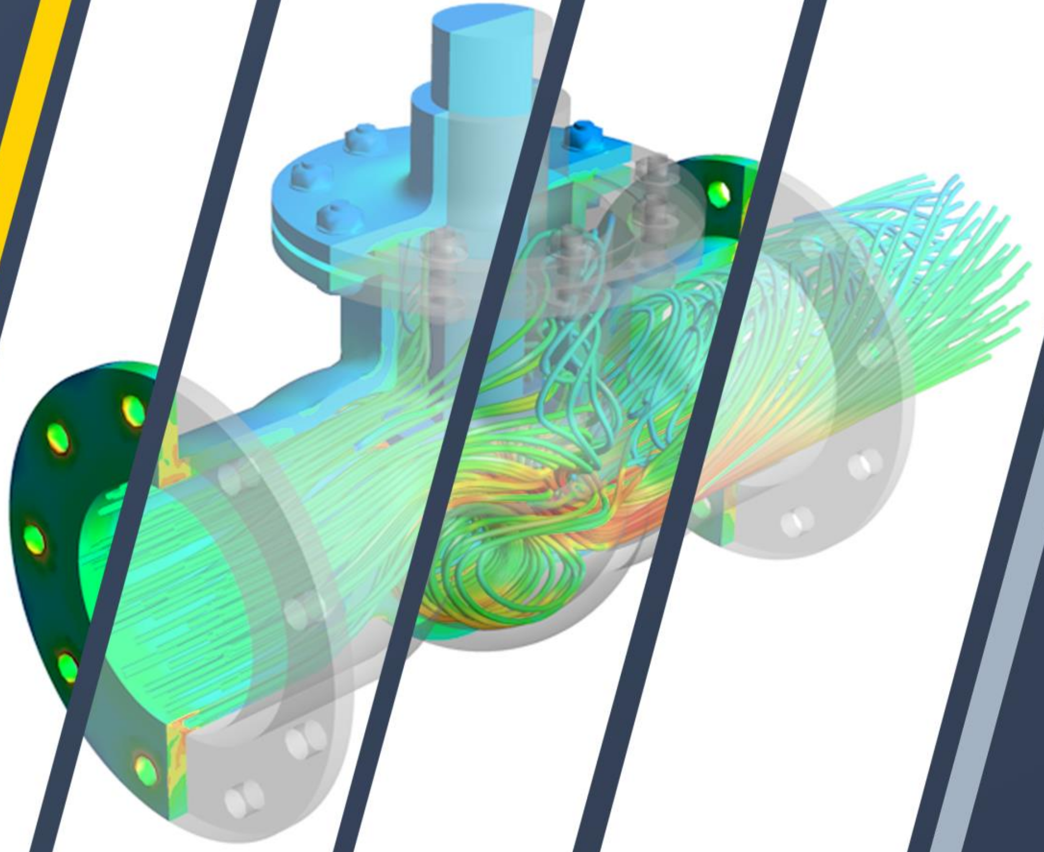
Lecture 02: Single Rotating Component Analysis



ANSYS CFX Rotating Machinery Modeling

Release 2019 R3

Outline

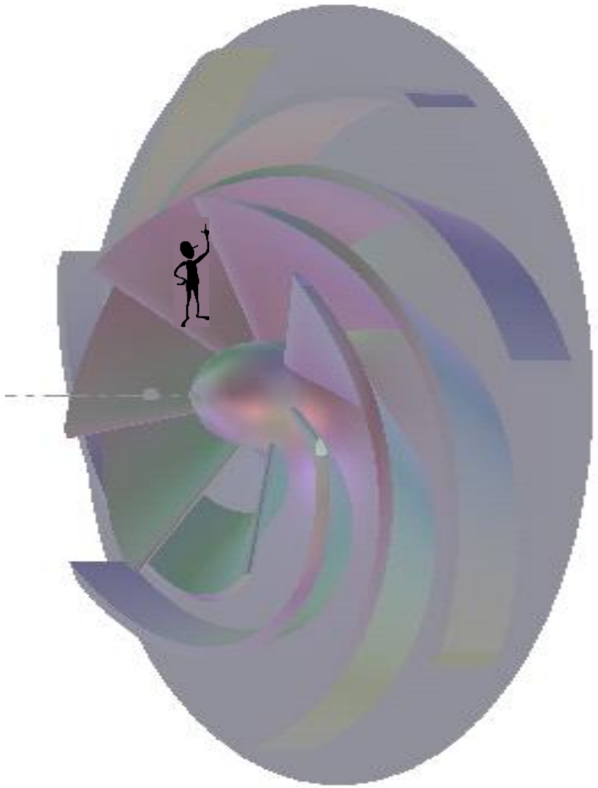
- Rotating Frames of Reference
  - Motivation
  - The Velocity Triangle
  - Governing Equations
- Single Rotating Component Analysis
  - Geometry Considerations
  - Workflow
    - Solver Settings and Physical Models
    - Boundary Conditions
    - Initialization and Solution
  - "False Swirl"
- Summary

Outline

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

Motivation

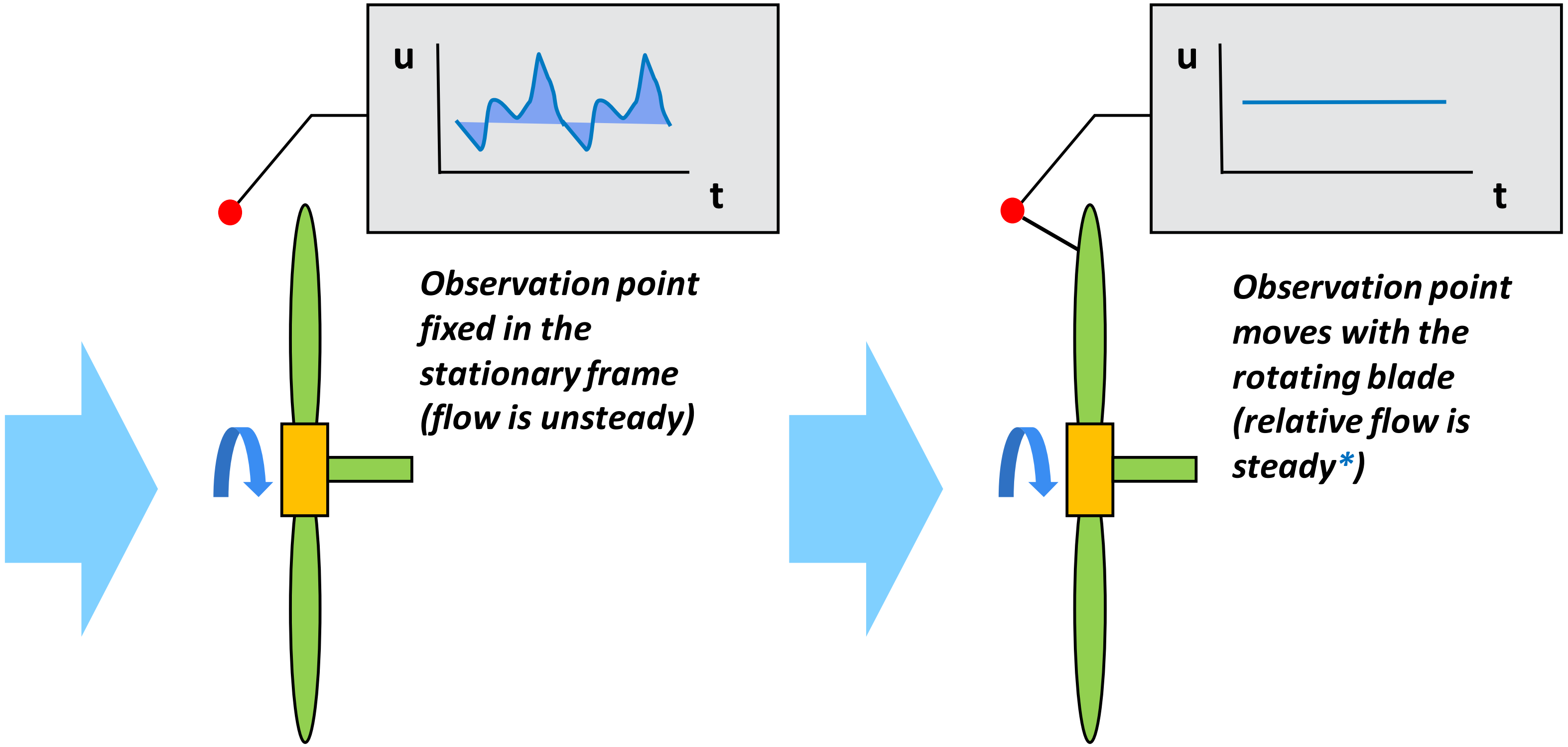
- Rotating machinery components can be best modeled using a **Rotating Frame of Reference (RFR)**
- Entire domain rotating at specified rate about an axis
  - For clarity, imagine yourself standing on the hub, moving with the blade row, looking out at the passage flow



