Lecture 02: Single Rotating Component Analysis

Release 2020 R2

Outline

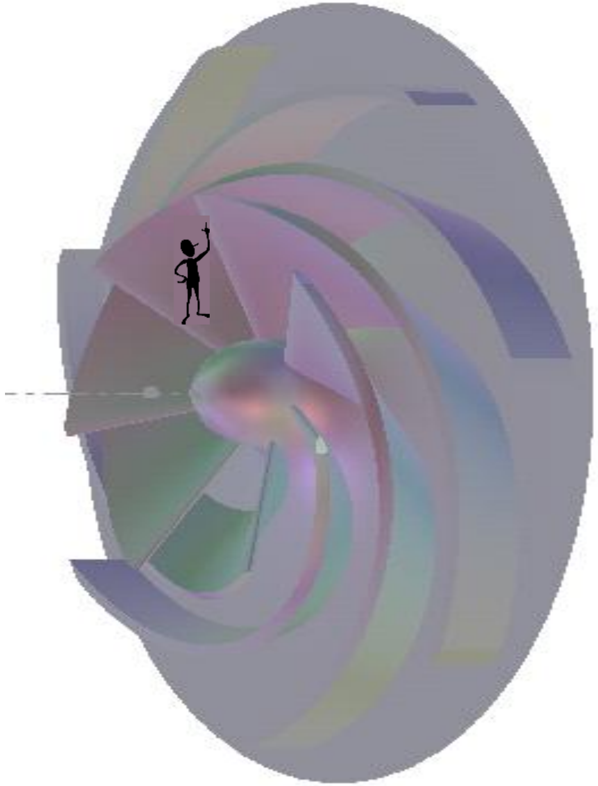
- Rotating Reference Frames
  - Motivation
  - Governing Equations
  - The Velocity Triangle
  - “False Swirl”
- Single Rotating Component Analysis
  - Geometry Considerations
  - Workflow
    - Solver Settings and Physical Models
    - Boundary Conditions
    - Initialization and Solution
- Summary

Outline

- Rotating Reference Frames
  - Motivation
  - Governing Equations
  - The Velocity Triangle
  - “False Swirl”
- Single Rotating Component Analysis
  - Geometry Considerations
  - Workflow
    - Solver Settings and Physical Models
    - Boundary Conditions
    - Initialization and Solution
- Summary

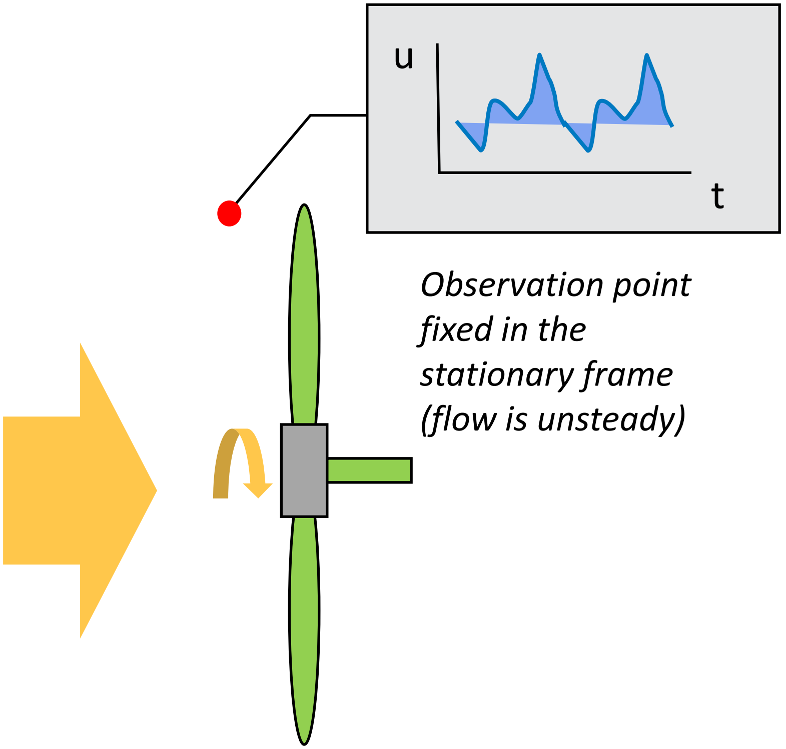
Motivation

- Rotating machinery components can be best modeled using a **Rotating Reference Frame (RRF)**
- Entire domain rotating at specified rate about an axis
  - For clarity, imagine yourself standing on the hub, moving with the blade row, looking out at the passage flow
- Why use a **Rotating Reference Frame**?
  - A flow field which is unsteady with respect to the stationary frame may be studied as steady with respect to the RFR (see next slide)
    - Easier to solve...
      - simpler BCs
      - lower computational cost
      - easier to post-process and analyze

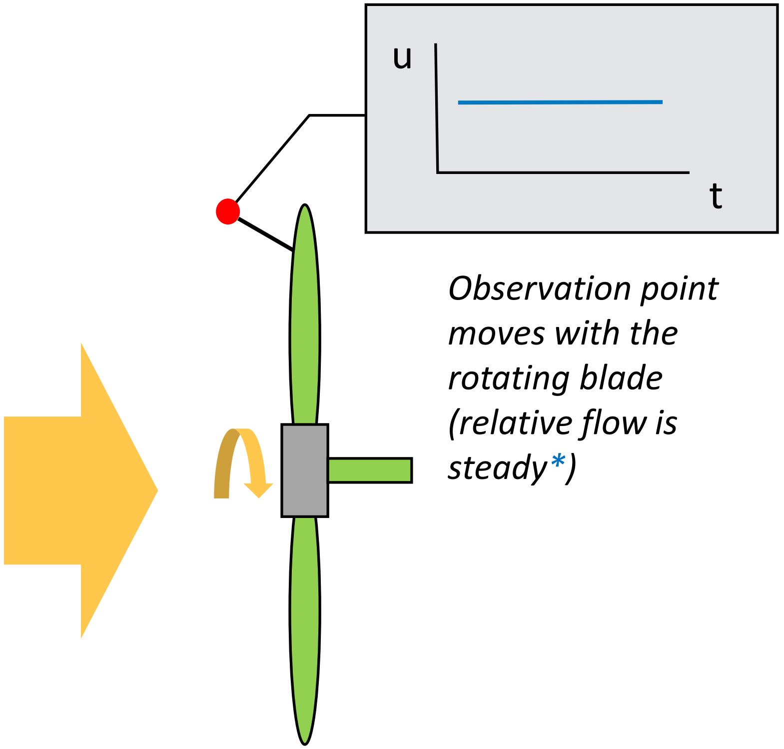


Flow Viewed in Stationary versus Rotating Reference Frame

Uniform freestream flow



Fixed point in absolute frame

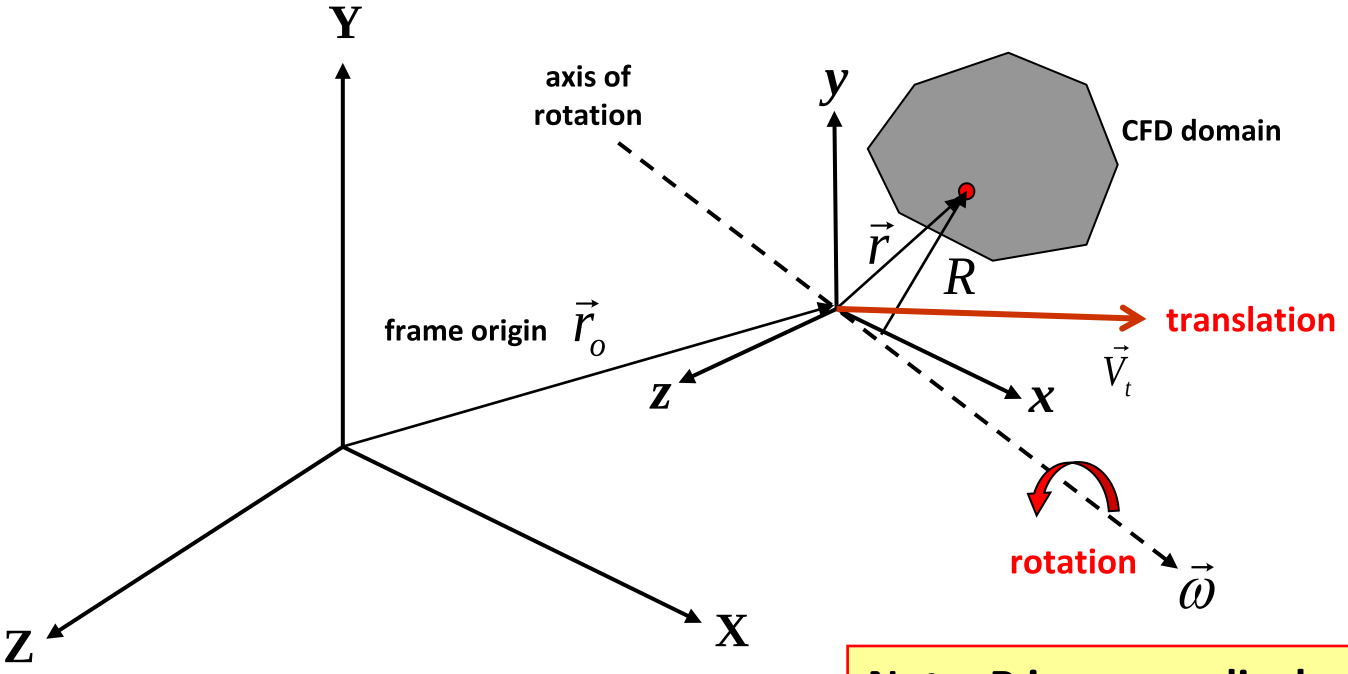


Fixed point in relative frame

* Flow can be unsteady in the RFR due to turbulence, local separation, etc. But mean flow remains steady

Moving Reference Frames in Fluent

- The CFD domain is moving with respect to the Global Coordinate system but is stationary in the moving Local Coordinate system
- The moving frame is undergoing a rotation and a translation as seen from the stationary frame
  - Only rotational motion is relevant to this course $V_t = 0$ → (see next slide)



Note: R is perpendicular to axis of rotation

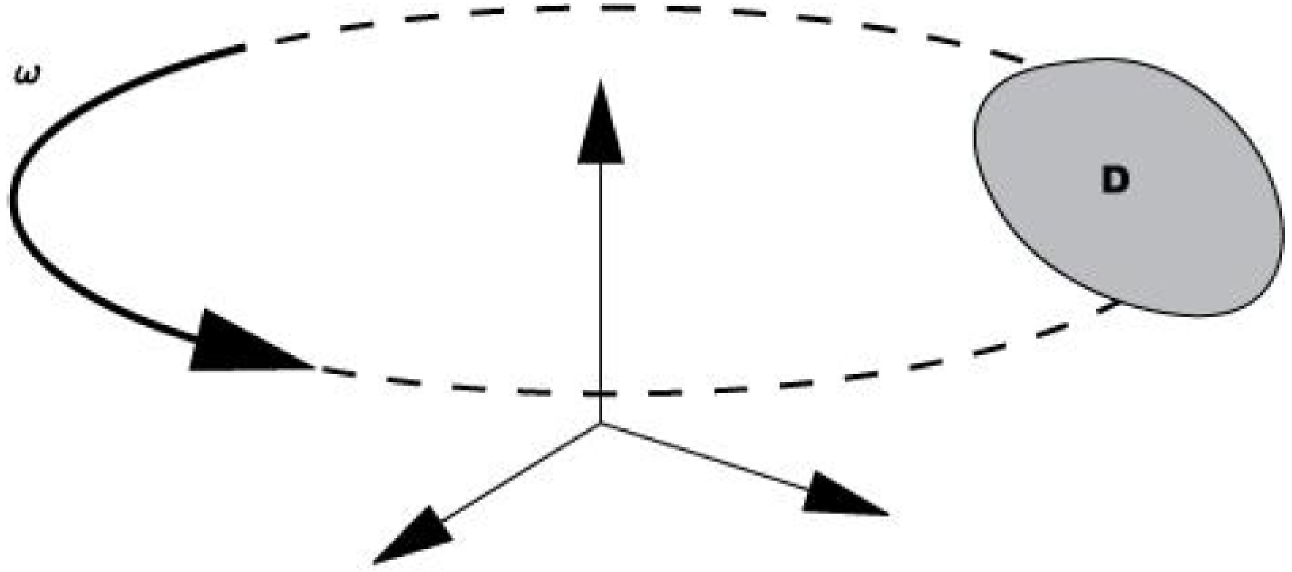
The velocity of the fluid can be defined with respect to both the stationary or moving frames:

Absolute velocity - Fluid velocity defined with respect to the stationary (absolute) frame

