# ANSYS Fluent Rotating Machinery Modeling

# Workshop 04.2: Axial Fan Stage

Release 2020 R2

Ansys

Introduction

- Workshop Description:
  - This Workshop deals with the Fluent setup and solution for an axial fan stage

- Learning Aims:
  - Setting up a steady stage calculation comprising a rotor and a stator
    - Defining a rotating frame
    - Applying rotational periodicity
    - Creating named expressions and report plots for monitoring the pressure rise and the power consumption of the fan rotor
    - Creating a Mixing Plane, General Turbo Interface
  - Solving and monitoring convergence
  - Visualizing the pressure distribution on the impeller walls and the relative velocity vectors at midspan
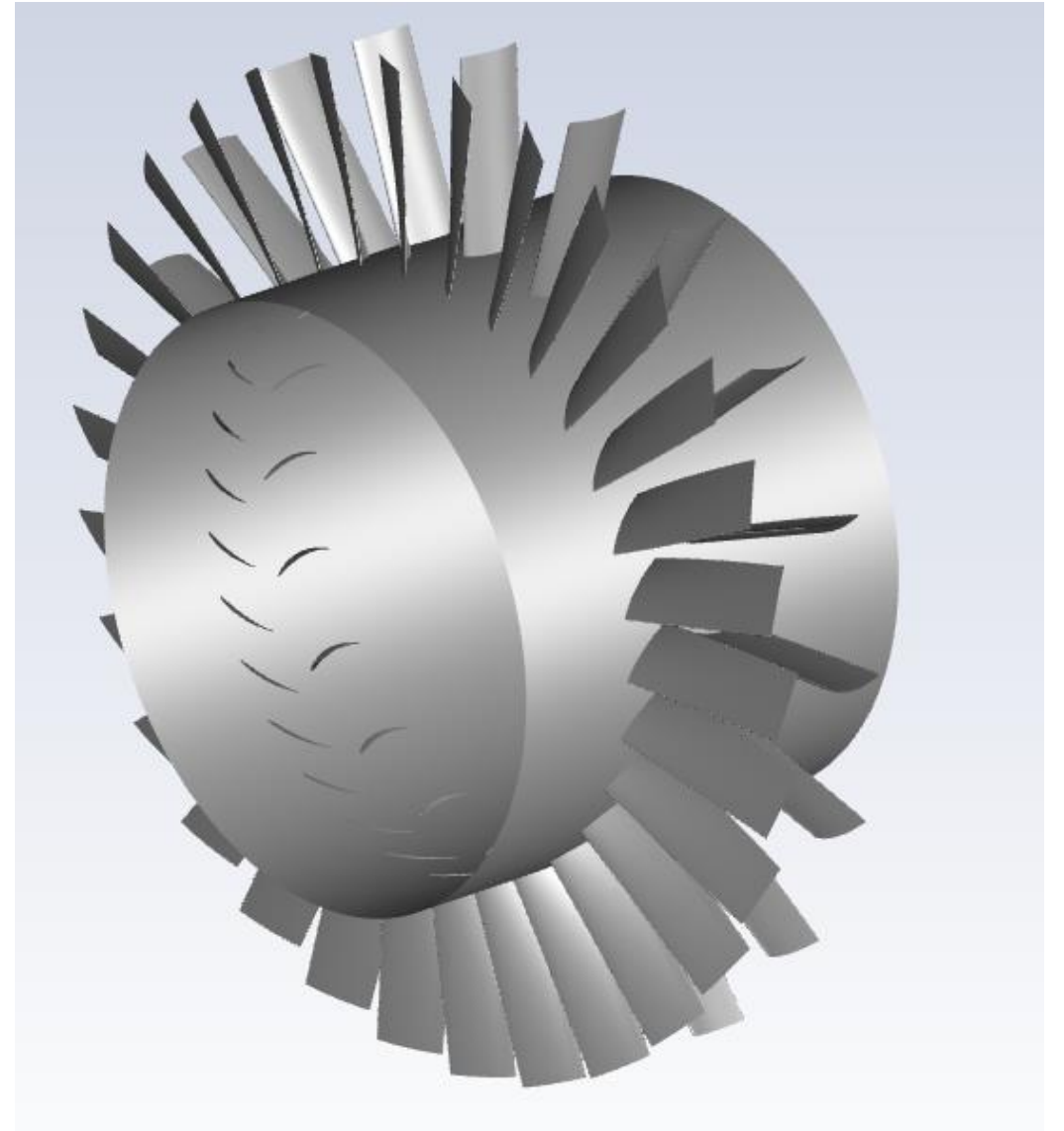
Fan Model

- A rotating component, followed by a stationary 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 for the rotor and for the stator with periodic boundaries

- Fan data
  - Fluid = Air Ideal Gas
  - Operating Pressure = 1 bar = 100,000 Pa
  - Speed = 2880 rpm
  - Number of rotor blades = 30
  - Number of stator blades = 20
  - Axis or rotation = z-axis
  - Conditions
    - Inlet
      - P_t = 1.0 bar, T_t = 288 K
    - Outlet
      - Mass Flow Rate = 0.3 kg/s (one passage)


Stator Blades (20)

Rotor Blades (30)

Ansys
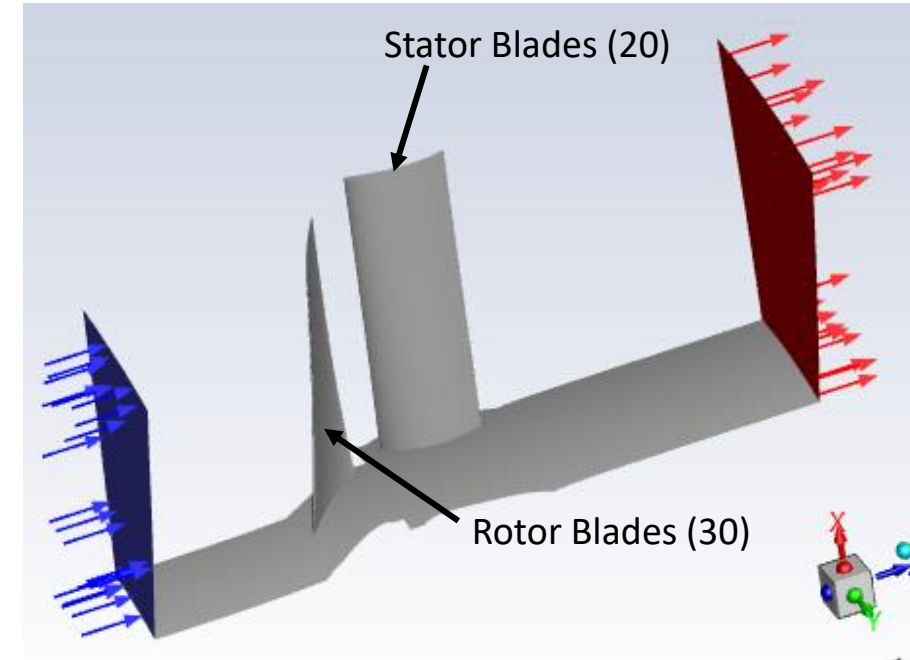
Start Fluent Launcher

- Start *Fluent Launcher* in *Solution* mode

- 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 97,000 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 5 Processes (so that each Processor is solving for not much less than 20,000 cells)
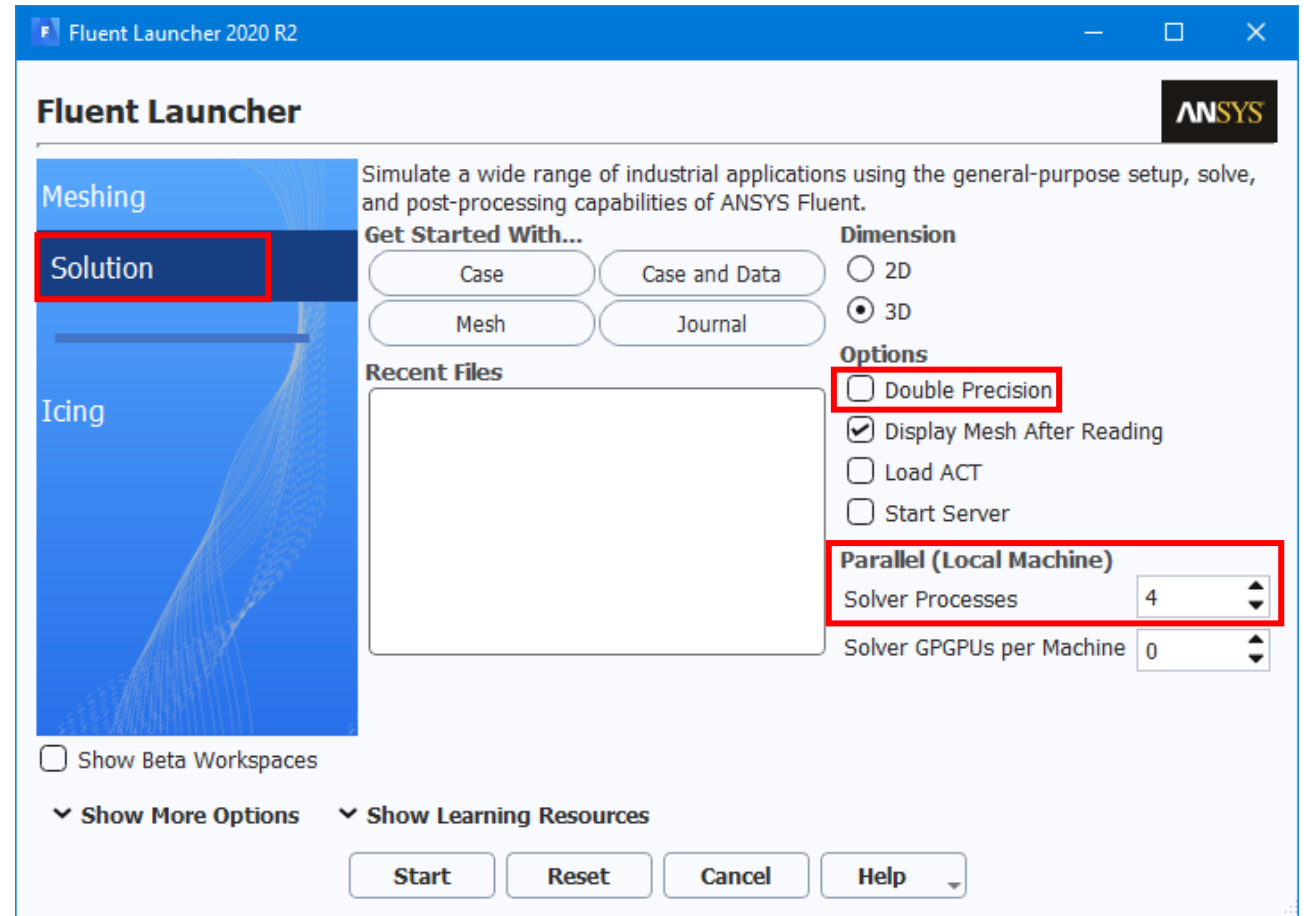
Fluent

- In the Fluent window read the mesh provided with the workshop inputs
  - File>Read>Mesh
  - Browse to file *AxialFanStage.msh*

- In the graphics window, you should see the single passage geometry of the axial fan stage as shown on the right
  - Note that in the image shown, the shroud boundary and one of the periodic boundaries were hidden (*LMB* to select a surface in the graphics window, *RMB>Hide>Selected*)

- The mesh comprises one rotor and one stator passage

- 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 97,000
  - *Quality > Evaluate Mesh Quality* will show you a *Maximum Aspect Ratio* of 5.53e+02 <1000
    - This justifies the choice of starting Fluent in Single Precision
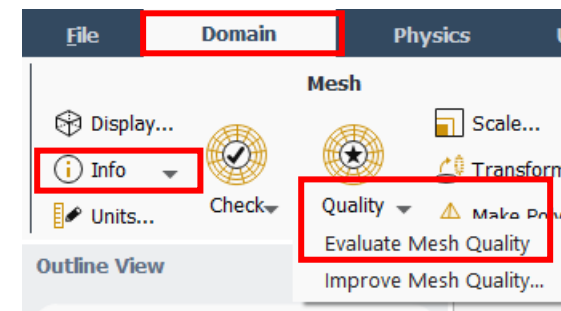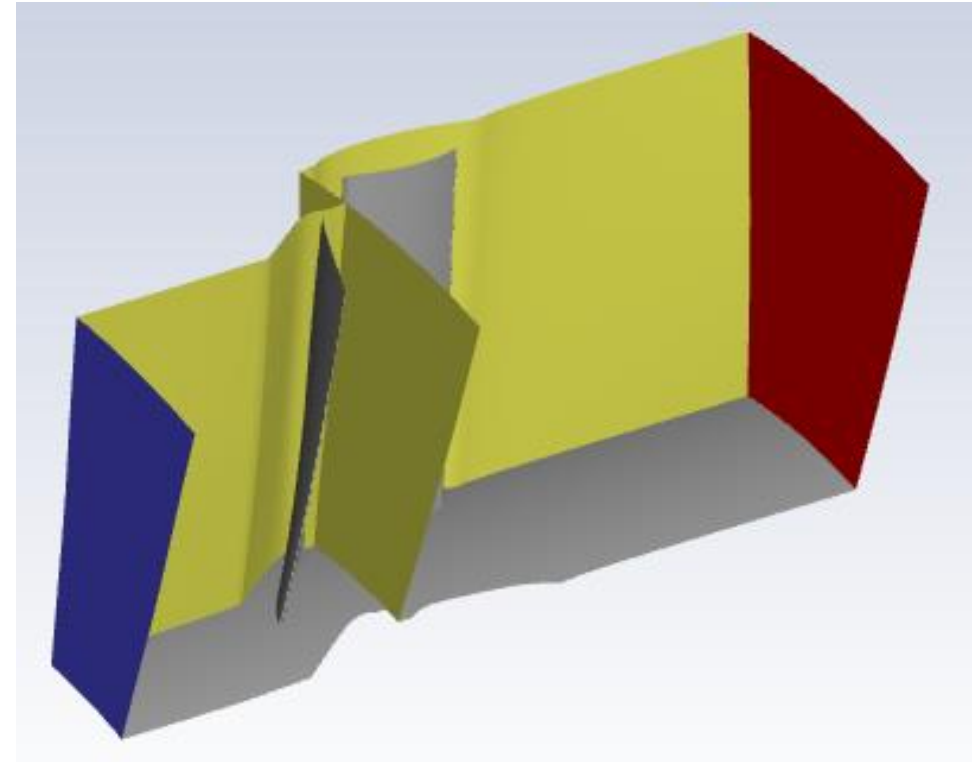
Physics: General & Operating Conditions

- Retain the default solver settings of *Pressure-Based* solver with *Absolute Velocity Formulation*
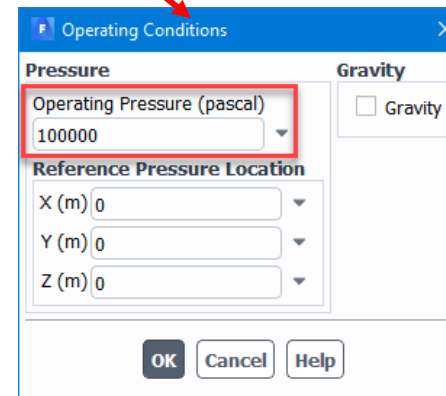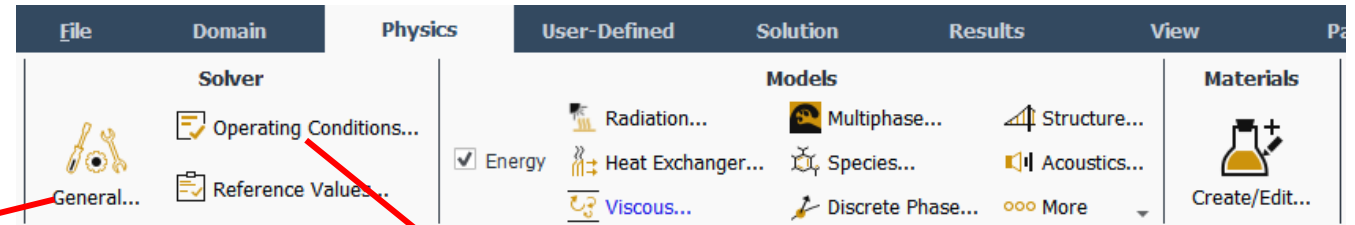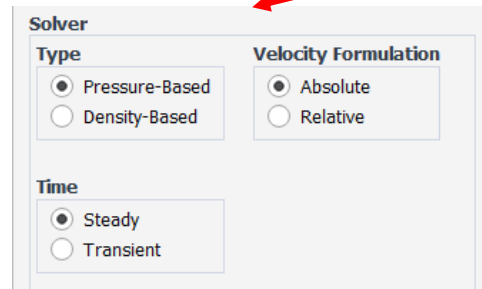
- In the *Operating Conditions* panel set the *Operating Pressure* to 100,000 (Pa)