- Why use an **RFR**?
  - 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 RFR

Uniform freestream flow



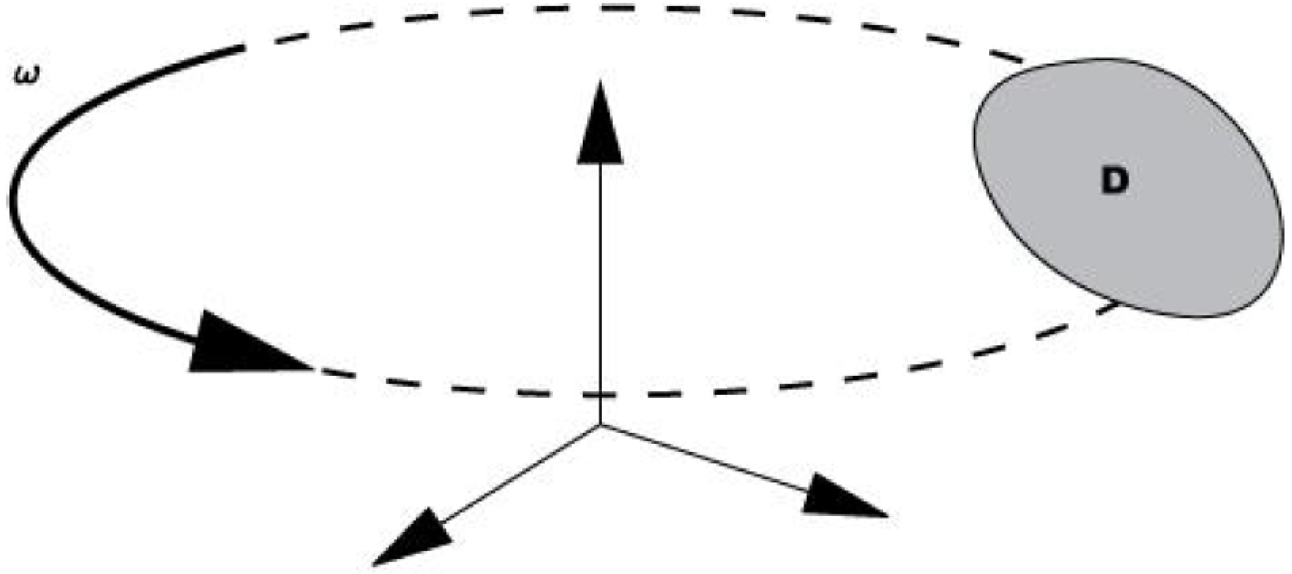
Fixed point in absolute frame

Fixed point in relative frame

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

Rotating Frame of Reference Illustration

- The CFD domain is rotating at an angular velocity ω with respect to the Global Coordinate system but is stationary in RFR
  - ω may be constant or a function of time

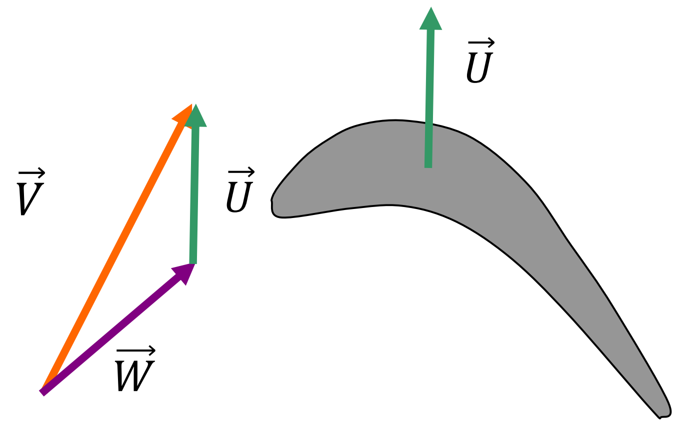


The Velocity Triangle

- 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

Governing Equations for Rotating Frame of Reference

- Equations are similar as for stationary frame
  - Relative velocities are used *
  - Additional source terms in momentum equations for the effects of the Coriolis and centrifugal forces

    * Absolute velocities can be used for the advection term in the momentum equations when enabling the Alternate Rotation Model for reducing the “False swirl”, see later slides

$$\boldsymbol{S}_{M,\mathrm{rot}} = \boldsymbol{S}_{\mathrm{Cor}} + \boldsymbol{S}_{\mathrm{cfg}}$$

$$\boldsymbol{S}_{\mathrm{Cor}} = -2\,\rho\,\boldsymbol{\omega} \times \boldsymbol{U}$$

$$\boldsymbol{S}_{\mathrm{cfg}} = -\rho\,\boldsymbol{\omega} \times (\boldsymbol{\omega} \times \boldsymbol{r})$$

$\boldsymbol{r}$ is the location vector

$\boldsymbol{U}$ is the relative frame velocity

- In the energy equation, the advection and transient terms use rothalpy I, instead of total enthalpy

$$I = h_{\mathrm{stat}} + \frac{1}{2}U^2 - \frac{1}{2}\omega^2 R^2$$

Quantities in Absolute and Rotating Frames of Reference

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

- Some are not
  - Velocity

- Others require careful interpretation
  - Stationary versus rotating frame total pressure

Outline

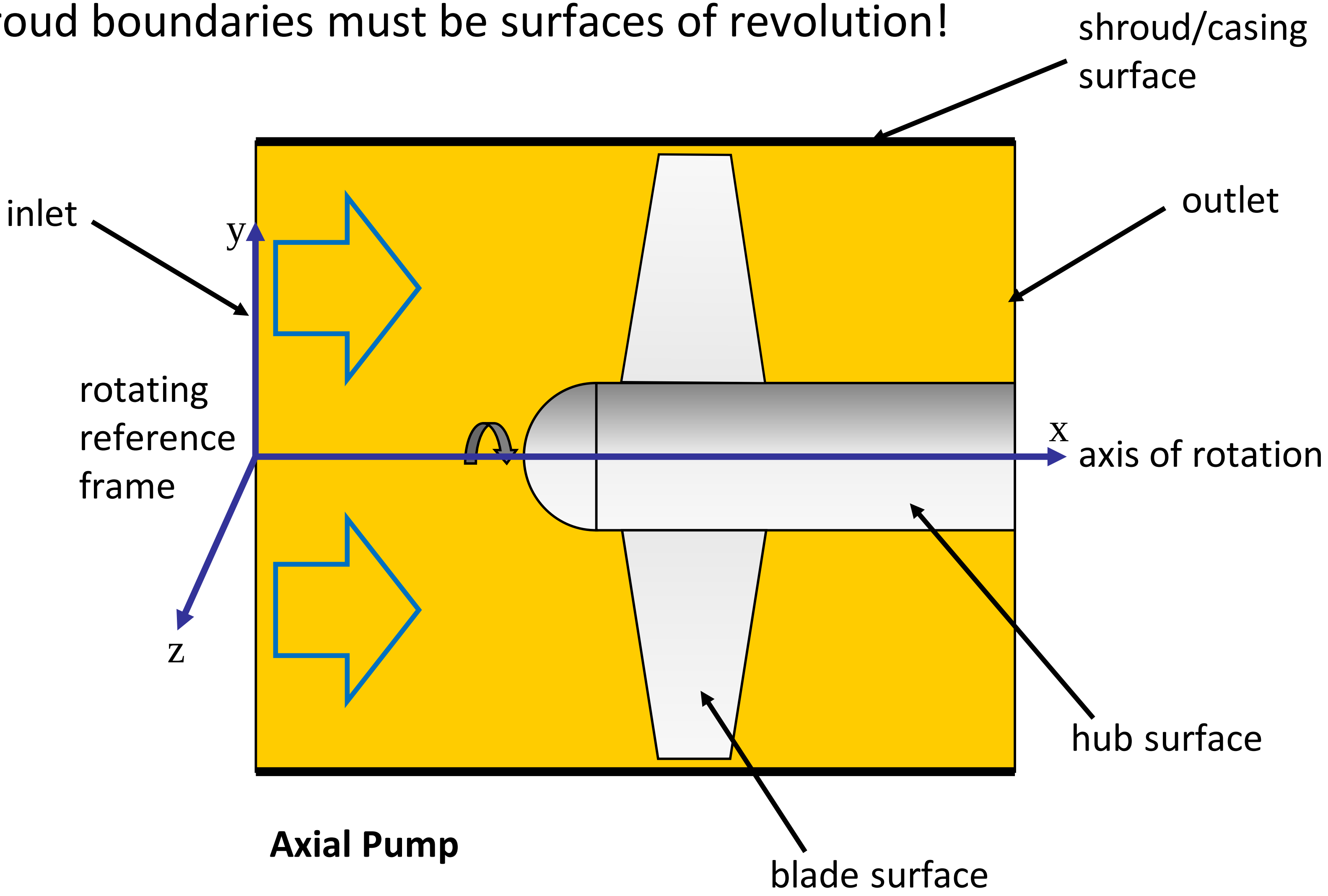
- Rotating Frames of Reference
  - Motivation
  - The Velocity Triangle
  - Governing Equations
- Single Rotating Component Analysis
  - Geometry Considerations
  - Workflow
    - Solver Settings and Physical Models
    - Boundary Conditions
    - Initialization and Solution
  - “False Swirl”
- Summary

