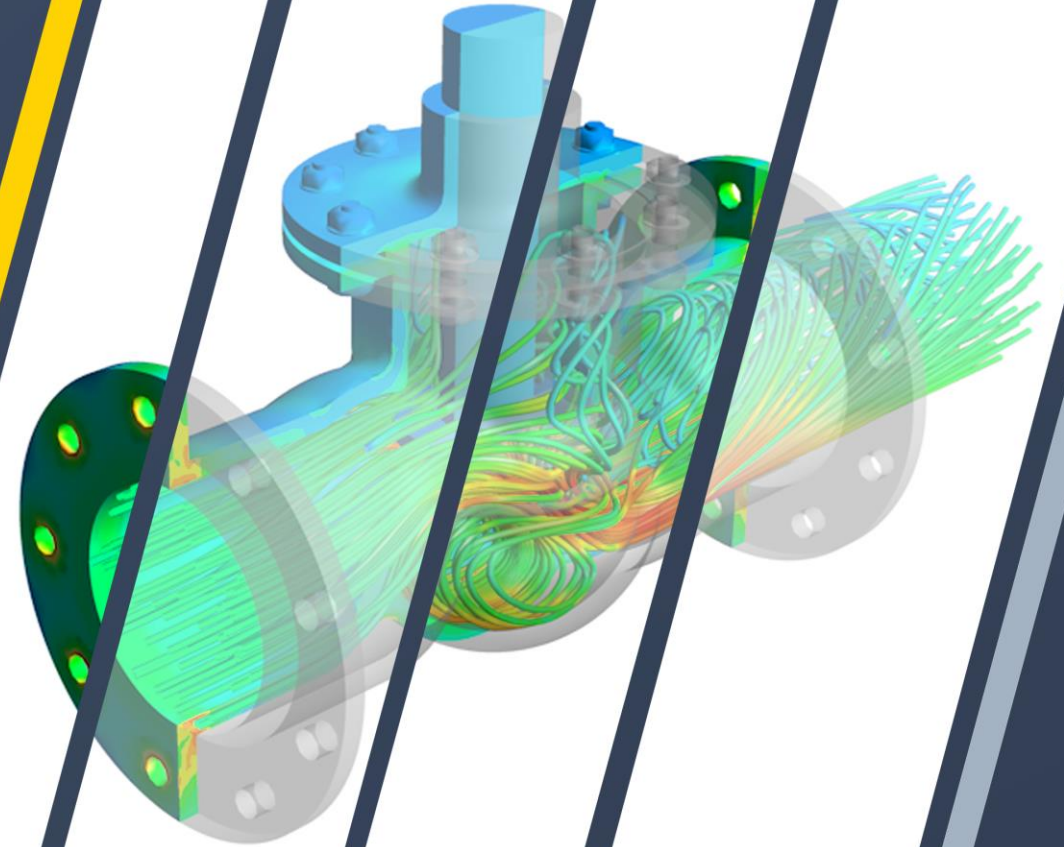# Workshop 04.1: Centrifugal Pump with a Volute

ANSYS CFX Rotating Machinery Modeling

Release 2019 R3

Introduction

- Workshop Description:
  - This Workshop will illustrate how to setup a simulation involving multiple frames of reference
- Learning Aims:
  - Setting up a case with multiple frames of reference
  - Copying/rotating single passage meshes to create full 360 impeller
  - Setting up domain interfaces
 - Post-processing multiple frames of reference solutions
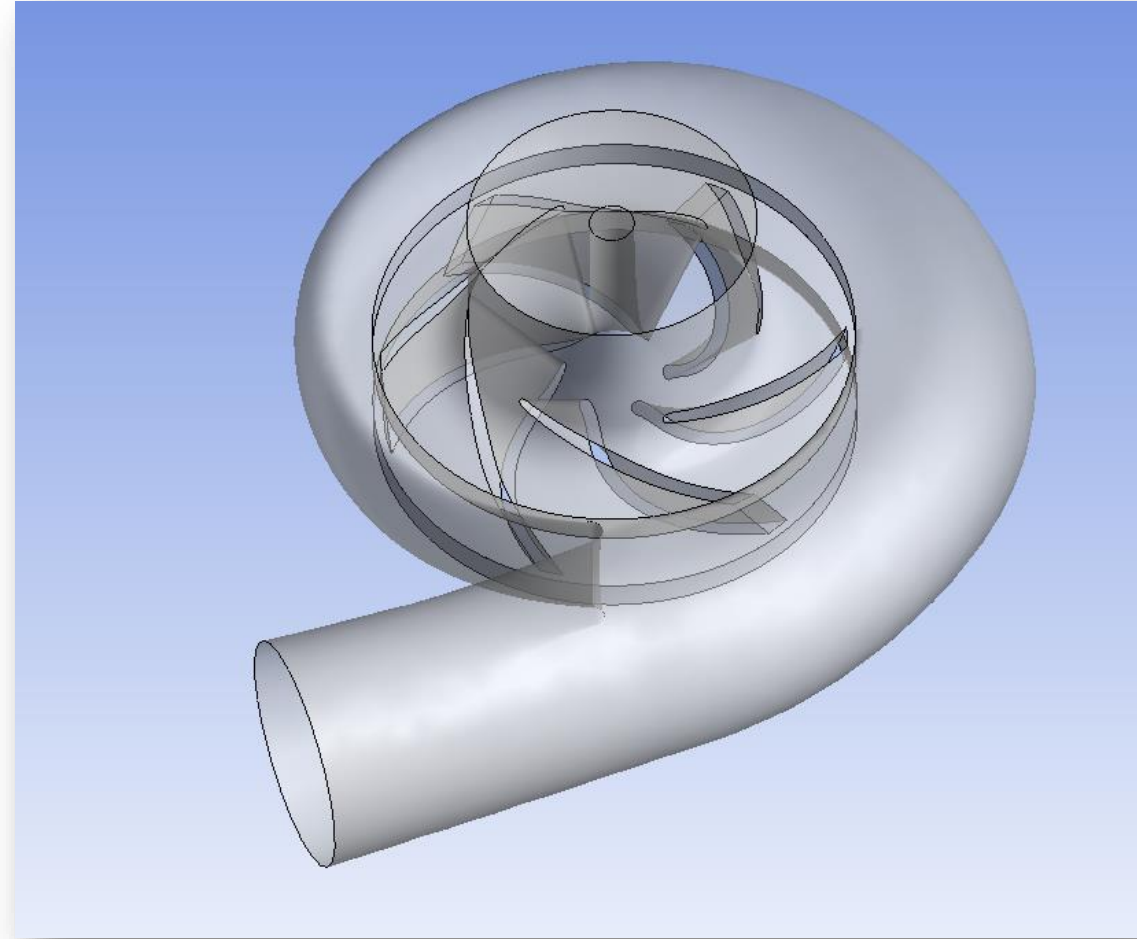
# Background

Domain Interfaces

ANSYS®
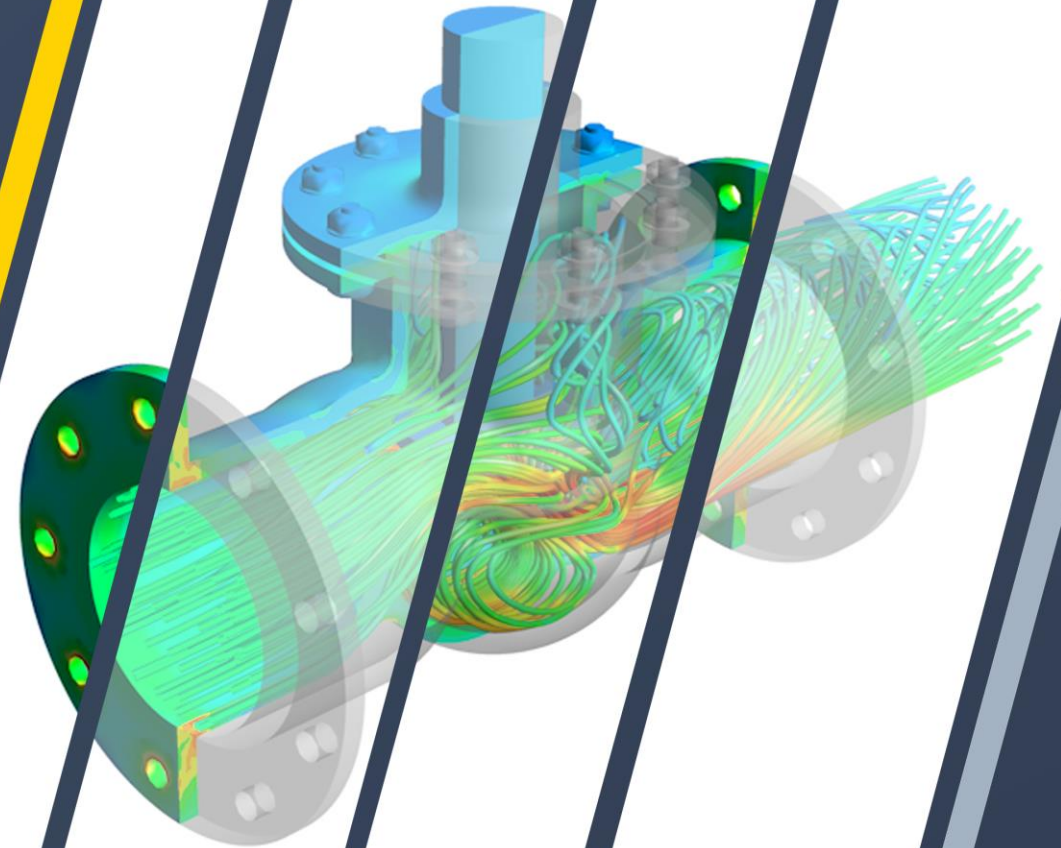
Background

- Recall that domain interfaces are used for:
  - Connection of mismatched meshes (hex to tet for example)
    - ➢ a single mesh file may contain non-matching mesh regions and require non-conformal interfaces
  - Changes in reference frames between domains
    - ➢ even if the mesh matches
  - Connect different types of domains together (e.g. Fluid to Solid)
 - Create periodic regions within a domain

© 2019 ANSYS, Inc.
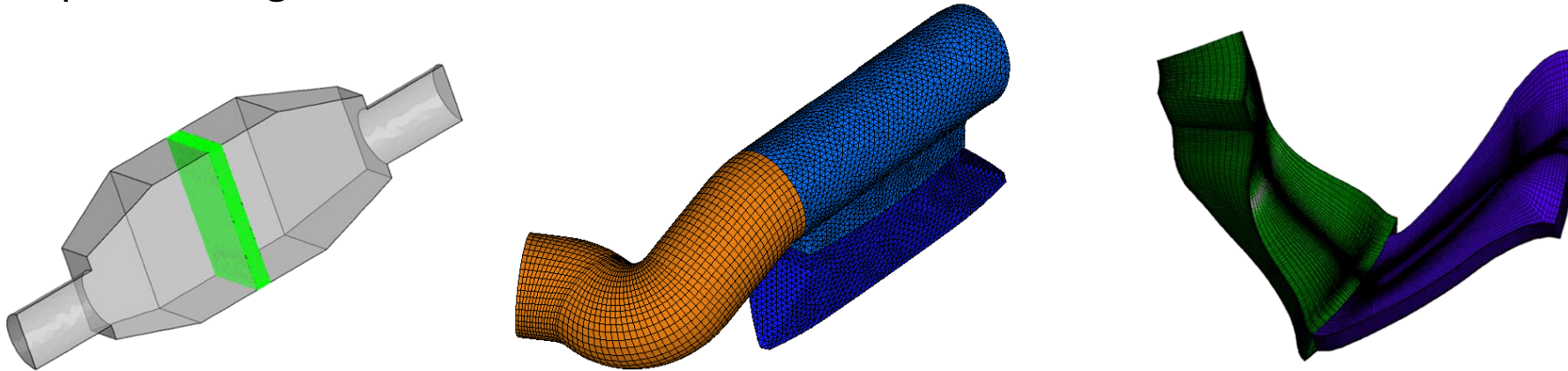
Inserting Domain Interfaces in CFX

- To create a domain interface right-click on the *Flow Analysis* or use the toolbar icon


© 2019 ANSYS, Inc.
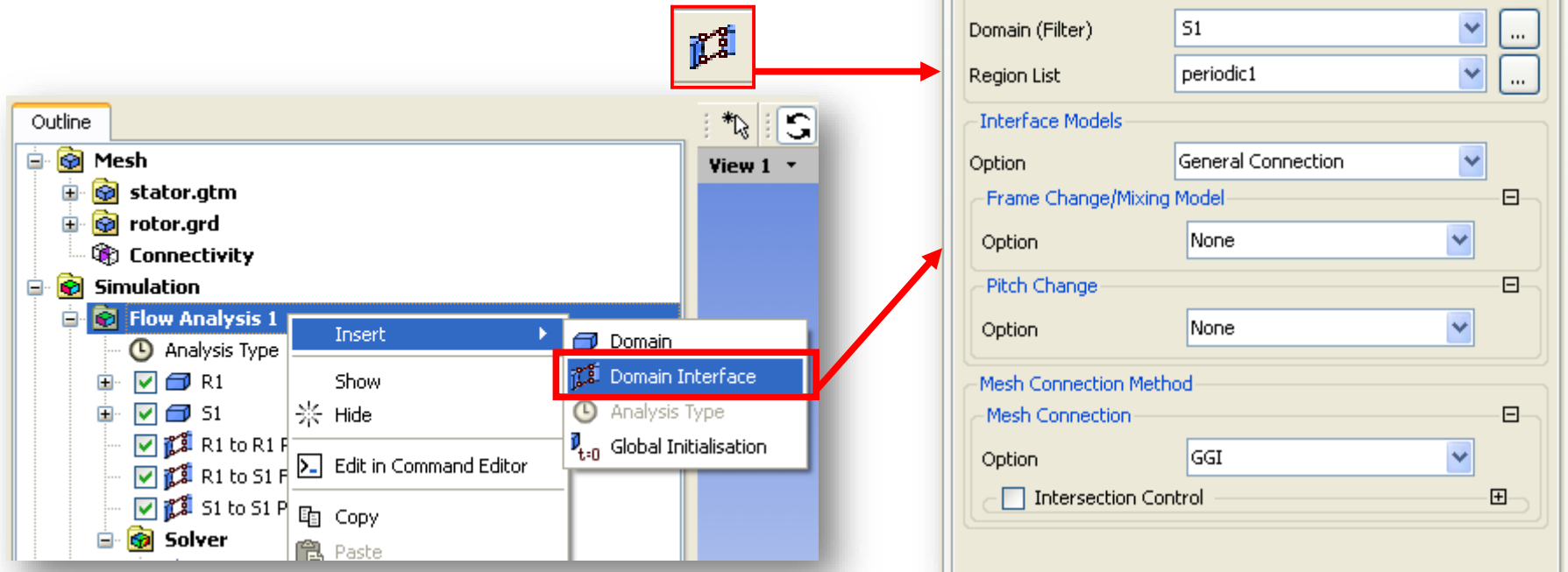
Domain Interfaces and Boundary Objects

- After creating a domain interface 3 new objects are created in the *Outline* Tree

- The interface object is at the *Flow Analysis* level
  - This is the object you should edit to make changes to the domain interface

- Within each domain a *Side 1* or *Side 2* boundary condition is automatically created
  - In general do not edit these objects
  - They will be automatically updated when changes are made to the interface object

The Side 1 and Side 2 boundary conditions

The Interface object
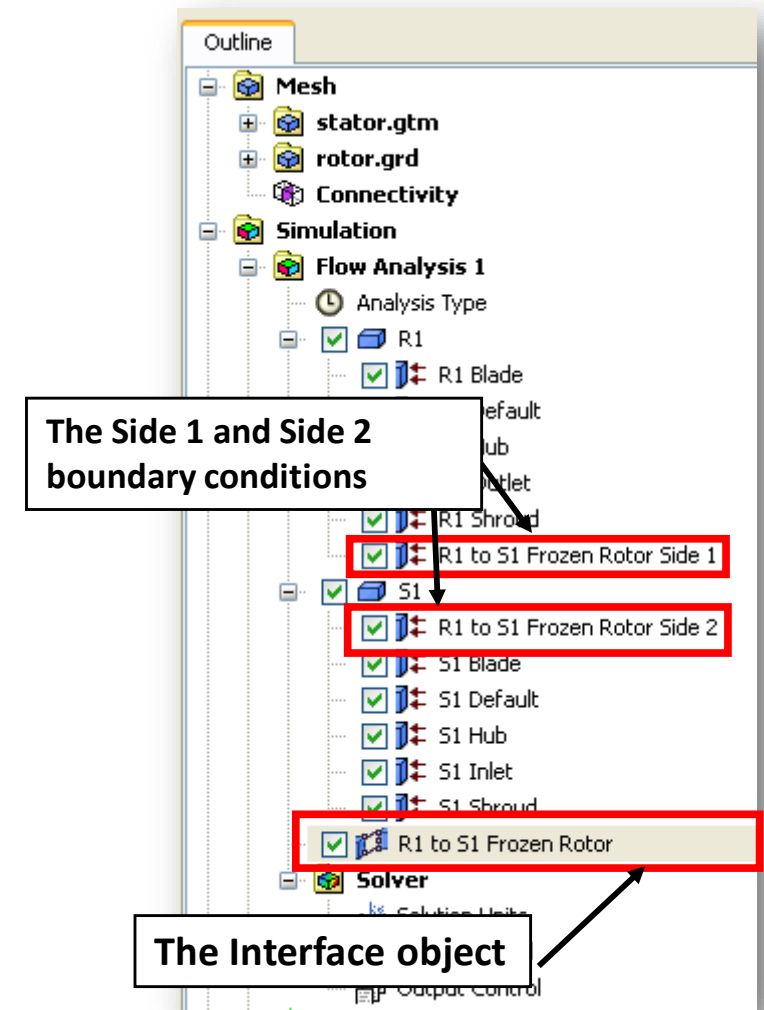
Domain Interfaces Panel