Physics: Energy and Viscous Model

- Enable the Energy equations

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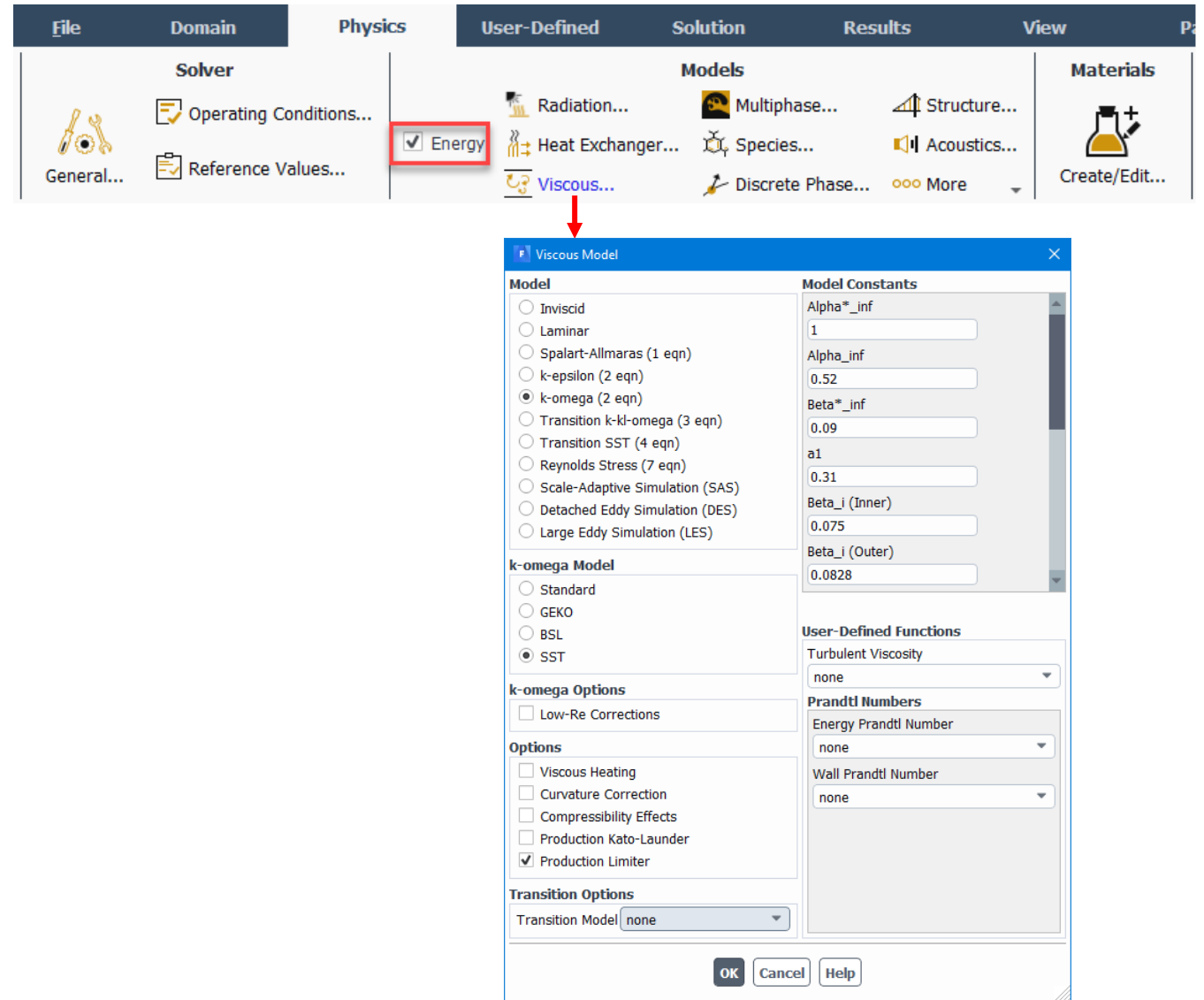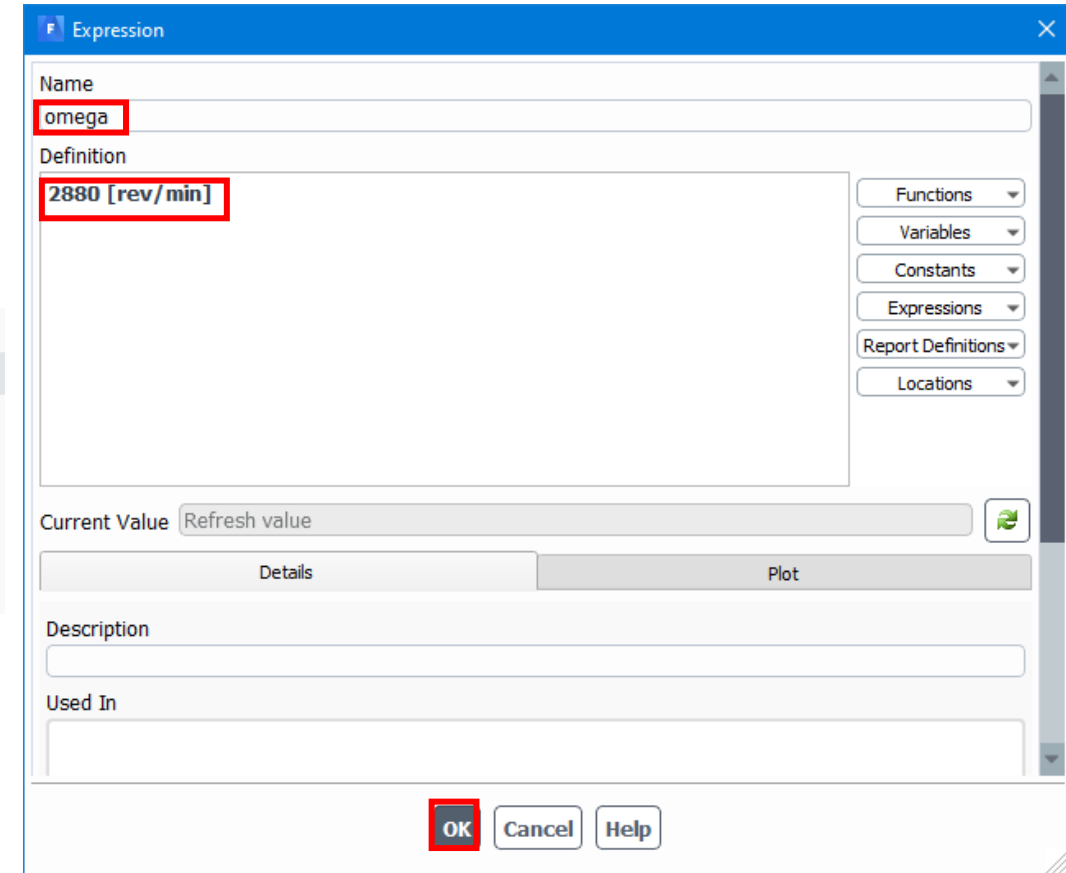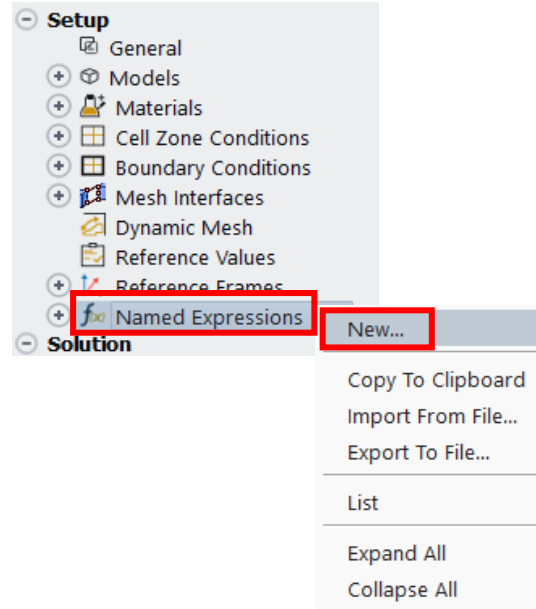
Physics: Materials

- The default material is Air with constant properties
  - We will need to change its Density to be a function of Temperature

- Click *Material > Create/Edit* in the *Physics* tab
  - Select *ideal-gas* from the *Density* drop-down list
 - Click *Change/Create* then *Close*
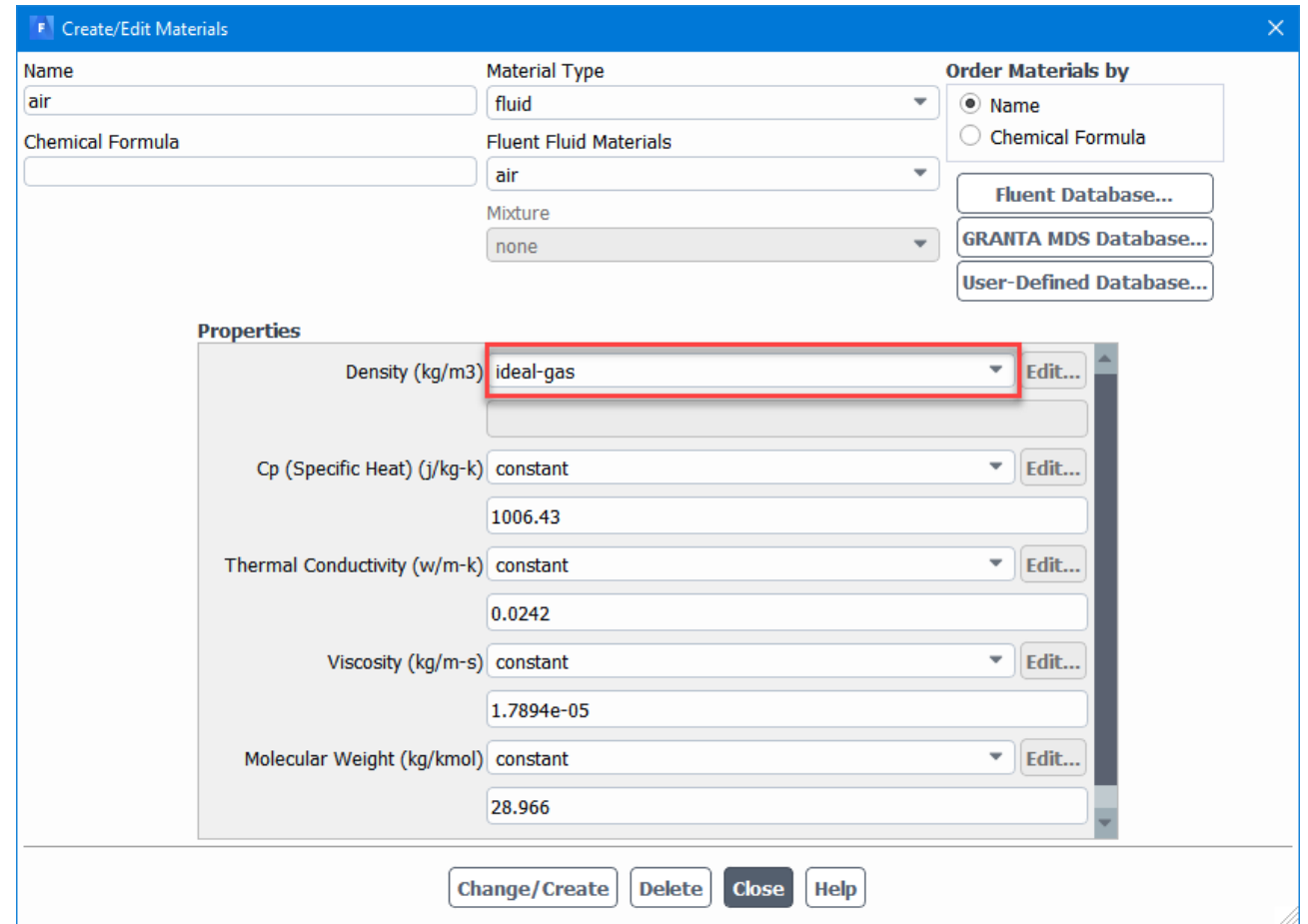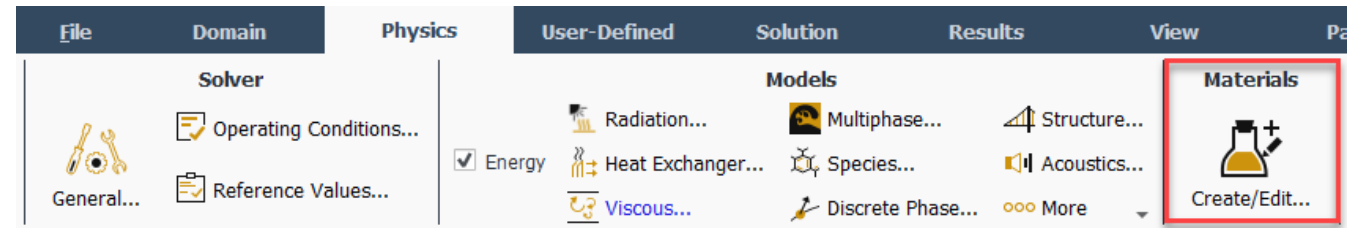
Create a Named Expression for Rotational Speed

- The fan rotor is rotating with a rotational speed of 2880 rpm

- We will need to set this in the conditions for the *r1* cell zone (see next slide)

- We are going to use a *Named Expression* for this

- In the *Outline*, *RMB* on *Named Expressions* and select *New…*

- In the *Expression* panel:
  - enter *omega* under *Name*,
  - *2880 [rev/min]* under *Definition*
 - Click *OK*

Physics: Cell Zone Conditions

- Edit the *r1* cell zone (rotor)
  - 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
  - Using the drop-down list next to *Speed* under *Rotational Velocity,* set this to *expression*
  - In the box next to *Speed*, type *omega* (i.e., the name of the expression created in the previous slide)
    - After typing the first few letters of *omega*, you will see that the name *omega* is highlighted in a blue box. You may left-click on it to select it, or continue typing the complete expression name
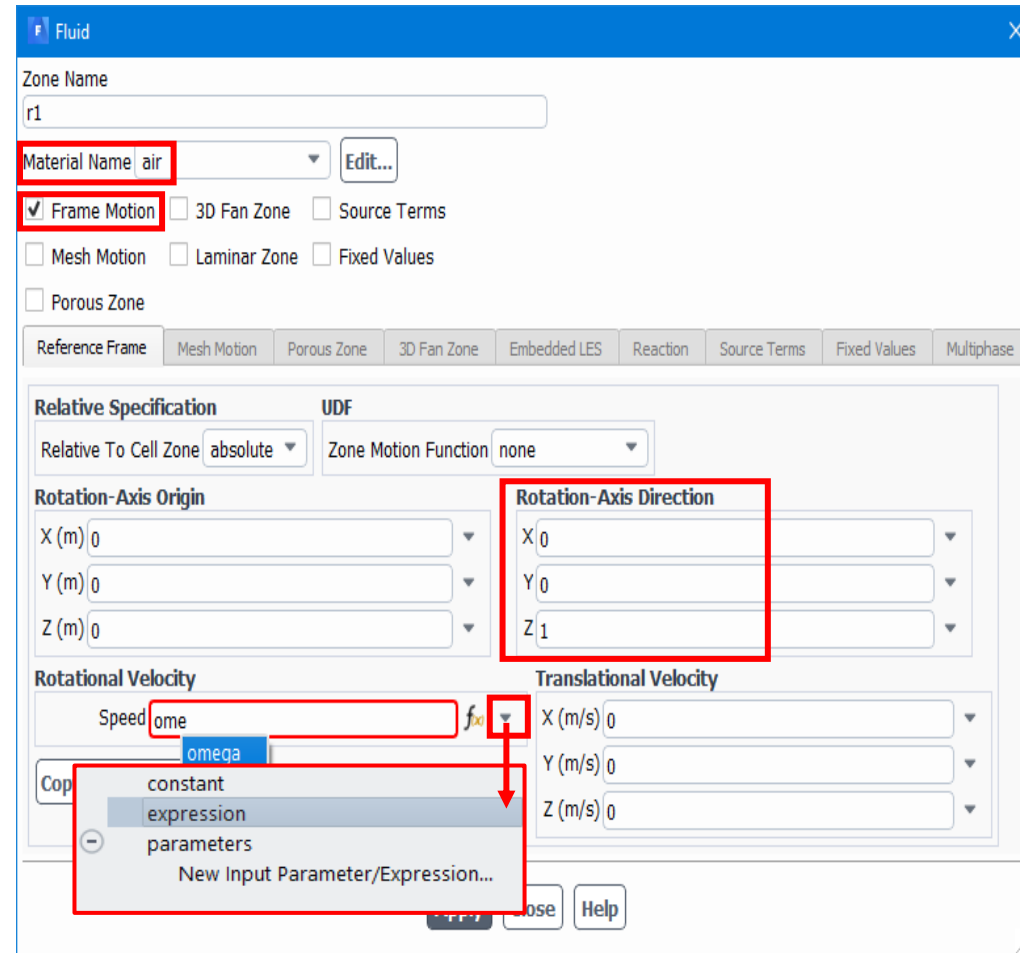
Physics: Cell Zone Conditions (2)

- Continuing from previous slide…

- The settings of cell zone *r1* should look as in the right-top image

- Click *Apply*

- The rotational speed has been set via an expression *omega= 2880 [rev/min]*
  - Sign verification: If you place your right thumb to point as the <u>positive</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>positive</u> number

