

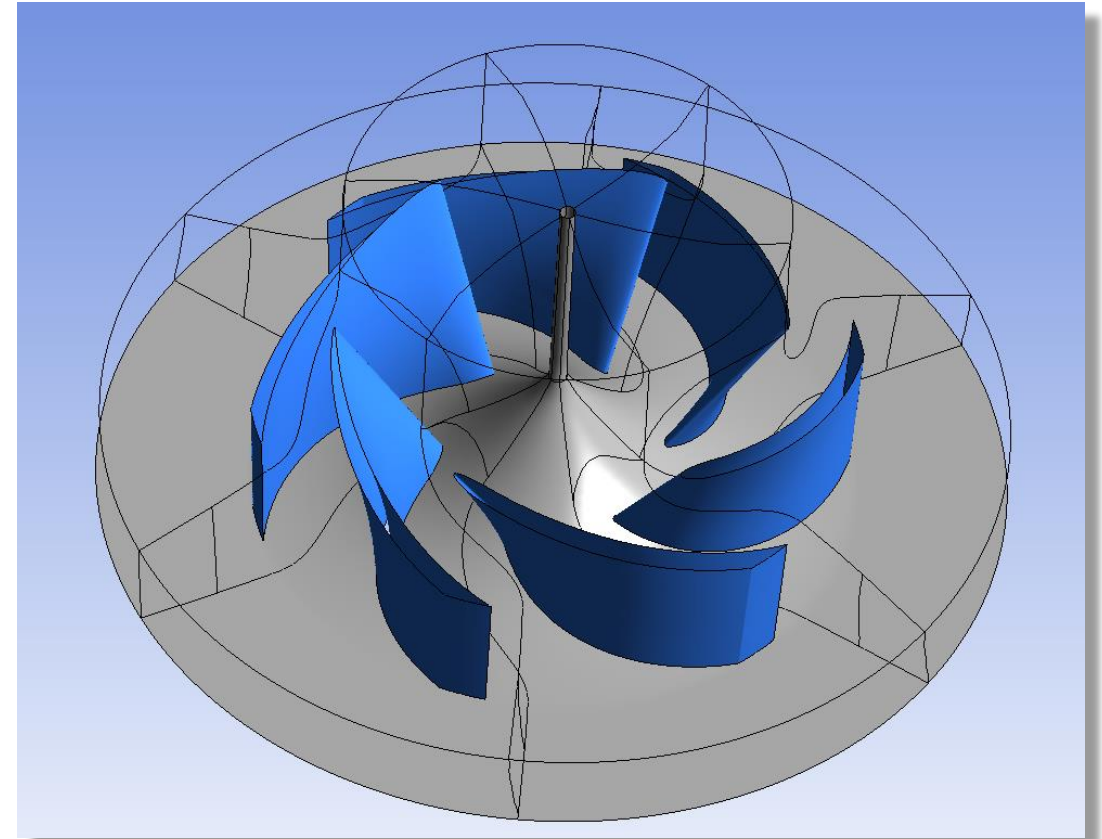
Workshop 02.1: Pump Simulation

Release 2020 R2



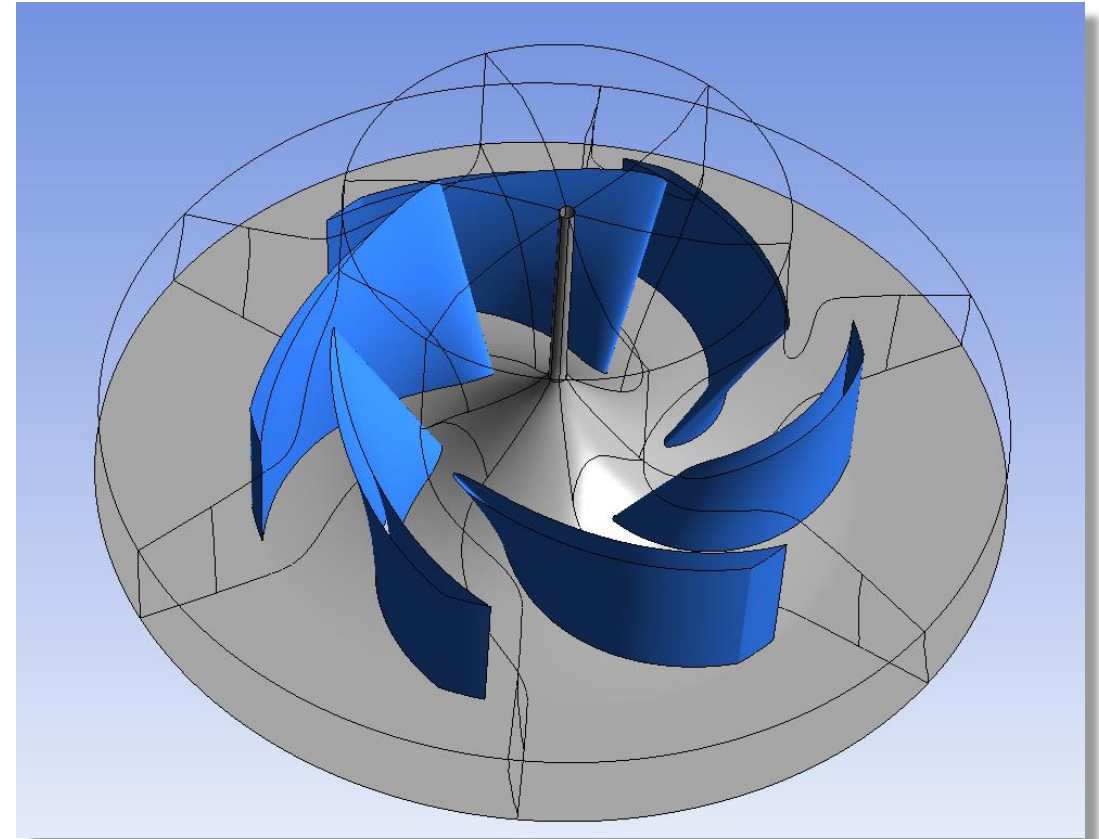
/ Introduction

- Workshop Description:
 - This Workshop deals with the Fluent setup and solution for a pump impeller
- Learning Aims:
 - Setting up a single rotating component
 - Defining a rotating frame
 - Applying rotational periodicity
 - Solving and monitoring convergence
 - Visualizing the pressure distribution on the impeller walls



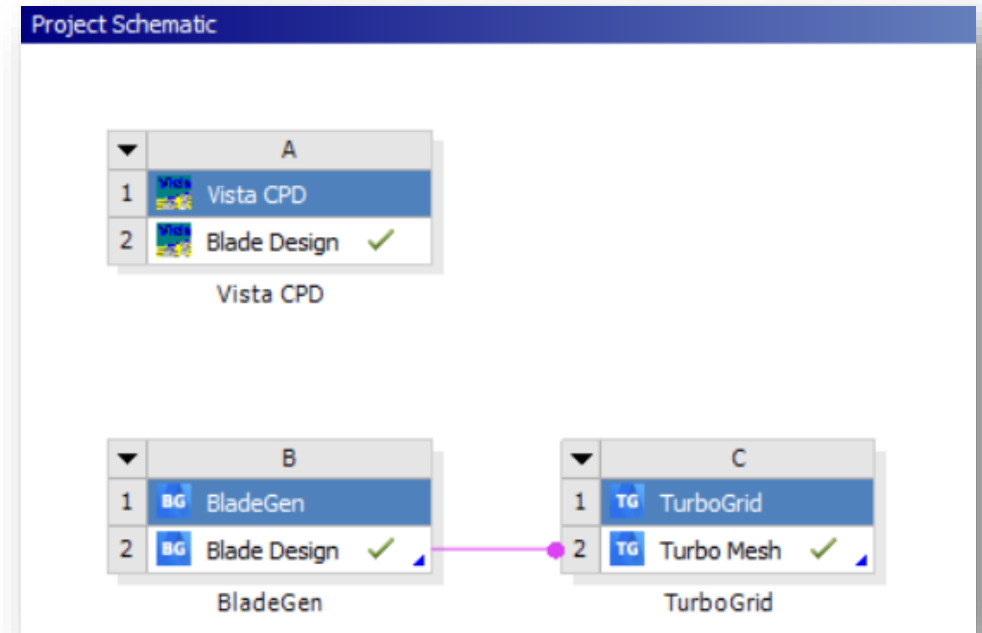
/ Pump Model

- Single rotating component
 - A moving reference frame is used to solve rotating components
 - As all blades are identical, we can reduce the problem size by modelling a single blade passage with periodic boundaries
- Pump data
 - Fluid = Water
 - Speed = 2000 rpm
 - Number of Blades = 6
 - Flow Rate = 83.76 kg/s
 - Flow Rate for one passage = $83.76/6 \text{ kg/s} = 13.96 \text{ kg/s}$
 - Axis of rotation = z-axis



