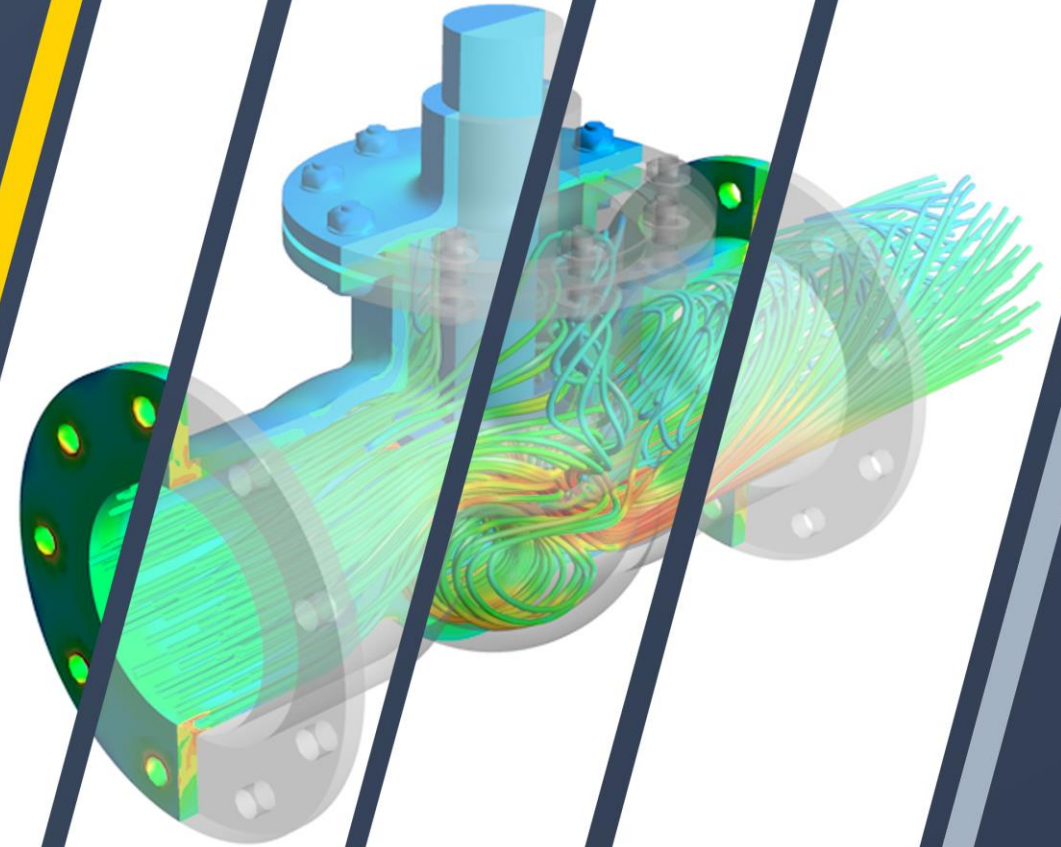




Workshop 02.1: Pump Simulation

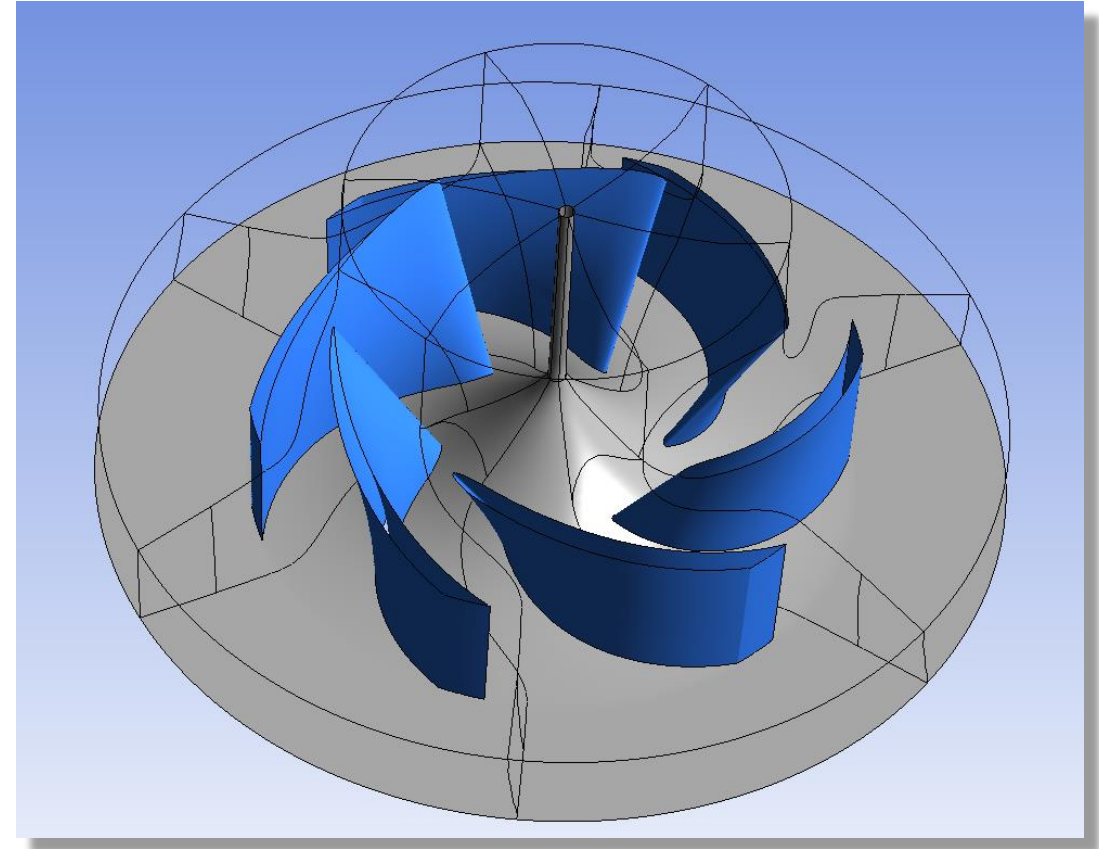
ANSYS CFX Rotating Machinery
Modeling

Release 2019 R3



Introduction

- Workshop Description:
 - This Workshop deals with the CFX setup and solution for a pump impeller
- Learning Aims:
 - Setting up a single frame of reference solution
 - Creating CEL expressions to monitor during the solution
 - Solving the case and convergence



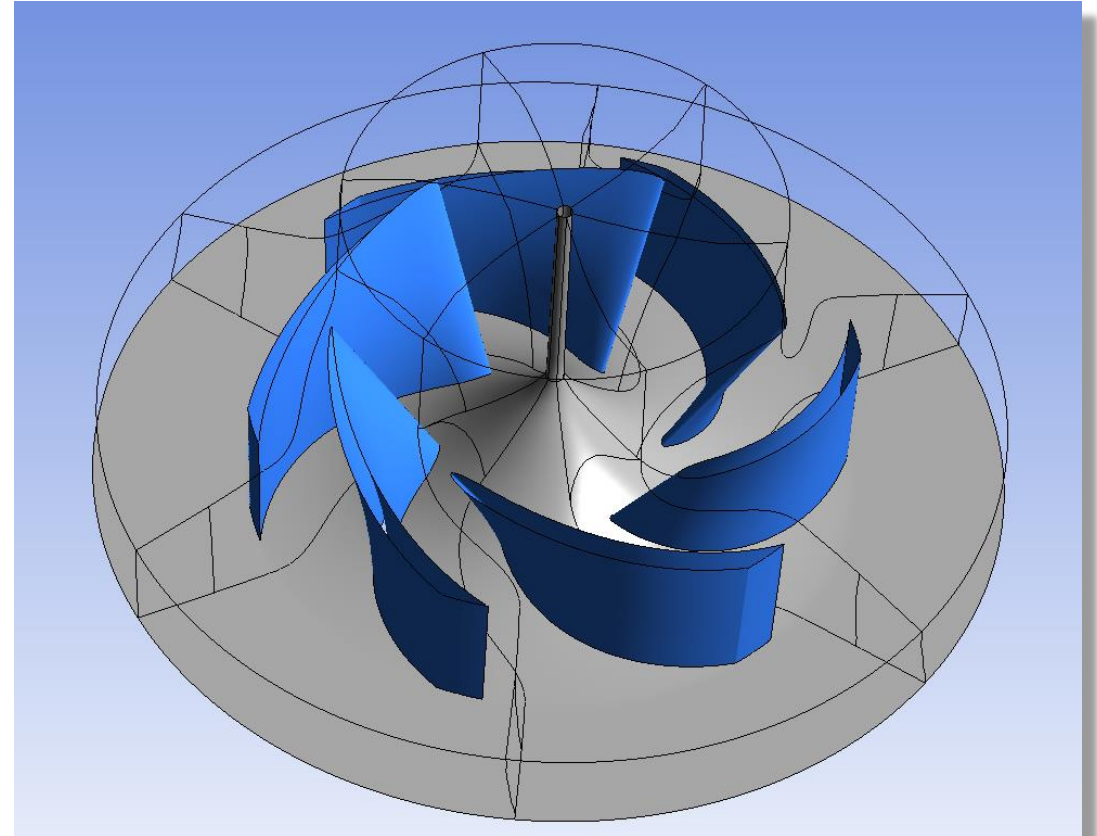
Pump Model

- Single Frame of Reference
 - As discussed in the lecture, single frames of reference are used to solve rotating components
 - Since each blade is the same, we can reduce the problem size by modelling a single blade passage with periodic boundaries
 - CFX Pre has a wizard to facilitate the setup of these single passage cases