- Expressions are used for a consistent setup and may also be used for the calculation of key targeted quantities
  - *omega* will be later used, in conjunction with a Force report for the moment, for creating a report for the rotor power
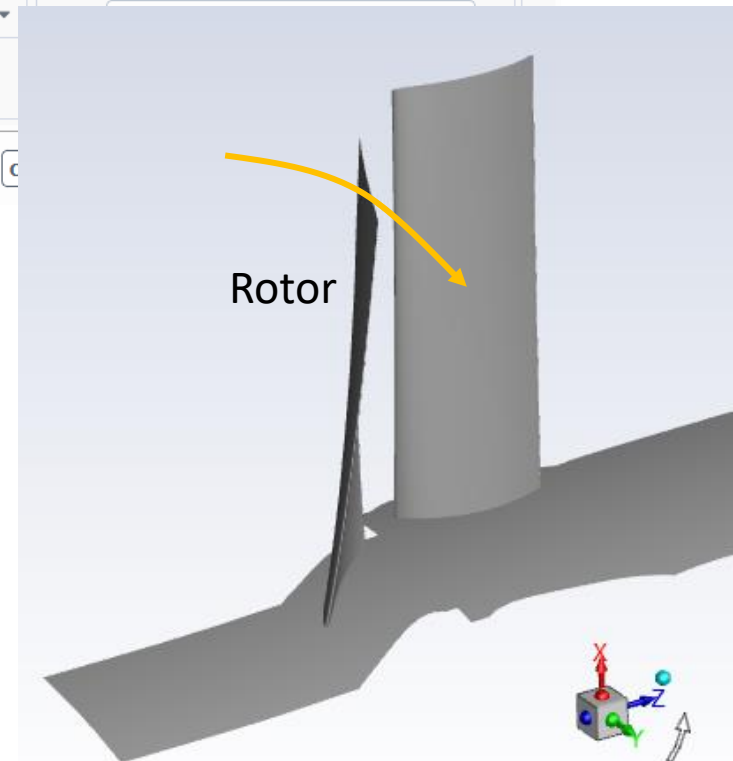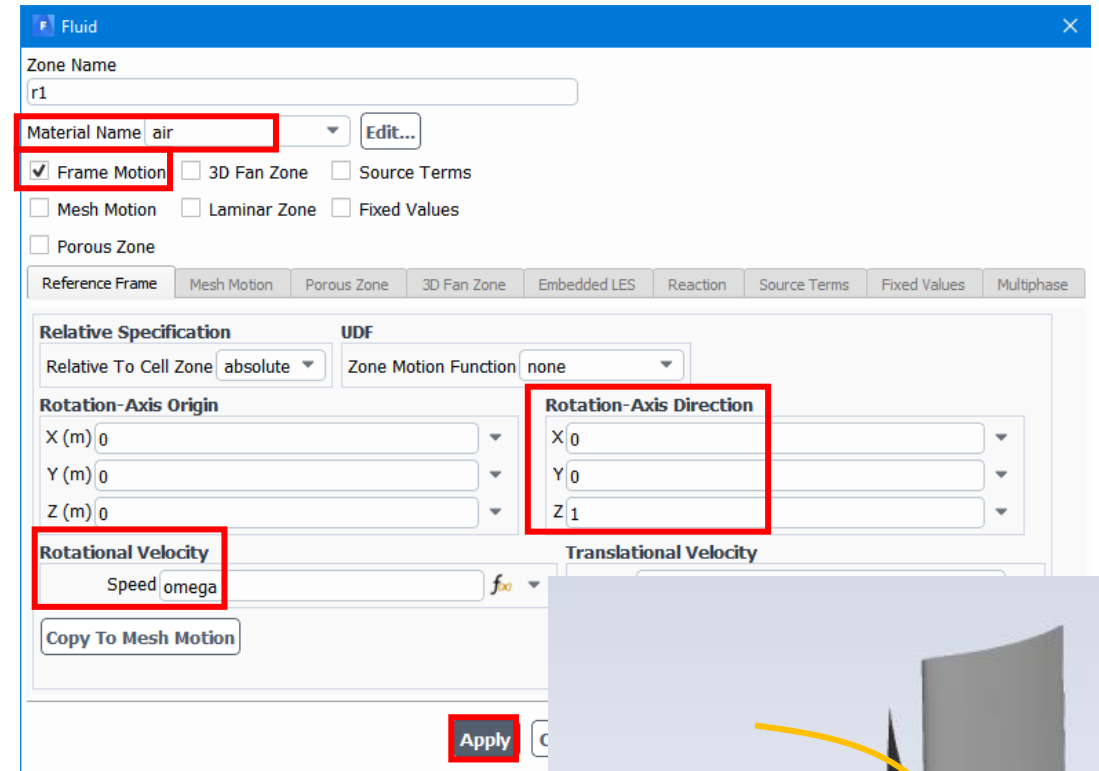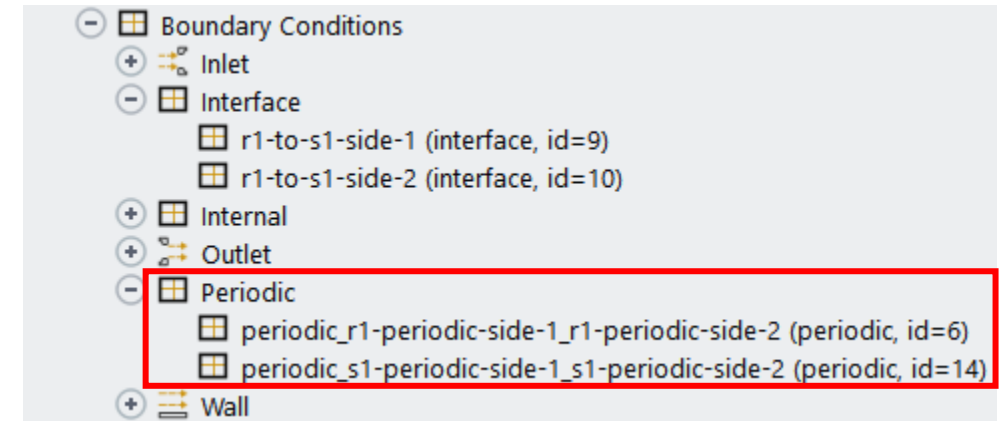
Create Rotational Periodic Zones

- In the *Boundary Conditions* of the *Outline View* expand the *Interface* branch

- Select *r1-periodic-side-1* and *r1-periodic-side-2*
  - 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 7 deleted
    Created a conformal periodic boundary.

- The periodic pair is placed under the newly created *Periodic* branch in the *Outline*

©2020 ANSYS, Inc. Unauthorized use, distribution, or duplication is prohibited.
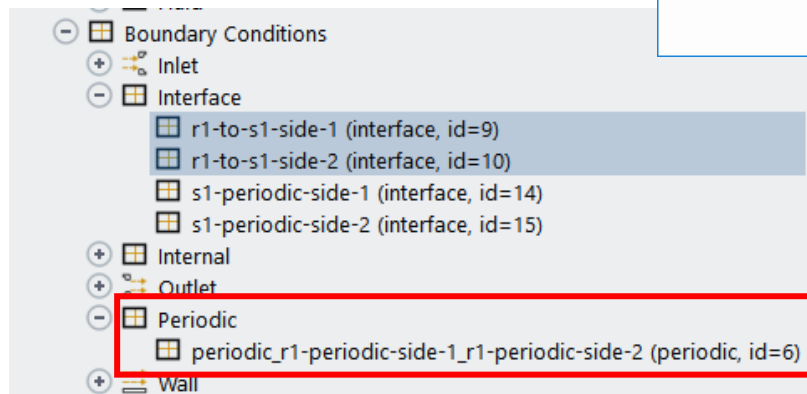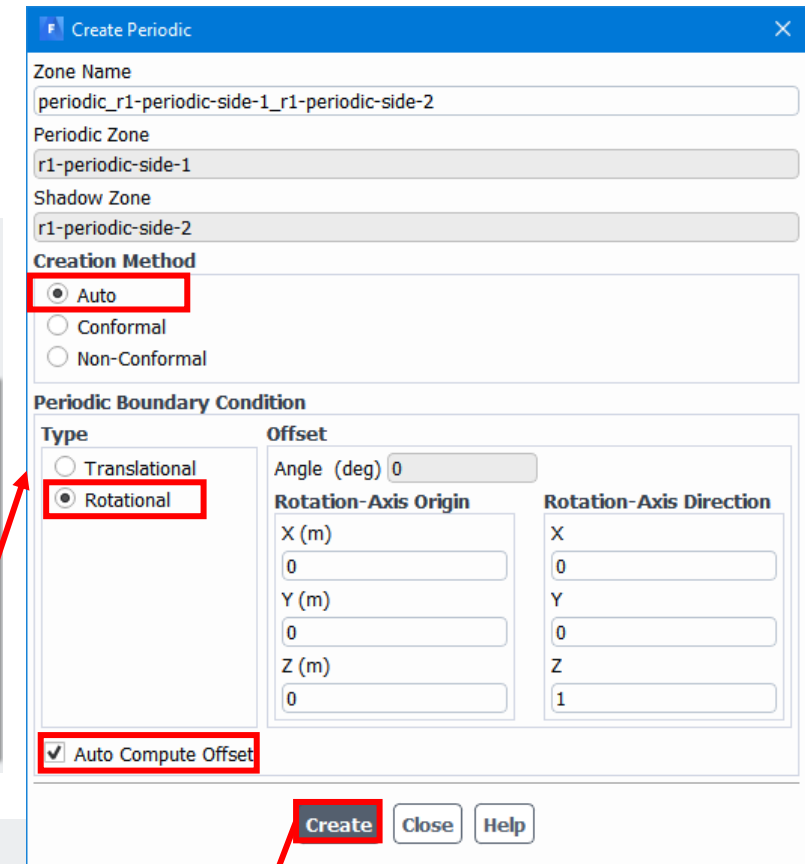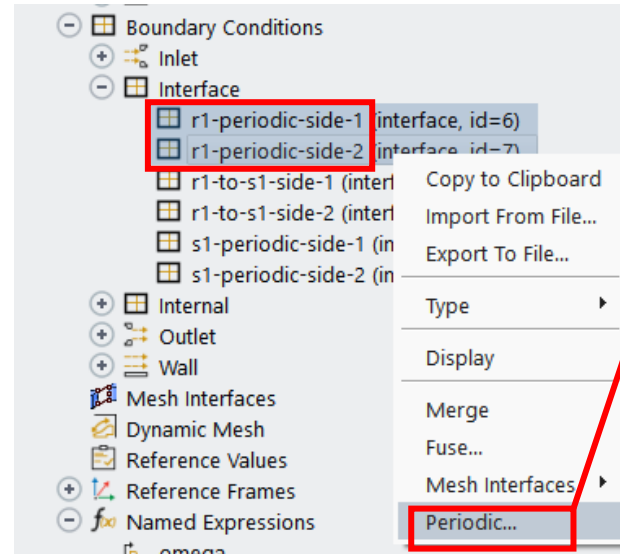
Create Rotational Periodic Zones (2)

- In the same way Create Rotational Periodic Zones for
  s1-periodic-side-1 & *s1-periodic-side-2*

- 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
  - Periodic zone 6 corresponds to the rotor and its rotation angle was computed to 12 degrees = 360 degrees / 30 blades (the rotor has 30 blades)
  - Periodic zone 14 corresponds to the stator and its rotation angle was computed to 18 degrees = 360 degrees /20 blades (the stator has 20 blades)
  - At this stage we can also see a Warning about an unassigned zone for interface 10. This will be corrected later by defining the rotor/stator GTI interface

©2020 ANSYS, Inc. Unauthorized use, distribution, or duplication is prohibited.
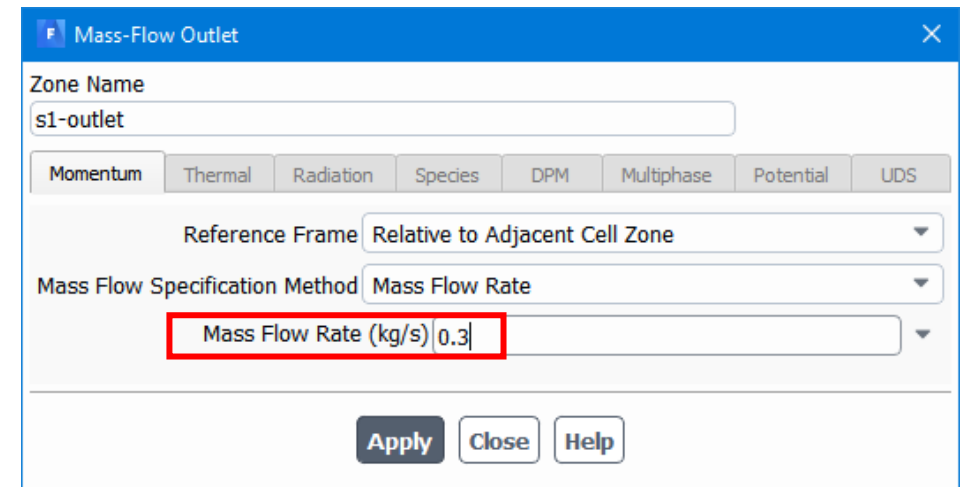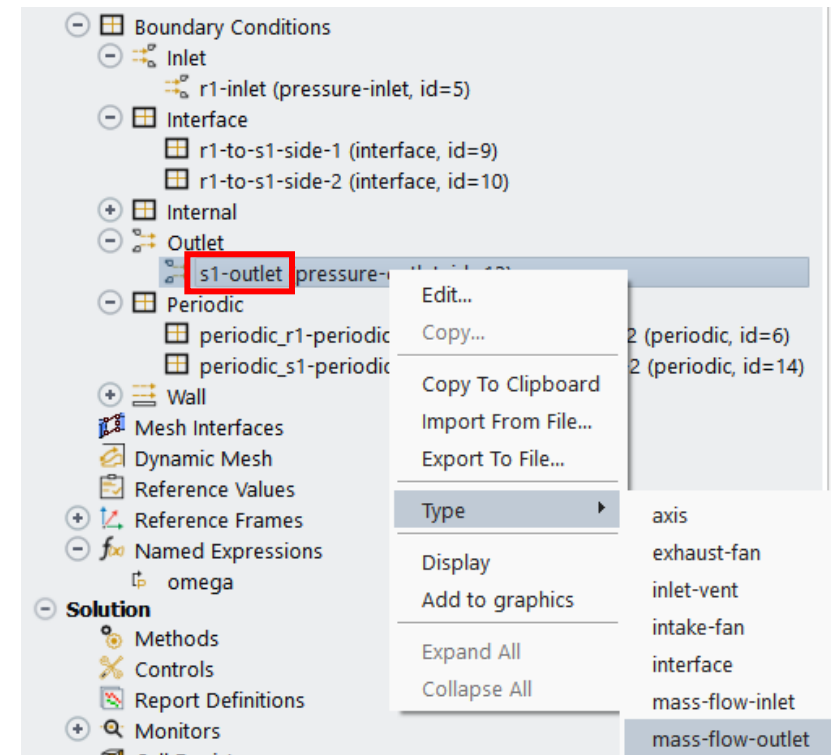
Boundary Conditions: Inlet

- Set the boundary conditions for *r1-inlet*
  - *RMB* on *r1-inlet* and set *Type* to *pressure-inlet*
  - Check *Prevent Reverse Flow*
  - Set a *Gauge Total Pressure* of 0 (pascal) at the inlet *
  - Set a Supersonic/Initial Gauge Pressure of -20 (pascal)
    - *Initial Gauge Pressure is set a few (pascal) lower than the Gauge Total Pressure. This will help in the flow field initialization (see slide 25)*
  - Accept all remaining defaults in the Momentum tab
  - In the *Thermal* tab, set a *Total Temperature* of 288 (k) (not shown), click *Apply* then *Close*

*   Note that the Operating Pressure was set to 100,000 (Pa) for this case. For this reason, the Gauge Total Pressure is set to 0 (Pa)
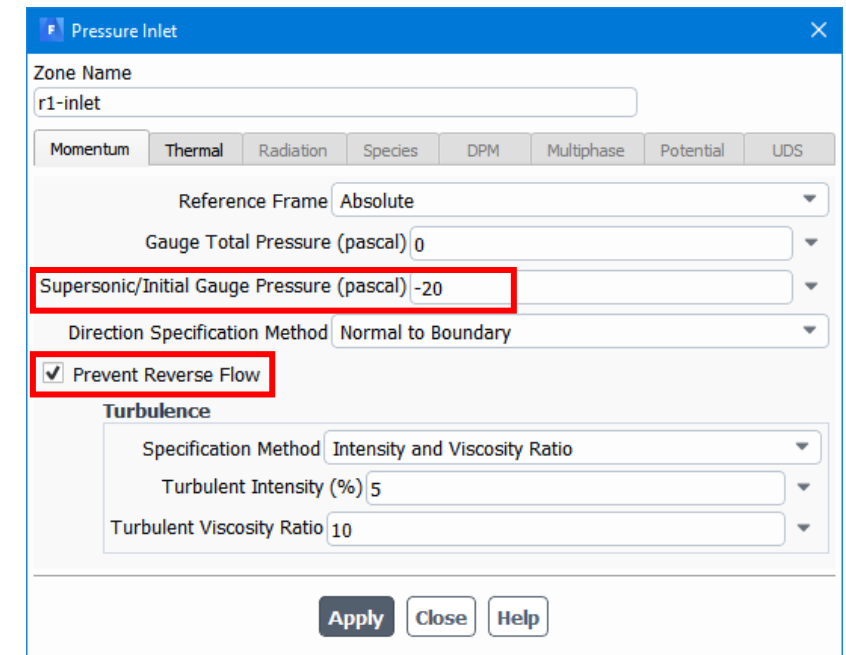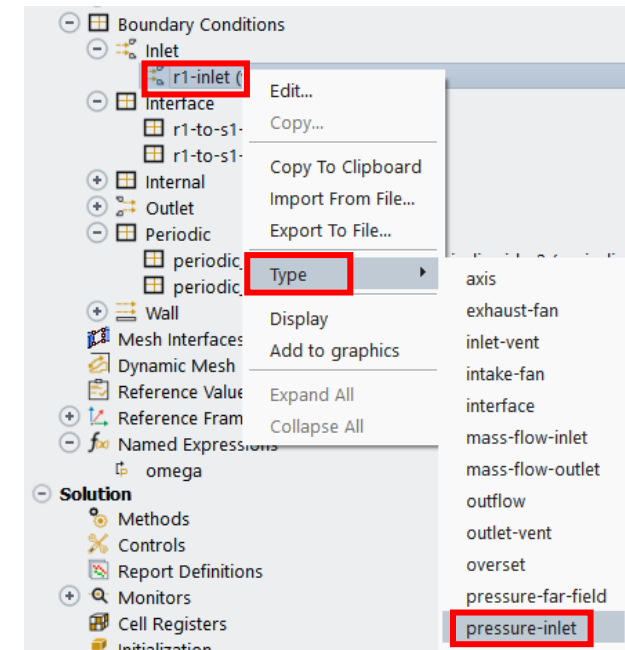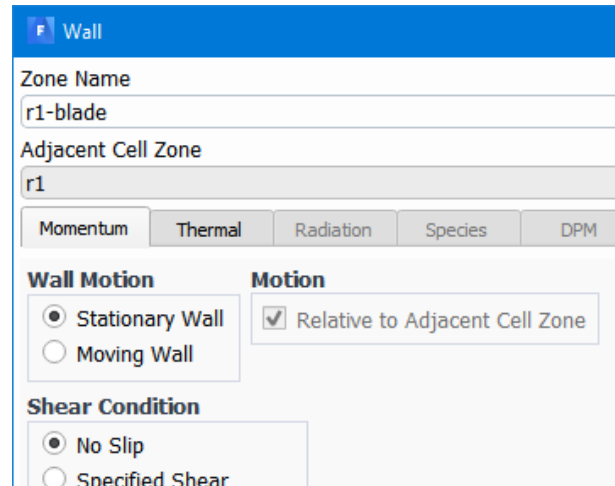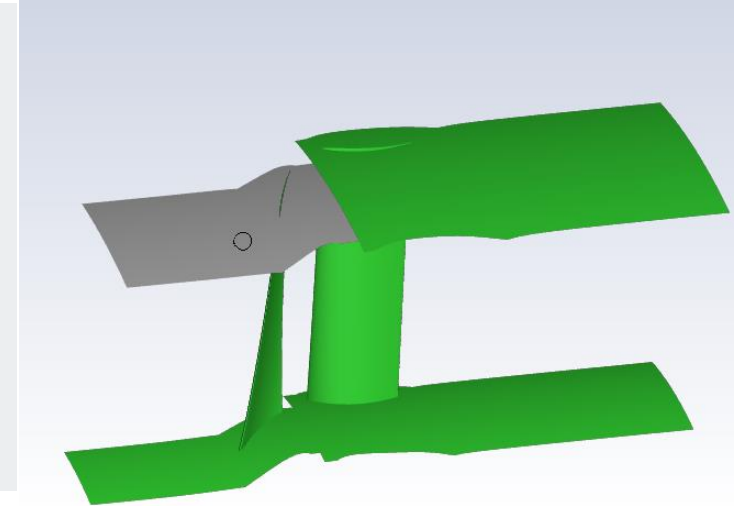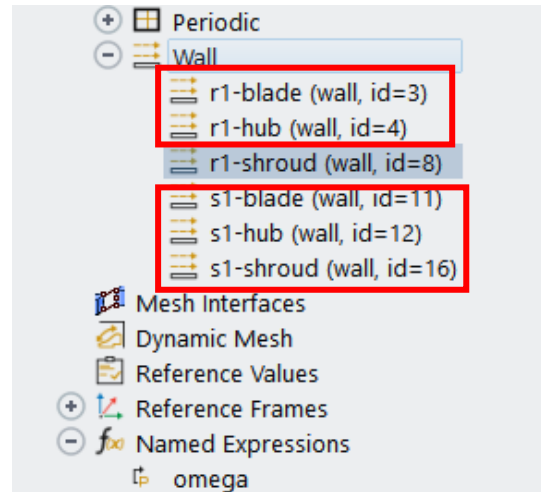
Boundary Conditions: Outlet