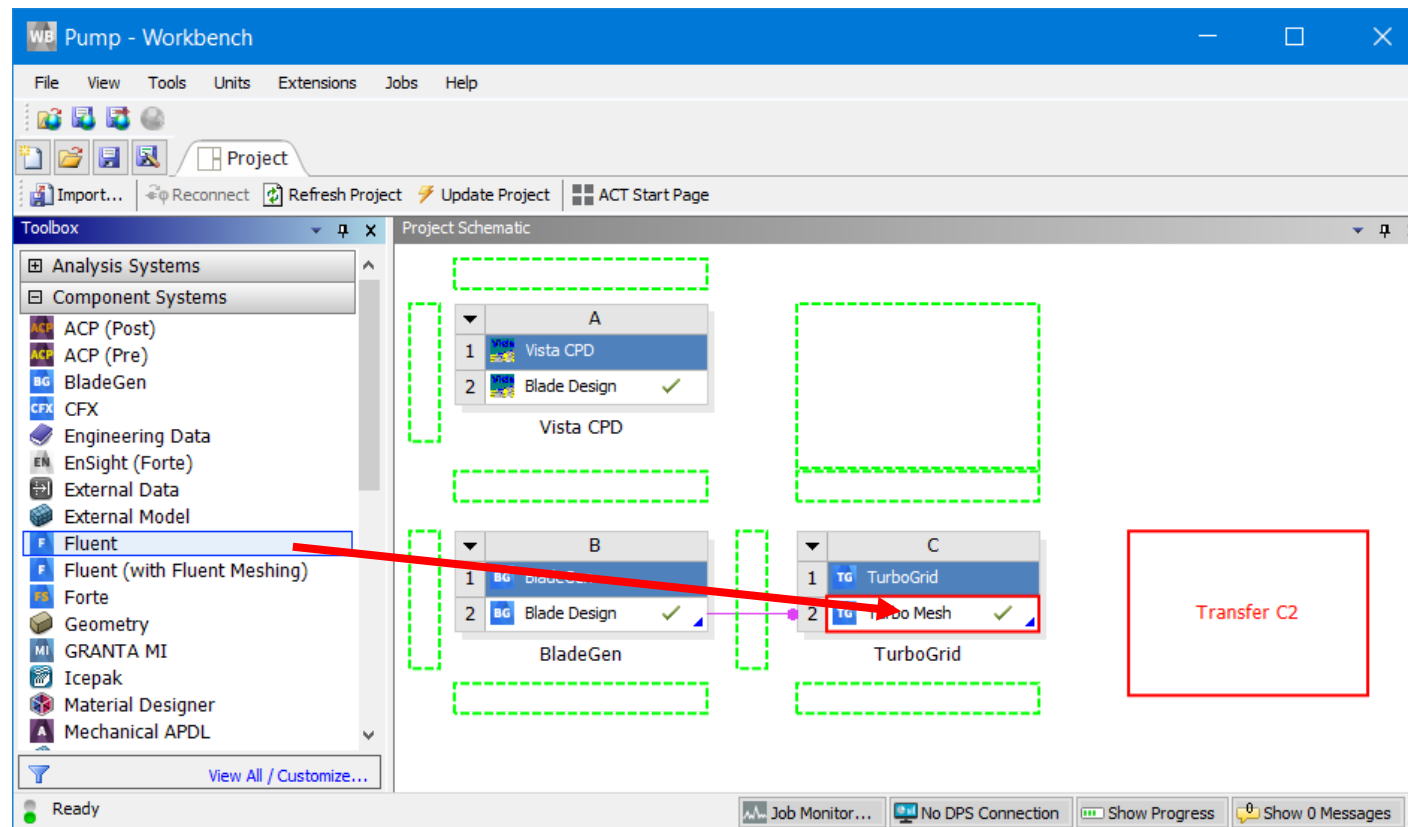
/ Load Workbench Project

- The geometry has been created using Vista CPD and the mesh has been created using TurboGrid
- The geometry and mesh for the pump are provided in a Workbench archive
- Open Workbench
 - 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*



