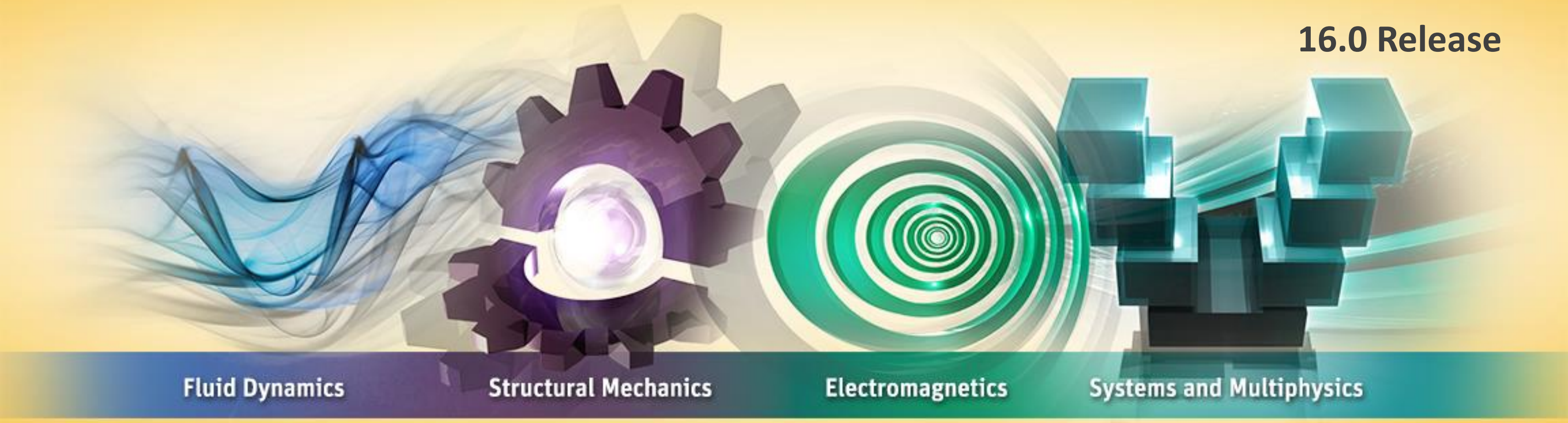
ANSYS

Lecture 8:
Domain Interfaces & Moving Zones

16.0 Release



Introduction to ANSYS CFX

Introduction

- **Lecture Themes:**
  - Domain Interfaces are important features that connect dissimilar domains or dissimilar meshes
  - Many CFD applications across industries involve systems or devices with moving parts. CFX offers many different models for rotating machinery, for arbitrary prescribed motion and for objects whose motion is determined by the flow.
- **Learning Objectives:**
  - You will be able to use domain interfaces and will become familiar with the CFX models for systems with moving parts and when a particular model is applicable.

ANSYS

Lecture 8 - 1:
Domain Interfaces

16.0 Release

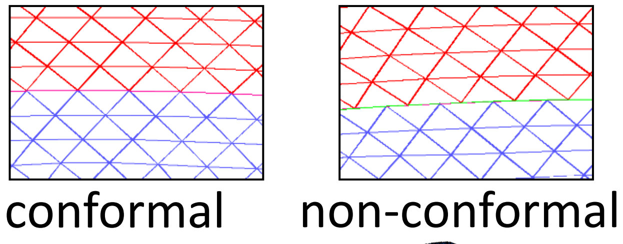
Fluid Dynamics Structural Mechanics Electromagnetics Systems and Multiphysics

Introduction to ANSYS CFX

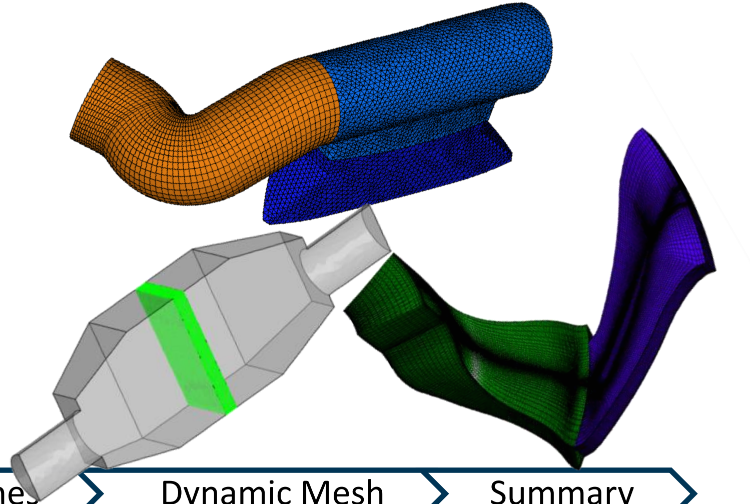
Realize Your Product Promise®

Overview

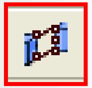
- Where two 3D mesh regions meet :
  - Two elements share one face (conformal) – no interpolation necessary → 1:1 mesh interface
  - Every element has its own face (non-conformal) – interpolation of fluxes is necessary → Generalized Grid Interface (GGI)

