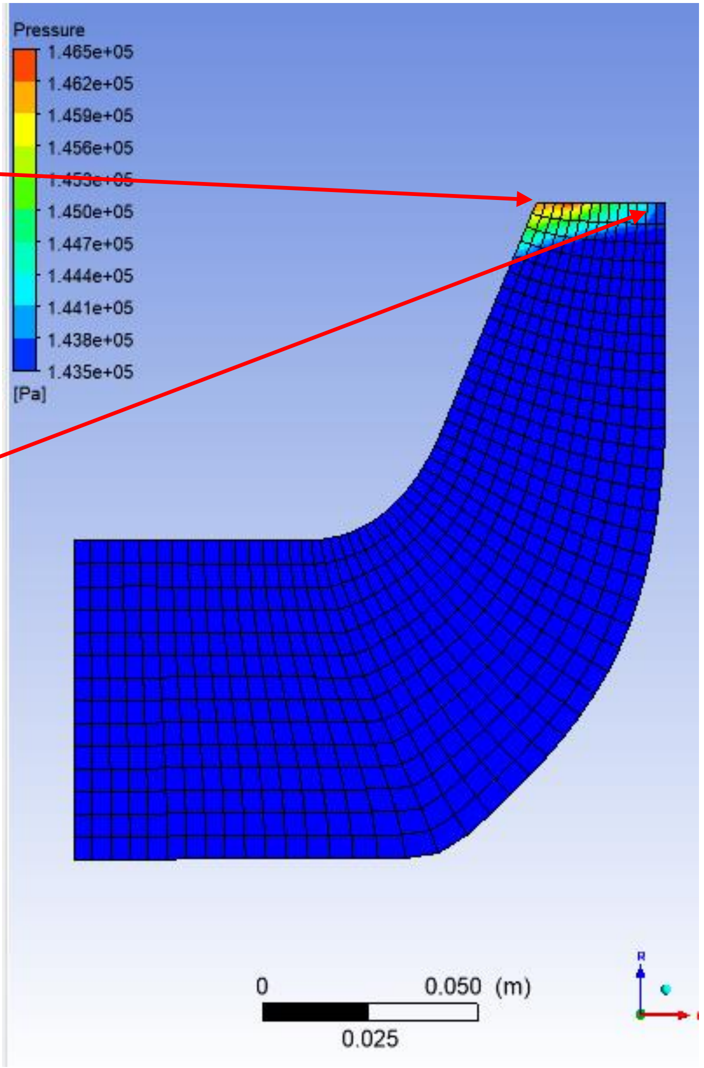
*The *Min* and *Max* values of 143500 and 146500 are taken from the x-axis limits of the chart in the previous slide. You might have to give different values depending on own results



© 2019 ANSYS, Inc.

Compare to Meridional Plot





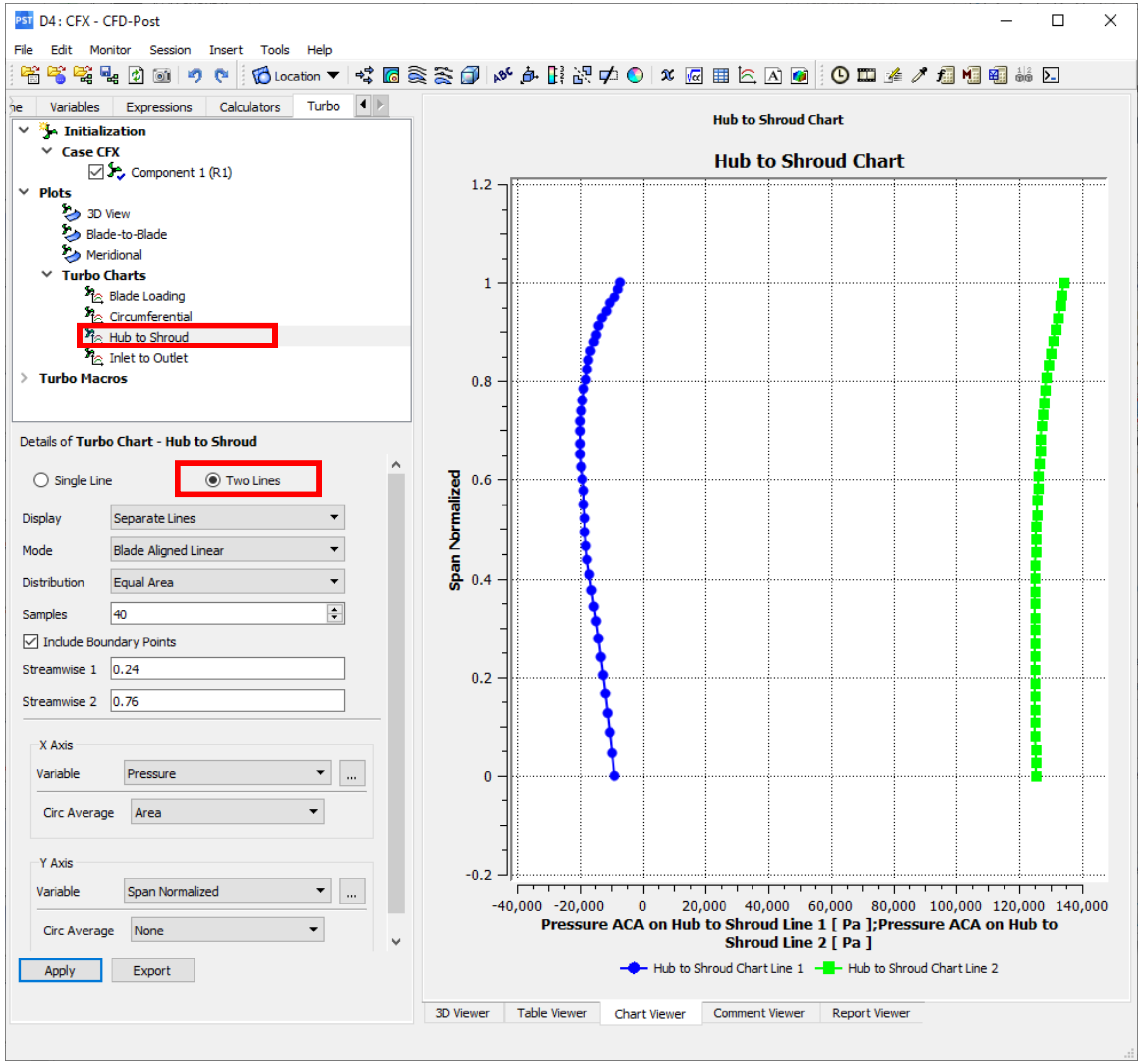
© 2019 ANSYS, Inc.

