ANSYS Fluent Rotating Machinery Modeling

# Workshop 02.2: Radial Compressor

Release 2020 R2

Ansys

Introduction

- Workshop Description:
  - This Workshop deals with the Fluent setup and solution for a radial compressor impeller

- Learning Aims:
  - Setting up a single rotating component
    - Defining a rotating frame
    - Applying rotational periodicity
    - Using Exit Mass Flow Correction as outlet boundary condition
    - Creating input and output Workbench parameters
  - Solving and monitoring convergence
    - Creating Named Expressions
  - Visualizing the pressure distribution on the impeller walls
  - Creating a speedline using parameters and design points in Workbench
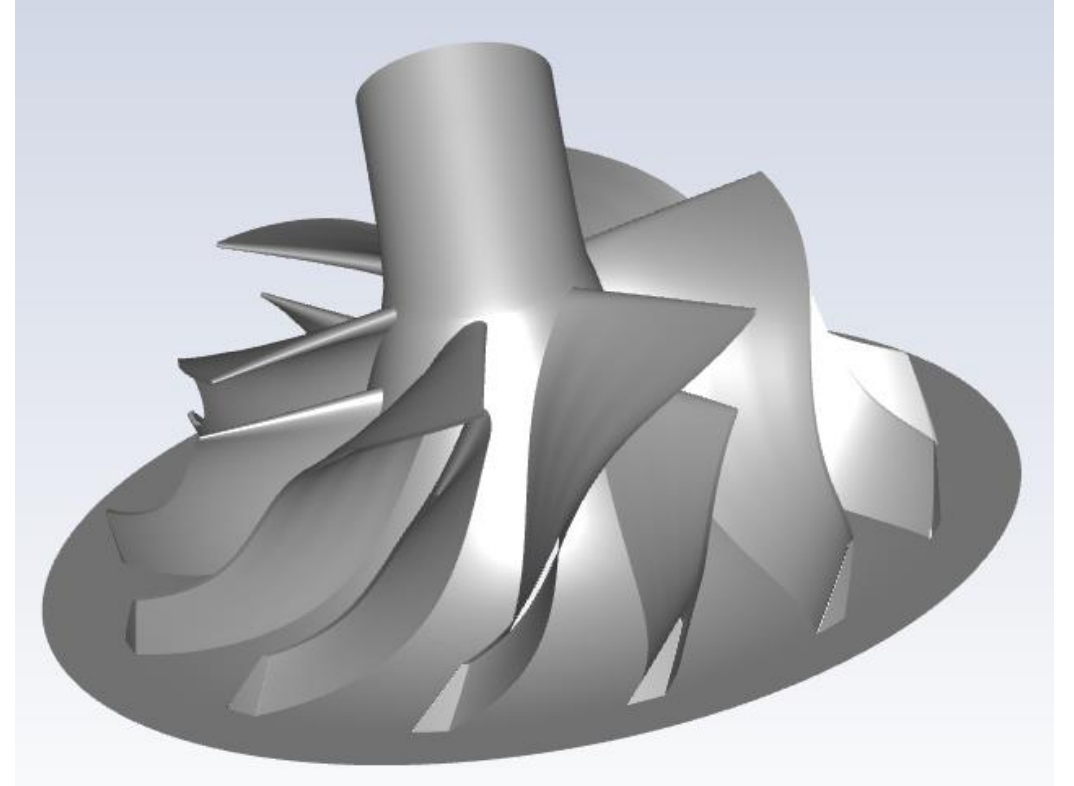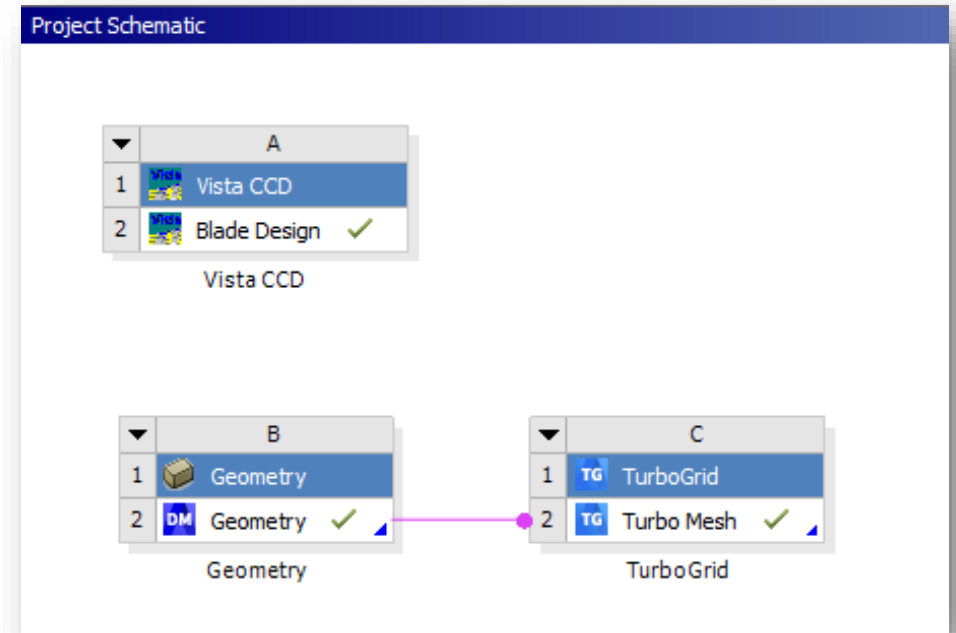
Pump Model

- Single rotating component
  - A moving reference frame is used to solve the rotating component
  - Due to rotational periodicity, we can reduce the problem size by modelling a single blade passage with periodic boundaries

- Compressor data
  - Fluid = Air Ideal Gas
  - Speed = 155733 rpm
  - Number of main blades = 6
 - Number of splitter blades = 6
  - Axis or rotation = z-axis
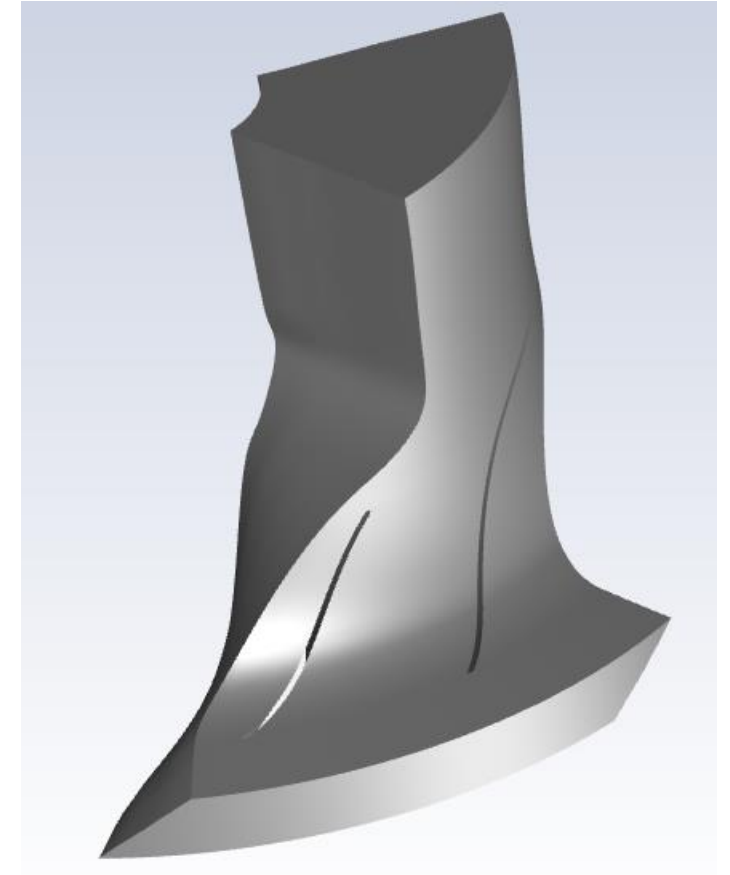
Load Workbench Project

- The geometry has been created using Vista CCD and the mesh has been created using TurboGrid

- The geometry and mesh for the radial compressor are provided in a Workbench archive

- Open Workbench
  - In the Workbench main menu *File > Open…*
    - In the *Open* dialogue box *Browse to TurbochargerCompressorFluentStartingPoint.wbpz* provided with the workshop inputs and click *Open*
    - In the *Save As* dialogue box edit the *File Name* to *TurbochargerCompressor.wbpj* and click *Save*