- Set the boundary conditions for *s1-outlet*
  - *RMB* on *s1-outlet* and set *Type* to *mass-flow-outlet*
  - Set the *Mass Flow Rate = 0.3(kg/s)*
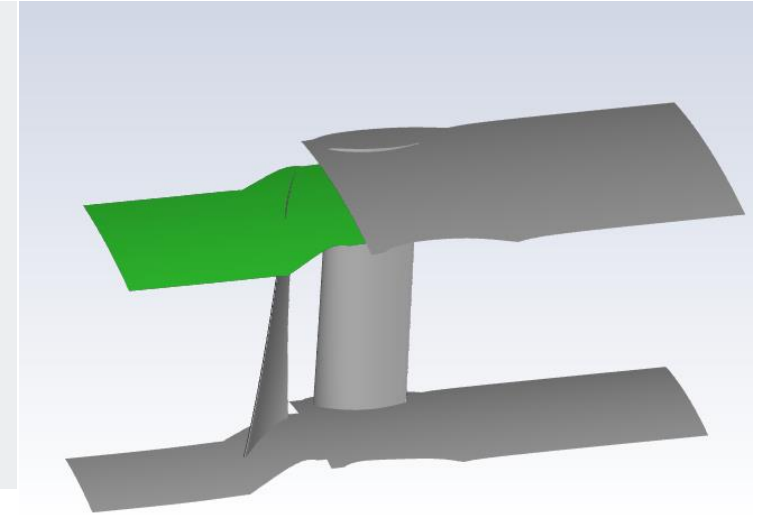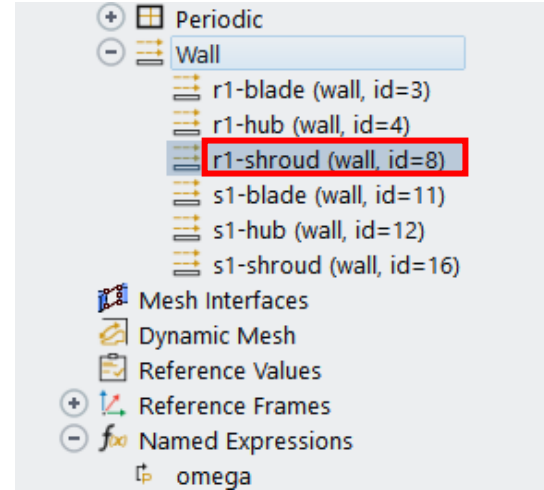 - Click *Apply*

Boundary Conditions: Automatically Set Walls

- From the remaining zones under *Wall* in the *Outline View*, all boundaries marked with red boxes correspond to walls which can be left to the default wall boundary condition setting:

  - *Stationary Wall*

  - *Relative to Adjacent Cell Zone*

  - *No Slip*

©2020 ANSYS, Inc. Unauthorized use, distribution, or duplication is prohibited.

Boundary Conditions: Rotor Shroud Wall

- The rotor shroud wall *r1-shroud*, belongs to a rotating cell-zone but is stationary in the absolute frame
  - Double click on *r1-shroud*
  - 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

Define a Mixing Plane Interface

- Having concluded with all rotational periodic zones and basic boundary conditions, you will now define a *Mixing Plane General Turbo Interface*
  - In the *Domain* tab under *Turbo Model* group check *Enable* and click *Turbo Create…*
  - In the *Create/Edit Turbo Interfaces* panel:
    - Set mesh interface name to *r1-to-s1*
    - Check *General Turbo Interface*
    - Select the two sides of the interface
    - Select *Mixing Plane*
    - Click *Create/Edit*
      - Ignore any error message in the console after clicking *Create/Edit*
  - Do a mesh check and make sure that no error message appears in the console
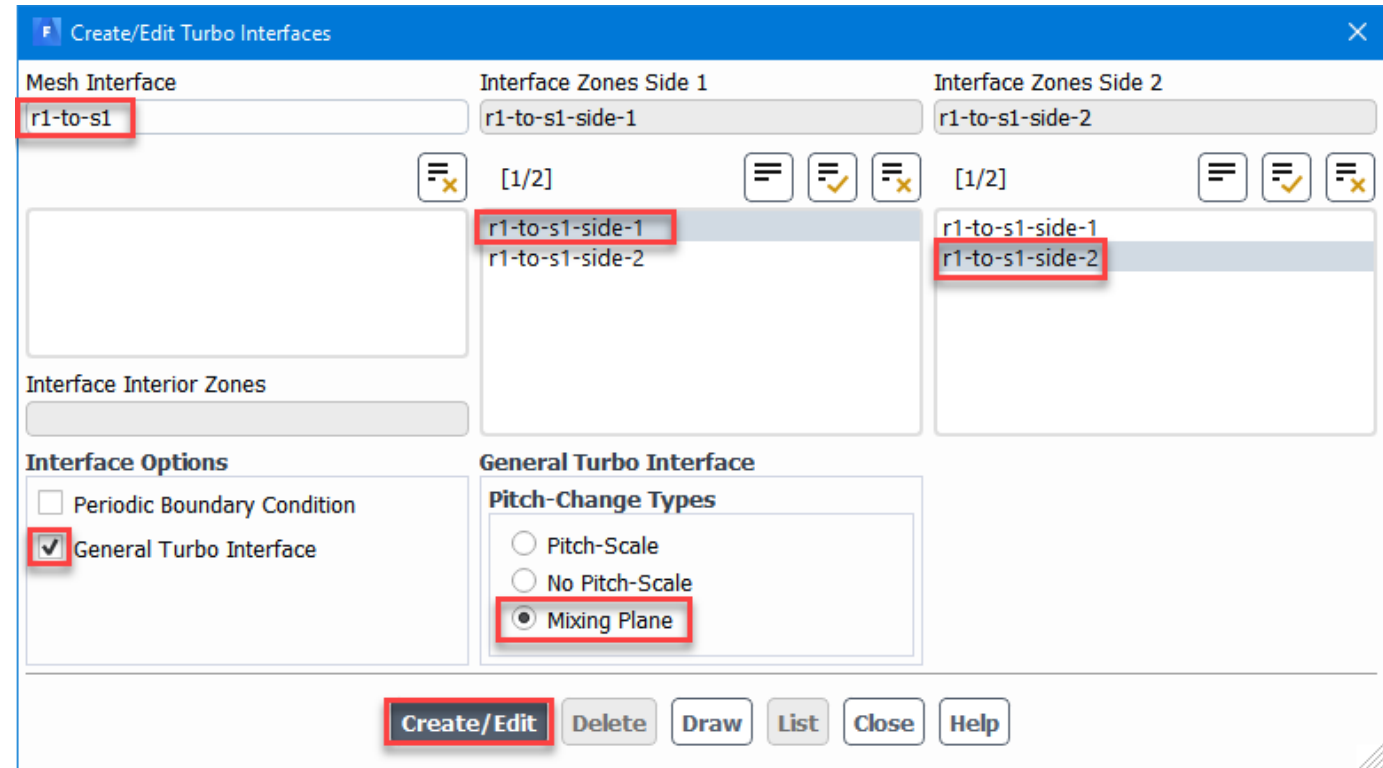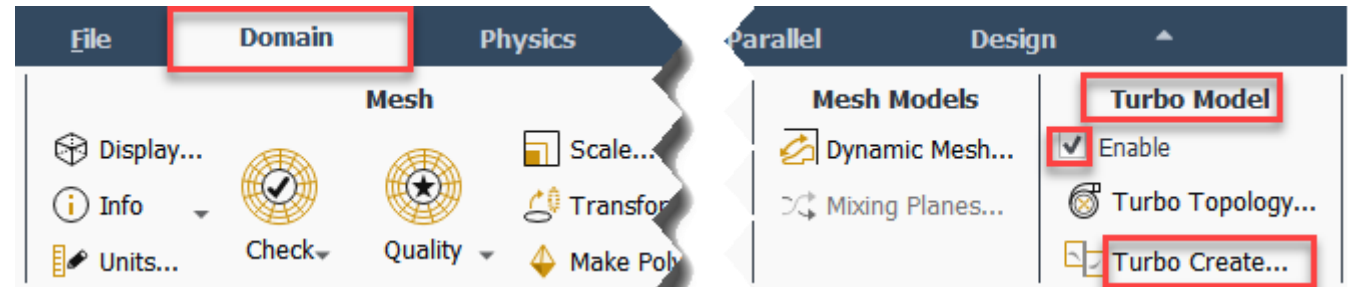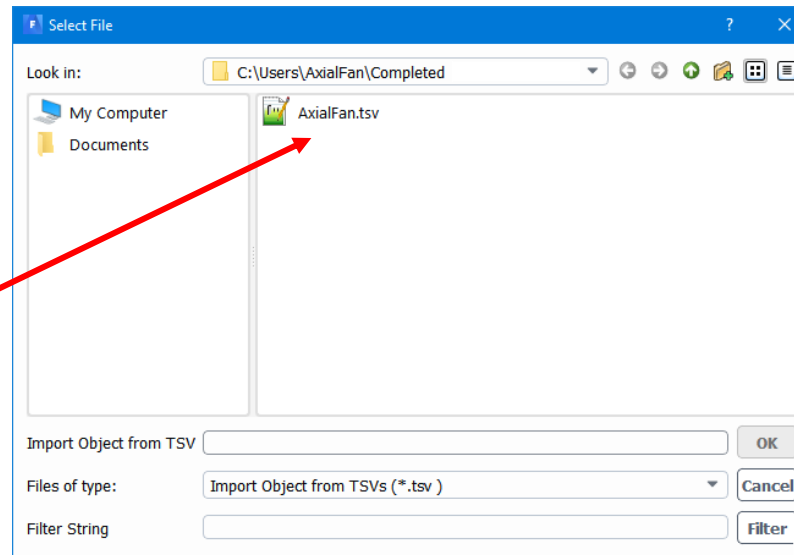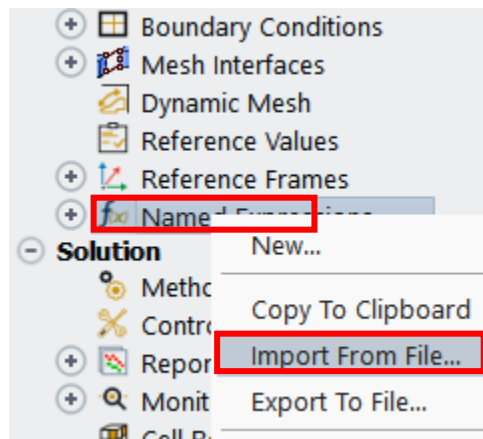
Solution: Moment Report Definition

- In the *Solution* tab create a new *Force Report Definition* for the *Moment* about the *z-axis,* with the following settings:

  - *Name = moment*

  - *Report Output Type = Moment*

  - *Boundaries = r1-blade, r1-hub*

  - *Report File = checked*

  - *Report Plot = checked*

  - *Moment Axis → keep the default corresponding to the z-axis)*

  - Click *OK*
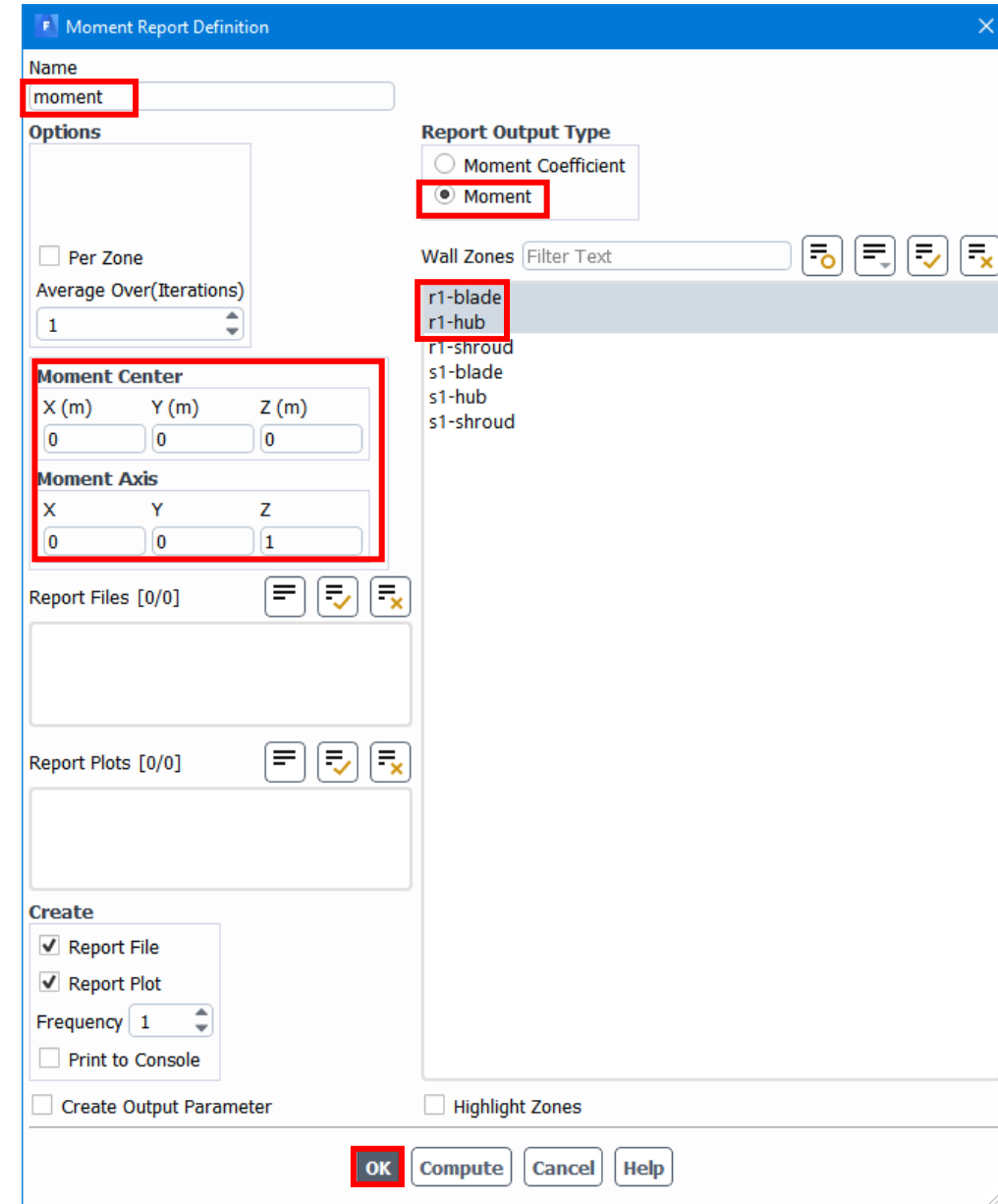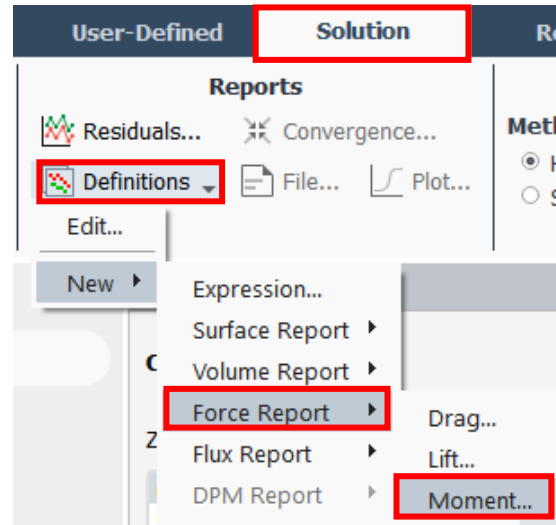
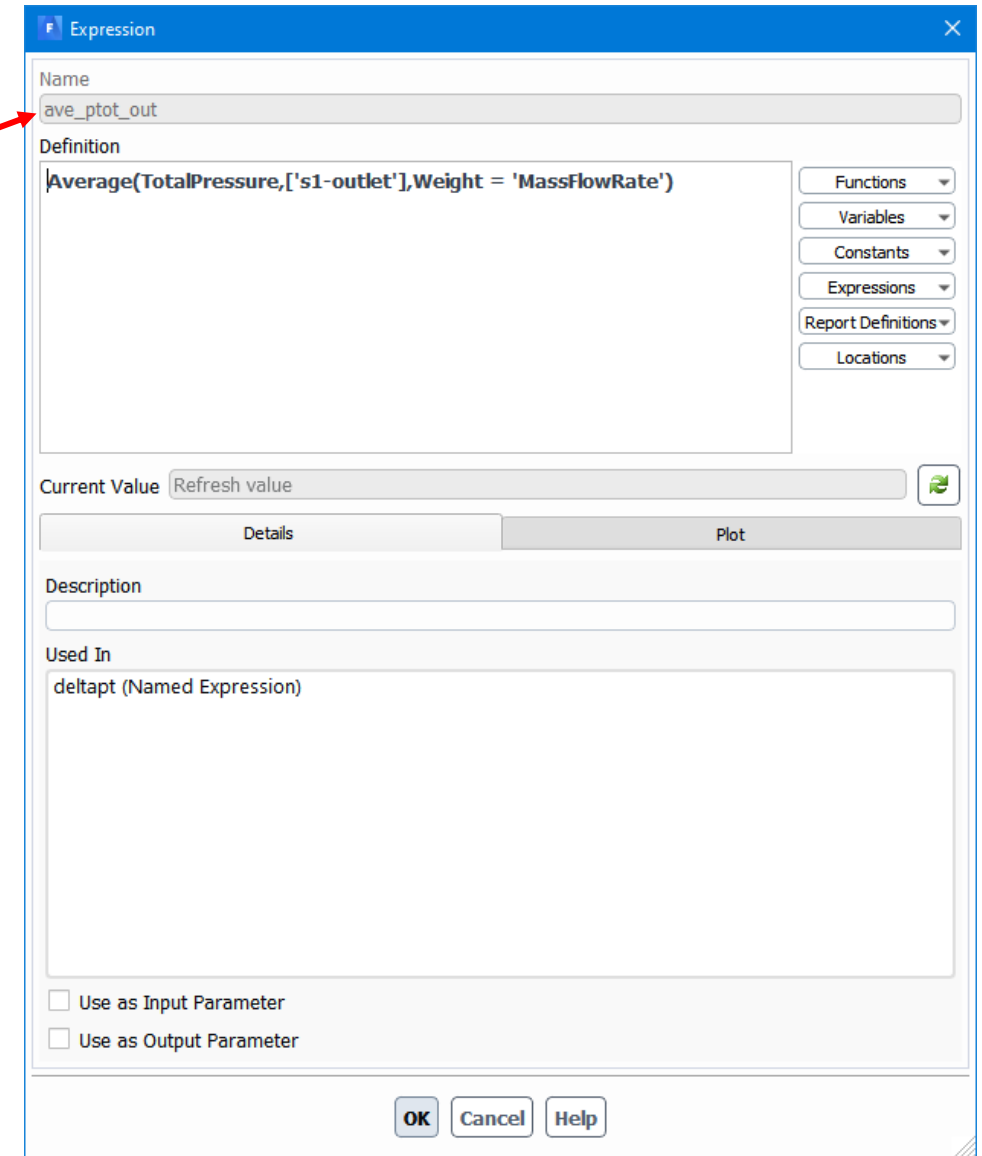# New Method for Report Definitions Using Named Expressions

