ANSYS Fluent Rotating Machinery Modeling

# Workshop 04.1: Pump with Volute

Release 2020 R2

Ansys

Introduction

- ## Workshop Description:
  - This Workshop deals with the Fluent setup and solution for Pump
    - The model consists of a pump impeller, connected to a casing (the volute)

- ## Learning Aims:
  - Setting up a steady stage calculation comprising a rotor and a stator
    - Defining a rotating frame
    - Creating a Frozen Rotor, General Turbo Interface
    - Creating named expressions and report plots for monitoring the head and the power consumption of the pump impeller
  - Solving and monitoring convergence
  - Visualizing the pressure distribution and velocity vectors on a plane of constant axial coordinate
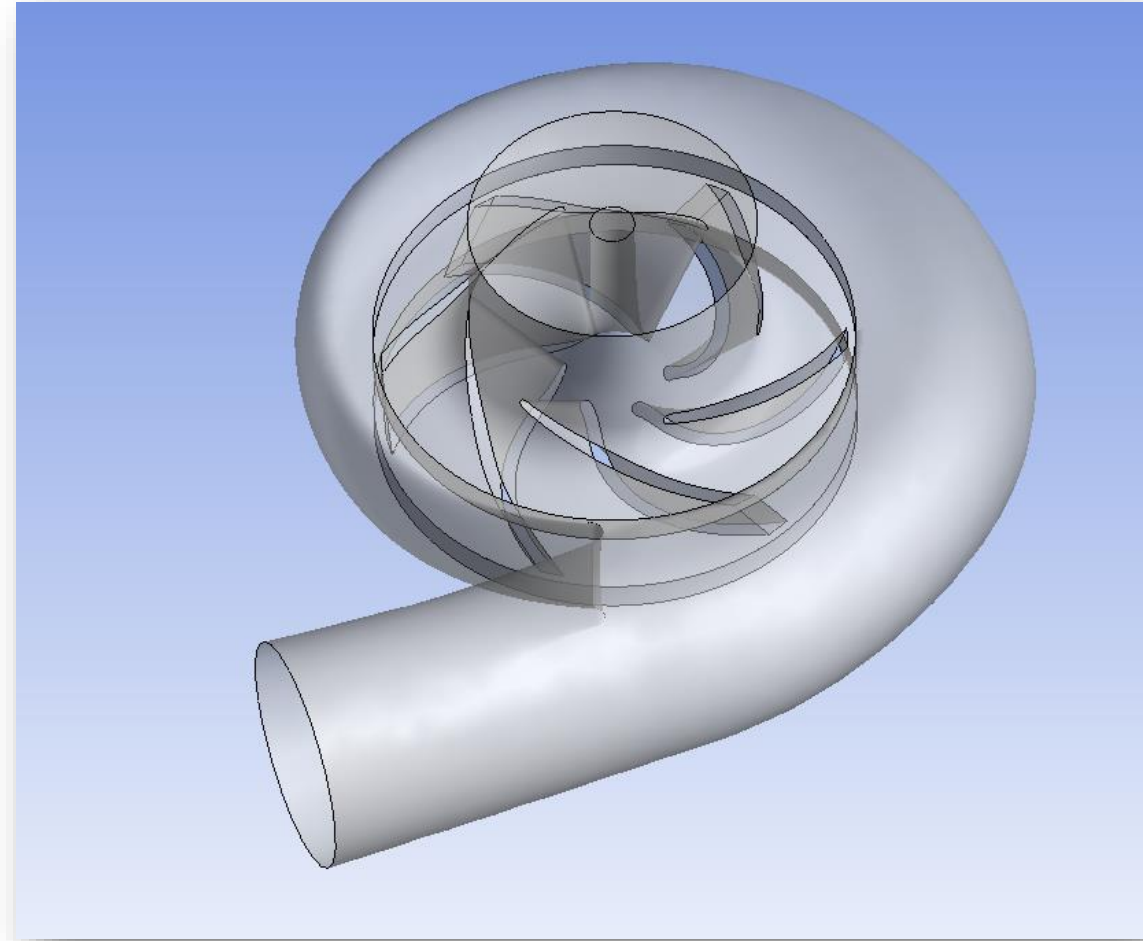
Pump Model

- A rotating component (impeller), followed by a stationary component (volute)
  - A moving reference frame is used to solve the rotating component

- Pump data
  - Fluid = Water
  - Speed = 1450 rpm
  - Number of Blades = 6
 - Flow Rate = 77.5 kg/s
  - Axis or rotation = z-axis
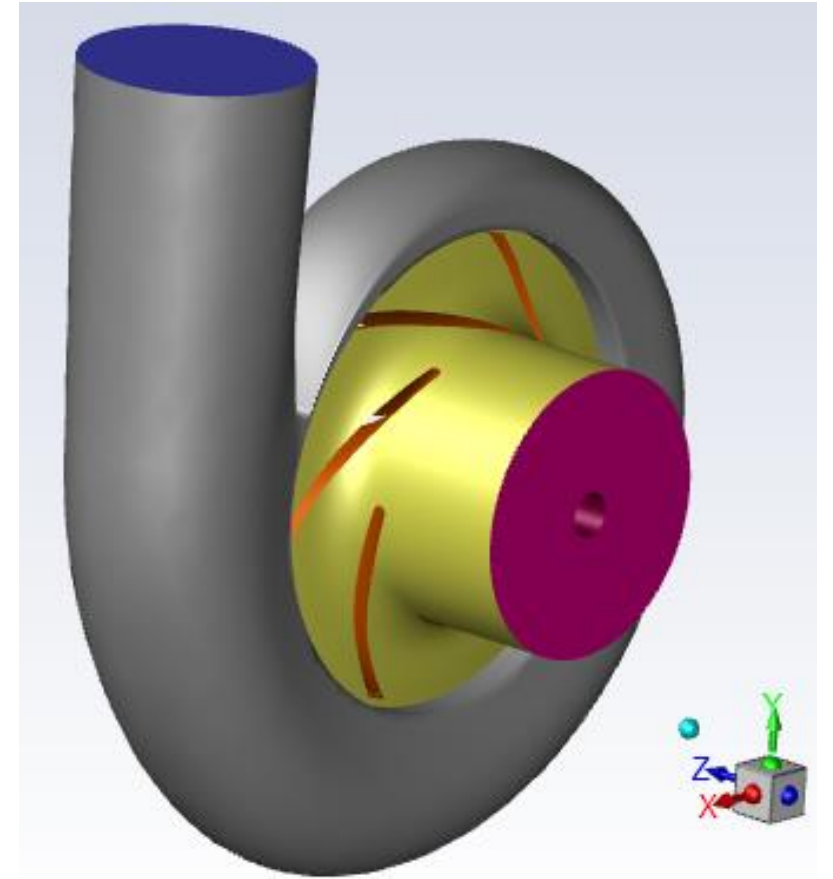
Start Fluent Launcher

- 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 1,100,000 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 55 Processes (so that each Processor is solving for not less than 20,000 cells)
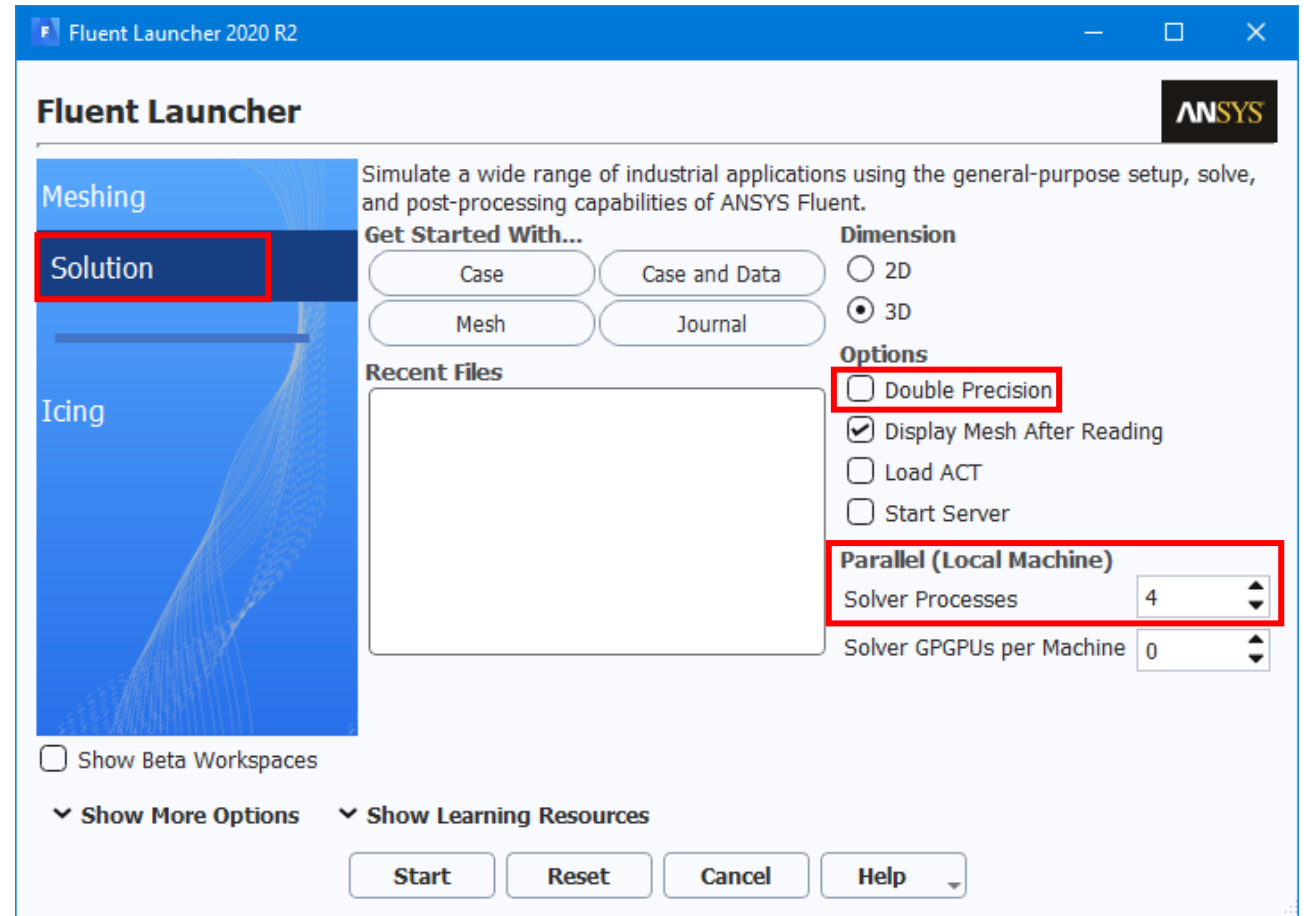
Fluent

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

- In the graphics window, you should see the geometry of the pump and volute as shown on the right
  - The mesh comprises the complete impeller and the volute

- 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 1,100,000
  - *Quality* > *Evaluate Mesh Quality* will show you a *Maximum Aspect Ratio* of 1.78e+02 <1000
   - This justifies the choice of starting Fluent in Single Precision
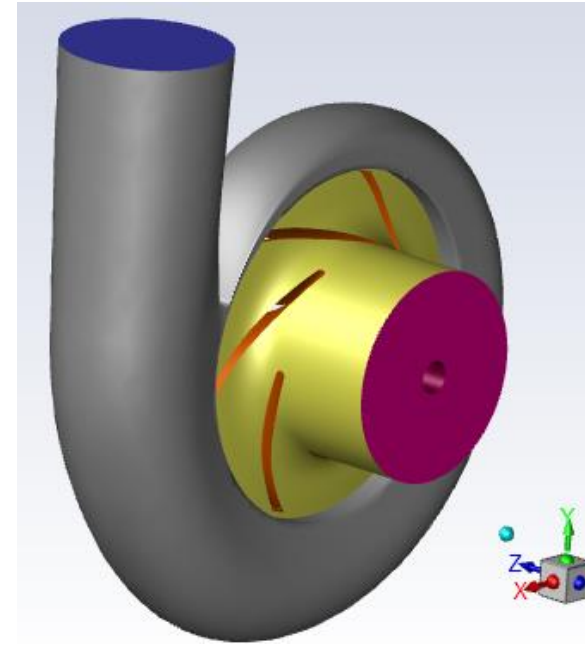
For cases with two different mesh files, one for the impeller and one for the volute:
- Read first one mesh file *File>Read>Mesh...*
- When the first mesh is displayed read the second mesh using *Append* in the *Zones* group of the *Domain* tab
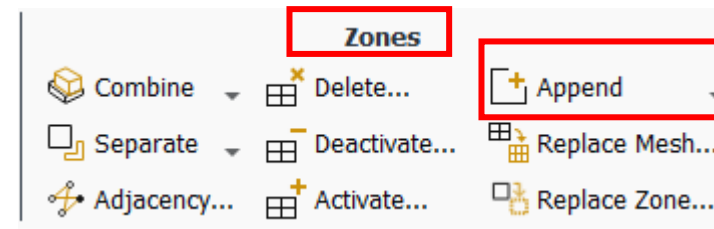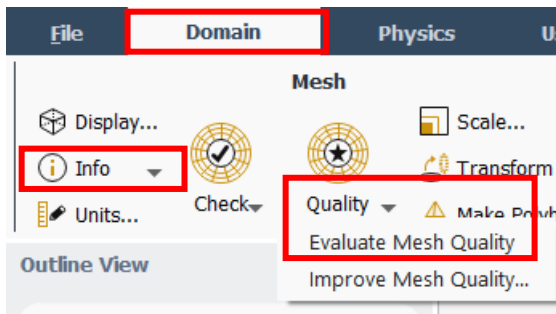
Physics: General & Operating Conditions