Pump Speed = 2000 rpm

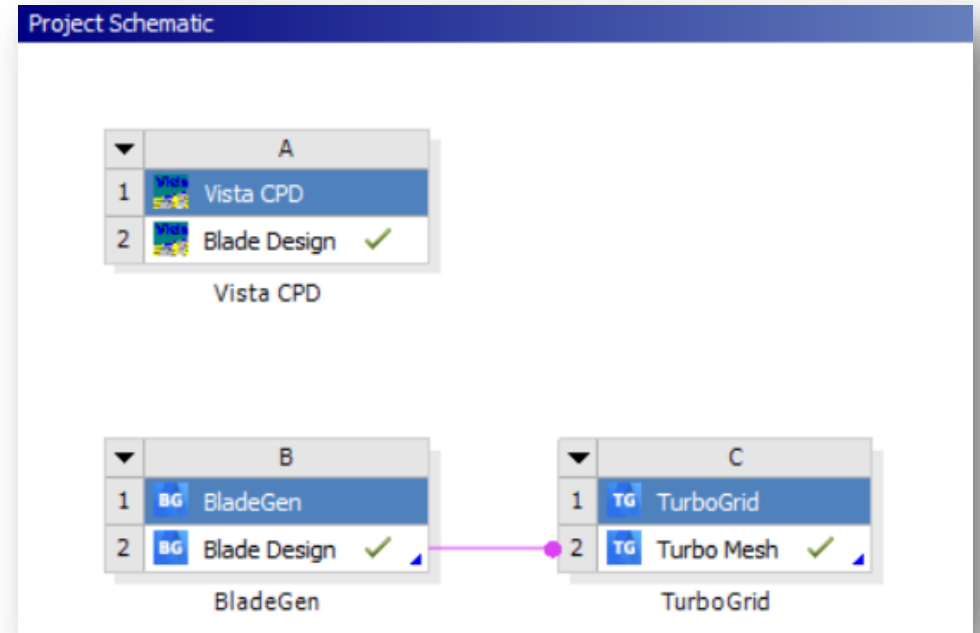
Pump Flow Rate = 83.76 kg/s

Fluid = Water



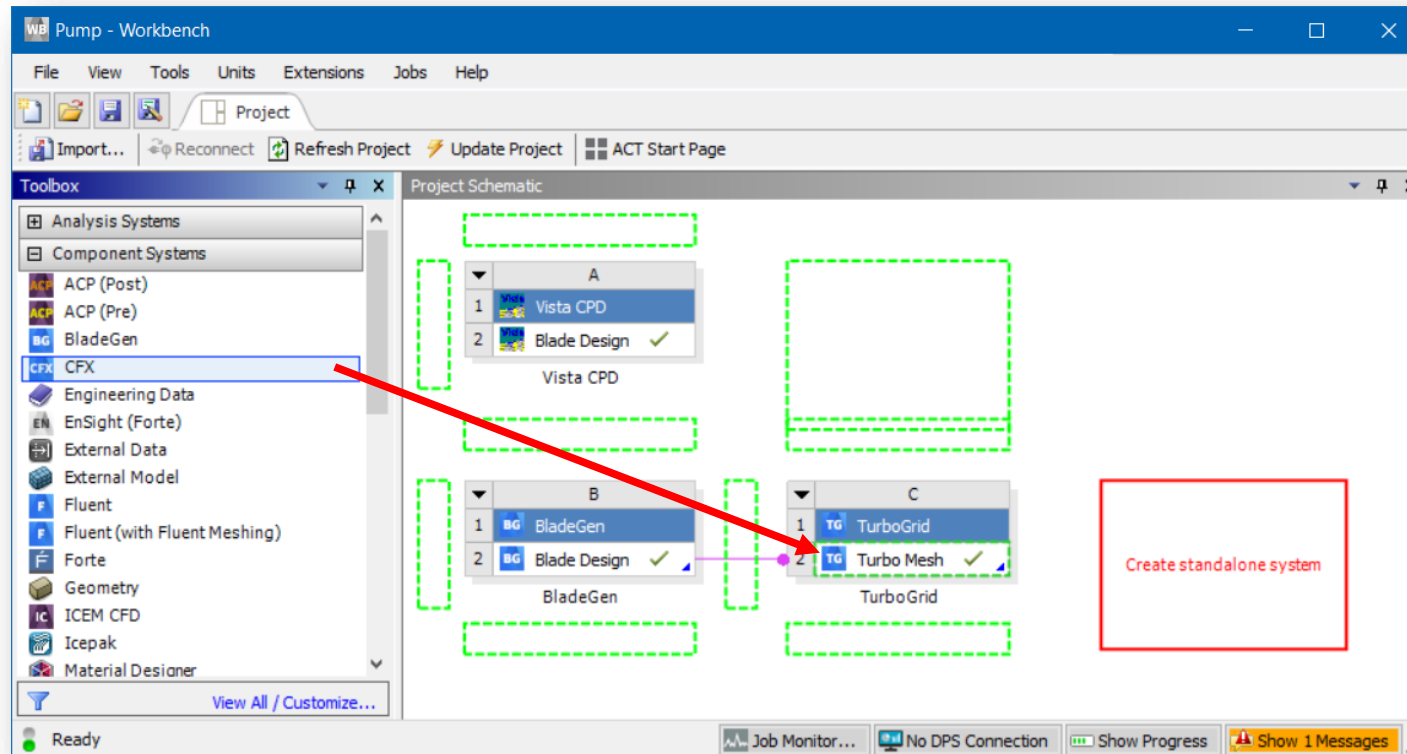
Load Workbench Project

- The geometry has been created using Vista CPD and the mesh has been created using TurboGrid
- Load the geometry and mesh for the pump to analyze
- Open Workbench:
 - *Start > ANSYS 2019 R3 > Workbench 2019 R3*
 - In the Workbench main menu *File > Open...*
 - In the *Open* dialogue box *Browse to Pump_mesh.wbpz* provided with the workshop inputs and click *Open*
 - In the *Save As* dialogue box edit the *File Name* to *Pump.wbpj* and click *Save*



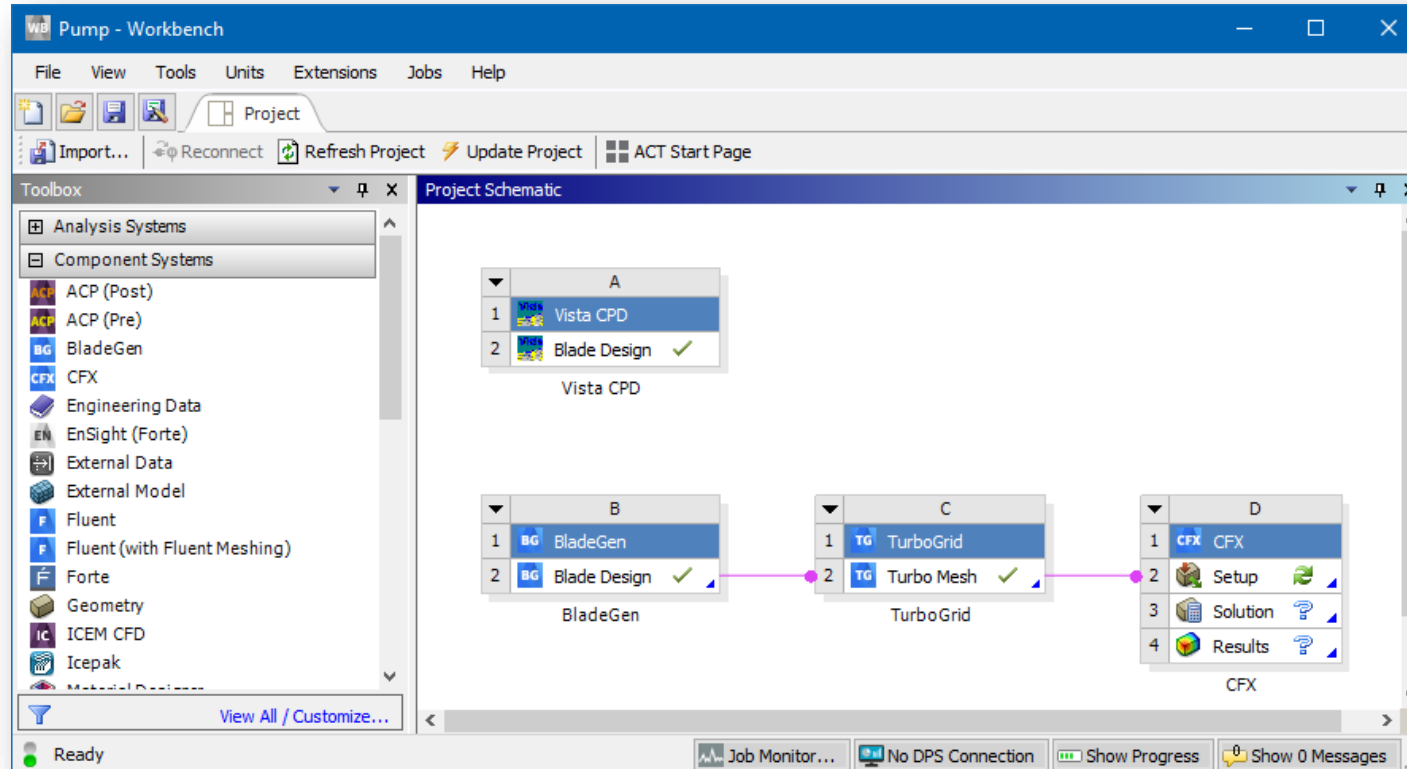
Create a CFX Component

- Add a *CFX Component* to *Project Schematic*
 - Find *CFX* under *Component Systems*
 - Drag *CFX* and drop it to cell *C2* to transfer the *TurboGrid* mesh automatically to *CFX*



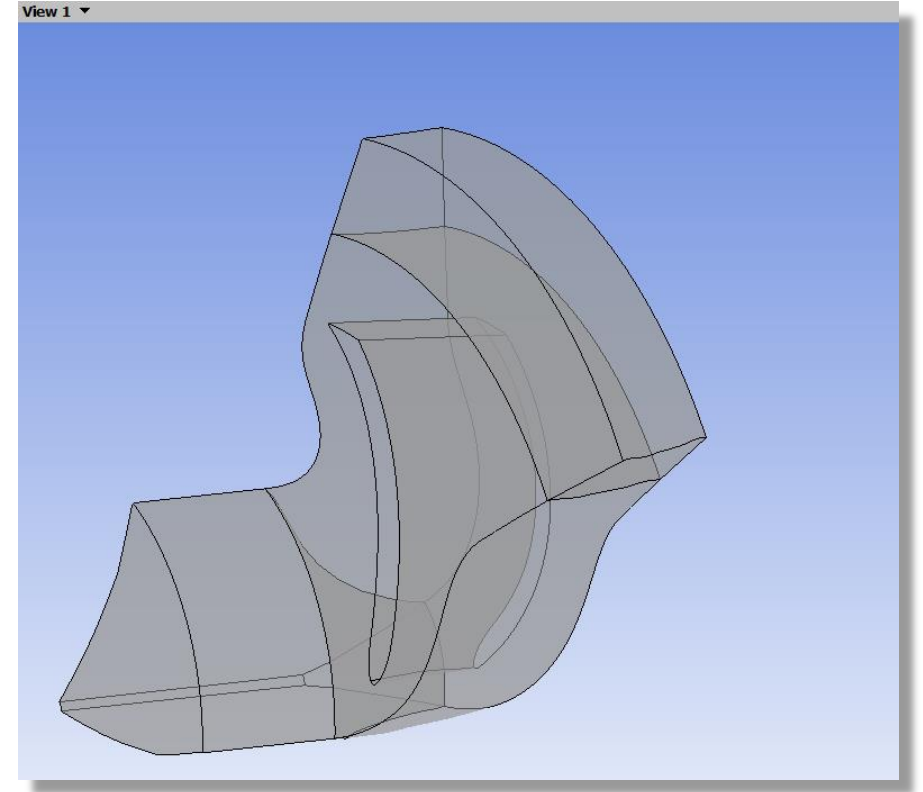
Launch CFX Pre

- Double click on the *Setup* cell *D2* to launch *CFX-Pre*



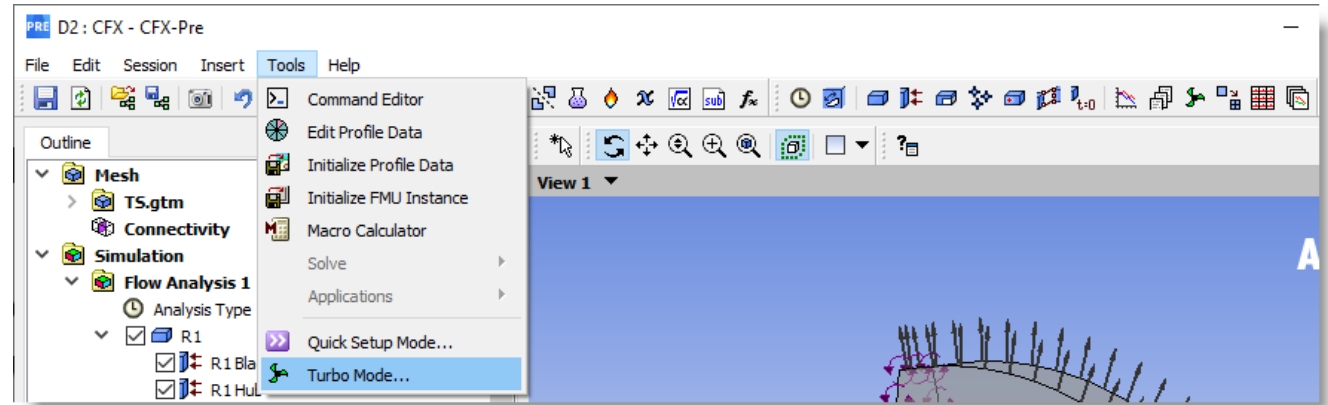
CFX-Pre

- Once *CFX-Pre* opens, you should see the single passage geometry of the pump as shown on the right