- Domain Interfaces connect two sets of surfaces together
  - *Side 1* and *Side 2*

- First select the domain combination to be connected

- Then select the *Side 1* and *Side 2* surface sets
  - The *Domain* (Filter) just limits the scope of the *Region List* to make selection easier

- The *Interface Models* and *Mesh Connection* Method control how data is transferred across the interface
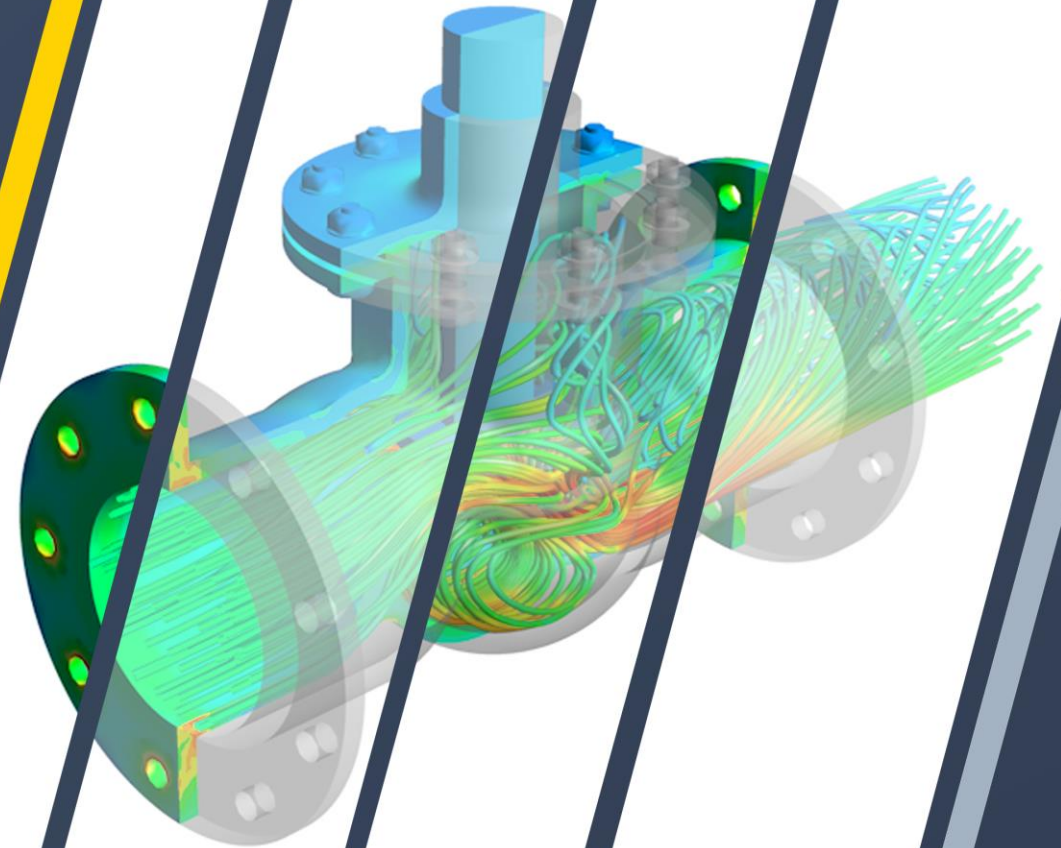
© 2019 ANSYS, Inc.

ANSYS®
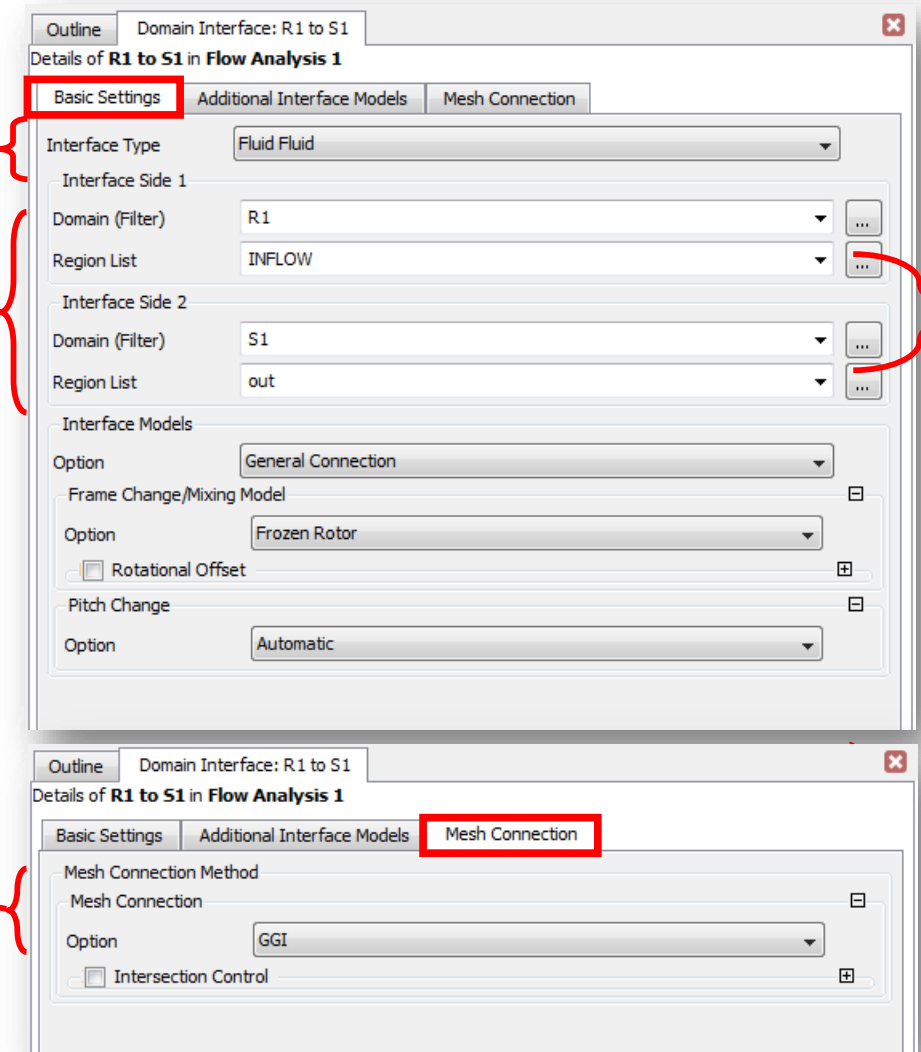
Workshop Instructions

Setup

- Start Workbench and *File>Open* the file: *PumpVolute_mesh.wbpz* provided with the workshop inputs
- *In the Save As dialogue box edit the File Name to PumpVolute.wbpj and click Save*
- This archive contains a mesh for a single passage impeller and a mesh for a volute
- Drag and drop a *CFX Component System* on top of the *Turbo Mesh* cell (*B2*)
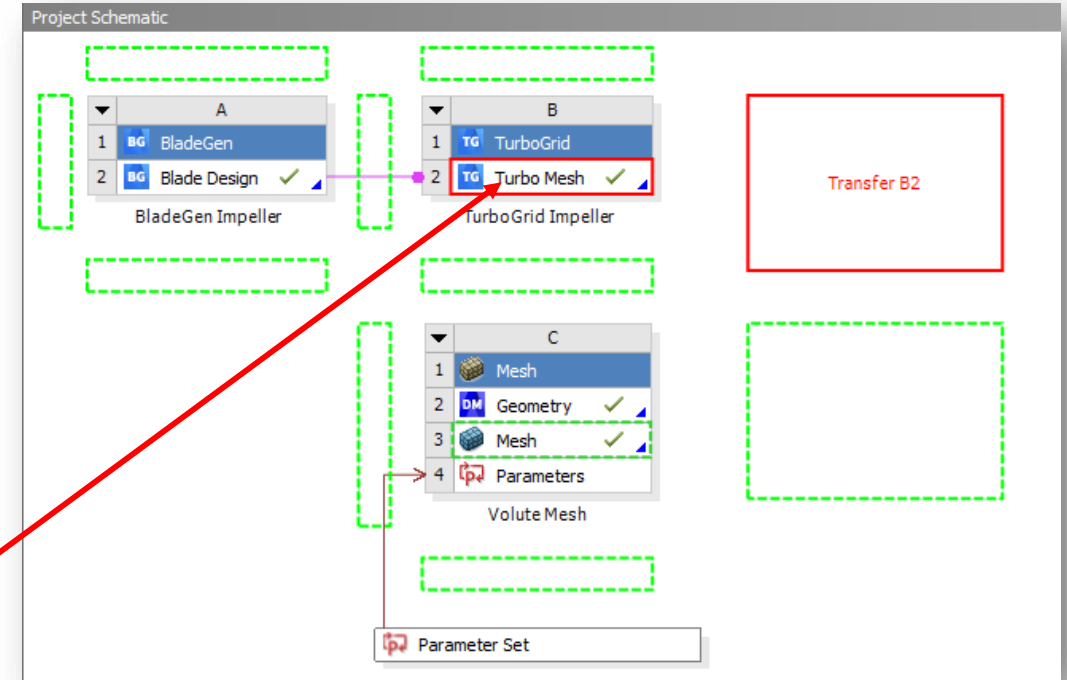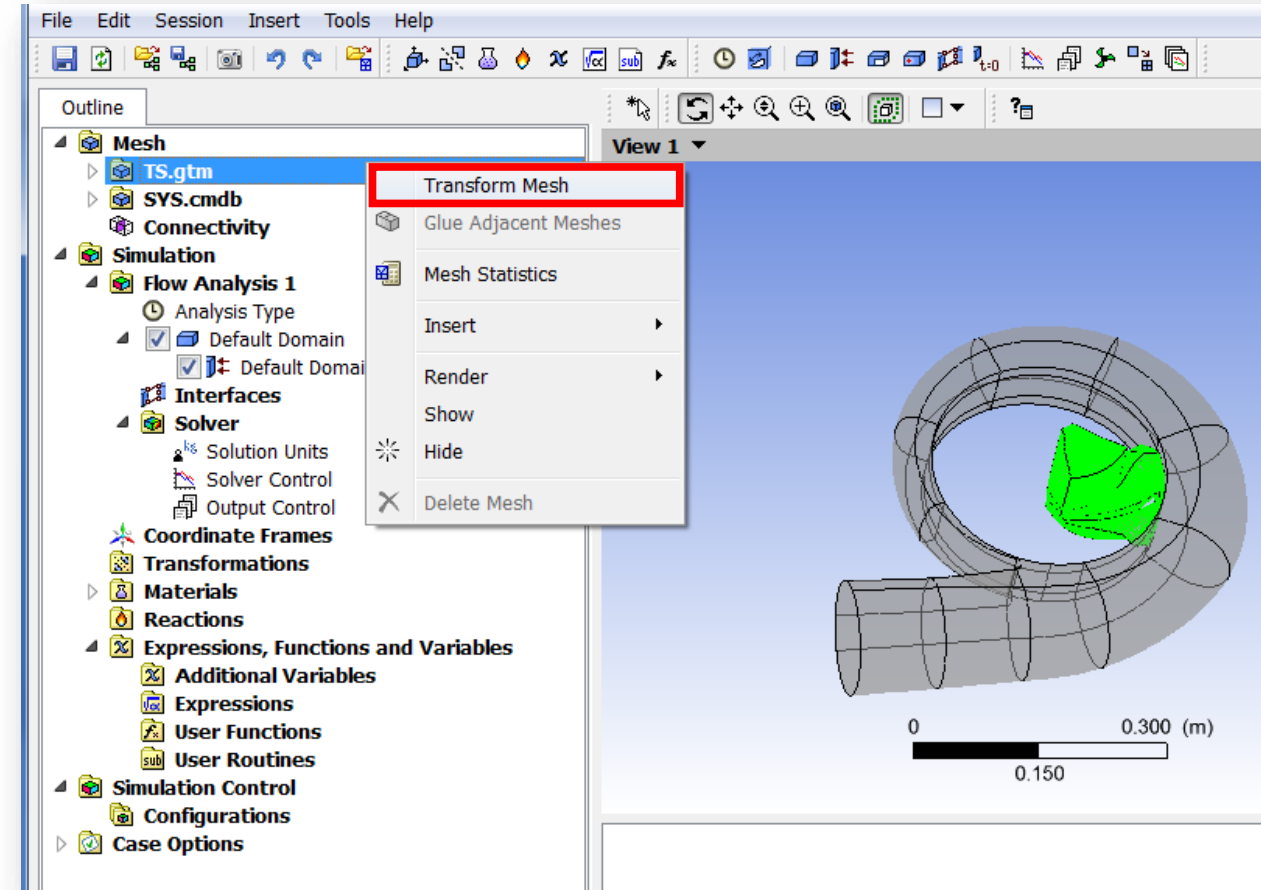
Setup

- Connect the *Volute Mesh* (*D3*) to the *CFX Setup* cell (*C2*)

- Right click on the *Volute Mesh* cell (*D3*) and *Update*

- Double click on the *CFX Setup* cell to open *CFX-Pre*


© 2019 ANSYS, Inc.
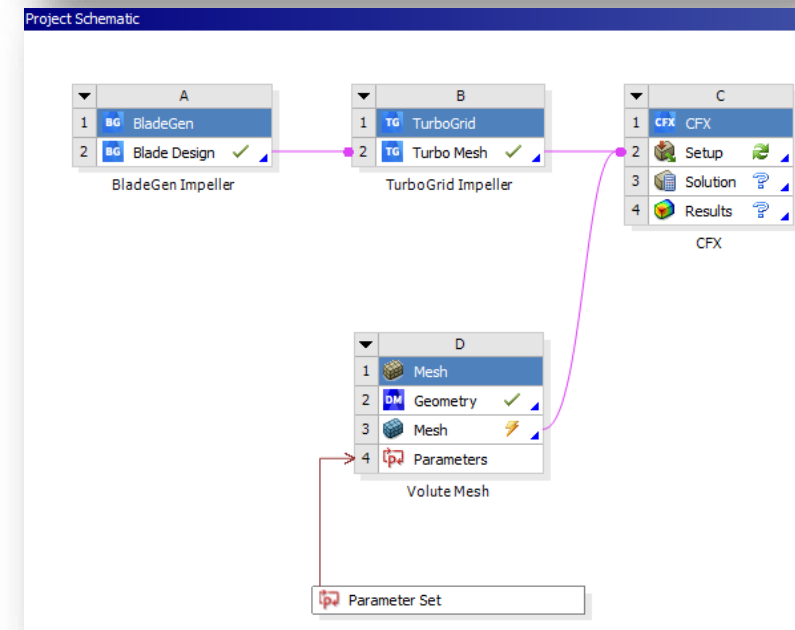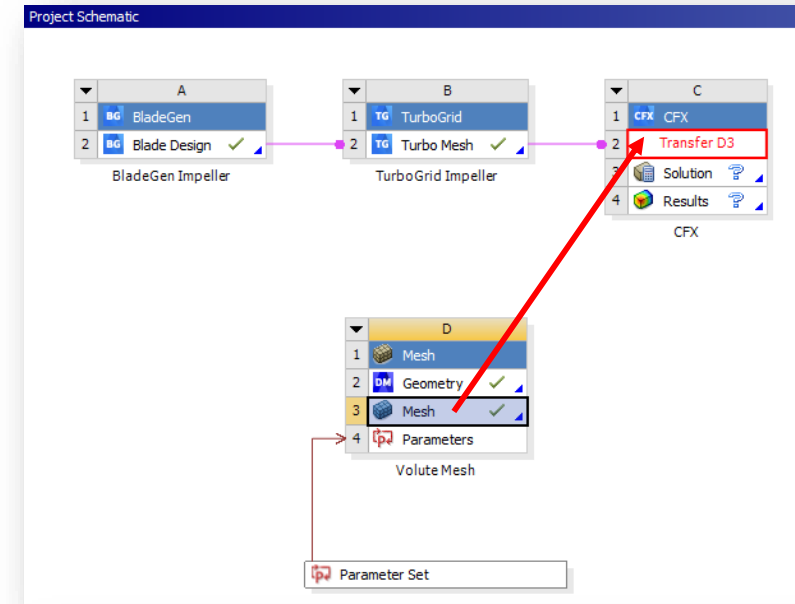
Setup

- When you open *CFX-Pre*, you will see that there is only a single passage of the pump impeller mesh
- We want to have the full 360°
- RMB on *TS.gtm* in *Outline* Tree
  - Select *Transform Mesh*

© 2019 ANSYS, Inc.