Project Schematic

	A
1	Vista CCD
2	Blade Design ✓

Vista CCD

	B
1	Geometry
2	Geometry ✓

Geometry

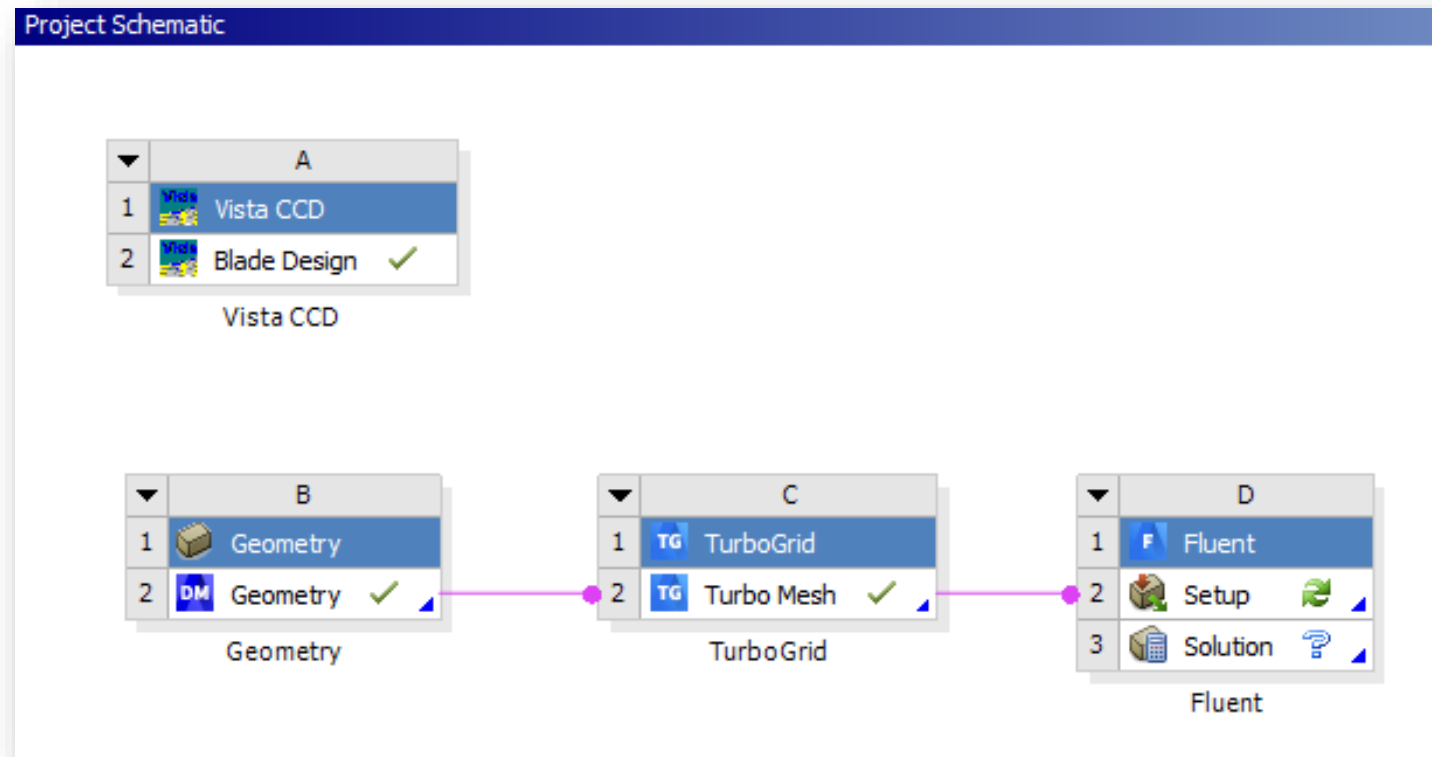|	C
1	TurboGrid
2	Turbo Mesh ✓

TurboGrid

Workbench: Create a Fluent Component

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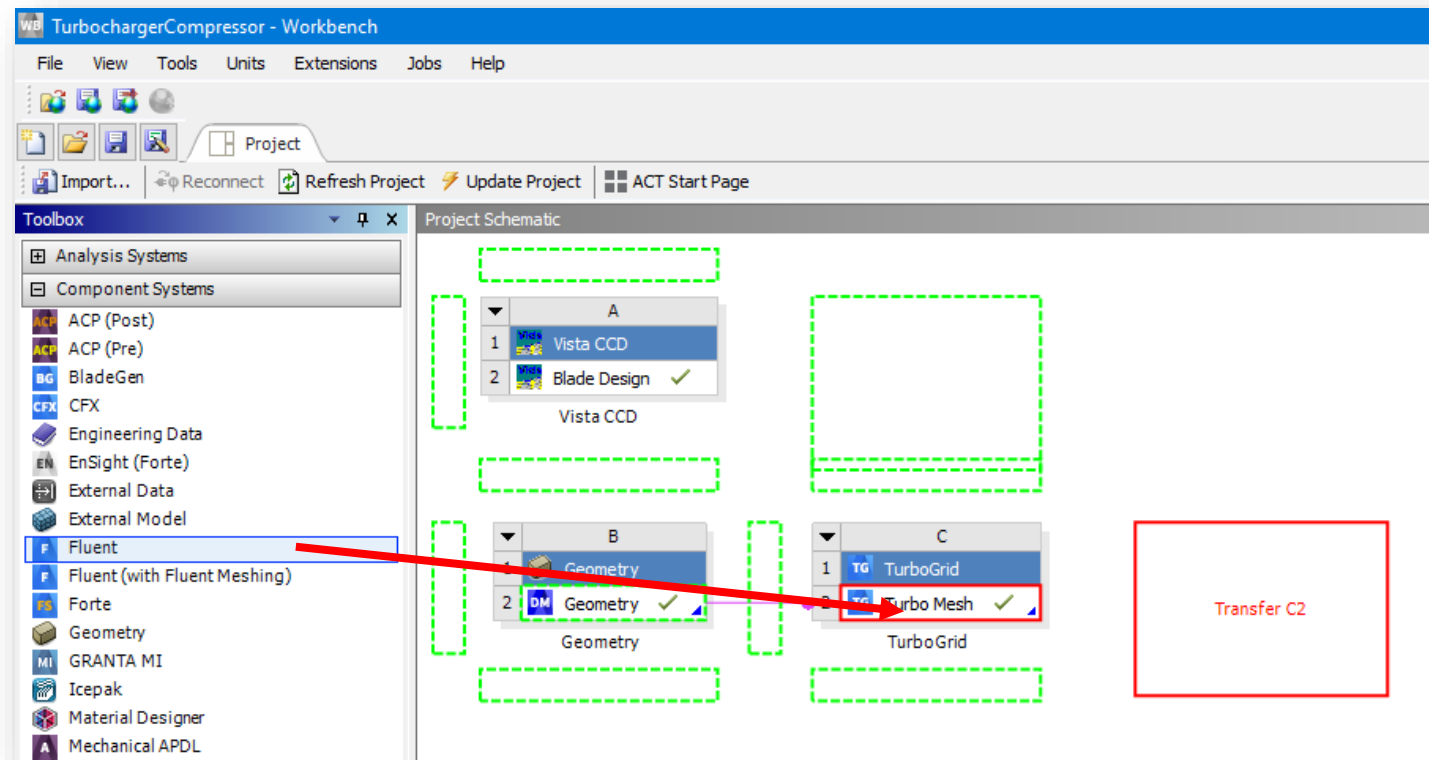
Workbench: Launch Fluent

- Double click on the *Setup* cell *D2* to launch Fluent

Fluent Launcher Settings

- In the *Fluent Launcher*

- Do not check *Double Precision* as the Maximum Aspect Ratio is much smaller that 1000 (see next slide)

- Set the number of Processes for Parallel to 4
  - The mesh size for this case is approximately 180,000 cells (see next slide)
  - If you have enough Parallel licenses and more than 4 cores available, you may set up a number of up to 9 Processes (so that each Processor is solving for not less than 20,000 cells)
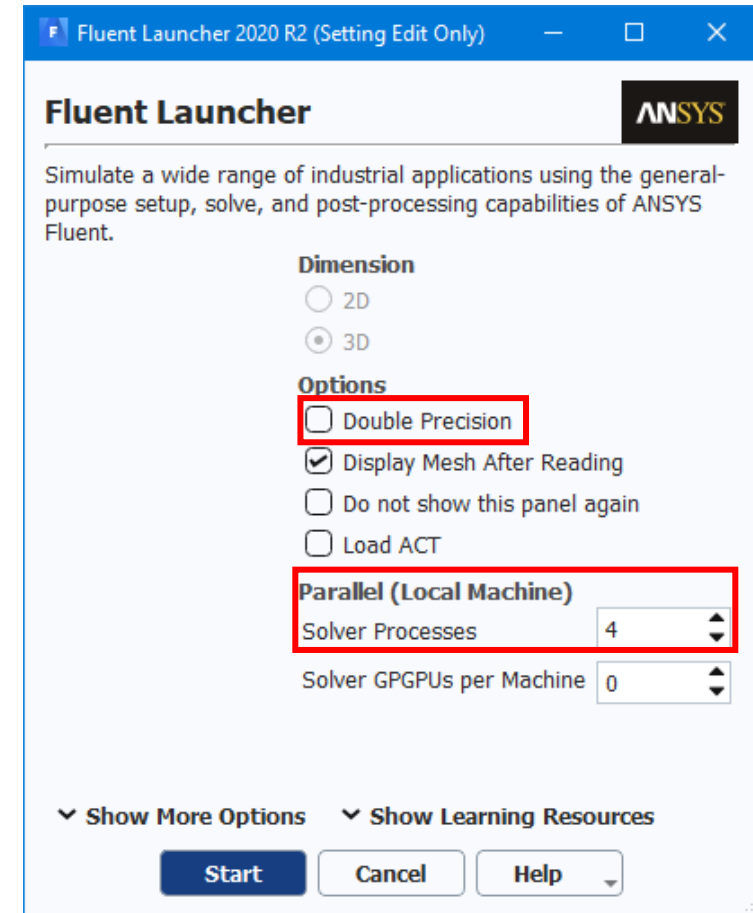
Fluent

- In the Fluent window, you should see the single passage geometry of the radial compressor as shown on the right

- The mesh corresponds to a 60-degree sector of the complete domain

- It is always a good practice to check the mesh size and the mesh quality in the *Mesh* group of the *Domain* tab
  - *Info > Size* will give you the number of cells in the Fluent *Console*, which is approximately 180,000
  - *Quality > Evaluate Mesh Quality* will show you a *Maximum Aspect Ratio* of 1.53e+02 <1000
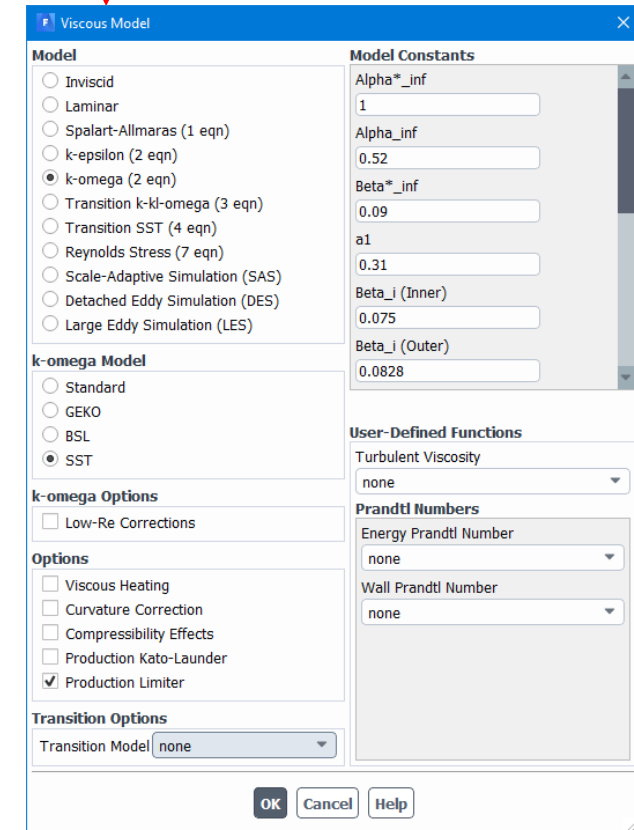   - This justifies the choice of starting Fluent in Single Precision
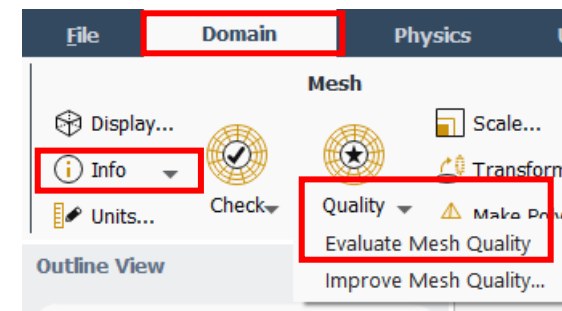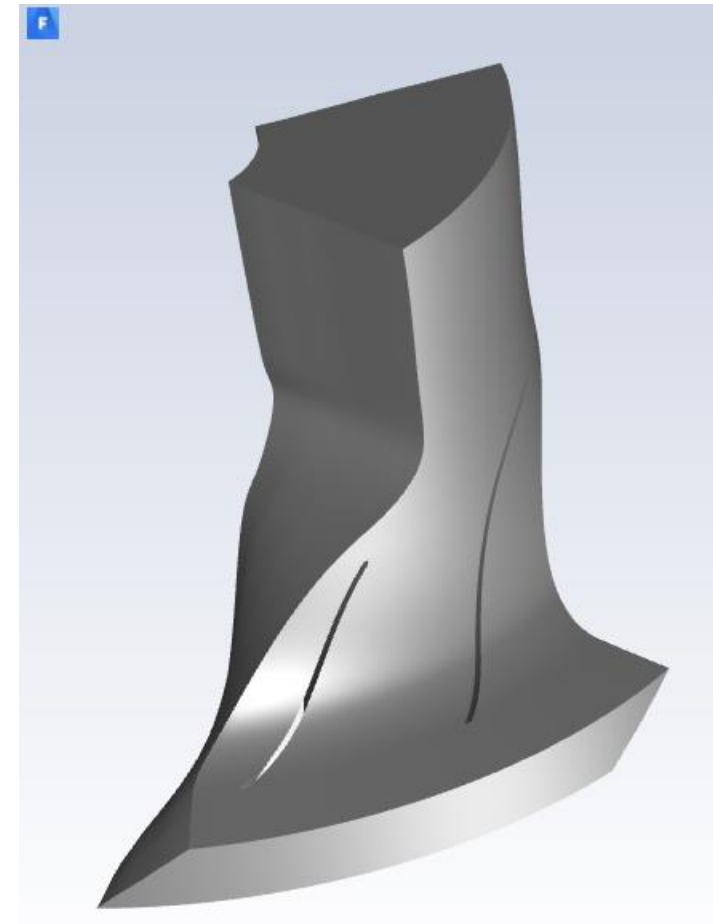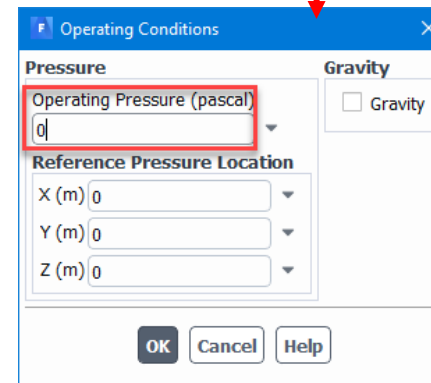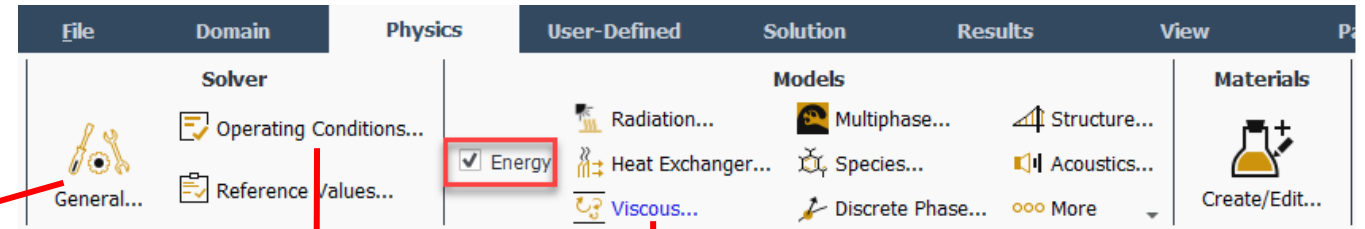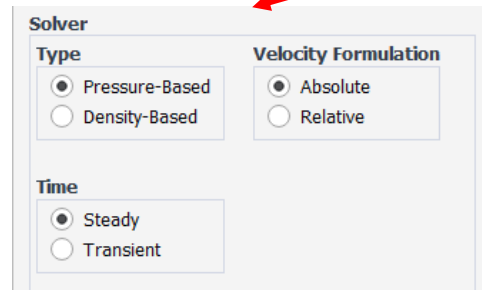
Domain: General Settings

- The rotational speed of the impeller is given in rpm

- Set the units for angular velocity to rpm

- Click *Close*
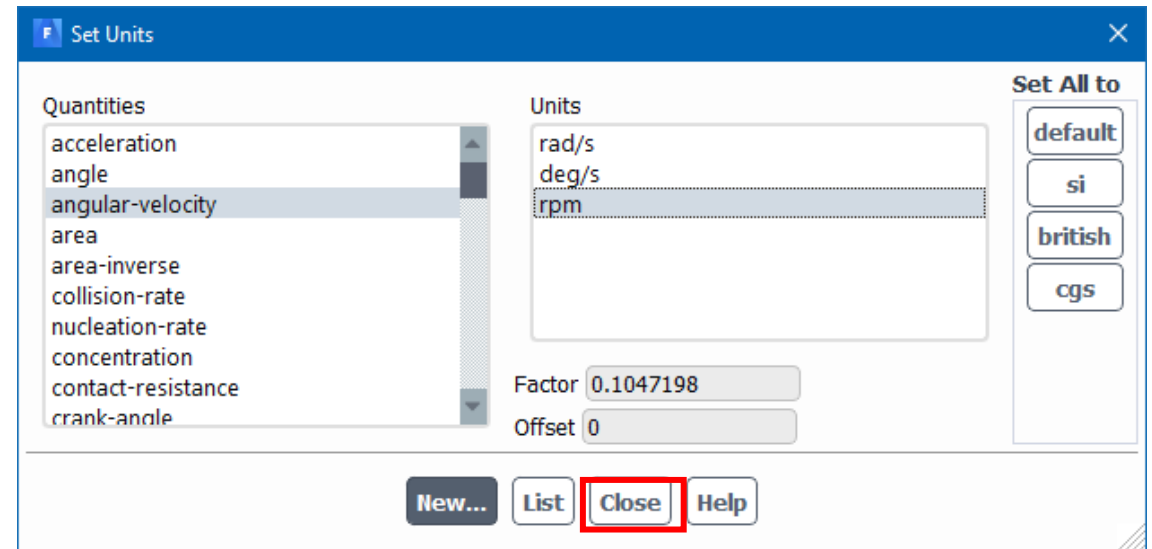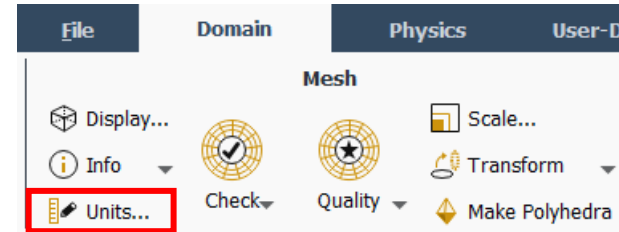
Physics: General, Operating Conditions, Energy and Viscous