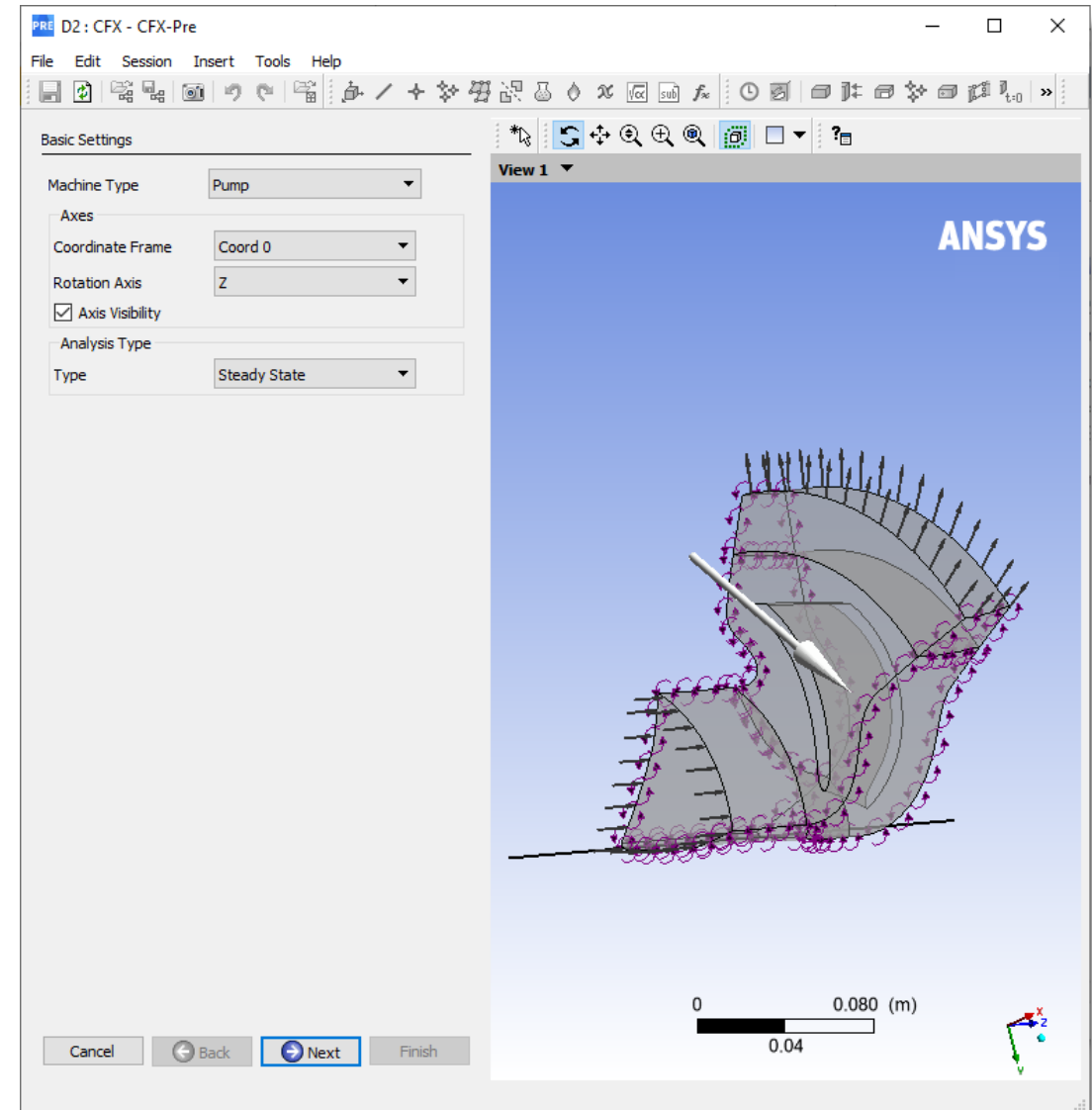
Turbo Mode

- *CFX-Pre* has a *Turbo Mode* which makes it easy to setup turbo cases
- To enable *Turbo Mode*:
 - *Tools > Turbo Mode*



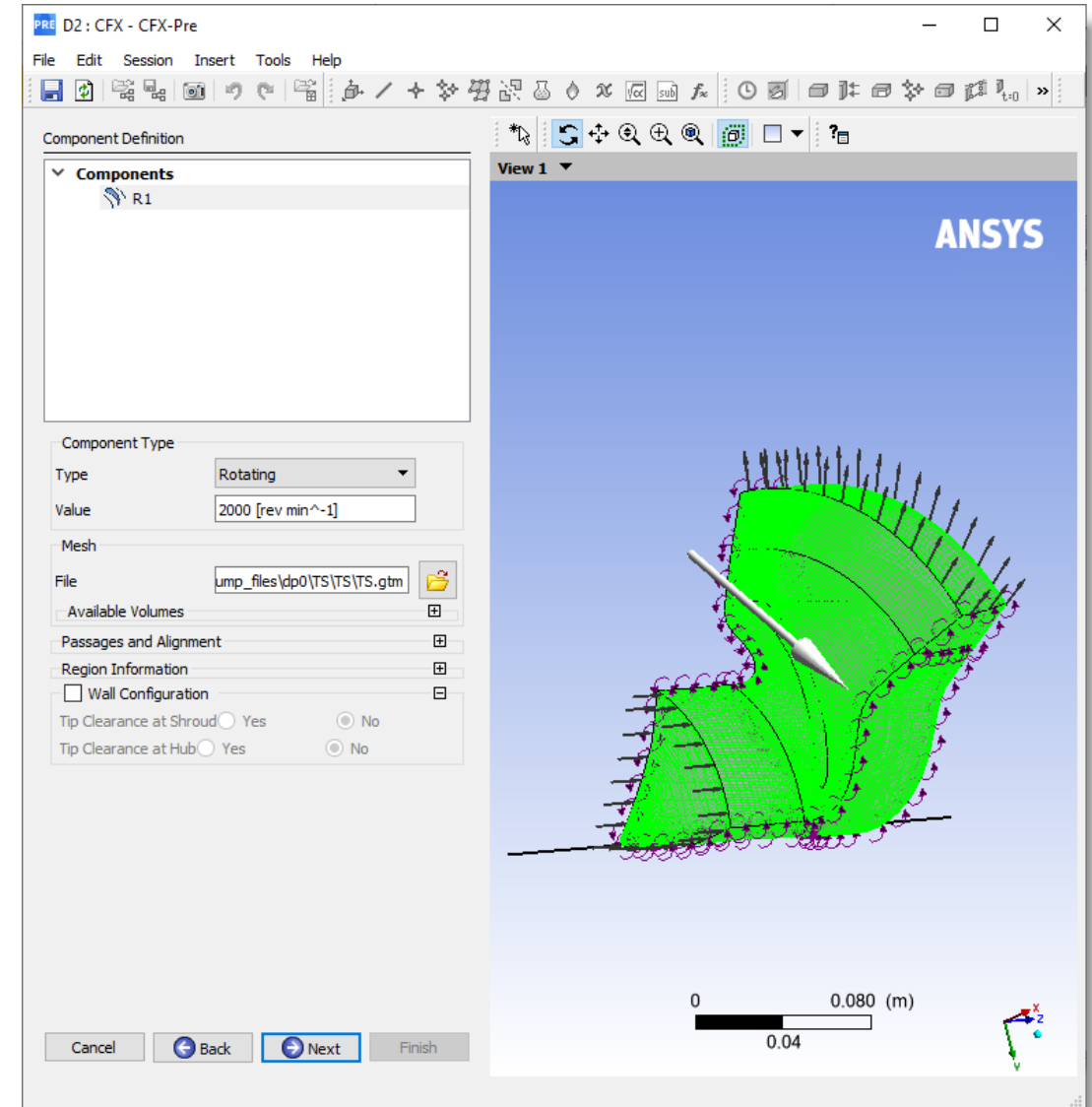
Turbo Mode – Basic Settings

- On this panel you will define the type of machine, rotation axis and whether the case is steady or unsteady
- Set the inputs to this panel as follows:
 - *Machine Type = Pump*
 - *Rotation Axis = Z*
 - *Analysis Type = Steady State*
- Click *Next* at the bottom



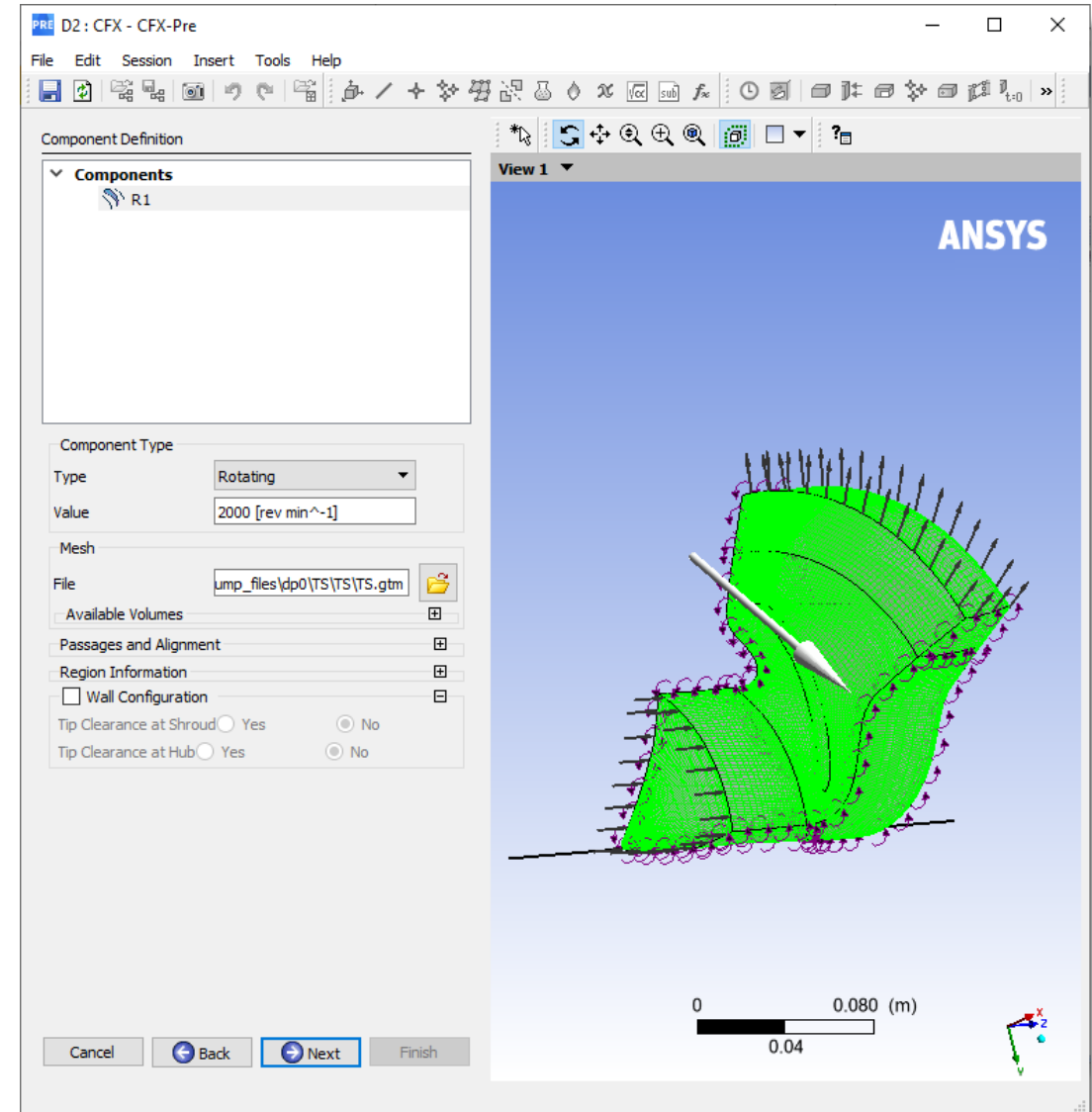
Rotational Speed in CFX

- Defining Rotational Speed
 - CFX uses the right hand rule to define rotation
 - Place your thumb of your right hand in the positive axis of rotation direction (here Z)
 - Your fingers will curl in the positive rotational direction
 - An arrow is shown in the viewer to indicate the direction of rotation
 - This should point in the direction the pump rotates
 - It is correct in this case, but if it was not you would specify a negative rotational speed to switch the direction