- Retain the default solver settings of *Pressure-Based* solver with *Absolute Velocity Formulation*
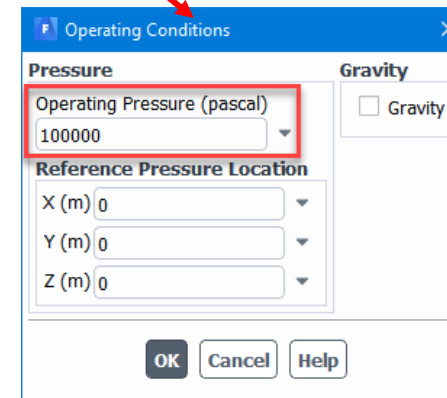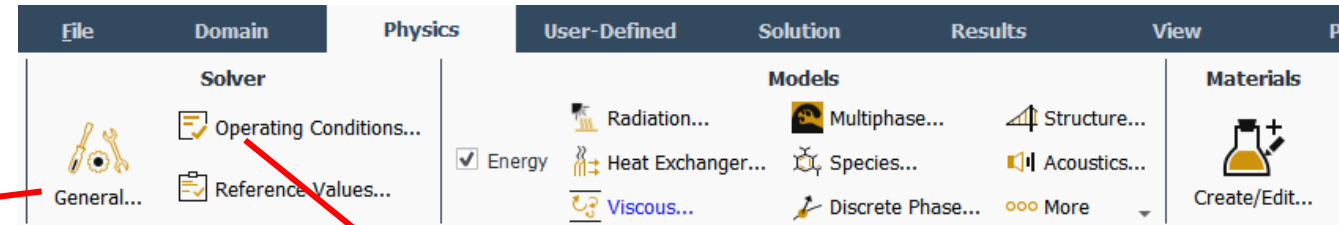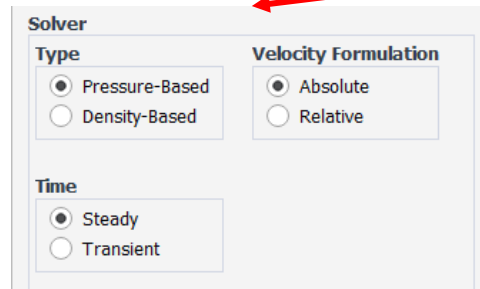
- In the *Operating Conditions* panel set the *Operating Pressure* to 0 (Pa)

  - The pressure difference (inlet to outlet) is expected to be much larger that 1 bar for this case

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
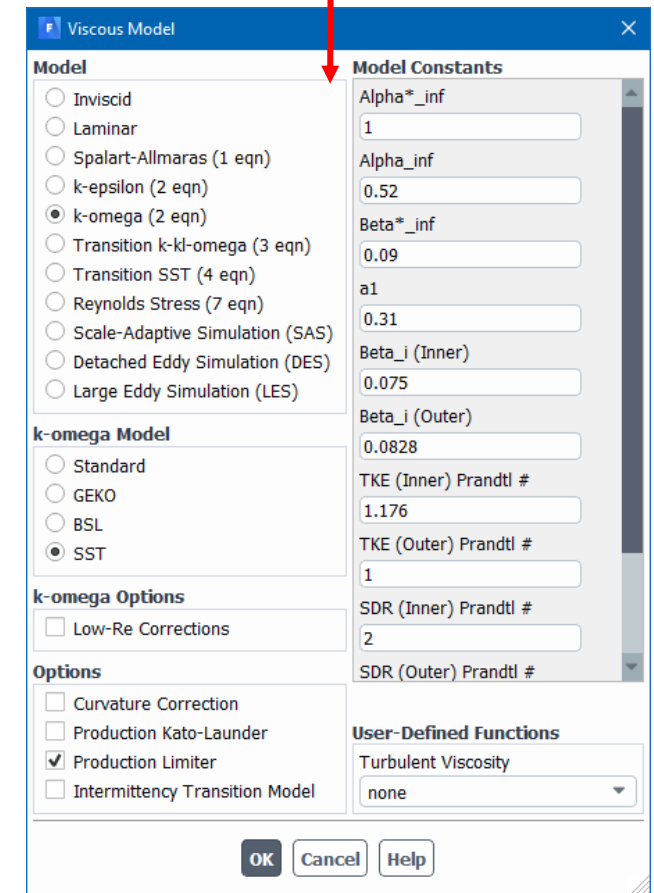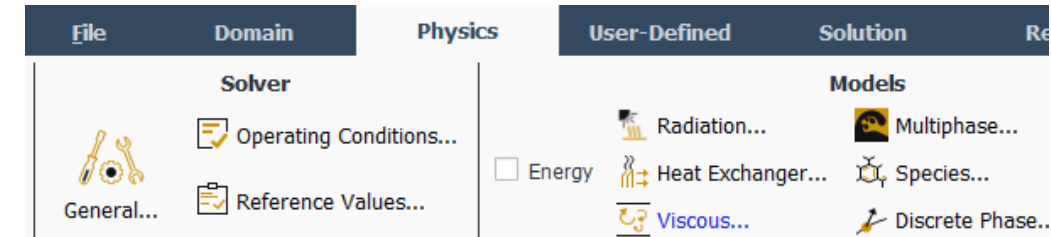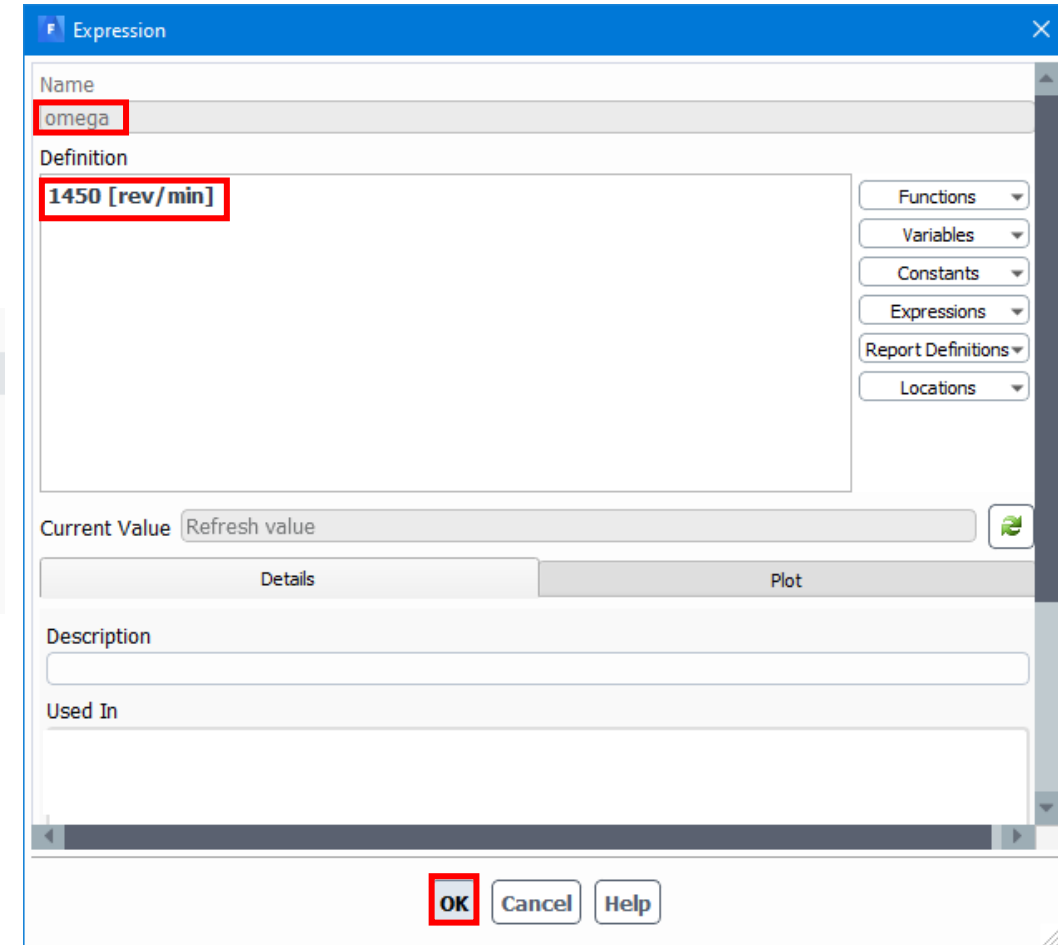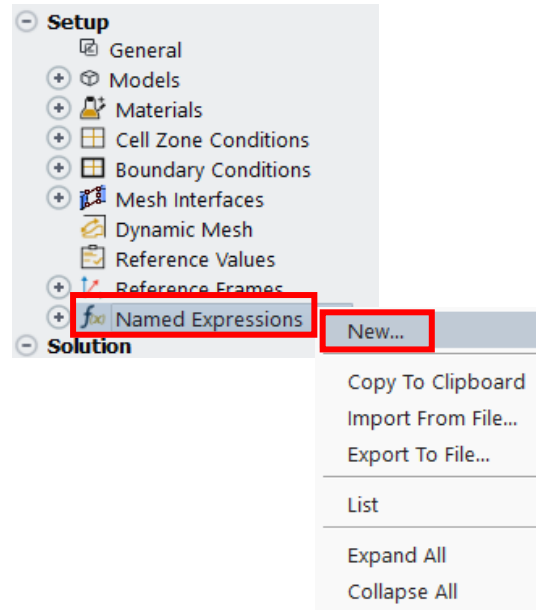
Physics: Materials

- 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
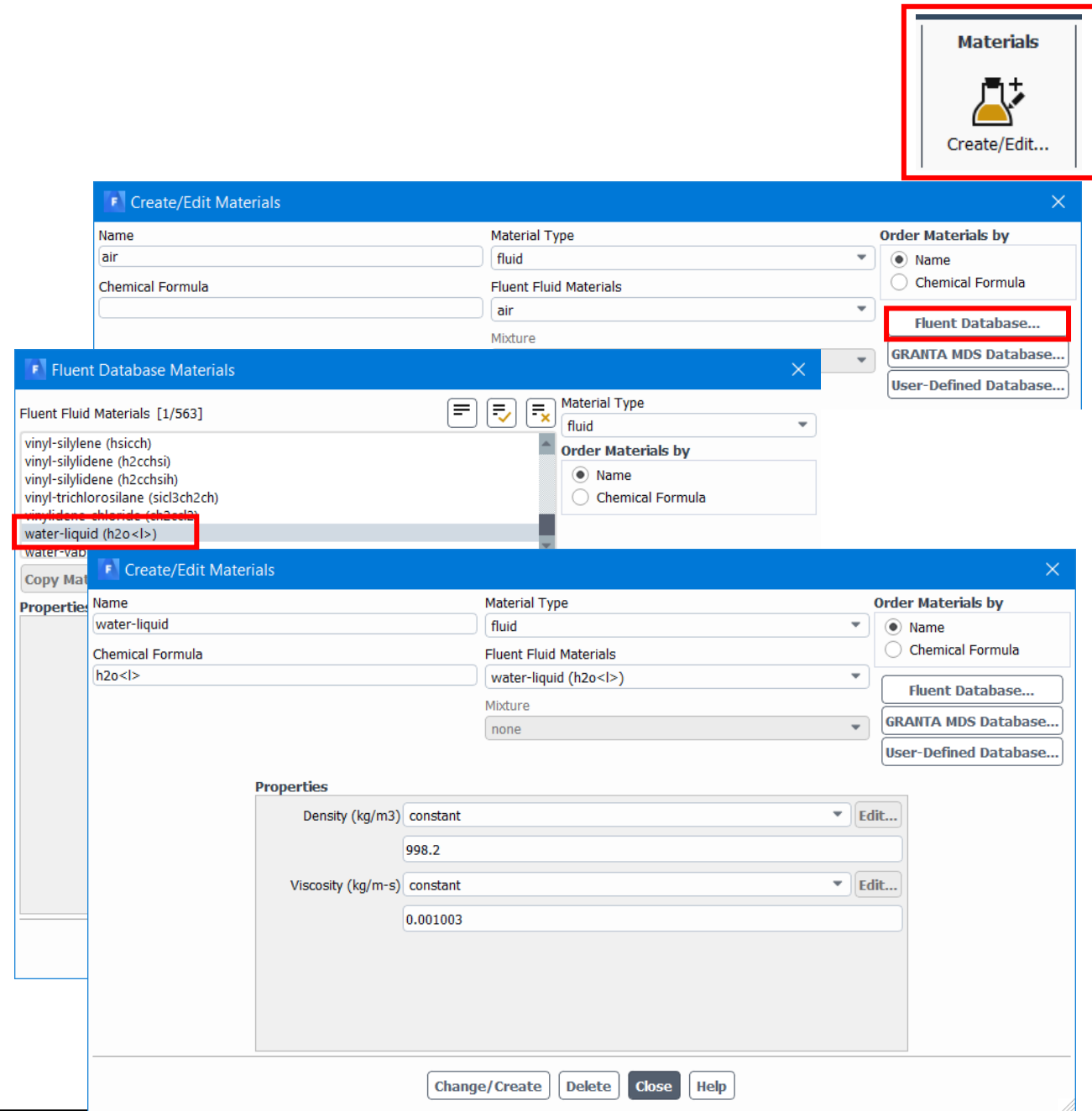
Create a Named Expression for Rotational Speed

- The pump impeller is rotating with a rotational speed of 1450 rpm

- We will need to set this in the conditions for the *impeller* 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*,
  - *1450 [rev/min]* under *Definition*
 - Click *OK*

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
  - 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
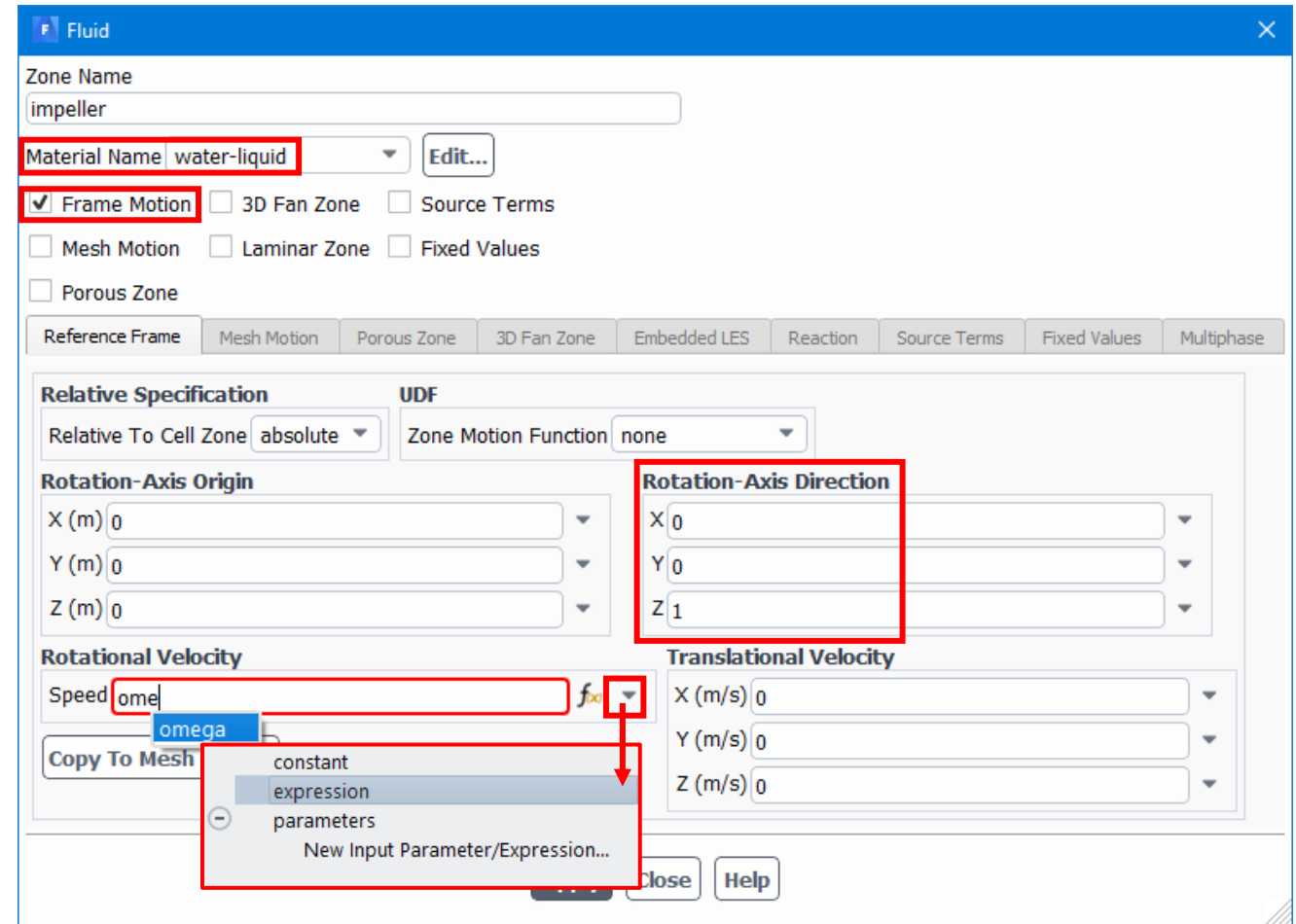
Physics: Cell Zone Conditions (2)