Setup

- In the *Mesh Transformation Editor*
    - Set *Transformation* to *Turbo Rotation*
    - Set *Passages in 360* to 6
    - Set *Passages to model* to 6
    - Check the *Automatic Transformation Preview* checkbox
        - ➤ To verify the mesh to be generated is correct
    - Click *Apply*
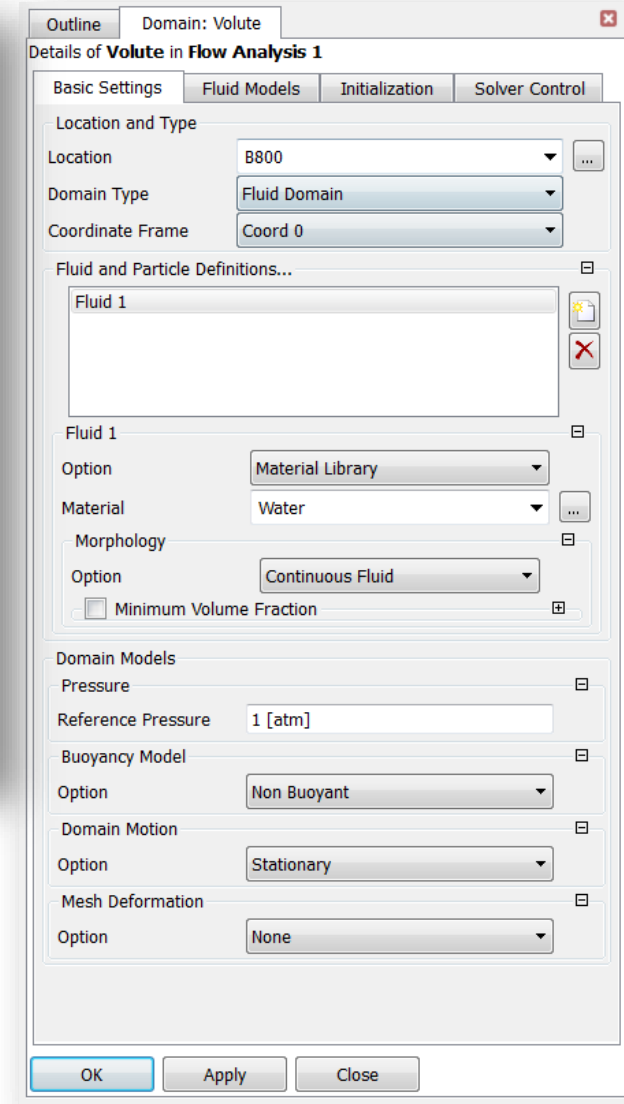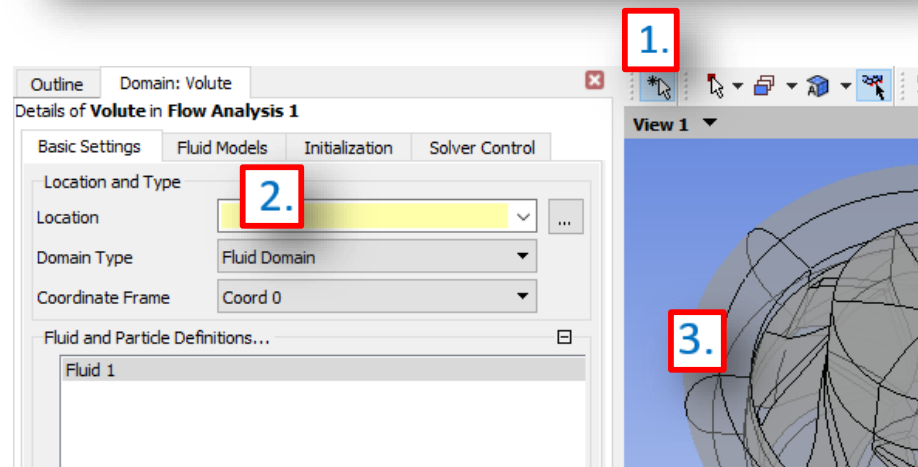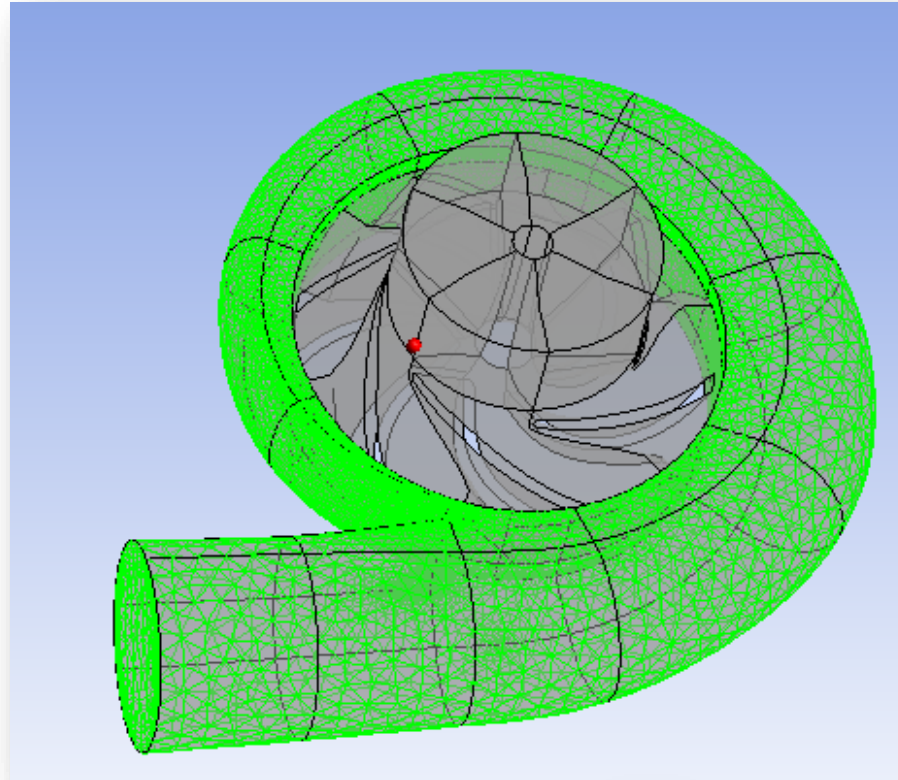   - *Close* the panel
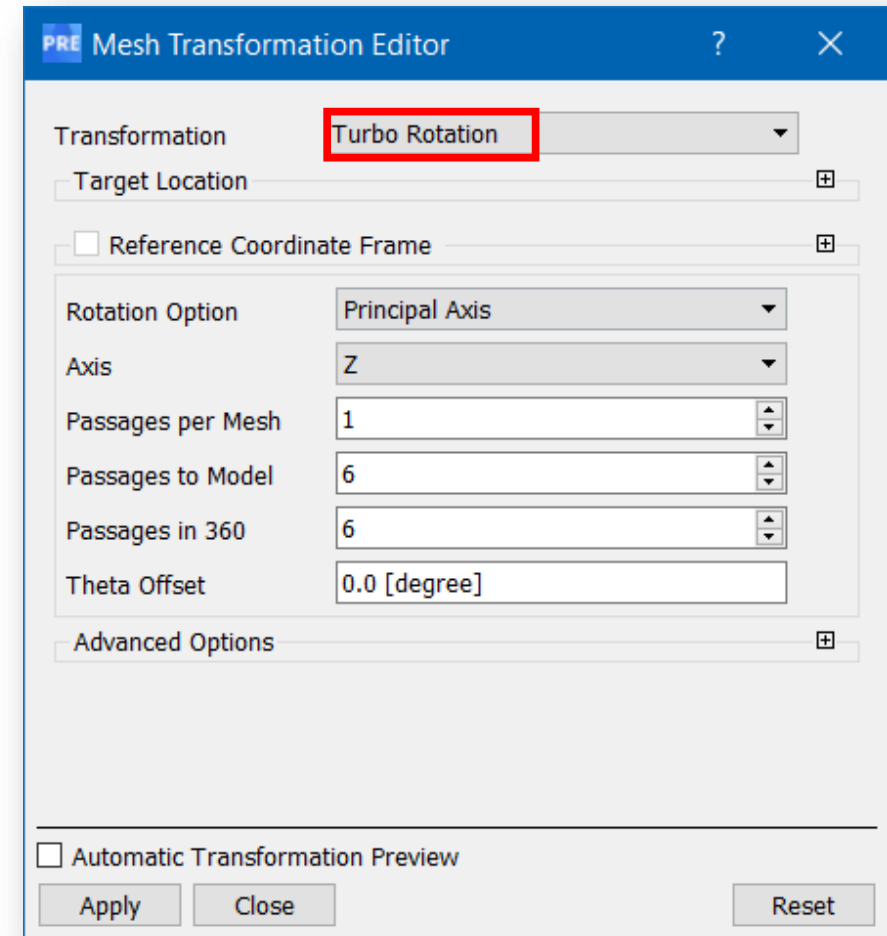
Setup

- Create a domain for the volute:
  - *Insert > Domain*
 - *Name: Volute*
  - *Location: B800* *
  - *Material: Water*

* Alternatively, you may pick the volute mesh location by:

1. Activating *Select*
2. Clicking in the box next to Location (it will become yellow)
3. Clicking on any the volute in the graphics window

Setup

- Create an expression for rotational speed:

- RMB on *Expressions* in the *Outline*
  - *Insert > Expression*
  - *Name* → *myRPM*
  - Enter *1450 [rev min^-1]* and *Apply*


© 2019 ANSYS, Inc.
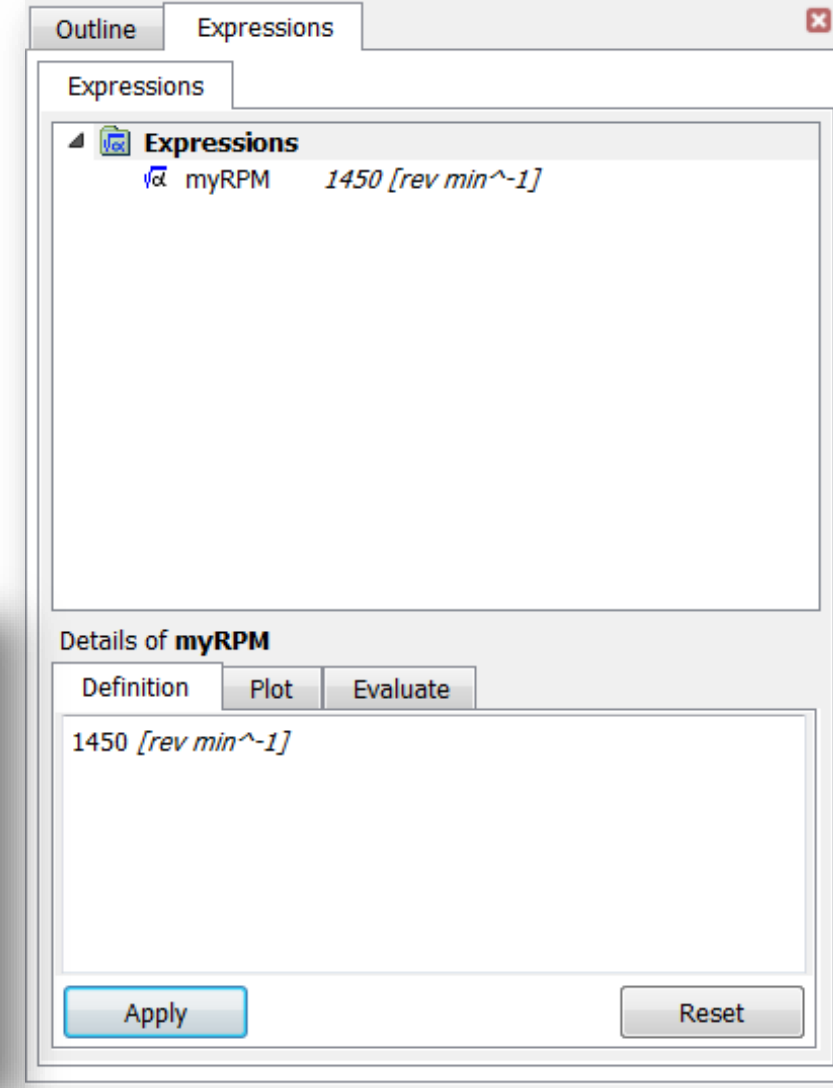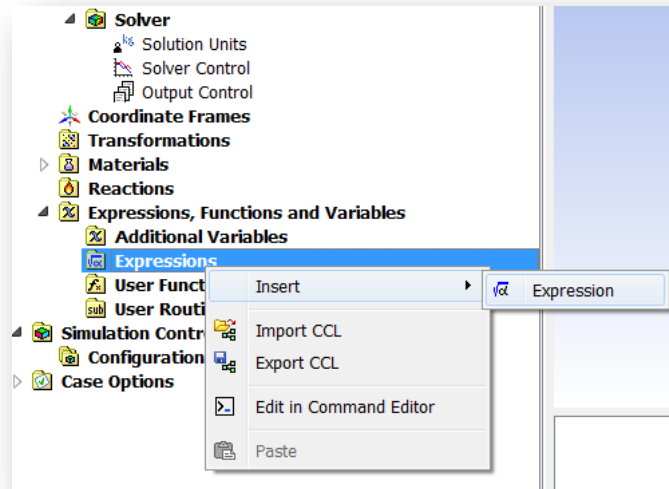
Setup

- Rename *Default Domain*:
  - Right click on *Default Domain*
  - Select *Rename*
  - Type *Impeller* and hit the *Return* key
 - Double click on the *Impeller Domain* in the *Outline* Tree to modify