Hub to Shroud

- Create total pressure profiles at the leading and trailing edge in the same chart
  - Modify *Hub to Shroud Chart*
  - Select *Two Lines*
  - Set *Mode* to *Blade Aligned Linear*
  - Set *Streamwise 1* to 0.24
  - Set *Streamwise 2* to 0.76
  - Click *Apply*

The *Blade Aligned Streamwise* Variable is defined such that it equals the following values at specific locations:

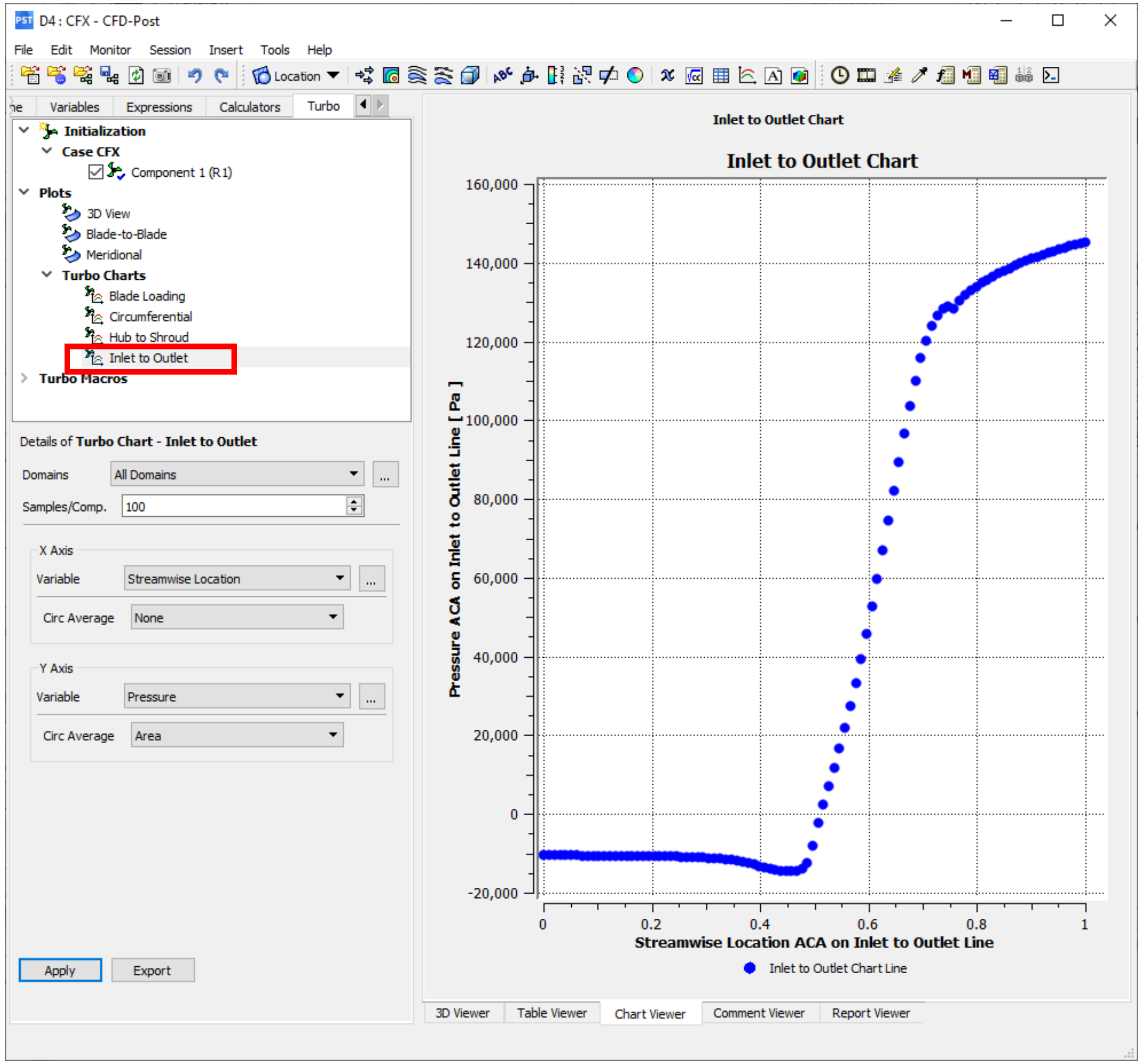
- Inlet to component = 0
- Blade leading edge = 0.25
- Blade trailing edge = 0.75
- Outlet to component = 1



© 2019 ANSYS, Inc.

Inlet to Outlet

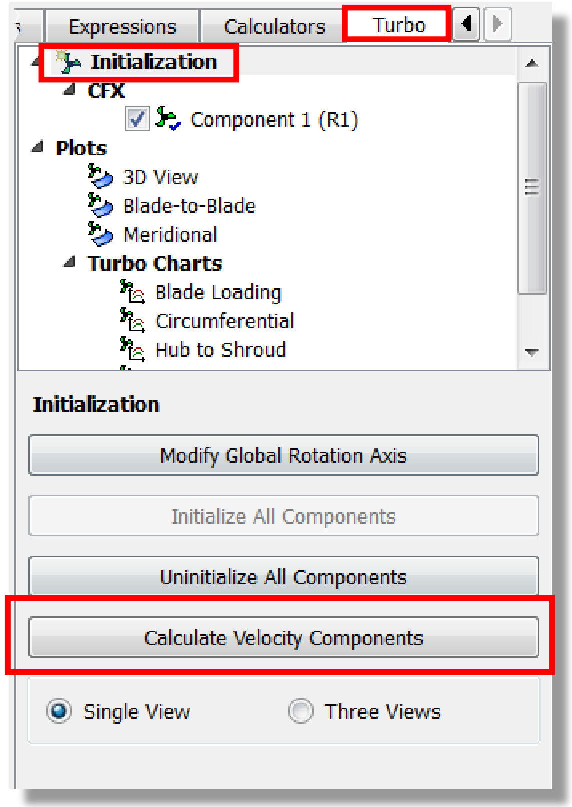
- To visualize how the pressure increases from inlet to outlet:
  - Select *Inlet to Outlet* under *Turbo Chart*
  - Set *Samples/Comp.* to 100
  - Set *X Axis Variable* to *Streamwise Location*
  - Set *Y Axis Variable* to *Pressure*
  - Click *Apply*



© 2019 ANSYS, Inc.