- Continuing from previous slide…

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

- Click *Apply*

- The rotational speed has been set via an expression *omega= 1450 [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 impeller power
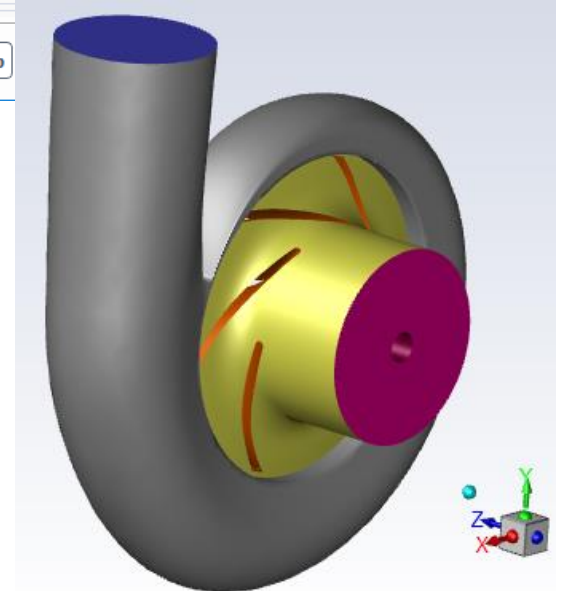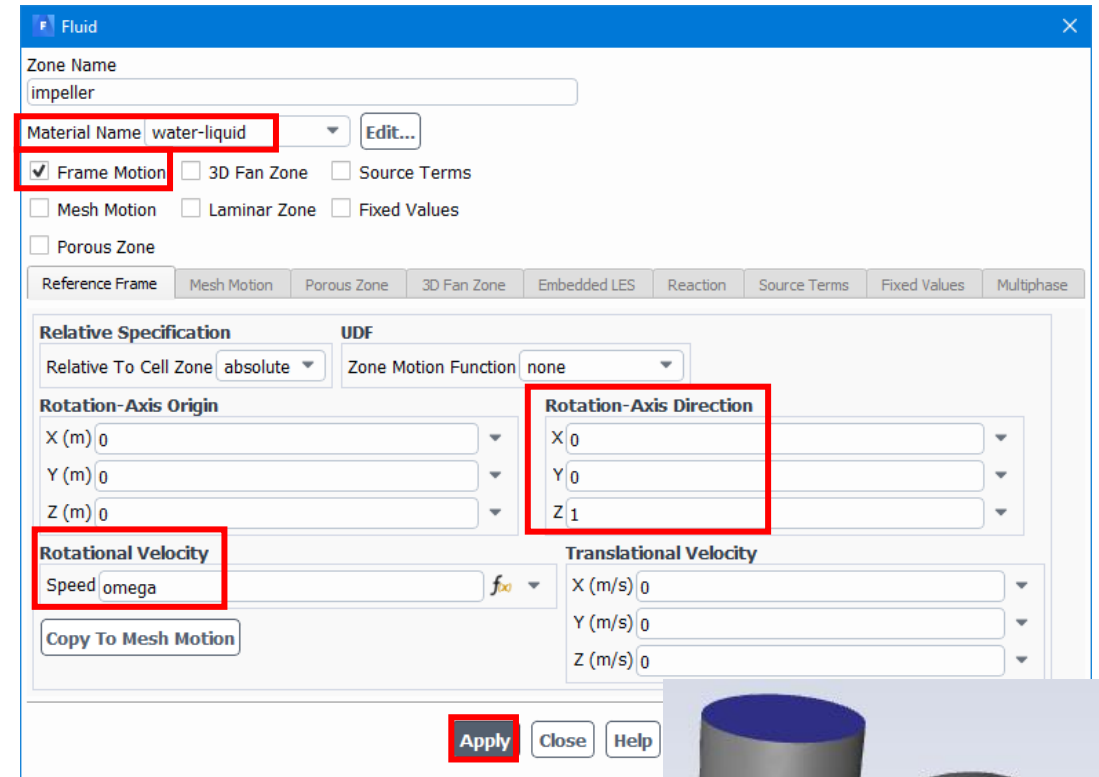
Physics: Cell Zone Conditions (3)

- Edit the *volute* cell zone
  - Select *water-liquid* as *Material Name*
 - Click *Apply*

Boundary Conditions: Inlet

- Set the boundary conditions for *impeller-inlet*
  - *RMB* on *impeller-inlet* and set *Type* to *pressure-inlet*
  - Check *Prevent Reverse Flow*
  - Set a *Gauge Total Pressure* of 100000 (pascal) at the inlet *
  - Set a Supersonic/Initial Gauge Pressure of 98000 (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 24)
  - Accept all remaining defaults in the Momentum tab
  - Click *Apply*

  * Note that the Operating Pressure was set to 0 (Pa) for this case. For this reason, the inlet Gauge Total Pressure is set to 100000 (Pa)
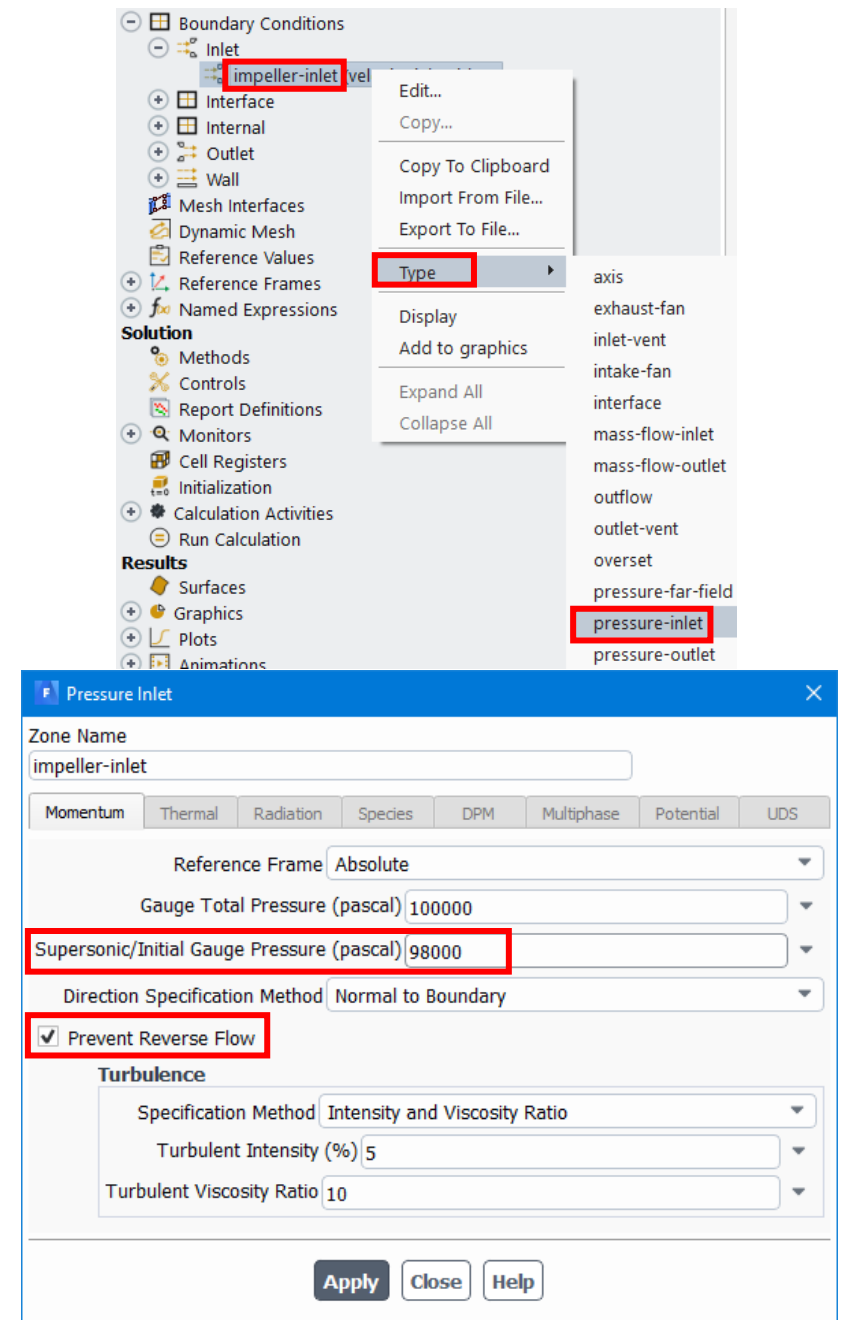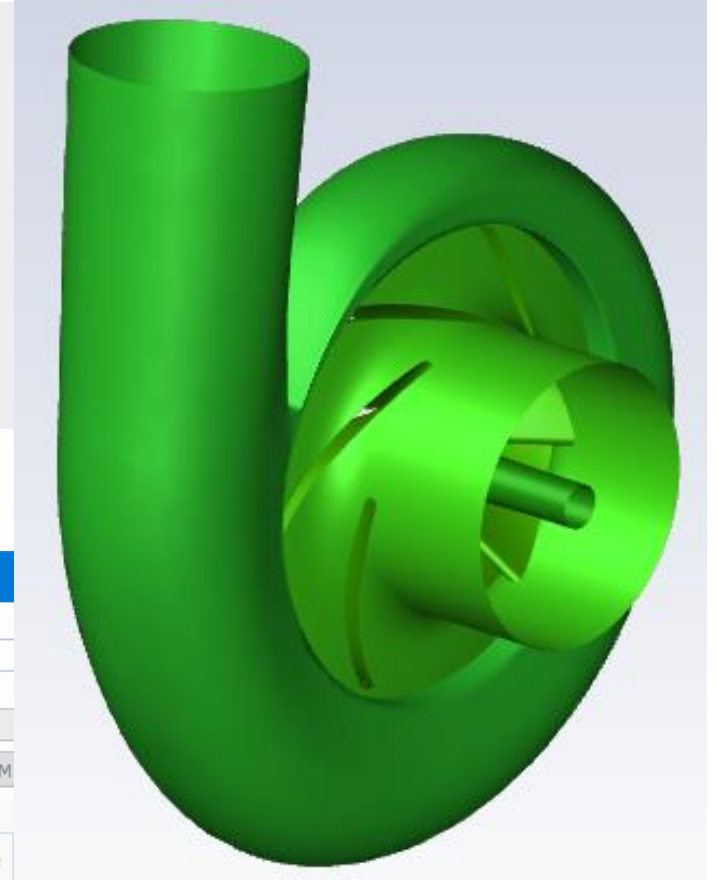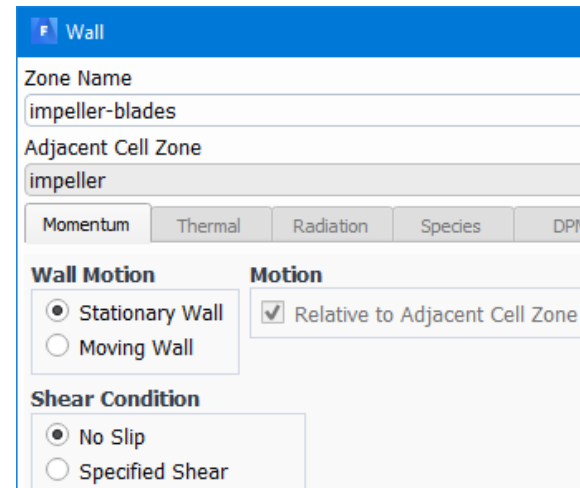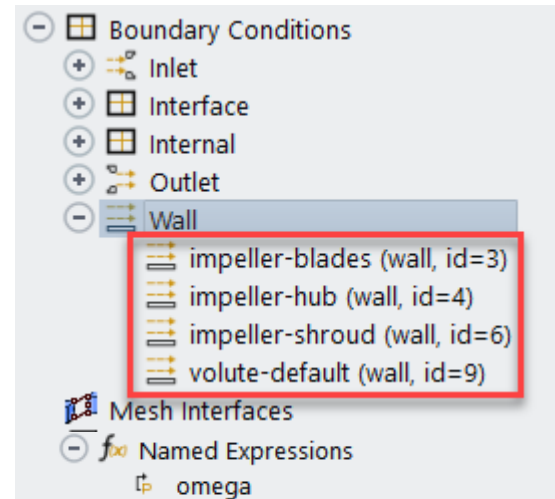
Boundary Conditions: Outlet