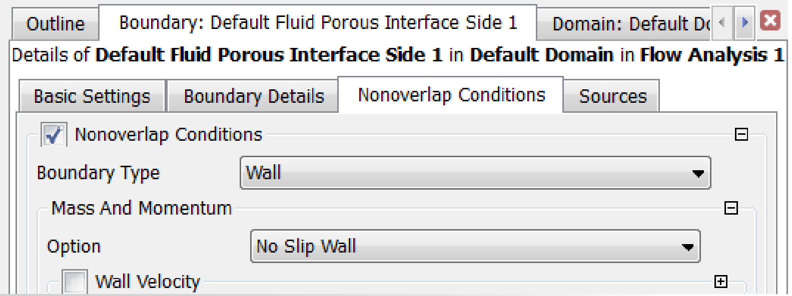


- Domain Interfaces are used for:
  - Connection of unmatched meshes
  - Changes in frames of reference
  - Connect different types of domain, e.g. Fluid & Solid
  - Periodic regions within a domain



Domain Interfaces and Boundary Objects

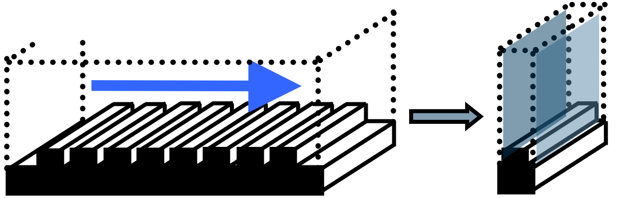
- Create a domain interface by right-clicking on ‘Flow Analysis’ or use the toolbar icon 
- 3 new objects in outline tree
  - Interface object → edit to modify the domain interface
  - Side 1 & Side 2 boundary condition →optionally specify sources, set wall boundary conditions for non-overlap regions

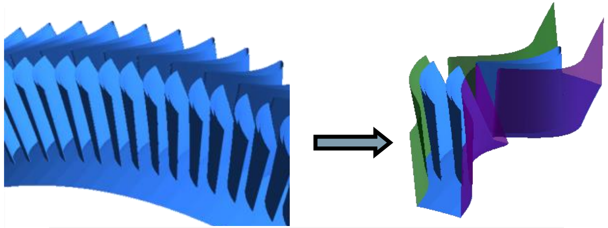


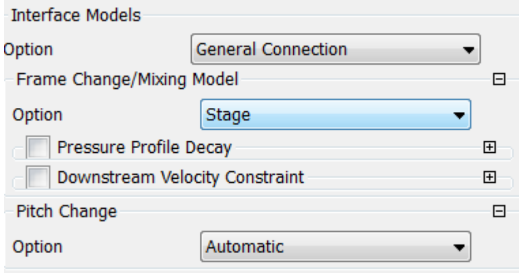
Interface Models

Available Interface Models are:

- Translational Periodicity
  - Allows for either the mass flow rate or the pressure change across the interface to be specified
- Rotational Periodicity
- General Connection
  - For all other types of connections
  - A Frame Change/Mixing Model and a Pitch Change can be set where there are multiple frames of reference. These are discussed in the Moving Zones section