Calculate Velocity Components

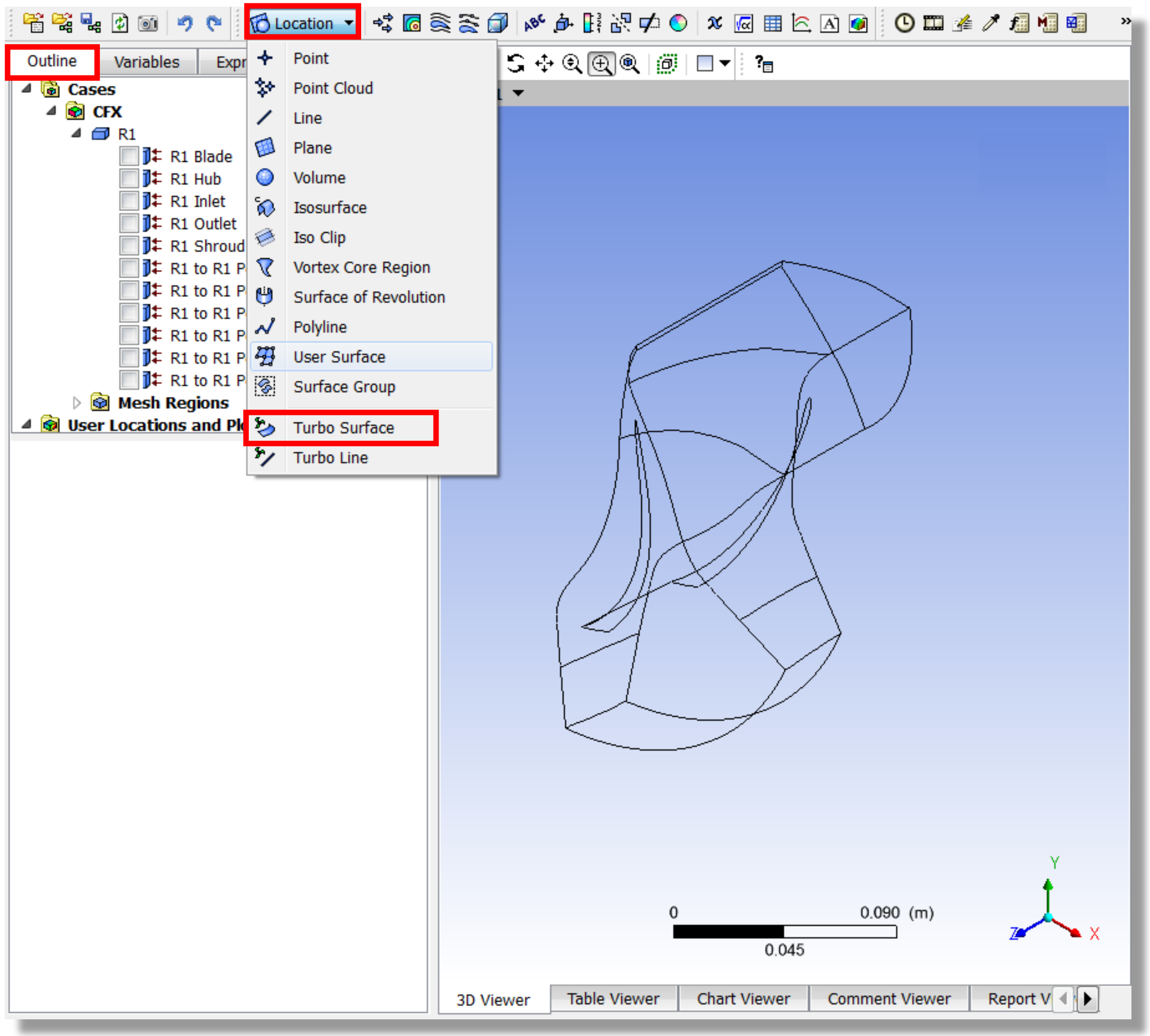
- Some additional variables useful for turbo post processing can be created
  - Still in the *Turbo* Tab
  - Double click *Initialization*
  - Click *Calculate Velocity Components*
- This creates a number of new velocity variables, such as:
  - *Velocity Axial*
  - *Velocity Radial*
  - *Velocity Circumferential*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



© 2019 ANSYS, Inc.

Turbo Surface

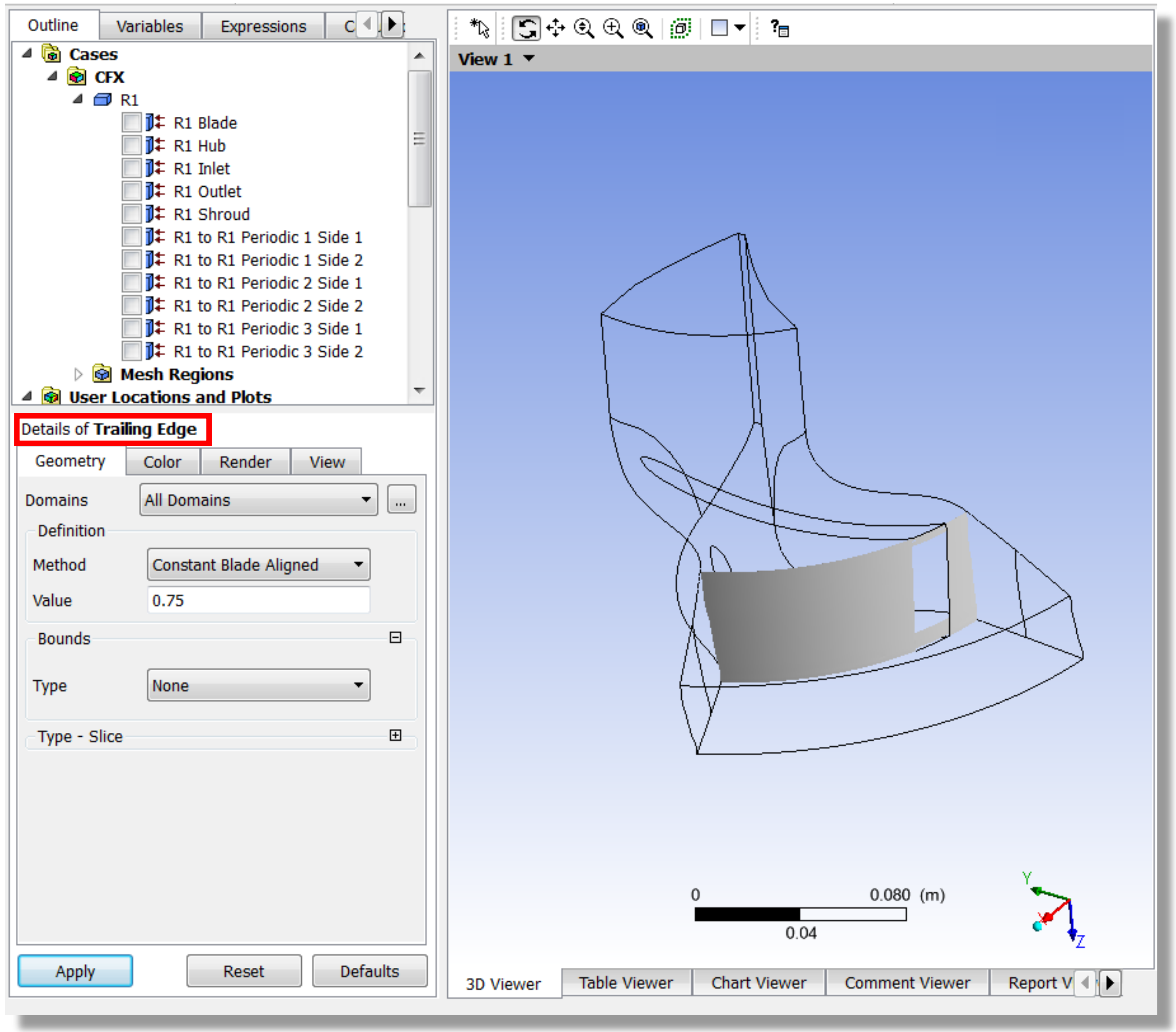
- You can create Turbo Surfaces in the standard 3D Viewer
  - Switch to the *Outline* Tab at the top left
  - *Location > Turbo Surface*
  - Rename the *Turbo Surface* to *Trailing Edge*



© 2019 ANSYS, Inc.