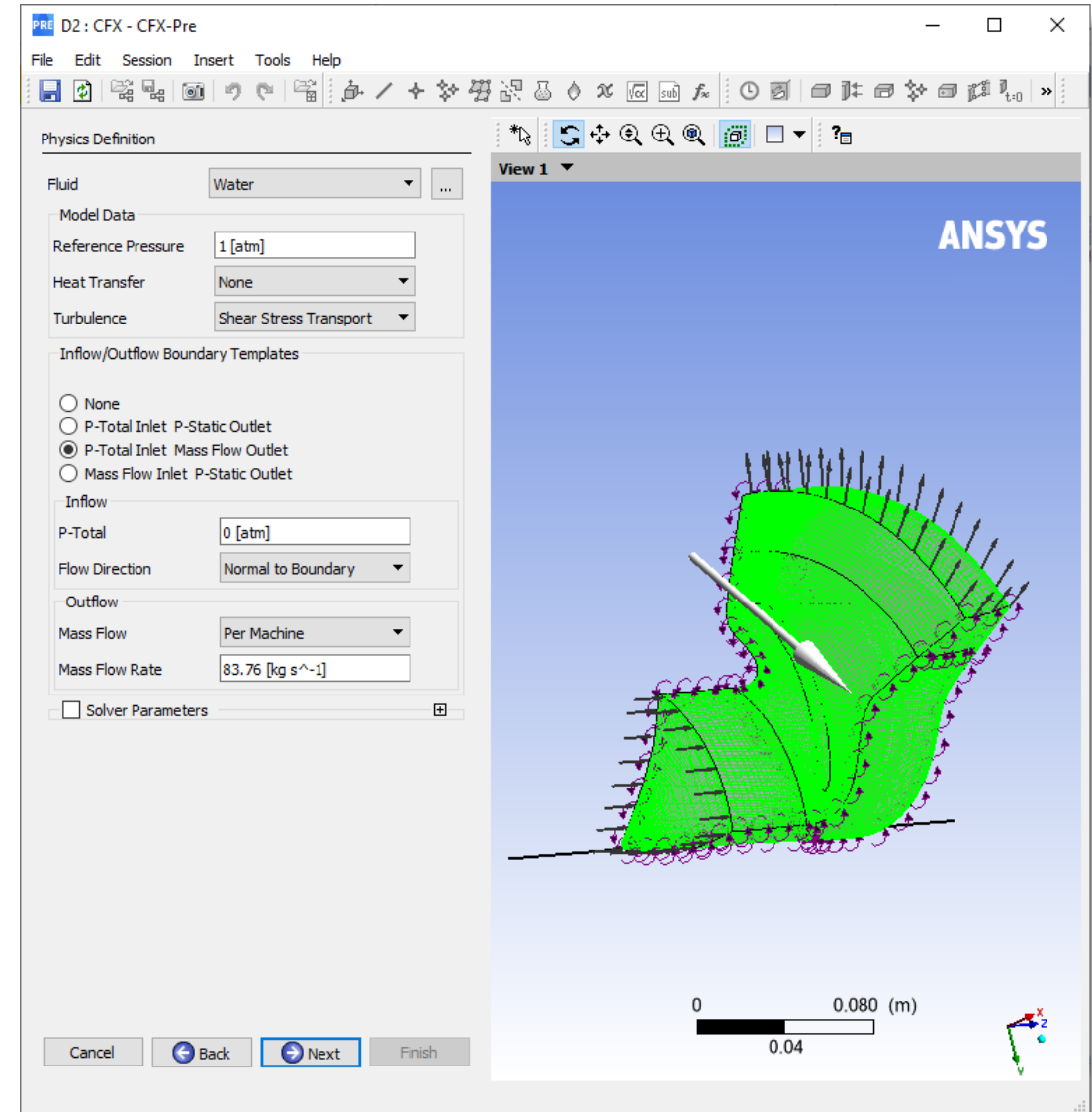
Turbo Mode – Component Definition

- On this panel you will define each component as stationary or rotating
 - You can also create more passages, and identify whether the case has a tip clearance
 - In this case we need to specify the *Component Type* as *Rotating* with a rotational speed
- Ensure the following are set:
 - *Rotational Speed*
 - *Value = 2000 [rev min⁻¹]*
- Click *Next* at the bottom



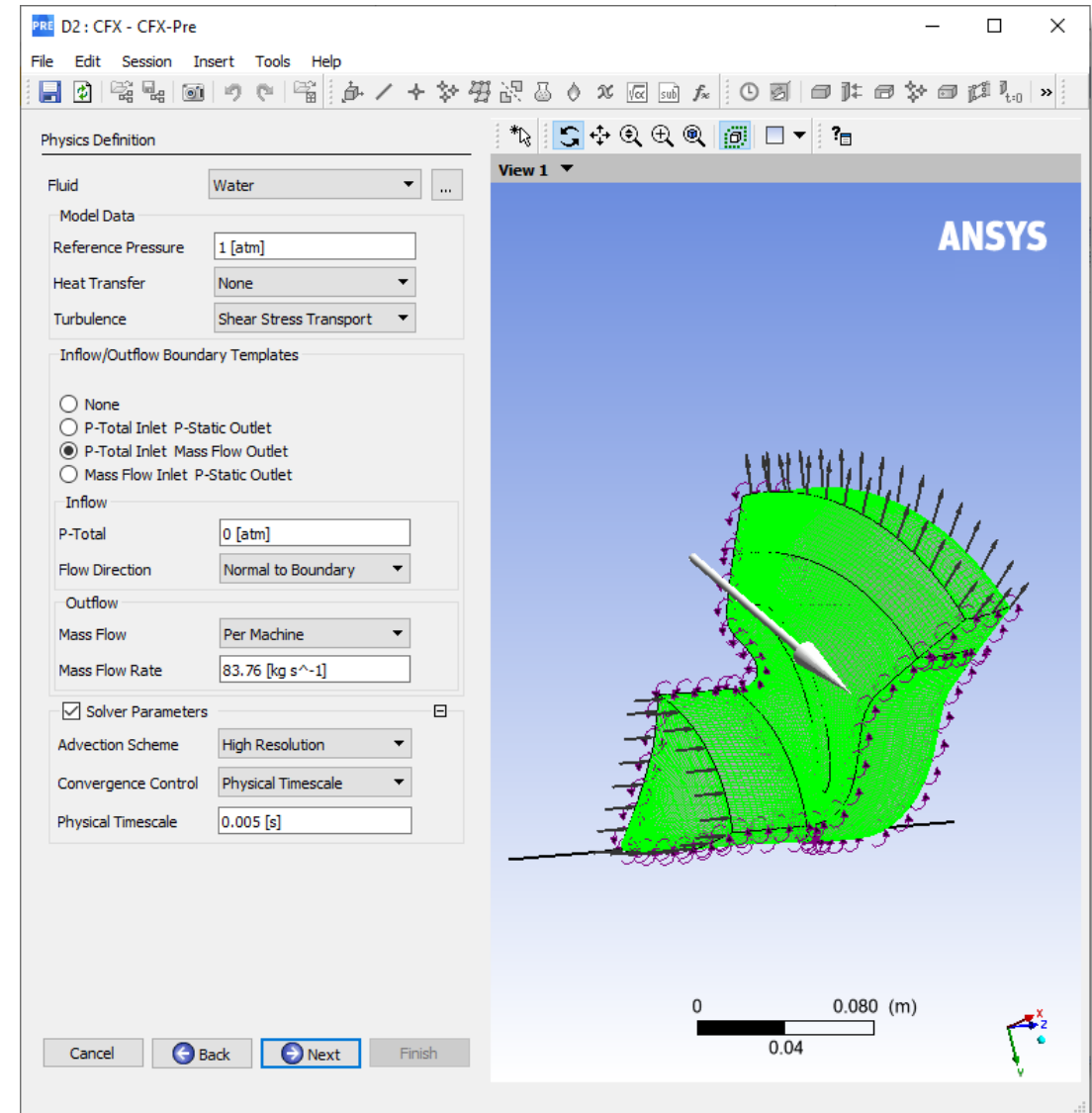
Turbo Mode – Physics Definition

- On this panel you will define a number of physical model settings:
 - Type of Fluid
 - Turbulence Model
 - Energy Model
 - Inlet/Outlet Boundary information
 - Solver Parameters
- See next slide



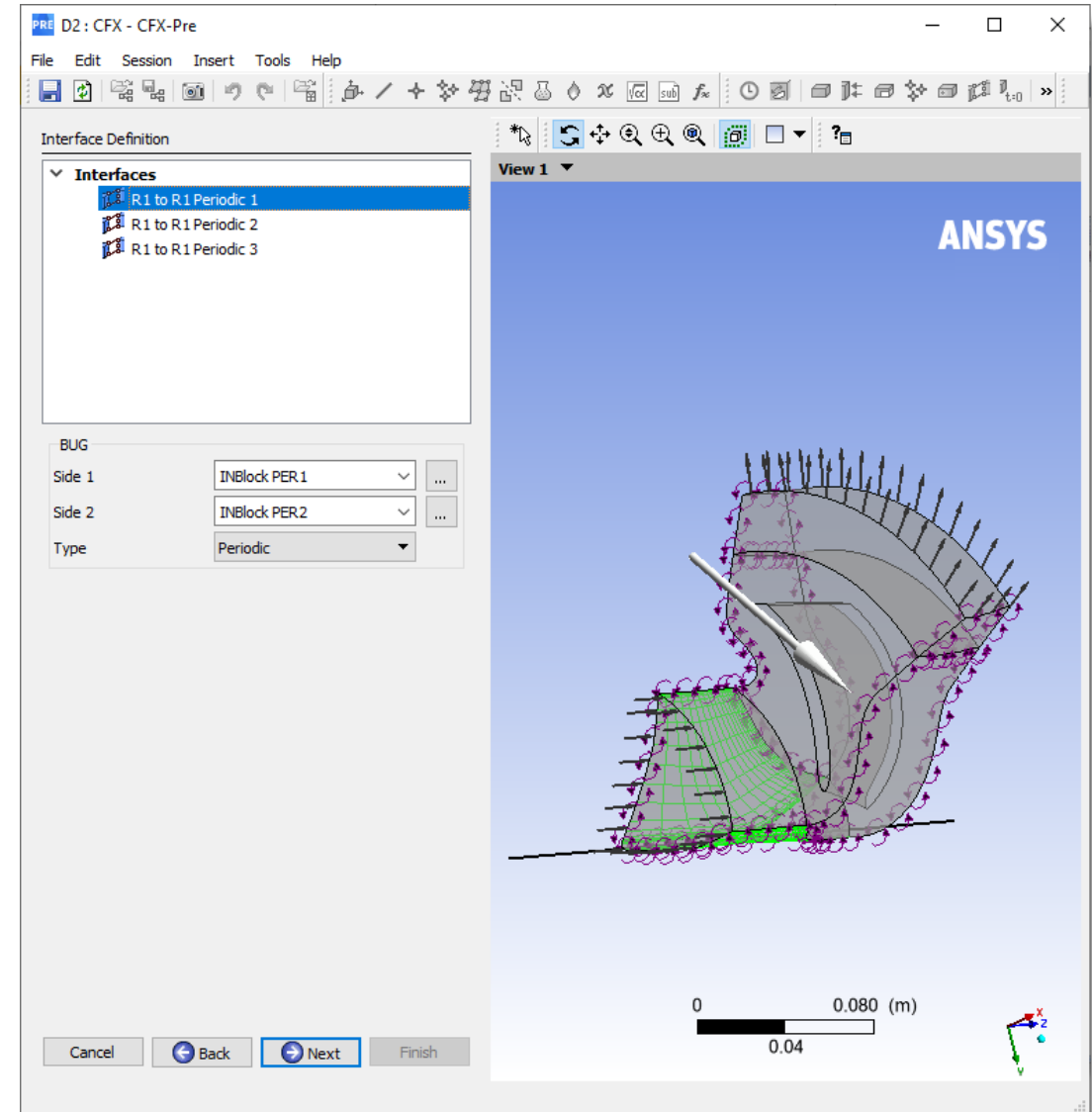
Turbo Mode – Physics Definition

- Set the following:
 - *Fluid = Water*
 - *Turbulence = Shear Stress Transport*
 - *Inflow/Outflow Boundary Template = P-Total Inlet Mass Flow Outlet*
 - *Inflow P-Total = 0 [atm]*
 - *Outflow*
 - *Mass Flow = Per Machine*
 - *83.76 [kg s⁻¹]*
 - *Tick on Solver Parameters*
 - *Under Solver Parameters choose the following:*
 - *Advection Scheme → High Resolution*
 - *Convergence Control → Physical Timescale*
 - *Physical Timescale → 0.005 [s]*
- Click *Next* at the bottom