© 2019 ANSYS, Inc.
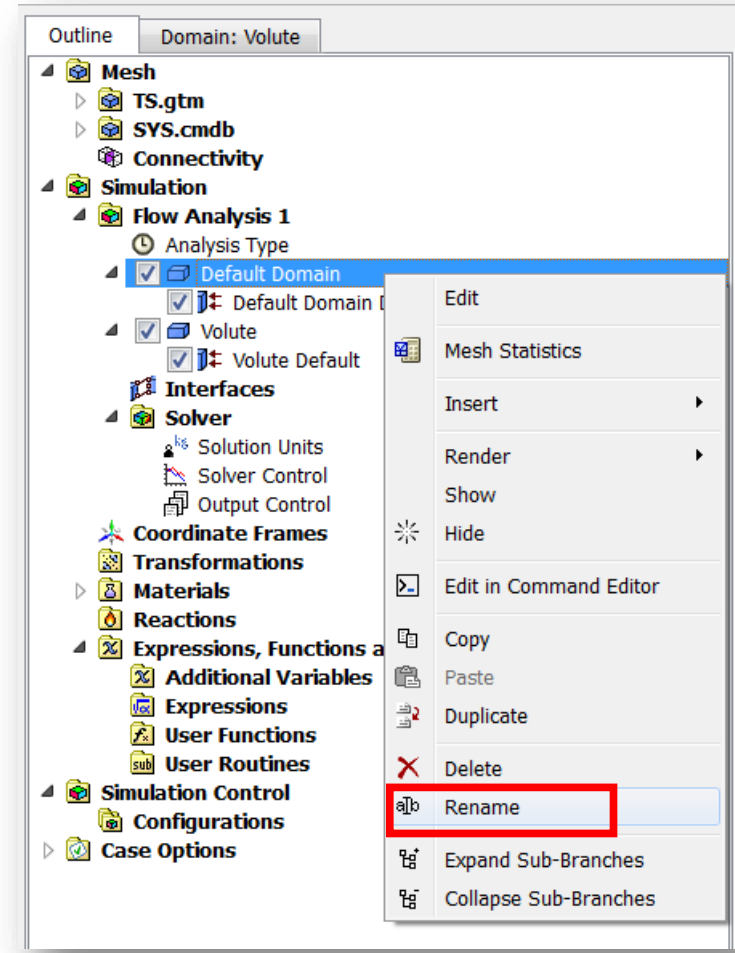
Setup

- *Impeller Domain*:
  - Set *Domain Motion* to *Rotating*
  - Set rotational speed to *myRPM*
    - ➢ Use *Enter Expression* button
  - *Reference Pressure* to 0 [atm]
  - Note that the *Material* is already set to *Water*
  - In the *Fluid Models* tab (not shown)
    - ➢ Set *Heat Transfer Option* to *None*
    - ➢ Select the *Shear Stress Transport* turbulence model
  - Click *OK* to close panel
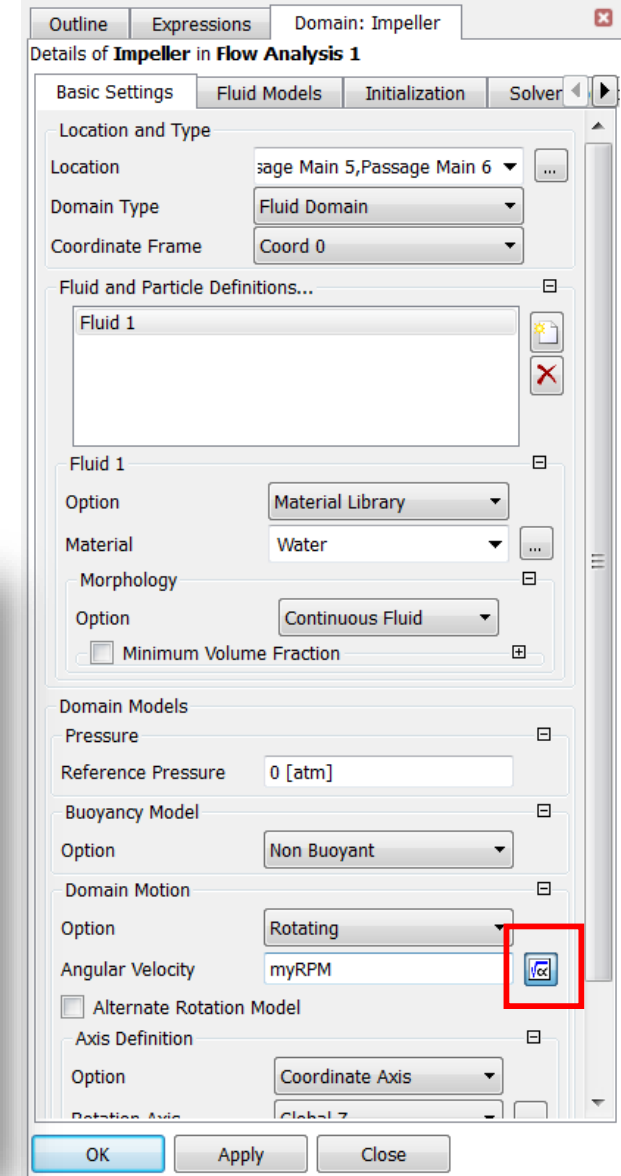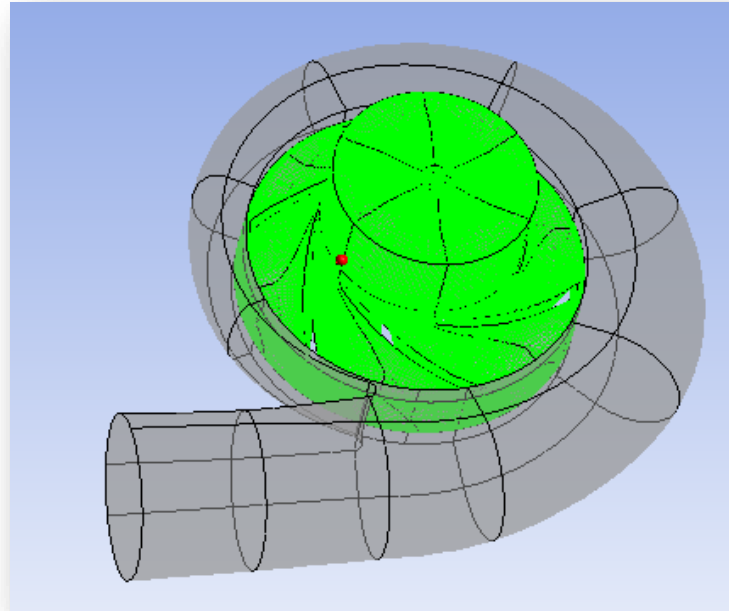
Setup

- Create a domain interface between the impeller and volute:
  - RMB on *Interfaces* in *Outline* Tree
  - *Insert > Domain Interface*
  - *Name*: *Impeller to Volute*
  - Settings: as shown on the right
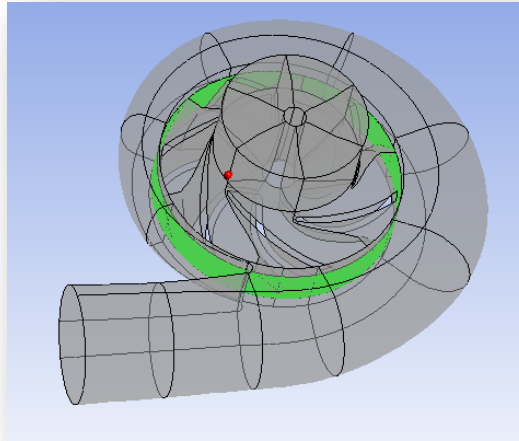
Always specify pitch angles when the interface spans the entire circumference
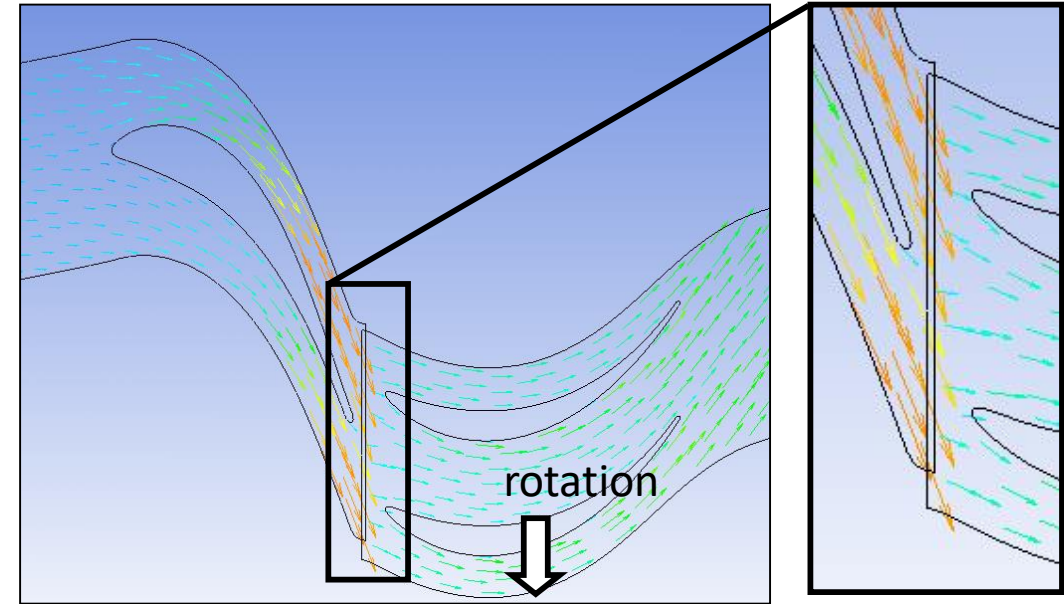
Setup

- Note that *CFX-Pre* has automatically created two separate regions that have the same name as the interface with the suffixes *Side 1* and *Side 2*
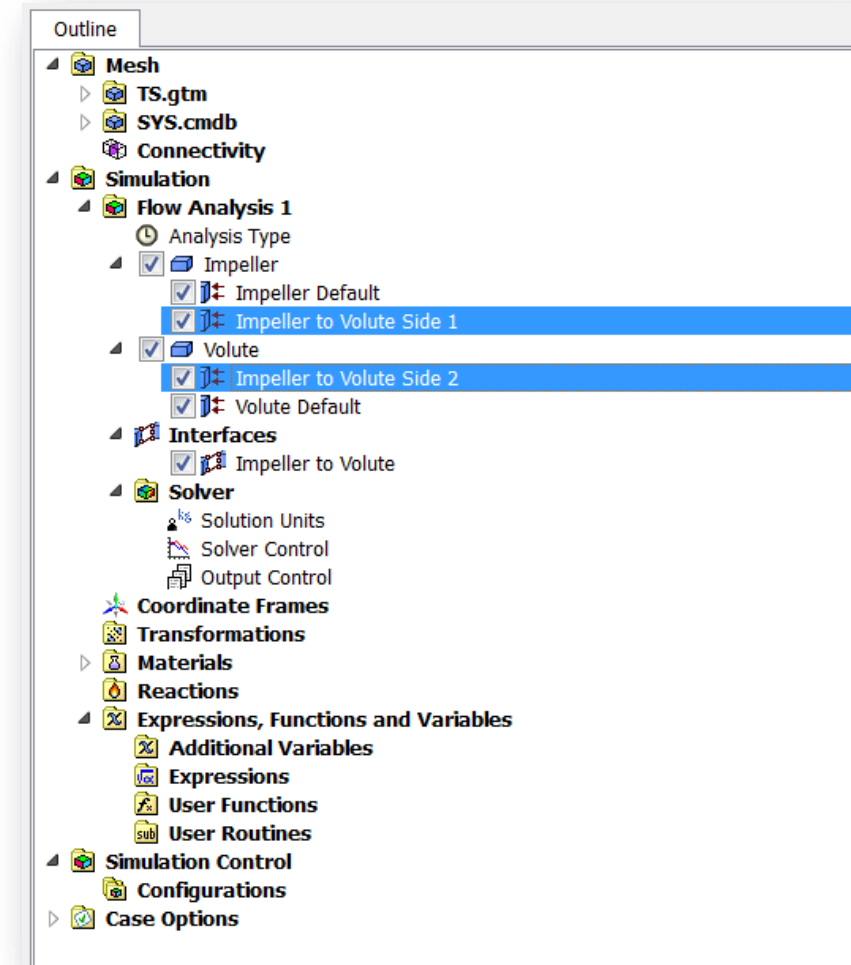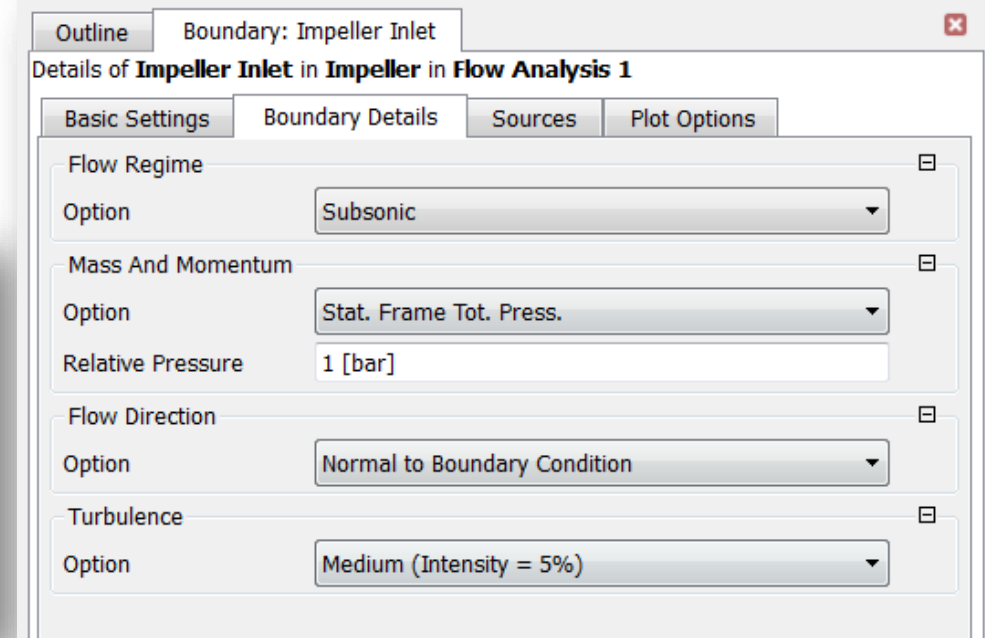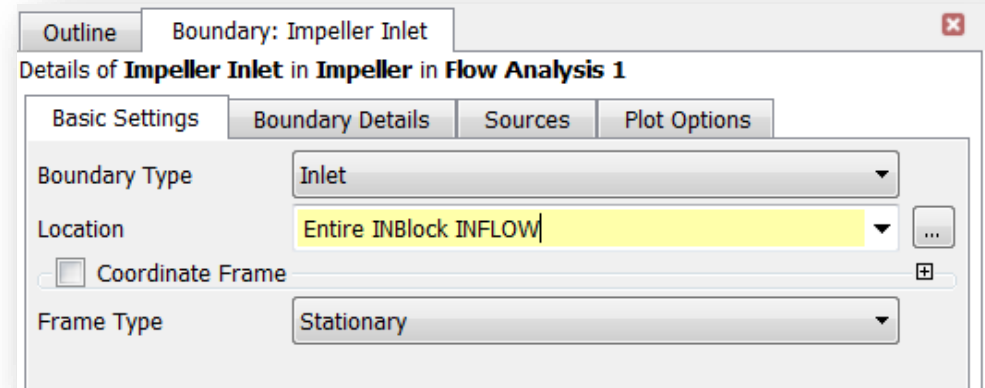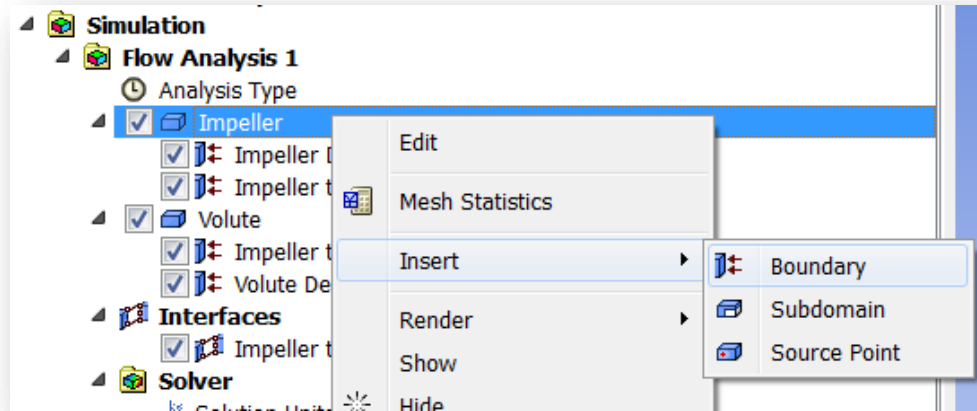
Setup

- Frozen Rotor Concept:
  - Components have a fixed relative position, but the appropriate frame transformation and pitch change is made
  - Most useful when the circumferential variation of the flow is large relative to the component pitch
  - Good approximation for a pump in a volute – each blade passage will observe a different back pressure based on the circumferential position within the volute

- Frozen Rotor Usage:
  - The quasi-steady approximation involved becomes small when the through flow speed is large relative to the machine speed at the interface
  - This model requires the least amount of computational effort of the three frame change models
  - Transient effects at the frame change interface are not modeled
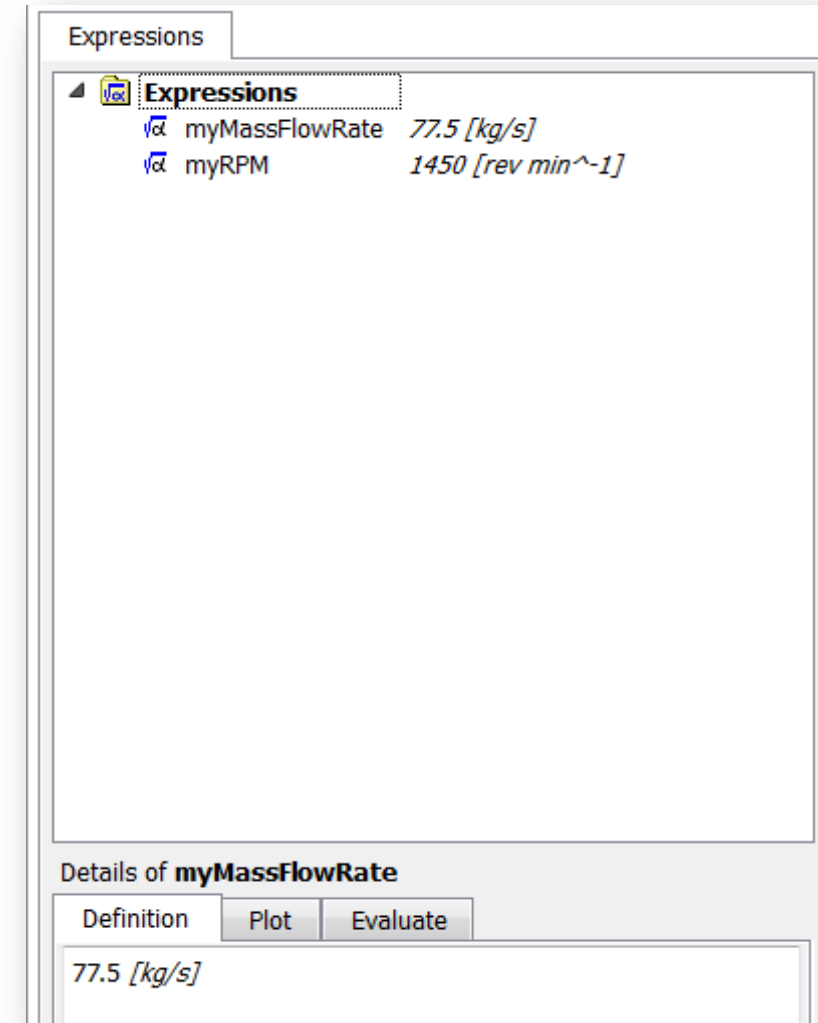