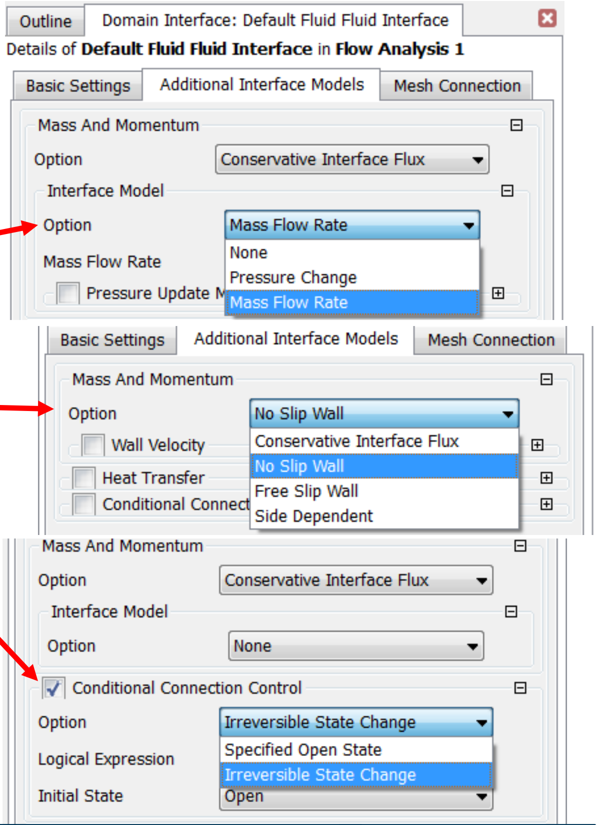






Additional Interface Models

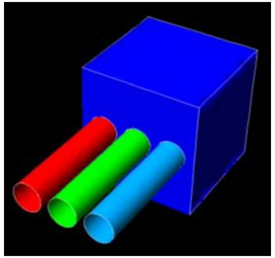
- Represent source or sink of momentum, e.g. fan or porous wall, by specifying a pressure change or a mass flow rate between side 1 and side 2
- When Interface Model = General Connection
  - Set up infinitely thin walls
  - Create a switch, using Conditional Connection Control, to determine whether the interface is open to flow or closed, i.e. a wall
    - Define when interface is open or its state changes by providing a logical expression, e.g. a function of time or pressure difference

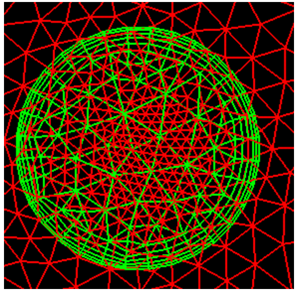


Introduction | Domain Interfaces | Rotating Zones | Dynamic Mesh | Summary

Mesh Connection Method

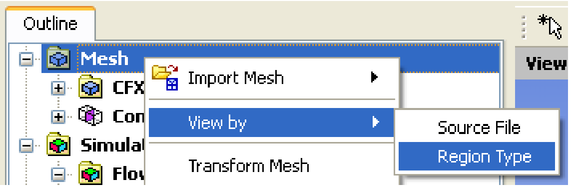
- **1:1**
  - Not available for Fluid-Porous and Porous-Porous
  - Available for Fluid-Fluid and Solid-Solid only within a single domain
  - Not recommended for Fluid – Solid, Solid – Solid, Solid-Porous interfaces
- **GGI**
  - If nodes on the 2 sides not aligned they should have similar edge lengths
  - If the region for one side ≠ the other, the connection will be between the mutually overlapping surfaces
  - use more memory and CPU than 1:1 connections
- **Automatic**
  - tries to make a 1:1 connection if possible, otherwise GGI
  - GGI connection always used for Fluid-Solid, Solid-Solid and Solid-Porous

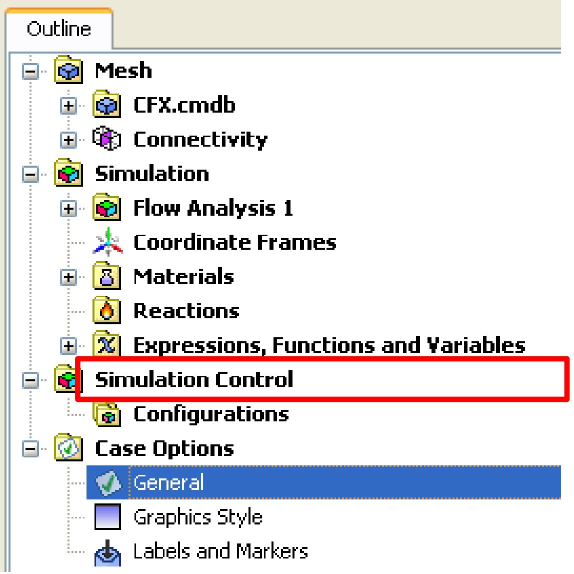




Automatic Domain Interfaces

- CFX-Pre automatically creates domain interfaces within a single mesh assembly
  - Connect multiple domains within the assembly
  - Connect non-matching meshes within the assembly
  - Right-click on Mesh > View by > Region Type to see a list of assemblies in the mesh
- Always check the automatic interfaces to make sure they are correct!
- Can disable automatic interface creation→ Case Options > General in the Outline tree
- Create interfaces manually between mesh