- Set the boundary conditions for *volute-outlet*
  - *RMB* on *volute-outlet* and set *Type* to *mass-flow-outlet*
  - Set the *Mass Flow Rate = 77.5(kg/s)*
 - Click *Apply*
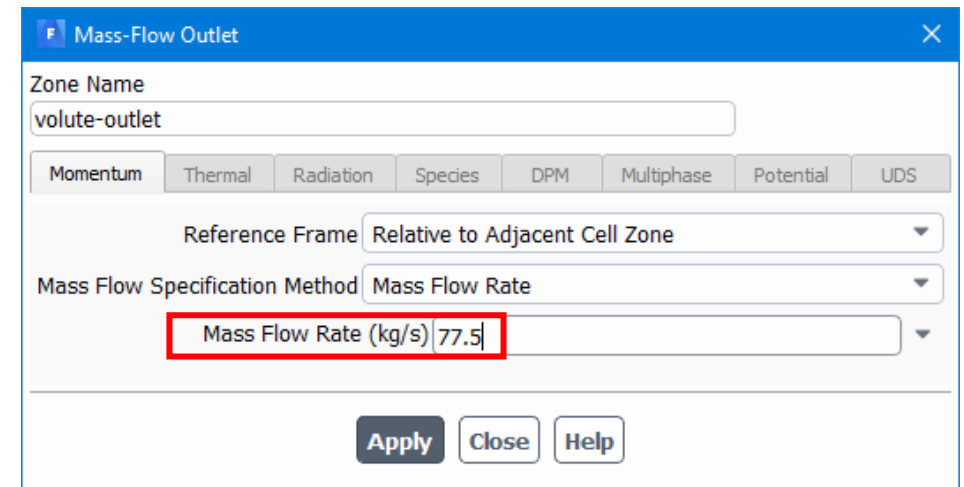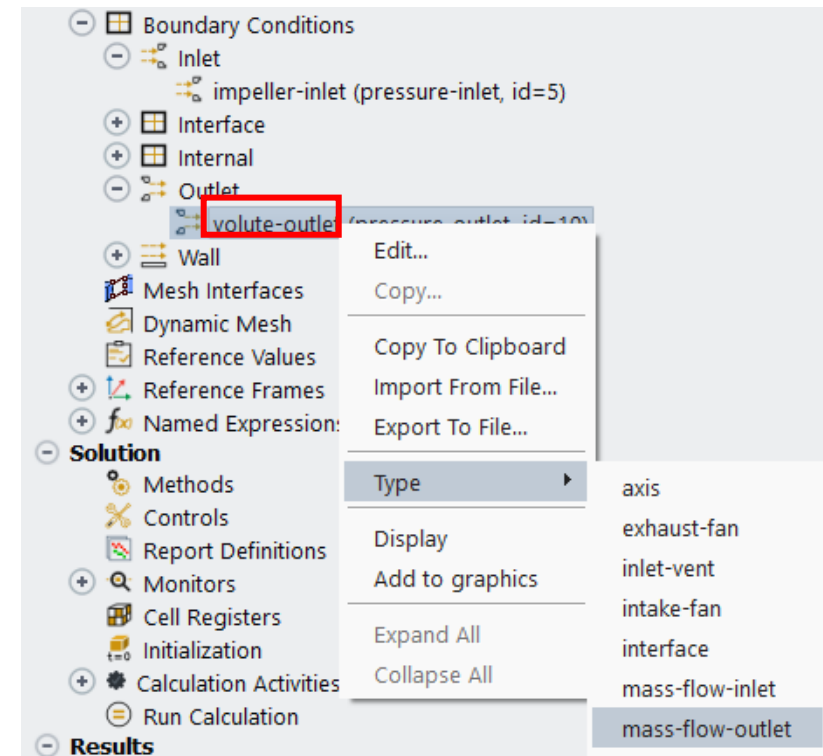
Boundary Conditions: Automatically Set Walls

- From the remaining zones under *Wall* in the *Outline View*, all boundaries correspond to walls, which can be left to the default wall boundary condition setting:
  - *Stationary Wall*
 - *Relative to Adjacent Cell Zone*
  - *No Slip*

Define a Frozen Rotor Interface

- Having concluded with all basic boundary conditions, you will now define a *Frozen Rotor 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 *impeller-to-volute*
    - Check *General Turbo Interface*
    - Select the two sides of the interface
    - Select *No Pitch-Scale*
    - 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
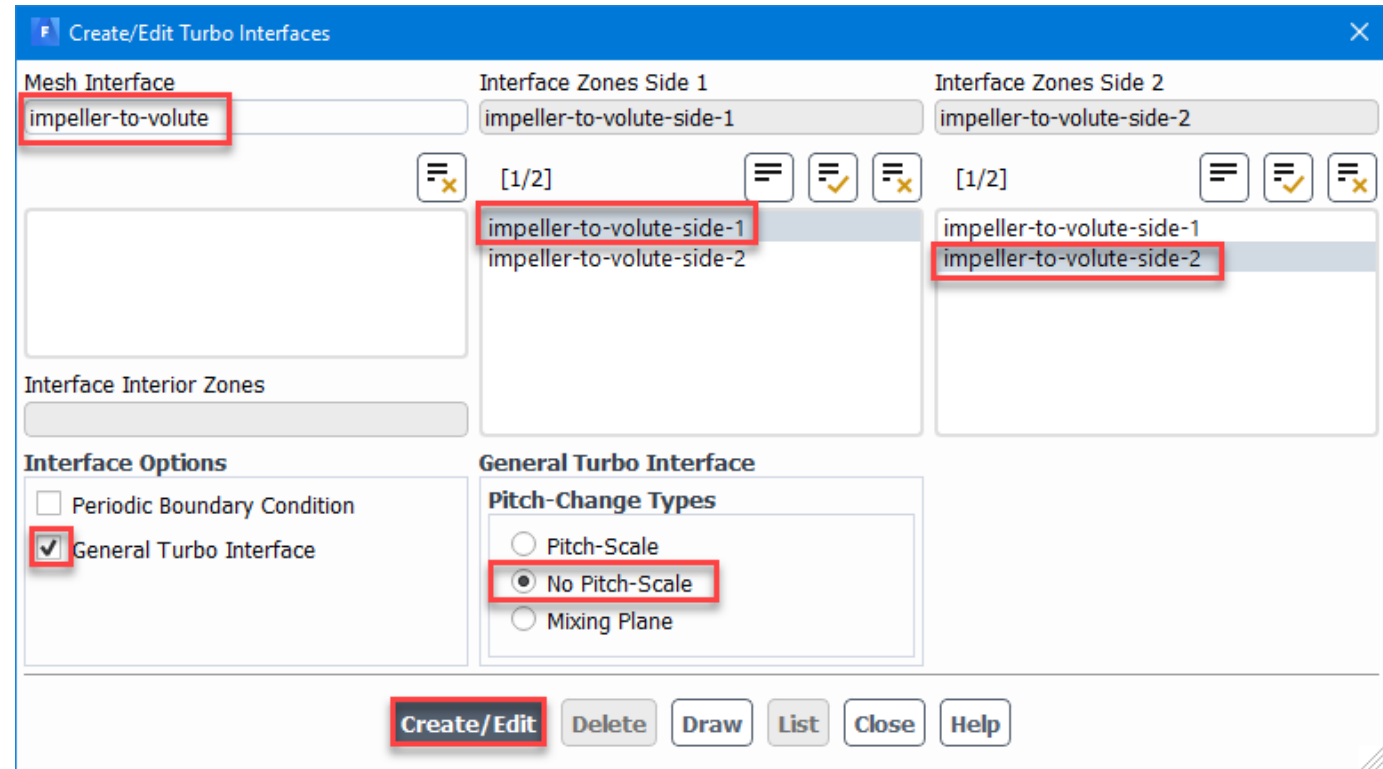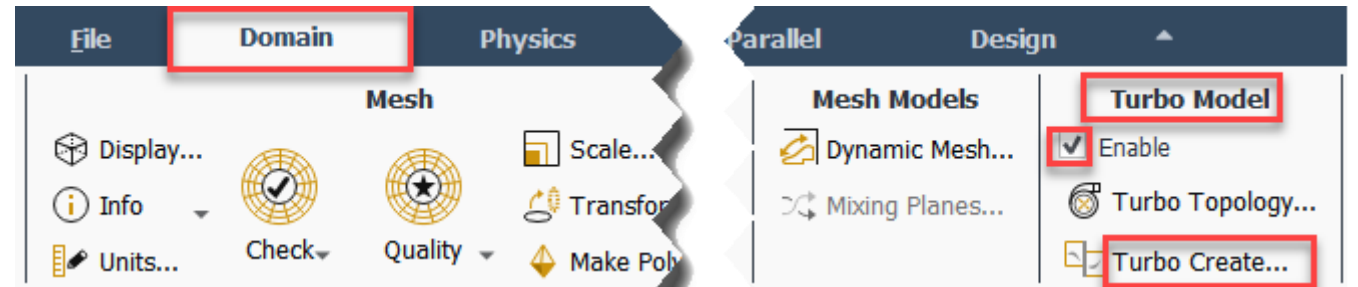
Solution: Moment Report Definition

- 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 = all 3 impeller walls*

  - *Report File = checked*

  - *Report Plot = checked*

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

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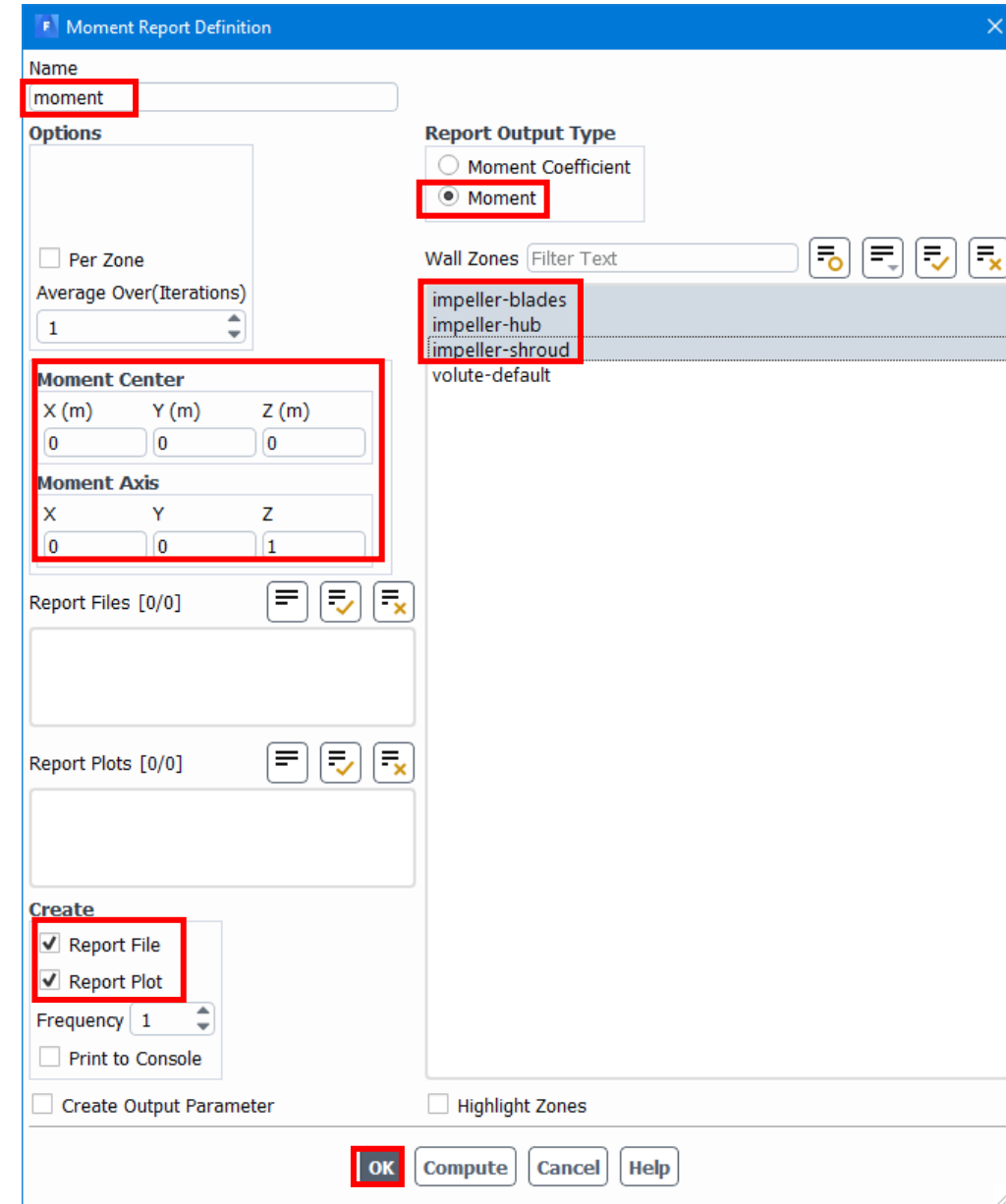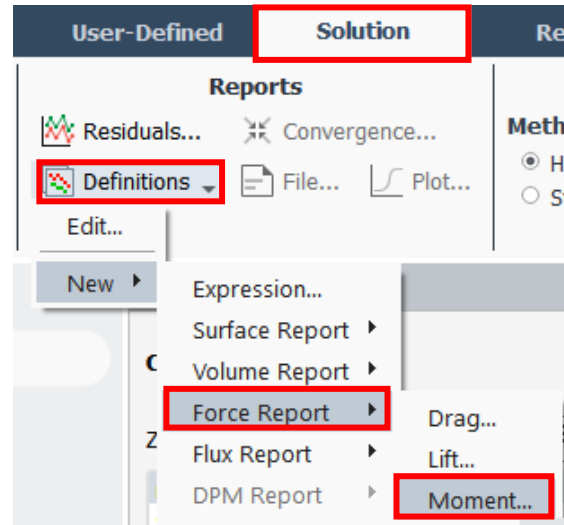
New Method for Report Definitions Using Named Expressions