Turbo Surface

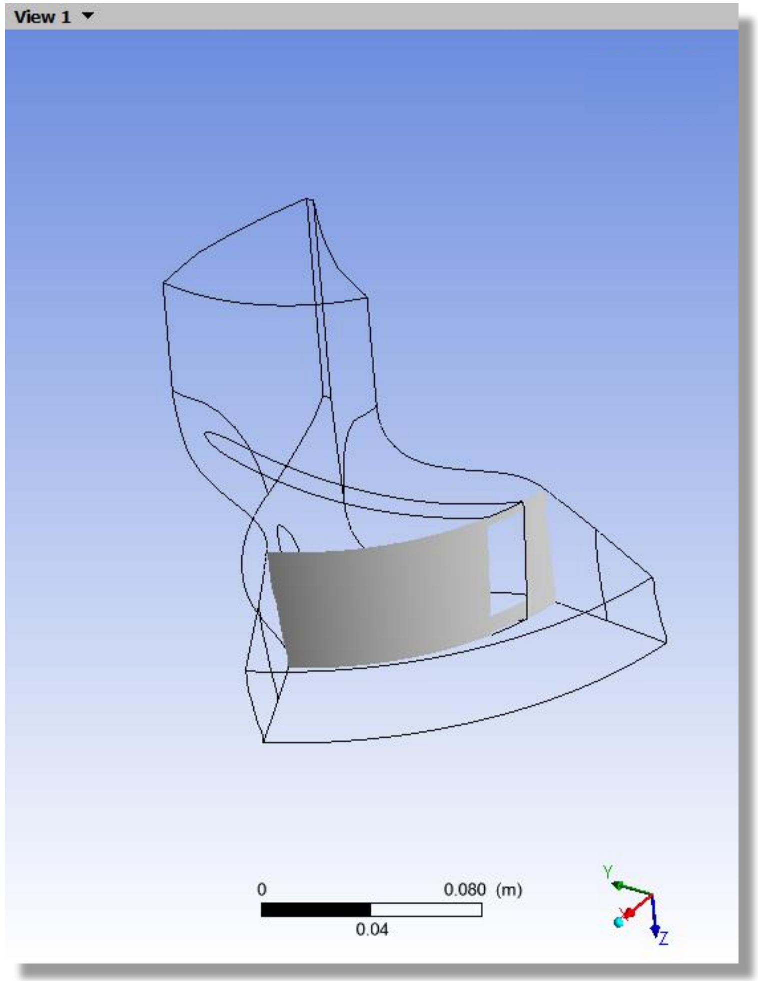
- Details of *Trailing Edge Turbo Surface*
  - Set *Method* to *Constant Blade Aligned*
  - Set *Value* to 0.75
- We can see this surface intersects the trailing edge of the blade
- Move the plane slightly downstream so that it is entirely behind the trailing edge
  - Set *Value* to 0.755



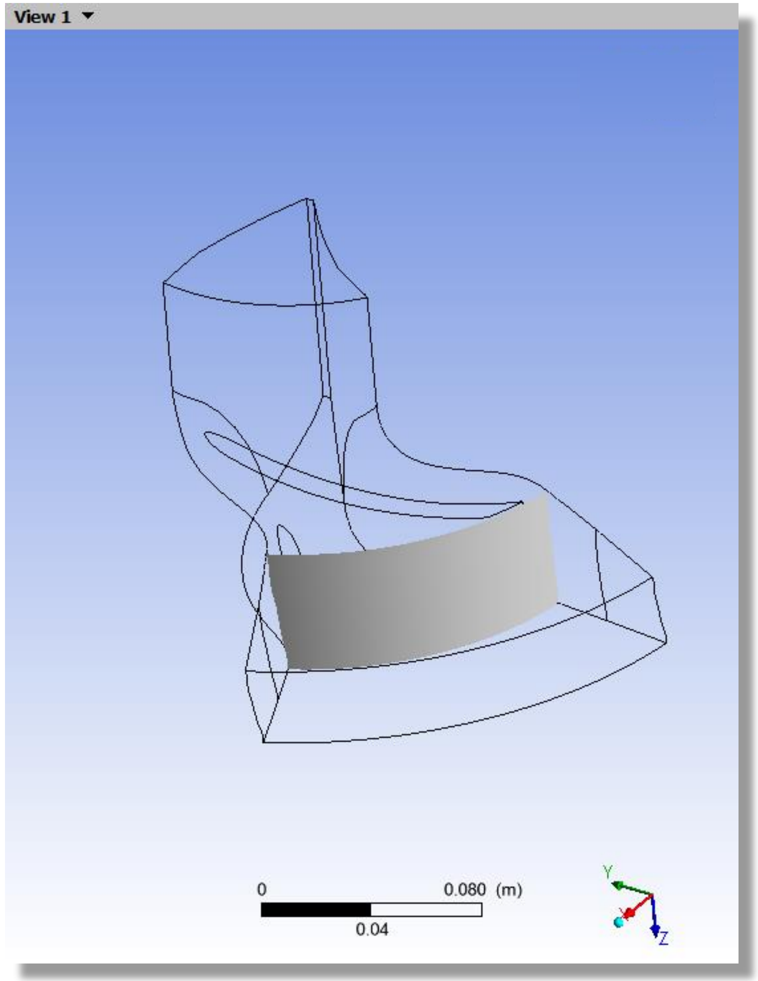
© 2019 ANSYS, Inc.

Constant Blade Aligned

Constant Blade Aligned = 0.75



Constant Blade Aligned = 0.755

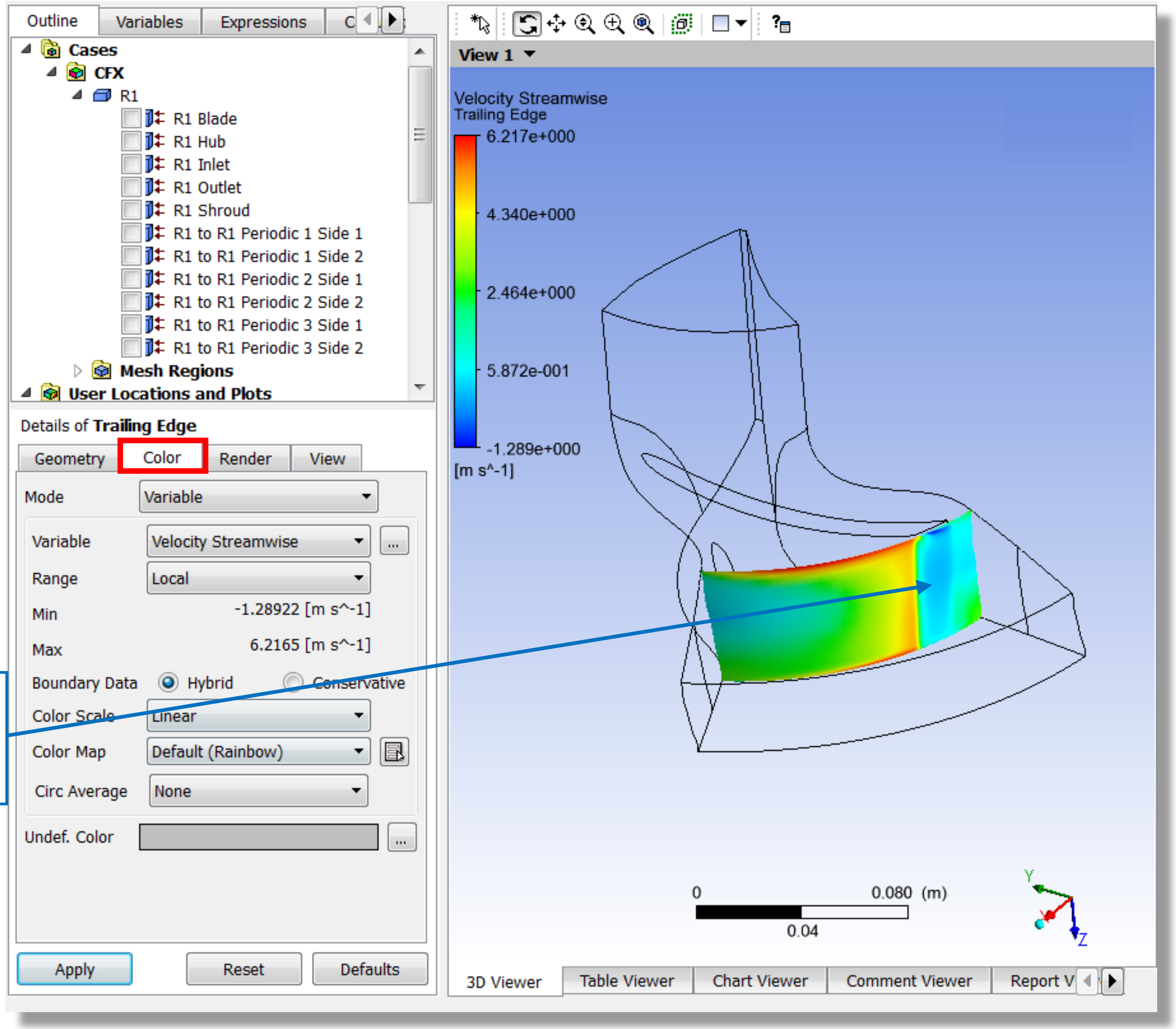


© 2019 ANSYS, Inc.