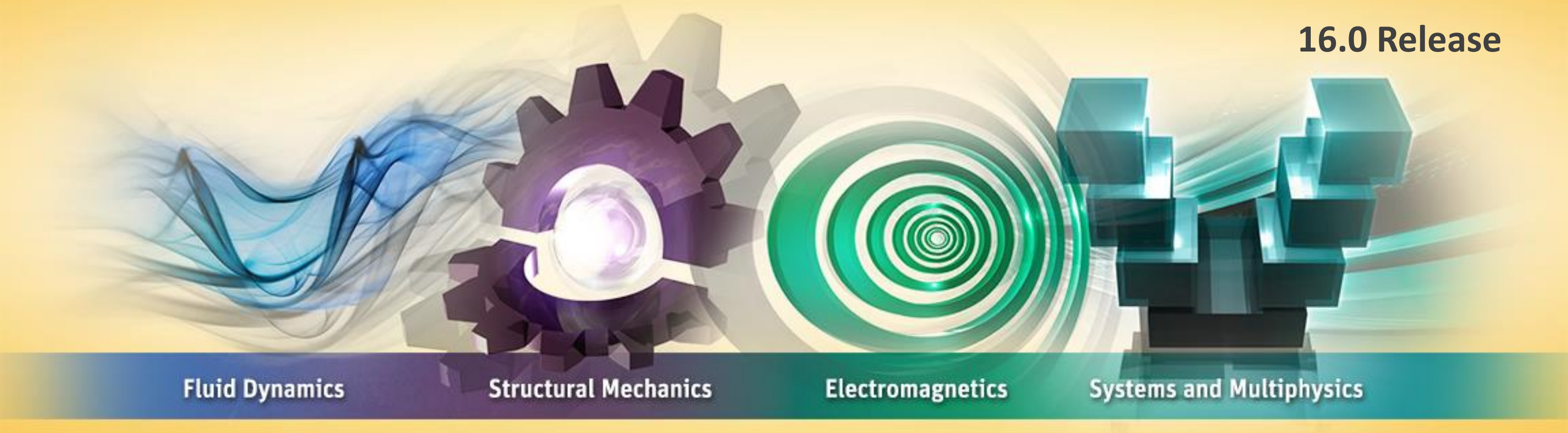




ANSYS

Lecture 8-2:
Moving Zones

16.0 Release

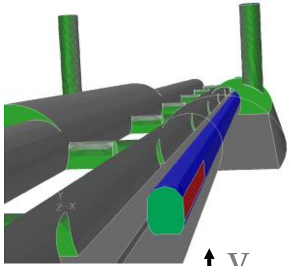


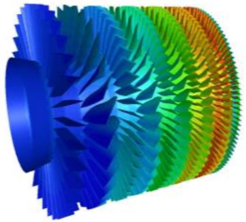
Introduction to ANSYS CFX

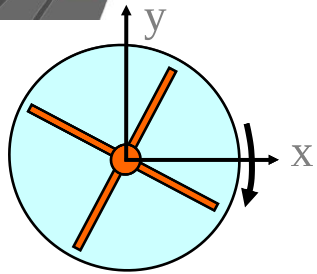
Realize Your Product Promise®

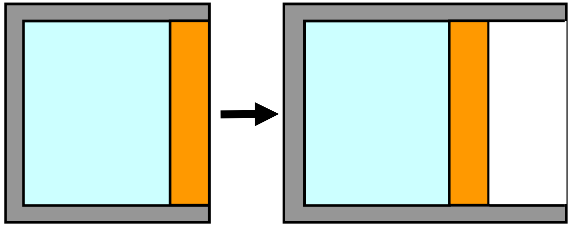
Introduction

- **Moving Frames of Reference**
  - Domain moves with coordinate system
  - Equations are modified to account for moving frame
  - Rotation or translation (translation in solids only)
  - To follow the motion of the body, the topology of the mesh does not need to be updated
- **Mesh Motion /Mesh Deformation**
  - Domain position and shape are tracked with respect to a stationary reference frame
  - Solutions are unsteady or approach equilibrium
  - Domain can change shape as a function of time, topology of the mesh might need to be updated



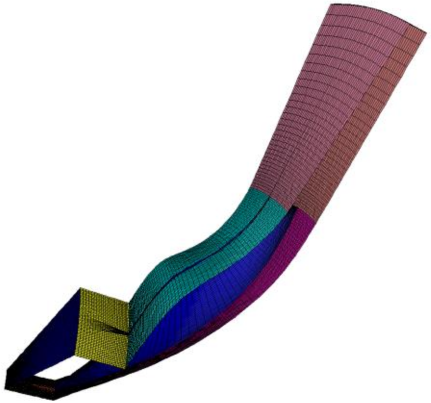






Rotating Equipment – Moving Frames of Reference

- **Why use a rotating frame of reference?**
  - Unsteady flow field when viewed in a stationary frame can become steady when viewed in a rotating frame →easy to solve
  - Additional acceleration terms are added to the momentum equations
- **Can employ rotational-periodic boundaries for efficiency (reduced domain size)**
- **Limitation:**
  - Existing unsteadiness in the rotating frame due to turbulence, circumferentially non-uniform variations in flow, separation, etc.
    - Example: vortex shedding from fan blade trailing edge

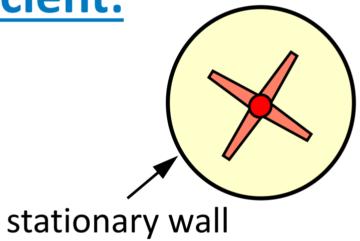


Centrifugal Compressor (single blade passage)