- 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 *PumpVolute.tsv* is provided with the workshop inputs, containing the syntax for 5 named Expressions:

```
name              definition          description        input-parameter output-parameter
"ave_ptot_in"  "Average(TotalPressure,['impeller-inlet'],Weight = 'MassFlowRate')"   ""          #f        #f
"ave_ptot_out" "Average(TotalPressure,['volute-outlet'],Weight = 'MassFlowRate')"    ""          #f        #f
"deltapt"       "ave_ptot_out-ave_ptot_in"    ""         #f          #f
"den"           "998.2 [kg m^-3]"    ""          #f          #f
"head"          "deltapt/(den*g)"    ""          #f          #f
```
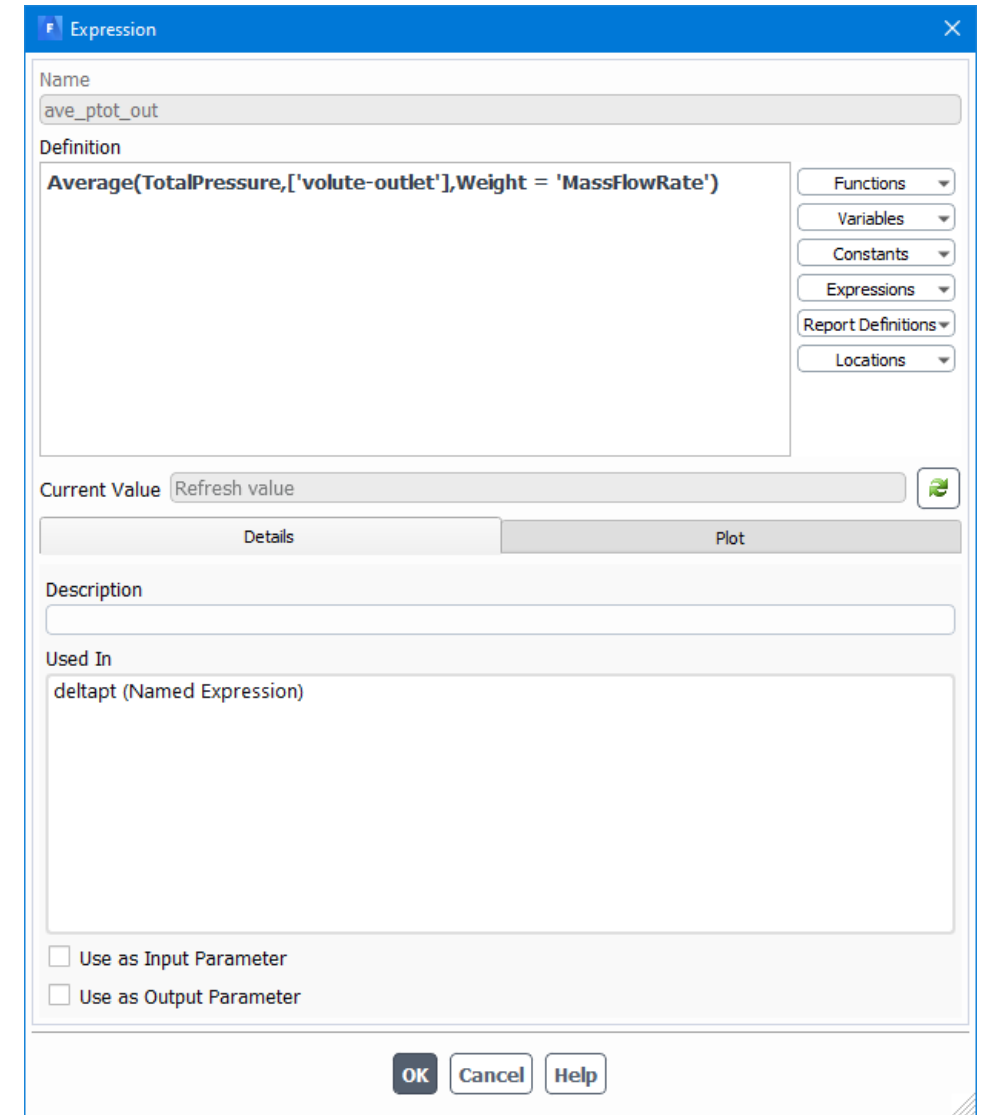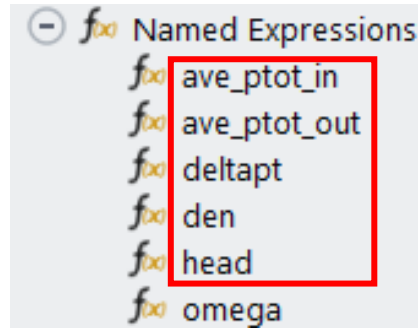
  You can import this file into the *Named Expressions* branch of the *Outline* using *RMB>Import From File…*
 - The 5 *Named Expressions* highlighted by a red box in the bottom-right image are created
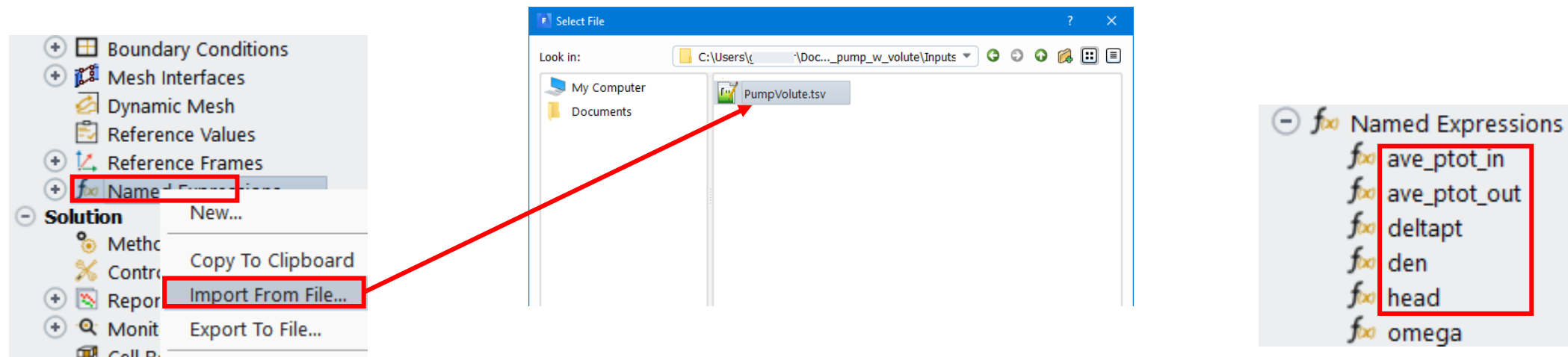
New Method for Report Definitions Using Named Expressions (2)

- 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 = head
  - Expressions > head
  - Report File = checked
 - Report Plot = checked
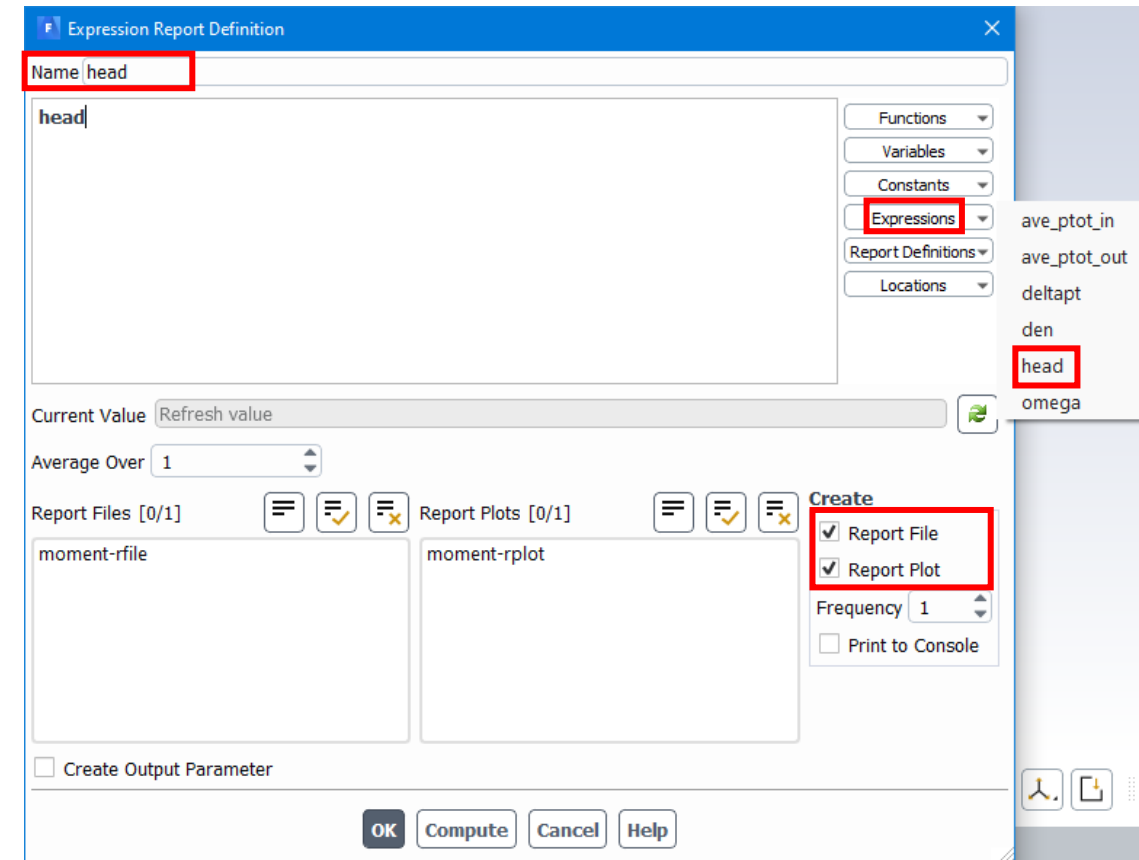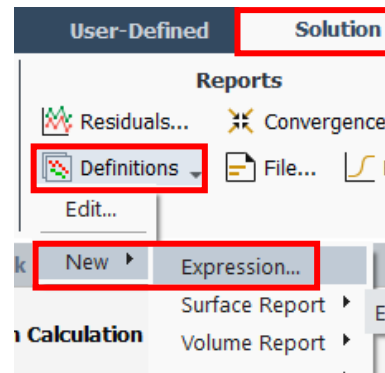
Solution: Create Report Definition for the Pump Power

- 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*

  - *Report File = checked*

  - *Report Plot = checked*
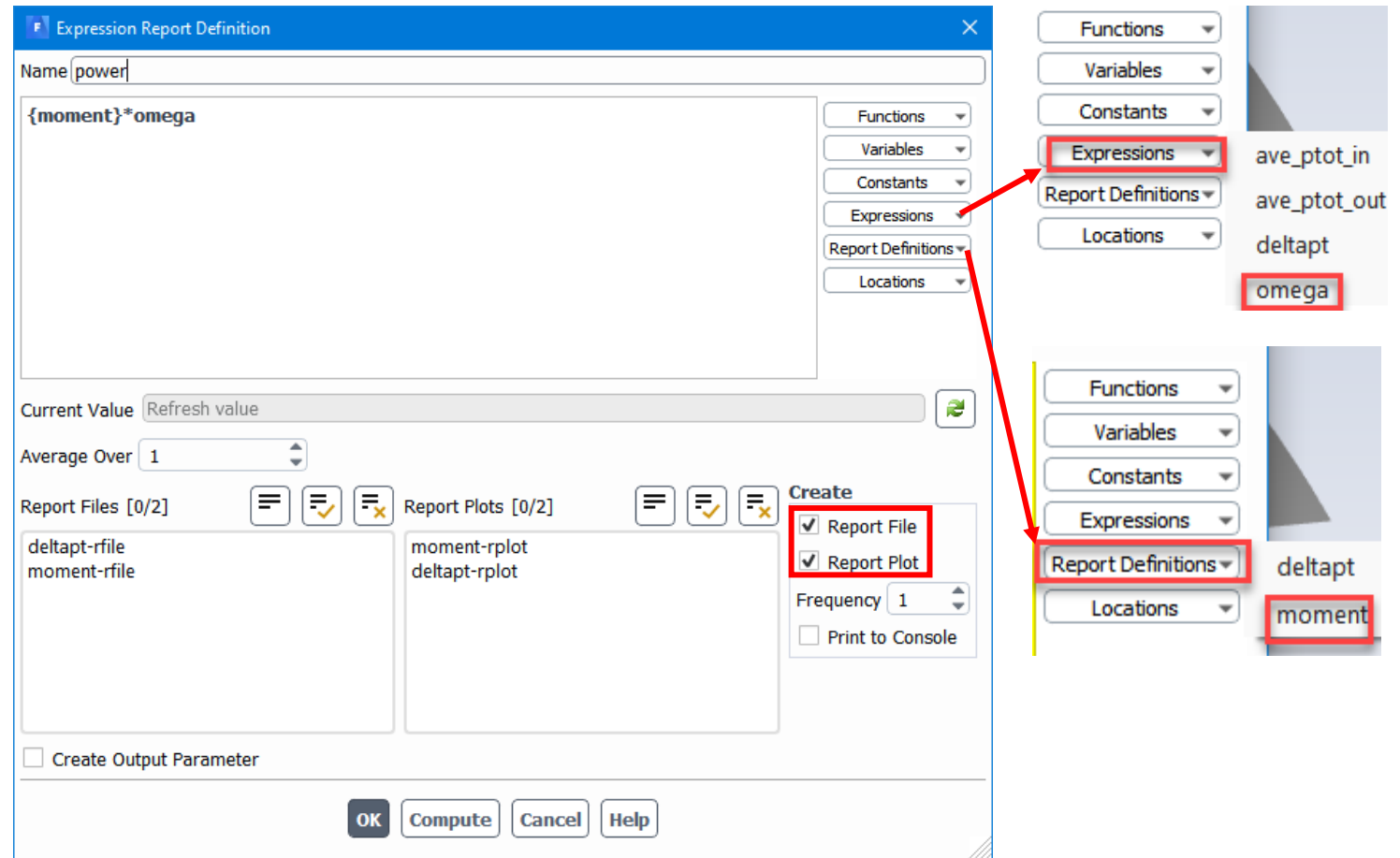
Solution: Solution Methods

- 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)
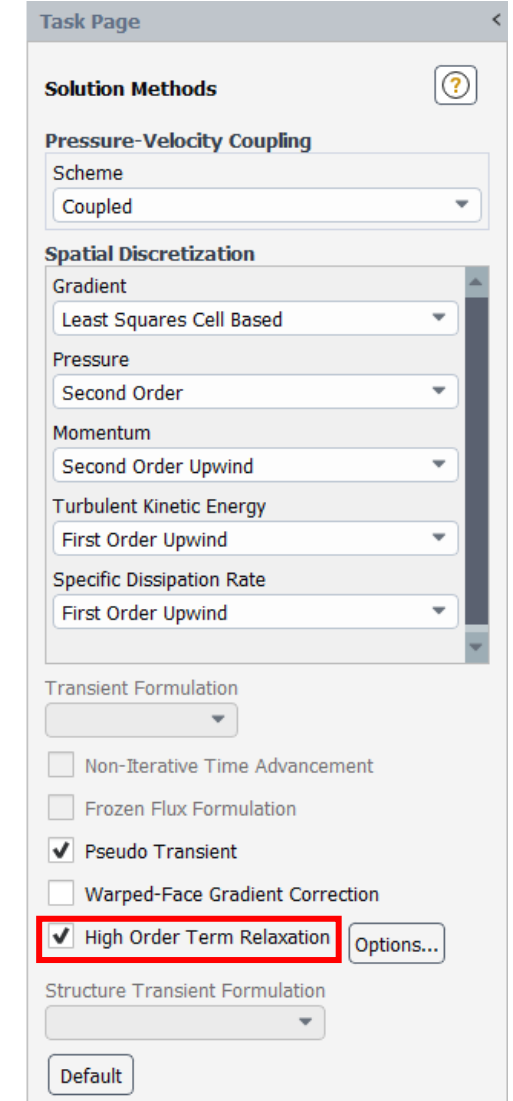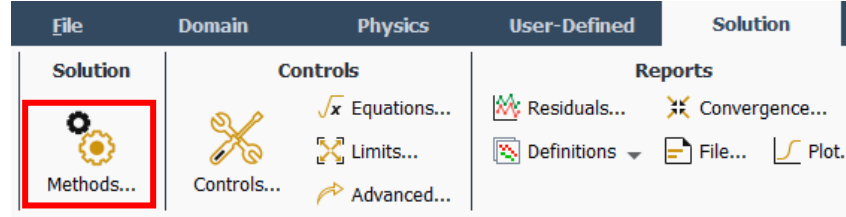
# Solution: Initialization (Best Practice Procedure)