- Retain the default solver settings of *Pressure-Based* solver with *Absolute Velocity Formulation*

- Set *Operating Pressure* to 0 (Pa)

- Enable the *Energy* equations

- Keep the default *SST k-omega Viscous Model*, which is the recommended turbulence model for turbomachinery simulations

Physics: Materials

- The default material is Air with constant properties
  - We will need to change its Density and Viscosity to be functions of Temperature

- Click on *Material > Create/Edit* in the *Physics* tab
  - Select *ideal-gas* from the *Density* drop-down list
  - From the *Viscosity* drop-down list select *sutherland*
  - In the *Sutherland Law* dialog box that opens retain the default settings and click *OK*.
  - Click *Change/Create* then *Close*

Physics: Cell Zone Conditions

- Edit the *inlet* cell zone
  - Leave the default *air* as *Material Name*
  - Enable *Frame Motion*
  - The default *Rotation-Axis Direction* is the z-axis and is suitable for this case
  - Set *Rotational Velocity* to -155733 (rpm)
    - Sign verification: If you place your right thumb to point as the <u>negative</u> z-axis, your fingers are curling (in this case) to the same direction with the rotation direction of the impeller. Therefore, the Rotational Velocity was set to a <u>negative</u> number

- Copy the settings of the *inlet* cell zone to the remaining 2 zones
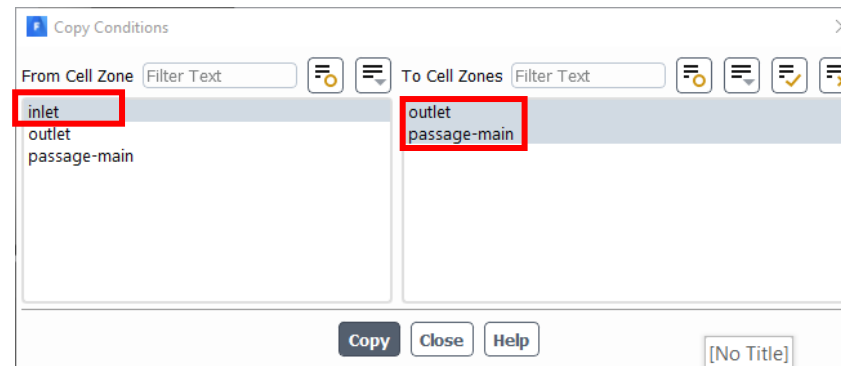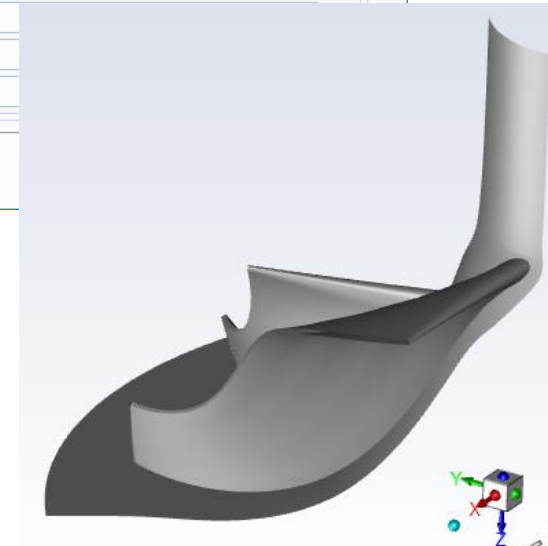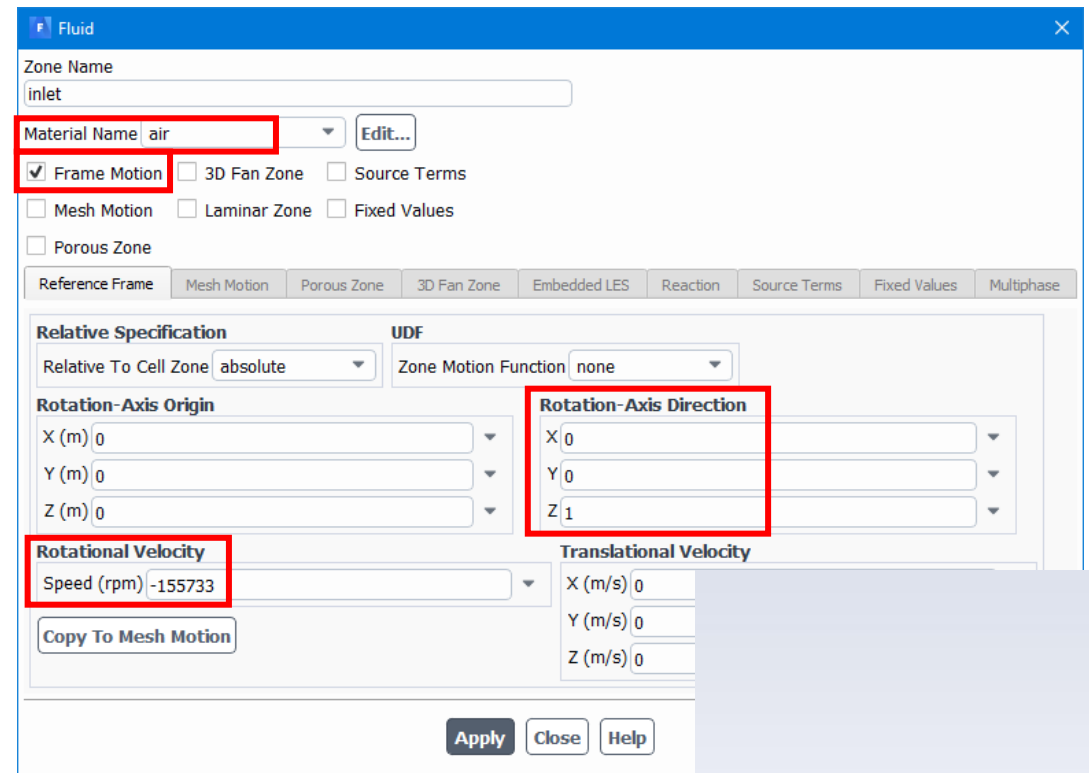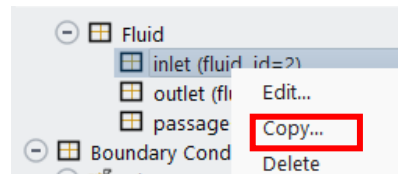
Create Rotational Periodic Zones

- In the *Boundary Conditions* of the *Outline View* expand the *Wall* branch
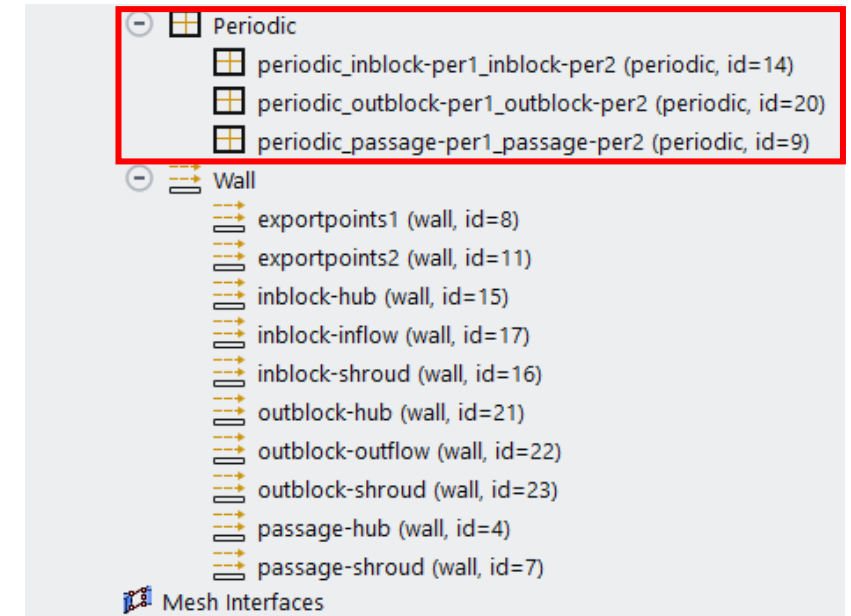
- Select *inblock-per1* and *inblock-per2*
  - Use *Ctrl + RMB* for multiple selections

- *RMB > Periodic*

- Select *Rotational* as *Type* and leave all the rest to default values
  - Note that the *Rotation-Axis Direction* is automatically set to the one defined in the cell-zone Frame *Motion*
  - It is important to always first define the axis of rotation in the cell-zone conditions before creating the Rotational Periodic boundaries
  - Look for *Console* message:
    Zone 18 deleted
    Created a conformal periodic boundary.

- The periodic pair is placed under the newly created *Periodic* branch in the *Outline*
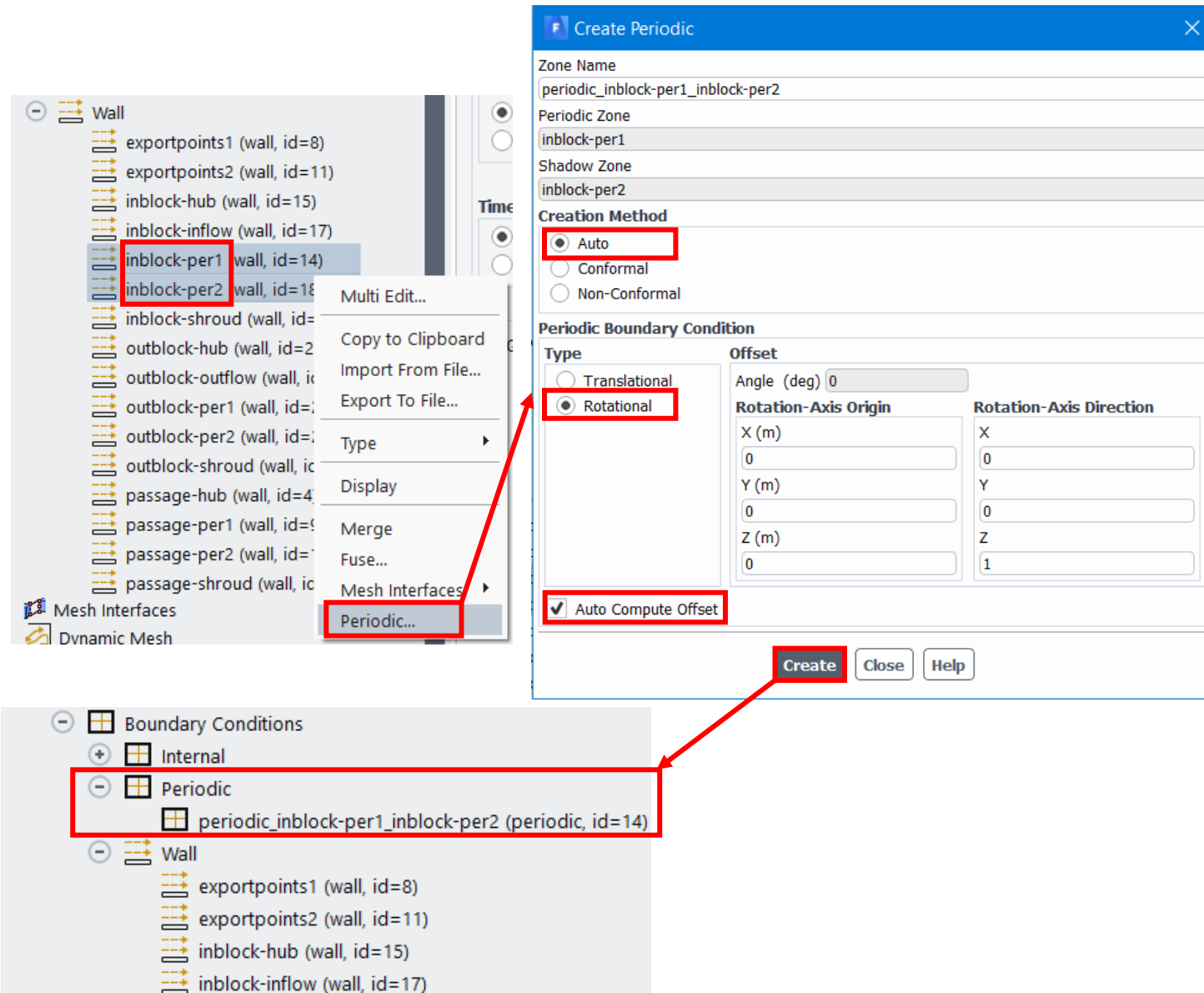
Create Rotational Periodic Zones (2)

- In the same way Create Rotational Periodic Zones for:

- *outblock-per1* and *outblock-per2*

- *passage-per1* and *passage-per2*
  - This interface will fail using *Auto Compute Offset*
    - Error: Auto computation of Rotational Offset is not successful, please enter Angle.
  - Do a mech check
    - Remember, always do a mesh check after creating rotational periodic boundaries
    - This will give you a summary of all Periodic zones including the rotation angle and the axis of rotation
 - Manually give an offset of 60 degrees for this periodic boundary

Boundary Conditions: Inlet

- Set the boundary conditions for *inblock-inflow* zone
  - *RMB* on *inblock-inflow* and set *Type* to *pressure-inlet*
  - Check *Prevent Reverse Flow*
  - Set a *Gauge Total Pressure* of 101325 (pascal) at the inlet *
  - Set a Supersonic/Initial Gauge Pressure of 99000
    - Initial Gauge Pressure is set 1% to 2% lower than the *Gauge Total Pressure.* This will help in the flow field initialization (see slide 26)
  - Accept all remaining defaults in the Momentum tab
  - In the thermal tab, set a *Total Temperature* of 288.15 (k) (not shown) and click *Apply* then *Close*

  *   Note that the Operating Pressure was set to 0 (Pa) for this case. For this reason, the Gauge Total Pressure is set to 101325 (Pa) which is equal to one atmosphere
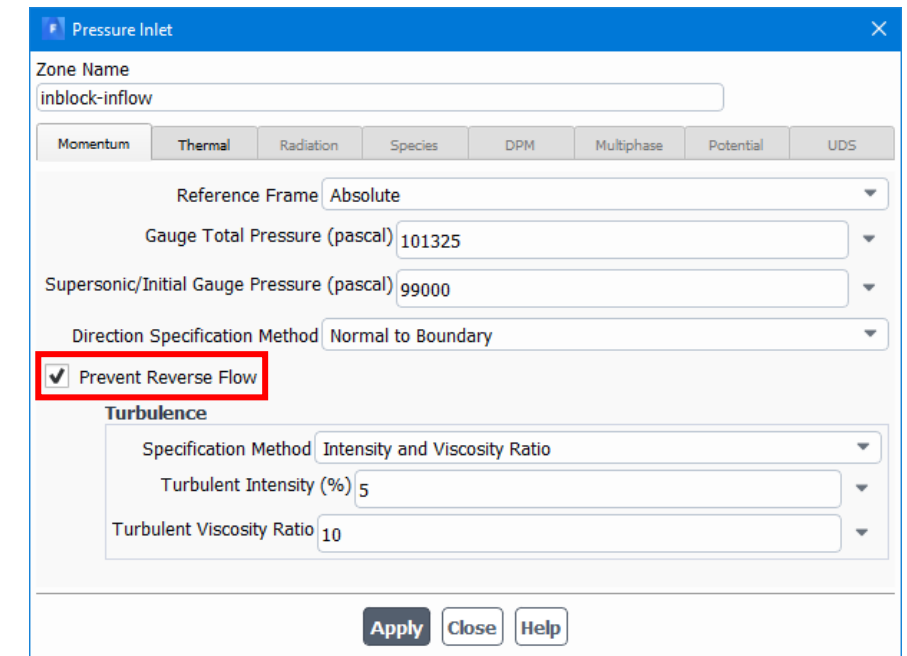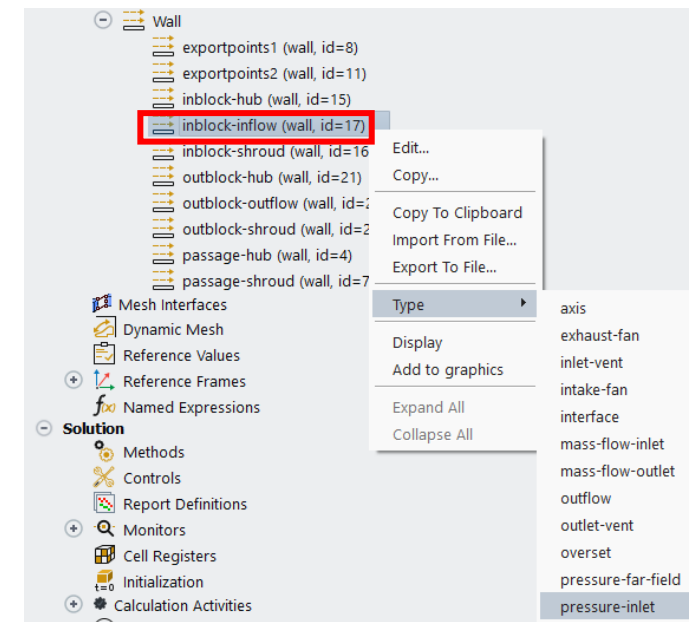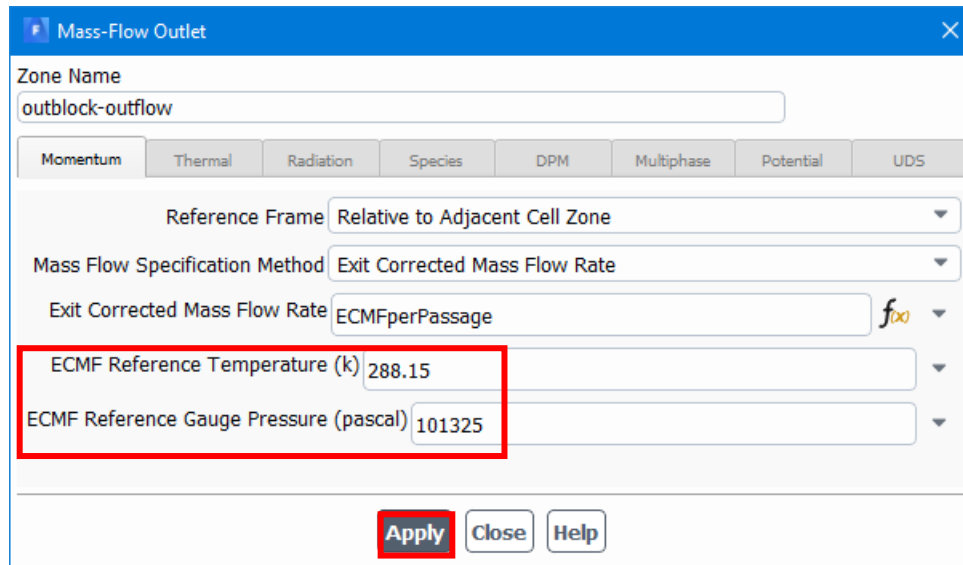
Boundary Conditions: Outlet

- Set the boundary conditions for *outblock-outflow* zone
  - *RMB* on *outblock-outflow* and set *Type* to *mass-flow-outlet*
  - Set *Mass Flow Specification Method* to *Exit Corrected Mass Flow Rate*
  - From the drop down next to *Exit Corrected Mass Flow Rate* select *New Input Parameter/Epression…*
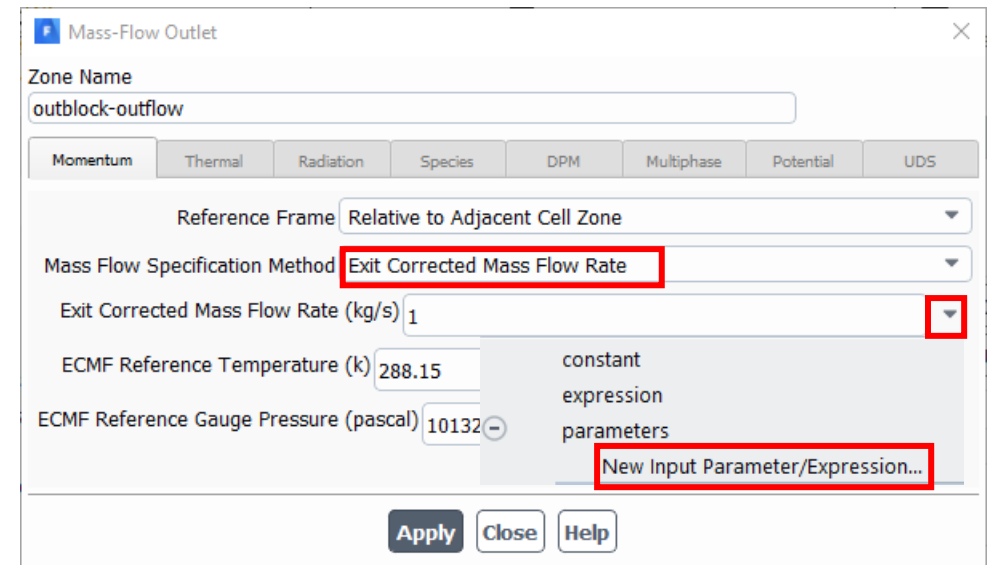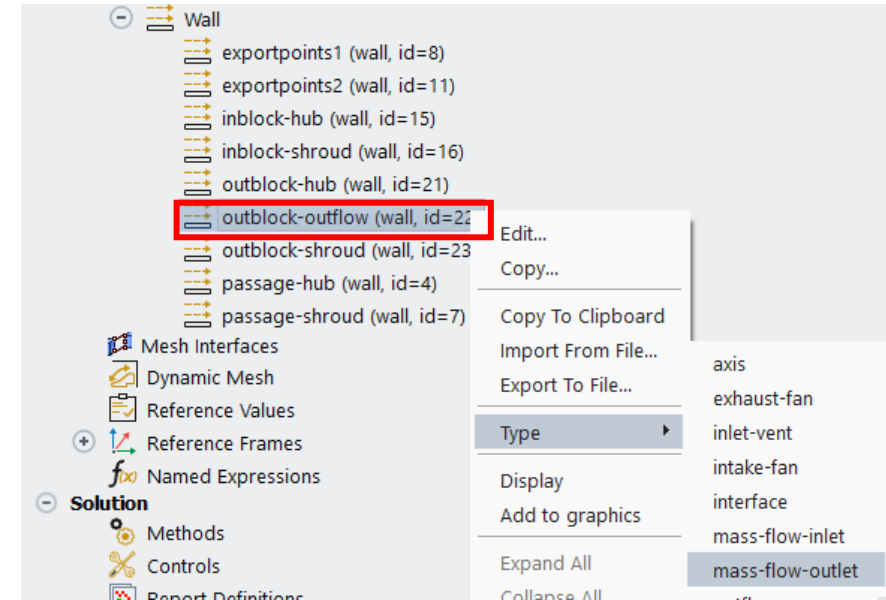
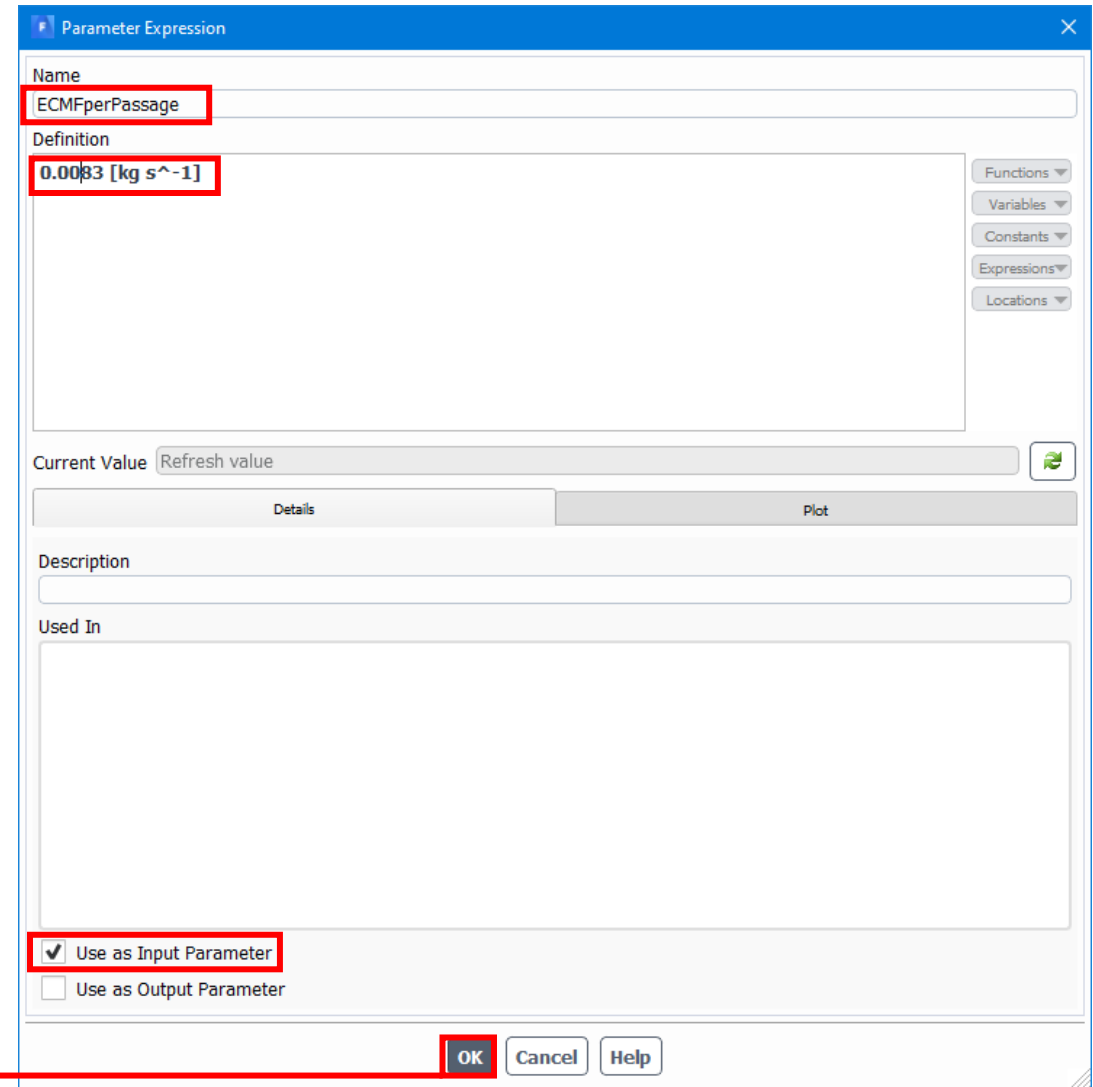# Boundary Conditions: Outlet (2)