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!



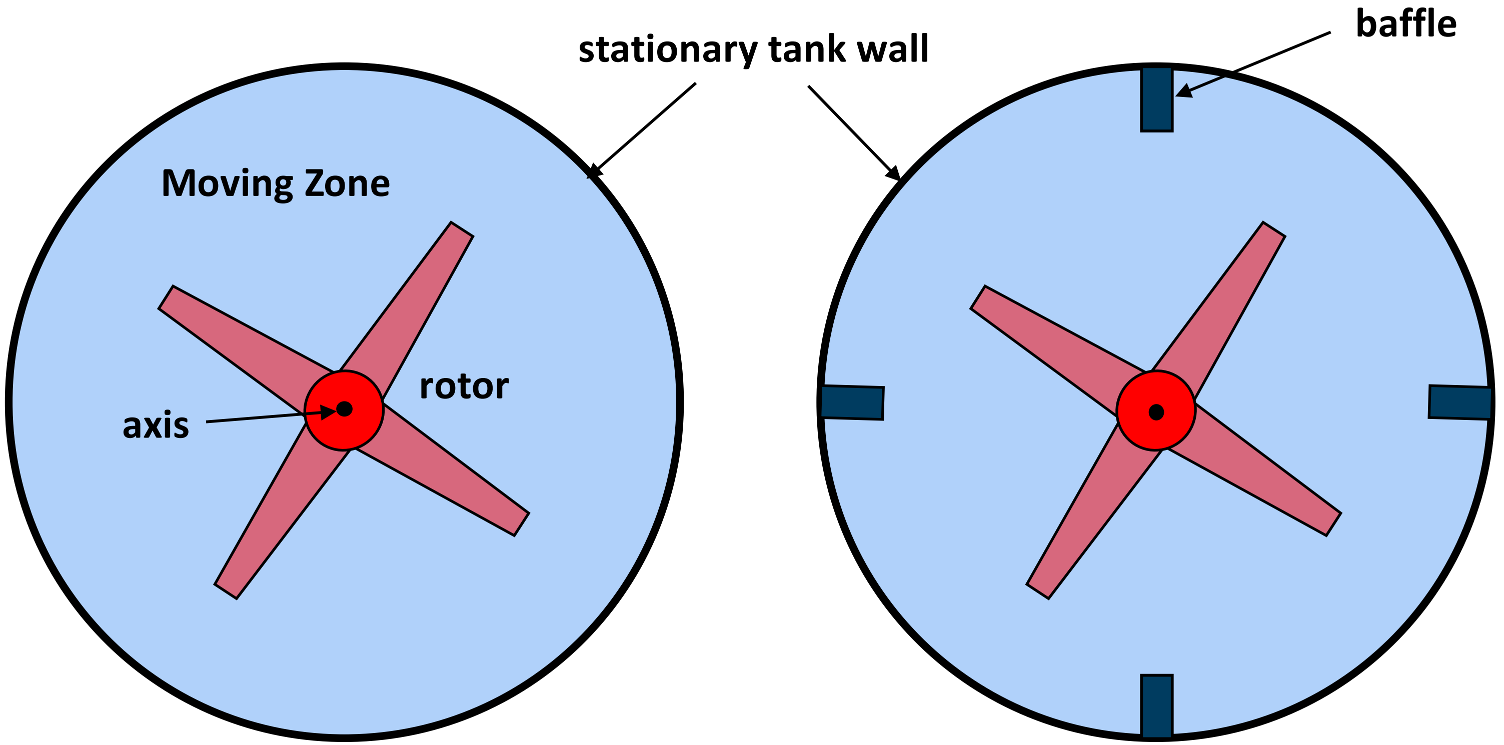
Axial Pump

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 Frame of Reference can be used

Single Rotating Frame of Reference 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)

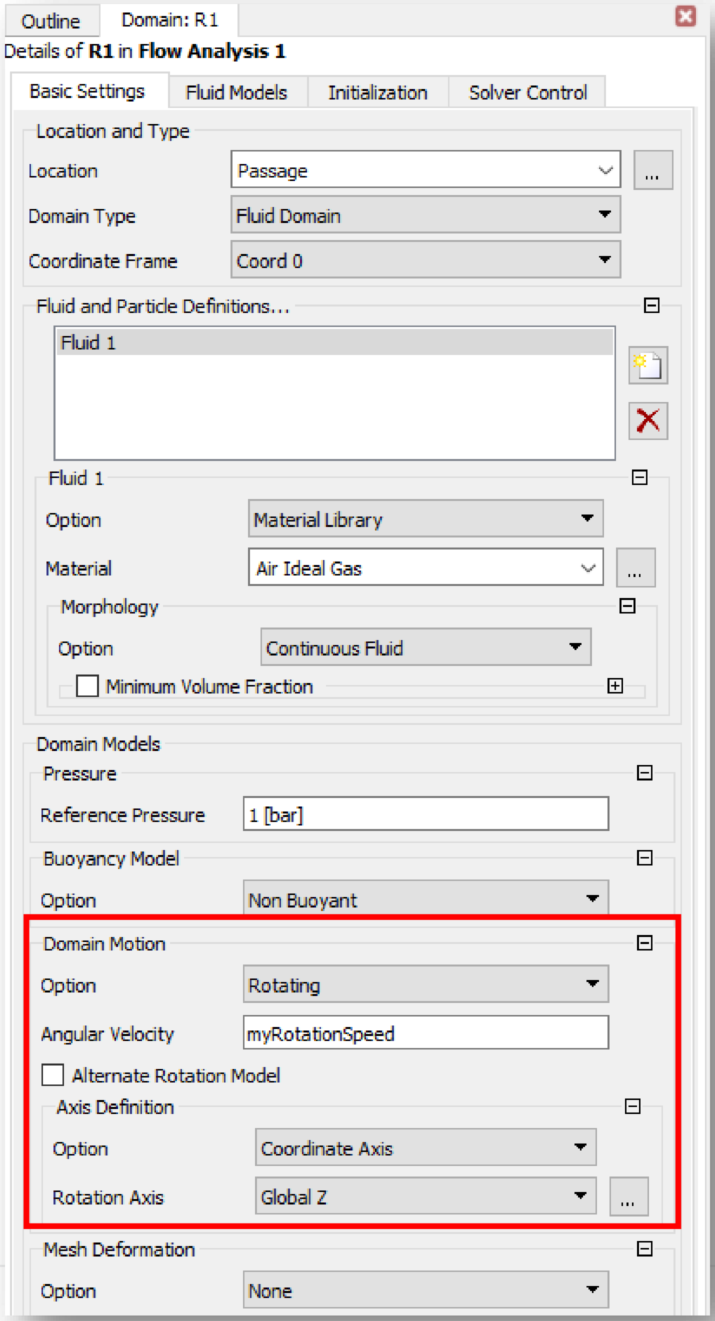
CFD Workflow for Single Rotating Component Analysis