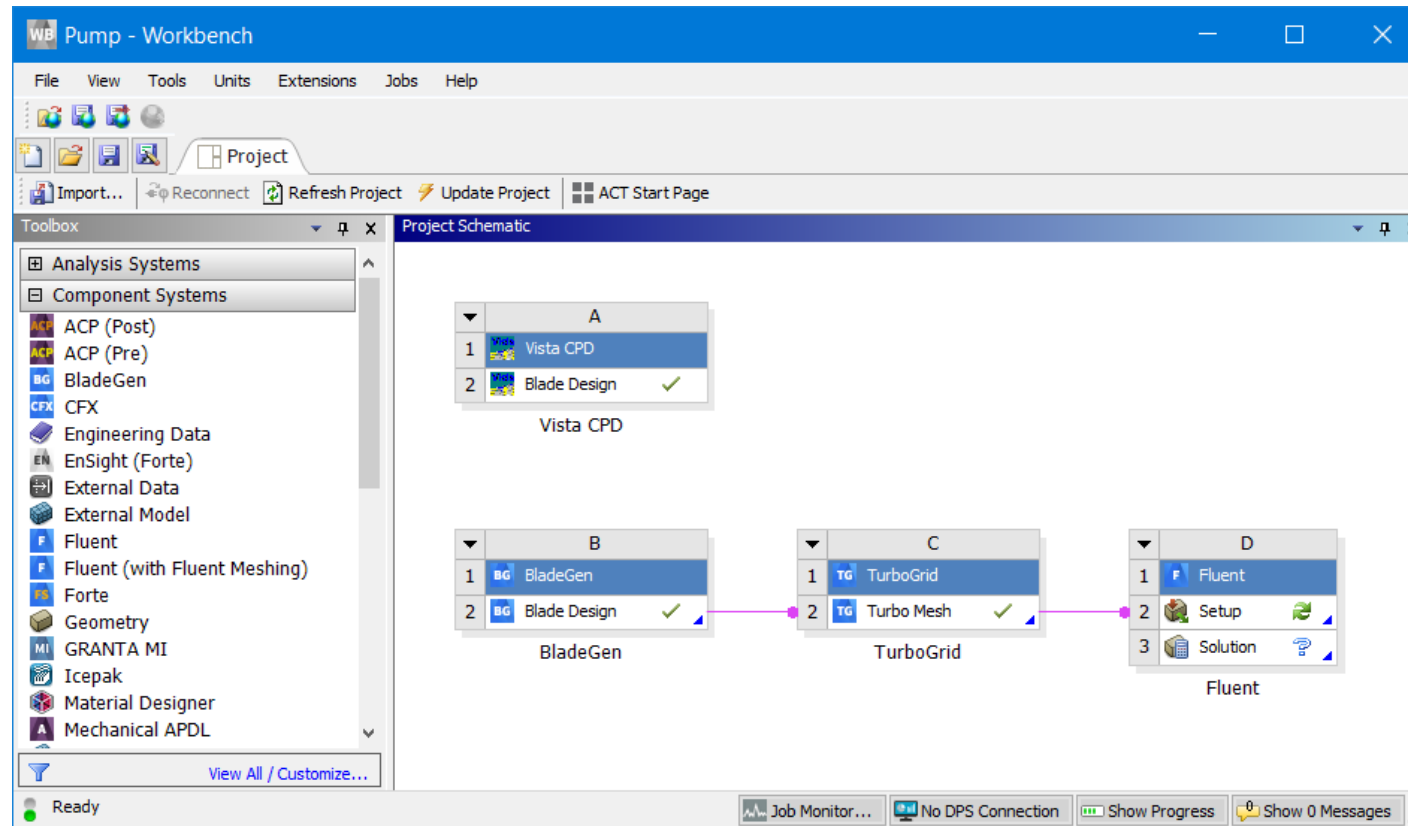
Create a Fluent Component

- 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



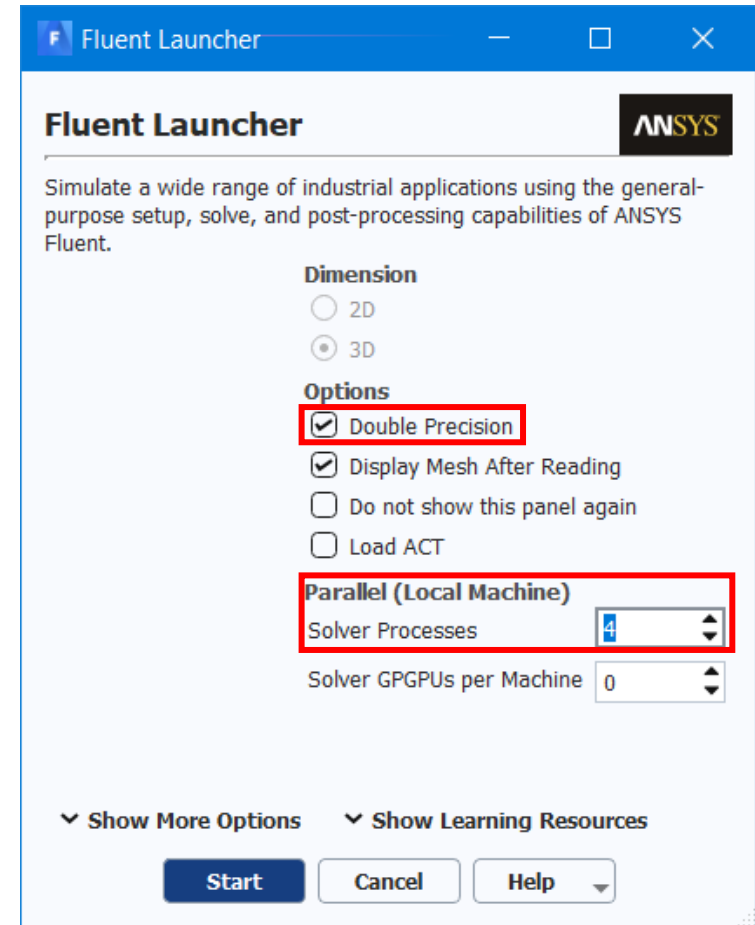
Launch Fluent

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



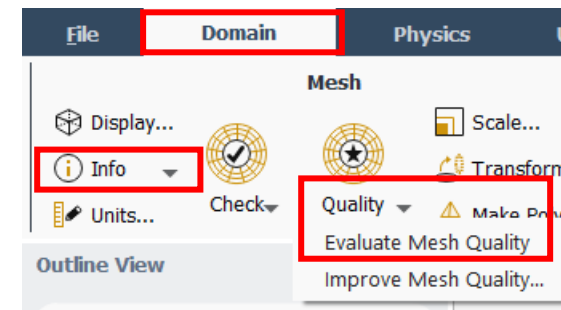
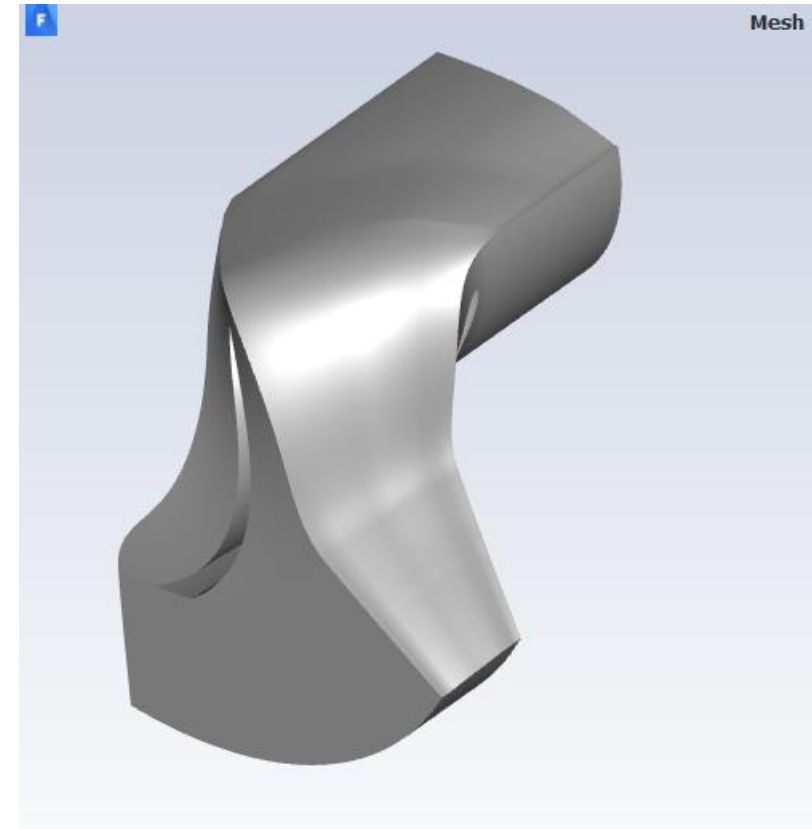
/ Launch Fluent

- In the *Fluent Launcher* select *Double Precision*
 - Double Precision is recommended in most turbomachinery simulations due to the usual high aspect ratios in the boundary layer (order 1000 and more...)
 - The Maximum Aspect Ratio for the given mesh is 1.18539e+03 (see next slide)
- Set the number of Processes for Parallel to 4
 - The mesh size for this case is approximately 250,000 cells (see next slide)
 - If you have enough Parallel licenses and more than 4 cores available, you may set up to 12 Processes (so that each Processor is solving for not less than 20,000 cells)



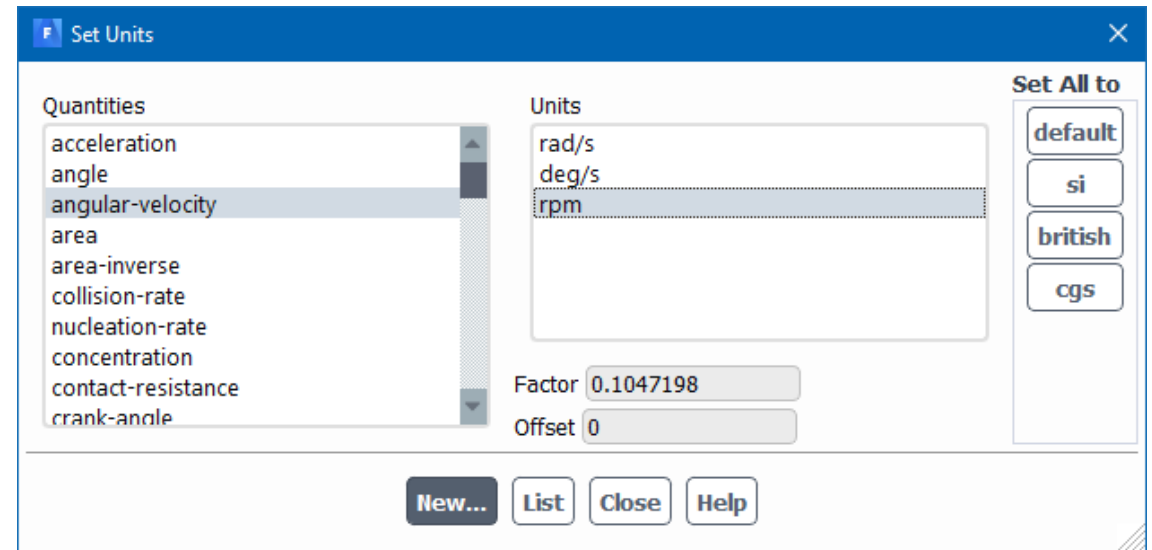
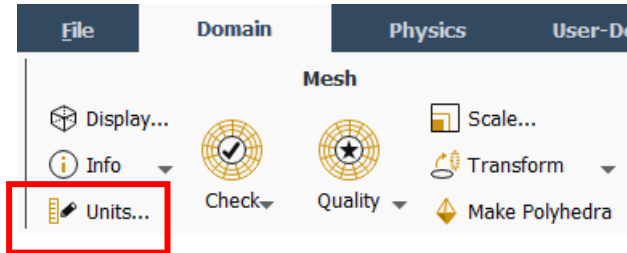
Fluent

- In the Fluent window, you should see the single passage geometry of the pump as shown on the right
- The mesh corresponds to a 60-degrees 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 250,000
 - *Quality* > *Evaluate Mesh Quality* will show you a *Maximum Aspect Ratio* of $1.18539e+03$
 - This justifies the choice of starting Fluent in Double Precision



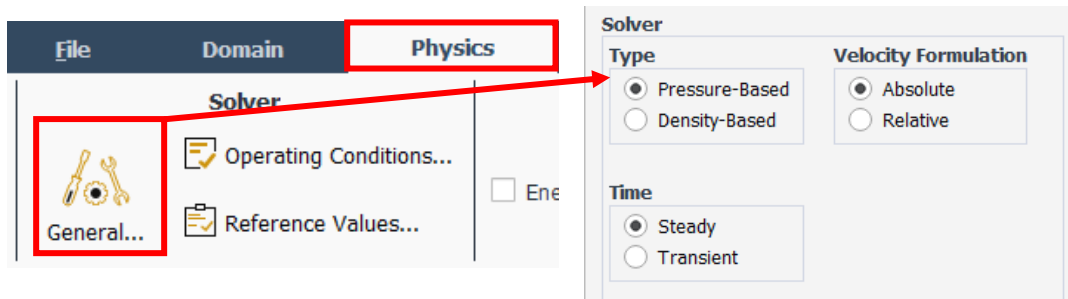
/ General Settings

- The pump rotational speed is given in rpm
- Set the units for angular velocity to rpm
- Click *Close*