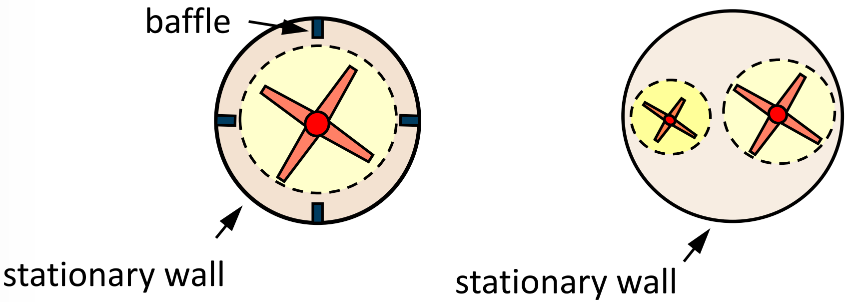
Single vs. Multiple Frames of Reference

- Multiple Frames of Reference required if:
  - Stationary walls ≠ surfaces of revolution
  - Multiple rotating domains rotating
    - at different rates
    - about different axes
- Interface between rotating and stationary zone must be a surface of revolution with respect to the axis of rotation
- Interfaces are always treated as GGIs

SFR is sufficient:

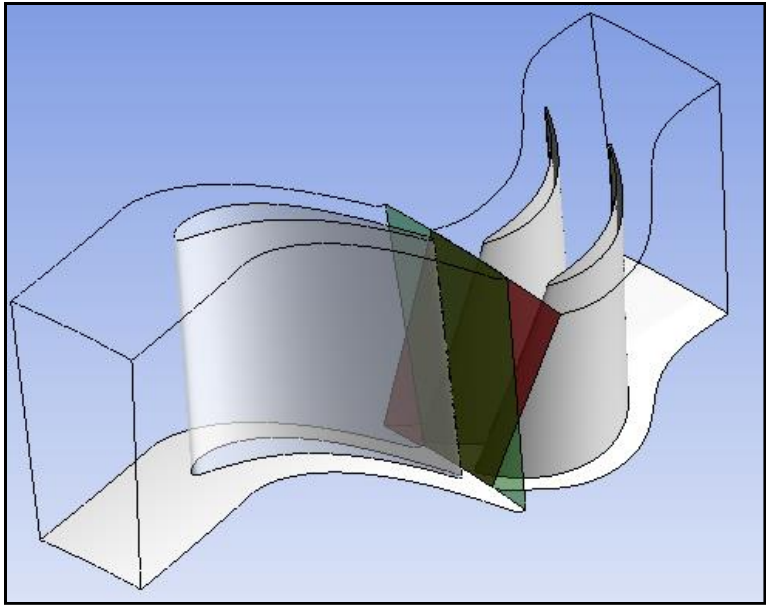


MFR is necessary:



Pitch Change

- Where there is rotational symmetry, rotational periodicity reduces the size of model, compared with full 360°
- Pitch change accounts for differences in rotational symmetry between domains in an MFR model
- Pitch Change is best set by
  - Specified Pitch Angles = specify Pitch angle on both sides
- The two sides do not need to line up physically



Steady State Modeling Approaches

- Single Reference Frame (SFR)
  - Domain is referred to a single moving reference frame
- Frozen Rotor (MFR)
  - Fixed relative positions
  - Fluxes scaled by pitch change → pitch change should be close to 1 for accuracy
  - Transient Interaction effects are ignored
- Stage (MFR)
  - Circumferential averaging of fluxes in bands at the domain interface
  - Incurs one-time mixing loss – suitable when relative motion at interface is large enough to mix out any upstream velocity profile
  - Accounts for time-averaged interaction effects