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)

Fluid Domain Settings

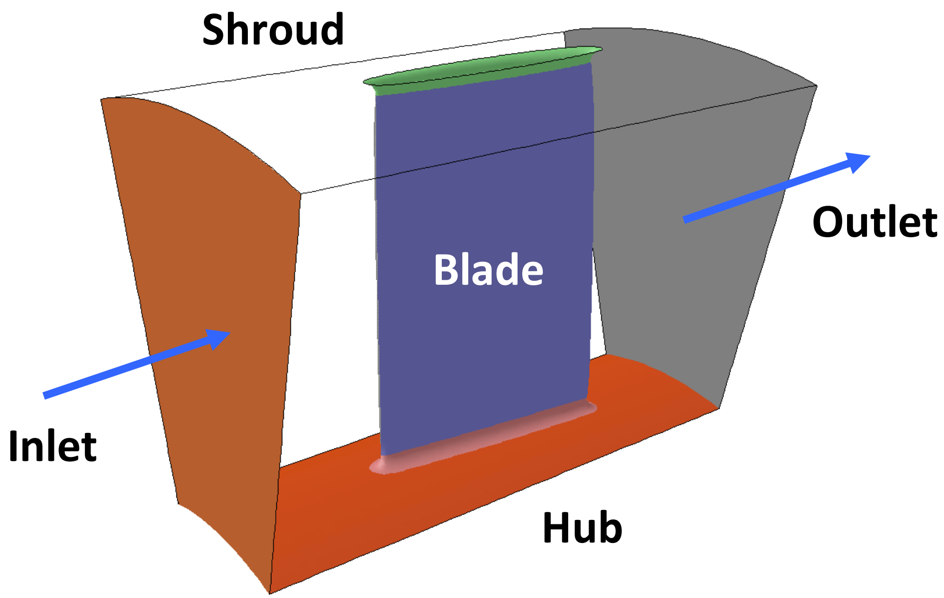
- Define Domain Motion
  - *Option: Rotating*
  - Set *Angular Velocity*
  - Define *Rotation Axis*
- Setup all other physical models as usual
  - Material Properties
  - Reference pressure
    - Set to zero for compressible cases
  - Physical models
    - Turbulence models
      - SST model is recommended for most cases
    - Heat transfer
      - Total energy with viscous work terms for compressible flows
    - …

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



Boundary Conditions

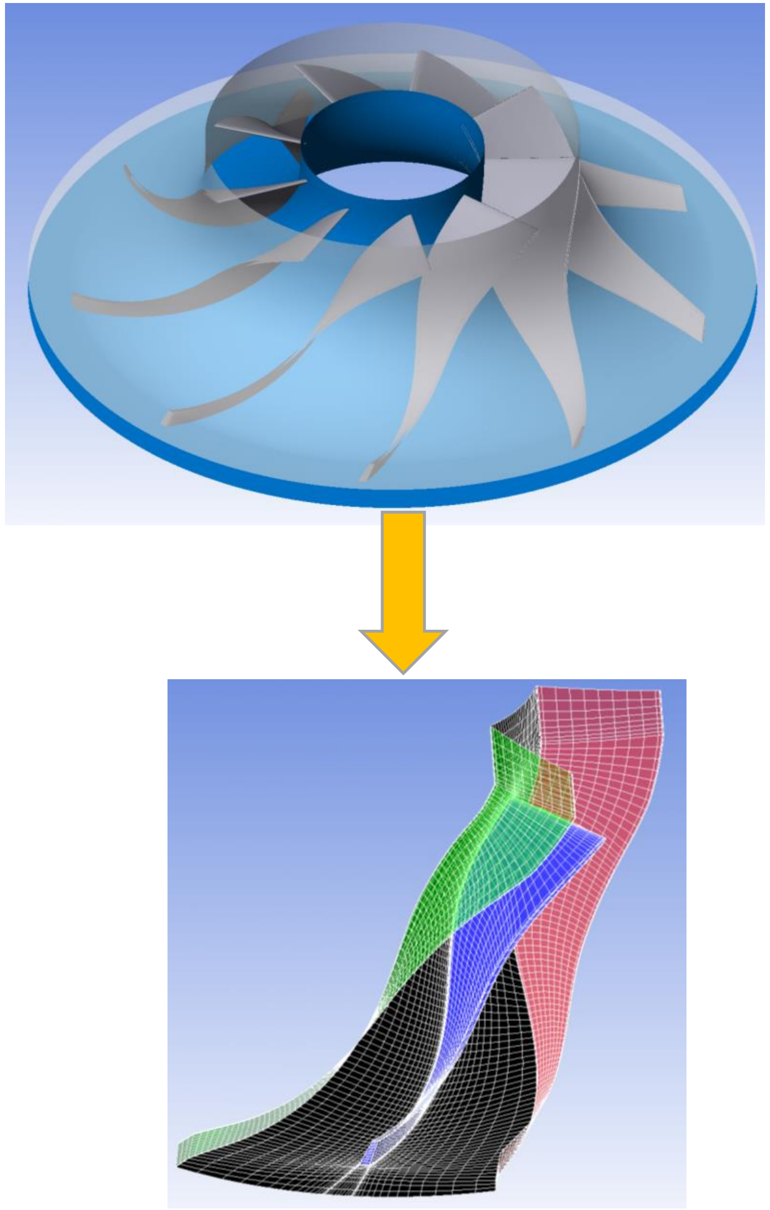
- Periodic interfaces
- Inlets
- Outlets
- Walls
- …



Axial Pump IGV

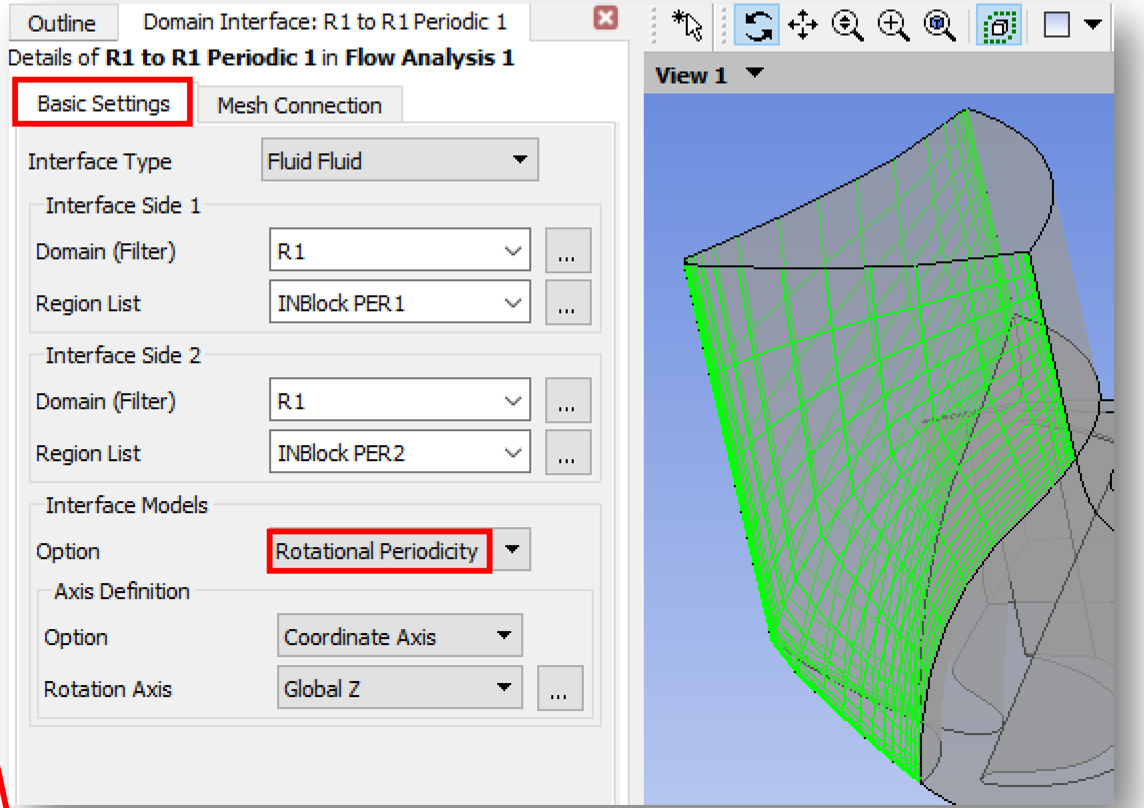
Rotational Periodicity

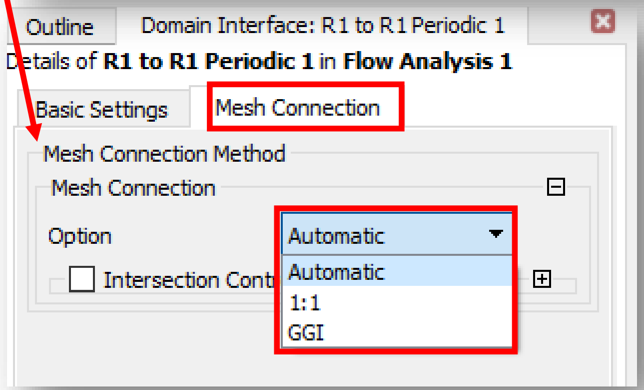
- Take advantage of rotational periodicity to reduce domain size
  - Solve for one blade passage only instead of the full wheel
- Must ensure both geometry and flow are periodic