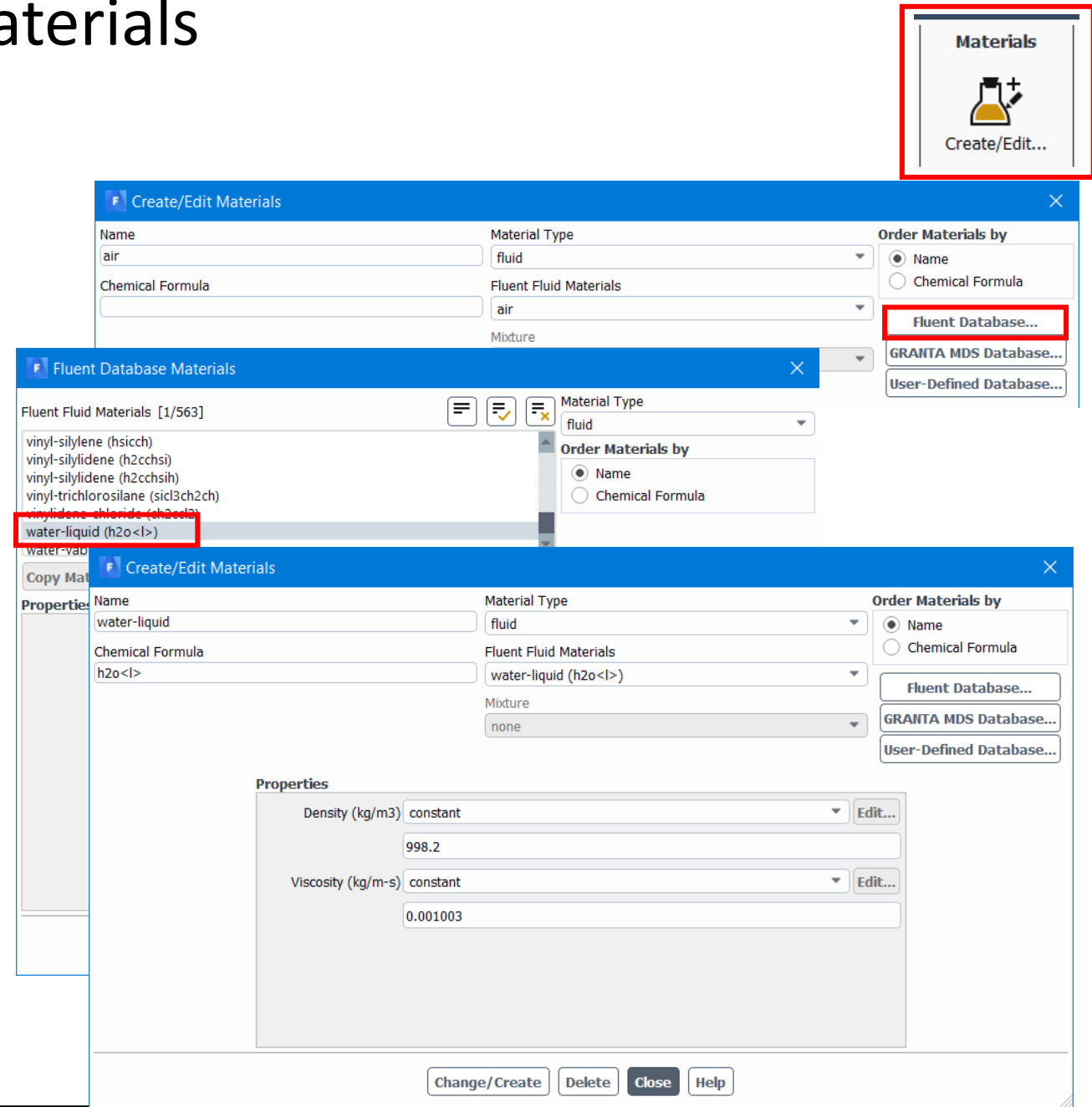


Physics: General Solver and Materials

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

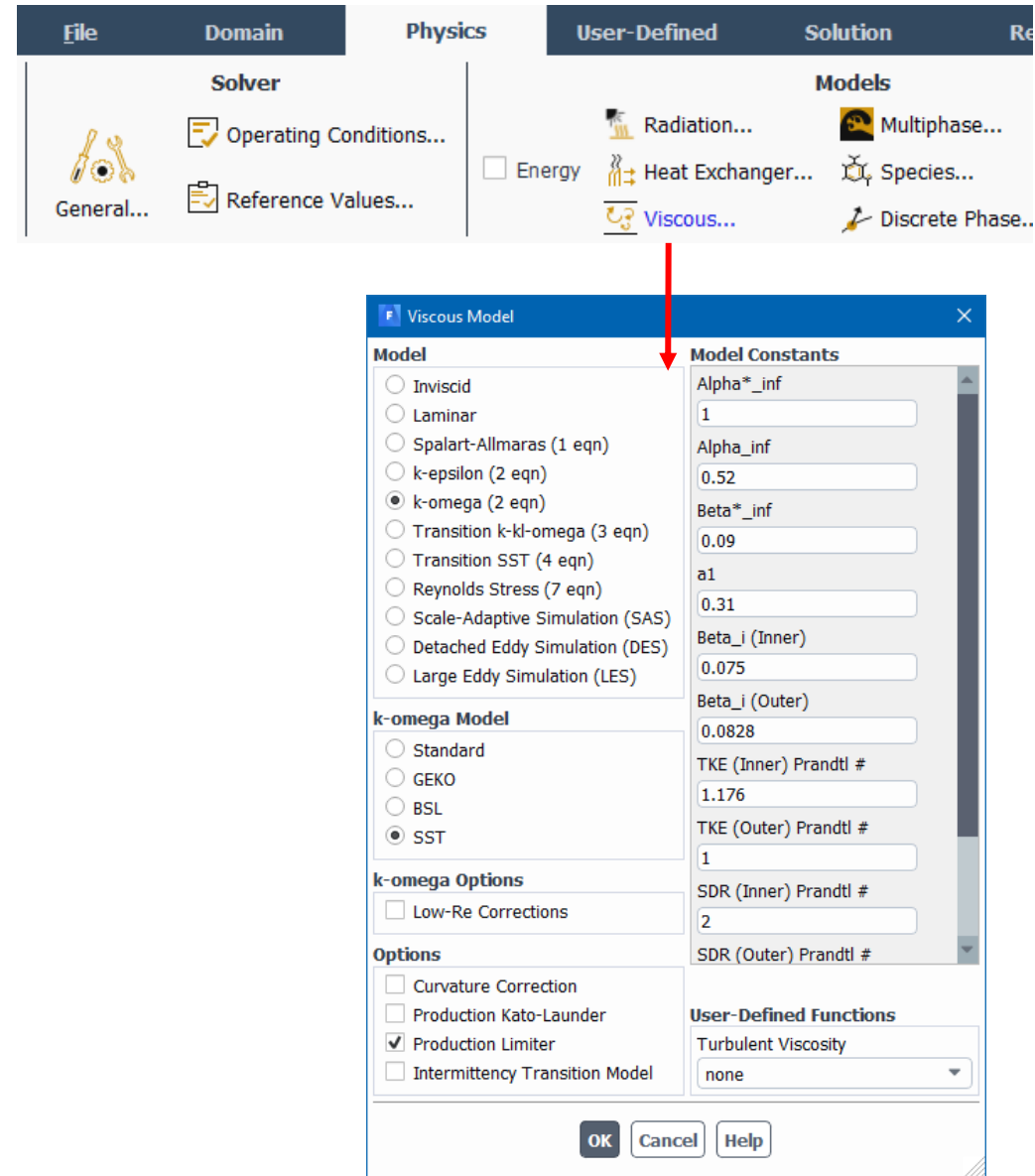


- Make Water material available:
 - From the *Fluent Database...* select water-liquid(h2o<l>) and click *Copy* and *Close*
 - Click *Change/Create* and close the *Create/Edit Materials* dialog box.



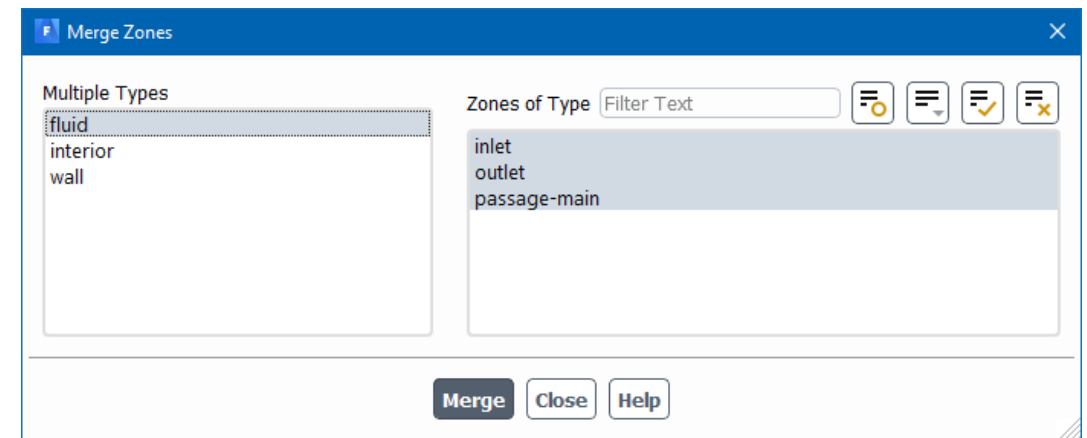
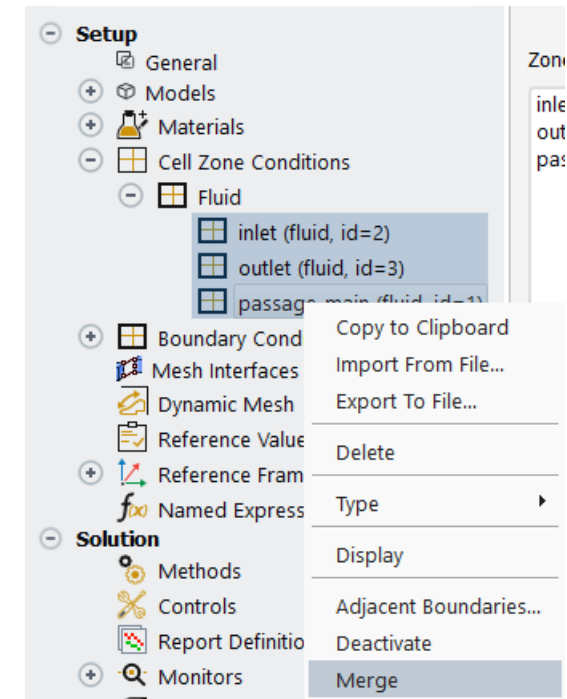
Physics: Turbulence Model

- Keep the default *SST k-omega Viscous Model*
- The *SST k-omega model* (default *Viscous Model* starting with 2020 R1) is the recommended turbulence model for turbomachinery simulations



Merge the 3 Cell Zones

- In the *Outline View* expand *Cell Zone Conditions* and then *Fluid*
 - You can see three zones: *inlet*, *outlet*, and *passage-main*
 - These names were automatically created in Turbogrid
- Merge the three fluid zones into a single zone
 - Select all 3 zones (holding down Ctrl) and *RMB* > *Merge*
- You can see in the *Outline View* that now only a single zone is present, *inlet*
- Rename it to *impeller*
 - *RMB* on the *inlet* cell zone in the *Outline*, select *Edit*, type in the new name and click *Apply*



Physics: Cell Zone Conditions

- Edit the *impeller* cell zone
 - Select *water-liquid* 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 2000 (rpm)
 - Sign verification: If you place your right thumb to point as the z-axis, your fingers are curling (in this case) to the same direction with the rotation direction of the pump blade. Therefore, the *Rotational Velocity* was set to a positive number

Fluid

Zone Name
impeller

Material Name water-liquid Edit...