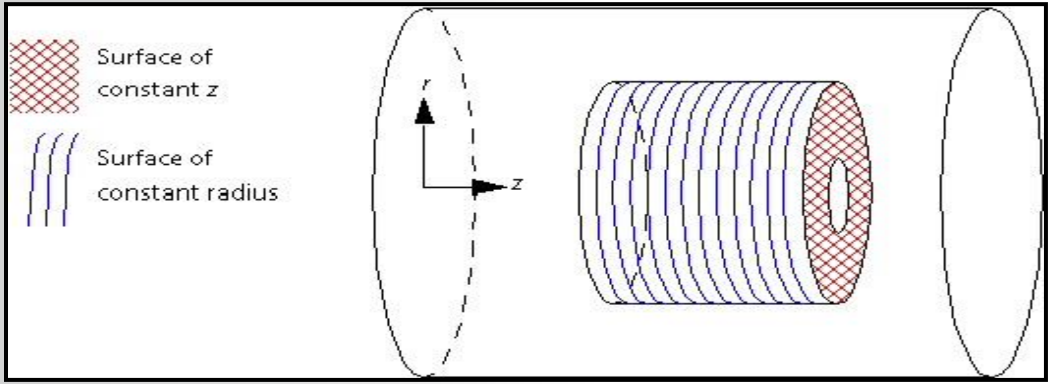
Unsteady Modeling Approaches

- **Transient rotor-stator (MFR)**
  – Predicts true transient interactions between a stator and rotor passage
  – Computationally expensive
  – Pitch change should be exactly 1
- **Transient Blade Row (TBR)**
  – Removes problem of unequal pitch → possible to model just one or two passages per row rather than the full 360. Methods are:
    - Profile Transformation
    - Time Transformation
    - Fourier Transformation

Interfaces

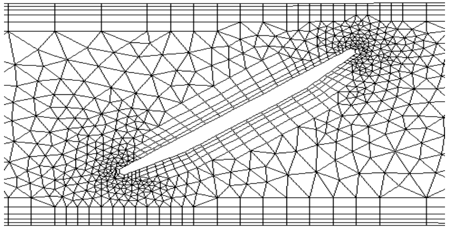
An intersection algorithm is used to find the overlapping parts of each mesh face at the interface

- **Interfaces that contain surfaces of both constant r and z have to be split.**

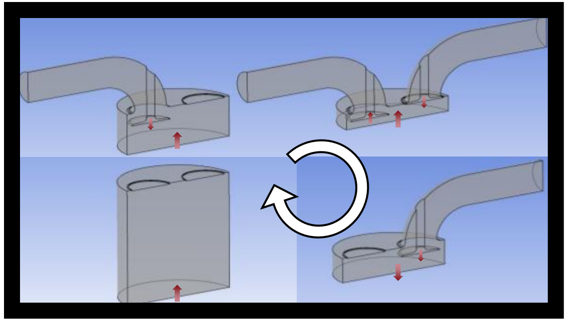


Mesh Deformation

- **Mesh Deformation: moving boundaries or objects**
- **Nodal displacements are calculated, mesh adjusts to accommodate them**
- **Boundary / object motion can be moved based on:**
  - **Specified displacement via Expressions**
  - **Coupled motion: two-way Fluid Structure Interaction**
  - **Sequential loading of pre-defined meshes**



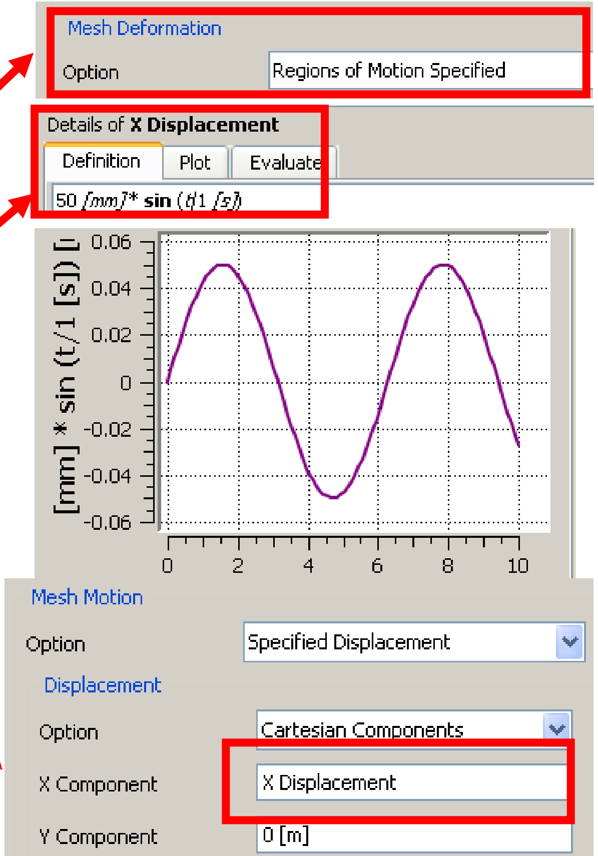
- **Examples:**
  - **Piston moving inside a cylinder**
  - **Flap moving on an airplane wing**
  - **Valve opening and closing**



- **Tutorials 22 & 23**

Mesh Deformation

- **Describing Mesh Motion:**
  - **Basic Domain Settings : "Mesh Deformation" to "Regions of Motion Specified"**
  - **Create expressions for nodal displacements or physical nodal locations**
  - **Apply Mesh Motion equations at boundaries/subdomains that are moving**