rotation

Setup

- Setup Inlet boundary condition:
  - RMB on *Impeller* and *Insert > Boundary*
  - *Name: Impeller Inlet*
  - *Location*: *Entire INBlock INFLOW*
  - *Mass and Momentum Option: Stat. Frame Tot. Press.*
  - *Relative Pressure* = 1 [bar]
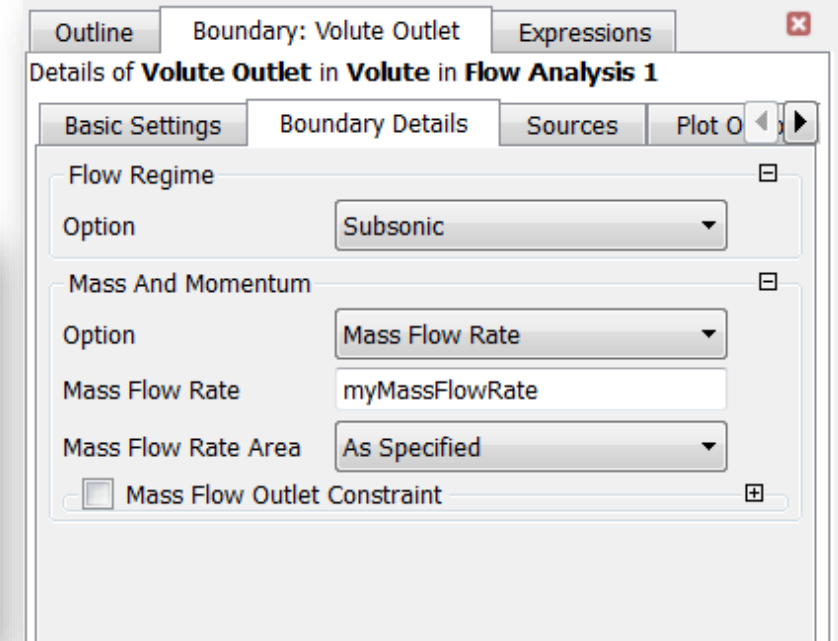 - Click *OK*
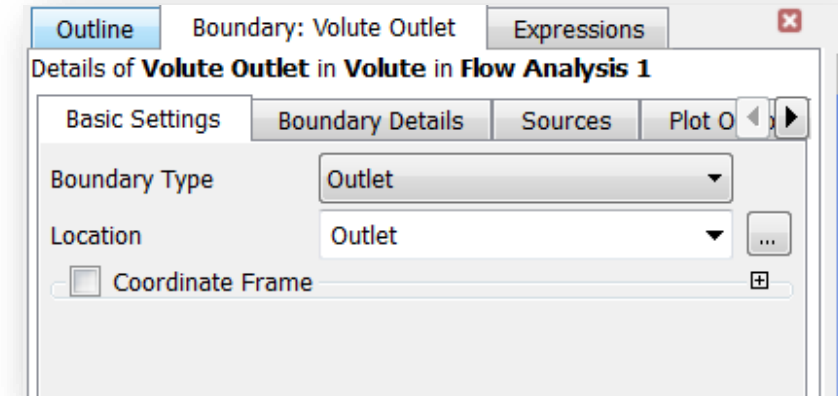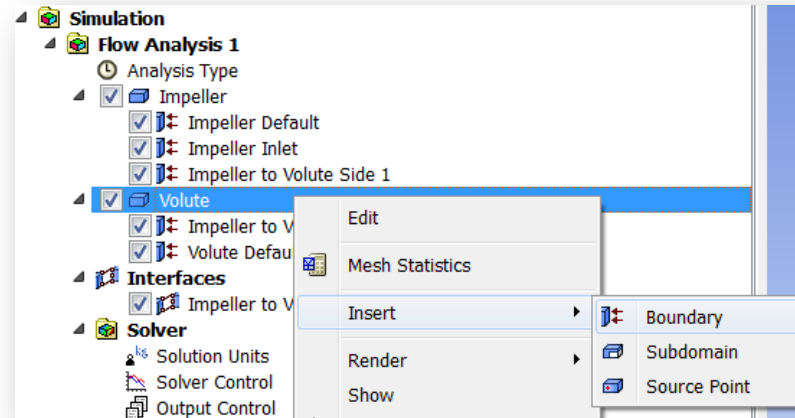
Setup

- Create an expression for mass flow rate
- RMB on *Expressions*
  - *Insert > Expression*
  - *Name: myMassFlowRate*
- Enter *77.5 [kg/s]* and *Apply*

Setup

- Setup Outlet boundary condition:
  - RMB on *Volute* and *Insert > Boundary*
  - *Name*: *Volute Outlet*
  - *Boundary Type*: *Outlet*
  - *Location*: *Outlet*
  - *Mass Flow Rate*: *myMassFlowRate*

© 2019 ANSYS, Inc.

Setup

- Create separate wall regions for:
  - *Impeller Hub*
    - ➤ *Regions*: *Entire INBlock HUB, Entire Passage HUB*
  - *Impeller Shroud*
    - ➤ *Regions*: *Entire INBlock SHROUD, Entire Passage SHROUD*
  - *Impeller Blades*
    - ➤ *Region*: *Entire BLADE*

© 2019 ANSYS, Inc.
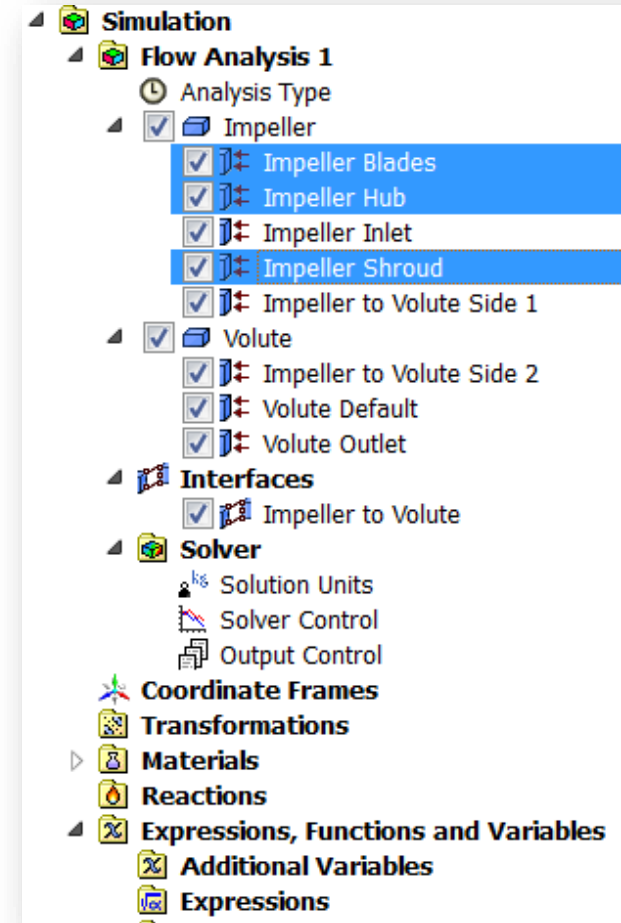
Setup

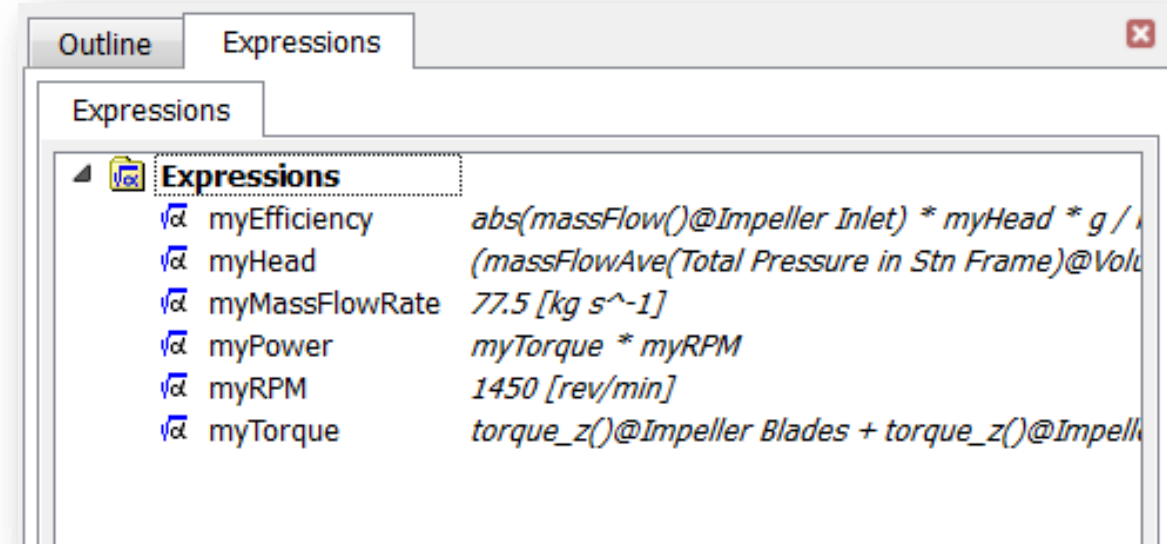- Create Additional Expressions that will be used for monitor points:
  - *myHead*
  - *myTorque*
  - *myPower*
  - *myEfficiency*

myEfficiency = abs(massFlow()@Impeller Inlet * myHead * g / myPower)

myHead = (massFlowAve(Total Pressure in Stn Frame)@Volute Outlet - massFlowAve(Total Pressure in Stn Frame)@Impeller Inlet)/ (g * ave(Density)@Impeller Inlet)

myPower = myTorque * myRPM

myTorque = torque_z()@Impeller Blades + torque_z()@Impeller Hub + torque_z()@Impeller Shroud

Setup

- Create Monitor Point for Torque
  - Double click *Output Control* in the *Outline* Tree
  - Switch to the *Monitor* Tab
  - Check the *Monitor Objects* checkbox
  - Click the Add New Item icon
  - Name the Monitor *Torque*
  - Change *Option* to *Expression*
  - Enter *myTorque* as the *Expression Value*
  - Click *Apply*

© 2019 ANSYS, Inc.
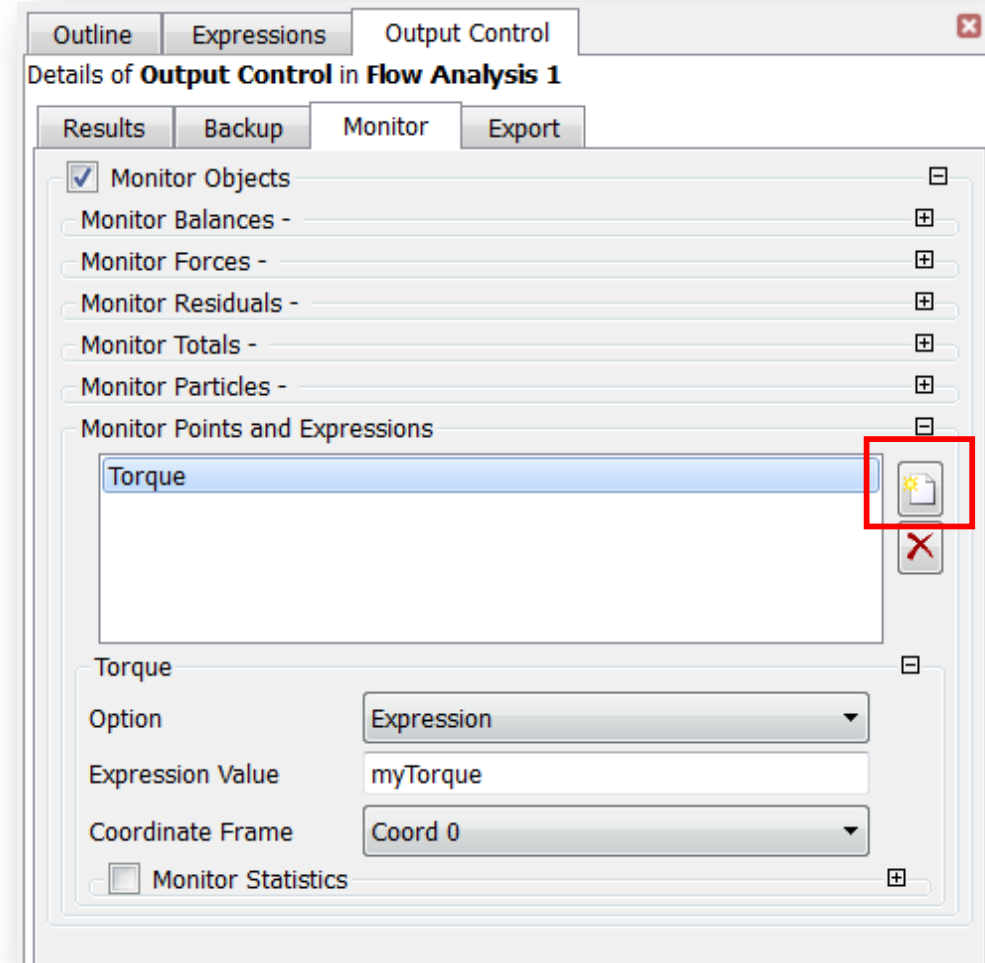
Setup

- Create monitor points in the same way for *Efficiency, Head* and *Power*

© 2019 ANSYS, Inc.
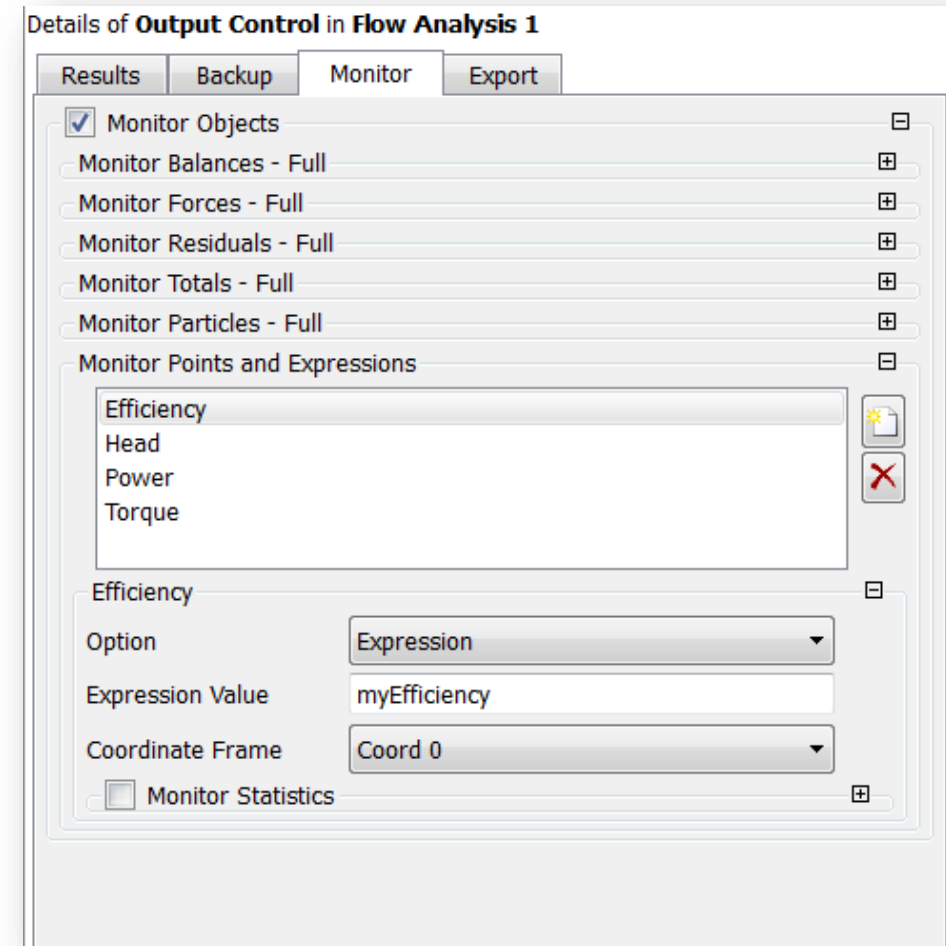
Setup

- Modify Solver Control
  - A best practice is to set the *Timescale* to be equal to 1/omega
    - Change *Timescale Control* to *Physical Timescale*
    - Set *Physical Timescale* to *1[rad]/myRPM*
  - Set the *Max. Iterations* to 300
  - Set the *RMS Residual Target* to 1e-5
  - Click *Apply*

© 2019 ANSYS, Inc.