☒ Frame Motion ☐ 3D Fan Zone ☐ Source Terms

☐ Mesh Motion ☐ Laminar Zone ☐ Fixed Values

☐ Porous Zone

Reference Frame Mesh Motion Porous Zone 3D Fan Zone Embedded LES Reaction Source Terms Fixed Values Multiphase

Relative Specification UDF
Relative To Cell Zone absolute Zone Motion Function none

Rotation-Axis Origin
X (m) 0
Y (m) 0
Z (m) 0

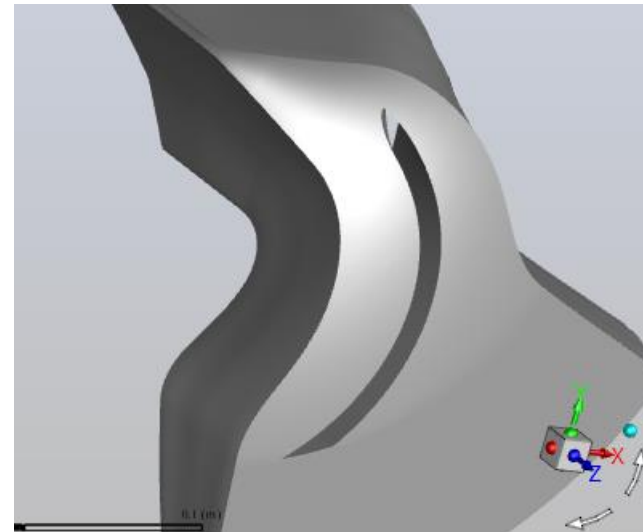
Rotation-Axis Direction
X 0
Y 0
Z 1

Rotational Velocity
Speed (rpm) 2000

Translational Velocity
X (m/s) 0
Y (m/s) 0
Z (m/s) 0

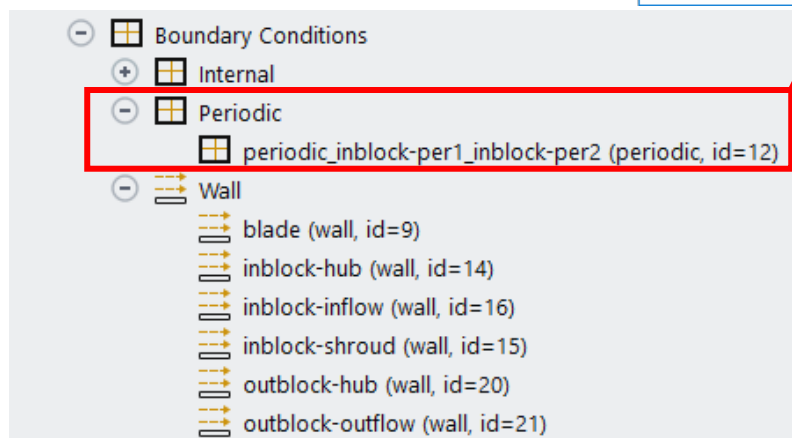
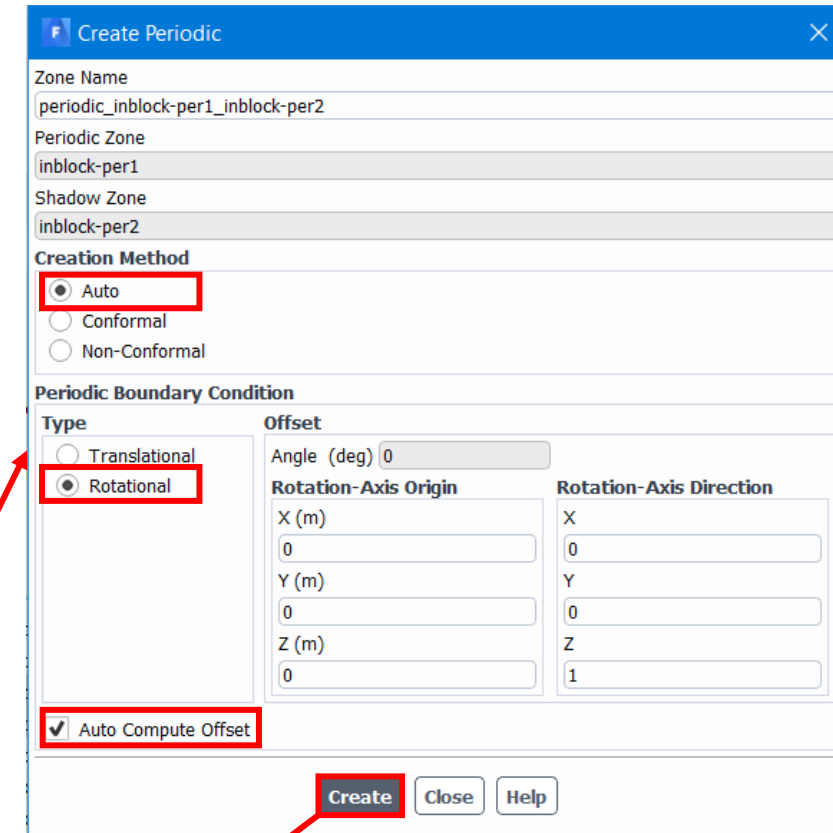
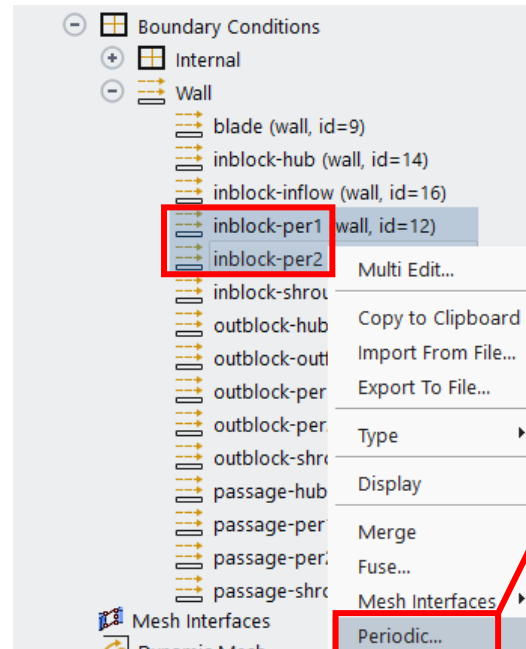
Copy To Mesh Motion

Apply Close Help



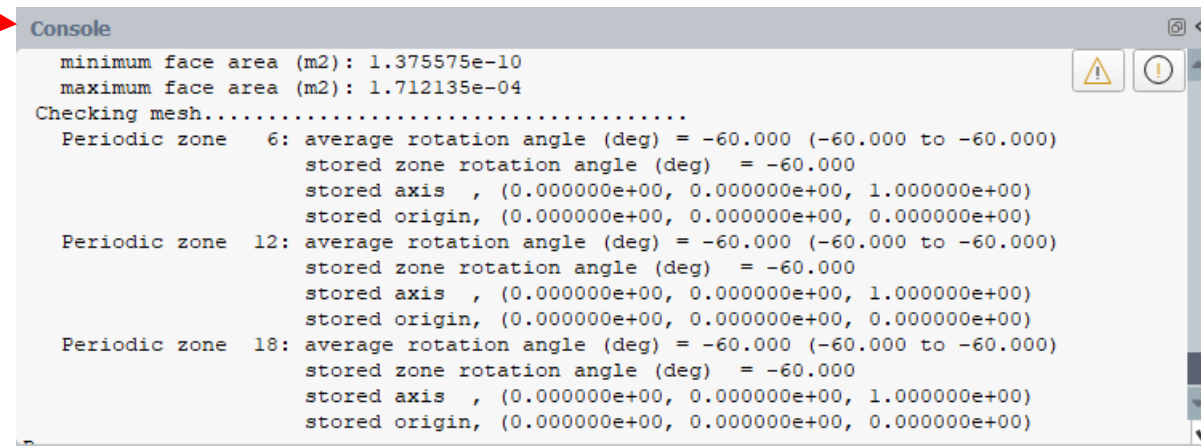
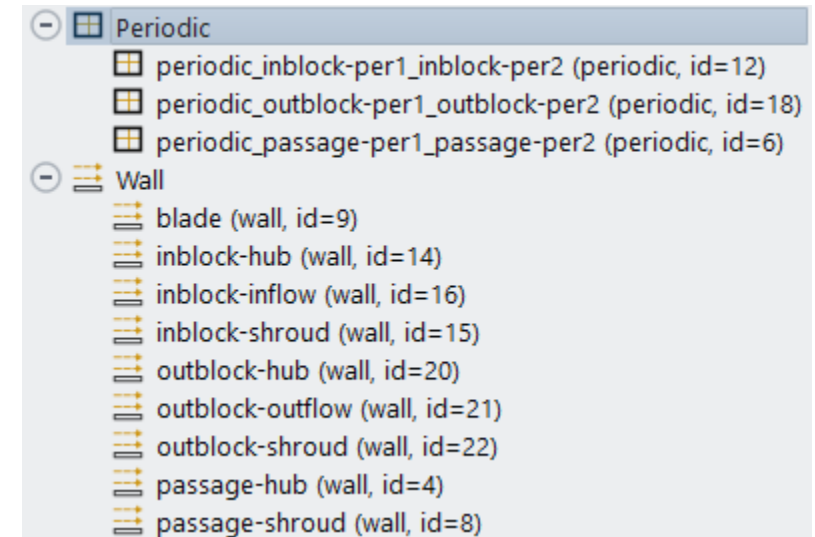
Create Rotational Periodic Zones