Relative velocity - Fluid velocity defined with respect to moving system

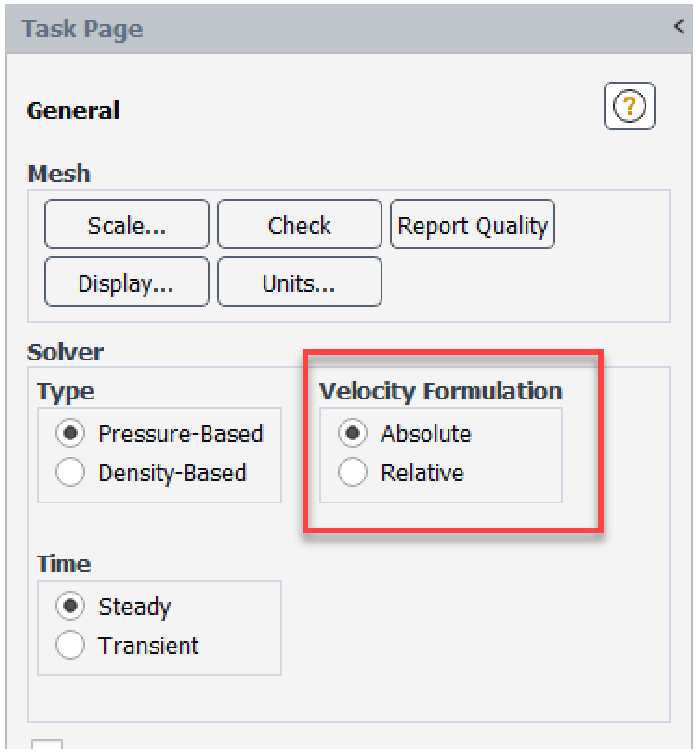
Rotating Reference Frame in Fluent

- Rotating Reference Frame in Fluent
  - A Moving Reference Frame with a translational velocity $V_t = 0$
- The CFD domain is rotating at an angular velocity ω with respect to the Global Coordinate system but is stationary in the Rotating Reference Frame
  - ω may be constant or a function of time



Governing Equations for Rotating Reference Frame

- Equations are similar as for stationary frame
- Relative or Absolute Velocity Formulations available
  - Relative Velocity Formulation
    - Obtained by transforming the stationary frame N-S equations to a rotating reference frame
    - Uses the relative velocity as the dependent variable in the momentum equations
    - Uses the relative total internal energy as the dependent variable in the energy equation
    - Recommended when most of the fluid in the domain is moving, as in the case of a large impeller in a mixing tank
  - Absolute Velocity Formulation
    - Derived from the relative velocity formulation
    - Uses the absolute velocity as the dependent variable in the momentum equations
    - Uses the absolute total internal energy as the dependent variable in the energy equation
    - Preferred in applications where the flow in most of the domain is not moving (for example, a fan in a large room)



Frame Acceleration in a Rotating Frame

- The frame rotational velocity is constant for most practical cases
- May also be a function of time $\vec{U} \equiv \vec{\omega}(t) \times \vec{r}$

- An additional rotational acceleration term $\vec{\alpha}$ is included in the momentum equation for a rotating reference frame :

$$\vec{\alpha} \equiv \frac{d\vec{\omega}}{dt}$$

Relative Velocity Formulation Accelerations Due to Rotating Frame

- Additional source terms in momentum equations for the effects of the Coriolis and centrifugal forces in the rotating “Non-inertial” reference frame

$$\rho[\underbrace{2\vec{\omega} \times \vec{W}}_{\text{Coriolis acceleration}} + \underbrace{\vec{\omega} \times (\vec{\omega} \times \vec{r})}_{\text{centrifugal acceleration}}] + \rho(\underbrace{\vec{\alpha} \times \vec{r}}_{\text{rotational acceleration}})$$

Zero for constant frame rotational velocity

Absolute Velocity Formulation Accelerations Due to Moving Frame

- Accelerations reduce to two terms
- Note that the frame acceleration term $\vec{\alpha}$ does not appear in the Absolute Velocity Formulation!

$$\rho[\vec{\omega} \times \vec{W} + \vec{\omega} \times (\vec{\omega} \times \vec{r})] = \rho(\vec{\omega} \times \vec{V})$$

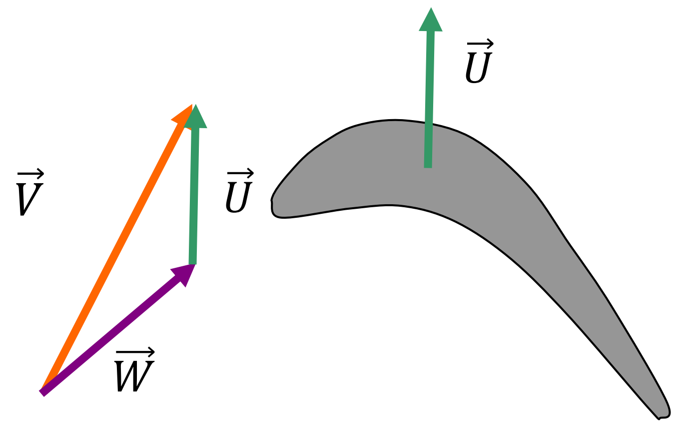
$\vec{V}$ = absolute velocity
$\vec{W}$ = relative velocity
$\vec{r}$ = location vector
$\vec{\omega}$ = rotation vector
See next slide

The Velocity Triangle

- The relationship between the absolute and relative velocities:

$$\vec{V} = \vec{W} + \vec{U}$$
$$\vec{U} \equiv \vec{\omega} \times \vec{r}$$

$\vec{V}$ = absolute velocity
$\vec{W}$ = relative velocity
$\vec{r}$ = location vector
$\vec{\omega}$ = rotation vector



- In turbomachinery, this relationship can be illustrated using the laws of vector addition. This is known as the Velocity Triangle

Quantities in Absolute and Rotating Reference Frames

- Some quantities are invariant to the reference frame
  - Static Pressure, Temperature

- Some are not
  - Velocity Magnitude, Total Quantities like Total Pressure, Total Temperature

False Swirl

- “False Swirl” is a term used to describe a particular type of numerical or discretization error
- In certain cases, this error introduces non-physical angular velocity to the stationary-frame flow in the vaneless space upstream and downstream of the blade row
- Causes incorrect leading-edge incidence (in the rotating frame), thus leading to inaccurate predictions of head and losses

False Swirl (2)

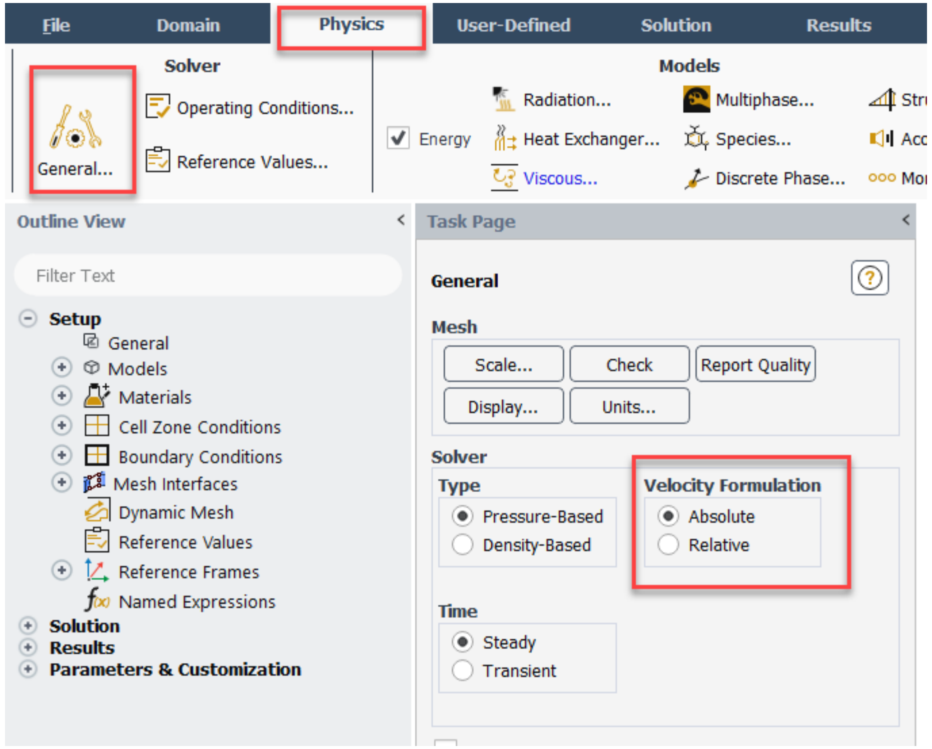
Errors are greatest when:

- Rotating-frame tangential flow is large compared with meridional flow
  - inducers and pumps with near tangential leading-edge angles, fans, propellers, blowers
- Mesh is coarse, particularly in the circumferential direction
- Blade number is low, hence domain has large circumferential extent

False Swirl (3)

Solution:

- Use Absolute Velocity Formulation
  - modified discretization which minimizes discretization errors for such cases
- Do a Multiple Frame Reference calculation (see lecture 4)
  - solve for the vaneless portion in the stationary frame
- Refine the mesh (requires large meshes, for low blade count)
- Concentrate mesh near the tip where errors are largest
- Increase mesh in circumferential direction
  - rule of thumb is about one node per degree



Ansys

Outline

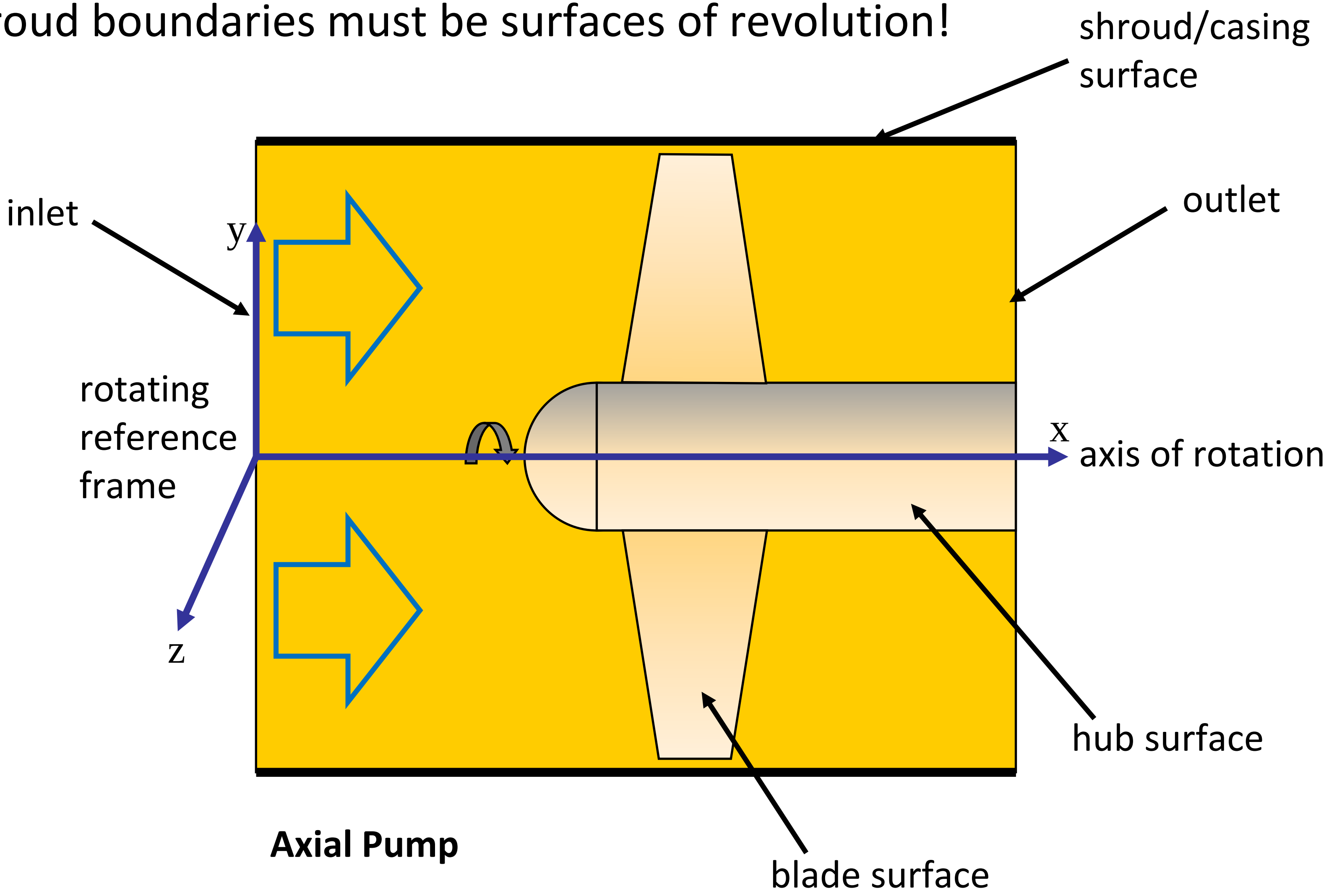
- Rotating Reference Frames
  - Motivation
  - Governing Equations
  - The Velocity Triangle
  - “False Swirl”
- Single Rotating Component Analysis
  - Geometry Considerations
  - Workflow
    - Solver Settings and Physical Models
    - Boundary Conditions
    - Initialization and Solution
- Summary

Single Rotating Component Analysis

- Can be applied to simple systems where the entire computational domain rotates
  - Impellers or rotors
  - Rotating containers
  - Disk cavities and rotating seals
- Special geometrical constraints apply (next slides)
- Can also be used for preliminary analysis of a single rotating component of a complex system
  - Example: Cavitation in a pump can be studied using a single rotating domain of the impeller

Single Rotating Component Analysis Geometry Requirements

Inlet, outlet, shroud boundaries must be surfaces of revolution!