- In the *Parameter Expression* panel set:
  - *Name = ECMFperPassage*
  - *Definition = 0.0083 [kg s^-1]*
  - *Use as Input Parameter = checked*

- Back to Mass-Flow Outlet panel
  - Set *ECMF Reference Temperature* and *Gauge Pressure as shown below*

Note on Exit Corrected Boundary Condition

- The Exit Corrected Boundary Condition is well defined all the way from choke to stall

  - In the past, a pressure outlet was used at choke while a mass flow boundary was used toward stall

  - Exit corrected mass flow will allow you to get the entire speedline with one boundary condition type

  - To estimate the exit corrected mass flow, we use the equation below

$$\dot{m}_{cor} = \dot{m}_{spec} \frac{\sqrt{T_{01}/T_{ref}}}{P_{01}/P_{ref}}$$

  - m_{spec} is the mass flow, T_{01} and P_{01} are the outlet temperature and pressure. We can know them either from Vista CCD, or from a first Fluent calculation using inlet total Pressure = 1 atm and outlet static pressure = 1 atm, which gives a first point in the Choke region
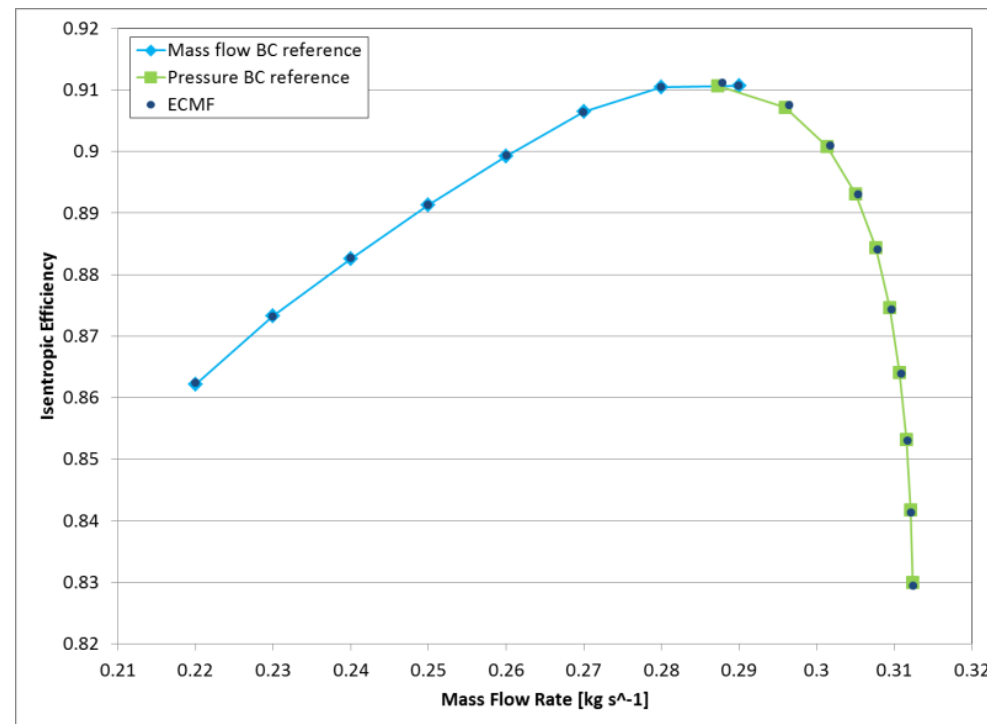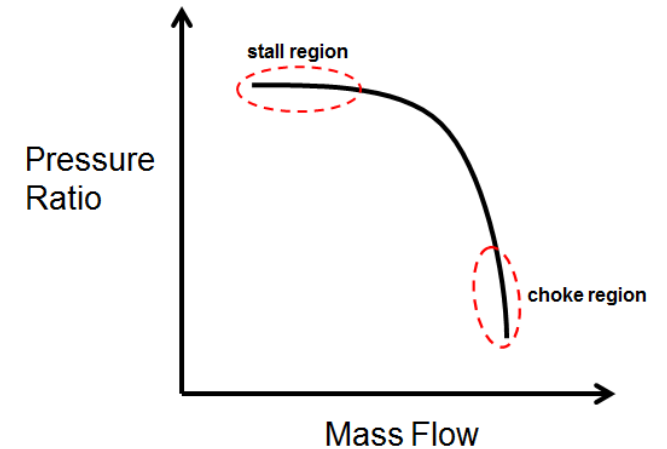
Speed line for compressor test case showing consistency of results between new exit-corrected mass flow BC and other BCs

Boundary Conditions: Automatically Set Walls

- From the remaining zones under *Wall* in the *Outline View*, the boundaries marked with red boxes correspond to walls which are stationary in the Moving Reference Frame of the 3 cell zones

  - Note that *exportpoints1* corresponds to the main blade and *exportpoints2* to the splitter blade. You may verify this by left-clicking on any of the two blades in the graphics window; the name of the boundary is shown in the quick property editor

- As the default settings of a *No Slip, Stationary Wall, Relative to Adjacent Cell Zone* is what we want, there is no need to change this default setting for any of these walls for this case
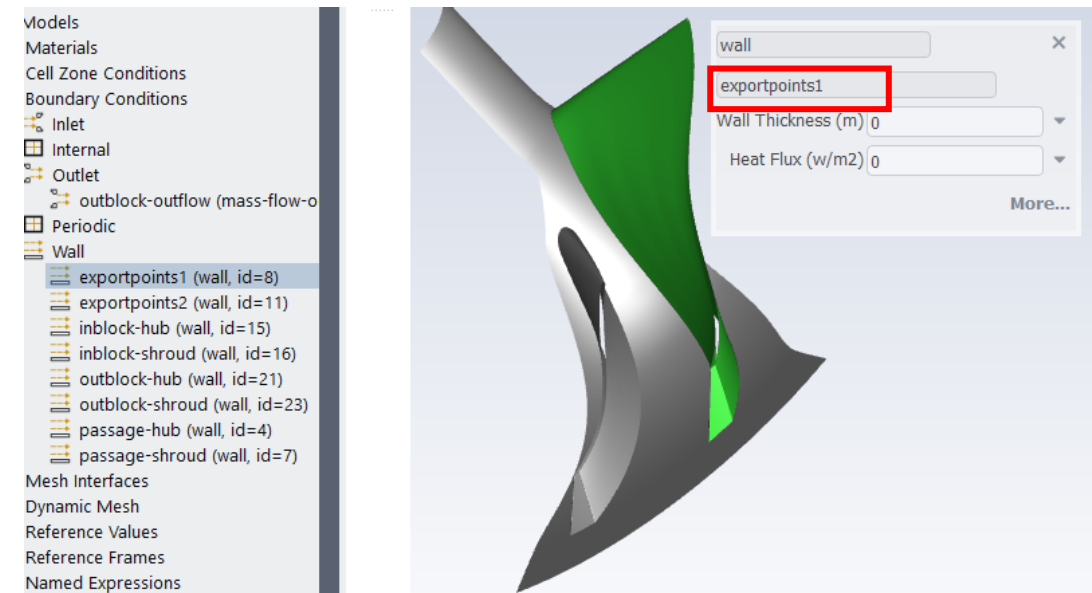
Boundary Conditions: Shroud Walls

- The 3 shroud walls, belong to rotating cell-zones but are stationary in the absolute frame. You will now set these walls using *Multi Edit…*

  - Select all 3 walls from the Outline using *Ctrl + LMB* and then *RMB > Multi Edit…*

  - Do all settings shown on the right and click *Apply*

Such walls are currently set as Moving Walls with Rotational Motion relative to the Absolute frame with a zero Rotational speed
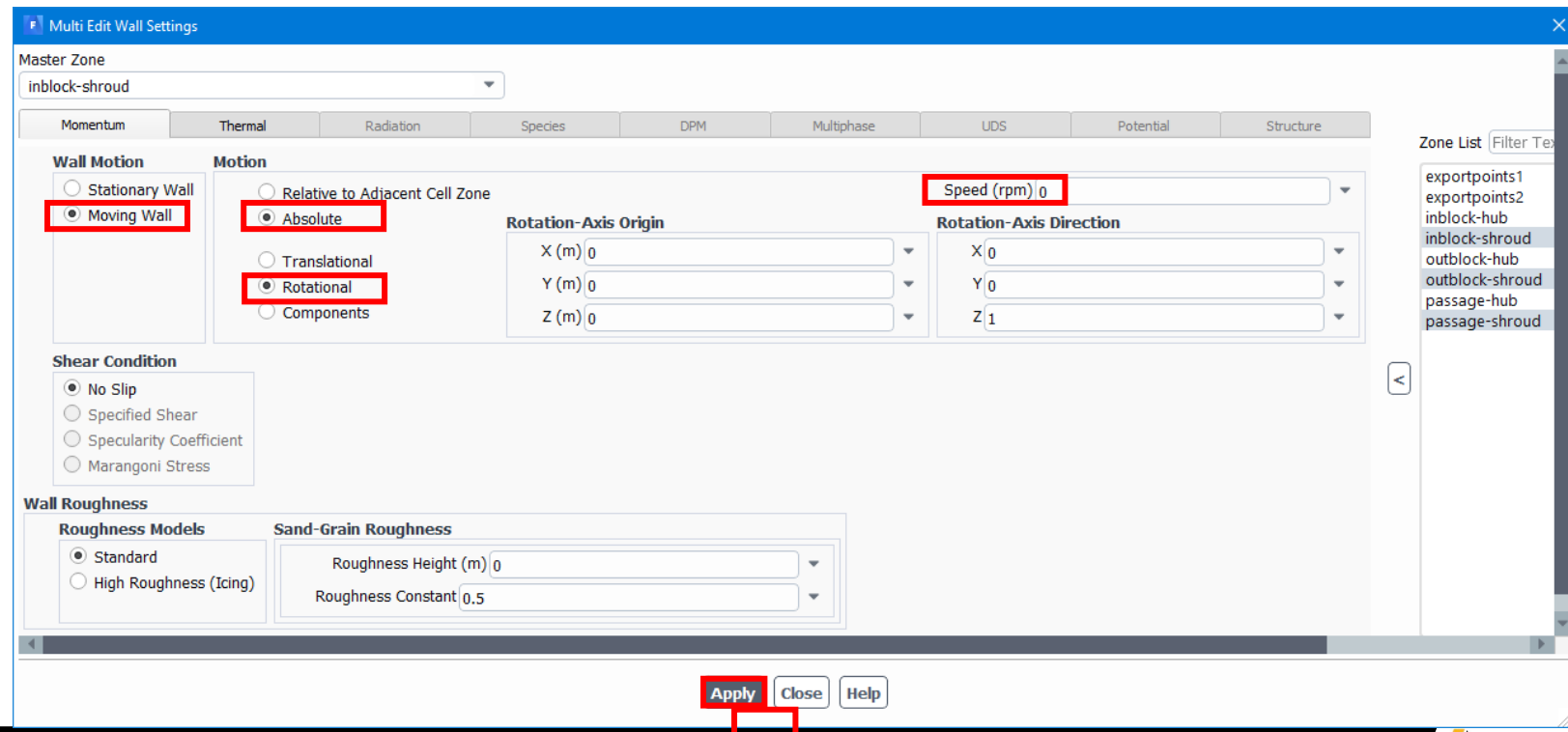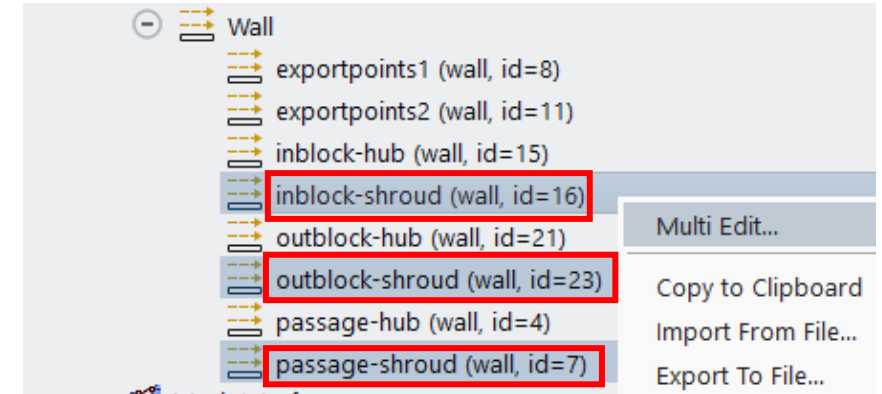
Solution: Report Definition Mass Flow Rate

- In the *Solution* tab create a new *Report Definition > Surface Report* about the *Mass Flow Rate,* with the following settings:
  - *Name = mass-inlet*
  - *Boundaries = inblock-inflow*
  - *Report File = checked*
  - *Report Plot = checked*
  - *Create Output Parameter = checked*
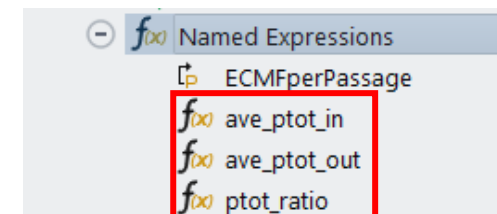
New Method for Report Definitions using Named Expressions