- In the *Outline View* expand the *Wall* branch
- Select *inblock-per1* and *inblock-per2*
 - Use *Ctrl + RMB* for multiple selections
- *RMB* > *Periodic*
- Select *Rotational* and leave all the rest to default values
 - Note that the *Rotation-Axis Direction* is 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 17 deleted
Created a conformal periodic boundary.
- The periodic pair is placed under the newly created *Periodic* branch in the *Outline*



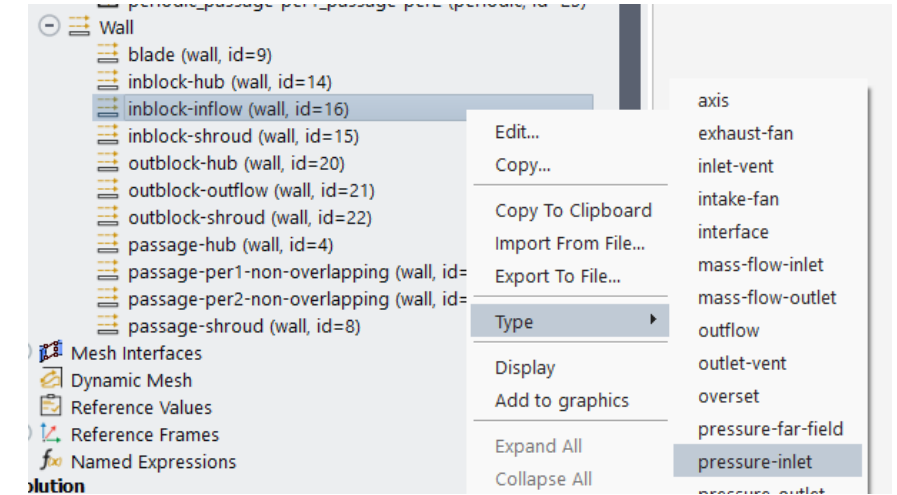
Create Rotational Periodic Zones (2)

- 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 mesh 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
 - The two successfully created periodic zones show a rotation angle of -60.000 degrees
 - Manually give an offset of -60.000 degrees for this periodic boundary

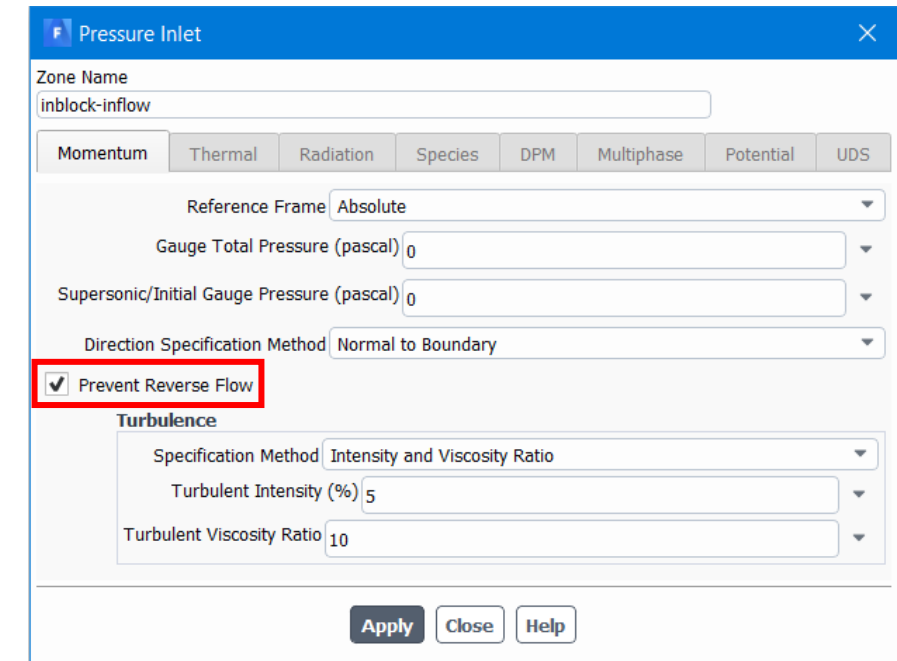
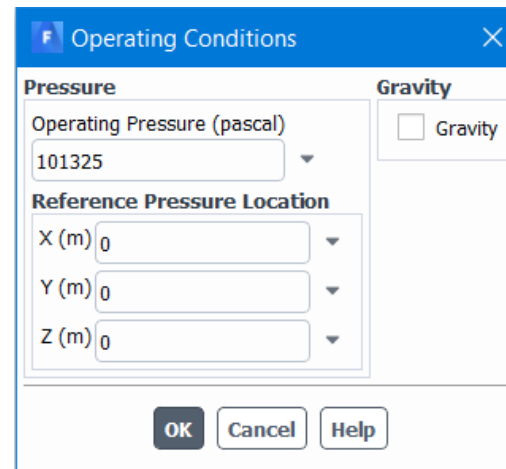


Boundary Conditions: Inlet

- Set the boundary conditions for *inblock-inflow* zone
 - RMB on *inblock-inflow* and set *Type* to *pressure-inlet*
 - Check *Prevent Reverse Flow*
 - We will use a Gauge Total Pressure of 0 (pascal) at the inlet (default value)
 - Accept all remaining defaults and click Apply

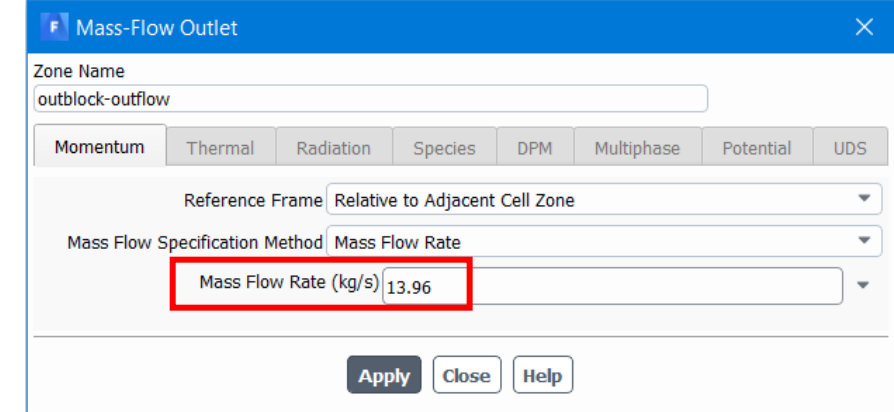
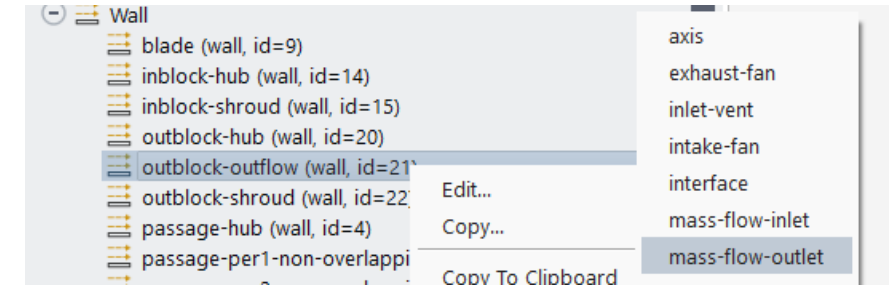


Note for Operating Pressure:
This is an incompressible flow case. Therefore, the Operating Pressure may be left to the default value of 101325 (pascal) and the Gauge Total Pressure at the inlet is given as 0 (pascal)



Boundary Conditions: Outlet

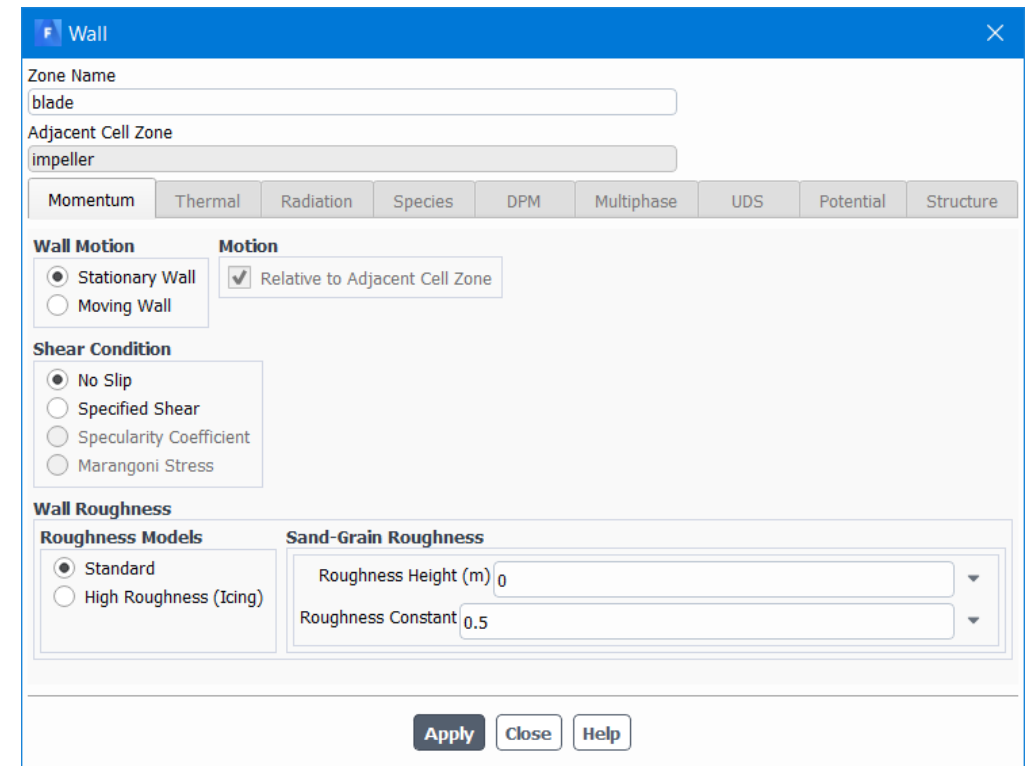
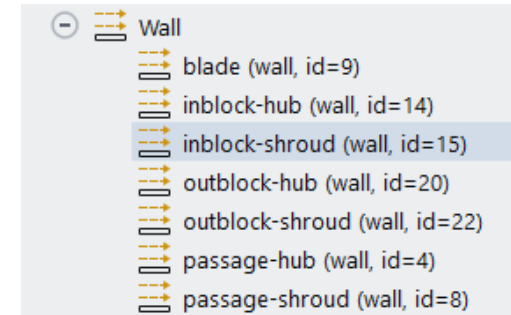
- Set the boundary conditions for *outblock-outflow* zone
 - *RMB* on *outblock-outflow* and set *Type* to *mass-flow-outlet*
 - Set Mass Flow Rate = 13.96 (kg/s)
 - This value corresponds to the mass flow rate for one passaged and was given in slide 3