Solver

- Close *CFX-Pre* and go back to the Workbench *Project Schematic*
- Save the *Workbench Project*
- Start the *CFX-Solver*
  - Double Click on the *Solution* cell *C3*
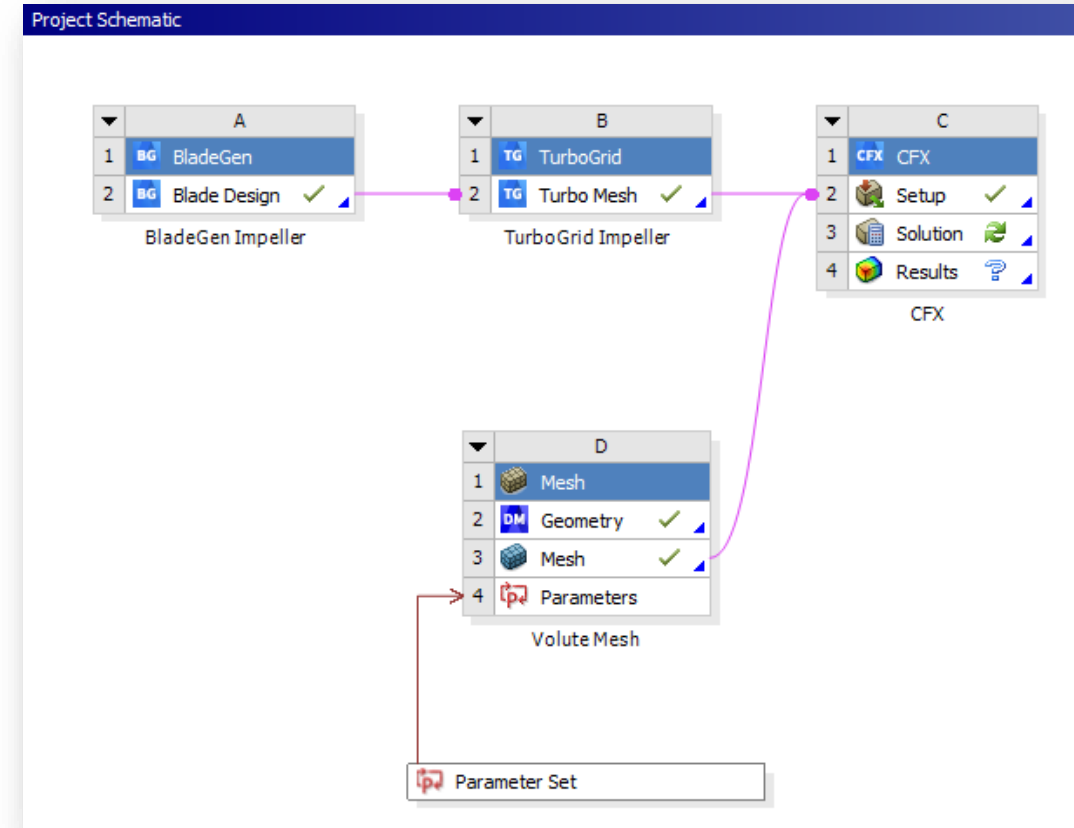
Solver

- On the *Run Definition* Tab, set the *Run Mode* to *Intel MPI Local Parallel* and set the number of *Partitions* to 4
  - Check the *Show Advanced Controls* checkbox
  - In the *Partitioner* and *Solver* tabs set *Memory Alloc Factor* to 1.2 (this is shown below for the *Solver* tab)
  - Click *Start Run*
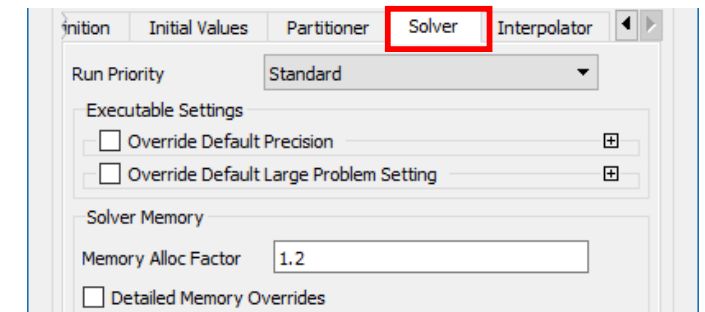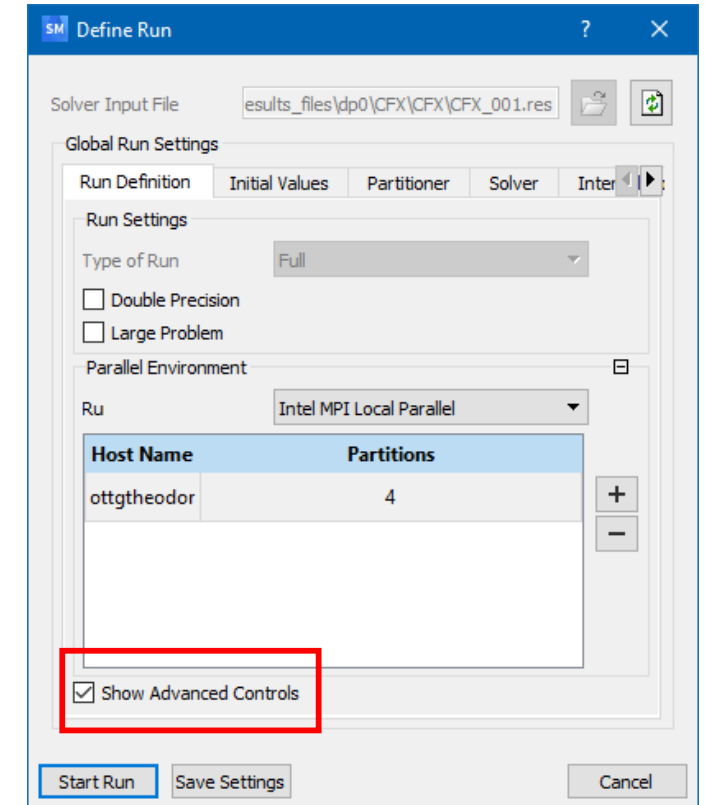
Solver

- Memory Allocation Factor:
    - CFX-Solver is written in Fortran
    - Requires memory to be allocated prior to run
    - Occasionally CFX underestimates the amount of memory required to run the simulation
   - In such cases the CFX-Solver ends up with an error related to not enough memory allocated: *** INSUFFICIENT MEMORY ALLOCATED ***
    - This could happen in the *Partitioner, Interoplator* or *Solver* tasks
    - When this happens, select the corresponding (*Partitioner, Interoplator* or *Solver*) Tab in the *Solver Manager* and set the *Memory Alloc Factor* somewhere in the range of 1.2 to 2 as shown in the previous slide for the *Solver.*
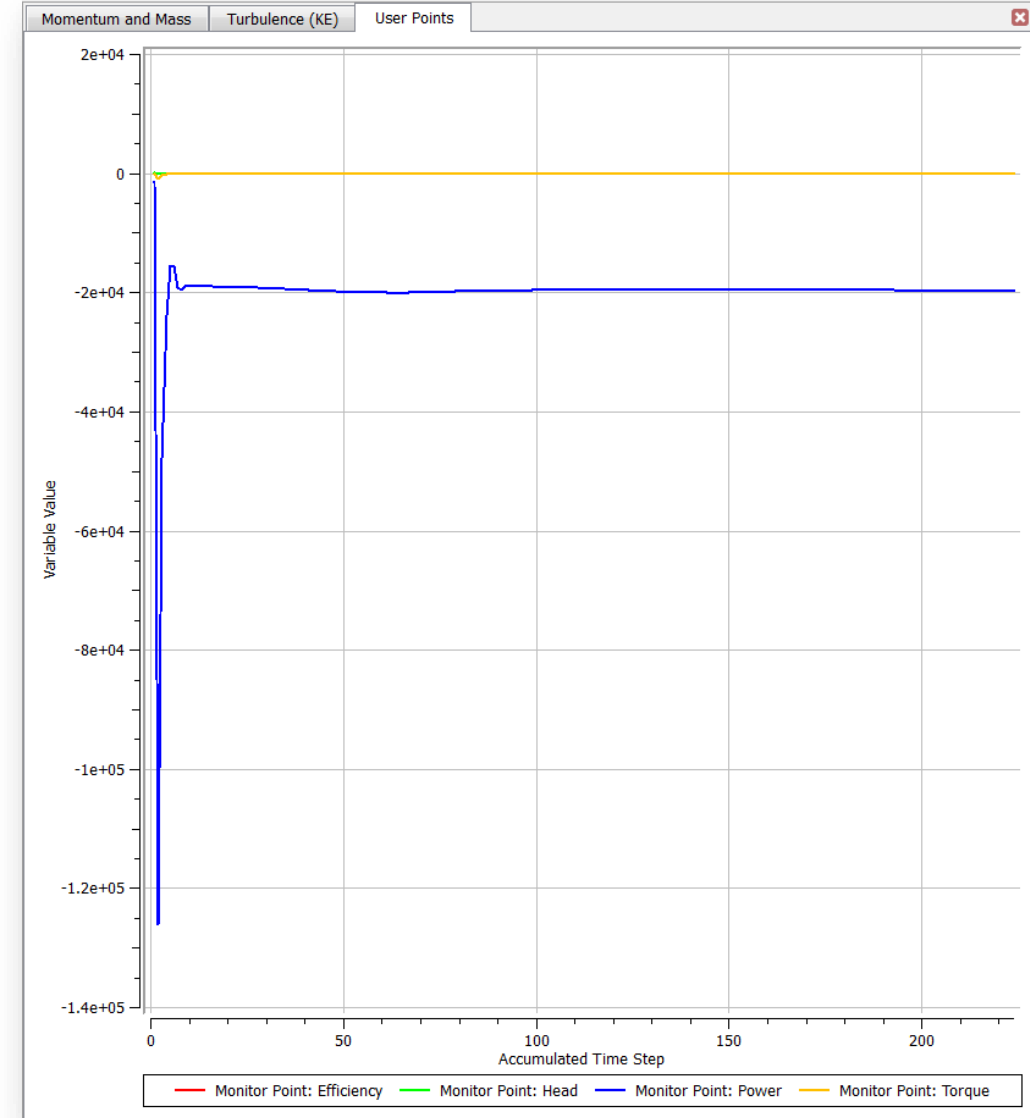
Solver

- This simulation will take roughly 40 minutes to run on 4 cores.
- To save time, you can open the Workbench project named: PumpVolute_solution.wbpz
  - This project includes a completed simulation


© 2019 ANSYS, Inc.
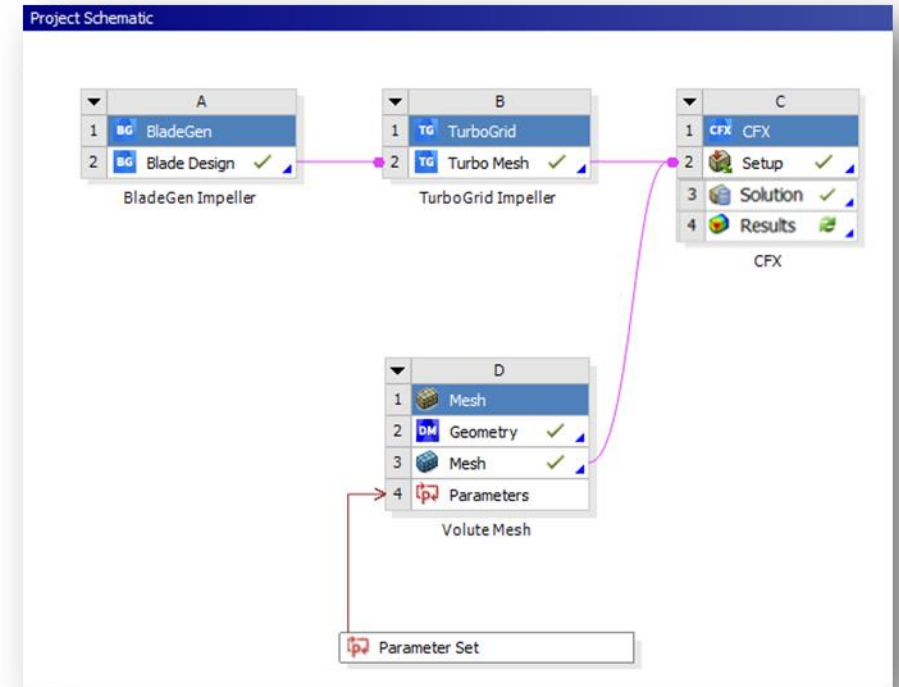
Solver

- If the *Solver Manager* is not already opened, open it by right mouse clicking on the *Solution* cell and selecting *Display Monitors*

  - Switch to the *User Points* Tab

  - We can see monitors for *Efficiency, Head, Power* and *Torque* that were created

  - Since the scale of these values is so different, let's view efficiency independently
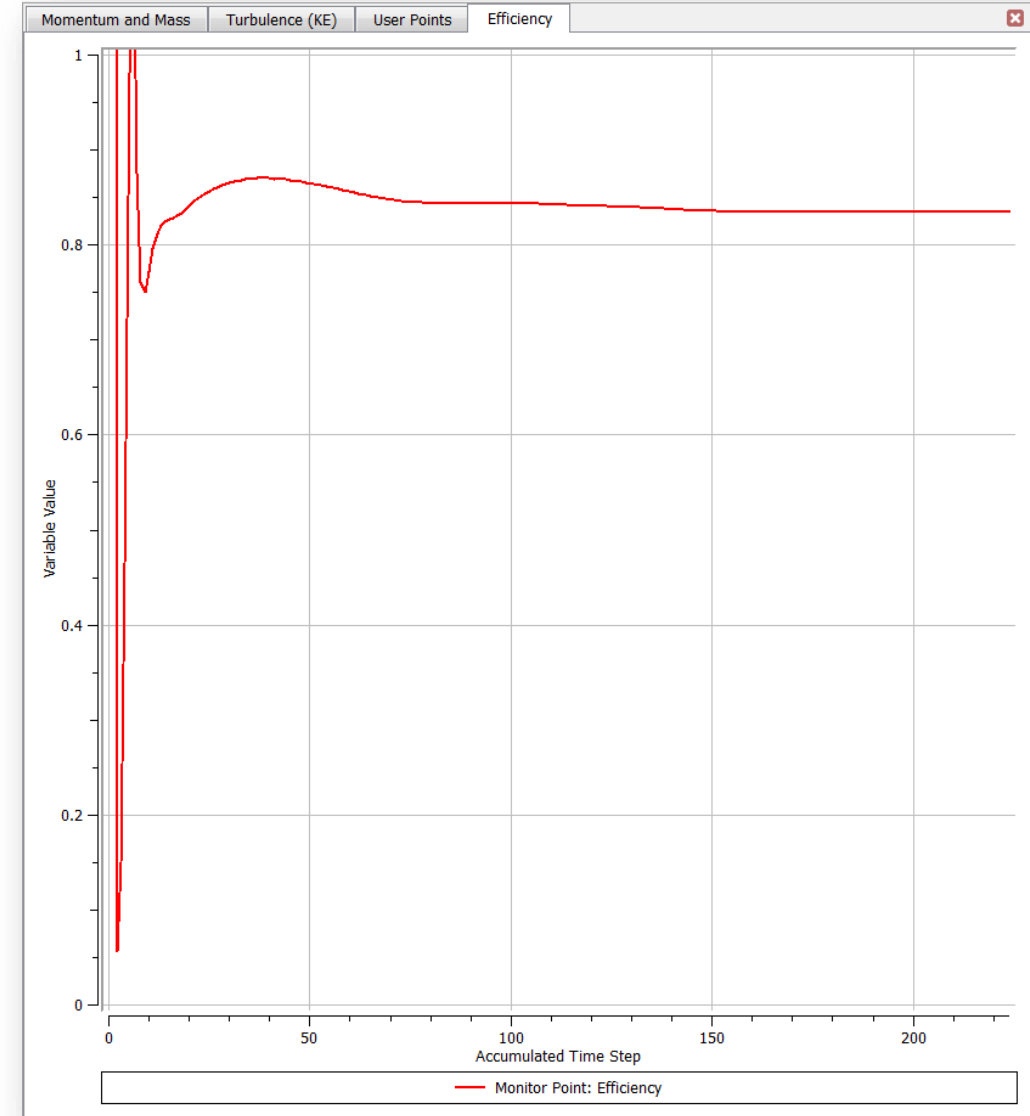

© 2019 ANSYS, Inc.