- You will now use a new method for creating Report Definitions and Output Parameters, based on Named Expressions
  - Named Expressions are introduced in the Fluent Getting Started course in the "Setting up Physics" lecture
  - In this workshop for convenience, a file *PressureRatio.tsv* is provided with the workshop inputs, containing the syntax for 3 named Expressions:

```
name              definition            description        input-parameter output-parameter
"ave_ptot_in"  "Average(TotalPressure,['inblock-inflow'],Weight = 'MassFlowRate')"   ""        #f        #f
"ave_ptot_out" "Average(TotalPressure,['outblock-outflow'],Weight = 'MassFlowRate')" ""        #f        #f
"ptot_ratio"   "ave_ptot_out/ave_ptot_in"     ""          #f          #f
```

  - You can import this file into the *Named Expressions* branch of the *Outline* using *RMB>Import From File…*
  - The 3 *Named Expressions* highlighted by a red box in the bottom-right image are created
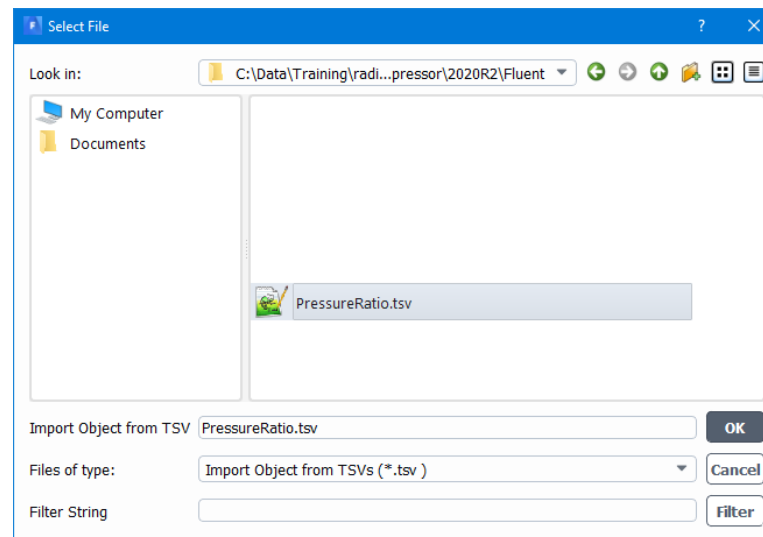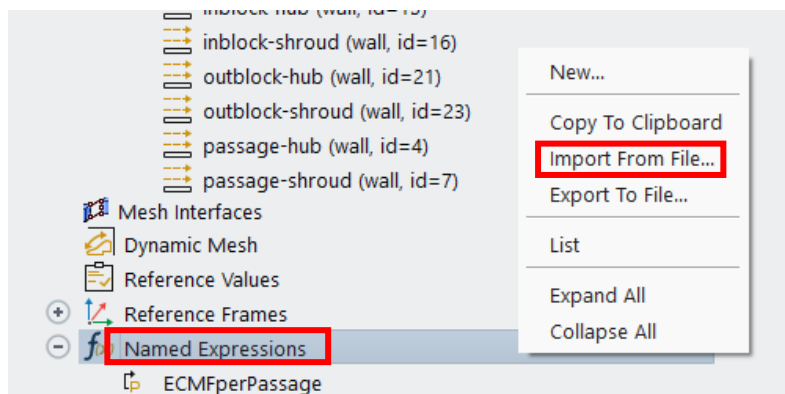
New Method for Report Definitions using Named Expressions (2)

- Double click on any of the new *Named Expressions* for examining its definition in the *Expression* editor

- *RMB* on *ptot_ratio* and select *Mark as Output Parameter*

  - This is a very convenient method for creating important output parameters, here for the absolute total pressure ratio

  - See under *Named Expressions* that you have one input parameter and one output parameter
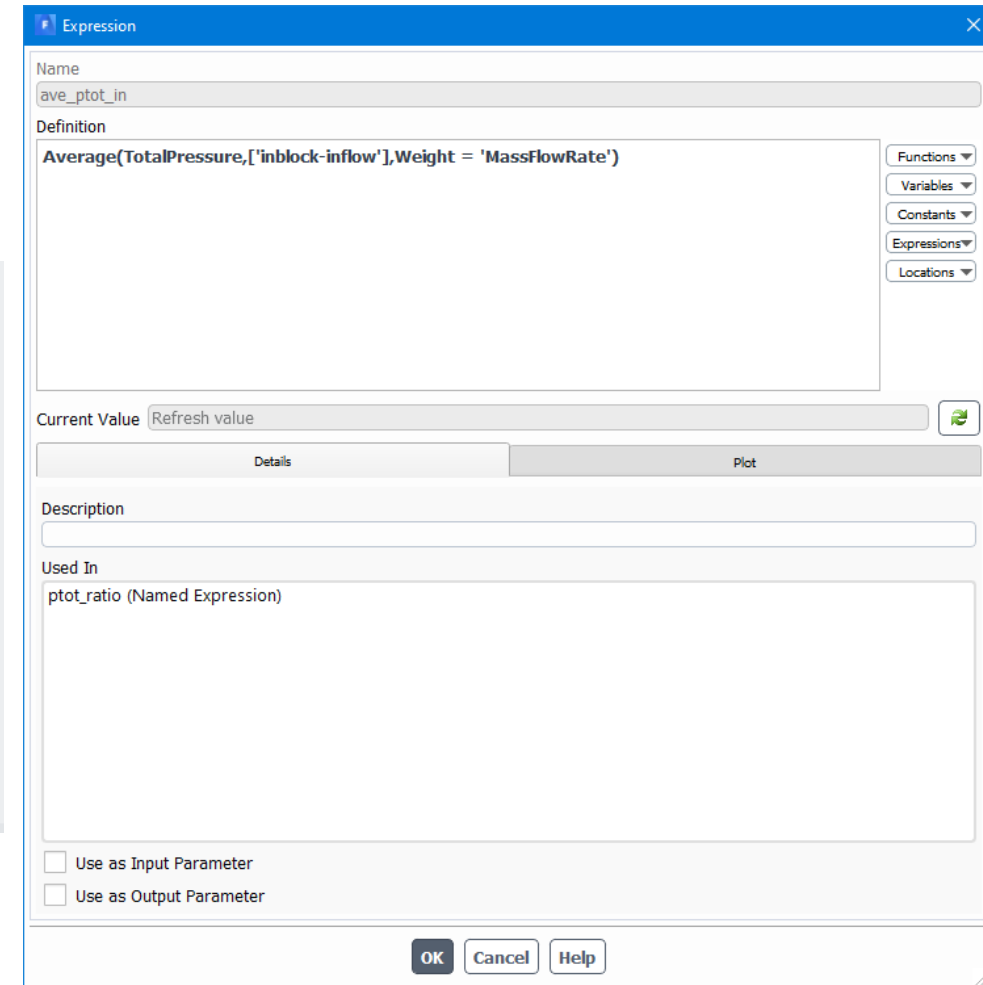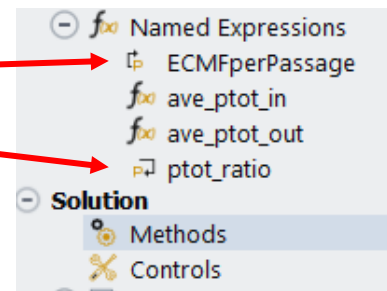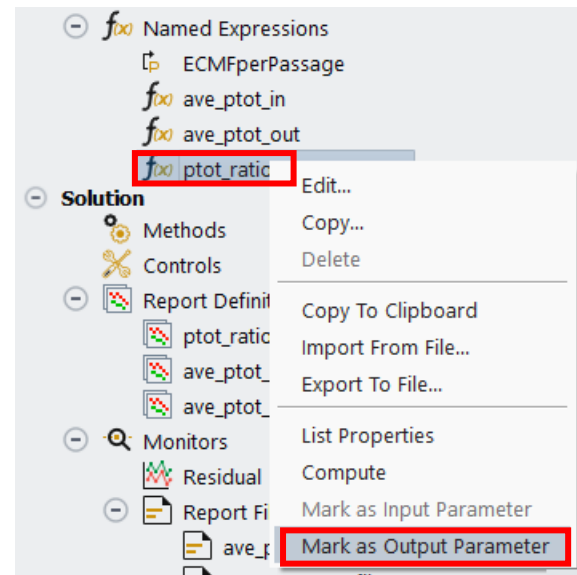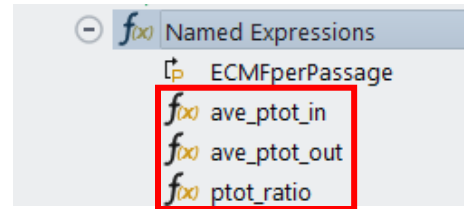
Solution: Create Report Definitions Using Named Expressions

- In the *Solution* tab click on *Definitions* in the *Reports* section and choose *New >Expression…*
Enter the following in the definition panel and click *OK*:
  - *Name = ave_ptot_in*
  - *Expressions > ave_ptot_in*
  - *Report File = checked*
  - *Report Plot = checked*

- In the same way create two more *Report Definitions* for the Named expressions *ave_ptot_out* and *ptot_ratio*
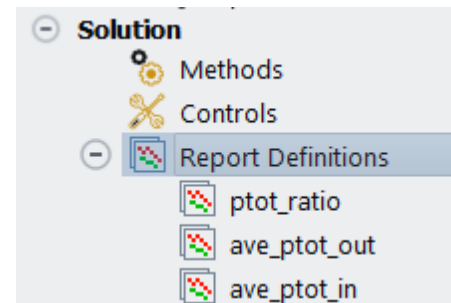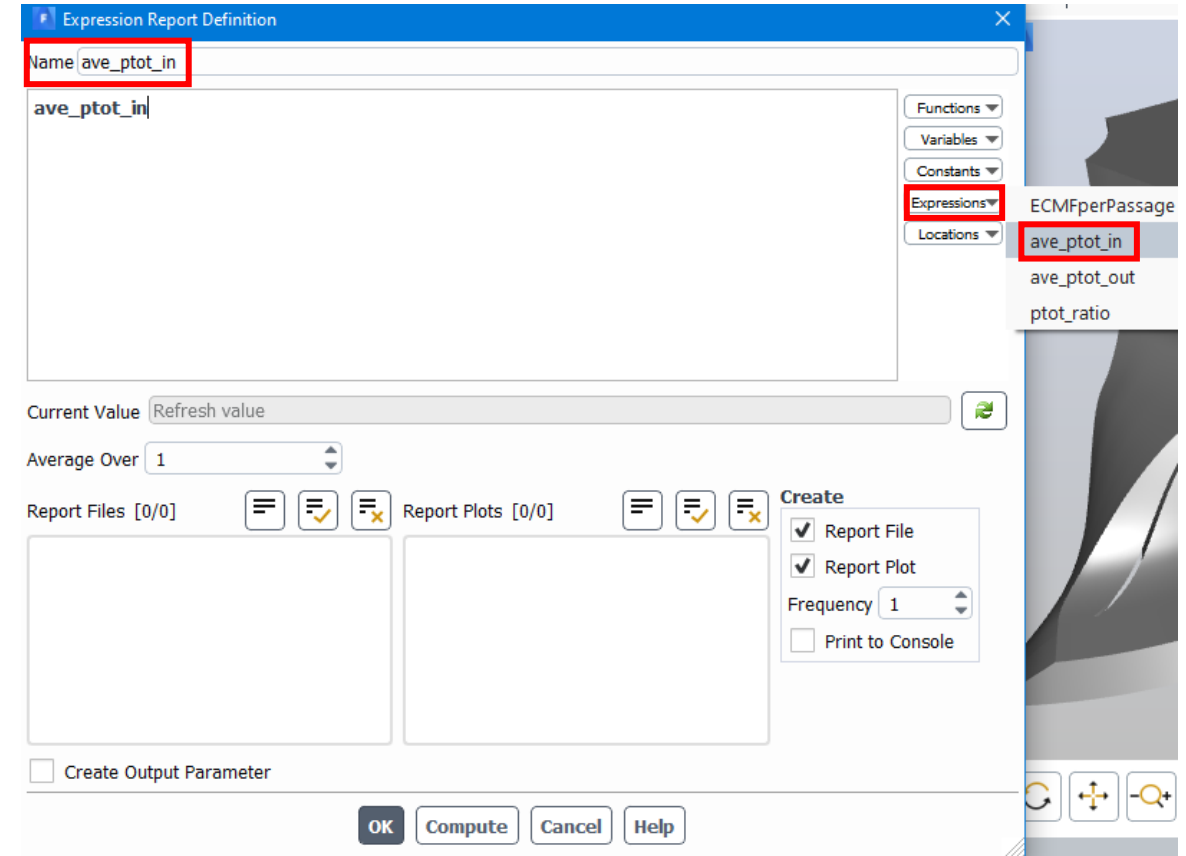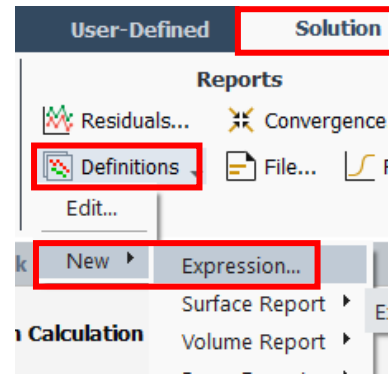
Solution: Solution Methods

- Always use the default *Coupled "Pseudo-Transient" Solver* for turbomachinery calculations
  - If for any reason the Solution method is set to some *Scheme* other than *Coupled,* click the *Default* button at the bottom of the panel