- 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 *AxialFan.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,['r1-inlet'],Weight = 'MassFlowRate')"       ""        #f       #f
"ave_ptot_out" "Average(TotalPressure,['s1-outlet'],Weight = 'MassFlowRate')"      ""        #f       #f
"deltapt"       "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

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 = deltapt*
 - *Expressions > deltapt*
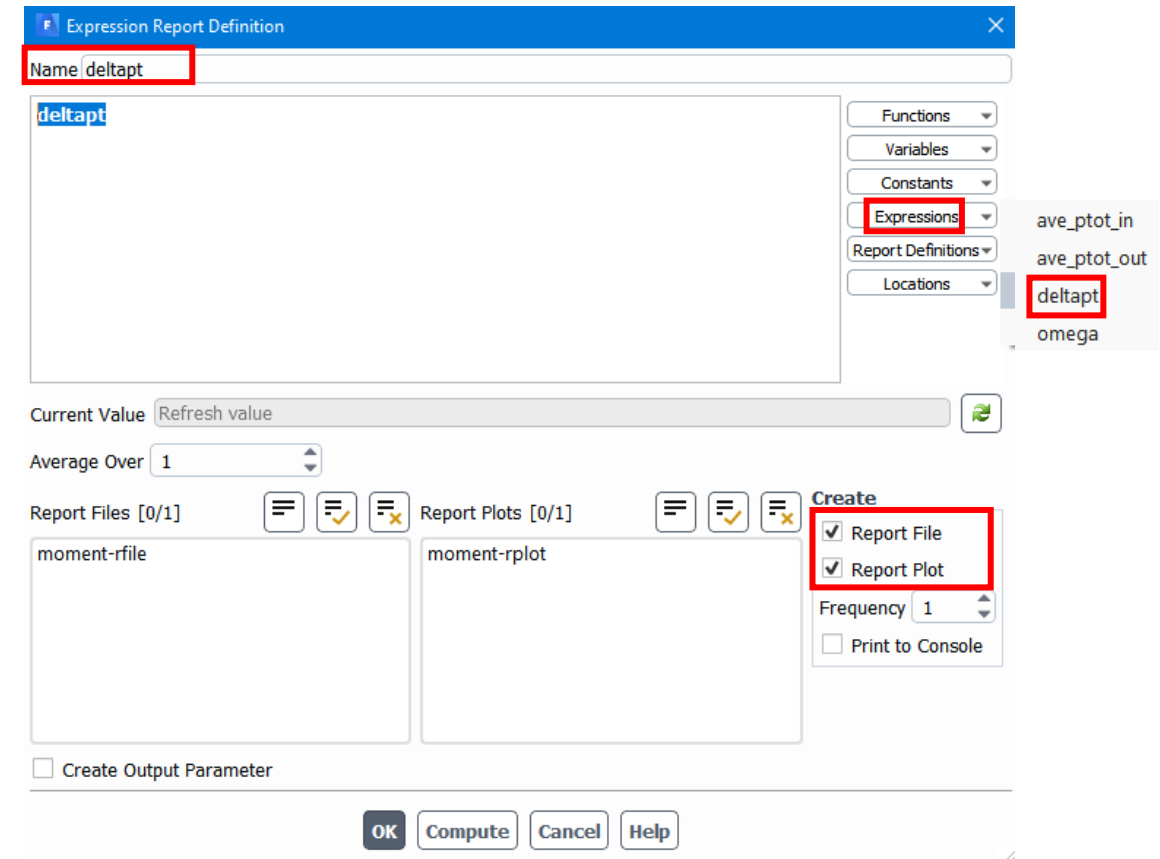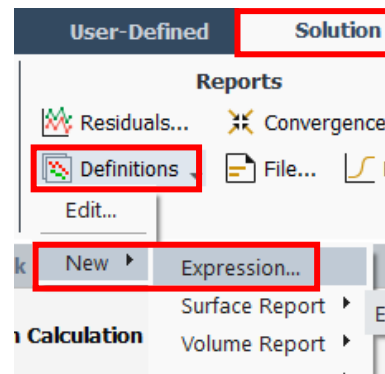
Solution: Create Report Definition for the Fan Power

- 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 = power*
  - *{moment}*omega*
    - This expression can be typed directly in the expression definition box, or one can use the drop-down lists of existing *Expressions* for *omega* and *Report Definitions* for *moment*
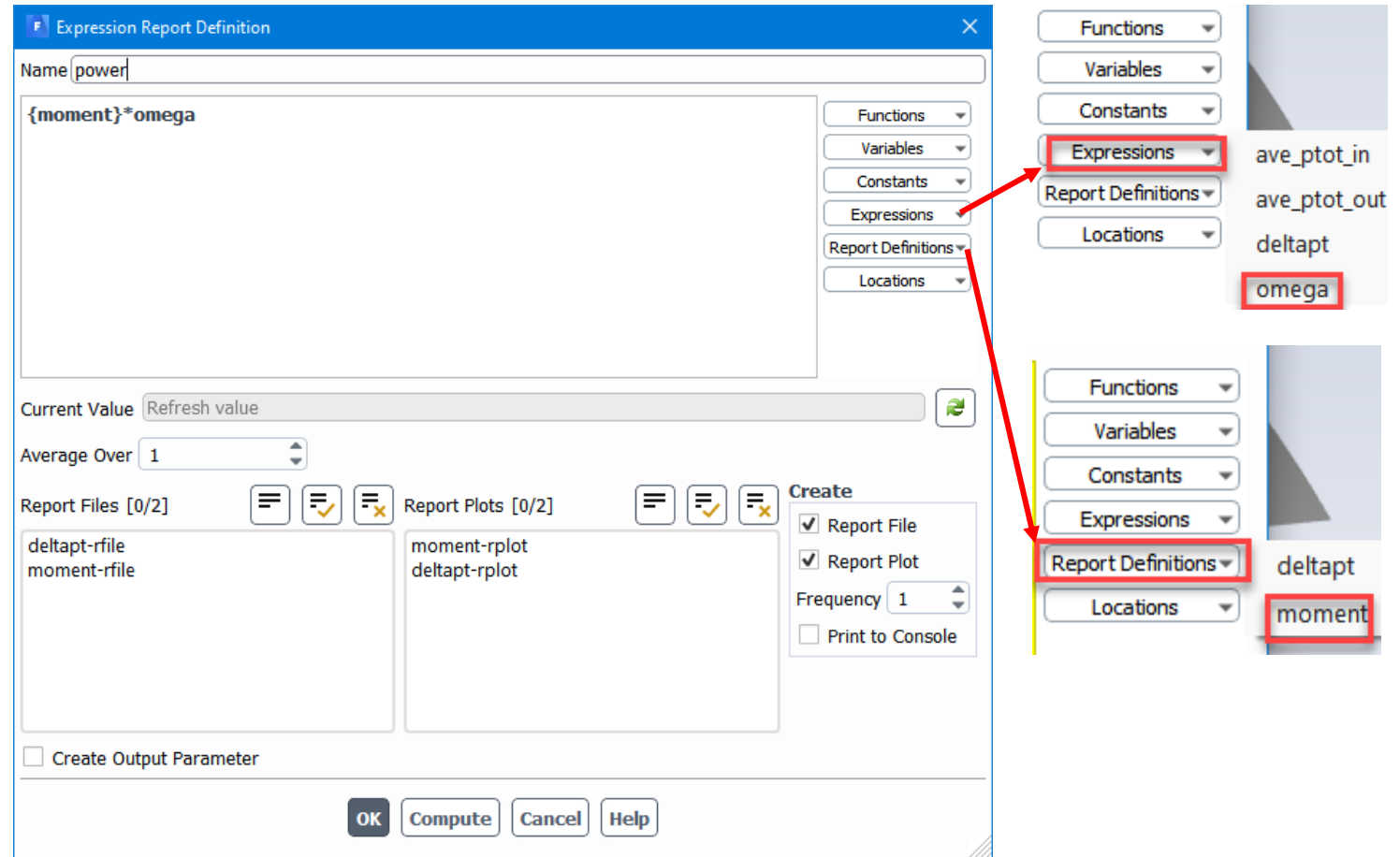
# Solution: Solution Methods

- 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)
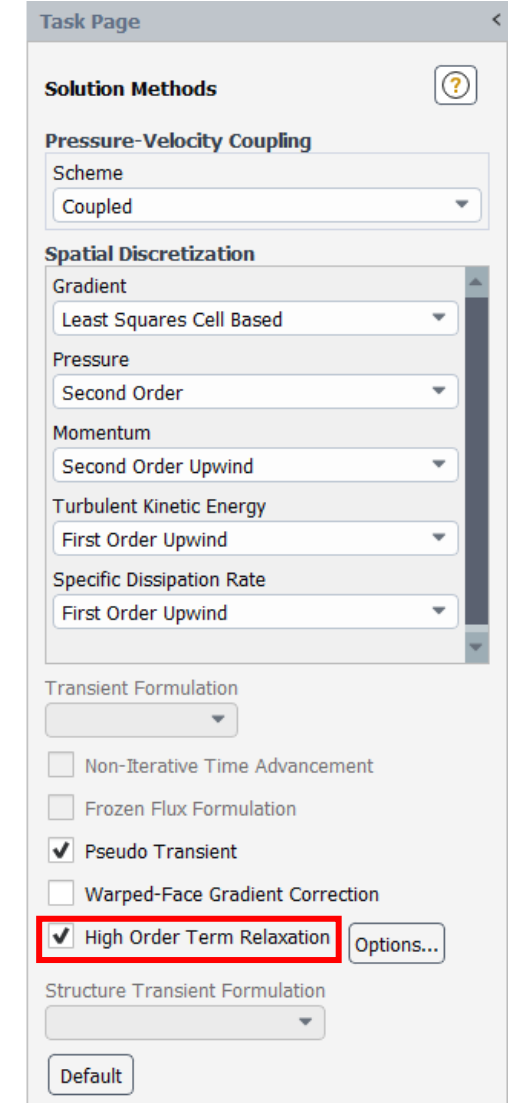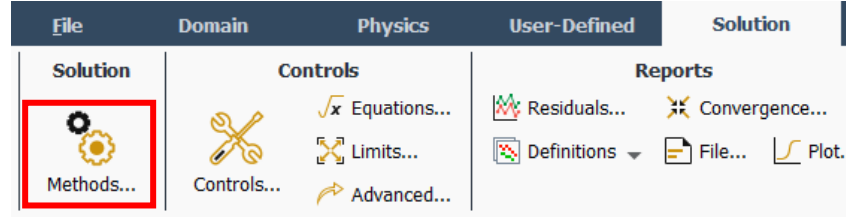
# Solution: Initialization (Best Practice Procedure)

- Perform a *Standard Initialization* using the inlet values
  - Remember we have set the *Initial Gauge Pressure*, a bit lower than the *Gauge Total Pressure* (slide 14)
    - 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*

- Save a Fluent .cas and .dat file
  - *File > write > Case & Data…*

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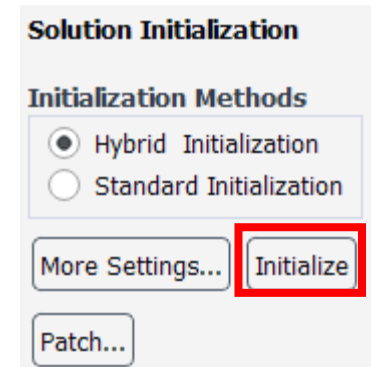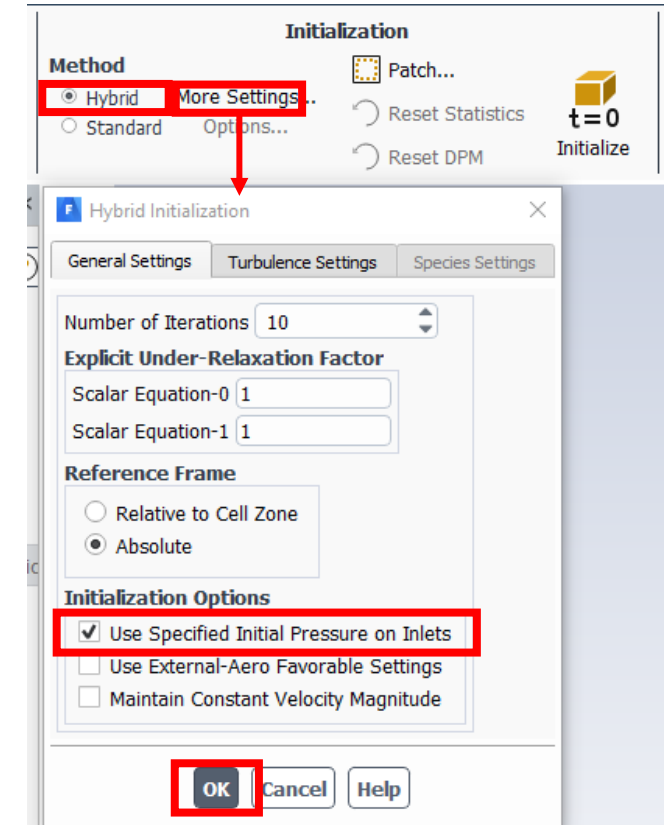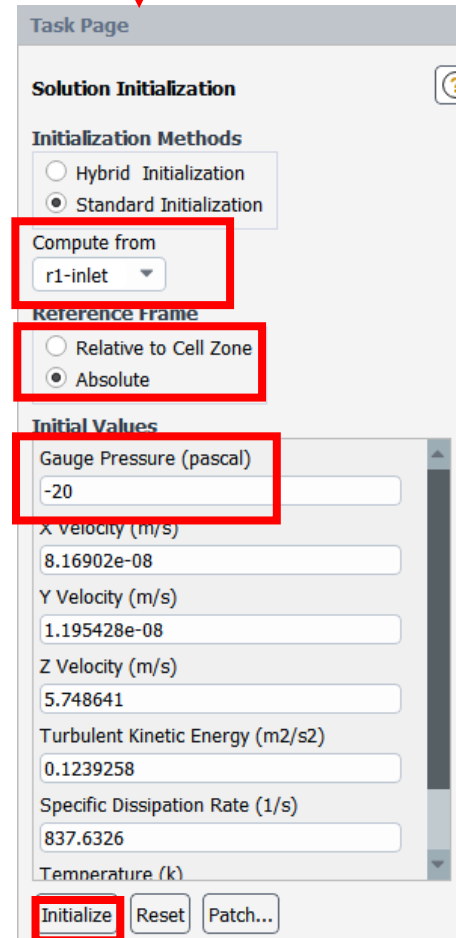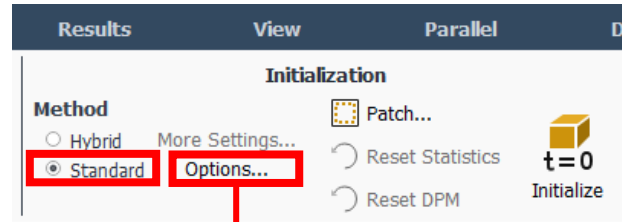
Solution: Run the Solver