Solver

- *Workspace > New Monitor*

- *Name*: *Efficiency*

- Under *Plot Lines* Tab, navigate to *User Point Efficiency*

- Under *Range Settings* Tab:
  - Set *Manual Scale (Linear)*
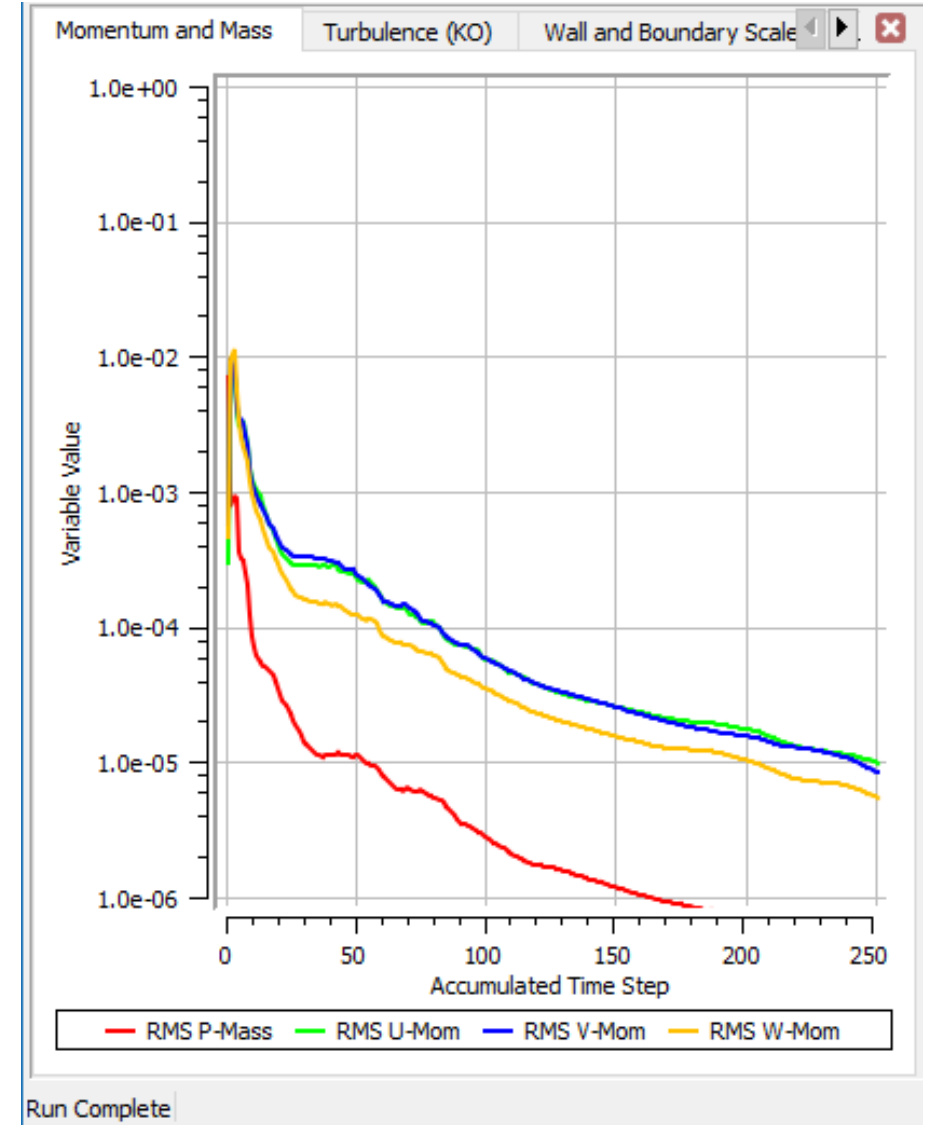 - *Lower Bound* = 0
  - *Upper Bound* = 1

- Click *OK*
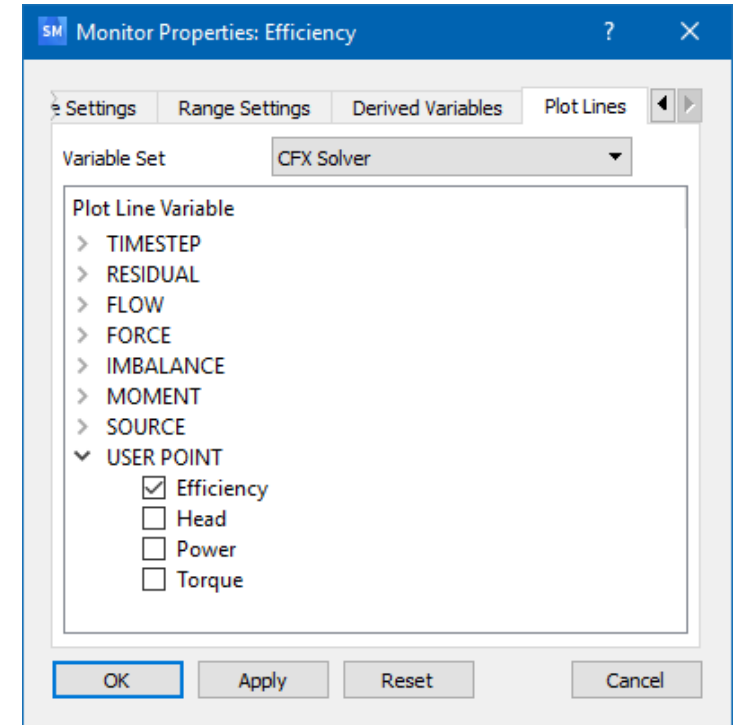
Solver

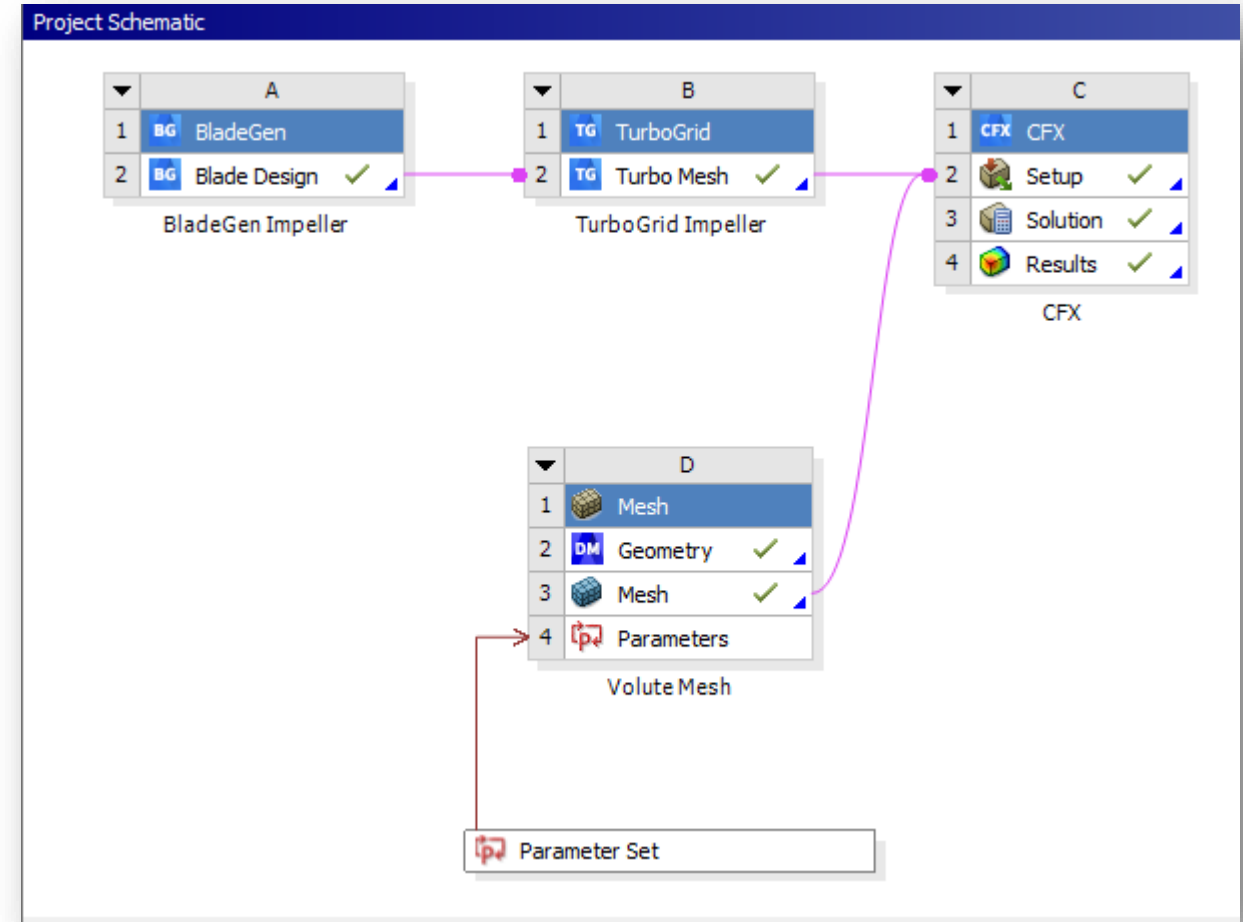- We can see that he efficiency has converged to approximately 87%

© 2019 ANSYS, Inc.

Solver

- Switching to the *Momentum and Mass* Tab, we can see that the residuals are all below 1.0e-5

- This case is well converged and we can now examine the results in *CFD-Post*

© 2019 ANSYS, Inc.

Post-Processing

- Double click on the *Results* Tab in Workbench to launch *CFD-Post*

Post-Processing

- We will initialize only the *Impeller* component on the *Turbo* Tab
  - Click on the *Turbo* Tab
  - If prompted to initialize all components, select no
  - Right click on *Component 1 (Impeller)* and select *Initialize*
  - You should receive a message stating *Failed to get outlines for turbo regions…*


© 2019 ANSYS, Inc.
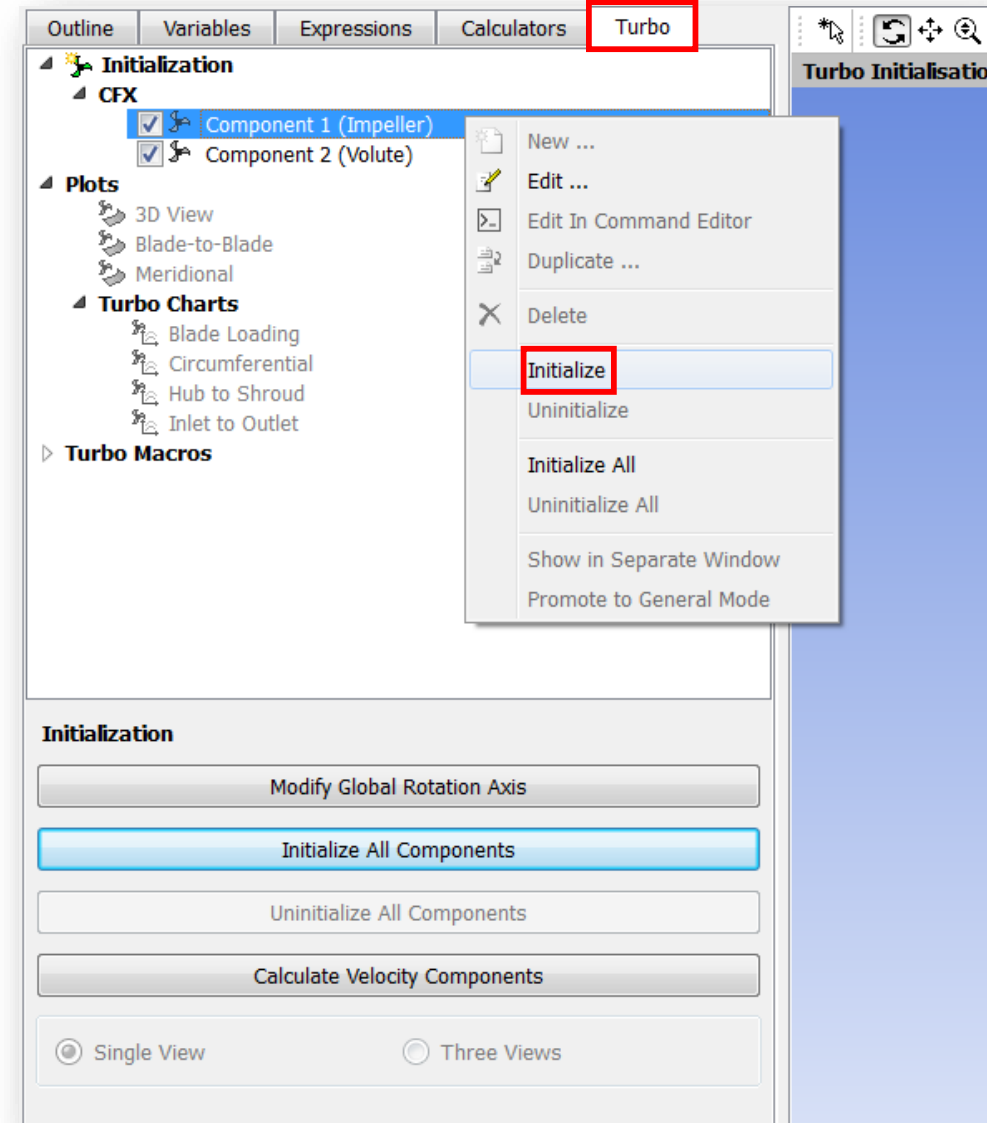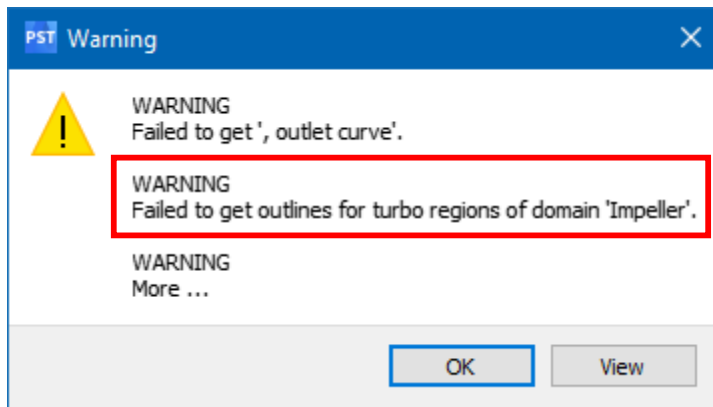
Post-Processing

- To resolve the error, double click on *Component 1*
  - Check the checkbox for *360 Case without Periodics* and click *Initialize*
  - The case should be properly initialized, indicated by the green background mesh
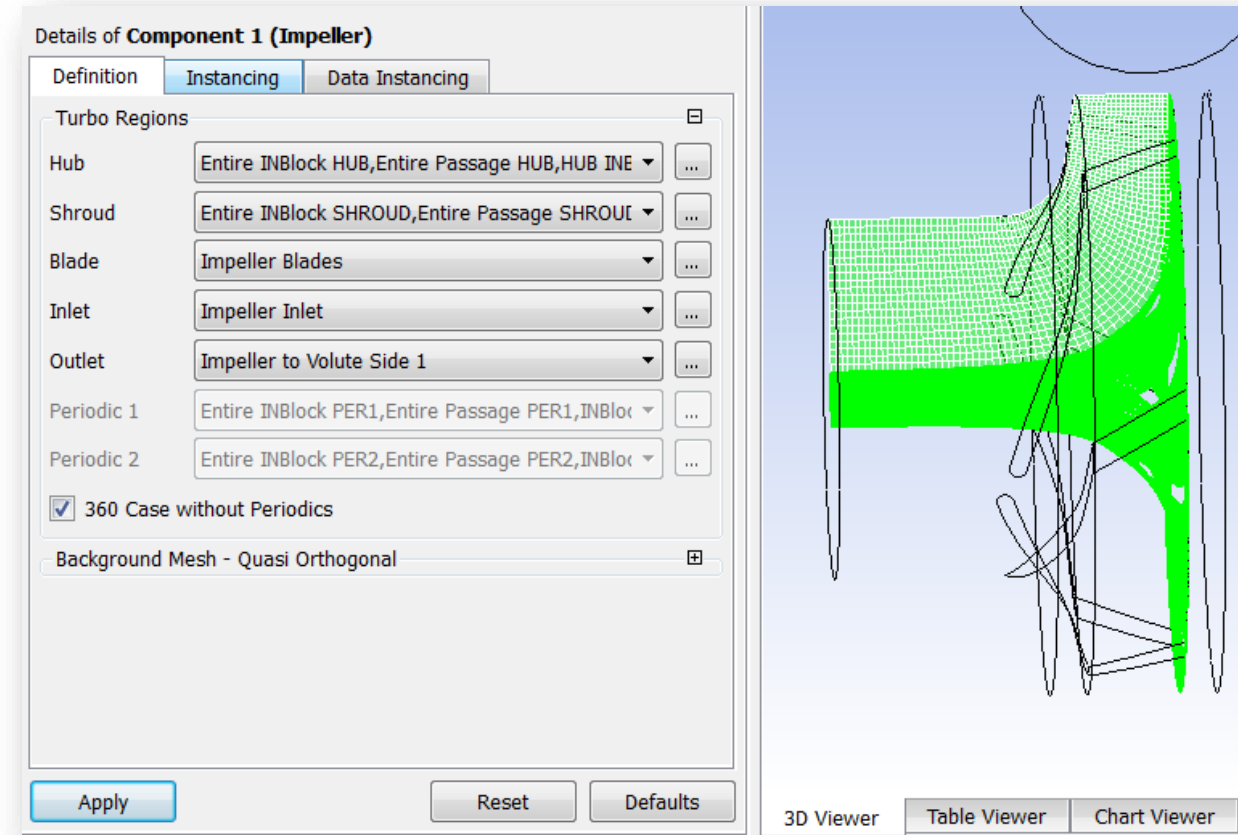
Post-Processing