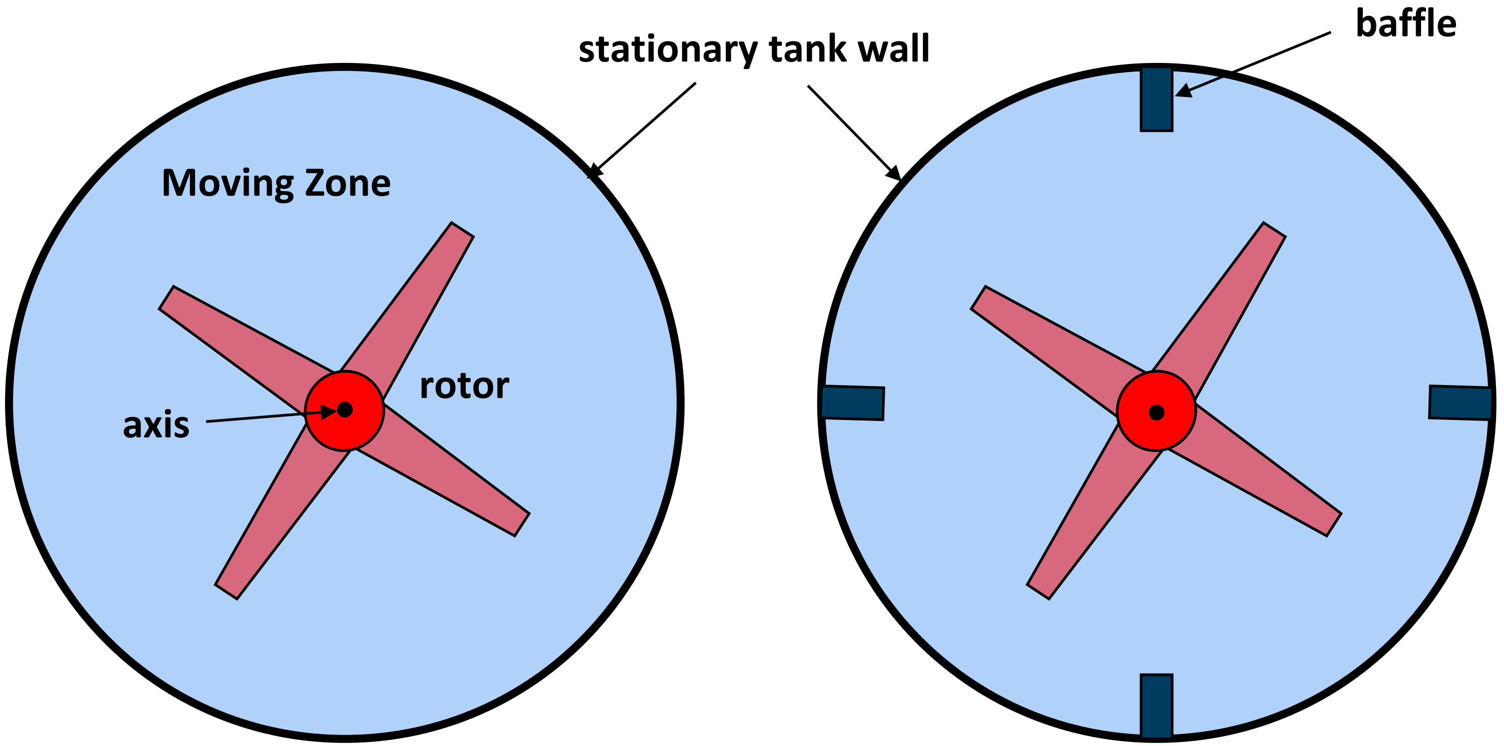
Axial Pump

Single Rotating Component Analysis Geometry Requirements (2)

- Boundaries which move with the fluid domain may assume any shape (example: impeller blades)
- Boundaries which are stationary (with respect to the absolute frame) must be surfaces of revolution about the axis of rotation
  - Walls (shroud or other)
  - Inlets
  - Outlets

Stationary Walls in Single Rotating Component Analysis

Mixing Tank Example



Single Rotating Reference Frame
can be used

Single Rotating Reference Frame
can NOT be used!
Stationary wall with baffles is not a
surface of revolution about the axis!

CFD Workflow for Single Rotating Component Analysis

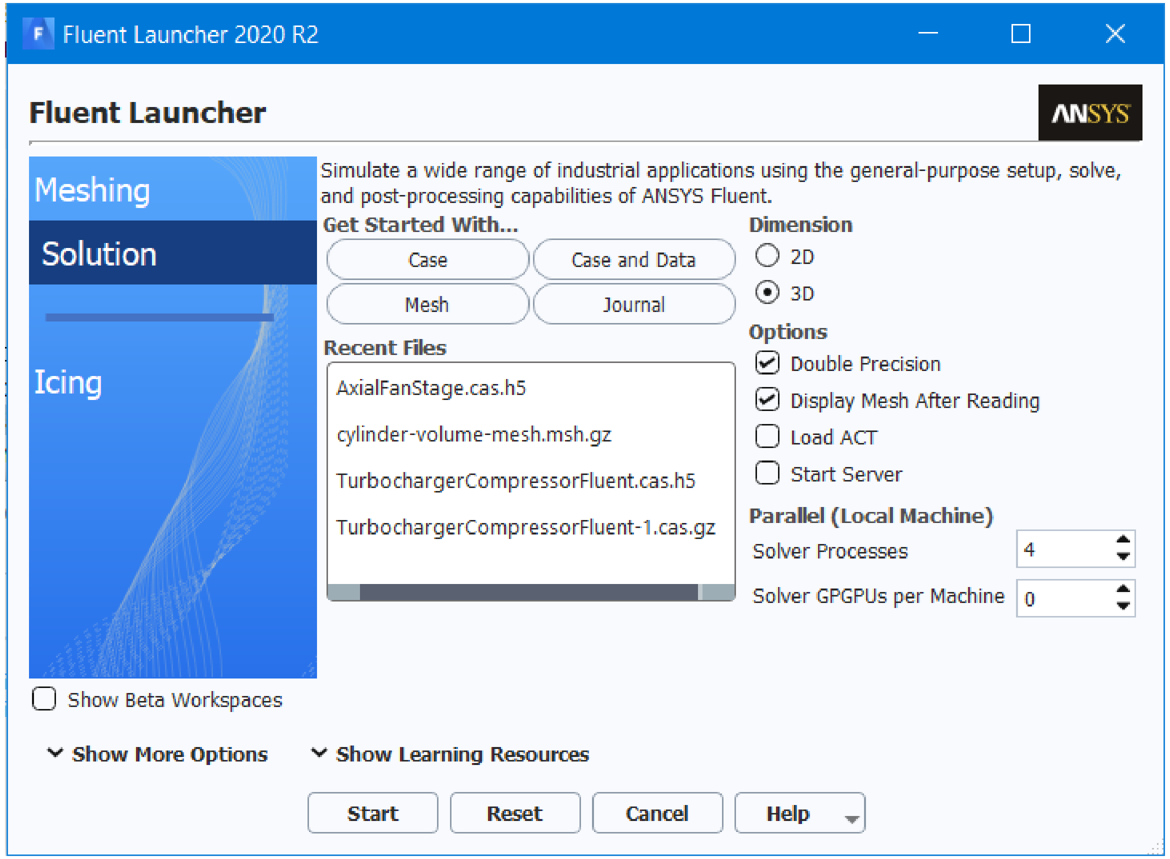
1. Define single rotating domain geometry
2. Generate a suitable mesh for the geometry
   - Hexahedral mesh is advantageous as the flow direction is aligned with the mesh
3. Set up the CFD model
   - Fluid Domain
     - Fluid material properties
     - Domain Motion
     - Physical models (turbulence, heat transfer,…)
   - Boundary conditions (periodic interfaces, inlets, outlets, walls,…)
   - Solver settings
   - Monitors
4. Run the calculation
5. Post-process the solution (covered in lecture 3)

CFD Workflow for Single Rotating Component Analysis

1. Define single rotating domain geometry
2. Generate a suitable mesh for the geometry
   - Hexahedral mesh is advantageous as the flow direction is aligned with the mesh
3. Set up the CFD model
   - Fluid Domain
     - Fluid material properties
     - Domain Motion
     - Physical models (turbulence, heat transfer,…)
   - Boundary conditions (periodic interfaces, inlets, outlets, walls,…)
   - Solver settings
   - Monitors
4. Run the calculation
5. Post-process the solution (covered in lecture 3)

Starting Fluent

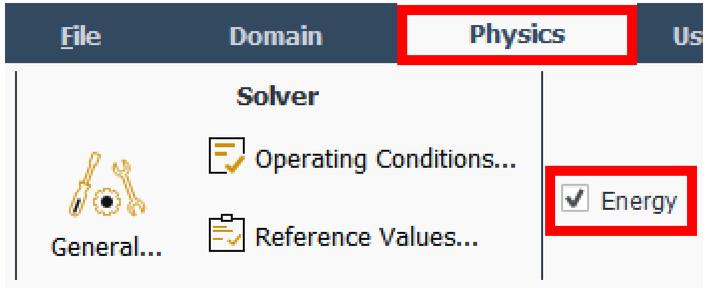
- Use double precision for the run
  - When aspect ratios in bladed components > 1000

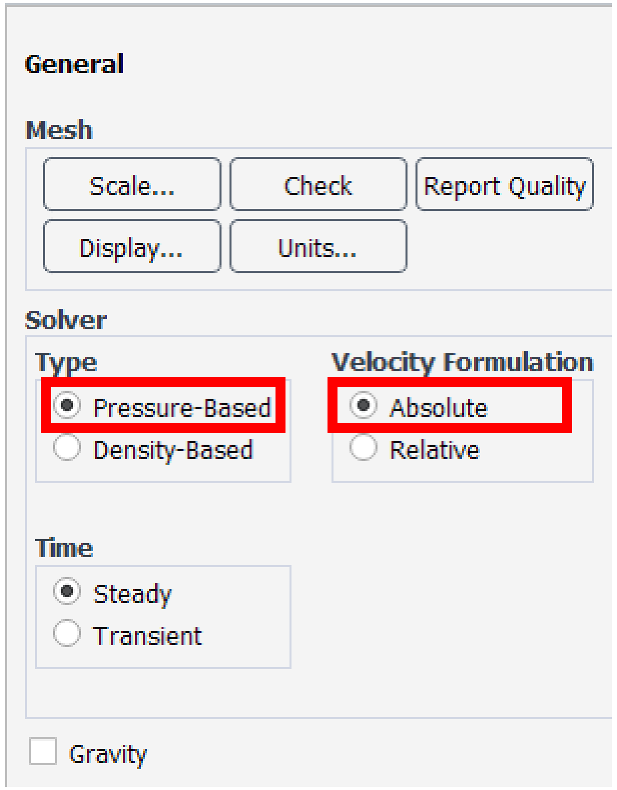


Physics: General Settings and Energy Equation

- Use the “Pressure-Based” solver
- Set “Velocity Formulation” to “Absolute”
- Turn on the solution of the “Energy” equation for compressible flows

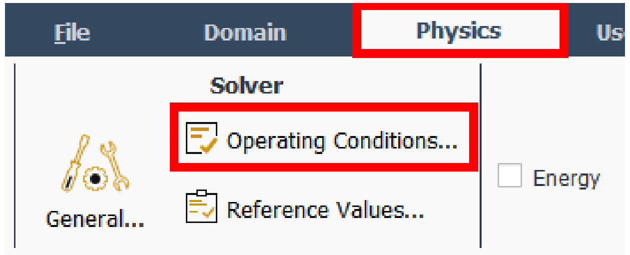
For compressible flows:

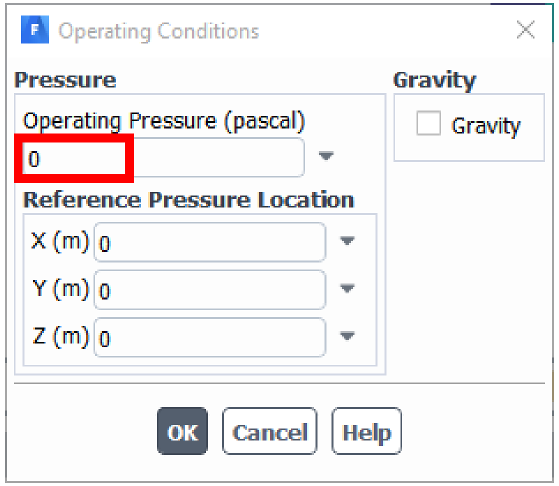




Physics: Operating Conditions

- Set a proper operating pressure
- Depends on the pressure rise or drop through the machine (ΔP) and the nominal operating pressure of the machine (P_{nom})
  - Set to 0 [Pa] if ΔP is of the order of, or larger than P_{nom}
    - True for most turbomachinery applications, e.g. steam and gas turbines, compressors, turbochargers,…
  - Set to P_{nom} (e.g. 1 [Atm]) if ΔP is at least one order smaller than P_{nom}
    - E.g., blowers and fans