- 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
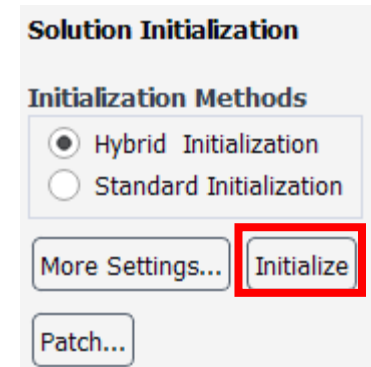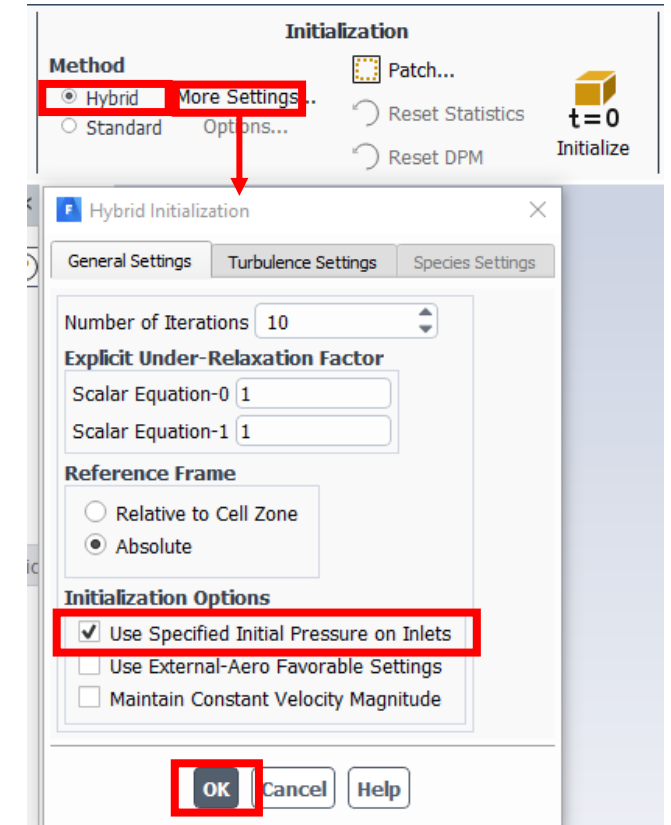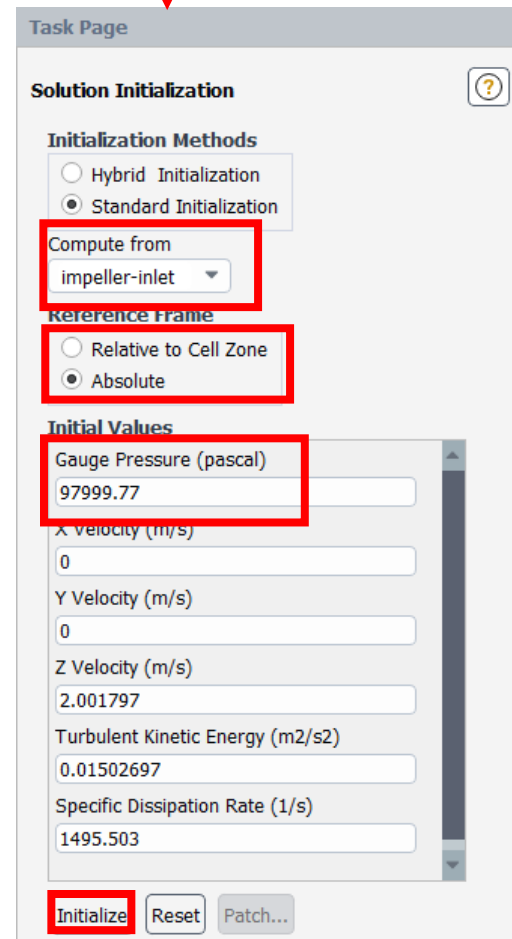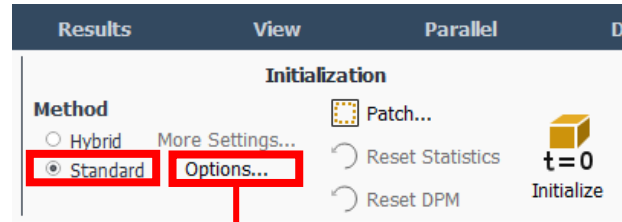
Solution: Run the Solver

- 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*
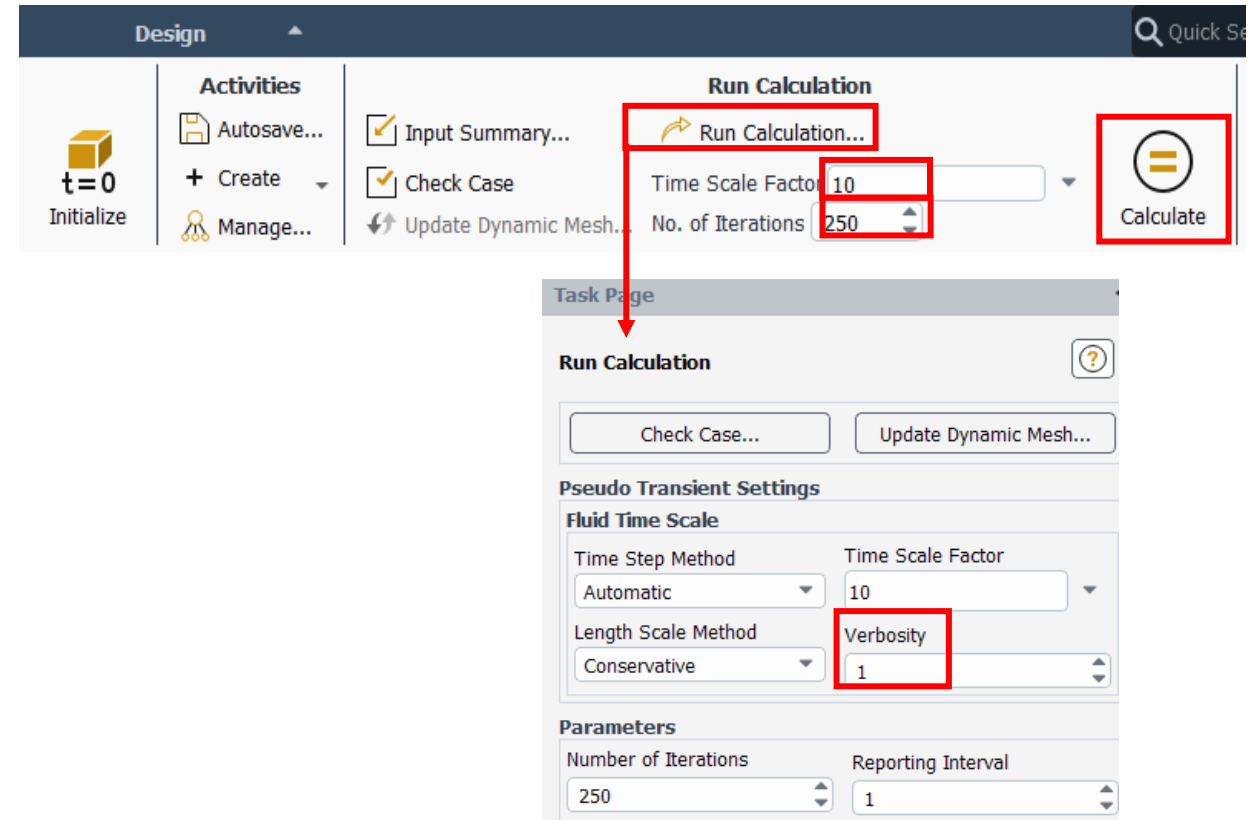
Solver Convergence

- The case does not converge well to a steady state solution after 250 iterations
  - Residual for continuity drops by 2 orders of magnitude
  - All rest residuals drop to values close to, or less than 1e-03
  - Residuals and report plots show a bouncy behavior

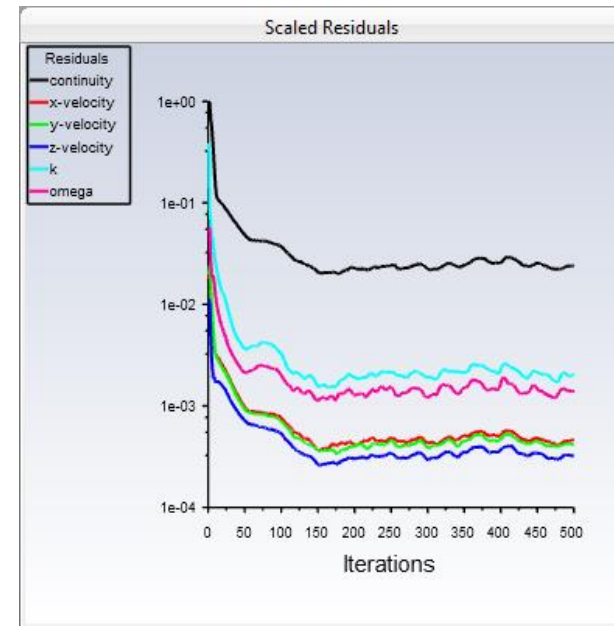Note: Residuals and monitor plots may differ between two computers or Ansys releases
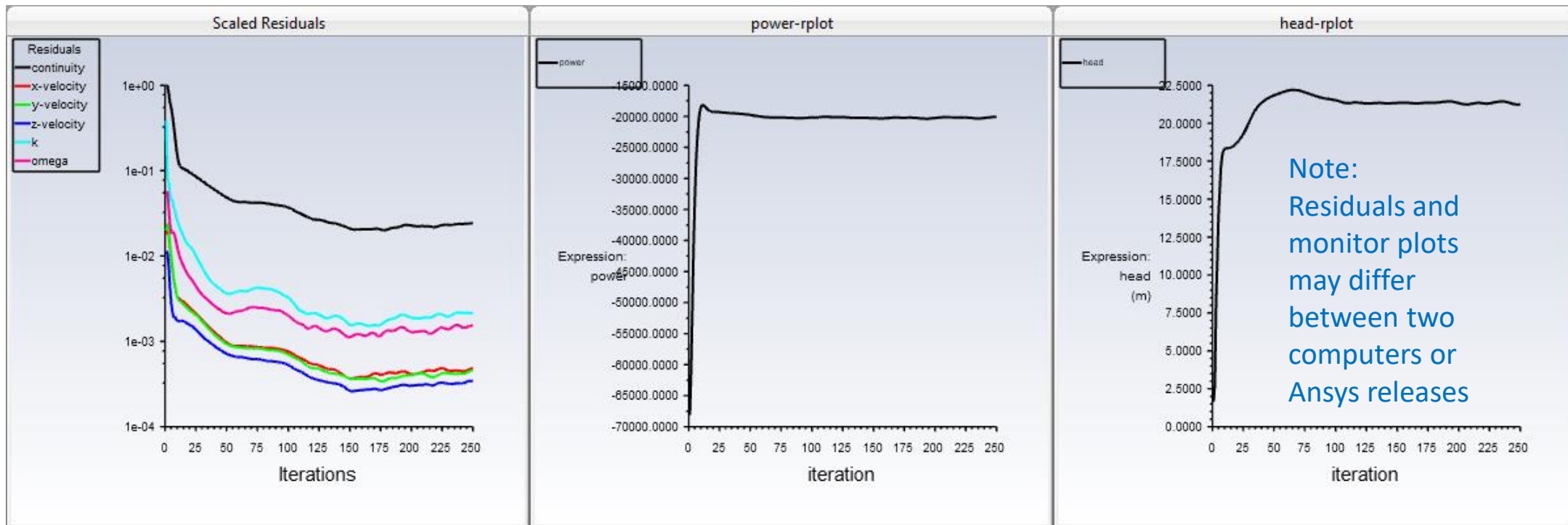
Check Solver Convergence Further

- 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
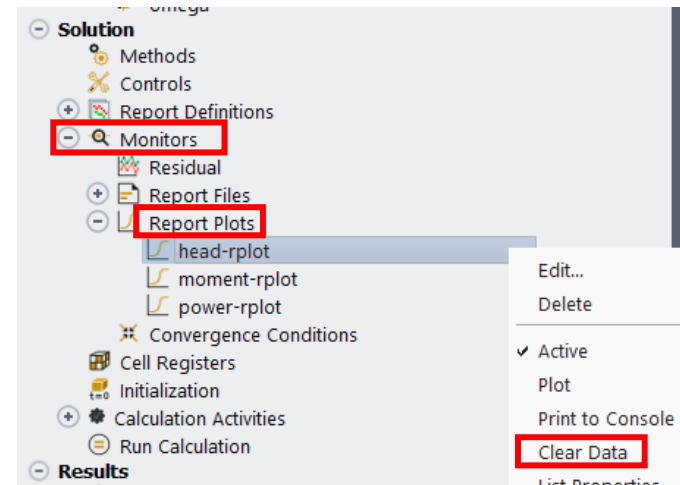
Solver Convergence