- 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*
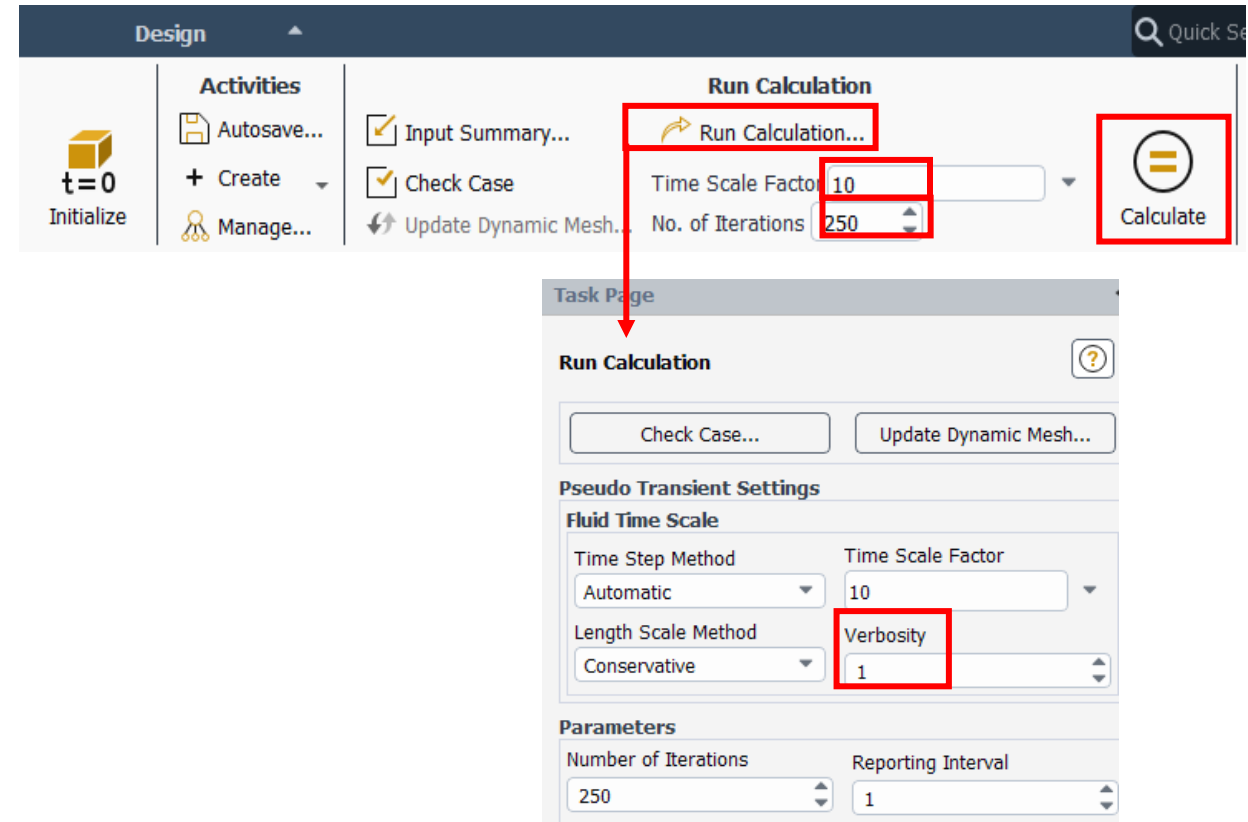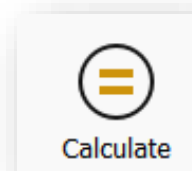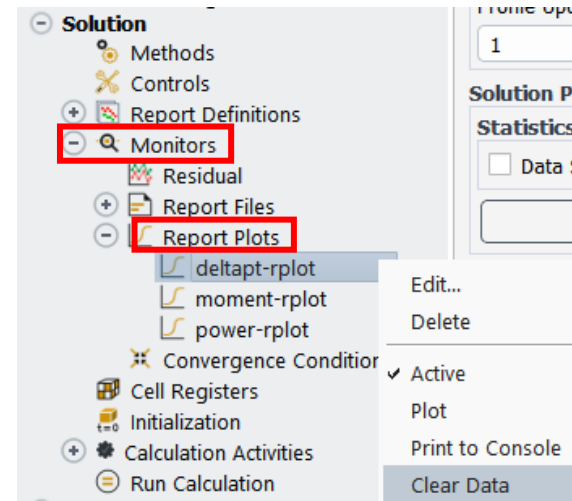
Solver Convergence

- The case does not converge well to a steady state solution after 250 iterations
  - Residuals and report plots show a bouncy behavior

Note: Residuals and monitor plots may differ between two computers or Ansys releases
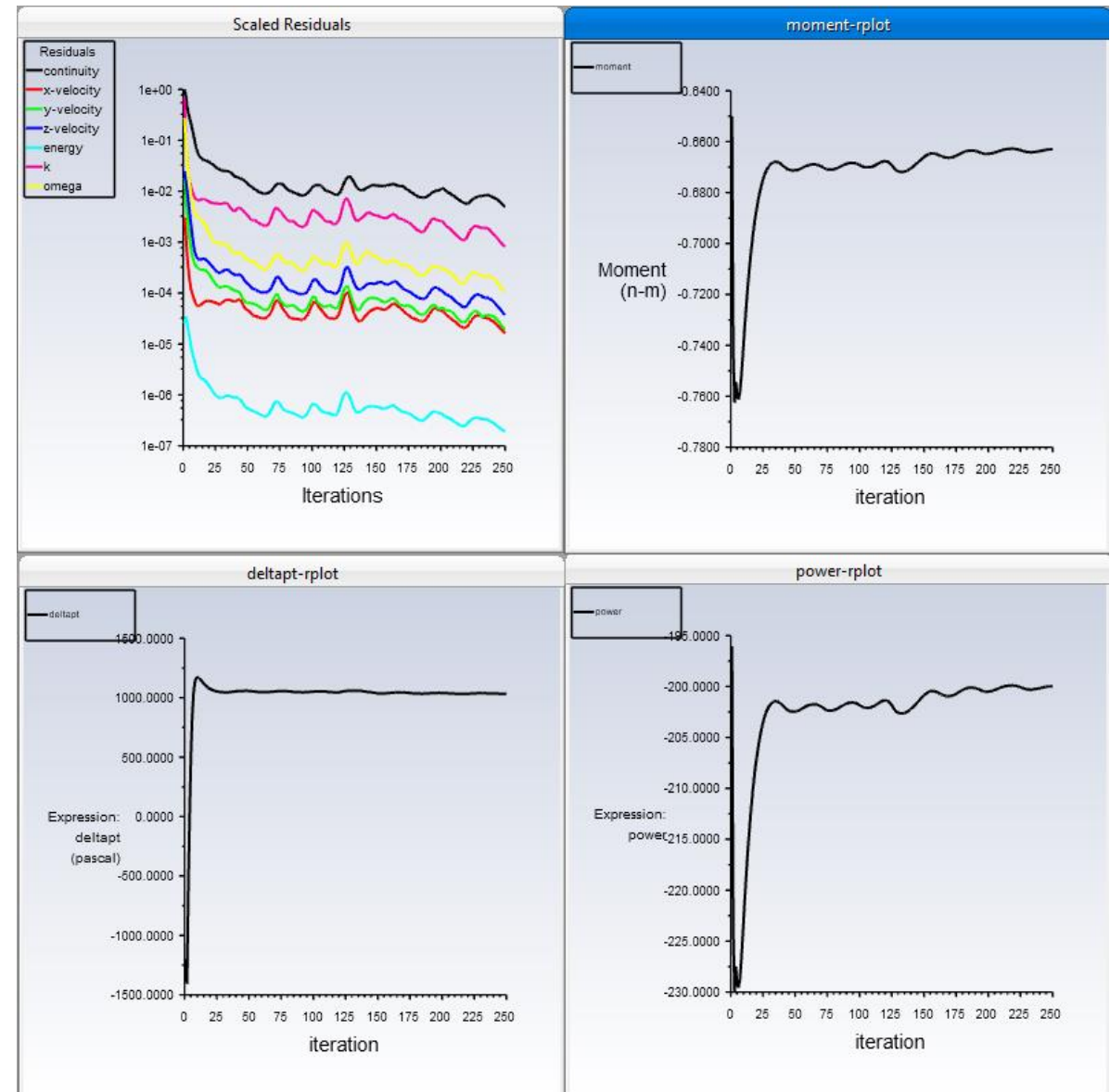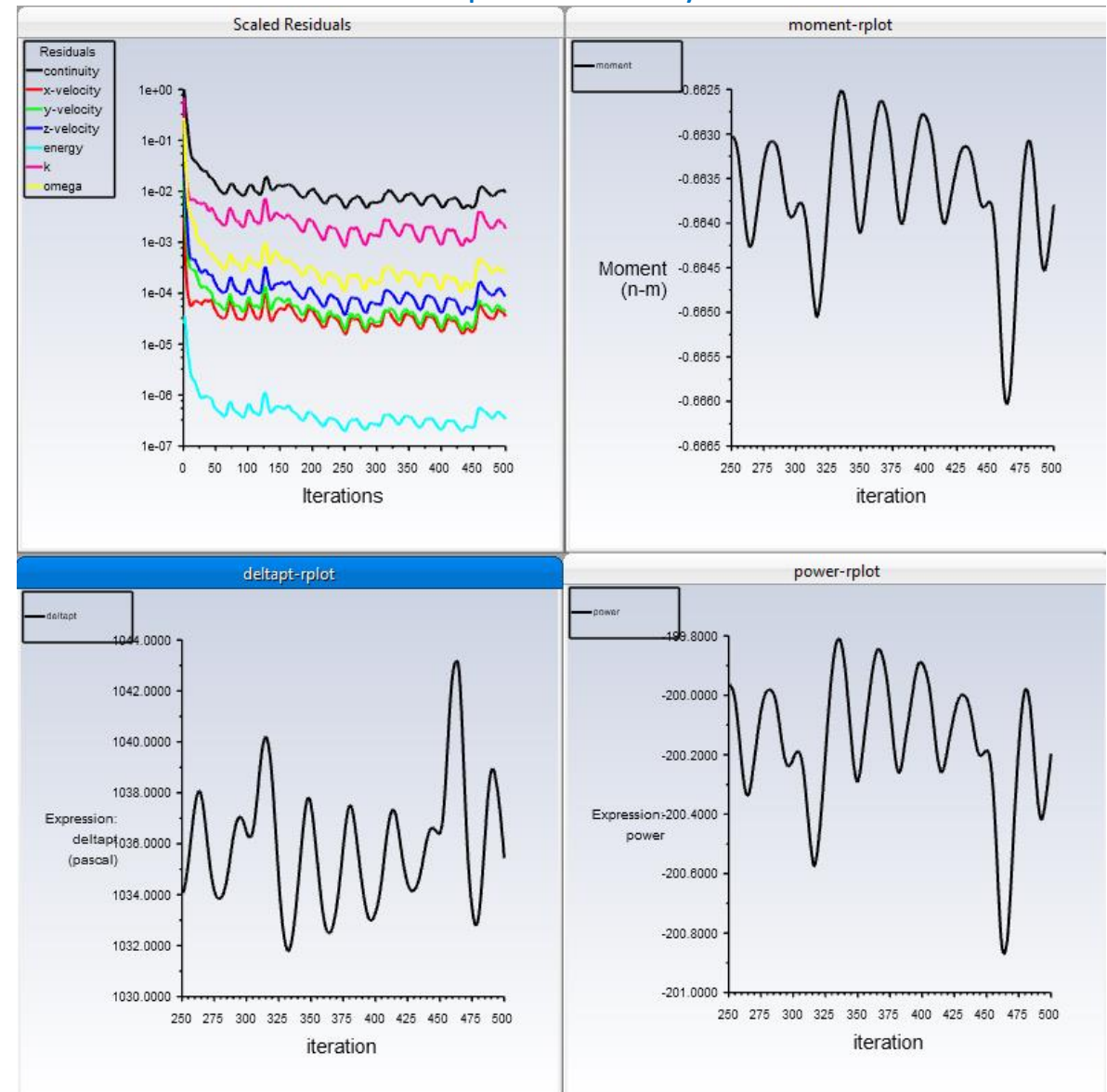
Check Solver Convergence Further

- It is a good idea to do a more detailed check of residuals and monitors.
  To do this:
  - Clear all *Report-Plots* data
    - This can be done by expanding the *Monitors* branch in the *Outline* and *RMB > Clear Data* for each *Report Plot*
      - This will reset the x- and y-axes limits of the report plots (see next slide)
  - Click *Calculate* in the *Solution* tab in the ribbon, for performing more solver iterations

Solver Convergence

- Residuals and report plots still show a bouncy behavior

  - Continuity residuals oscillate around 1e-2 and cannot be reduced further

  - Total pressure difference, moment and power monitors oscillate

  - Case is not converged to steady state in the strict sense

  - The targeted quantities of total pressure rise *deltapt* and power consumption *power* seem to be constant within 3 significant figures

    - *deltapt* ≈ 1035 (pascal)

    - *power* ≈ -200 (watt) per rotor passage

      - Total power consumed = -200x30=-6000 (watt)

  - To check the accuracy of these key quantities, it is a good idea to investigate the case by solving it also as transient. This is done in the next workshop
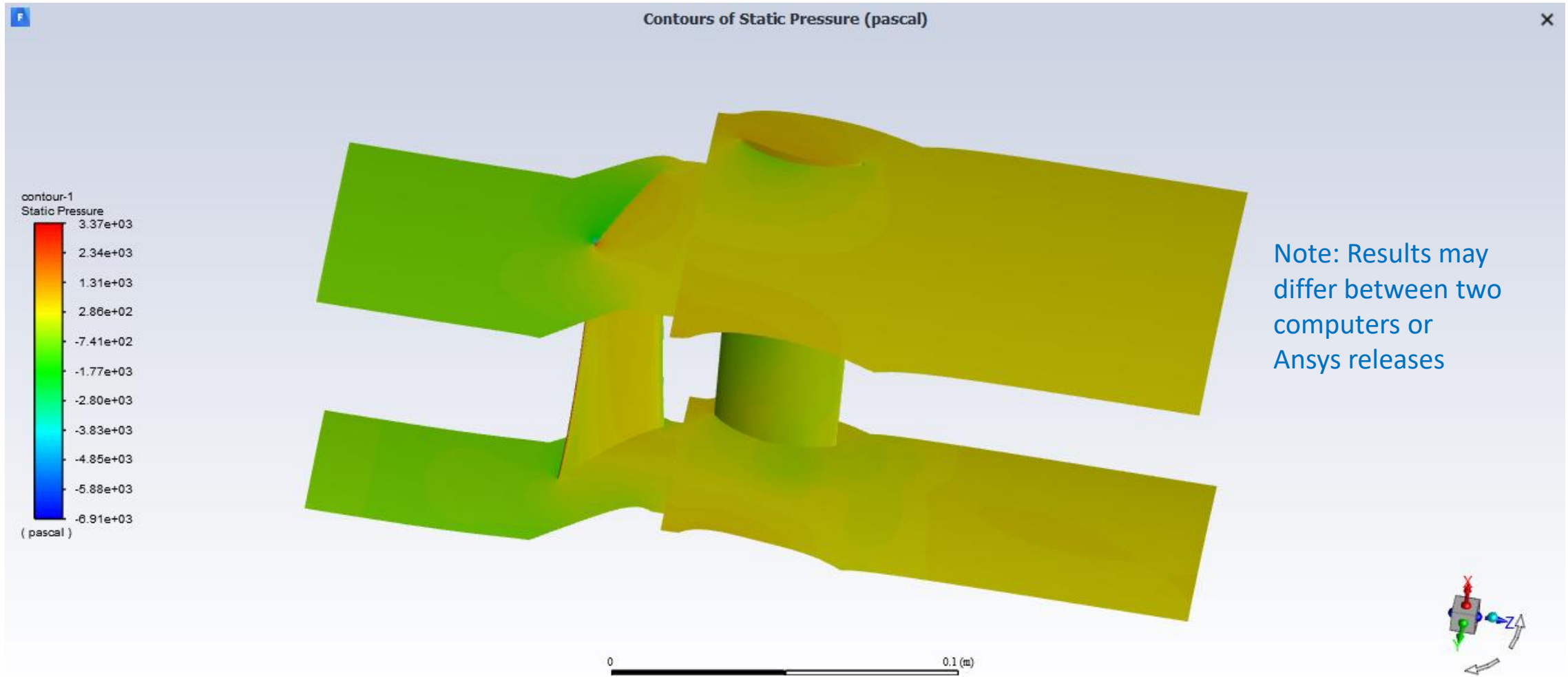
Pressure Contours on the Walls