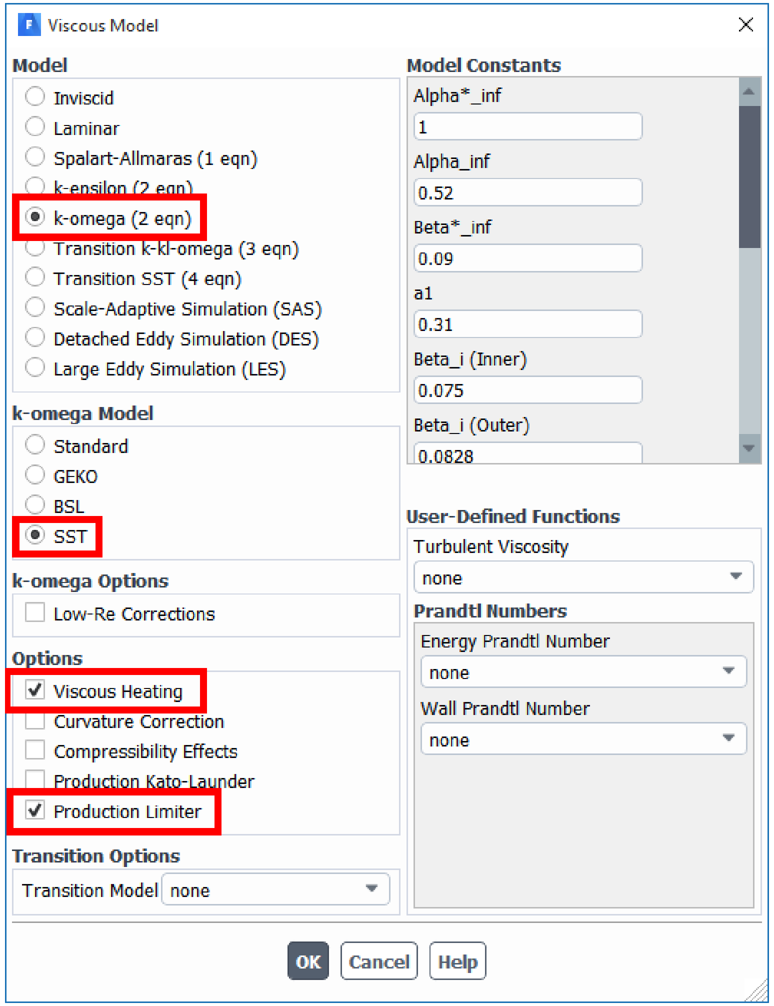


NOTE for gravity: Gravity is ignored for most rotating reference frame problems. Note that, unless aligned with the axis or rotation, the gravity vector will appear to be changing direction in a rotating frame and will NOT be a constant in a specific direction, as is usually assumed!

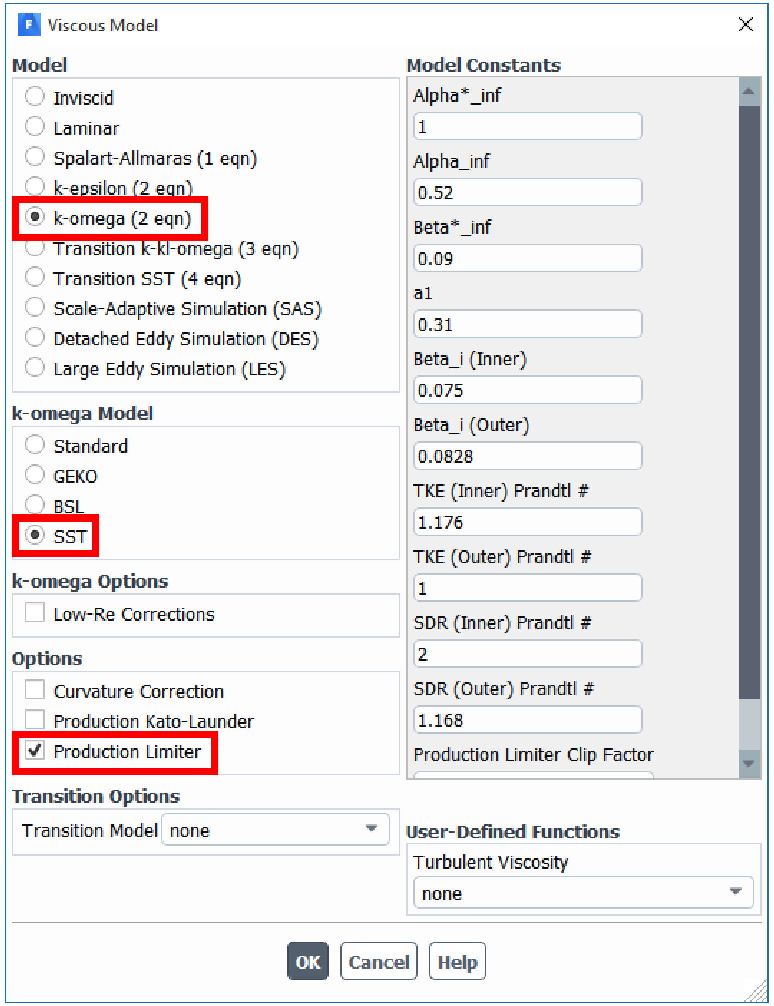
Physics: General Settings and Energy Equation

- Use "k-omega SST" turbulence model (default since 2020 R1)
- Activate the "Viscous Heating" term, if you solve the energy equation (compressible)

Energy equation: on

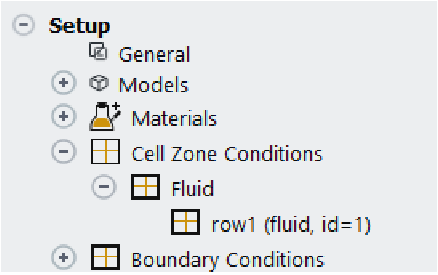


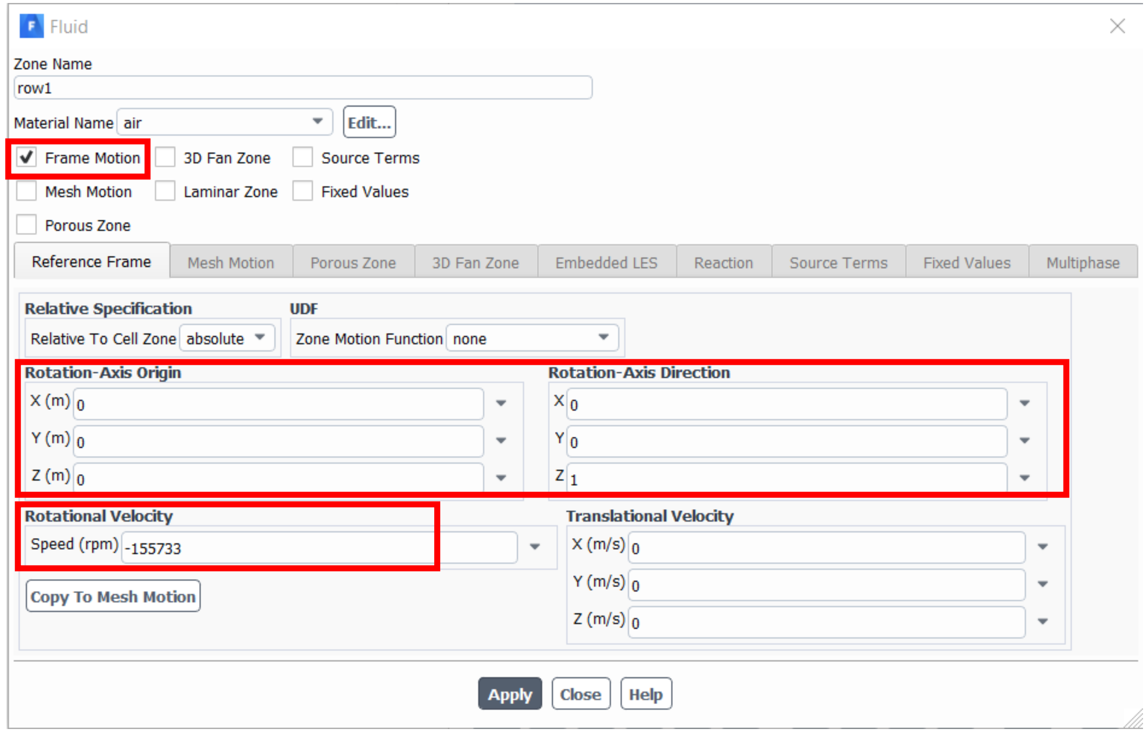
Energy equation: off



Physics: Cell Zone Conditions

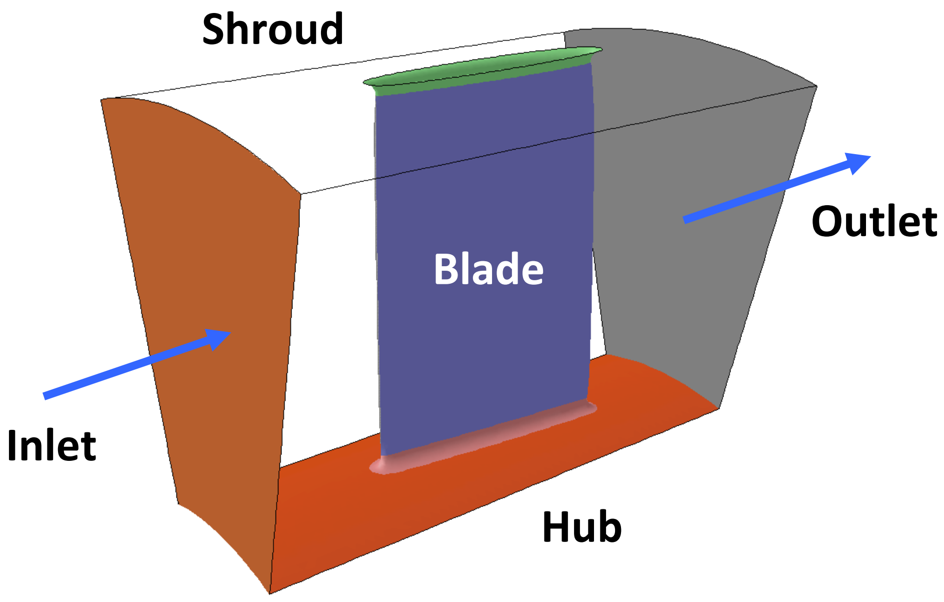
- Enable Frame Motion
- Define Rotational Speed
- Define Rotation Axis
  - It is important to set up the Rotational Axis properly, before setting up any rotational periodic interface





Boundary Conditions

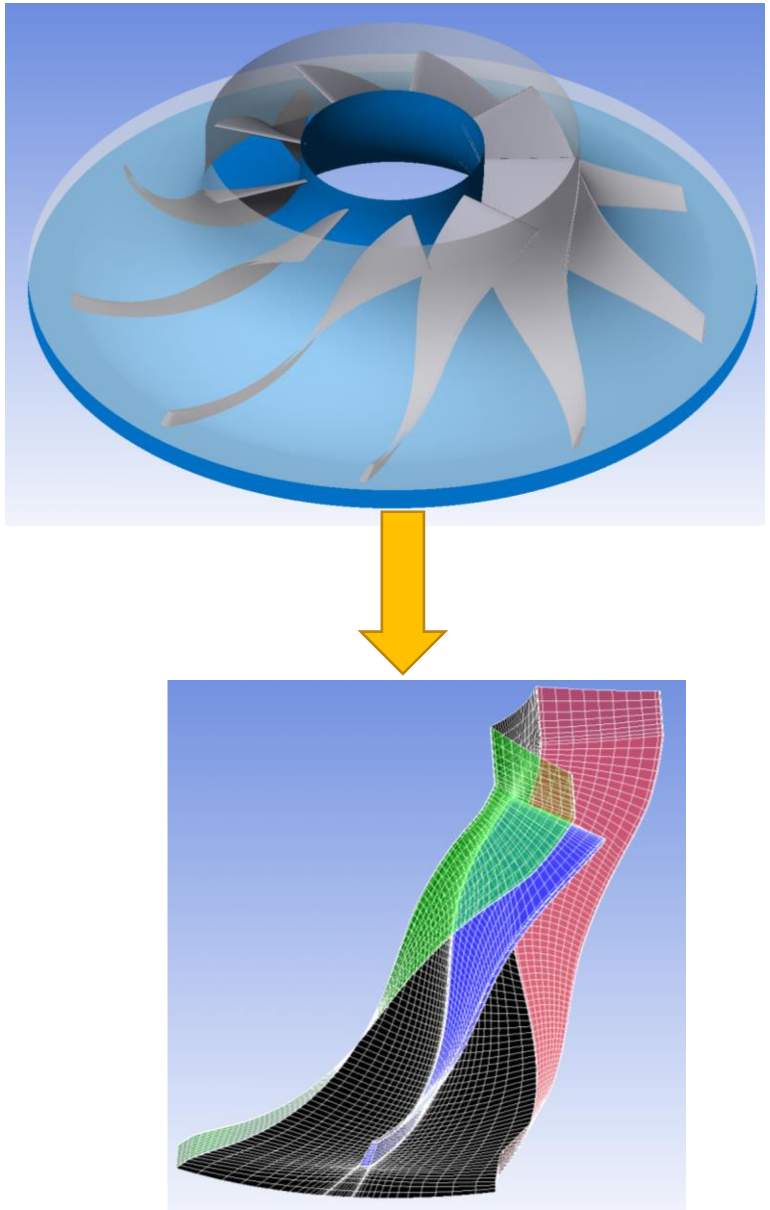
- Periodic interfaces
- Inlets
- Outlets
- Walls
- …