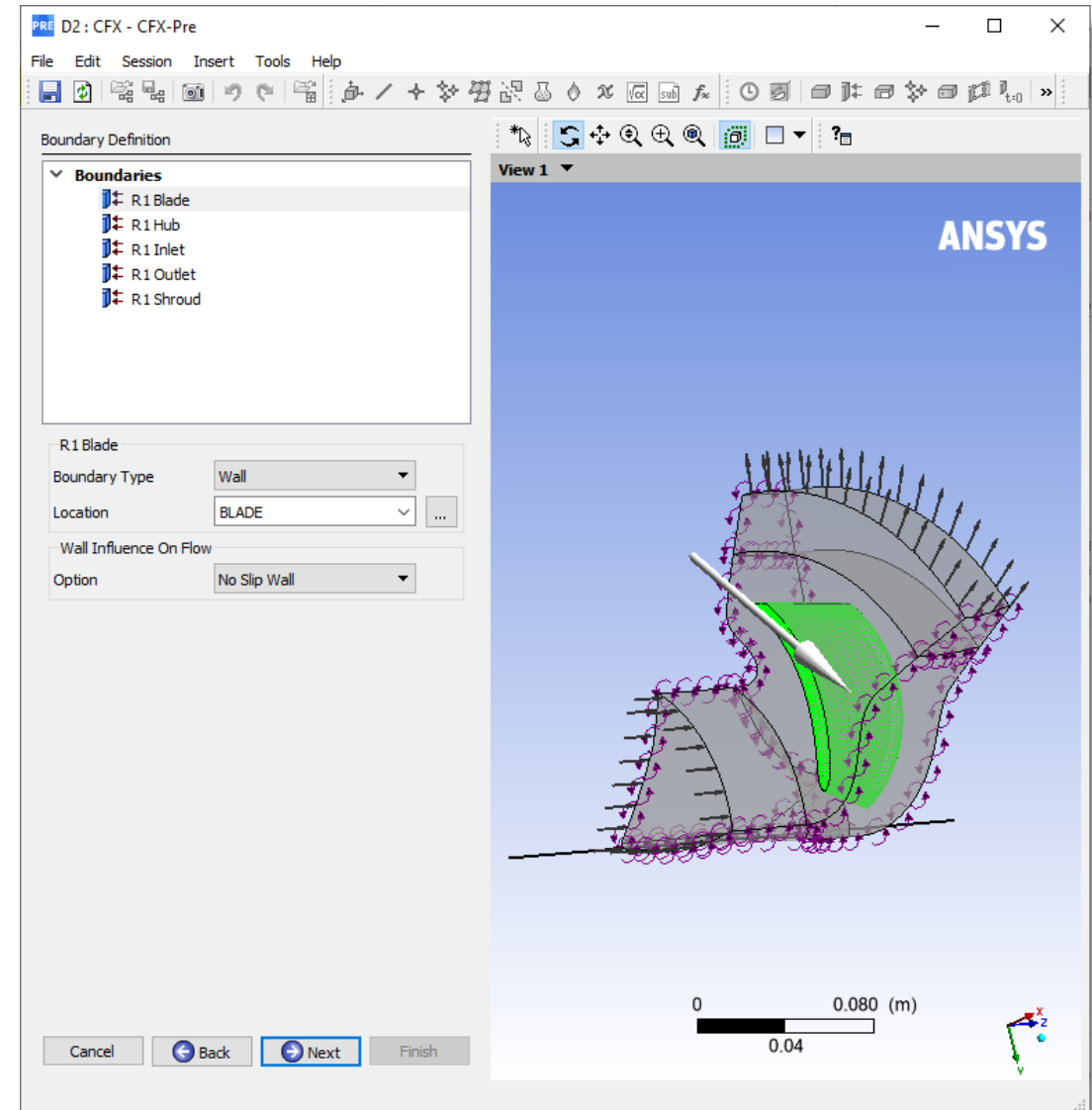
Turbo Mode – Interface Definition

- This panel is used to create interfaces
 - Note that Turbo Pre usually will create all Interfaces automatically
 - In this case it has correctly defined three periodic interfaces
 - Click on each Interface in the list to view and check they are correct (as highlighted in the graphics window)
 - Click *Next* at bottom



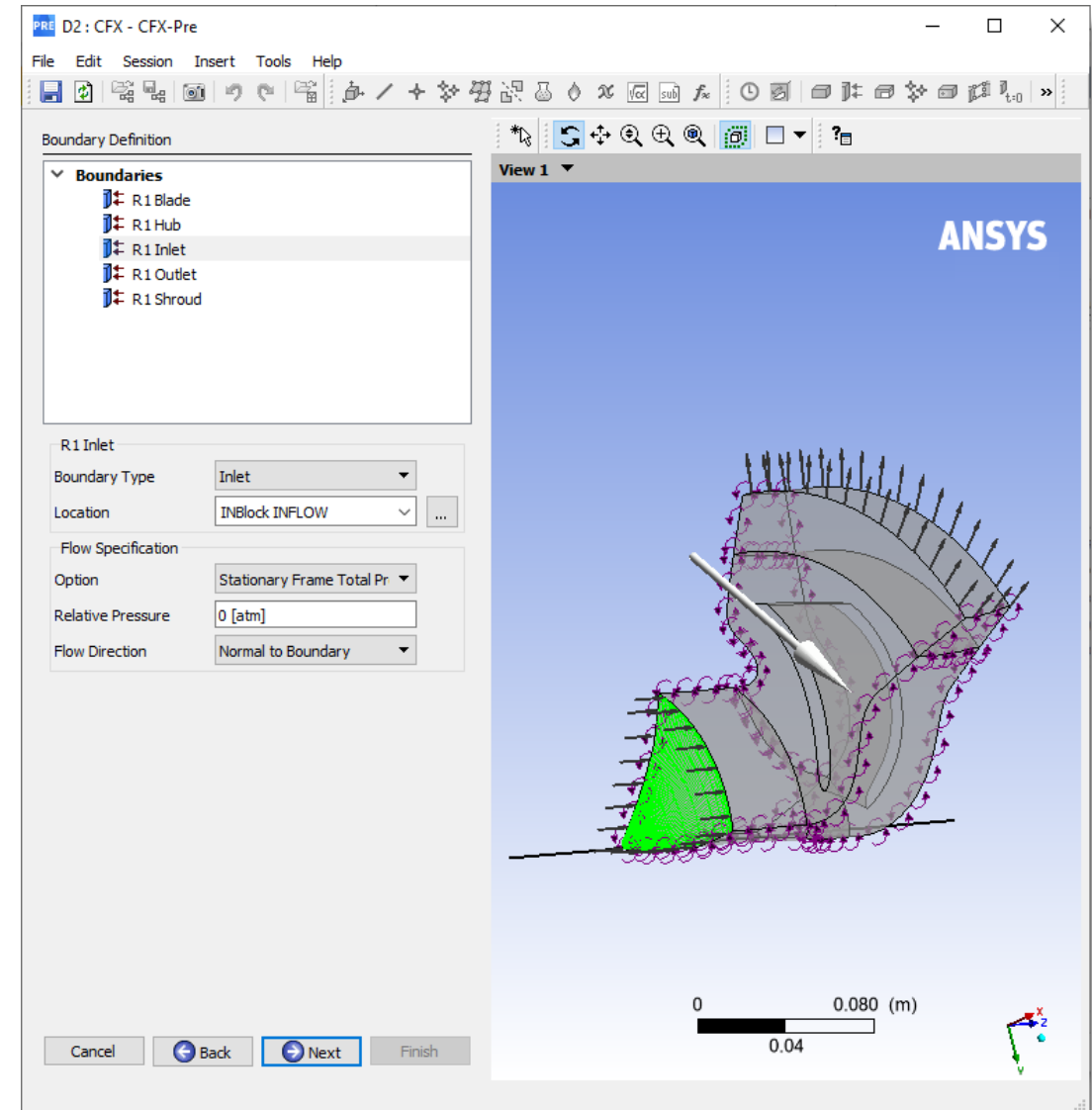
Turbo Mode – Boundary Definition

- This panel is used to create boundary conditions
 - CFX will usually create all required boundary conditions automatically
- Click on *R1 Blade*
 - We can see it has correctly created a boundary condition of a no slip wall for the blade
- Click on *R1 Hub* and *R1 Shroud*
 - These boundary conditions should be similar to *R1 Blade*