- Turn on *High Order Term Relaxation* (more stable)
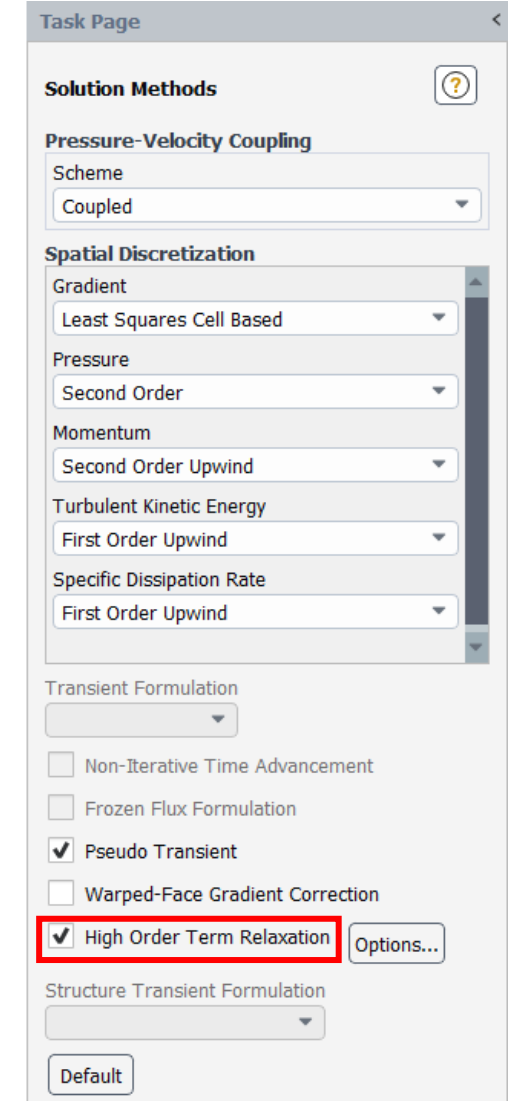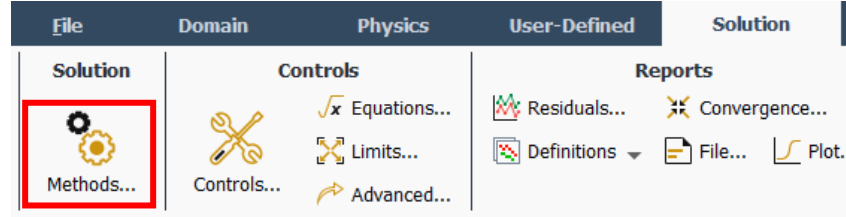
Solution: Initialization (Best Practice Procedure)

- Perform a *Standard Initialization* using the inlet values

  - Make sure you have set at the inlet boundary, an *Initial Gauge Pressure*, which is about 1 to 2% lower than the *Gauge Total Pressure* (this was already done on slide 15)

    - This will ensure a proper k and omega initialization

- Then perform a *Hybrid Initialization* after having checked the option *Use Specific Initial Pressure on Inlets*

Note: FMG initialization is currently not compatible with all General Turbo Interfaces
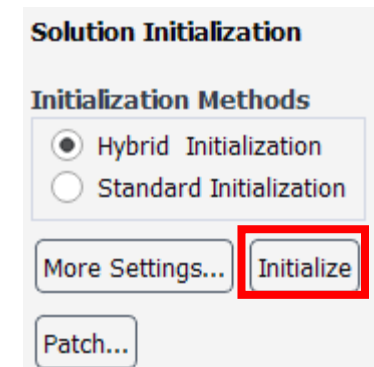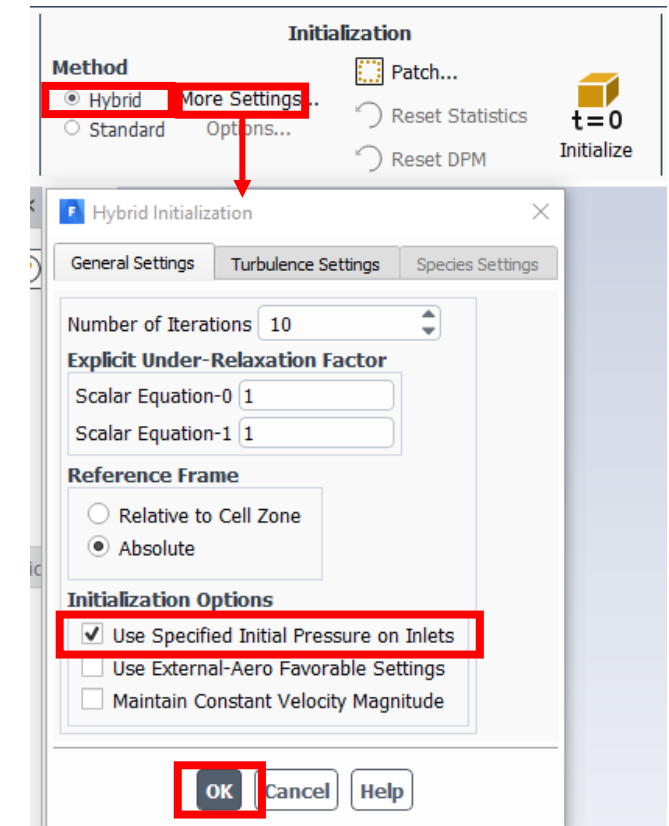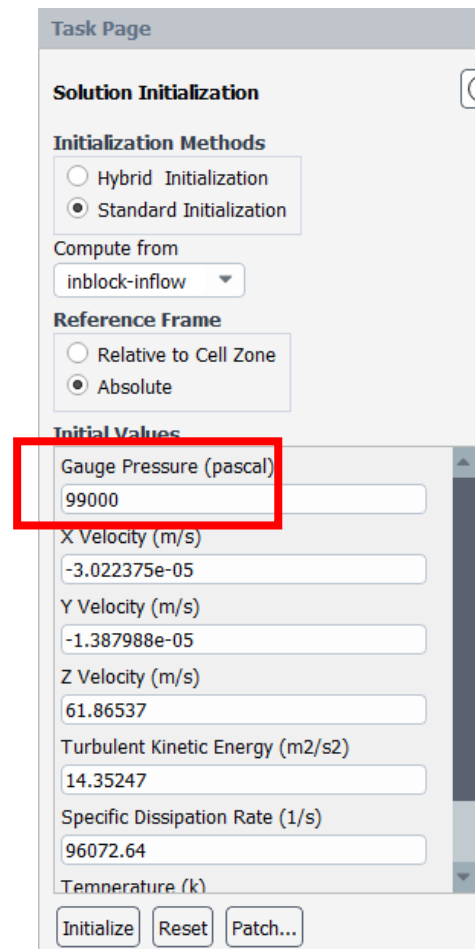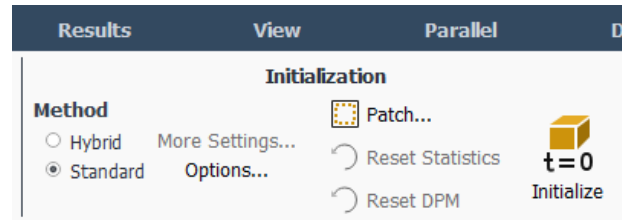
Solution: Run the Solver

- Click on *Run Calculation...*
  - Set *Verbosity* to 1
    - This will produce a more detailed runtime solver output, including the time step used by the pseudo-transient solver

- Set *Time Scale Factor* to 10

- Set *No. of Iterations* to 250

- Click *Calculate*
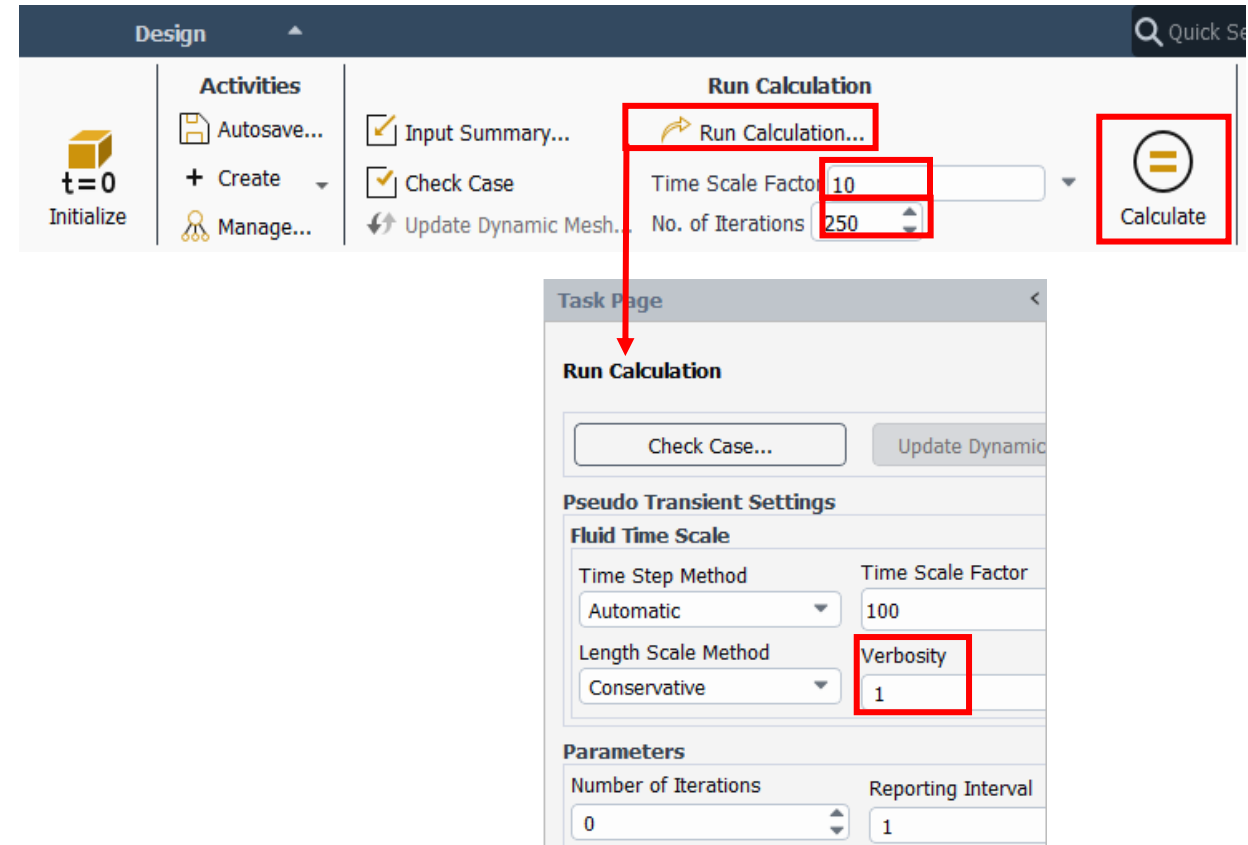
Solver Convergence

- The case converges fast in approximately 180 iterations:
  - All residuals drop below the target of 1e-03
  - The report plots for the inlet mass flow rate and total pressure ratio are not changing

Note: Residuals and monitor plots may differ between two computers or Ansys releases
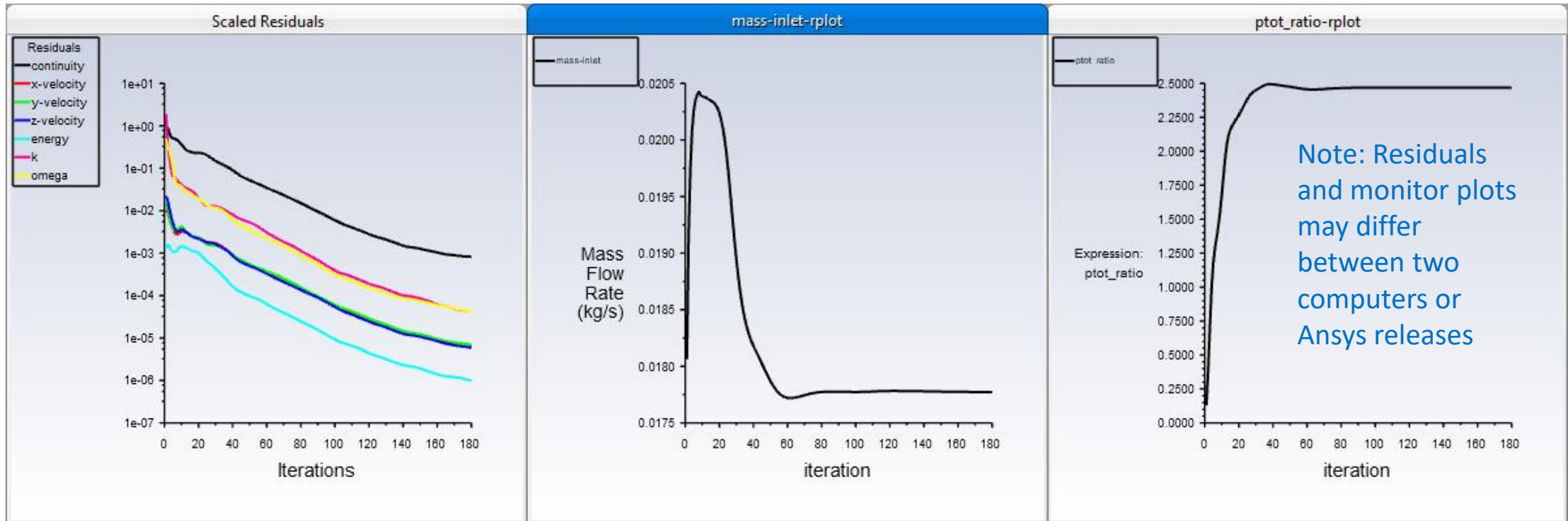
Results: Pressure Contours