- Create a *New Contour* plot of *Static Pressure* on all walls
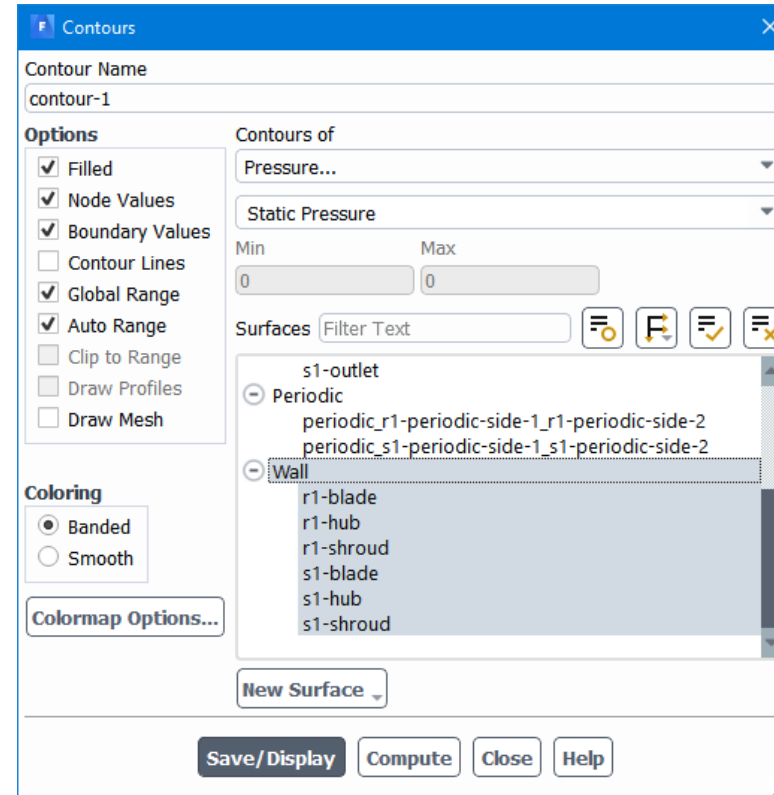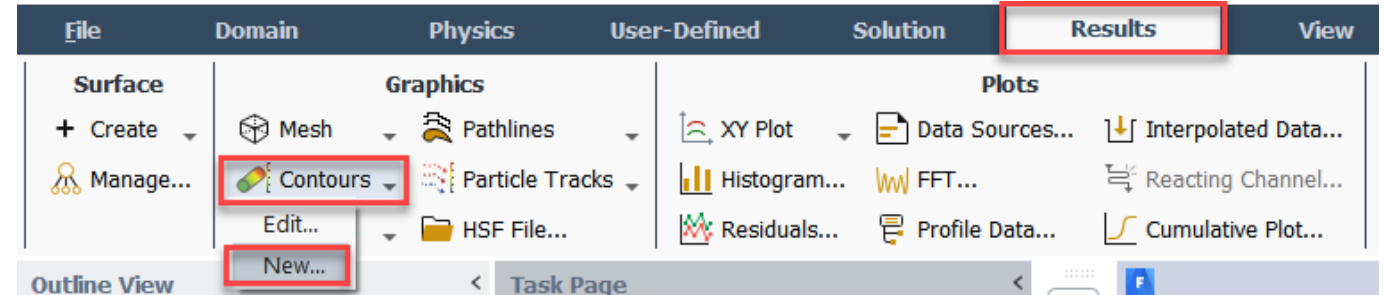
Pressure Contours on the Walls

Create a Vector Plot on an Iso-surface Constant Radial Coordinate

- You will now create a combined vector plot on an iso-surface of constant radial coordinate at mid-span

- This will include a relative-velocity vector plot for the rotor zone r1 and an absolute-velocity vector plot for the stator zone s1

- For this you will need to:
  - Define the reference zone for relative velocities and
  - Create:
    - 2 Iso-surfaces of constant radial coordinate, one for zone r1 and one for zone s1
    - 2 vector plots, one for zone r1 and one for zone s1
   - 1 scene combining the 2 vector plots
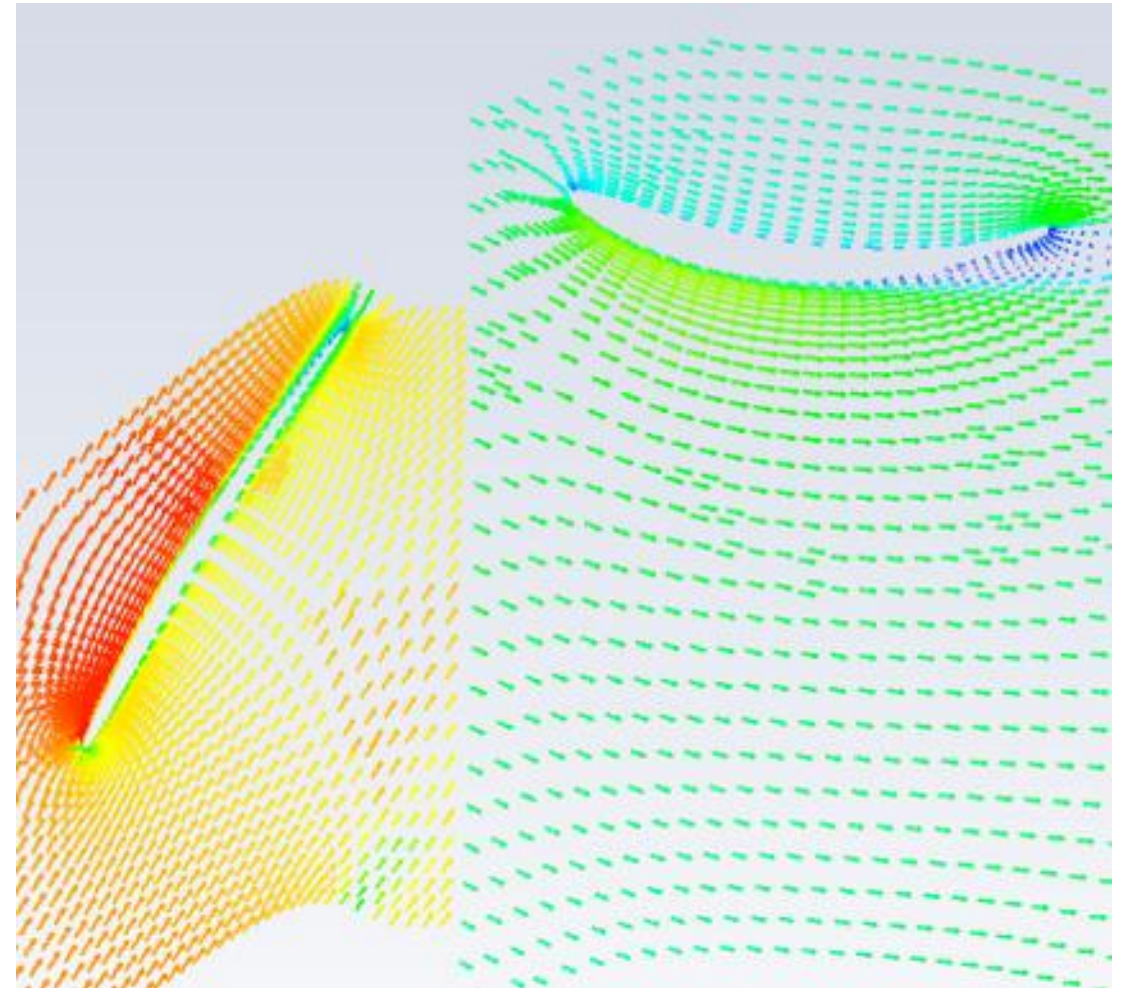
Define Reference Zone for Relative Velocities

- In the *Reference Values* panel set the *Reference Zone* to *r1*

  - In this model, we have one moving and one stationary fluid zone. The *Reference Zone* determines how the relative velocities are computed. See lecture 03 on Post-processing for more details
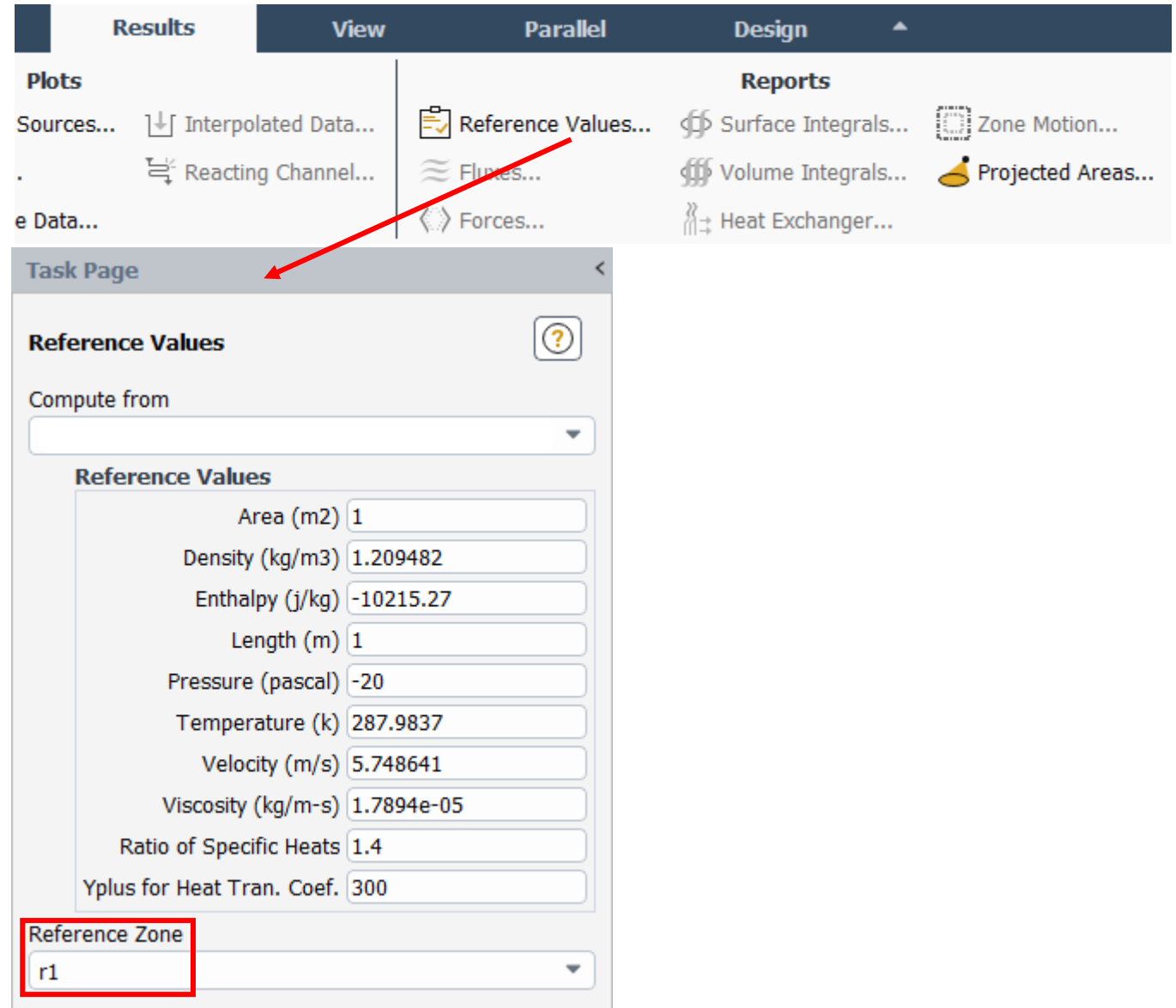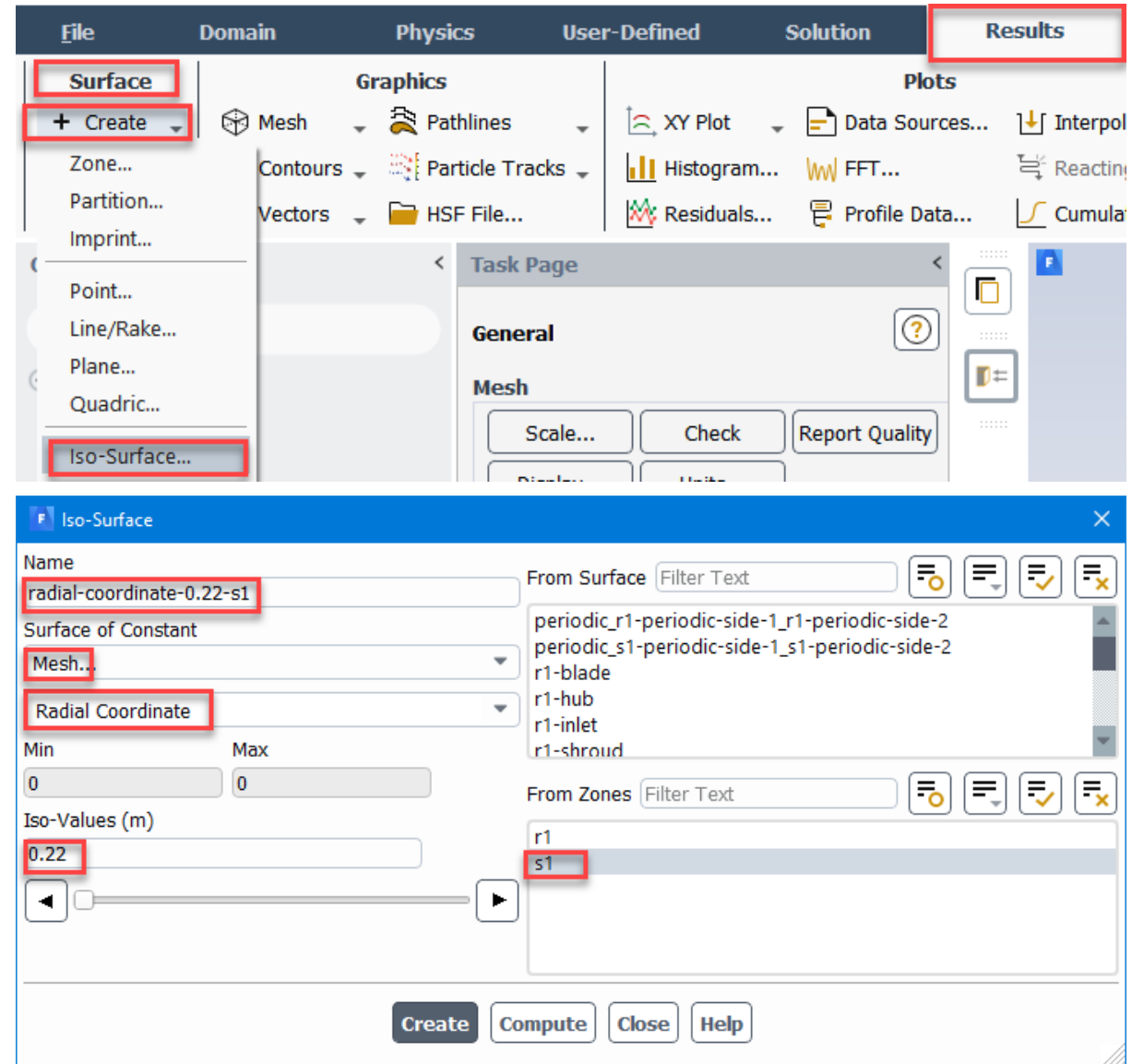
Create a Constant Radial Coordinate Iso-surface for Zone r1