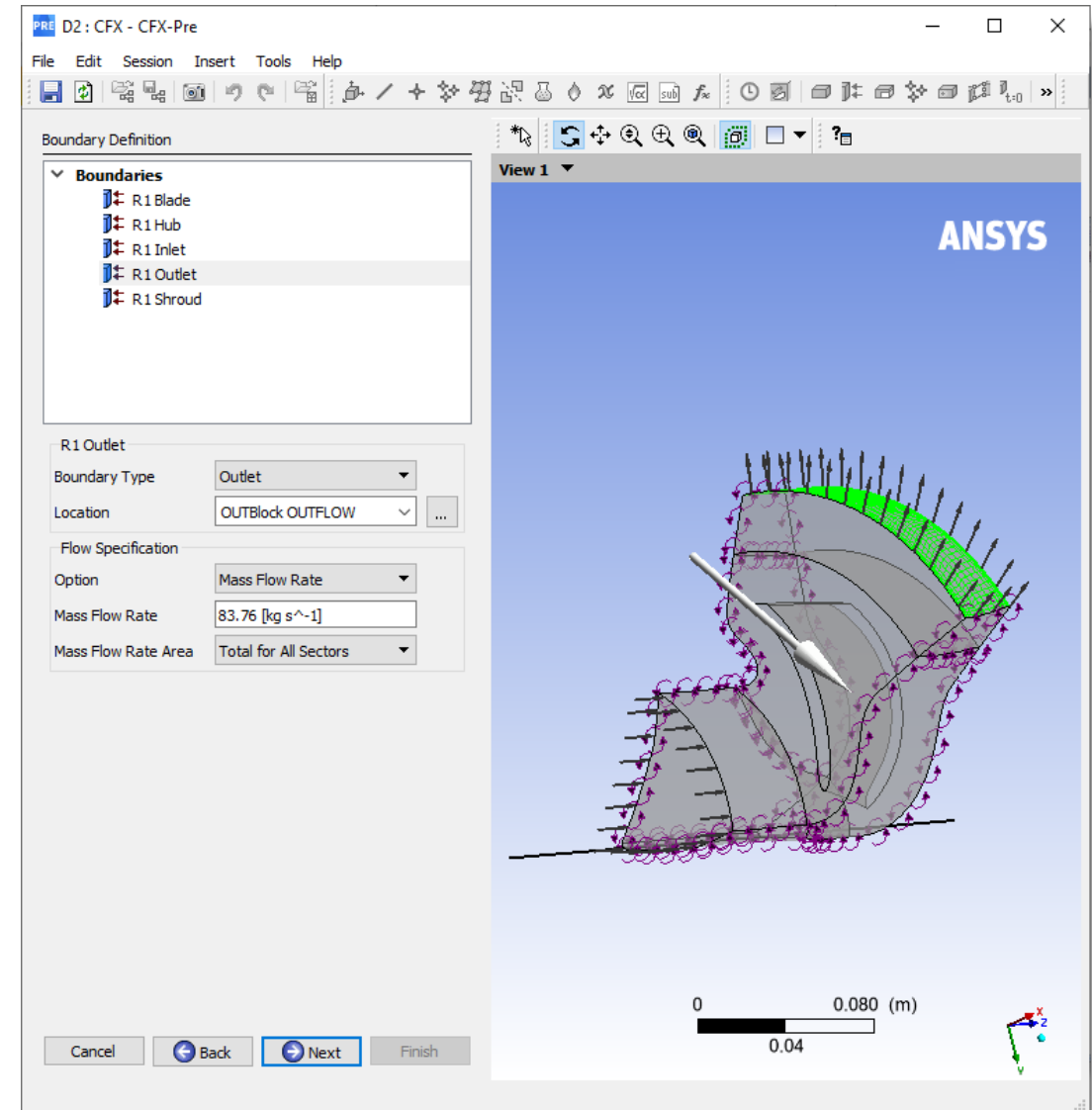
Turbo Mode – Boundary Definition

- Click on *R1 Inlet*
 - We can see it has correctly created an inlet boundary condition
 - The setting for Relative Pressure is taken from the value set previously in the Physics Definition panel
 - There is nothing to change, but it is good to double check your boundary conditions here



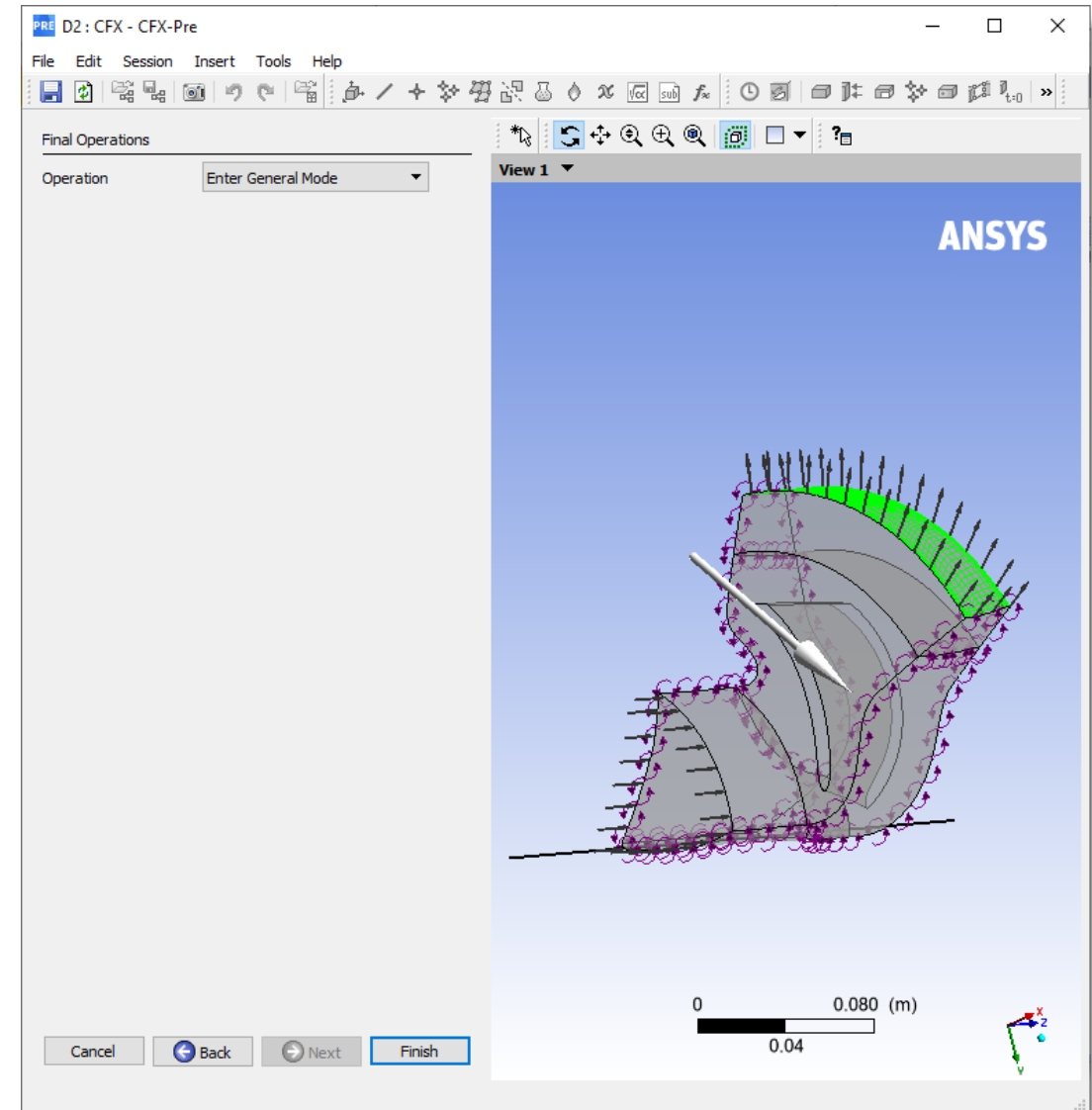
Turbo Mode – Boundary Definition

- Click on *R1 Outlet*
 - We can see it has correctly created an outlet boundary condition
 - The correct flow rate is taken from the value set previously in the Physics Definition panel
 - Note that the *Mass Flow Rate Area* is defined as *Total For All Sectors*
 - This is the total flow rate through all 6 blade passages
 - You could also specify the flow rate through a single passage (1/6th of the total flow rate) by changing this to *As Specified*
 - Click *Next* at bottom



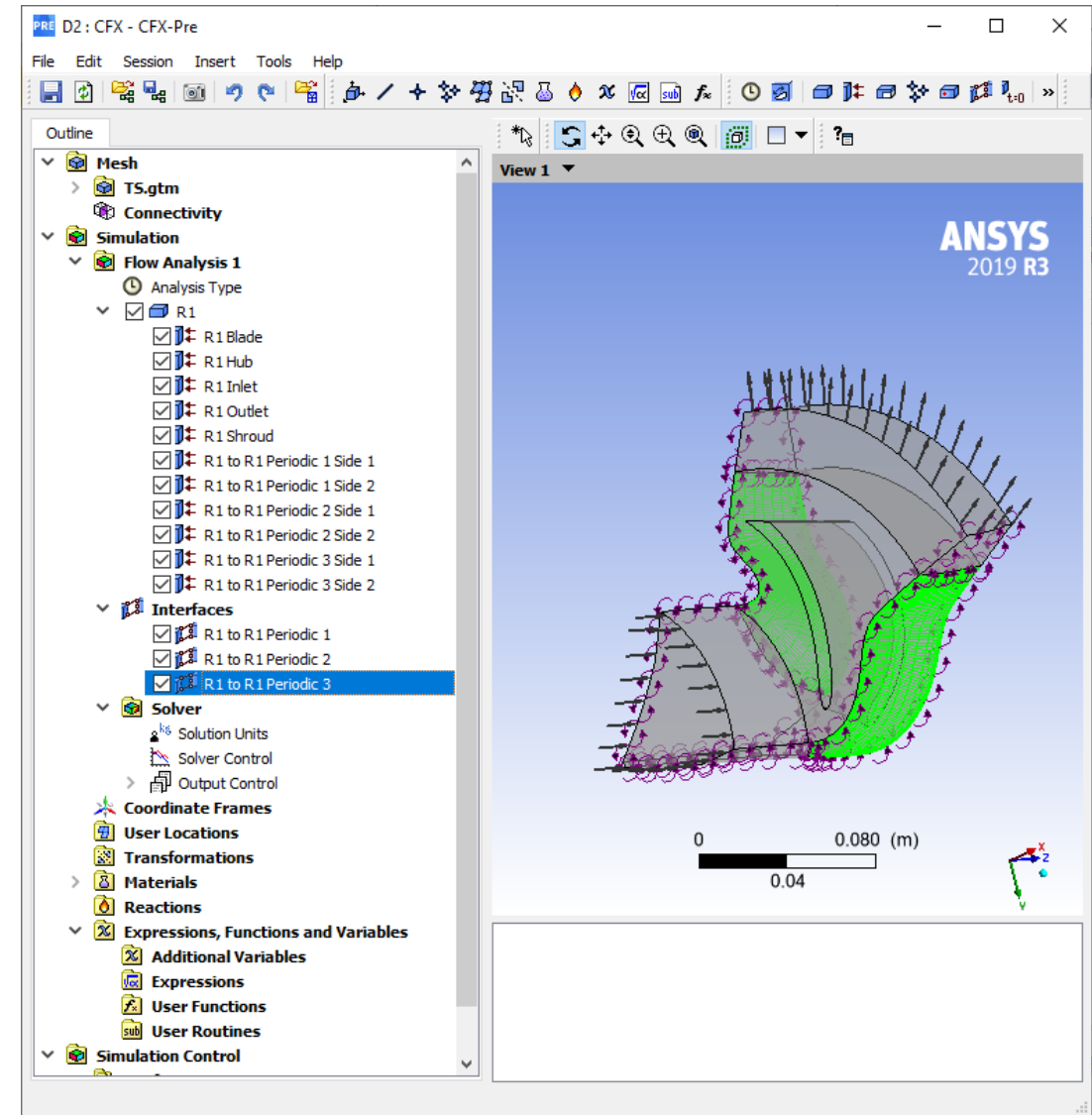
Turbo Mode – Final Operations

- The main setup is complete
 - We will enter the General Mode by clicking *Finish* at the bottom
 - This allows us to set the rest of the model up in the usual way for CFX cases