- Back on the *Outline* Tab, we will create a *Turbosurface* at 50% span

- In the toolbar, from the *Location* drop down select *Turbo Surface*

- In the *details of Turbo Surface 1*
  - *Constant Span*, 0.5
  - *Apply*
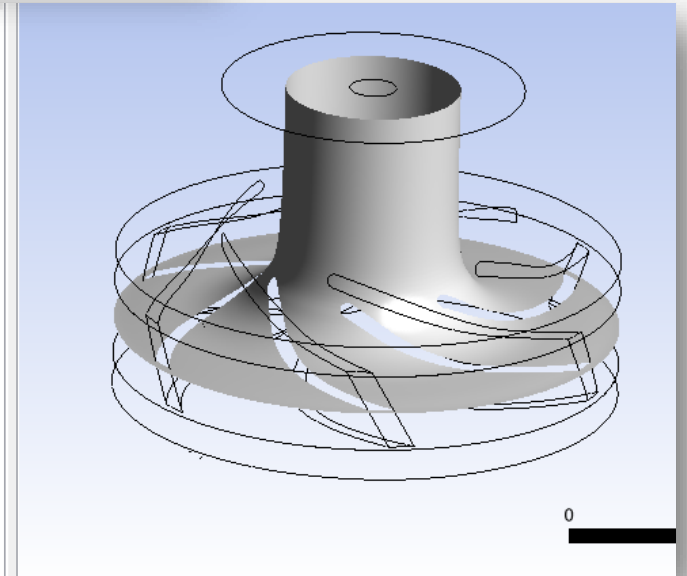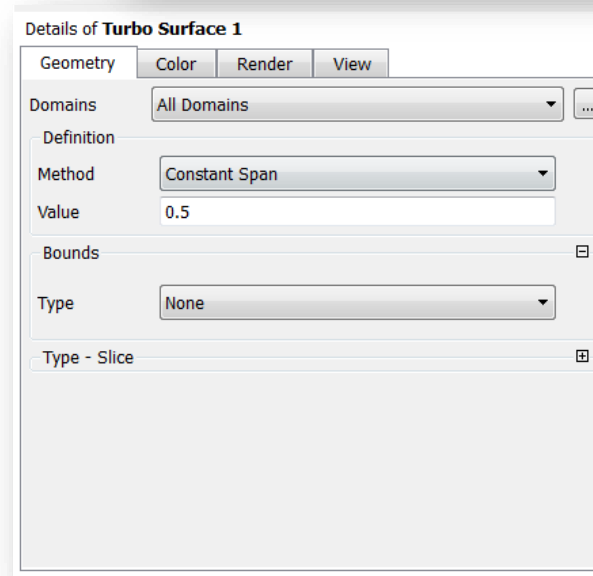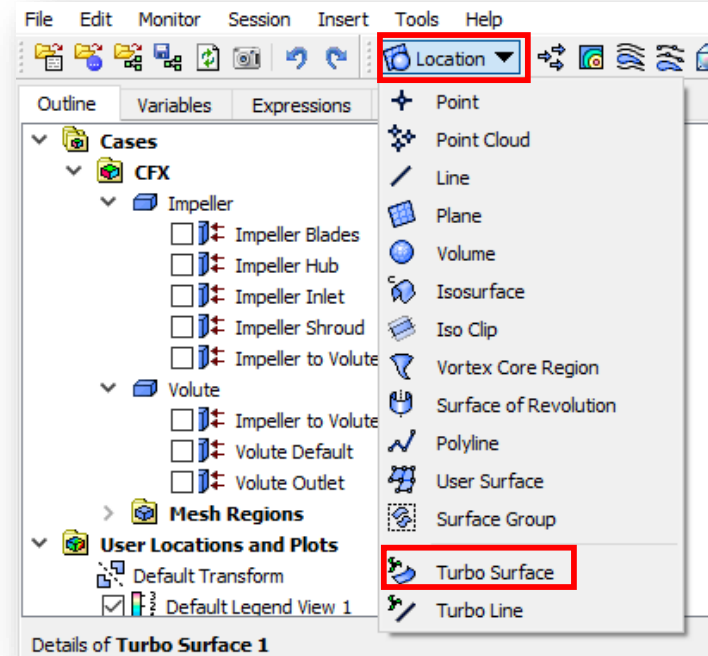
Post-Processing

- We will make a plane that is tangent to this in the volute domain

- In the toolbar, from the *Location* drop down select *Plane*

- In the *details of Turbo Surface 1*
  - *Domains: Volute*
  - *Method: XY Plane*
  - For *Z*, enter expression: *maxVal(z)@Turbo Surface 1*
  - *Apply*
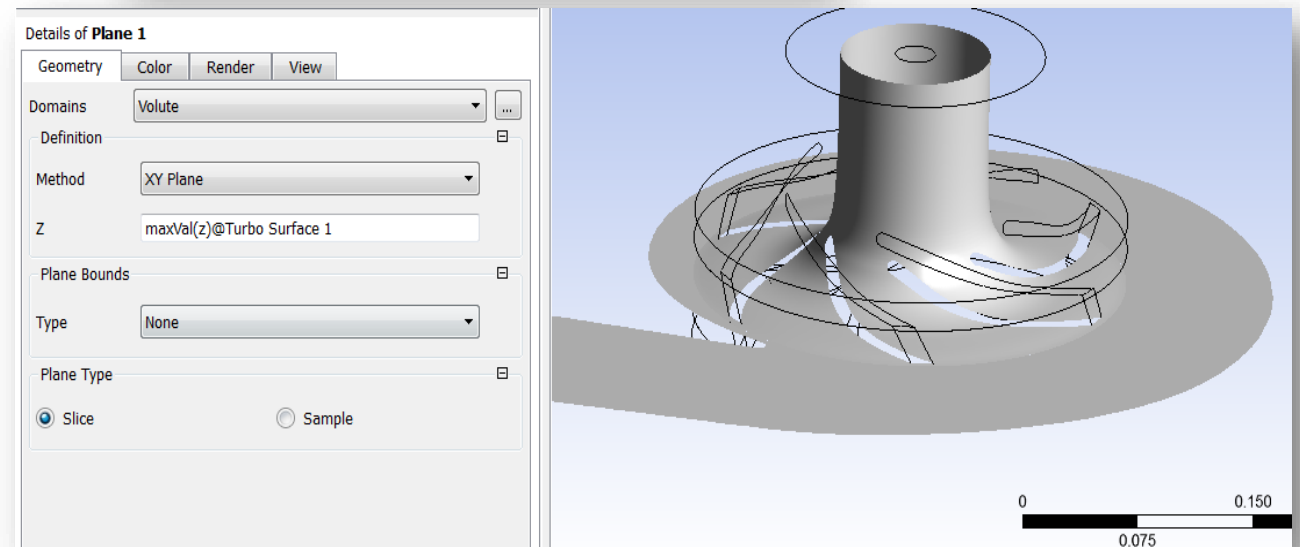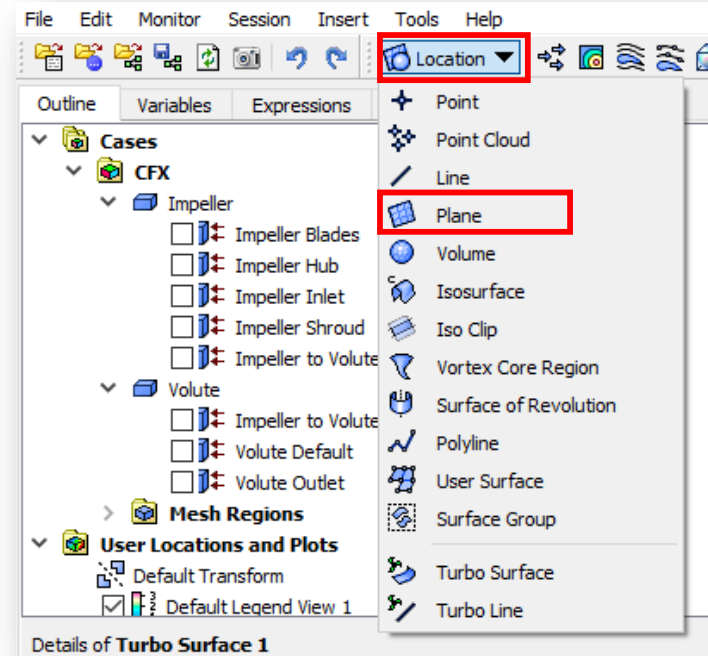
Post-Processing

- Make a contour plot of pressure on both Plane 1 and the Turbo Surface
  - Turn off the visibility of *Plane 1* and *TurboSurface 1*
  - *Insert > Contour*, accept default name
  - For *Locations*, use the three dots icon … to select both *Plane 1* and *TurboSurface 1*
  - Set the *Range* to *Local*
  - *Apply*
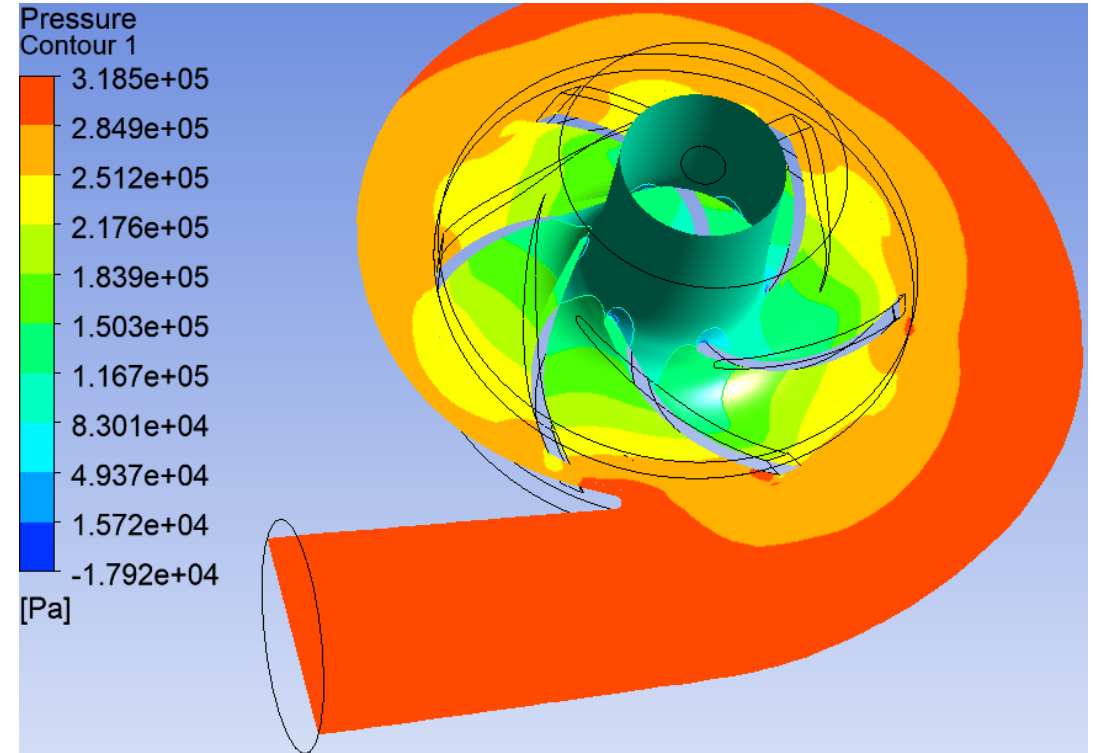
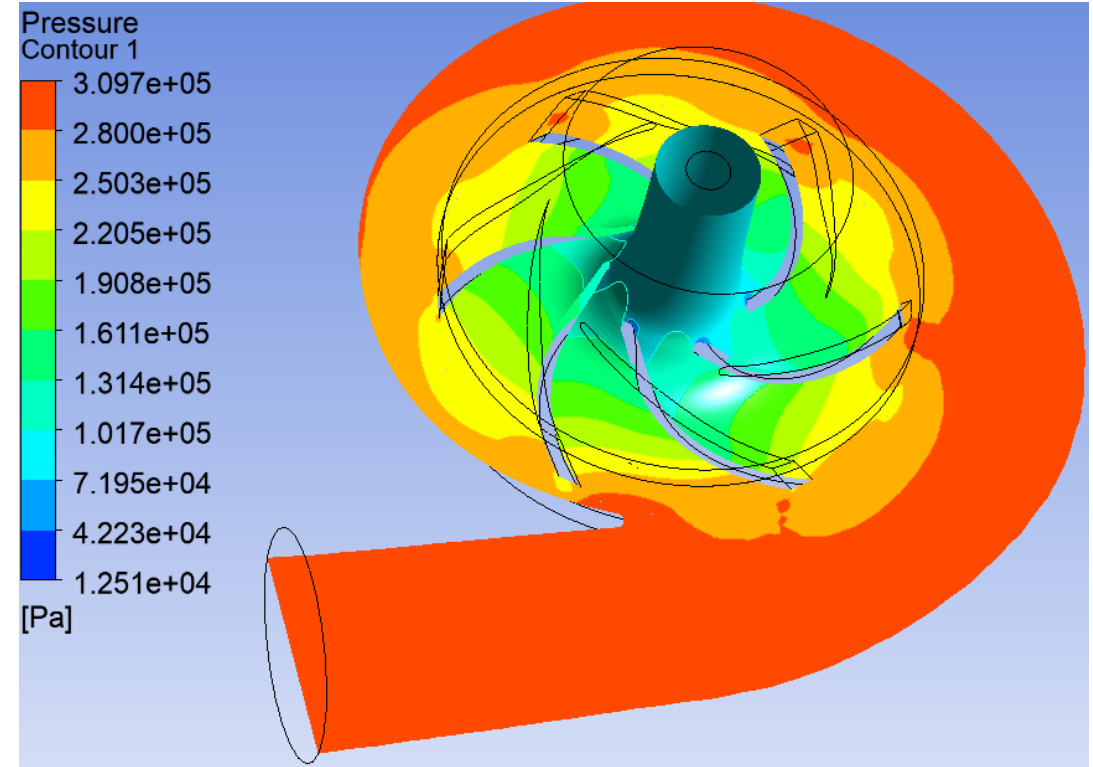# Post-Processing

- Modify *Turbo Surface 1* to visualize at 25% span
  - Double click *Turbo Surface 1*
  - Set *Value* to 0.25
  - *Apply*
  - Note that the *Plane 1* also updates position
- Note the flow non-uniformity downstream of the blades due to the use of a frozen rotor interface in this case

Post-Processing

- Make a table for the Expressions created:
  - Go to the *Table Viewer* Tab
  - Select *New Table* from the toolbar
  - In cell *A1*, enter the text *Head*
  - In cell *B1*, enter *=myHead* (or alternatively after entering "=" *RMB > Expressions* and selected expression *myHead*
  - In the same way, enter the texts as shown in column A and their corresponding expressions (created on slides 14, 21 and 24) in column B for Mass Flow Rate, Rotation Speed, Torque, Efficiency and Shaft Power
    - ➢ Note: In column B multiply the expression *myEfficiency* by *1 [rad]* and divide *myPower* by *1[rad]* for proper units
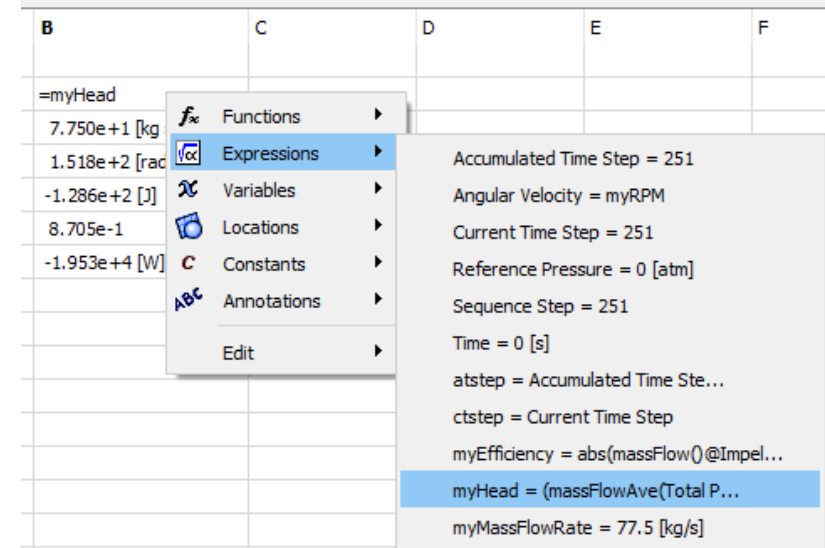

© 2019 ANSYS, Inc.

Post-Processing

- On your own:

- Create a *Vector* plot on *Plane 1* and *Turbo Surface 1*, and try plotting both *Velocity* and *Velocity in Stn Frame*

- On the *Turbo Tab > Initialization,* click *Calculate Velocity components*. Then try to calculate the average flow angle at the blade leading edge and trailing edge

Summary

- In this workshop, you have learned the following:
  - Combining meshes for different components in CFX Pre
  - Duplicating single passage meshes for full 360
  - Setting up multiple frames of reference with a domain interface
  - Setting up monitor points for quantities of interest
  - Determining convergence of a CFX solution
  - Post processing a mix of rotating and stationary components