- Residuals and report plots show a bouncy behavior
  - Case is not converging to a steady state
    - Continuity residuals oscillate around 1e-2 and cannot be reduced further
    - The targeted quantities of *head* and *power* seem to be constant within 2-3 significant figures
      - *head* ≈ 21.3 (m)
      - *power* ≈ -20200 (watt)
    - To check the accuracy of these key quantities, it is a good idea to investigate the case by solving it also as transient
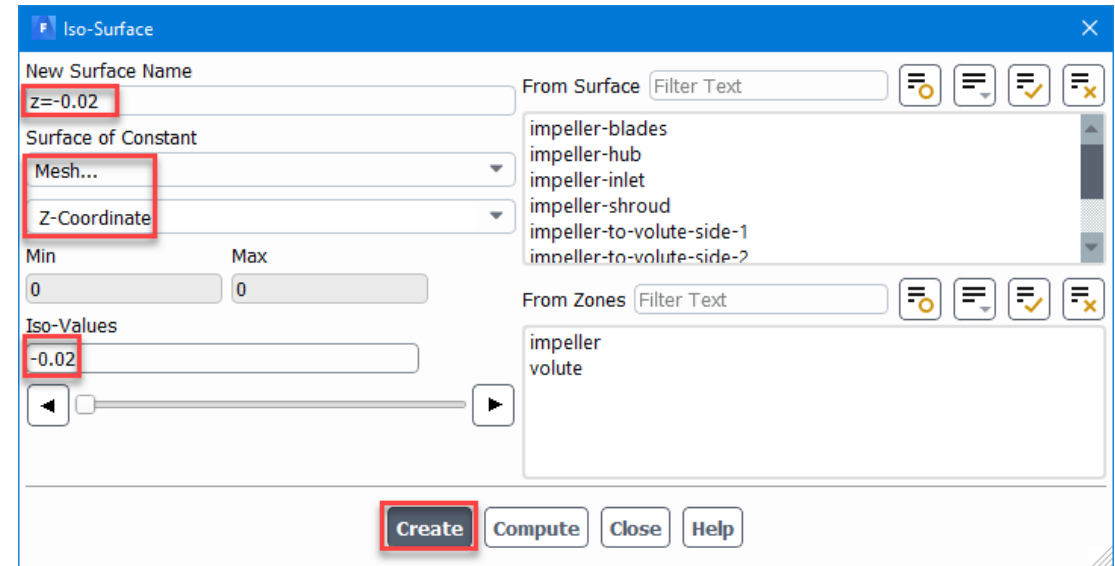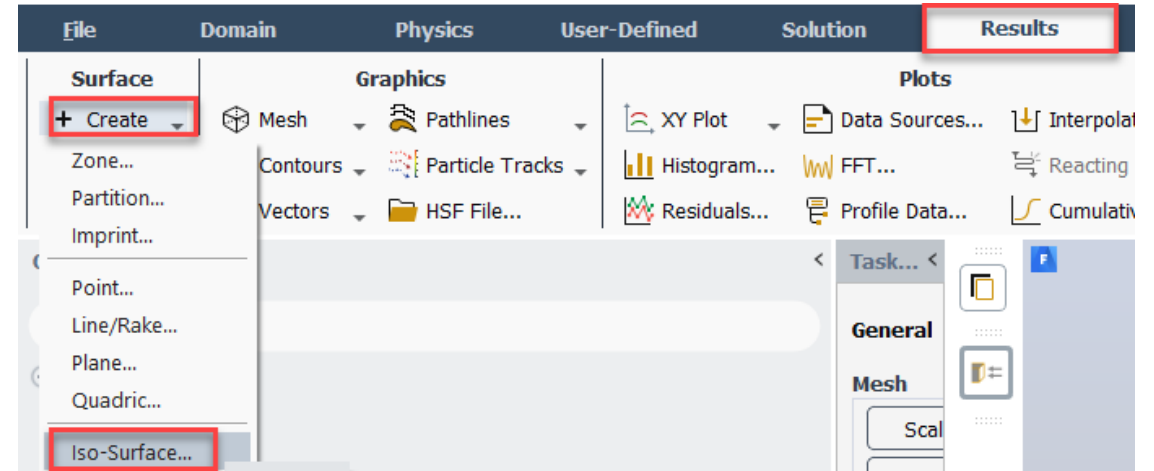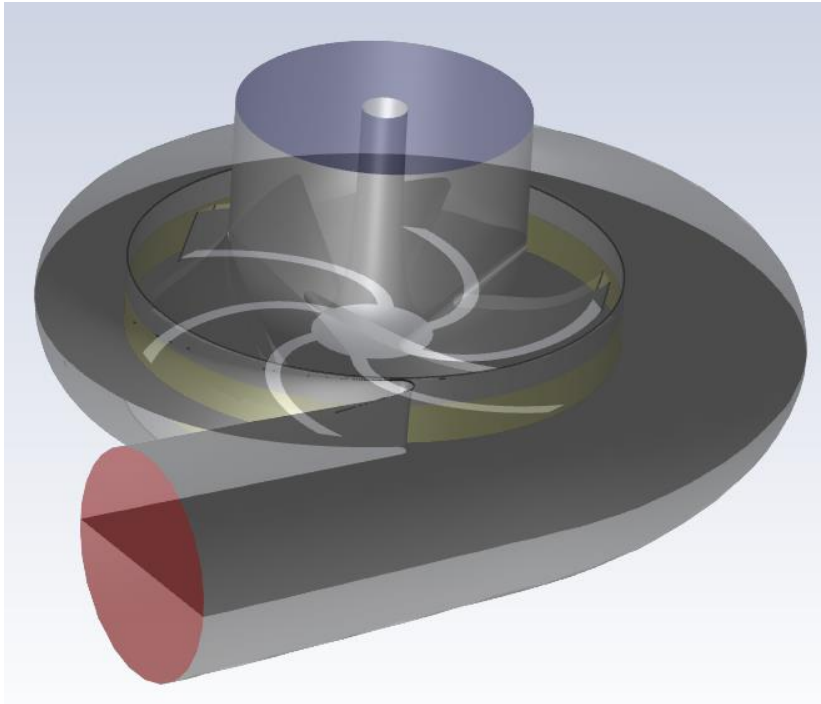
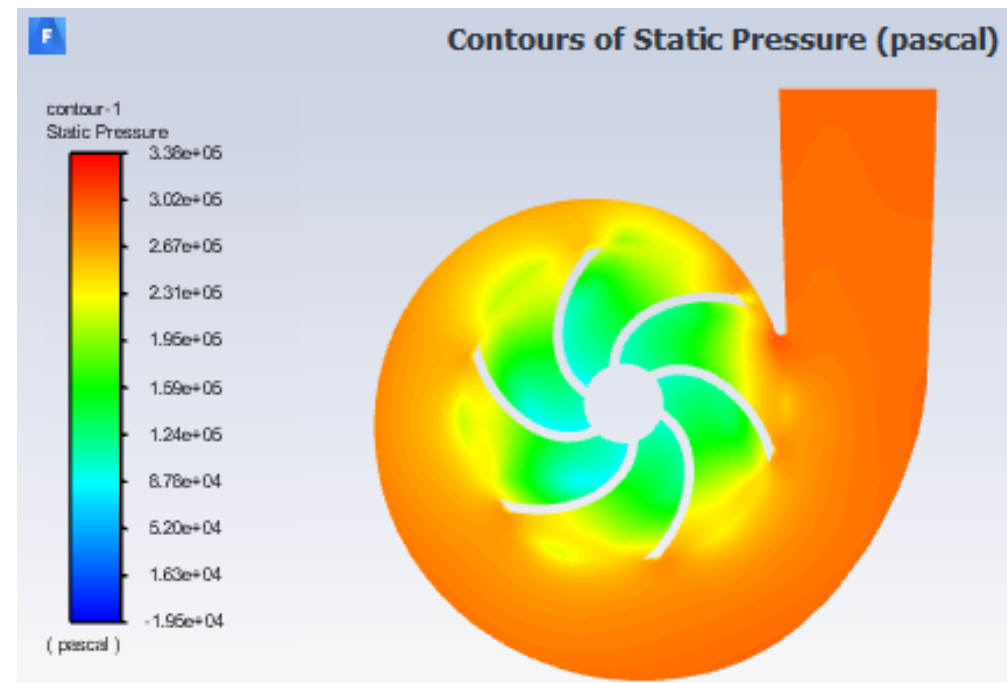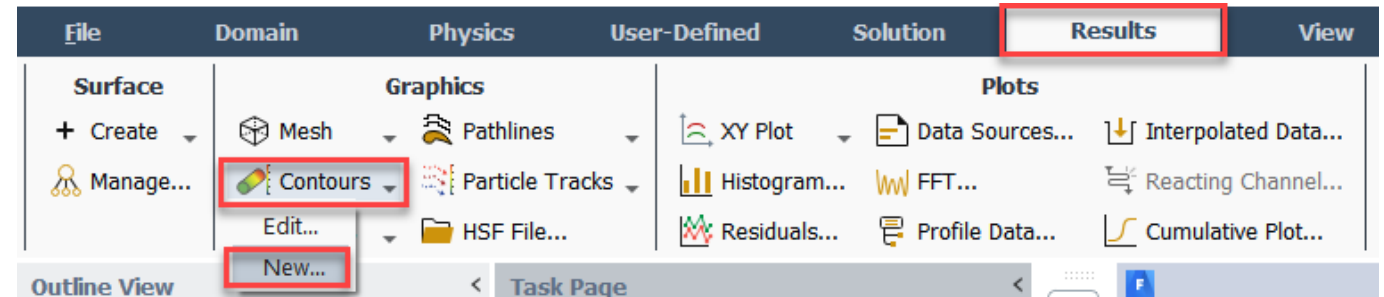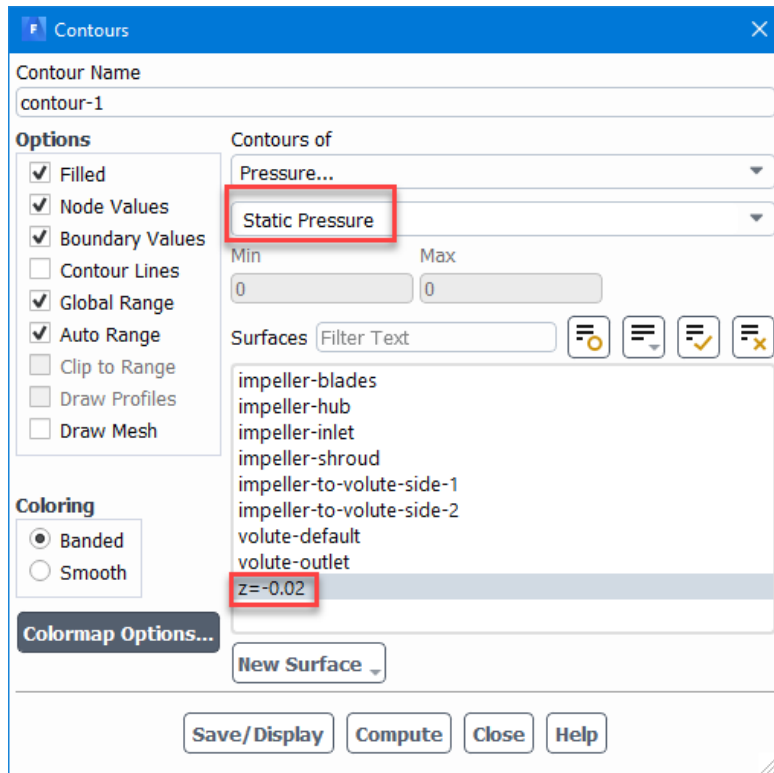Note: Residuals and monitor plots may differ between two computers or Ansys releases
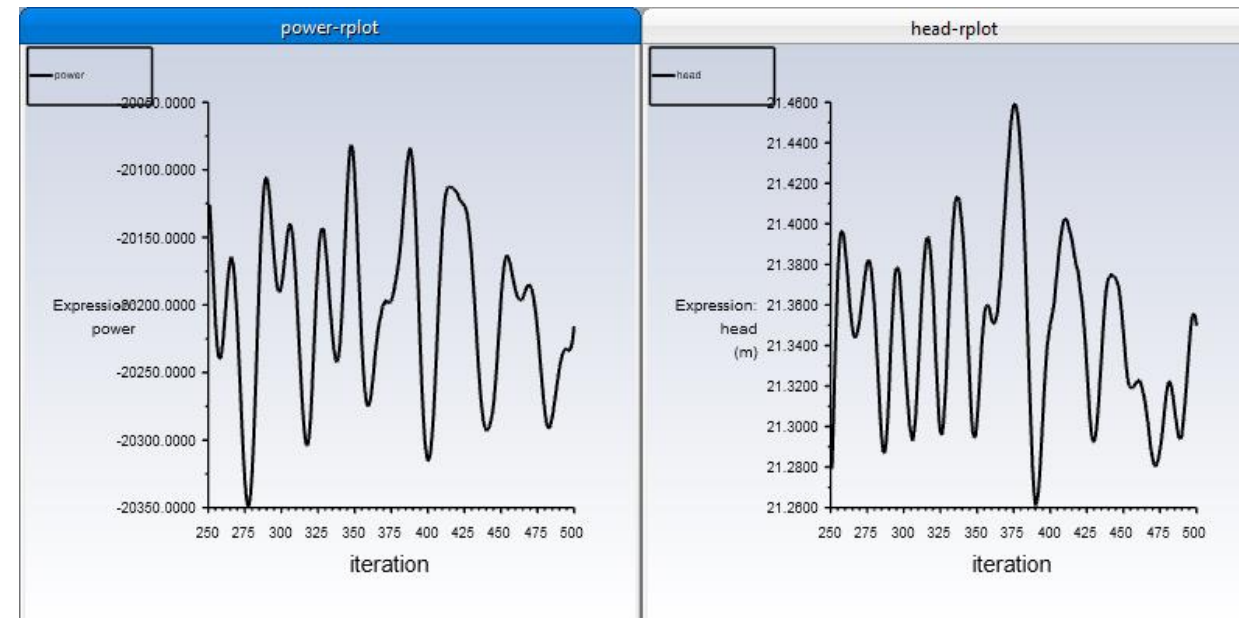
Create a Surface of Constant Axial Coordinate