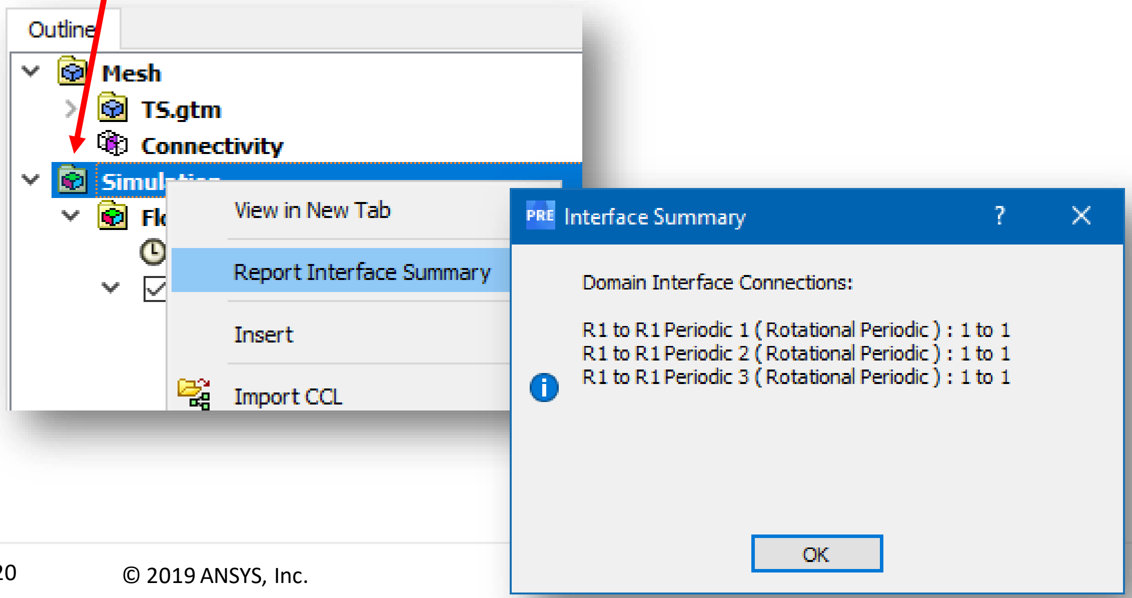


Create Periodic Interfaces

- Set *Interface Model Option to Rotational Periodicity*
- Can be conformal (1:1) or non-conformal GGI
  - Use the default *Automatic Mesh Connection Option*
  - Can check actual *Mesh Connection* in CFX-Pre
    - ➢ In the *Outline* RMB *Simulation > Report Interface Summary*