Axial Pump IGV

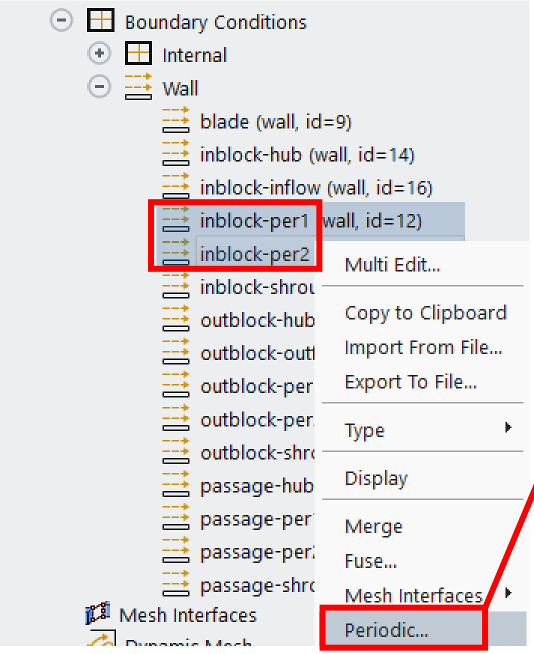
Rotational Periodicity

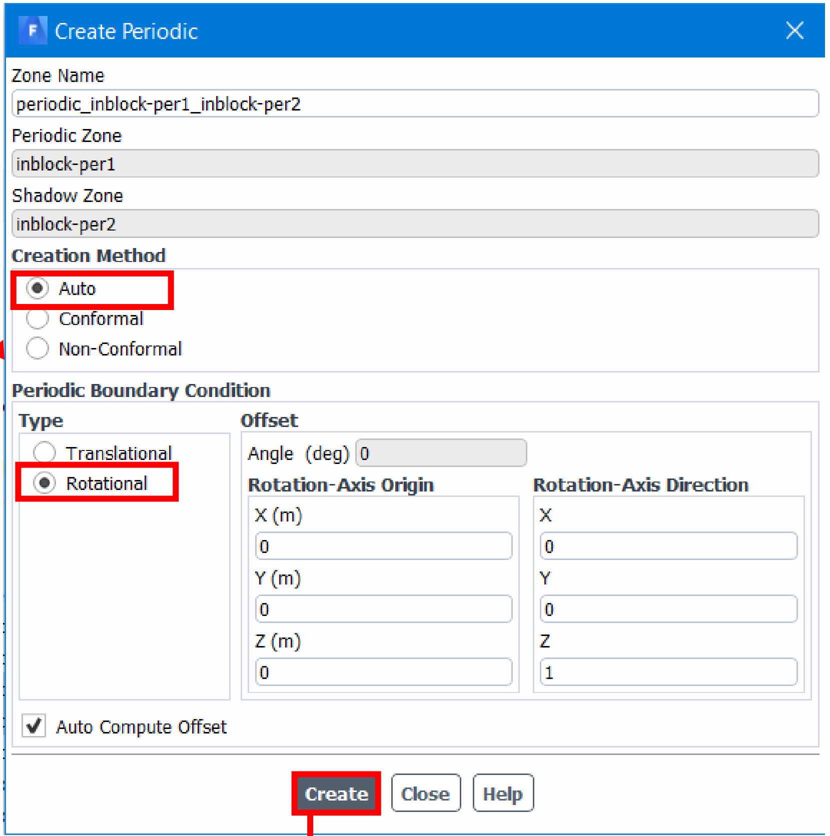
- Take advantage of rotational periodicity to reduce domain size
  - Solve for one blade passage only instead of the full wheel
- Must ensure both geometry and flow are periodic



Creating Periodic Interfaces

- Select the two sides of the rotational periodic boundaries under Boundary Conditions
  - Depending on their name they might initially reside under Wall boundaries
- RMB > Periodic
- Select Rotational
  - Rotation-Axis is automatically set to the cell-zone rotation axis previously set
- Check in console for message about conformal or non conformal periodic created
- A mesh check will summarize all periodic zones created

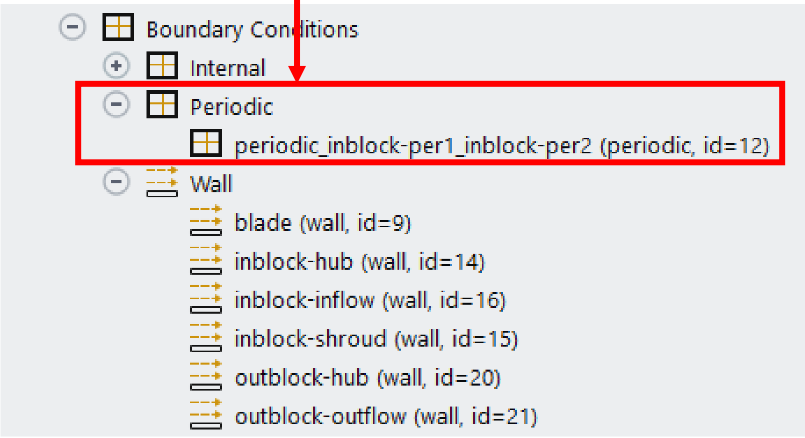




Console

```
     writing outblock-per2 (type wall) (mixture) ... Done.
     writing zones map name-id ... Done.
merged zones into inlet.

 zone 17 deleted
Created a conformal periodic boundary.
```



Console

```
   maximum face area (m2): 1.712135e-04
 Checking mesh.............................
   Periodic zone  12: average rotation angle (deg) = -60.000 (-60.000 to -60.000)
                      stored zone rotation angle (deg)   = -60.000
                      stored axis  , (0.000000e+00, 0.000000e+00, 1.000000e+00)
                      stored origin, (0.000000e+00, 0.000000e+00, 0.000000e+00)
Done.
```

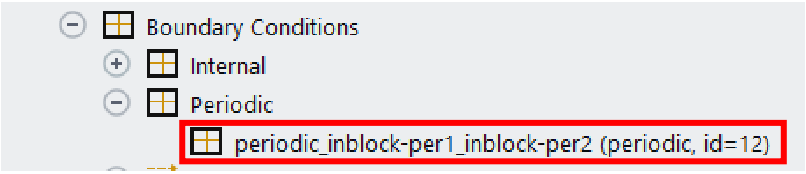

Ansys

Conformal vs. Non-Conformal Interfaces

- Conformal (1:1) interface
  - Console output:

```
  zone 17 deleted
Created a conformal periodic boundary.
```

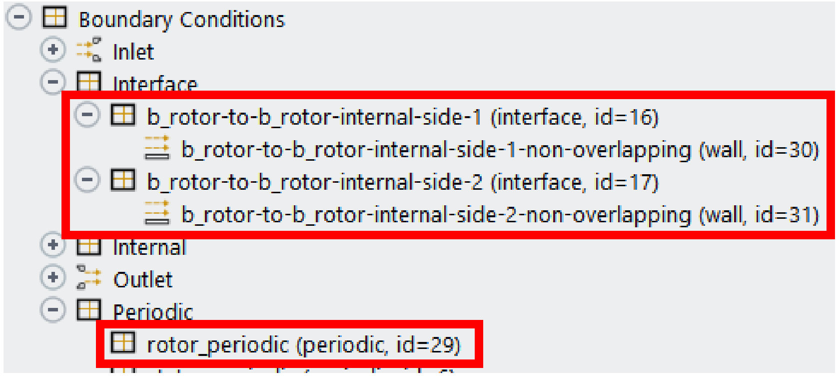
  - Visualization in Outline tree → Just one boundary patch is shown under Periodic



- Non-Conformal interface
  - Console output:

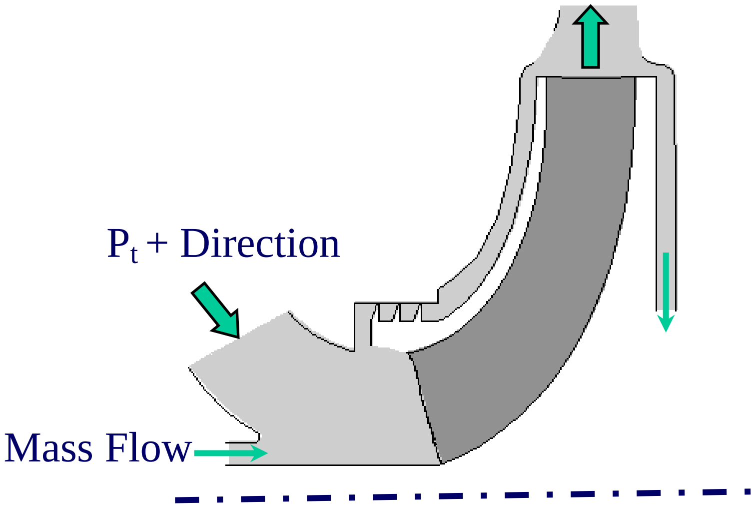
```
Changed zone 16 and 17 to type interface, and created a non-conformal periodic interface.
```

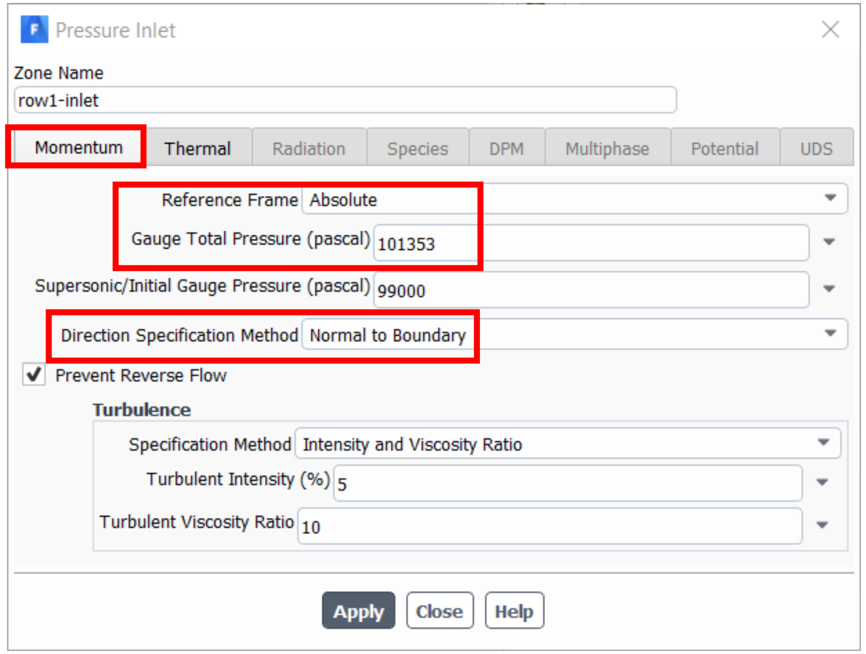
  - Visualization in Outline tree → Two boundary patches is shown under Interface

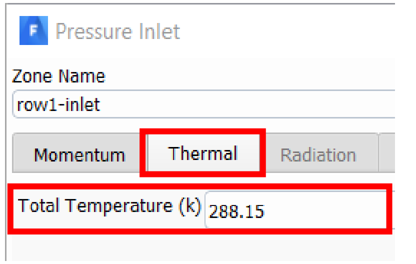


Boundary Conditions: Inlets

- Pressure inlet with absolute frame total pressure and direction is the preferred inlet boundary condition:
  - Inflow energy is usually known
    - For compressible flows, total temperature is also specified
  - Total pressure allows for gradients in computed inlet velocity and pressure
- Velocity- or mass-flow-inlets can be specified at inlet:
  - Usually less accurate
  - Often distorts the energy or total pressure profile, particularly if an impingement region is directly downstream
  - Can impose a known inlet velocity profile, e.g. if known from measurements