- Create an *Iso-Surface* of *z-Coordinate=-0.02 m*
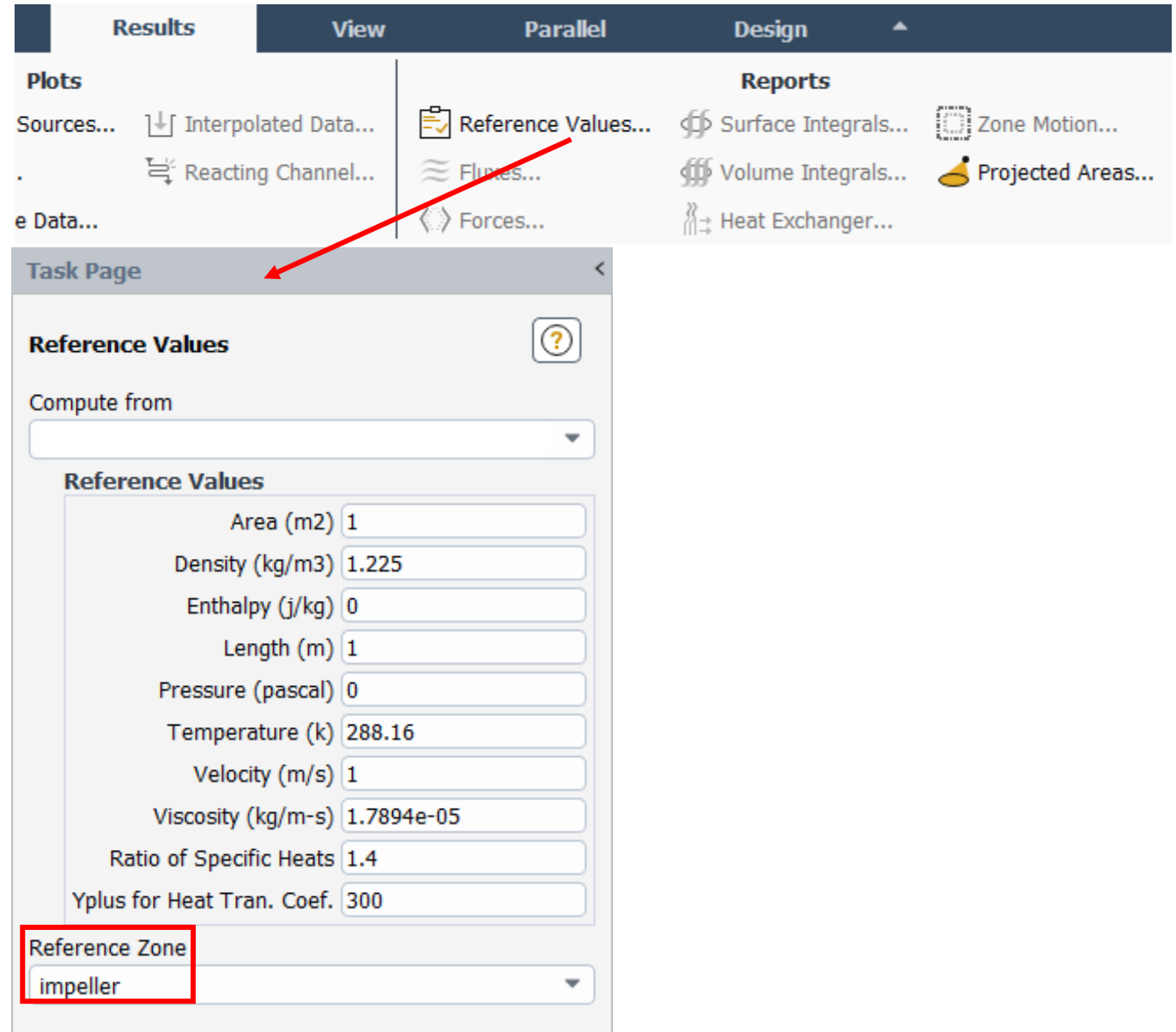
Pressure Contours on Surface z=-0.02

- Create a *New Contour* plot of *Static Pressure* on surface z=-0.02

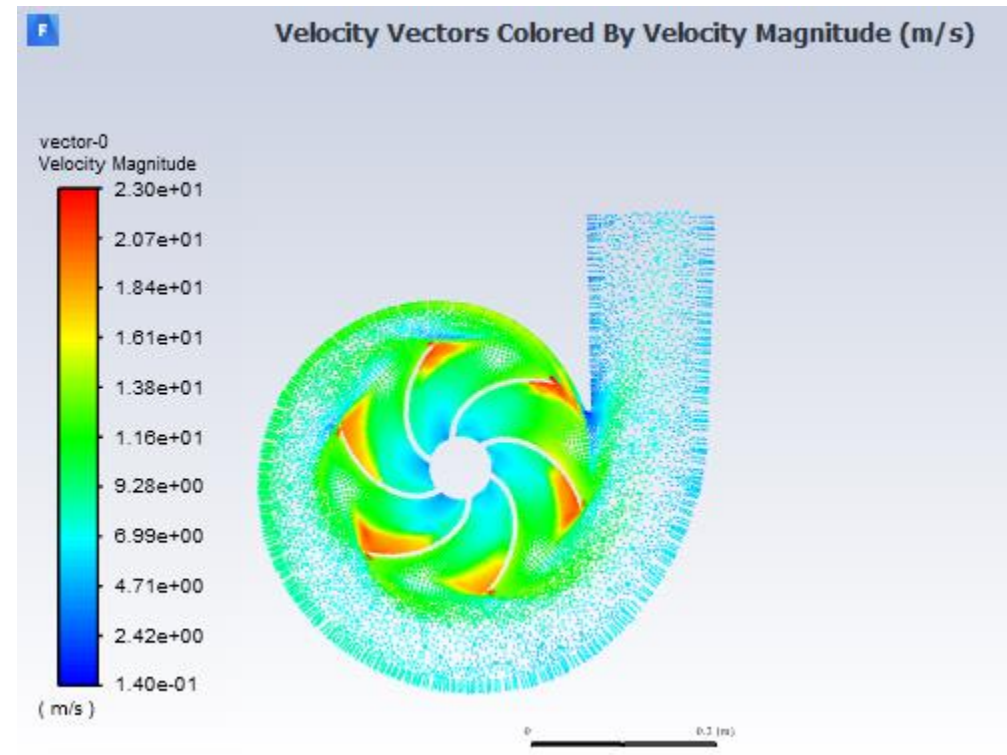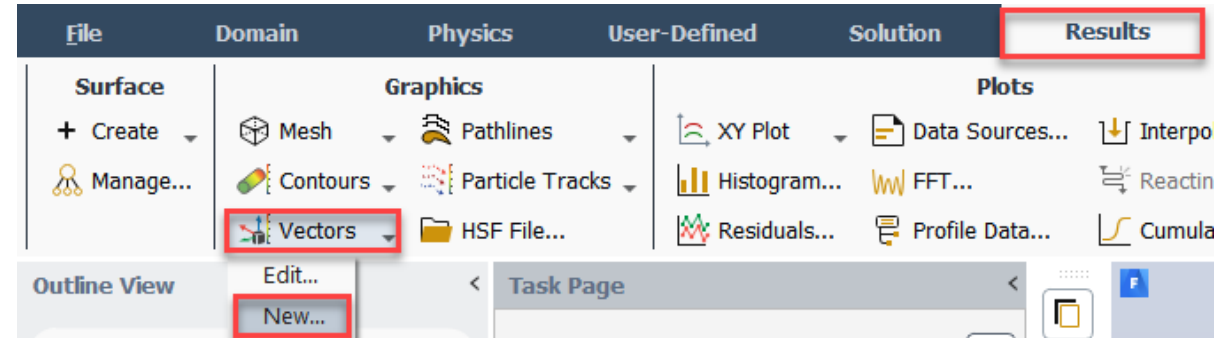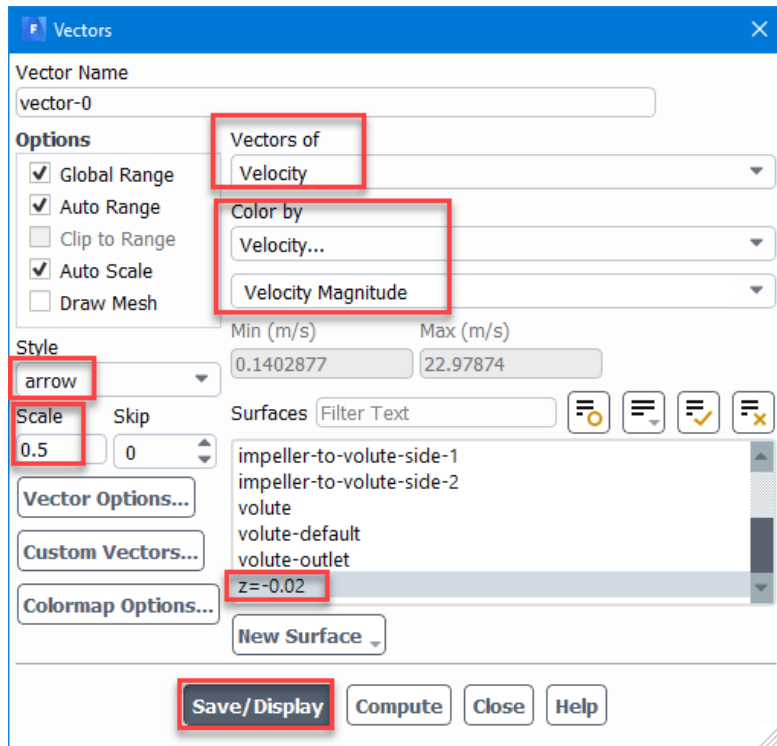Note: Results may differ between two computers or Ansys releases

Define Reference Zone for Relative Velocities

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

  - 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

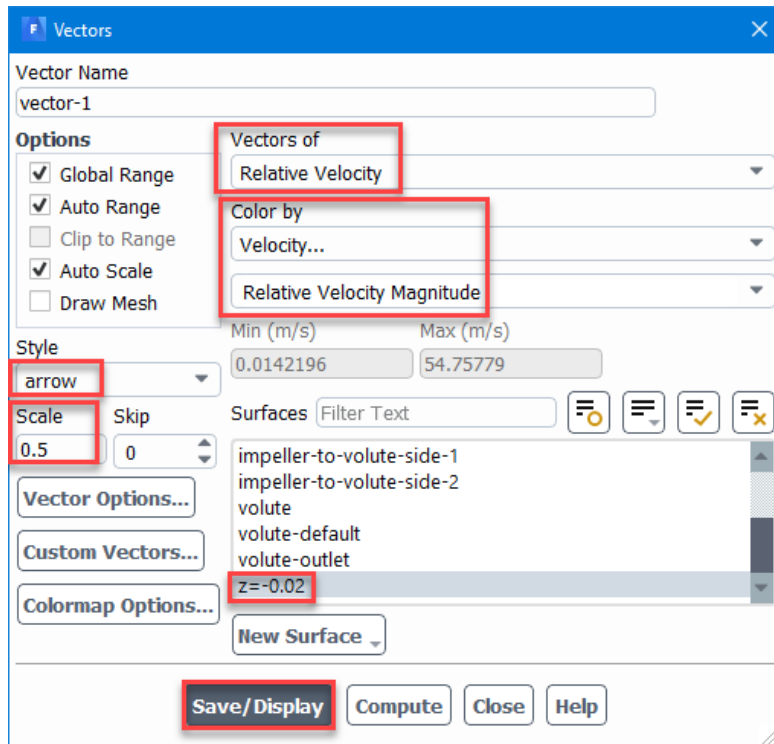# Velocity Vectors on Surface z=-0.02

- Create a *New Vector* plot of *Velocity* on Surface z=-0.02
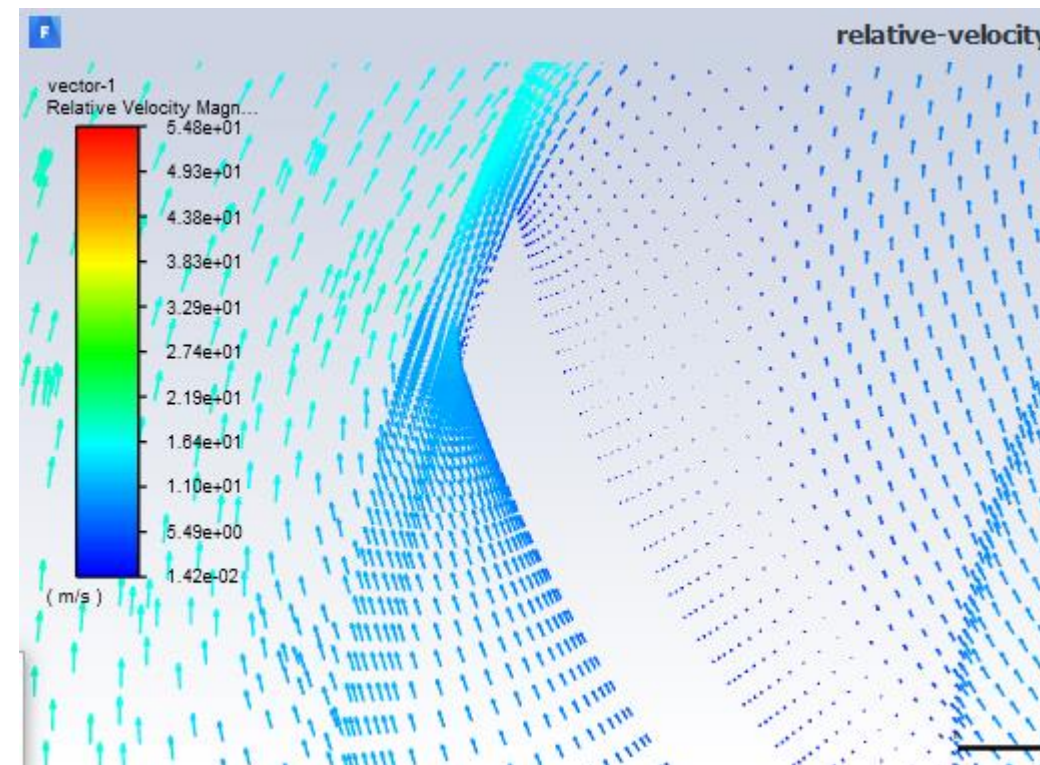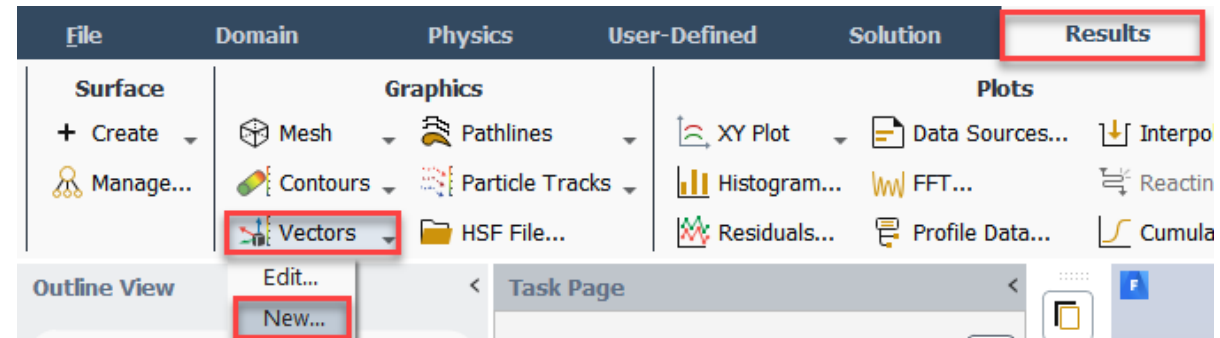

Note: Results may differ between two computers or Ansys releases

Relative Velocity Vectors on Iso-surface z=-0.02

- Create a *New Vector* plot of *Relative Velocity* on Surface z=-0.02

- Write the Fluent .cas and .dat files and exit Fluent

Note that you will have to set the *Reference Zone* as described on slide 30 and might also have to perform one more solver iteration, so that the relative velocity vectors look like the ones in the image here.

Summary

- This workshop has covered:
  - Setting up a steady stage calculation comprising a rotor and a stator
    - Defining a rotating frame
    - Creating a Frozen Rotor, General Turbo Interface
    - Creating named expressions and report plots for monitoring the head and the power consumption of the pump impeller
  - Solving and monitoring convergence
  - Visualizing the pressure distribution and velocity vectors on a plane of constant axial coordinate

Ansys

End of presentation