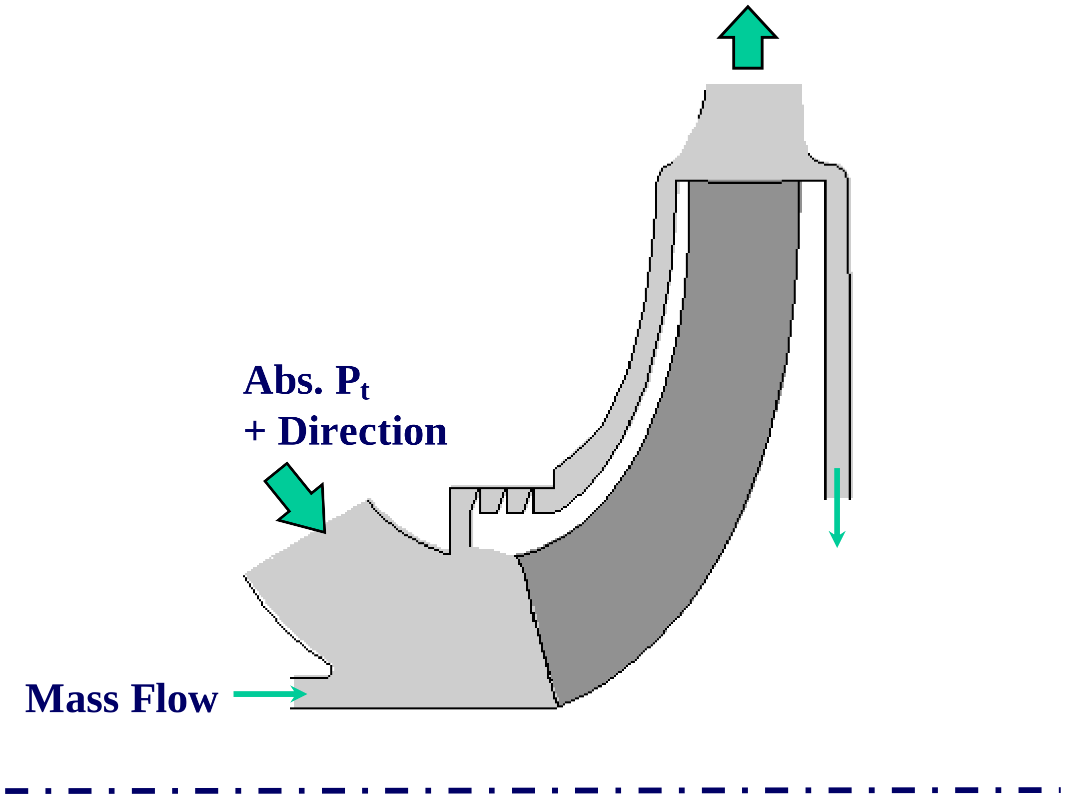






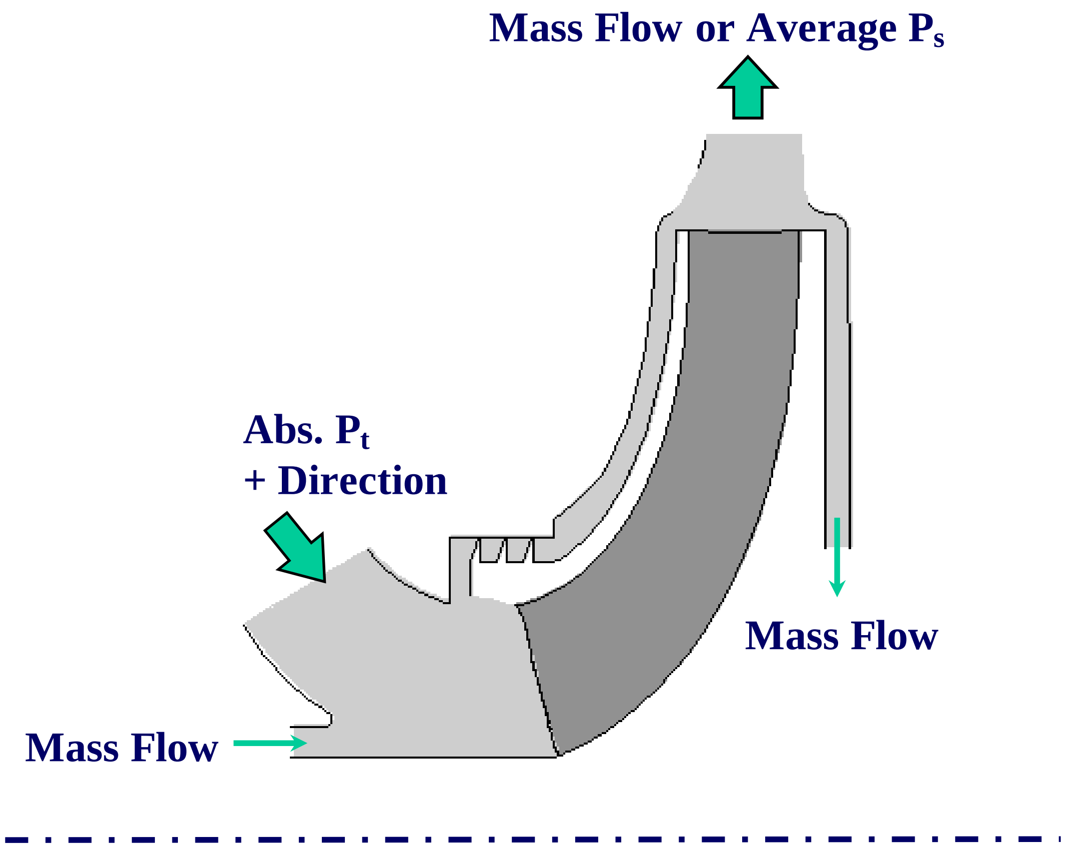
Boundary Conditions: Inlets

- Absolute total pressure and direction is the preferred inlet boundary condition:
  - ➢ Inflow energy is usually known
    - ▪ For compressible flows, absolute total temperature is also specified
  - ➢ Absolute total pressure allows for gradients in computed inlet velocity and pressure
- Velocity or mass 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



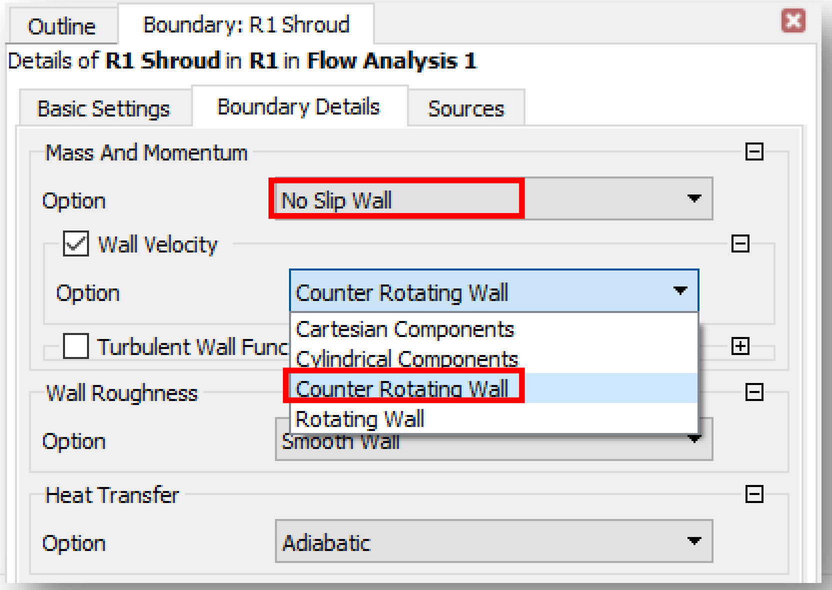
Boundary Conditions: Outlets

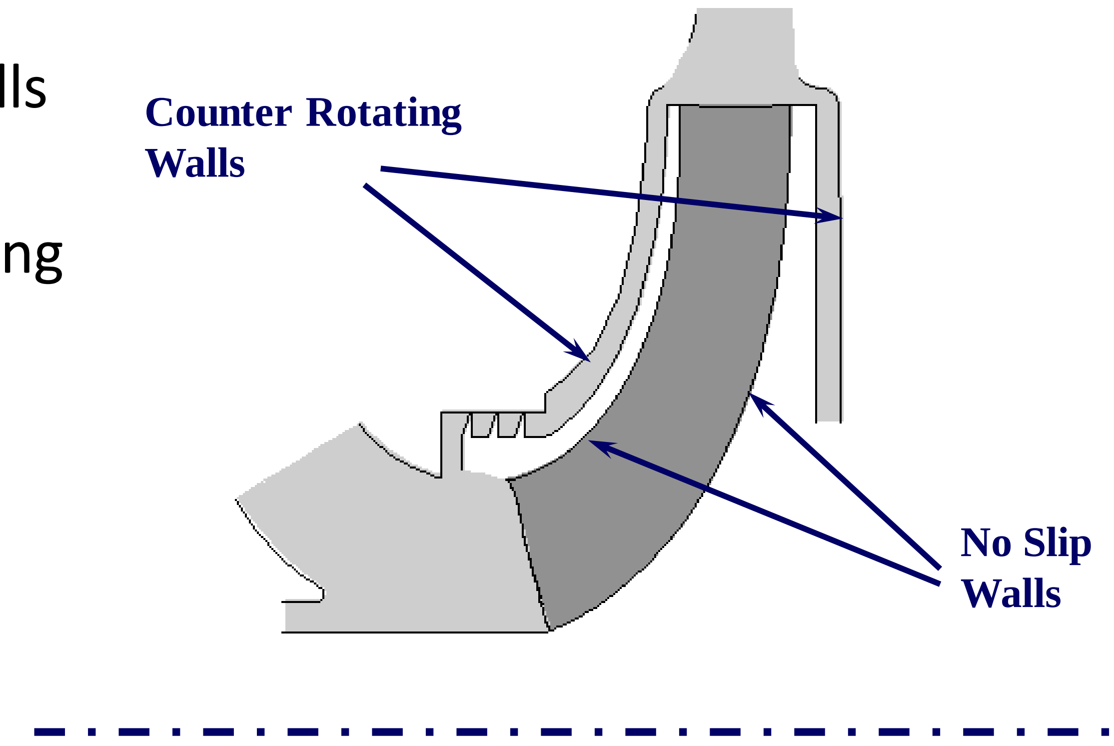
- *Average Static Pressure* or *Mass Flow Rate* is recommended at outlets
  - Do not constrain the distribution of velocity or pressure, this is part of the solution
  - Convergence is often better with *Mass Flow Rate*
- *Mass Flow Rate*: use for pumps and compressors (away from choke)
- *Average Static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



Boundary Conditions: Walls

- Everything is backward in the RFR (relative to our usual non-rotating viewpoint)
- In the rotating frame, most of the passage walls are seen as stationary (no slip walls w.r.t. RFR)
- All fixed (relative to the absolute frame) housing walls are seen as counter-rotating walls





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

Initialization

- Initial conditions in CFX-Pre
  - Default “Automatic” CFX-Pre initialization should fine for most applications
    - ➢ Based on boundary data to provide a good initial condition
- Initialization from an available completed simulation at a slightly different operating condition is a preferred option

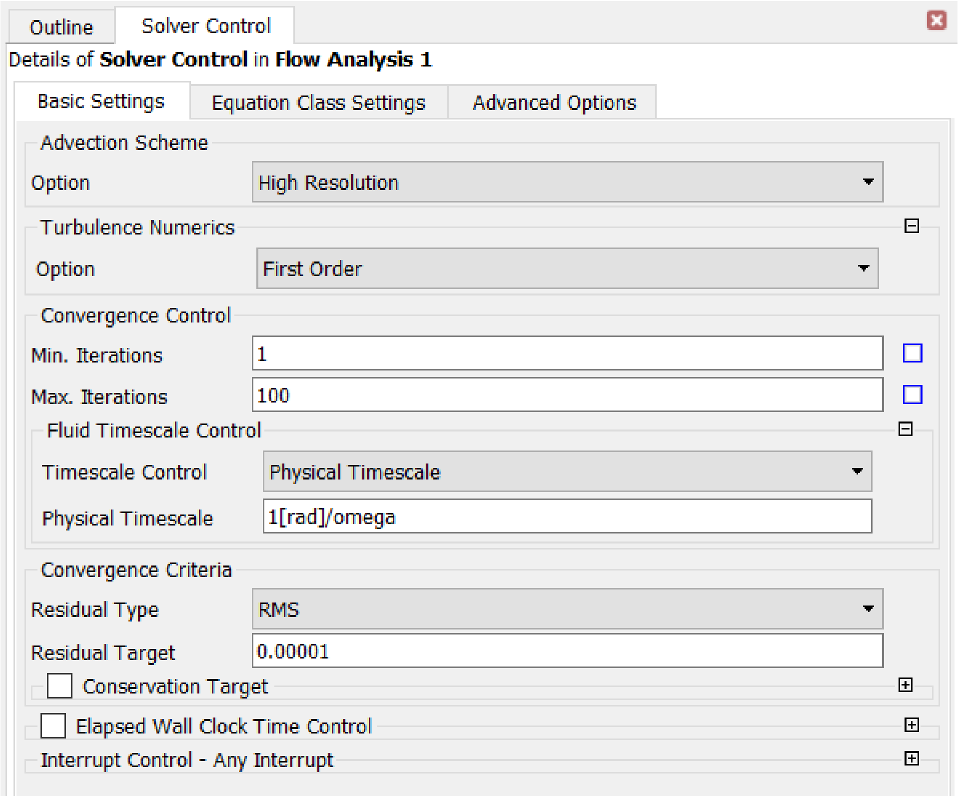
Solution Discretization and Controls (CFX)