- Create a contour plot on the blade and hub walls

- The pressure increases through the impeller passage, as expected

- Additional post-processing of this case can be done with CFD-Post using the method presented in workshop 03
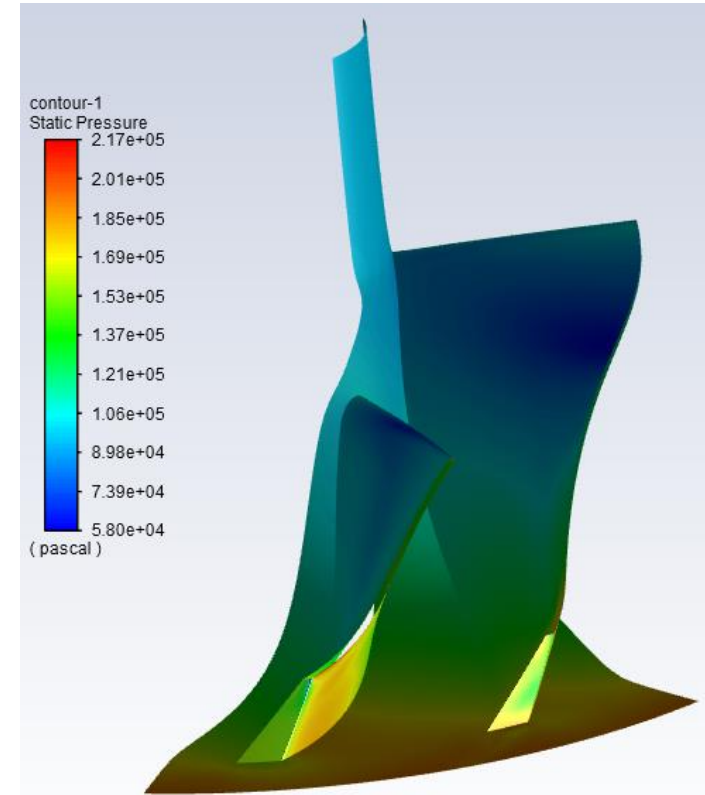
Note: Results may differ between two computers or Ansys releases

Creating a Speedline in Workbench

Workbench: Parameter Set

- In the Workbench window double click on *Parameter Set*
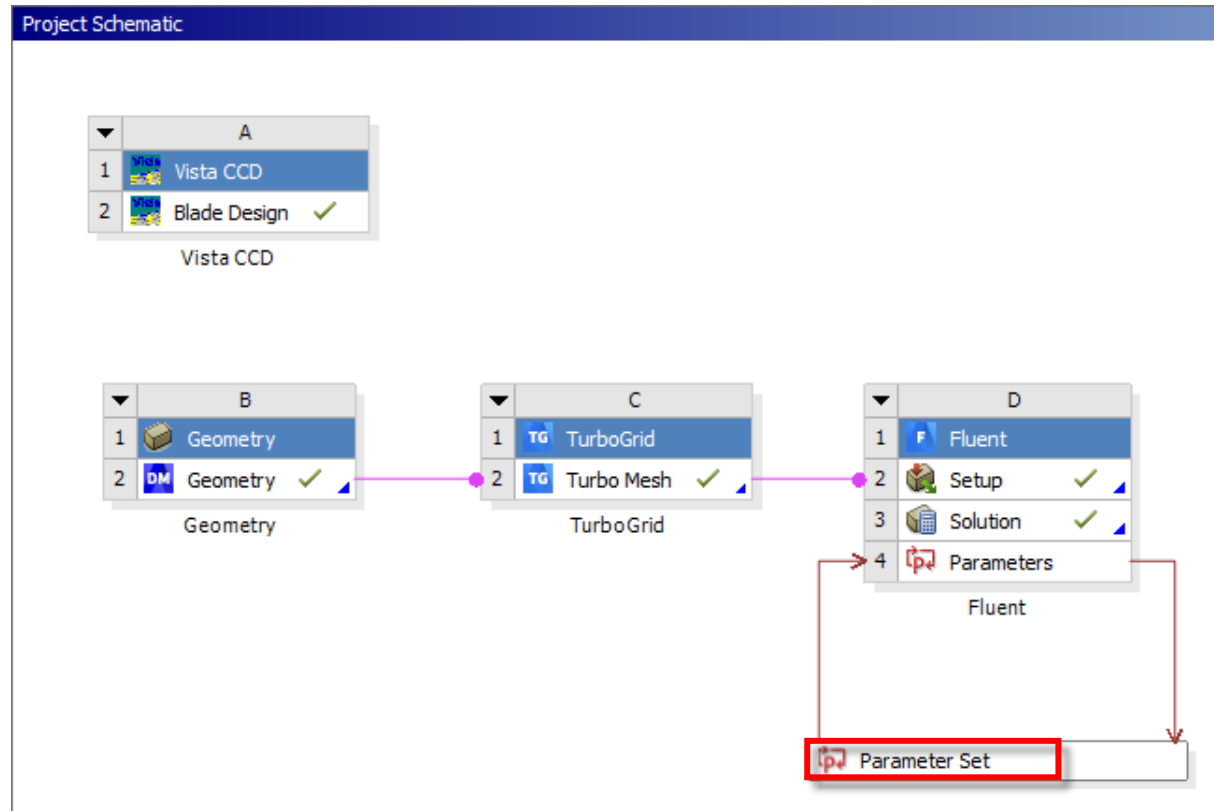
Workbench: Parameter Set

- In the Parameter Set tab
  - The *Outline of All Parameters* section shows the input and output parameters defined in Fluent
    - The total pressure ratio for this design point is 2.4671
    - The Mass Flow Rate for the 60-degrees passage is 0.017772 kg/s

Workbench: Speedline Calculation

- In the *Table of Design Points* add some more design points

- Enter values for *ECMFperPassage* of 0.05, 0.067, 0.01, and 0.0116 (column B)

- Select *Retain* for all design points (column E)

- Click *Update All Design Points*

Table of Design Points

	A	B	C	D	E	F	G
1	Name ▾	P1 - ECMFperPassage ▾	P2 - ptot_ratio ▾	P3 - mass-inlet-op ▾	Retain	Retained Data	Note ▾
2	Units	kg s^-1	▾	kg s^-1			
3	DP 0 (Current)	0.0083	2.4671	0.017772	☑	✓	
*					☐		

Table of Design Points

	A	B	C	D	E	F	G
1	Name ▾	P1 - ECMFperPassage ▾	P2 - ptot_ratio ▾	P3 - mass-inlet-op ▾	☐ Retain	Retained Data	Note ▾
2	Units	kg s^-1	▾	kg s^-1			
3	DP 0 (Current)	0.0083	2.4671	0.017772	☑	✓	
4	DP 1	0.005	⚡	⚡	☑	⚡	
5	DP 2	0.0067	⚡	⚡	☑	⚡	
6	DP 3	0.01	⚡	⚡	☑	⚡	
7	DP 4	0.0116	⚡	⚡	☑	⚡	
*					☐		

File   View   Tools   Units   Extensions   Jobs   Help

Project   Parameter Set ✕

Resume   ⚡⚡ Update All Design Points

Toolbox   Outline of All Parameters
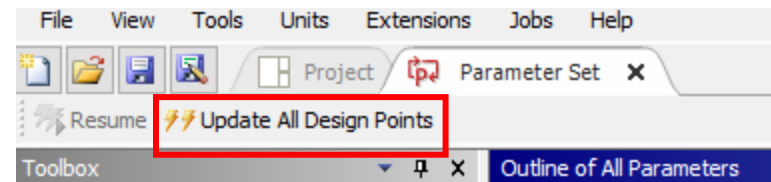
Workbench: Speedline Calculation (2)

- When the run is complete for all design points, you should see the table fully populated
- You may click on one of the down arrow buttons to sort the data by one of the parameters

Table of Design Points

	A	B	C	D	E	F	G
1	Name ▾	P1 - ECMFperPassage ▾	P2 - ptot_ratio ▾	P3 - mass-inlet-op ▾	☐ Retain	Retained Data	Note ▾
2	Units	kg s^-1 ▾		kg s^-1			
3	DP 0 (Current)	0.0083	2.4671	0.017772			
4	DP 1	0.005	2.5156	0.01098	☑	✓	
5	DP 2	0.0067	2.5534	0.014782	☑	✓	
6	DP 3	0.01	2.22	0.019451	☑	✓	
7	DP 4	0.0116	1.9657	0.020197	☑	✓	
*					☐		

(dropdown menu: Ascending, Descending, Sort Settings...)

Table of Design Points

	A	B	C	D	E	F	G
1	Name ▾	P1 - ECMFperPassage ▾	P2 - ptot_ratio ▾	P3 - mass-inlet-op ⇅	☐ Retain	Retained Data	Note ▾
2	Units	kg s^-1 ▾		kg s^-1			
3	DP 1	0.005	2.5156	0.01098	☑	✓	
4	DP 2	0.0067	2.5534	0.014782	☑	✓	
5	DP 0 (Current)	0.0083	2.4671	0.017772	☑	✓	
6	DP 3	0.01	2.22	0.019451	☑	✓	
7	DP 4	0.0116	1.9657	0.020197	☑	✓	
8					☐		

Note: Results may differ between two computers or Ansys releases

/Ansys

Workbench: Speedline Calculation (3)

- Make a plot of Total Pressure Ratio vs. Mass Flow Rate
  - Click *Parameter Chart 0*
  - Under *Properties*, set *X-Axis* to *P3*, and *Y-Axis* to *P2, to draw Ptot_ratio vs Mass-inlet*

- The speedline is displayed

- When done, save the Workbench project and exit Fluent
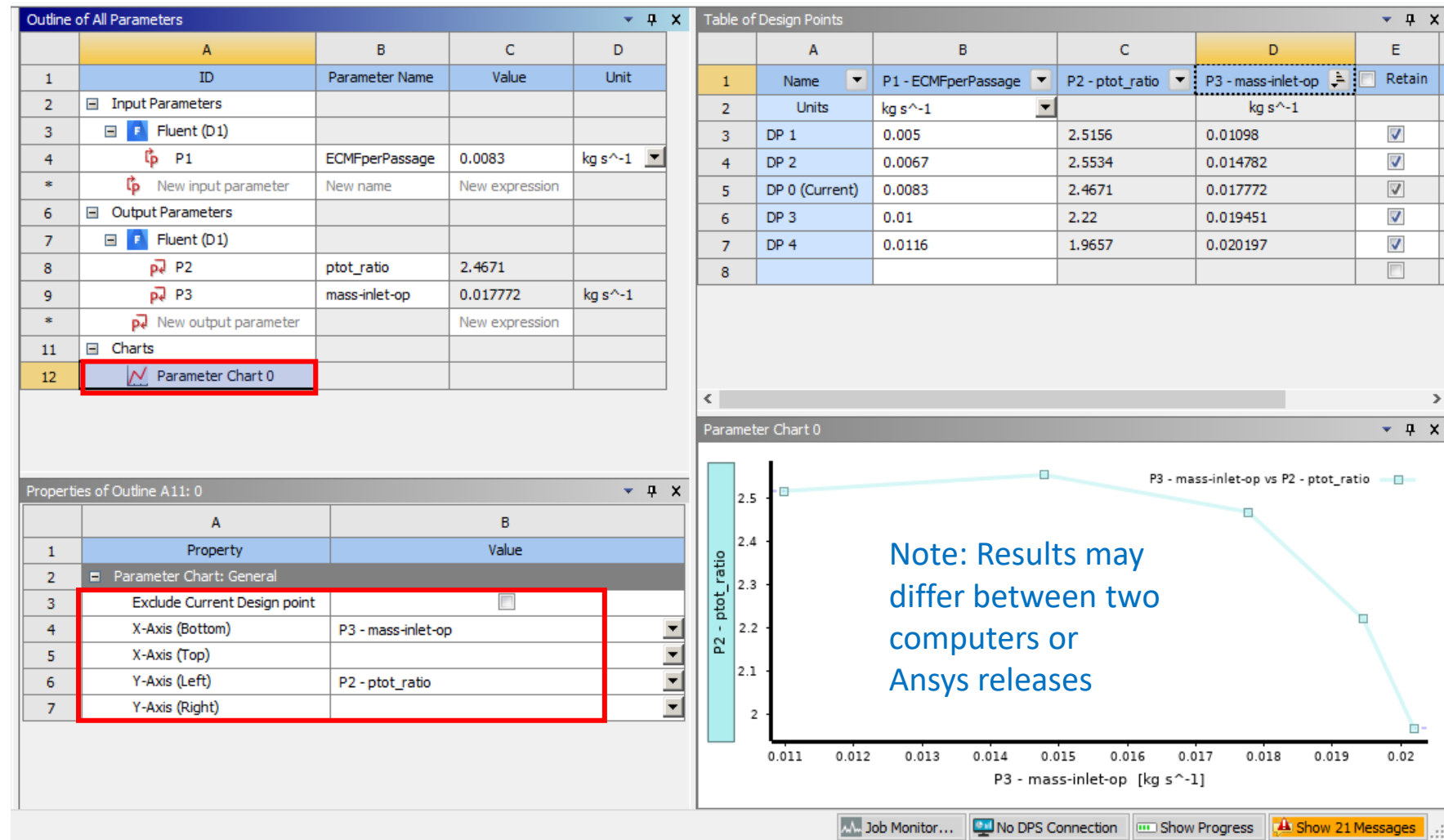
Summary

- This workshop has covered:
  - Setting up a single rotating component for a radial compressor impeller
    - Defining a rotating frame
    - Applying rotational periodicity
    - Using Exit Mass Flow Correction as outlet boundary condition
    - Creating input and output Workbench parameters
  - Solving and monitoring convergence
    - Creating Named Expressions
  - Visualizing the pressure distribution on the impeller walls
  - Creating a speedline using parameters and design points in Workbench

Ansys

End of presentation