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*

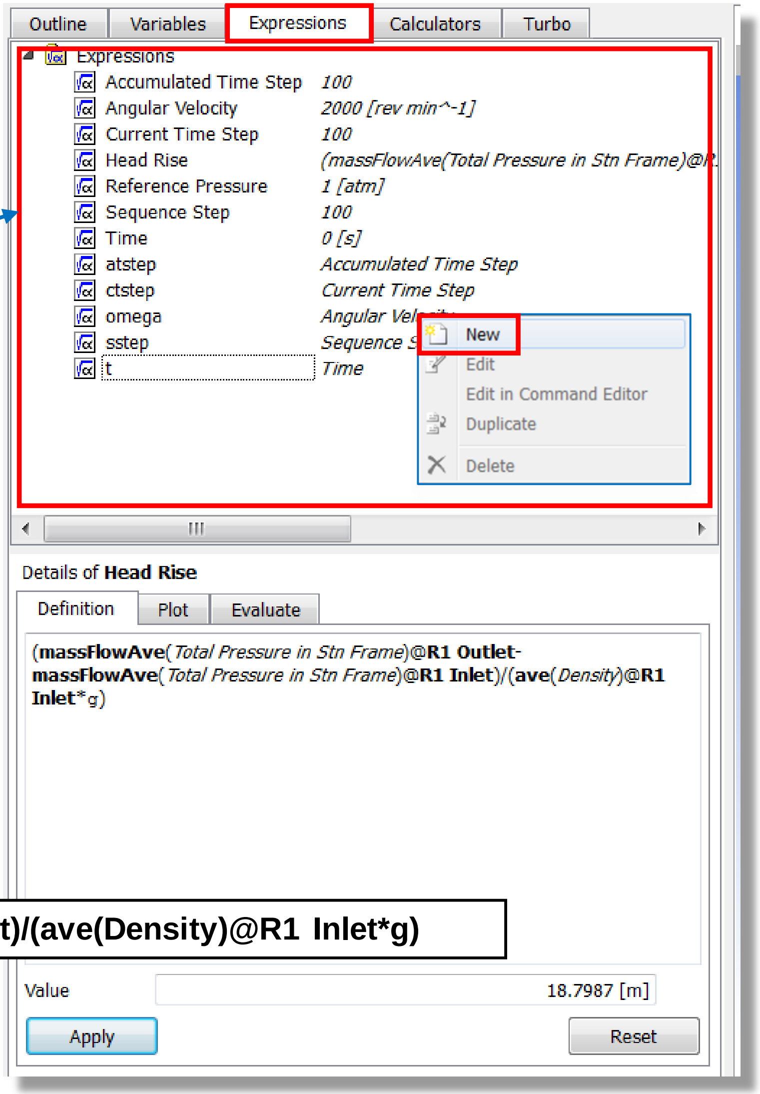
Note the low velocities in the wake of the blade



© 2019 ANSYS, Inc.

Quantitative Calculations

- Create an expression to define the head generated from inlet to outlet
  - Click on the *Expressions* Tab
  - Right click inside the *Expressions* window and select *New*
  - Name the Expression *Head Rise*
  - Type in the expression given in the box below
  - Click *Apply*
- The Head is 18.8 [m] for this pump



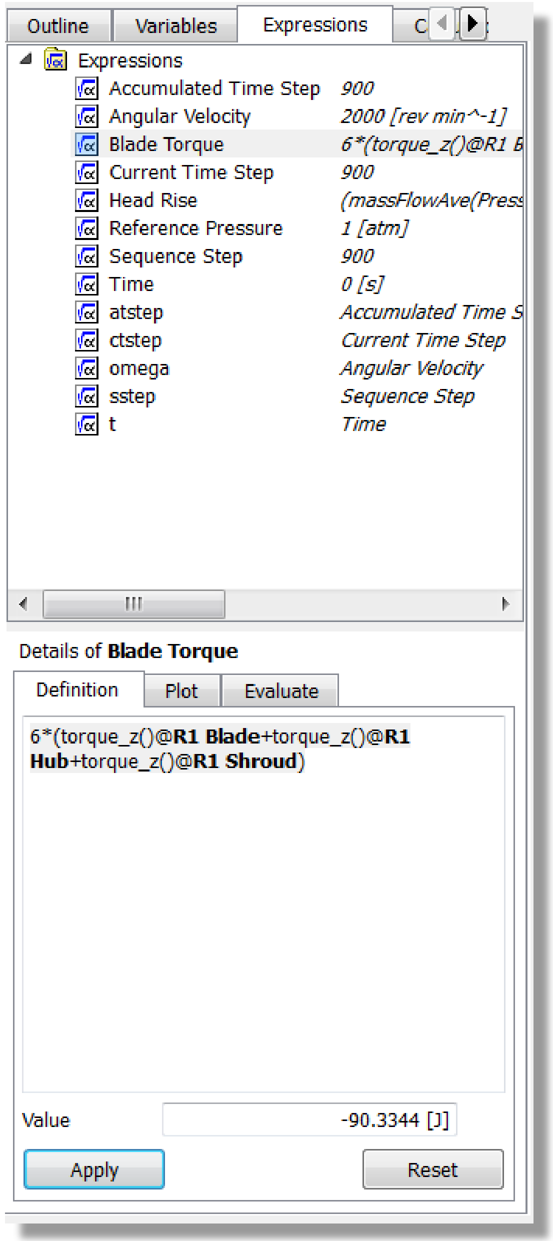
(massFlowAve(Total Pressure in Stn Frame)@R1 Outlet-massFlowAve(Total Pressure in Stn Frame)@R1 Inlet)/(ave(Density)@R1 Inlet*g)

© 2019 ANSYS, Inc.

Calculate Blade Torque

- To calculate blade torque
  - Create a new expression
  - Name *Blade Torque*
  - Enter expression given in the box below
  - Click *Apply*
- Torque is 90.33 [J] [i.e. N*m])
- Note that we multiplied the expression by 6 which is the number of blades
- We are also accounting for torque due to shear on hub and shroud

6*(torque_z()@R1 Blade+torque_z()@R1 Hub+torque_z()@R1 Shroud)



© 2019 ANSYS, Inc.