- In the *Results* tab create a Constant Radial Coordinate Iso-surface for Zone r1
  - *Surface of Constant Mesh…*
  - *Radial Coordinate*
  - *Iso-Value = 0.22*
  - This corresponds to a midspan surface. We will use it for creating various vector and contour plots
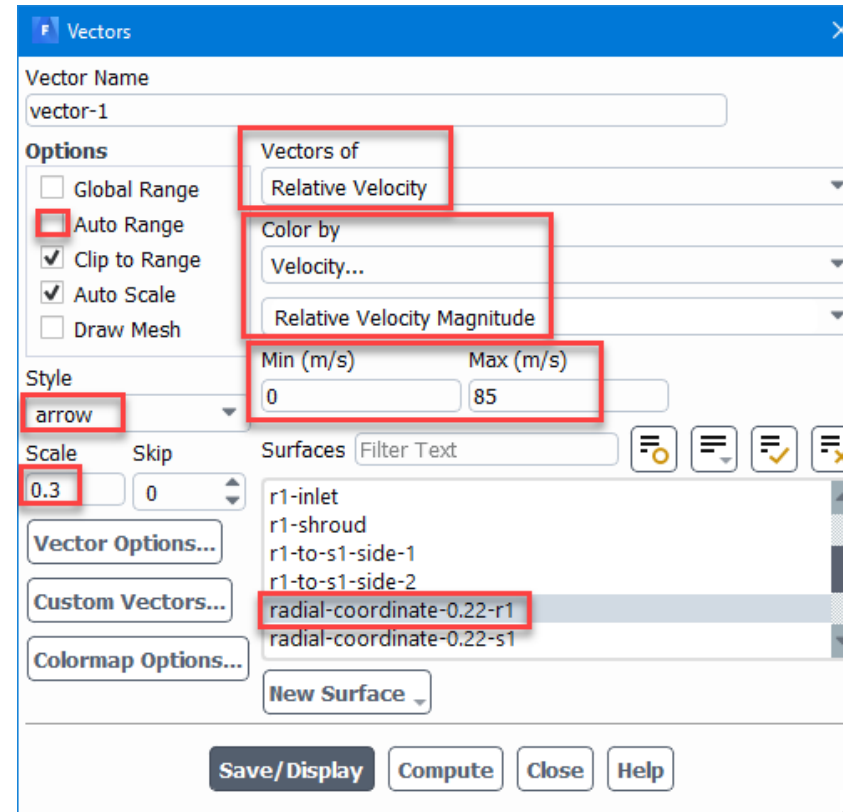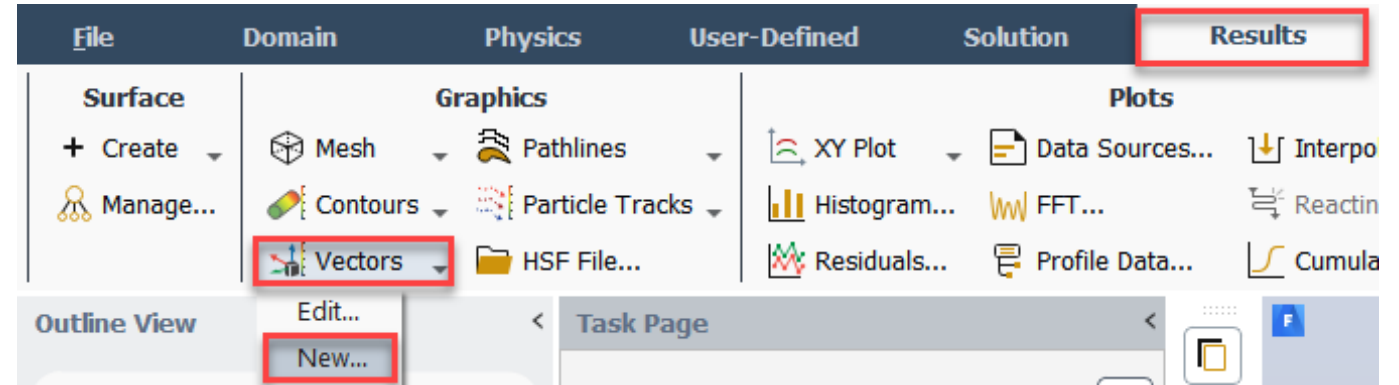 - Click *Create*
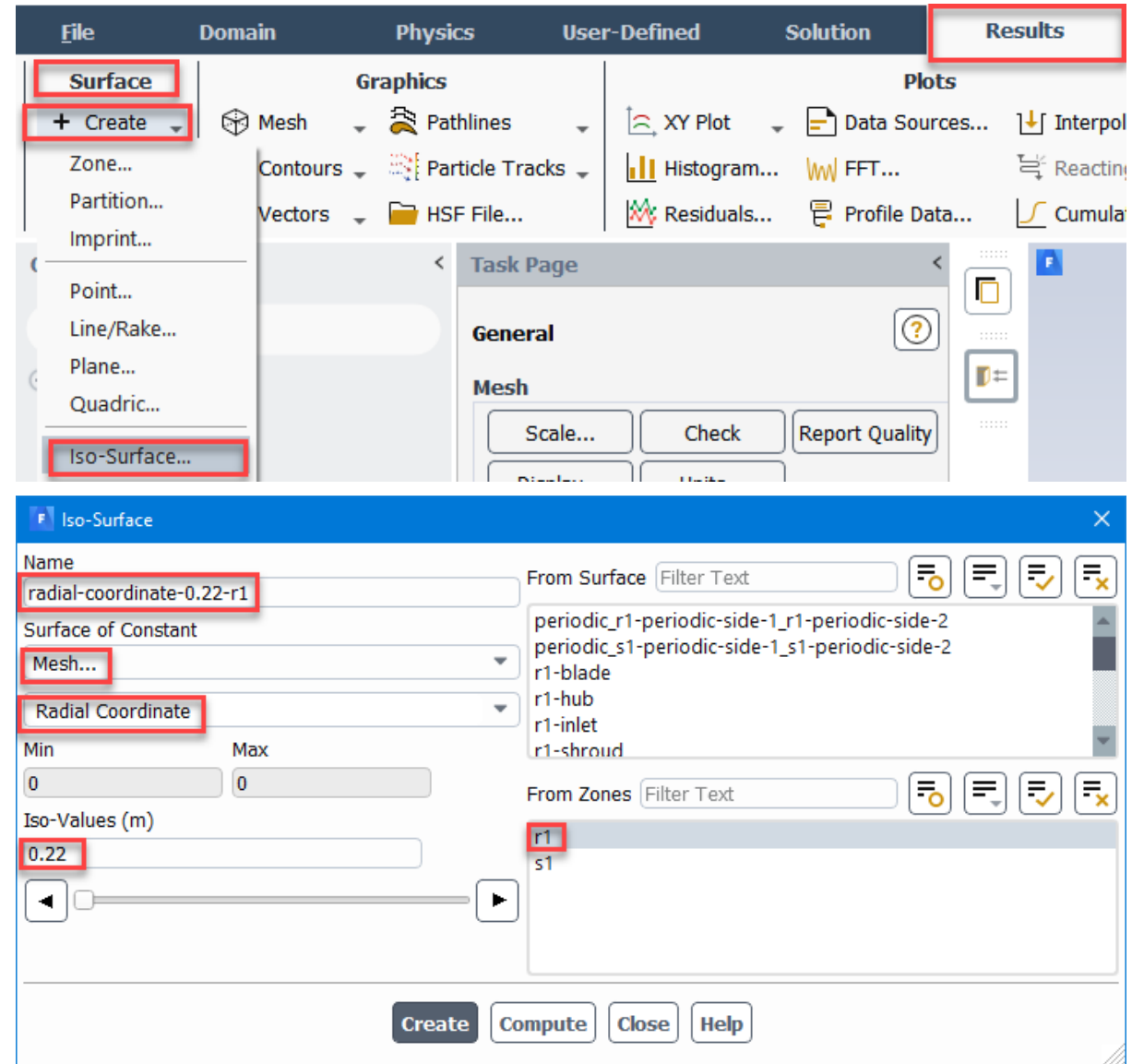
Create a Constant Radial Coordinate Iso-surface for Zone s1

- In the *Results* tab create a Constant Radial Coordinate Iso-surface for Zone s1

  - *Surface of Constant Mesh…*

  - *Radial Coordinate*

  - *Iso-Value = 0.22*

  - This corresponds to a midspan surface. We will use it for creating various vector and contour plots
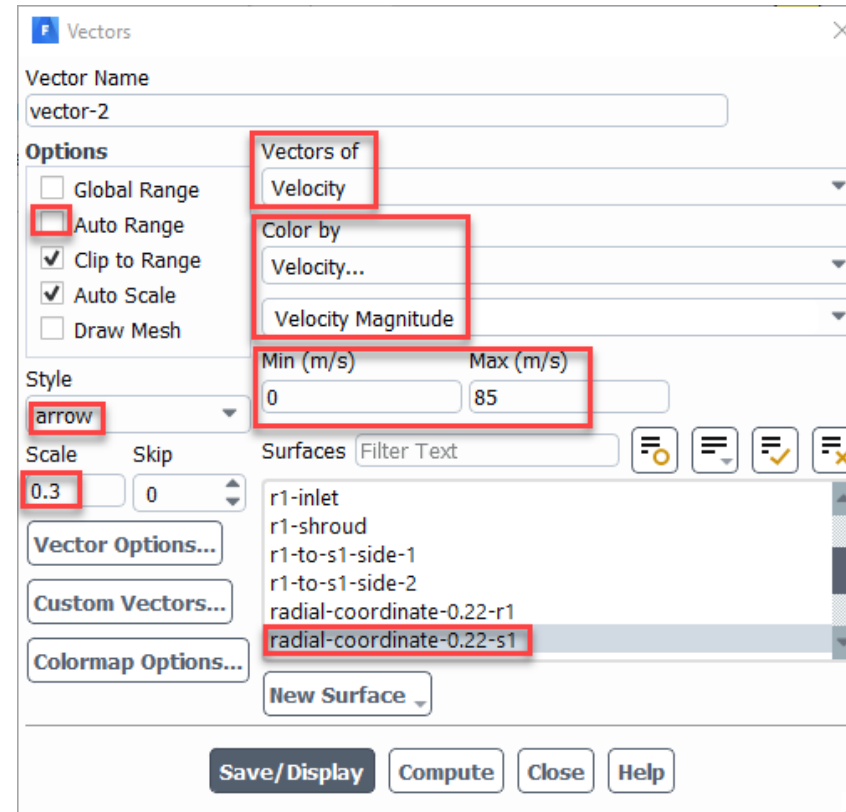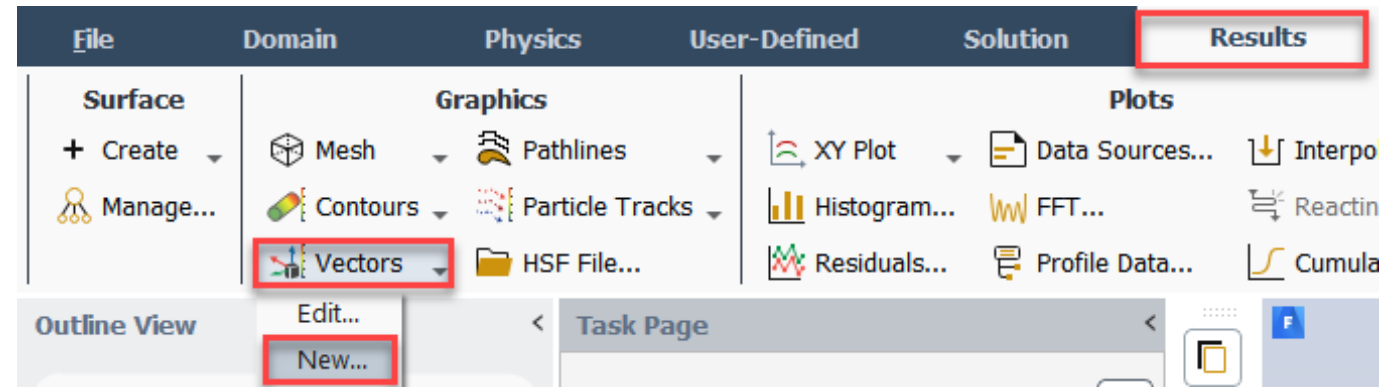
  - Click *Create*

Mid-span Relative Velocity Vector Plot Iso-surface for Zone r1

- Create a *New Vector* plot of *Relative Velocity* on the midspan plane *radial-coordinate-0.22-r1*
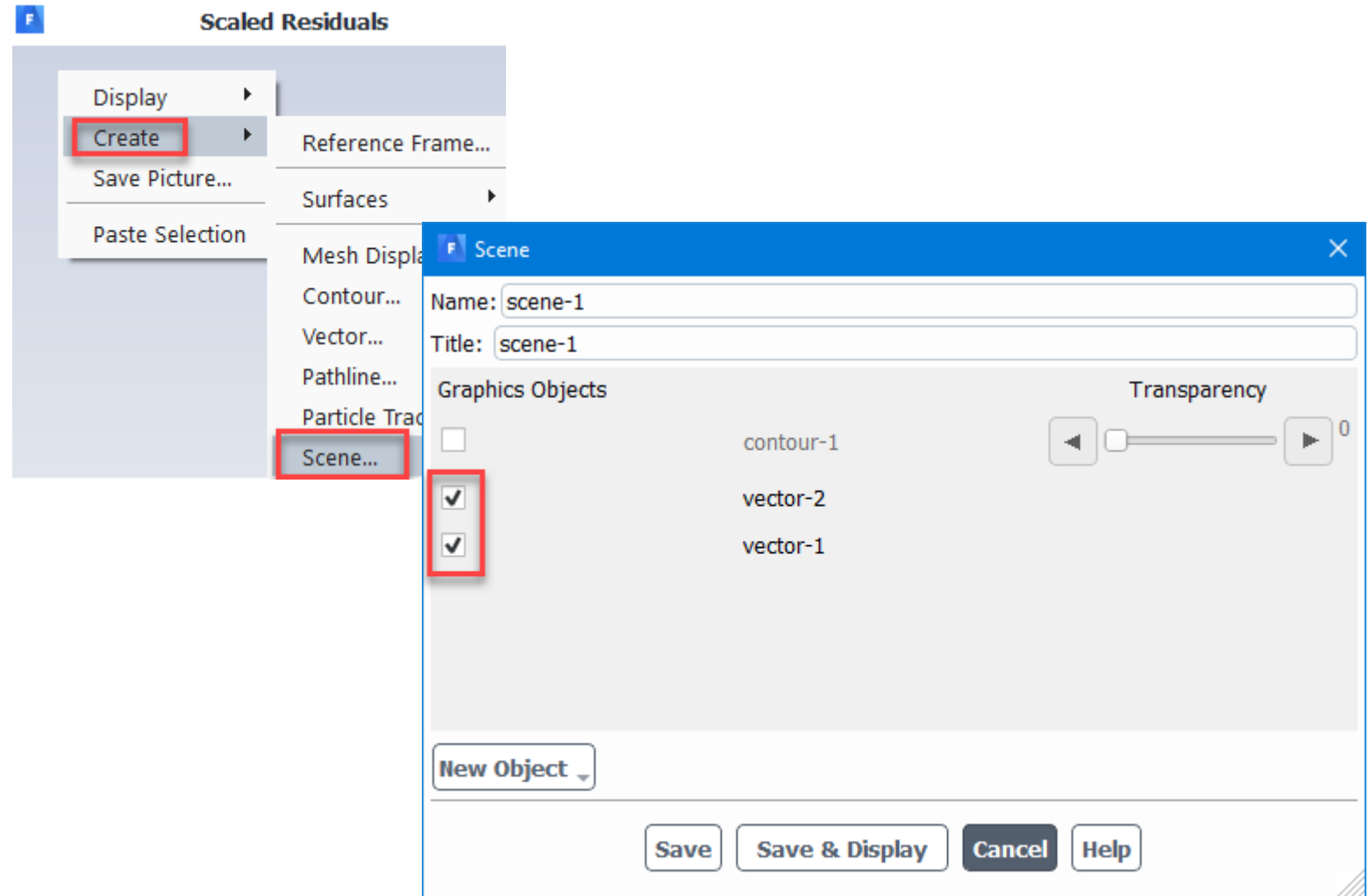
- Click *Save/Display*

Mid-span Velocity Vector Plot Iso-surface for Zone s1

- Create a *New Vector* plot of *Velocity* on the midspan plane *radial-coordinate-0.22-s1*
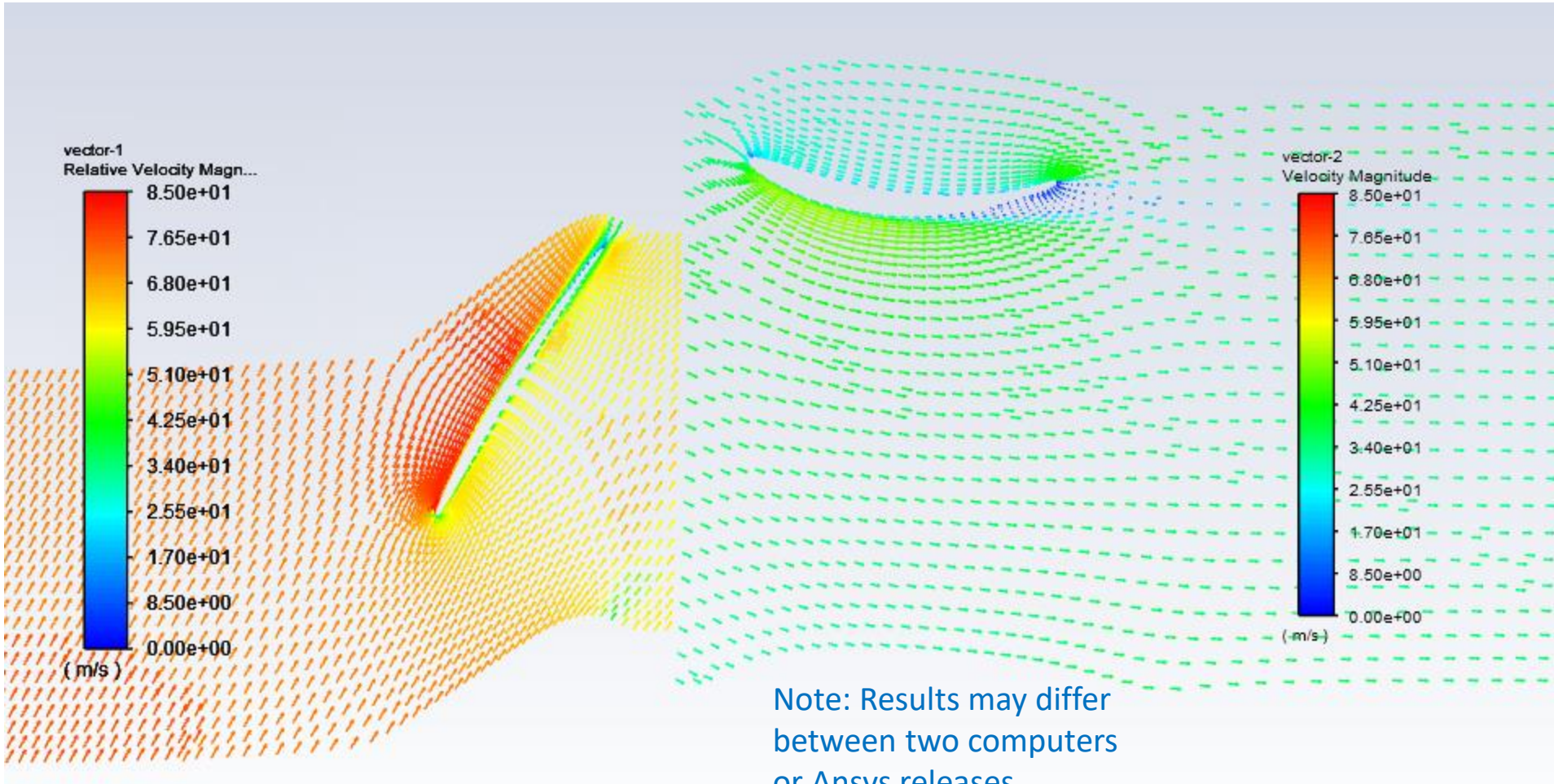
- Click *Save/Display*

Create a Scene for the 2 Vector Plots

- RMB on an empty in the graphics window and select *Create>Scene…*

- Select both vector plots and click *Save & Display*

Combined Vector Plot for r1 and S1

Note: Results may differ between two computers or Ansys releases

- When done, write the Fluent .cas and .dat files and exit Fluent

Summary

- This workshop has covered:
  - Setting up a steady stage calculation comprising a rotor and a stator
    - Defining a rotating frame
    - Applying rotational periodicity
    - Creating named expressions and report plots for monitoring the pressure rise and the power consumption of the fan rotor
    - Creating a Mixing Plane, General Turbo Interface
  - Solving and monitoring convergence
  - Visualizing the pressure distribution on the impeller walls and the relative velocity vectors at midspan

End of presentation