- Use CFD Best Practices for general aerodynamics problems
  - High Resolution for Advection Scheme (bounded second order)
    - ➢ Better prediction of head and losses *
  - First Order usually OK for turbulence
    - ➢ Use High Resolution for increased accuracy if you are modeling transition from laminar to turbulent flow
  - Use 0.1-10/omega (rotational speed in [rad/s]) for timescale **

  * 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

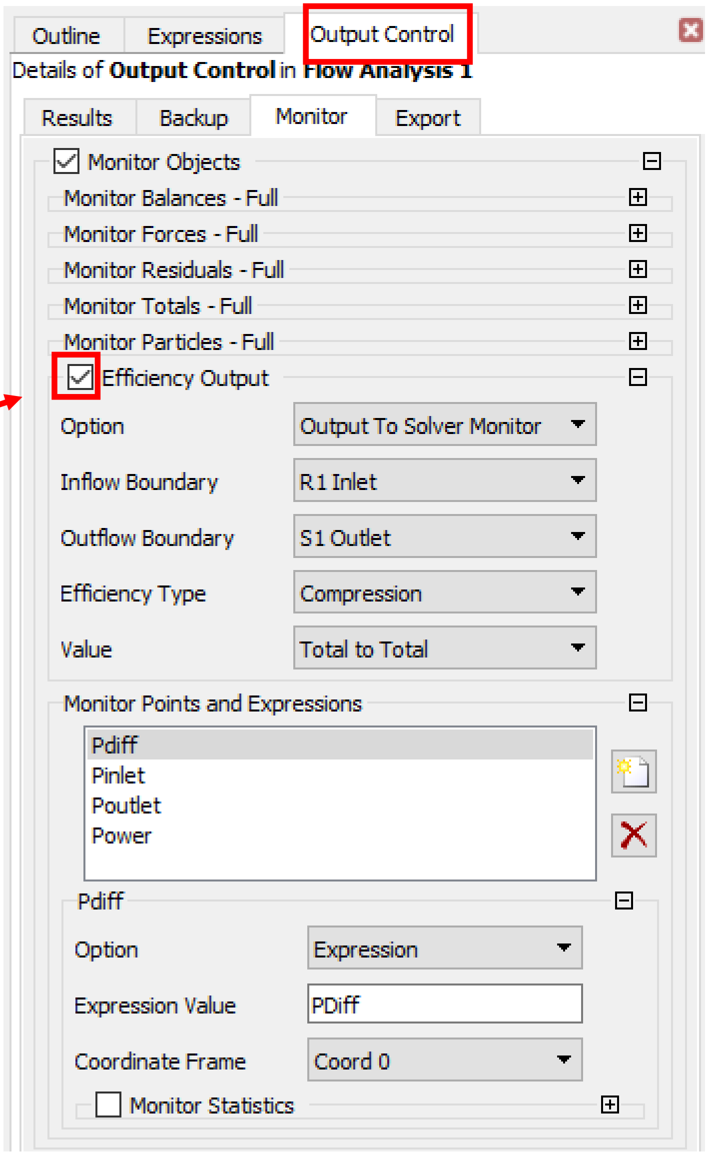
  ** A smaller time step (more relaxation) may be required when starting from a poor initial guess

  - ➢ After initial convergence, increase the time step
  - ➢ 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



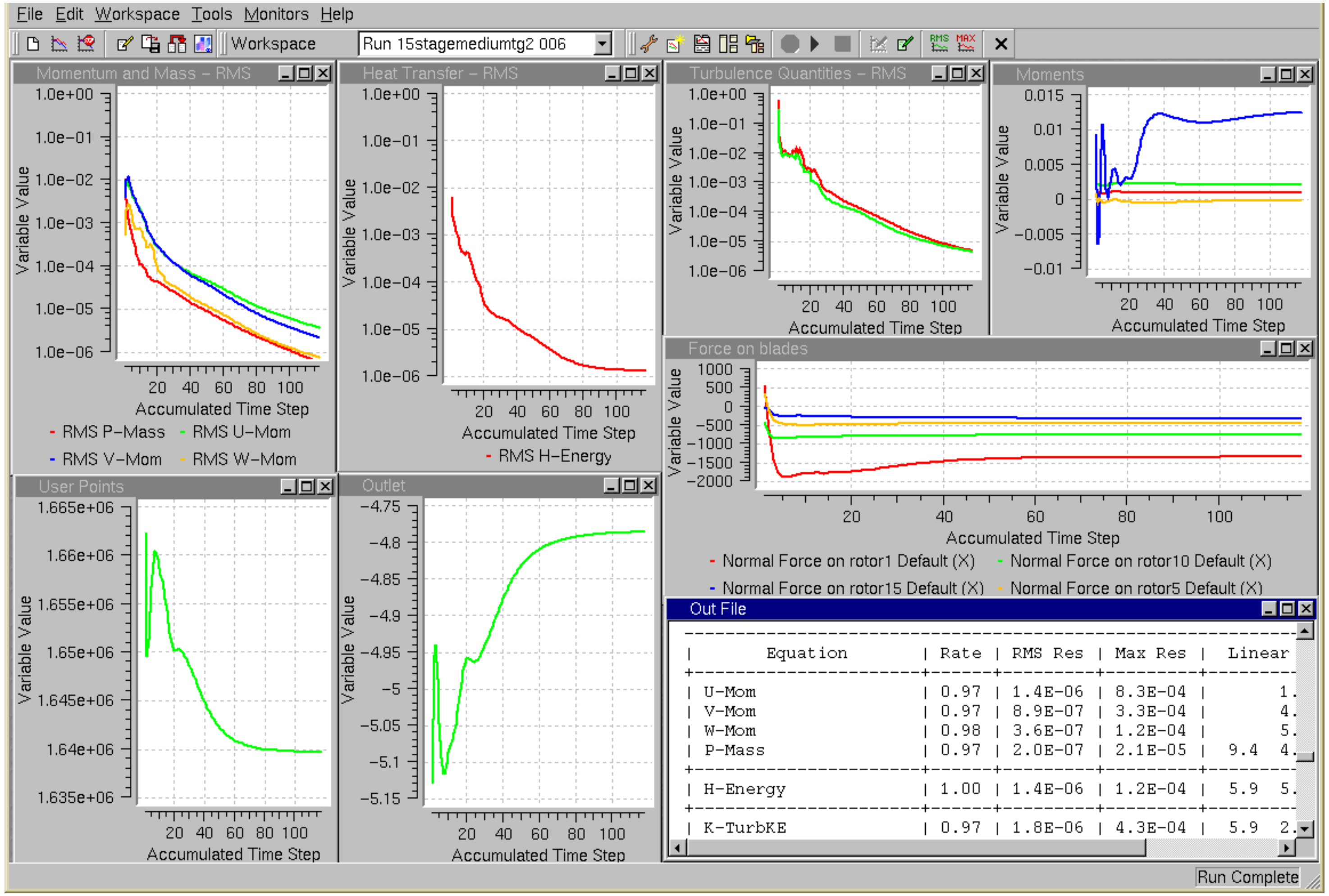
Output controls

- Create monitor points
  - Expressions for key parameters
    - ➢ Pressure head
    - ➢ Losses
    - ➢ …
  - For compressible cases enable Efficiency Output
    - ➢ Define Inflow and Outflow boundaries
    - ➢ Define Efficiency type
      - Expansion, Compression, both
    - ➢ Define Value option
      - Total to Total
      - Total to Static