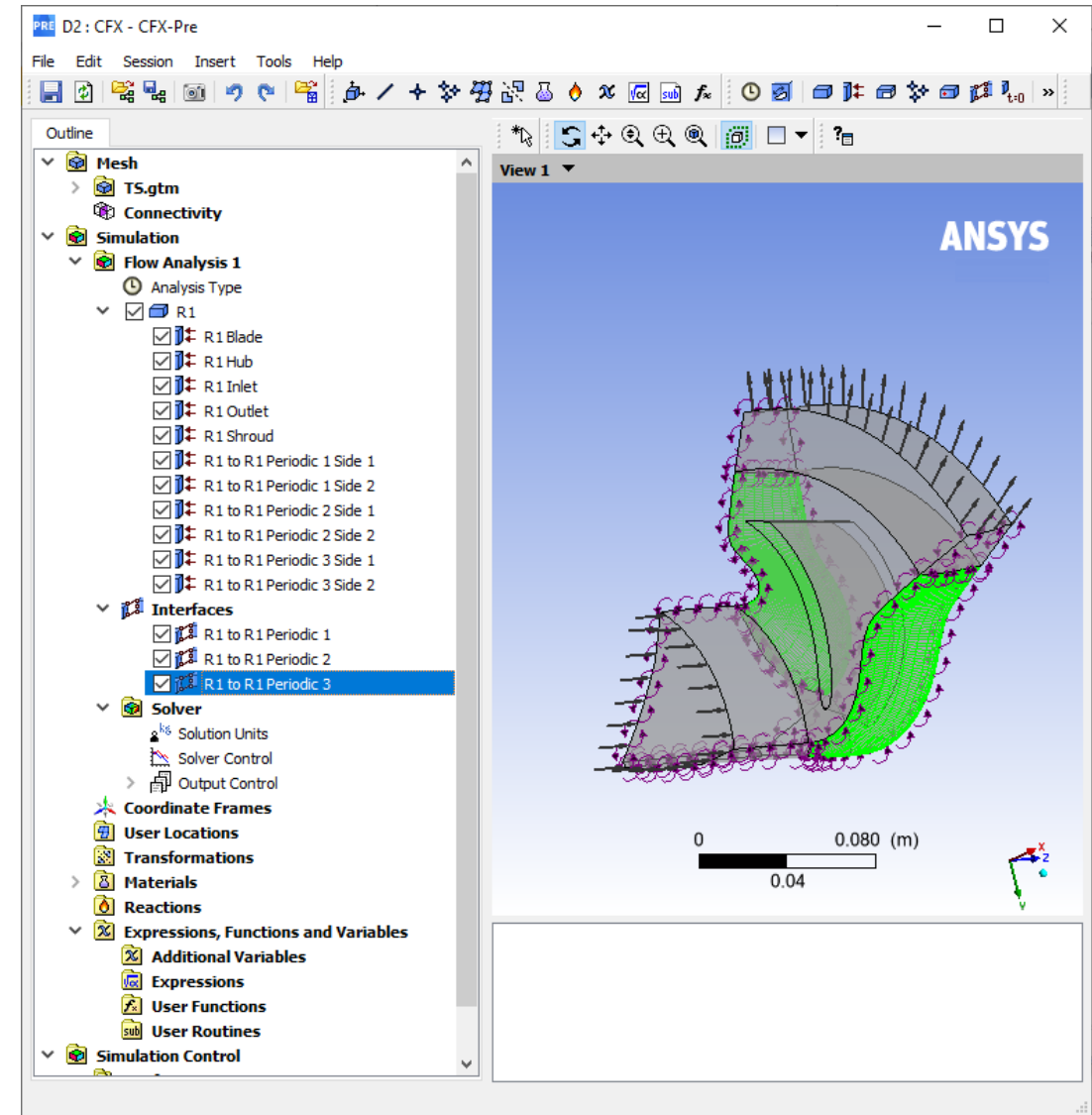
General Mode

- Interfaces
 - Interfaces are created to...
 - Make periodic zones for single passage models.
 - Connect mesh blocks together nonconformally
 - Note that a number of Domain Interfaces and Boundary Conditions have been created automatically
 - You can double click on them to review



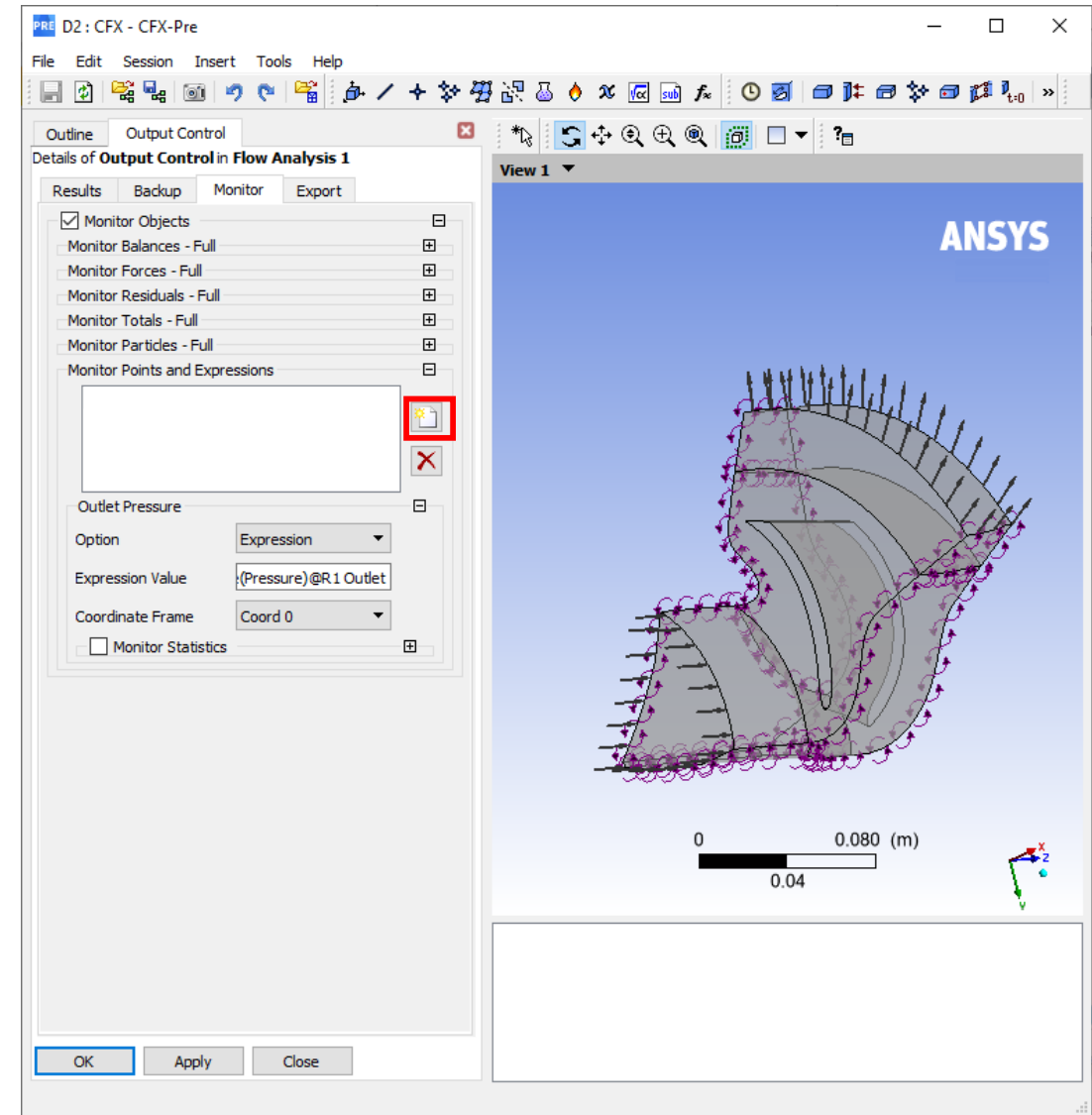
Monitors

- Creating a *Monitor Point* will help to determine convergence
- Typically you would want to create many monitor points for quantities of interest, such as head rise, efficiency, shaft power, torque, etc.
- In this case we will make a *Monitor Point* to monitor the average pressure at the outlet



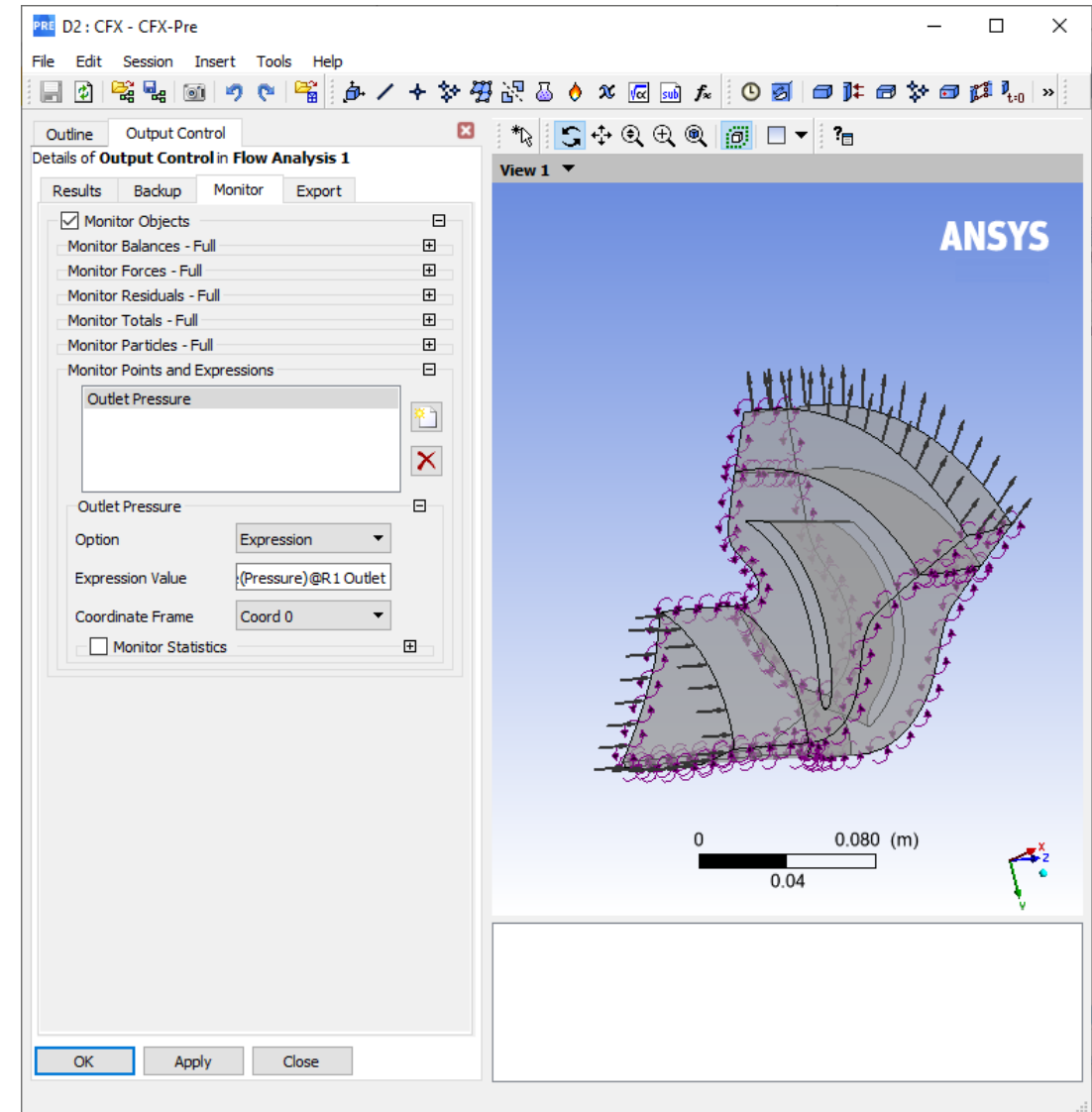
Monitors

- In the *Outline Tree*:
 - Double Click *Output Control*
 - Switch to the *Monitor* Tab
 - Check the *Monitor Objects* Checkbox
 - Click *New*
 - In text box that appears, name the monitor point *Outlet Pressure*