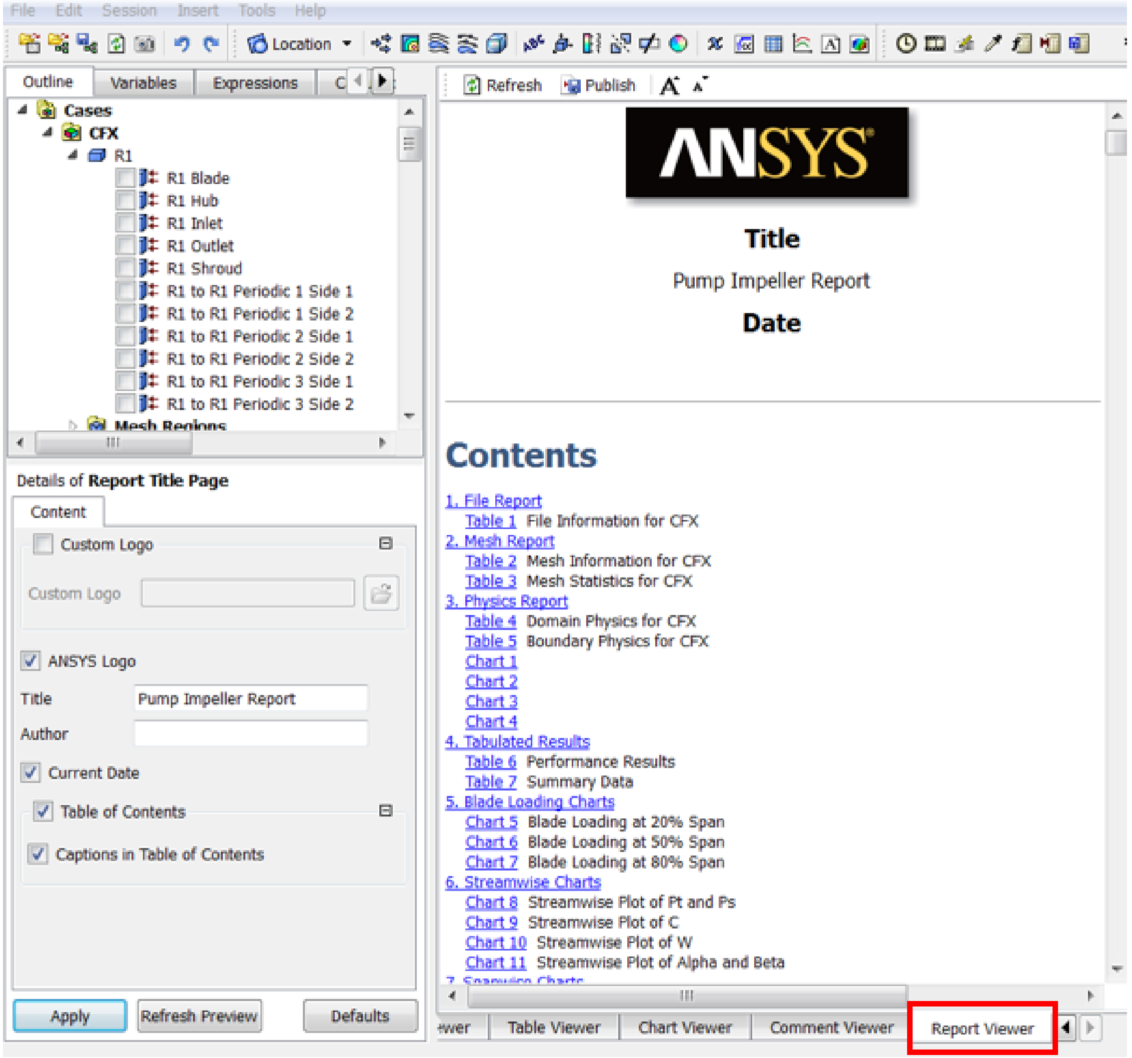
Quantitative Calculations - Expressions

- You could create many expressions to evaluate quantities of interest
  - *Head*
  - *Input Power*
  - *Mass or Volume Flow rate*
  - *Efficiency*
  - *Torque*
  - etc.
- However, there are built in reports which can do this for you!

© 2019 ANSYS, Inc.

Load Pump Report

- From the Main Menu, *File > Report > Load 'Pump Impeller Report' Template*
  - It will take a few minutes to generate the plots and quantitative data for the report
- Once complete, click on the Report Viewer tab
  - It will take a minute to generate the report
- Once complete, scroll through and examine the report



© 2019 ANSYS, Inc.

Examine Pump Report

- Go to section *4. Tabulated Results* of the report
  - Table 6. Performance Results
    - Total Efficiency of 80.2%
  - Table 7. Summary Data
    - Flow Angle: Alpha represents the Absolute flow angle (i.e. to compare to design velocity triangles)
    - Values are calculated at inlet, outlet, leading and trailing edge
    - Ratios and differences from TE-LE are calculated

4. Tabulated Results

The first table below gives a summary of the performance results for the pump impeller. The second table lists the mass or area averaged solution variables and derived quantities computed at the inlet, leading edge (LE Cut), trailing edge (TE Cut) and outlet locations. The flow angles Alpha and Beta are relative to the meridional plane; a positive angle implies that the tangential velocity is the same direction as the machine rotation.

Table 6. Performance Results

Rotation Speed	209.4400	[radian s^-1]
Reference Diameter	0.2081	[m]
Volume Flow Rate	0.0840	[m^3 s^-1]
Head (LE-TE)	18.7137	[m]
Head (IN-OUT)	18.7987	[m]
Flow Coefficient	0.0445	
Head Coefficient (IN-OUT)	0.0971	
Shaft Power	19252.1000	[W]
Power Coefficient	0.0054	
Total Efficiency (IN-OUT) %	80.1957	
Static Efficiency (IN-OUT) %	63.4938	