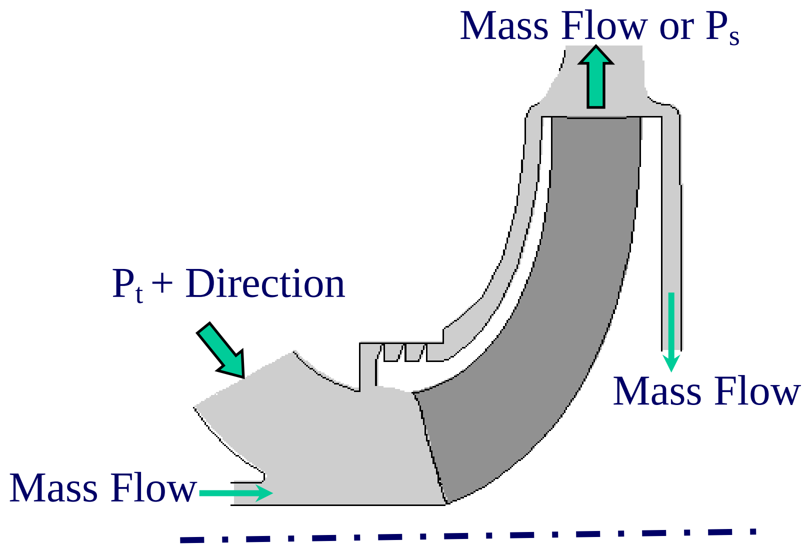


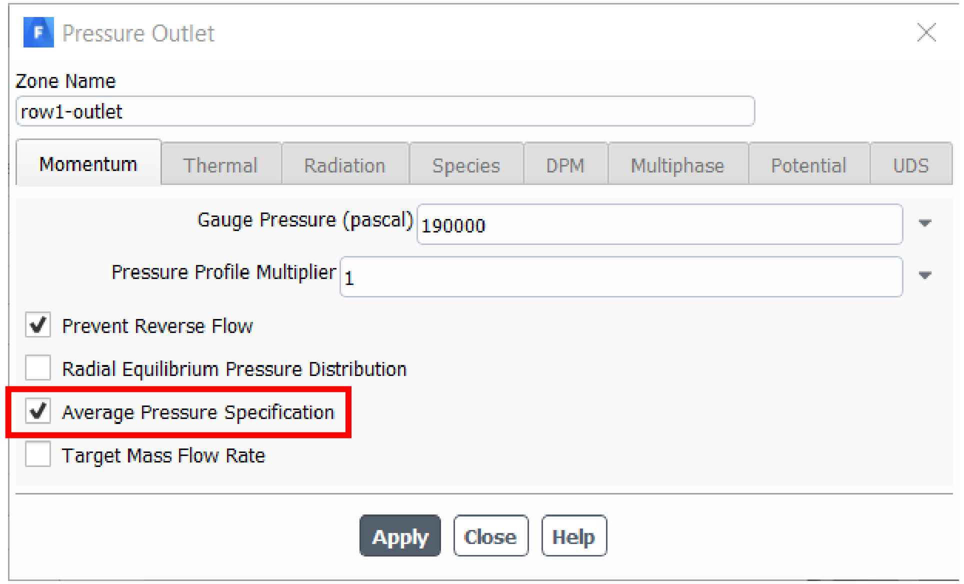




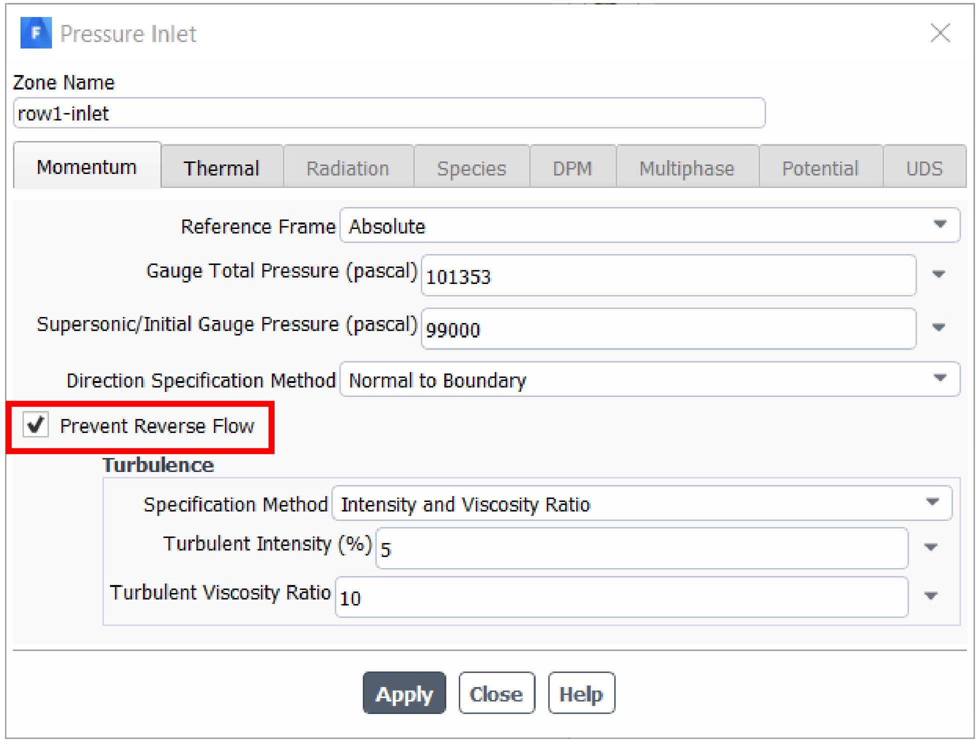
Boundary Conditions: Outlets

- *Average Static Pressure* or *Mass Flow Rate* is recommended at outlets
  - *Average* → Do not constrain the distribution of velocity or pressure, this is part of the solution
  - Convergence is often better with *Mass Flow Rate*
- *Mass Flow Rate*: use for pumps and compressors (away from choke)
- *Average Static Pressure*: use for turbines and compressors (near choke)
  - For axial turbomachines, use *Radial Equilibrium Pressure Averaging Option*
- To effectively compute speedlines for compressible flows use *Exit Corrected Mass Flow Rate*
  - *Functions well across the entire speedline*
  - *Improves stability for poor initial conditions*

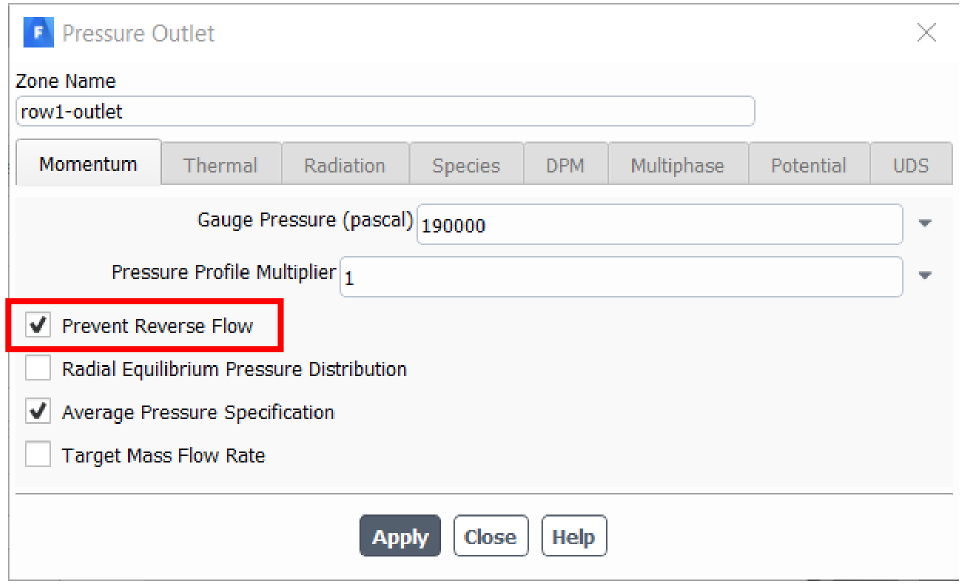




Boundary Conditions: "Prevent Reverse Flow" at Inlets and Outlets



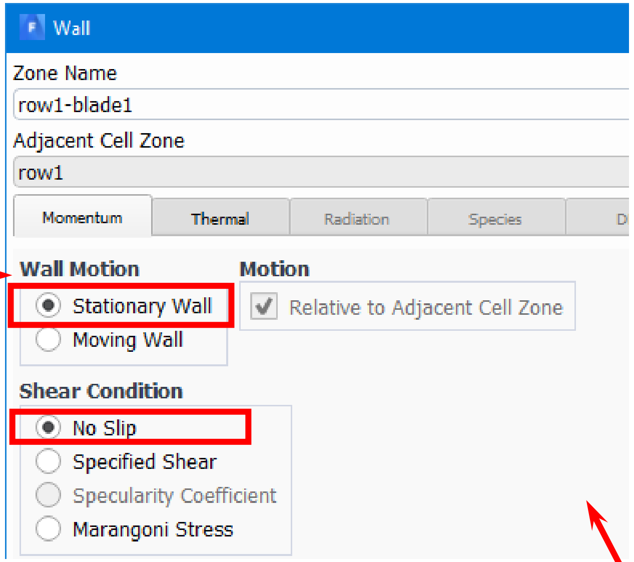
- Use the "Prevent Reverse Flow" option for all inlets and outlets
- In case of any problems with "artificial walls", consider expanding the numerical domain.

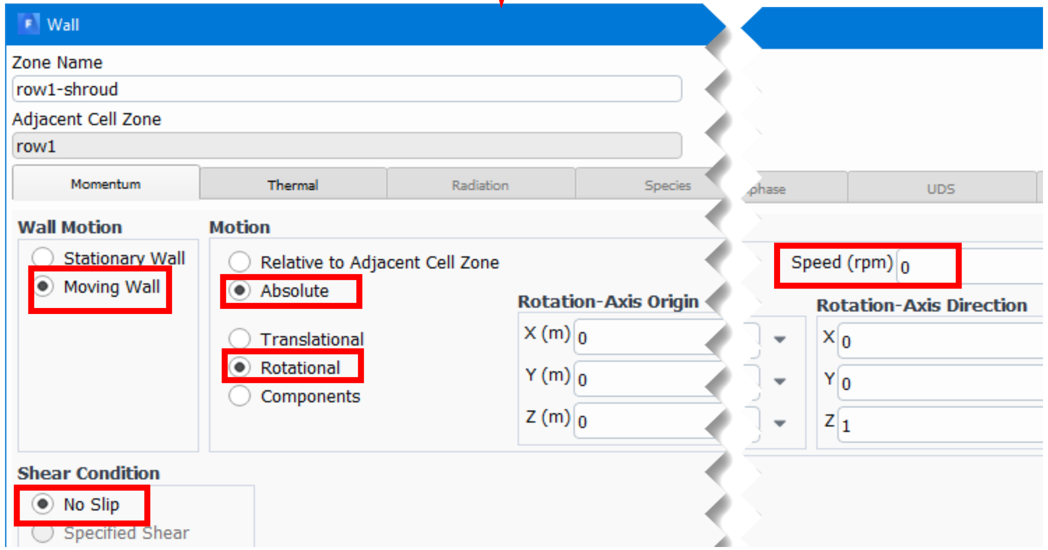


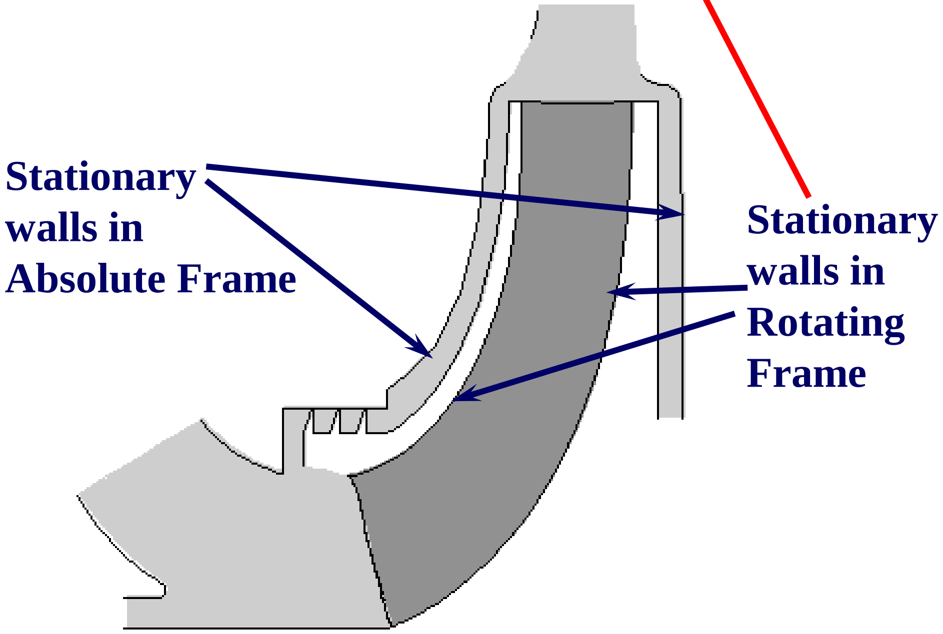
Boundary Conditions: Walls

- In the rotating frame, all rotating (with respect to the absolute system) walls are seen as stationary walls
  - Leave the default No Slip, Stationary Wall, Relative to Adjacent Zone.