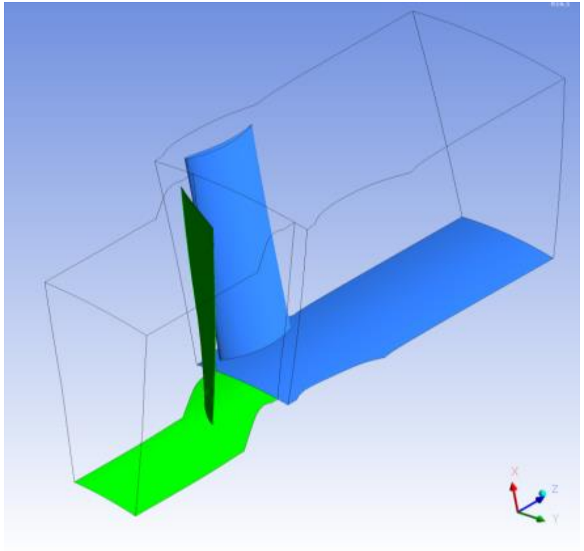


Summary

- Five approaches for moving zones
  - Single (Rotating) Reference Frame Model (SFR)
  - Multiple Frames of Reference (MFR)
    - Frozen Rotor
    - Stage
    - Transient Rotor Stator
  - Mesh Motion
- SFR, Frozen Rotor and Stage methods are steady-state approaches; Transient Rotor Stator is unsteady
- All physical models are compatible with moving reference frames or moving meshes (e.g. multiphase, combustion, etc.)

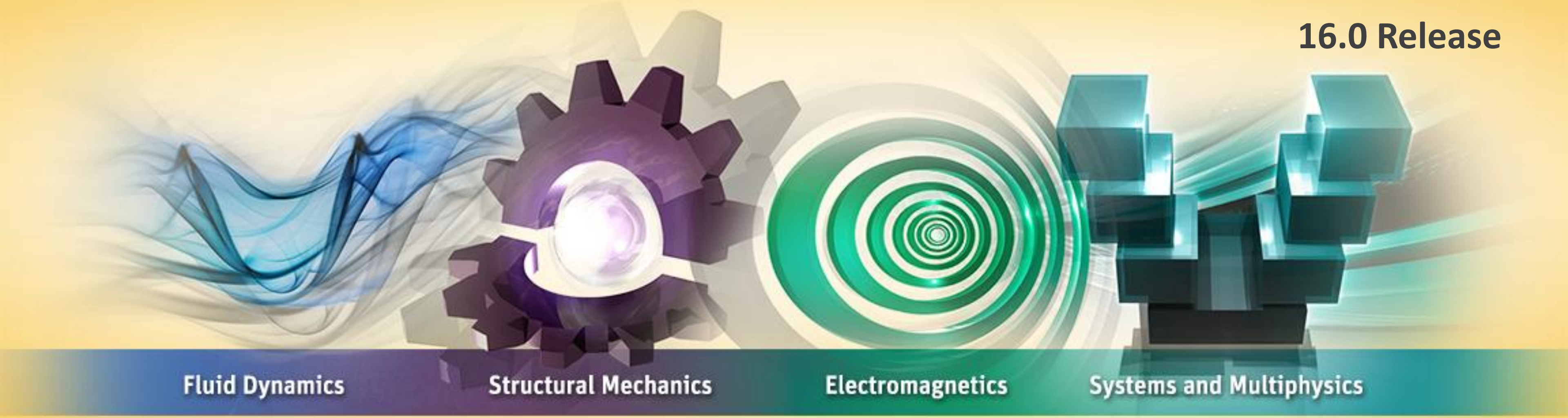
Workshop

- **Workshop 04 – Axial Fan Stage (MFR)**
  - Running ANSYS CFX outside ANSYS Workbench
  - Rotating domains
  - Basic setup using multiple frames of reference
  - Turbo-specific post-processing