Boundary Conditions: Walls

- The remaining zones under *Wall* in the *Outline View* correspond to walls which are stationary in the Moving Reference Frame of the *impeller* cell-zone
- Edit the boundary condition for *blade*
- The default settings of a *No Slip, Stationary Wall, Relative to Adjacent Cell Zone* is what we want
No need to change this default setting for any of the walls for this case

Note: In some cases a wall in a rotating zone can be stationary in the absolute frame. Example, the shroud casing wall of an unshrouded impeller. Such a wall is seen as a “counter-rotating wall” in the reference frame of the rotating zone. You will learn how to set up such walls in a next workshop



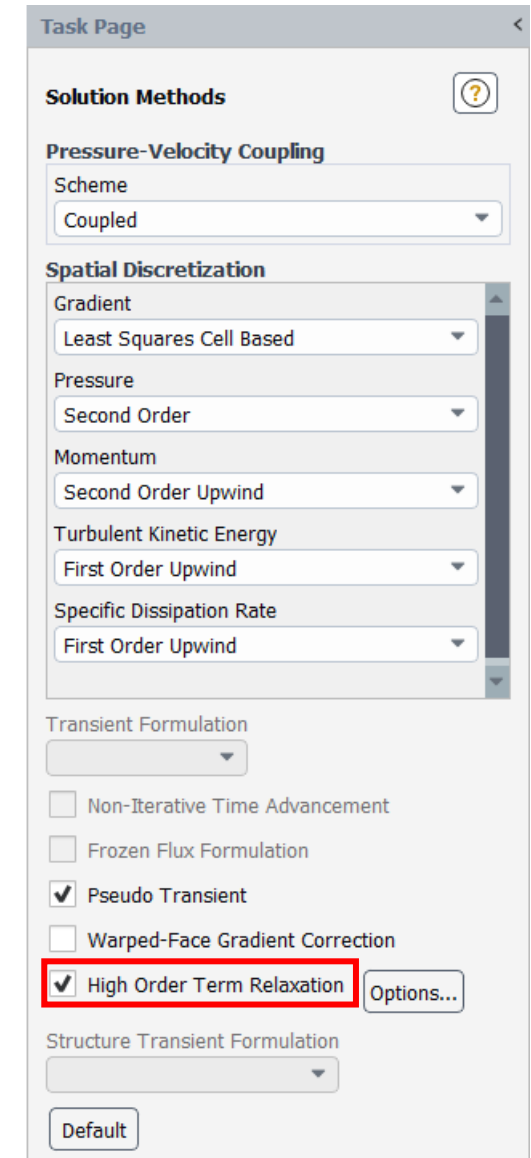
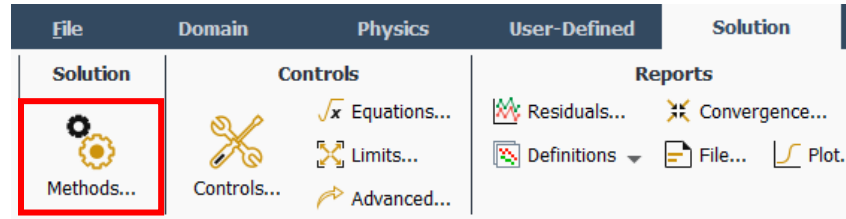
/ Solution: Report Definition

- In the *Solution* tab create a new *Mass-Weighted Average Surface Report Definition* with the following settings:
 - *Name* = *ave-press-out*
 - *Field Variable* = *Total Pressure*
 - *Surfaces* = *outblock-outflow*
 - *Report File* = *checked*
 - *Report Plot* = *checked*

The screenshot shows the 'Surface Report Definition' dialog box. The 'Name' field is set to 'ave-press-out'. The 'Report Type' is set to 'Mass-Weighted Average'. Under 'Options', 'Per Surface' is unchecked, and 'Average Over' is set to '1'. The 'Report Files' section is empty. The 'Report Plots' section is empty. In the 'Create' section, both 'Report File' and 'Report Plot' are checked. The 'Frequency' is set to '1'. 'Print to Console' and 'Create Output Parameter' are unchecked. The 'Field Variable' is set to 'Pressure...'. The 'Surfaces' list includes 'blade', 'inblock-hub', 'inblock-inflow', 'inblock-shroud', 'interface-inblock-outflow', 'interface-outblock-inflow', 'outblock-hub', 'outblock-outflow' (which is highlighted), 'outblock-shroud', 'passage-hub', 'passage-shroud', 'periodic_inblock-per1_inblock-per2', and 'periodic_outblock-per1_outblock-per2'. The 'New Surface' button is at the bottom right. The 'OK', 'Compute', 'Cancel', and 'Help' buttons are at the bottom.

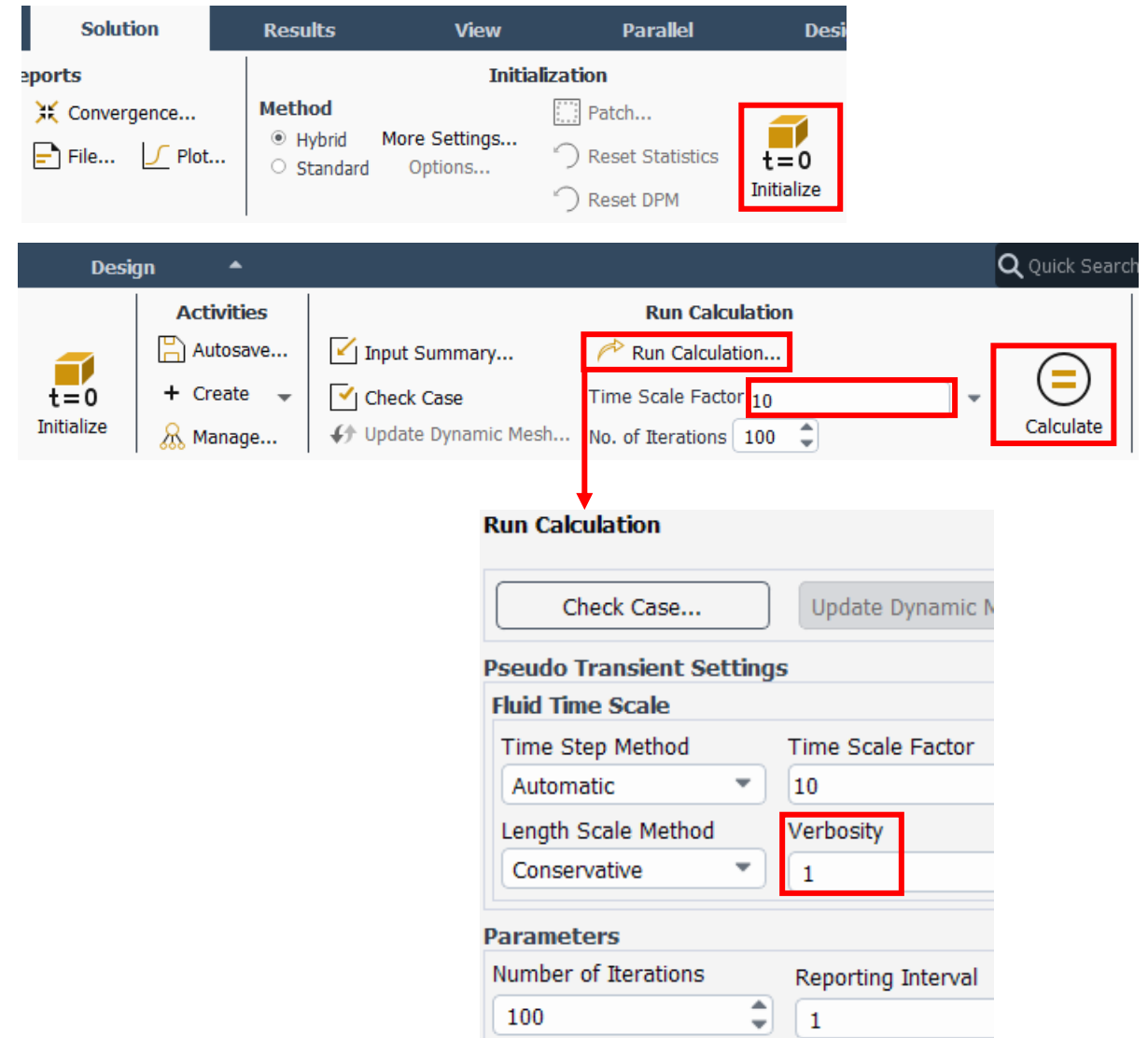
/ Solution: Solution Methods

- Always use the default *Coupled “Pseudo-Transient” Solver*
 - 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)