- All fixed relative to the absolute frame housing walls should be set as rotationally moving walls at a zero rotating speed relative to the Absolute Frame





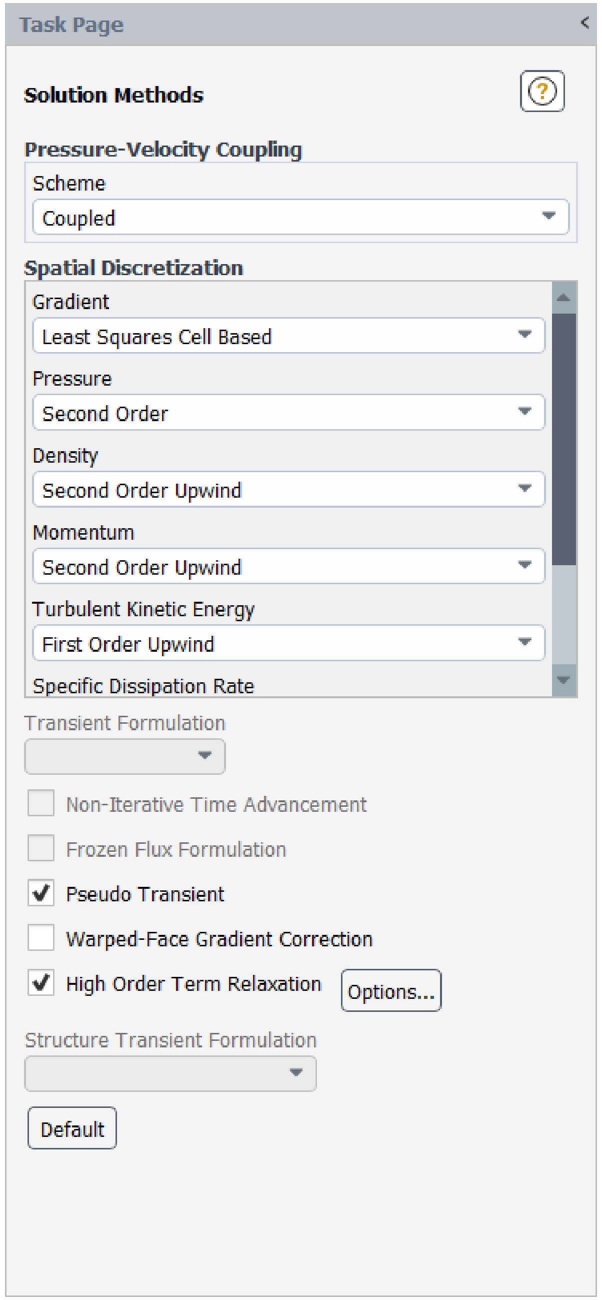


Ansys

Solution: Solver Settings – Solution Methods

- Use the default Coupled Solver
  - "Pseudo-Transient"
  - Spatial Discretization
    - Gradient: Least Squares Cell Based
    - Pressure: Second Order
    - Momentum and Energy: Second Order Upwind
      - Better prediction of head and losses *
    - Turbulence: First Order Upwind
      - First Order Upwind usually OK for turbulence
      - Use High Resolution for increased accuracy if you are modeling transition from laminar to turbulent flow
- Turn on "High Order Term Relaxation"

* Lower order discretization is more robust and stable but is less accurate. Discretization errors show up as false loss of total pressure, and hence errors in prediction of head and losses



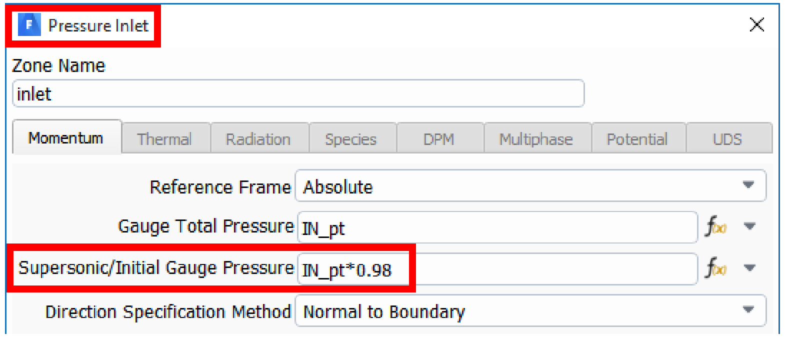
Solution: Initialization

- Good initialization of the solution is often the key to obtaining rapid and robust convergence of turbomachinery problems
  - Less of an issue for
    - Incompressible flows with velocity / mass flow inlets
      - Fixed flow rate provides stability to the calculation
    - Problems with favorable pressure gradients (e.g. turbines)
      - Less propensity for reverse flow at boundaries
  - More of an issue for
    - Compressible flows with adverse pressure gradients (e.g. compressors, diffusers)
      - Adverse pressure gradient leads to flow separation and reverse flows, solution instability

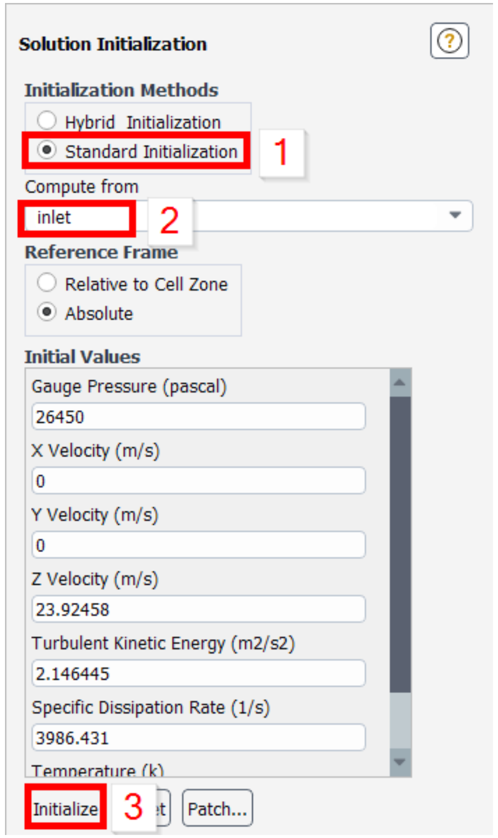
Solution: Initialization Procedure

1. Set an “Initial Gauge Pressure”, which is about 1 to 2% lower than the used inlet pressure
   - Set this at the inlet boundary
2. Perform a “Standard Initialization” using the inlet values (for a proper k and omega initialization)
3. Perform a “Hybrid Initialization” using the option “Use Specific Initial Pressure on Inlets”
   - It is a good idea to visualize the initial guess flow field before proceeding to the solution

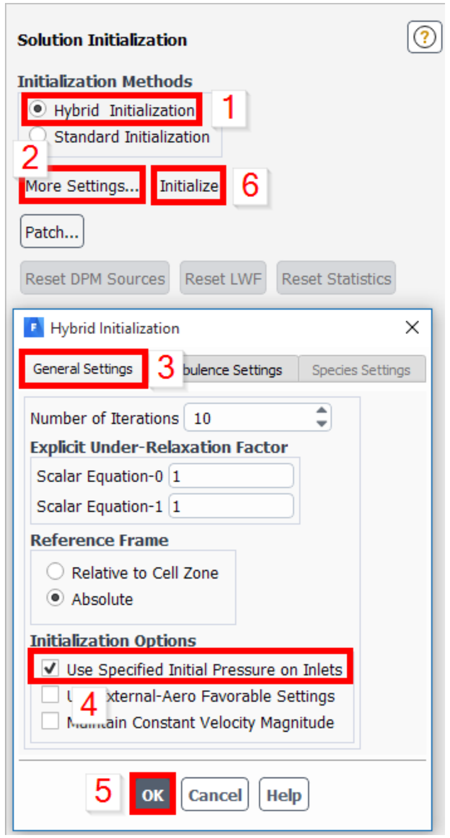
1.



2.

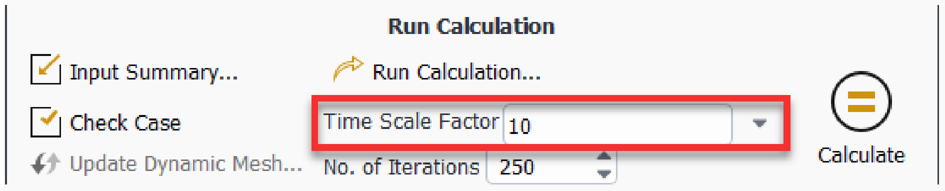


3.



Solution: Time Scale Factor for Steady State Simulations

- Use a "Time Scale Factor" between 5 to 10
  - A smaller Time Scale Factor, corresponding to a smaller time step (more relaxation) might be required when starting from a poor initial guess. After initial convergence, increase the time scale factor. Before accepting final results, make sure solution has run with a time step proportional to the largest time scale of the flow. Aim for 1/omega as final time step (corresponding to a "Time Scale Factor" of 10 in Fluent).

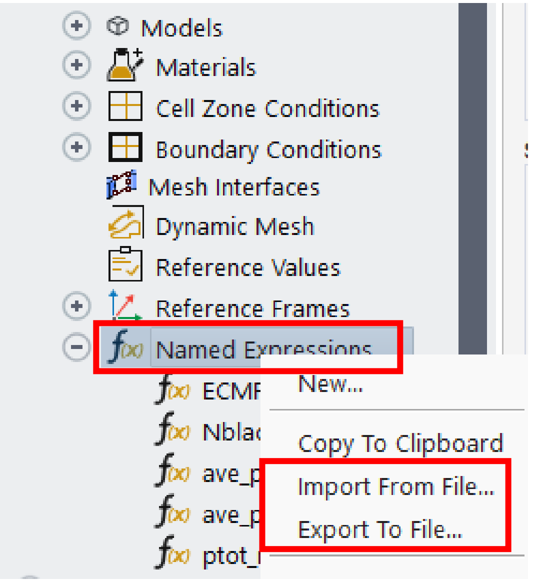


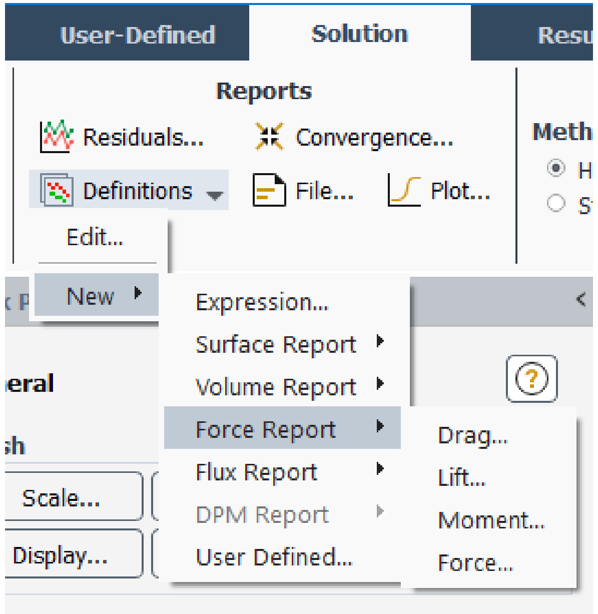
Ansys

Solution: Report Definitions & Plots

Create several report definitions to help you Judge convergence

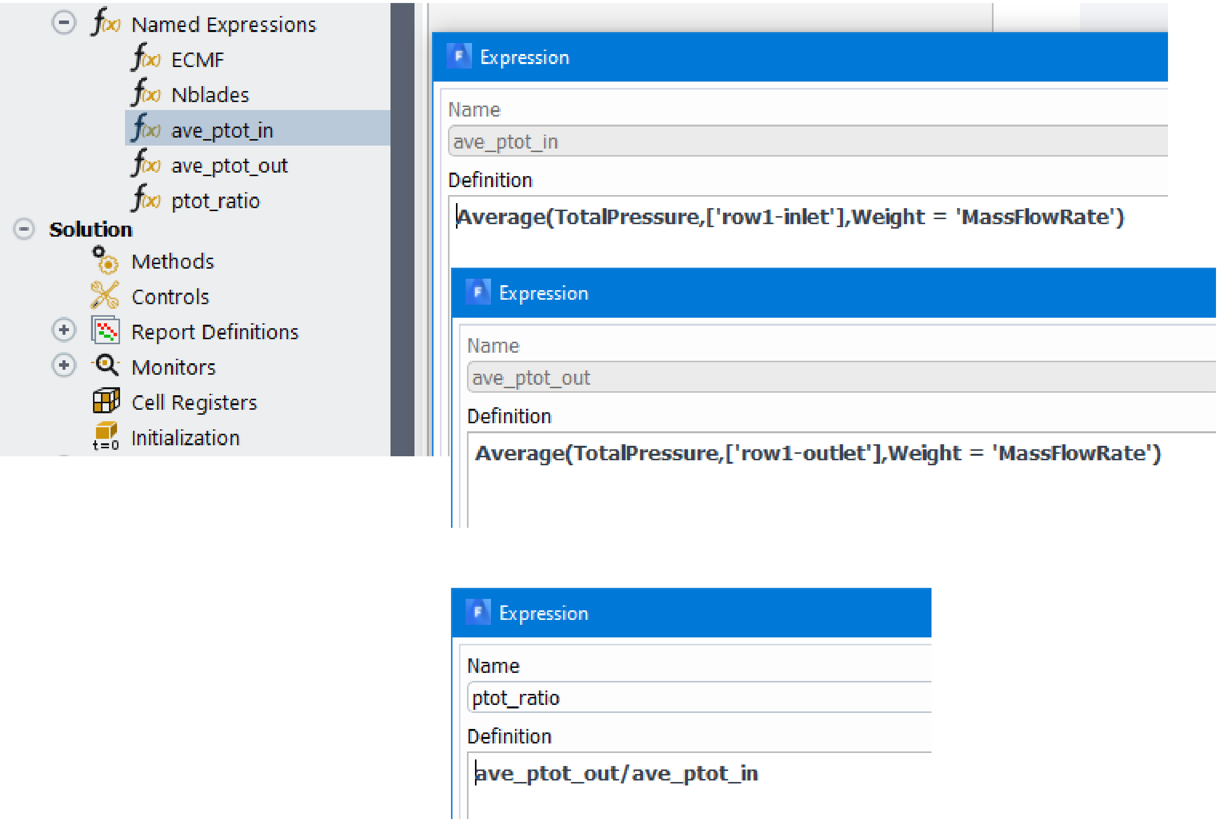
- Surface Reports
  - Best approach: Create Named Expressions of the following and use them in Report Definitions
    - Flow Rate
    - Mass-Weighted Average of Total Pressure, and Temperature at inlets and/or outlets
    - Area-Weighted Average of static Pressure at inlets and/or outlets
    - …
  - Can RMB on Named Expressions in the Outline and Export to a .tsv File. Can reuse this .tsv file in other similar cases (RMB Import from File)
- Force Report
  - Named expressions not yet available for forces (planned for next release)
  - Use the classic Force Report panel
    - Moment
    - Force