Table 7. Summary Data

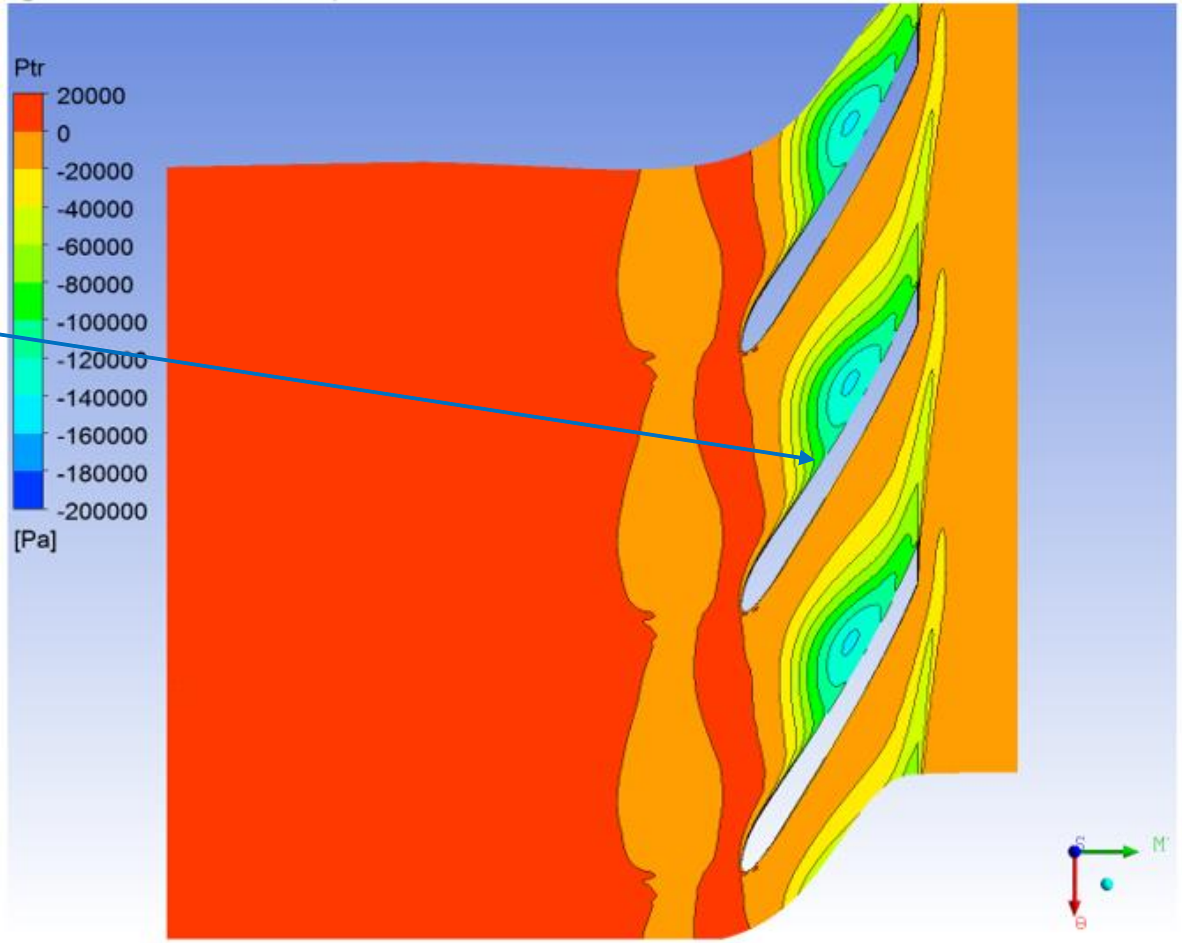
Quantity	Inlet	LE Cut	TE Cut	Outlet	TE/LE	TE-LE	Units
Density	997.0000	997.0000	997.0000	997.0000	1.0000	0.0000	[kg m^-3]
Pstatic	90822.3000	83304.8000	229592.0000	246604.0000	2.7561	146287.0000	[Pa]
Ptotal	101225.0000	101174.0000	284143.0000	285024.0000	2.8085	182968.0000	[Pa]
Ptotal (rot)	100514.0000	96139.4000	76207.5000	55710.0000	0.7927	-19931.9000	[Pa]
U	10.7251	12.4750	21.7896	32.4612	1.7467	9.3146	[m s^-1]
Cm	4.5618	5.4465	3.3458	3.0825	0.6143	-2.1007	[m s^-1]
Cu	0.0362	1.8640	11.0474	6.2849	5.9268	9.1834	[m s^-1]
C	4.5678	6.7835	11.7722	7.0842	1.7354	4.9887	[m s^-1]
Distortion Parameter	1.0007	1.1947	1.3000	1.3910	1.0881	0.1053	
Flow Angle: Alpha	0.4801	25.3026	72.7069	66.3892	2.8735	47.4043	[degree]
Wu	-10.6889	-10.6113	-10.7423	-26.1764	1.0123	-0.1309	[m s^-1]
W	11.7669	12.0128	11.2978	26.4387	0.9405	-0.7150	[m s^-1]
Flow Angle: Beta	-63.8893	-46.0883	-73.1813	-83.2342	1.5878	-27.0930	[degree]

© 2019 ANSYS, Inc.

Examine Pump Report

- Examine:
  - Figure 5. Contour of Ptr at 50% span
    - ➢ There is significant flow separation from the suction side of the blade

Figure 5. Contour of Ptr at 50% Span

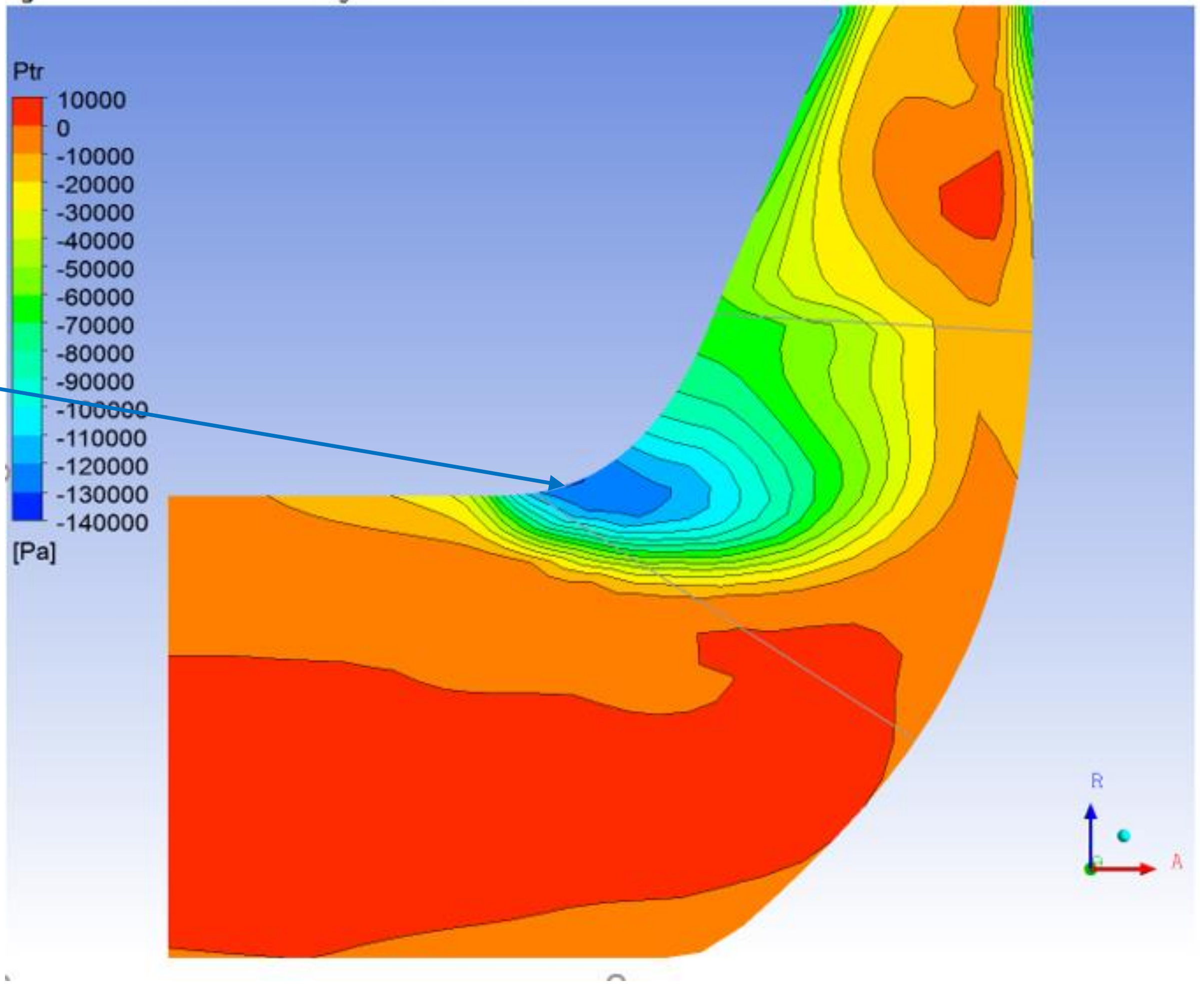


© 2019 ANSYS, Inc.