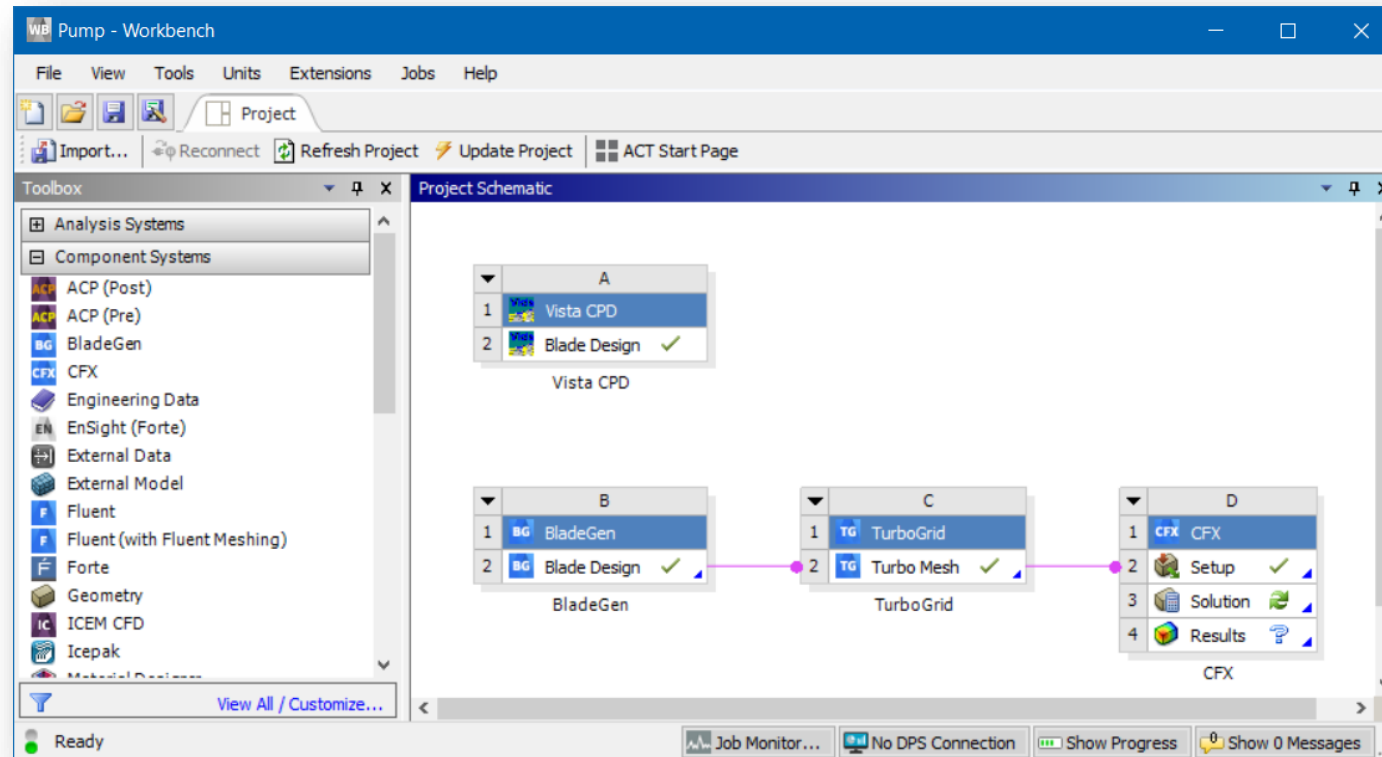
Monitors

- Under the *Outlet Pressure* Monitor:
 - *Option = Expression*
 - *Expression Value*
 - *massFlowAve(Pressure)@R1 Outlet*
 - Click *OK*
- Close CFX-Pre
- We are now ready to solve the case



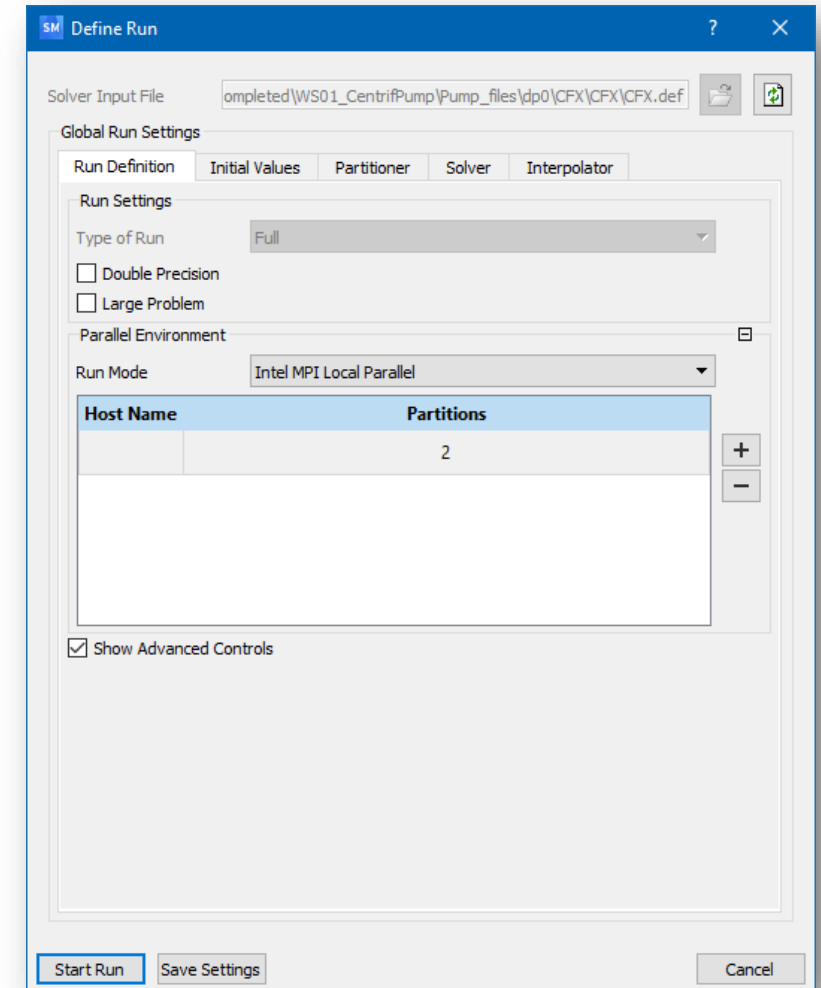
Solution Setup

- Go back to Workbench and double click the *Solution* cell *D3*



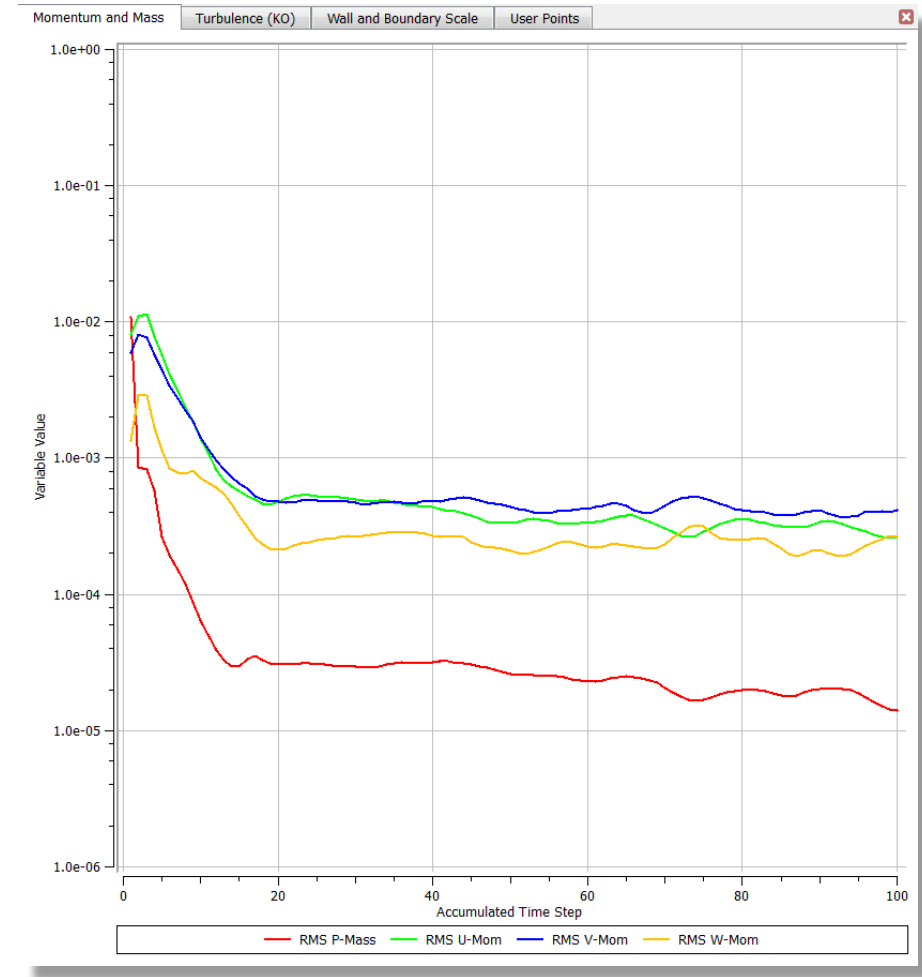
Solution Setup

- We will run the case using two cores
 - Set *Run Mode* to *Intel MPI Local Parallel*
 - Use 2 for the number of partitions
 - If you have more cores on the workstation, feel free to hit the *+* button to the right to increase the core count
- Click *Start Run*



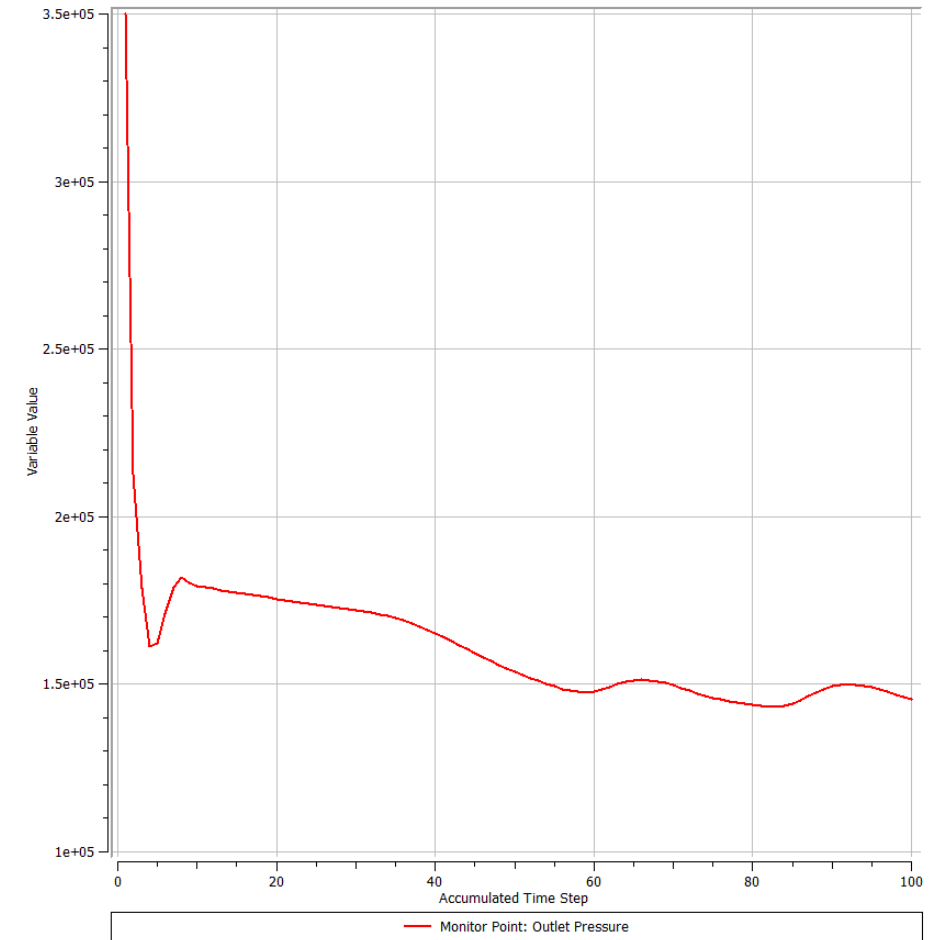
Solver Convergence

- As the solution converges, we can see that the residuals reduce somewhat however the case ends after 100 iterations which is the default maximum number of iterations
- Ideally we would have liked the *RMS Residuals* to decrease beyond $1e-4$ for all quantities



Monitor Solver Convergence

- The monitor point for outlet pressure seems to be oscillating near 150,000 Pa
- This case is not well converged
- In the next workshop we will examine why using CFD Post



Summary

- This workshop has covered:
 - Setting up a single frame of reference solution using CFX-Pre in turbo mode
 - Creating CEL expressions to monitor during the solution
 - Solving the case and judging convergence