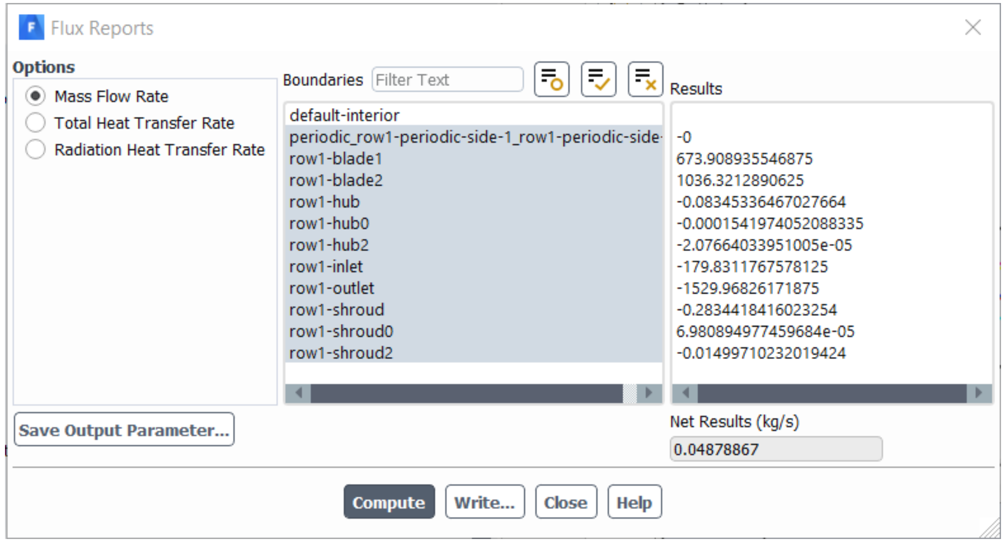
Example Report Definition for Compressor Pressure Ratio

- Create Named expressions for mass-weighted averaged Ptot at the inlet and outlet
- Create Named expression for the ratio
- Use this expression in Report definition

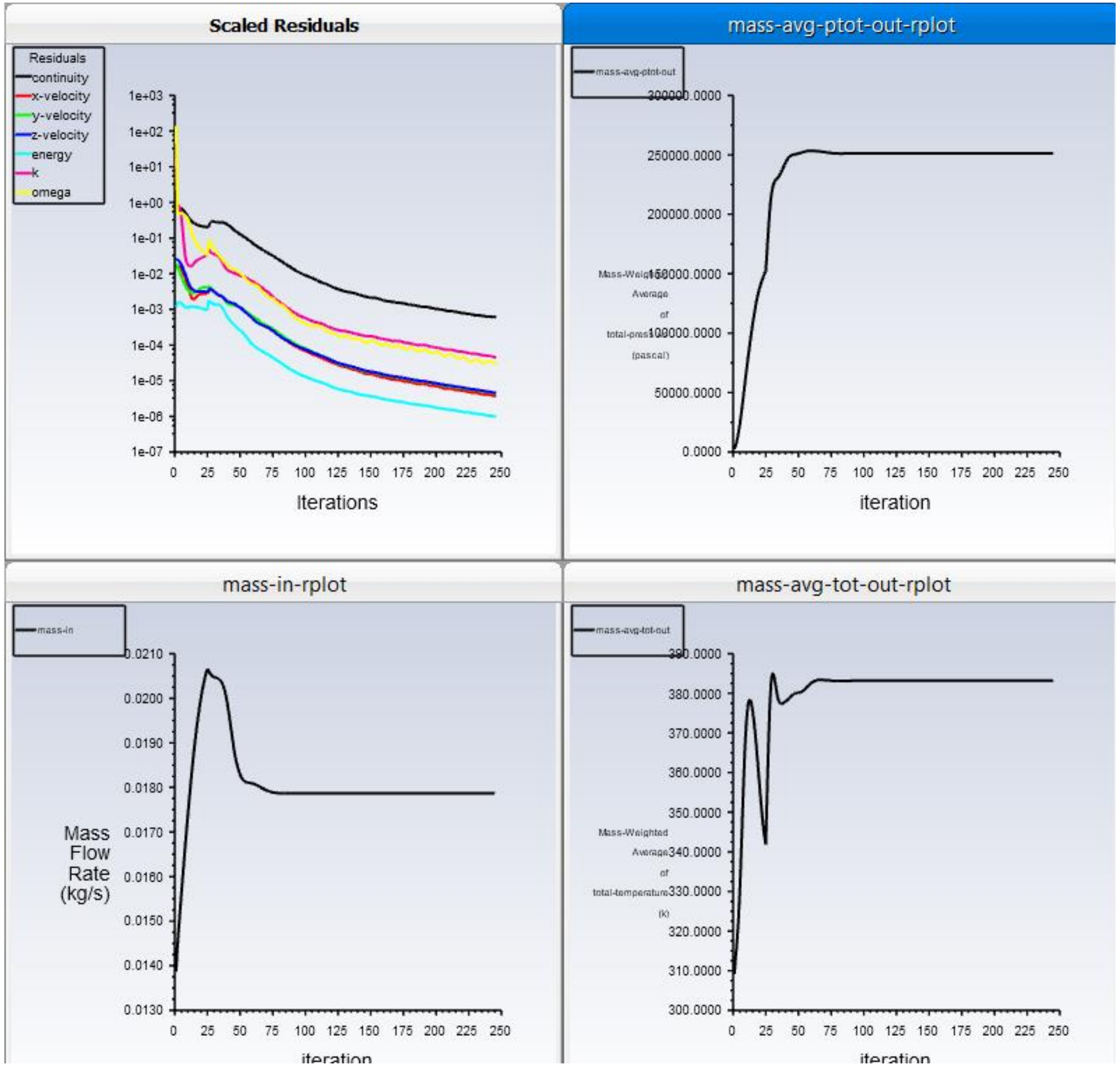


Solution: Convergence

- Judge convergence by monitoring
  - Solution residuals
  - Report Plots
  - Mass and Energy Report Fluxes



Note that for physically unstable or mildly unsteady flows, steady-state solution monitors may oscillate. Consider running the case further as transient



Ansys

Summary

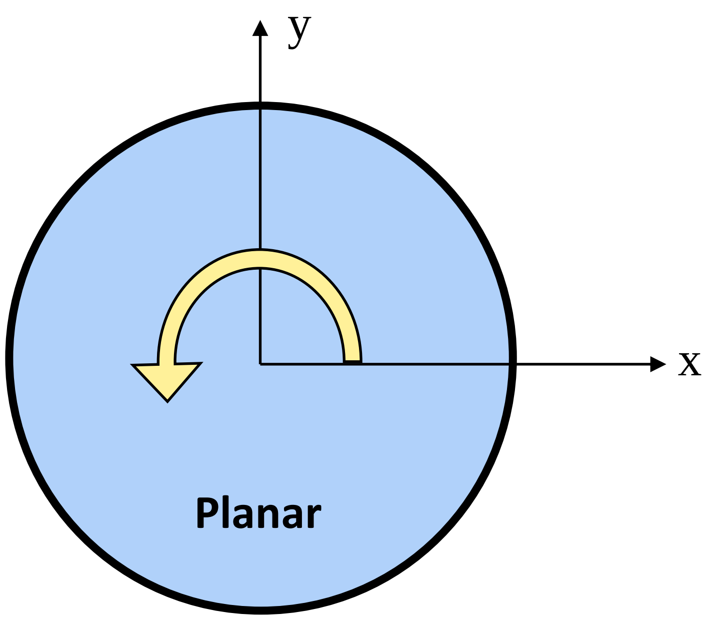
- Single Rotating Component Analysis is the simplest modeling approach for rotating machinery
- Applicable for Single rotating domains (e.g., rotor domain) without non axisymmetric stationary parts in the absolute frame
- Rotational Periodicity allows for reducing domain size (one passage vs. full 360 wheel)
- For steady state solution use time scales of the order of 1/omega

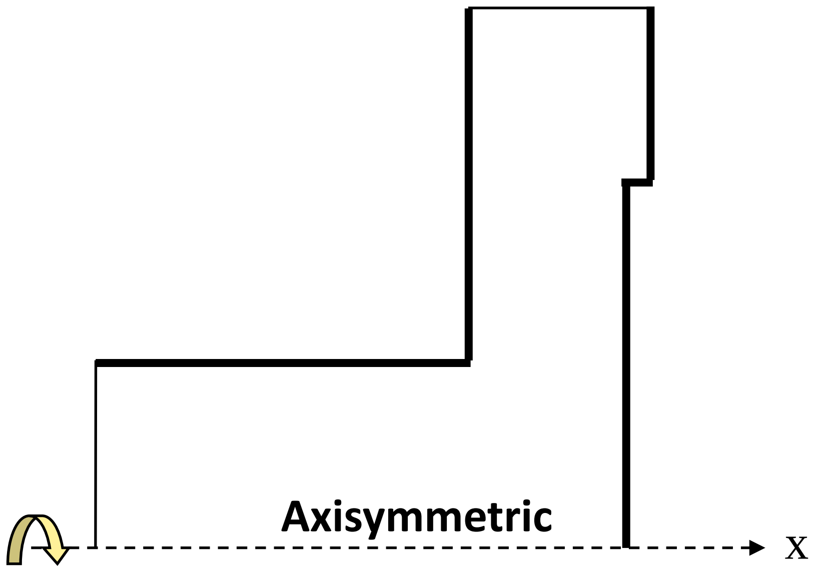
Appendix

Rotating Reference Frames and 2-D Geometries in Fluent

Single Moving Reference Frame Geometries: 2-D (Fluent)

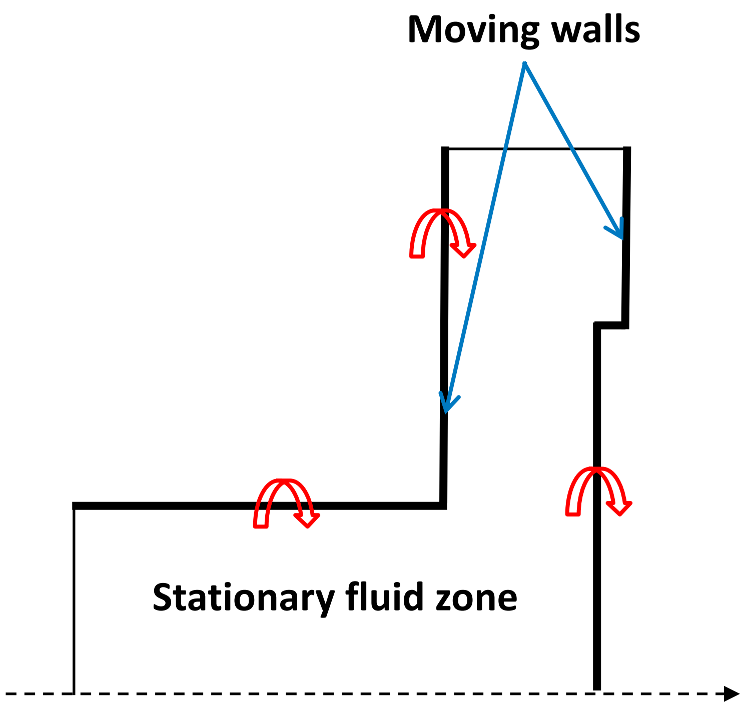
- 2-D planar
  - Geometries may rotate about axis normal to x-y plane with specified origin (periodic boundaries are permitted)
- 2-D axisymmetric, axisymmetric with swirl
  - Geometries rotate about the x-axis by default





Modeling Axisymmetric Geometries

- You do NOT need to use a rotating reference frame for 2-D and 3-D axisymmetric geometries!
  - Define fluid zones to be stationary frame and <u>prescribe tangential wall rotational motion for the moving walls</u>
  - Moving wall velocity is a valid BC in a stationary zone provided that the wall motion is purely tangential w.r.t. the cell zone
    - No BC velocity normal to the wall is permitted!



End of presentation