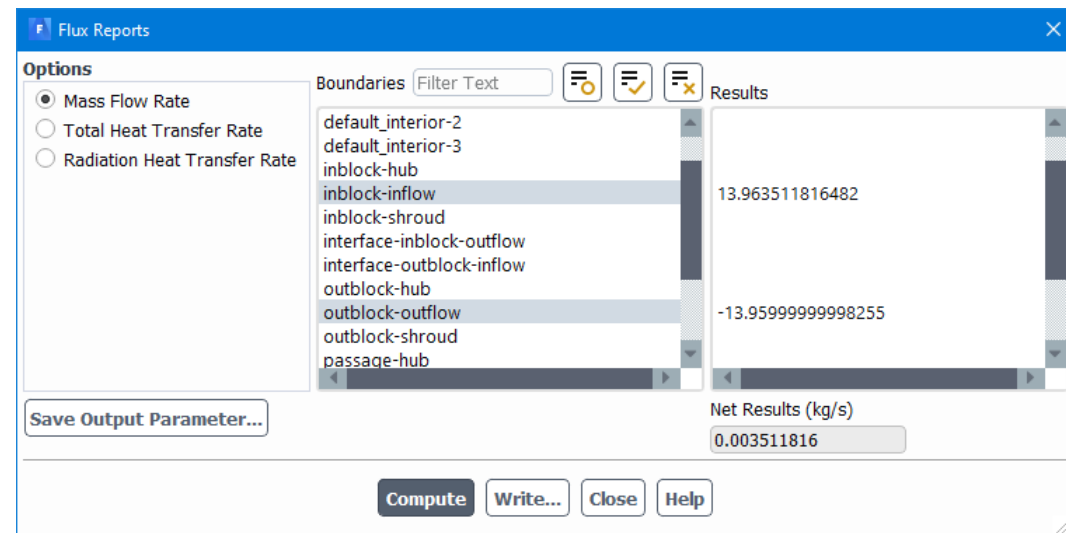
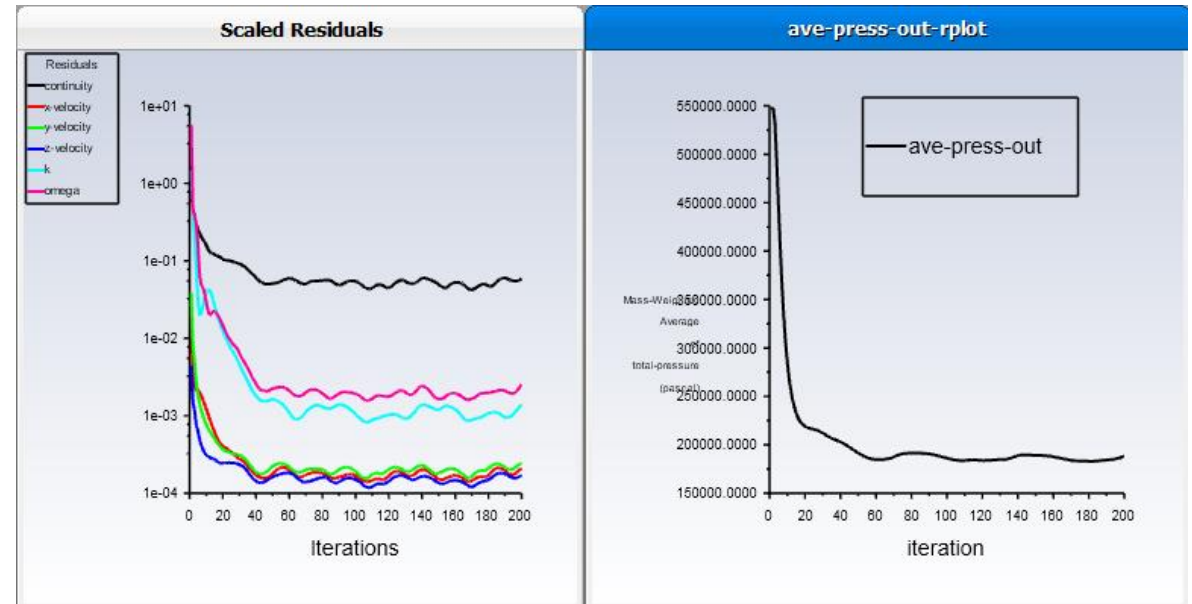
/ Solution: Initialize Solution and Run the Solver

- Perform a default *Hybrid Initialization*
 - This works fine for this case. A best practice initialization procedure will be used in the next workshops of this course
- 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 100
- Click *Calculate*
- Click OK in any Warning popup about Mesh and settings have changed



/ Solver Convergence

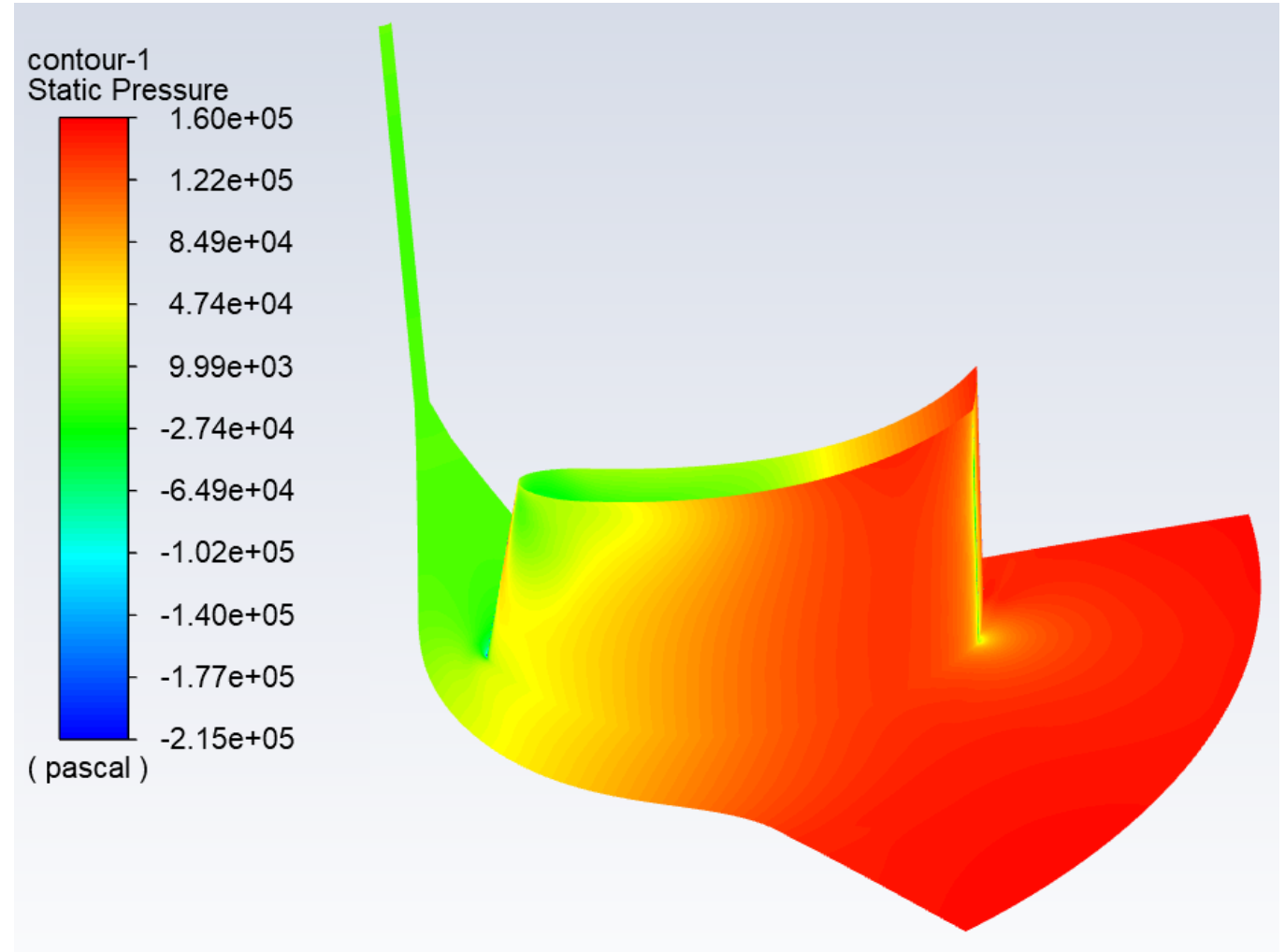
- As the solution progresses we can see that the residuals reduce somewhat; however the case ends after 100 iterations with the residuals and the report plot for average outlet pressure showing a bouncy behaviour
- Running 100 iterations further the convergence behavior is similar
- The mass Flow Rate Flux report shows an imbalance of 0.025 %
- This case is not well converged
- In workshop 03.1 “Pump Analysis using CFD-Post” we will examine why this is happening using CFD-Post



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 will be carried out in workshop 03
- When done, save the Workbench project and exit Fluent



Summary

- This workshop has covered:
 - Setting up a single rotating component
 - Defining a rotating frame
 - Applying rotational periodicity
 - Solving and monitoring convergence
 - Visualizing the pressure distribution on the impeller walls



End of presentation