Examine Pump Report

- Examine:
  - Figure 12. Contour of Mass Averaged Ptr on Meridional Surface
    - ➢ There is significant flow separation from shroud
- This plot and the one on the previous page indicate that the shroud is not designed correctly, hence the low pump efficiency and poor convergence
- The report can provide insights on how to improve your design

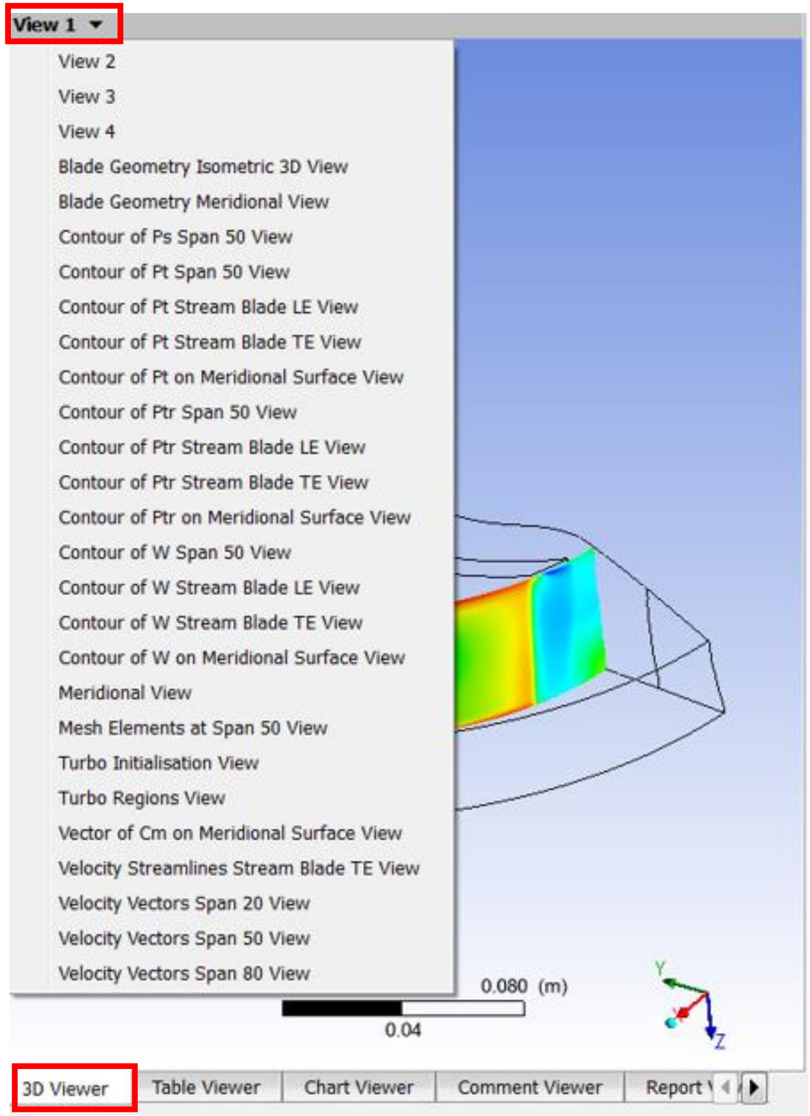
Figure 12. Contour of Mass Averaged Ptr on Meridional Surface



© 2019 ANSYS, Inc.

Add a plot to the Pump Report

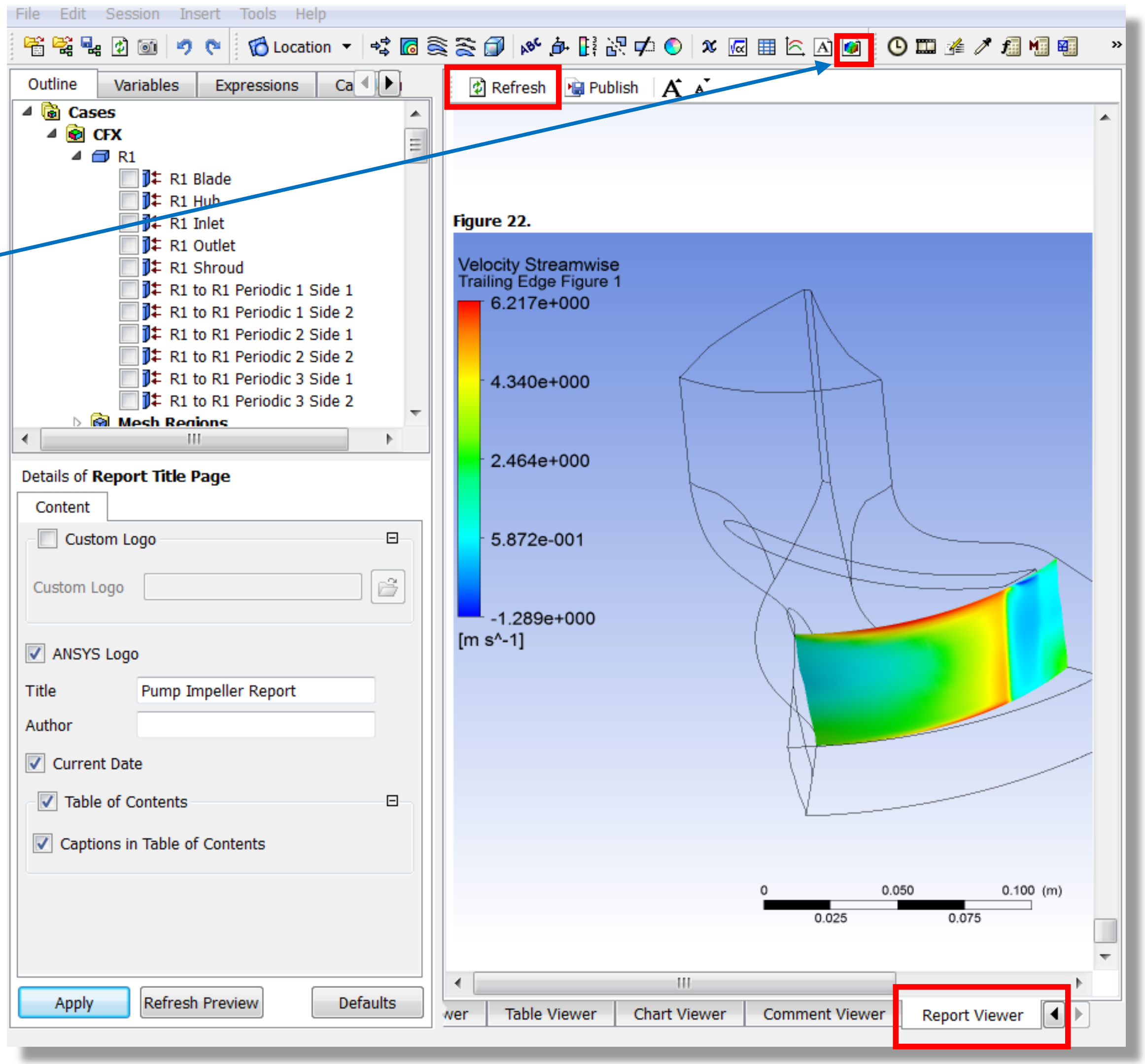
- Click on *3D Viewer* Tab at the bottom
- Switch to *View 1* at the top
- Add the colored turbo surface to the report (see next slide)



© 2019 ANSYS, Inc.

Add a plot to the Pump Report

- *Insert > Figure*
  - Alternatively select Figure from the toolbar
- *Name: Figure 1*
  - Click *OK*
- Click on the *Report Viewer* Tab
- Click *Refresh*
- Scroll to bottom of the window once the report is regenerated
- The new figure has been added



© 2019 ANSYS, Inc.

Summary

- This workshop demonstrated the following:
- Turbo Post Processing using the Turbo Tab
- Creating CEL expressions to evaluate global quantities for the pump
- Report Generation and customization

© 2019 ANSYS, Inc.