Solution

- Use double precision for the run
  - When aspect ratios in bladed components >> 1000
- Judge convergence by monitoring
  - Solution residuals
  - User monitor points (defined in CFX-pre)
  - CFX Solver-manager built-in monitors
    - ➢ Imbalances
    - ➢ Mass flow rates
    - ➢ Forces
    - ➢ Moments

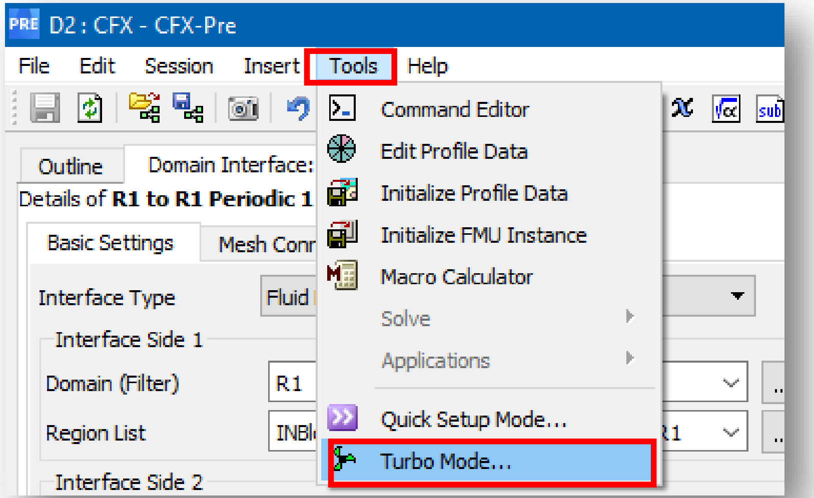


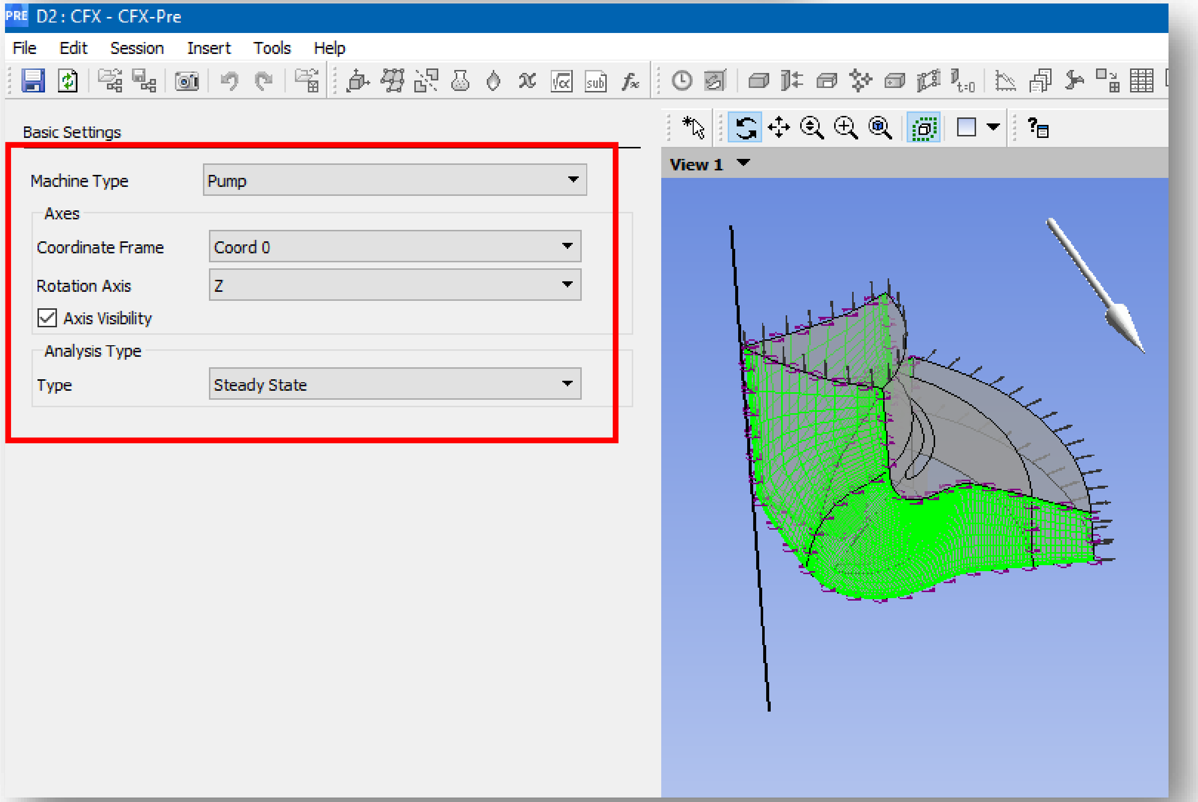
- Note that for unstable or mildly unsteady flows, steady-state solution monitors may oscillate
  - Consider running the case further as transient

CFX-Pre Turbo Mode

- Wizard for easy setup of any rotating machinery case
  - Walks you through the various settings
  - Automated setup when mesh was generated in TurboGrid
    - ➢ Domain
    - ➢ Interfaces
    - ➢ Boundary conditions
    - ➢ Solver settings

Complete workflow shown in workshop 1





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

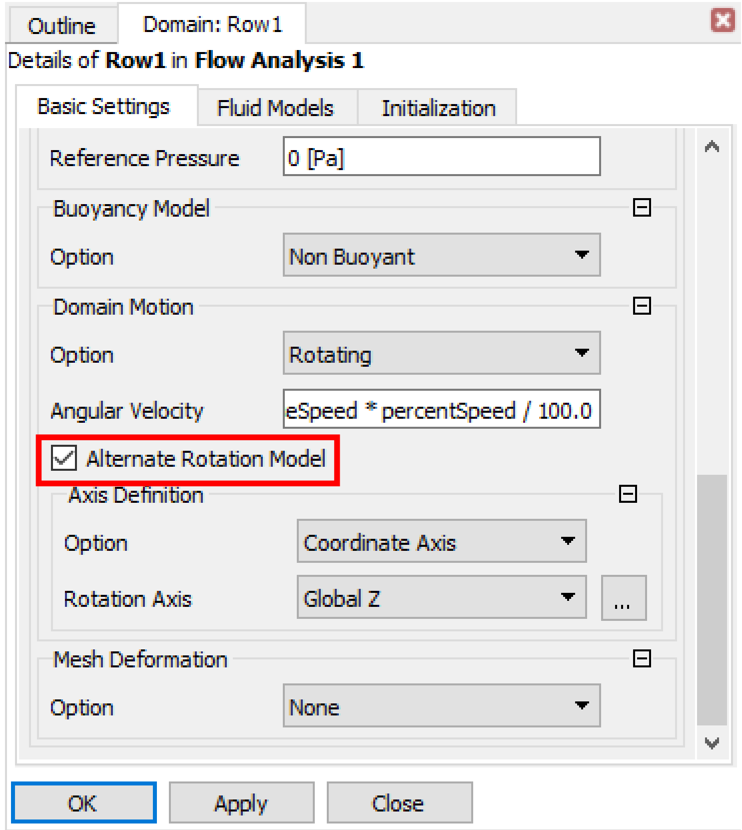
- Mesh is coarse, particularly in the circumferential direction

- Blade number is low, hence domain has large circumferential extent

False Swirl (3)

Solution:

- Use Alternate Rotation Model
  - modified discretization which minimizes discretization errors for this case
- Do a Multiple Frame of 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



Best Practices for Turbomachinery

- Online documentation
  CFX Reference Guide Chapter 11
  CFX Best Practices Guide for Turbomachinery

Chapter 11: CFX Best Practices Guide for Turbomachinery

Turbomachinery applications can generally be divided into three main categories: gas compressors and turbines, liquid pumps and turbines, and fans and blowers. Each category is discussed in a separate section below.

This guide describes best practices for setting up simulations involving:

- Gas Compressors and Turbines
- Liquid Pumps and Turbines
- Fans and Blowers
- Frame Change Models
- Domain Interface Setup
- Transient Blade Row

This guide is part of a series that provides advice for using CFX in specific engineering application areas. It is aimed at users with moderate or little experience using CFX for applications involving turbomachinery.

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