Appendix: Fluid/Porous Interface

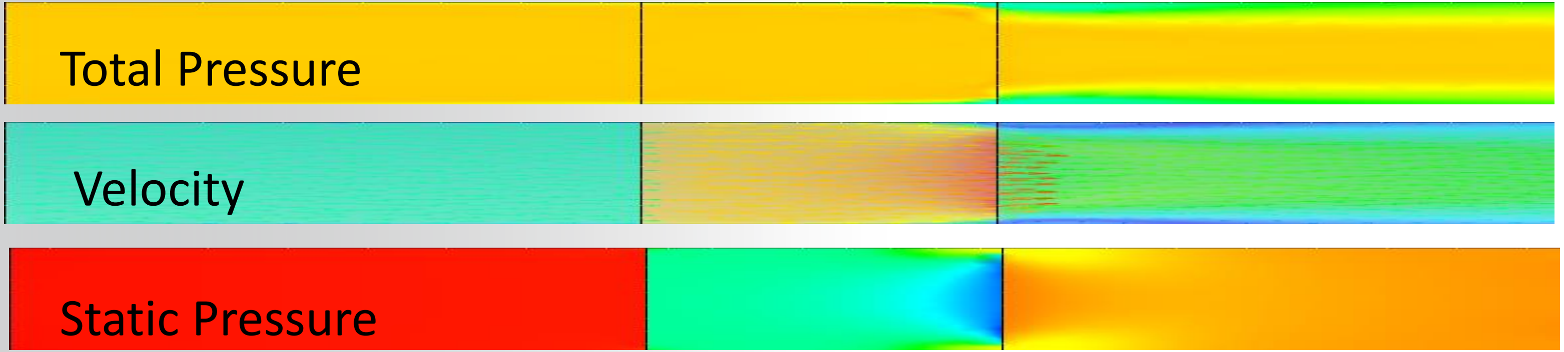
16.0 Release



Introduction to ANSYS CFX

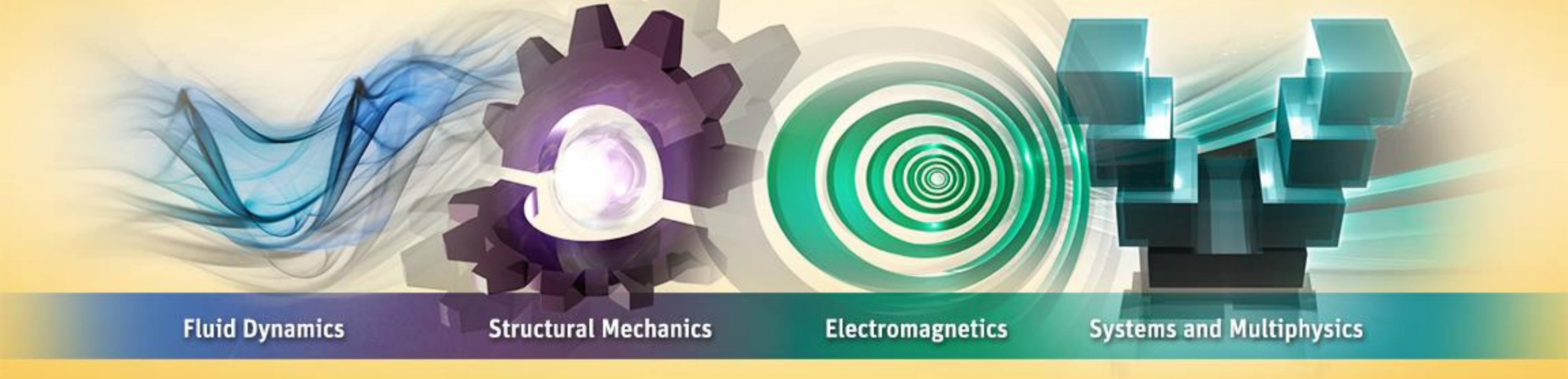
Porous Interface Usage

- Domain interfaces involving porous domains are always treated as GGI
- Total Pressure is unchanged across the interface
  - Static pressure will show a discontinuity at the interface
- Total Enthalpy (Total Energy) is unchanged across interface
  - May see a discontinuity in Enthalpy (Temperature) in high speed flows



Appendix: Multiple Frames of Reference (MFR)

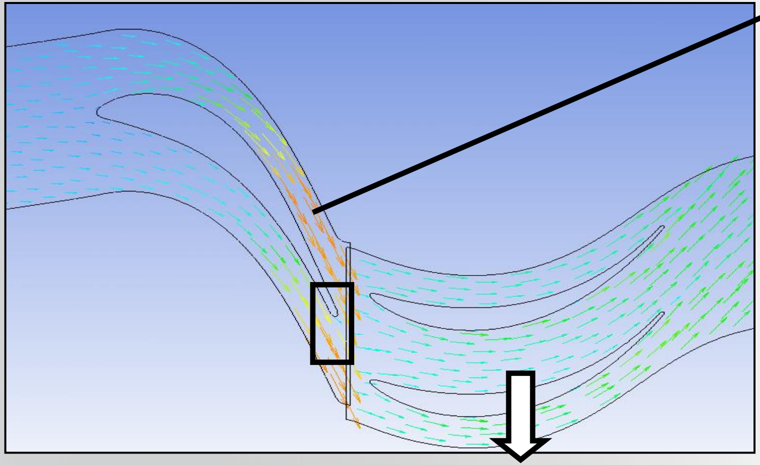
16.0 Release



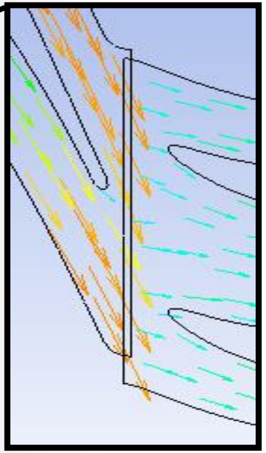
Introduction to ANSYS CFX

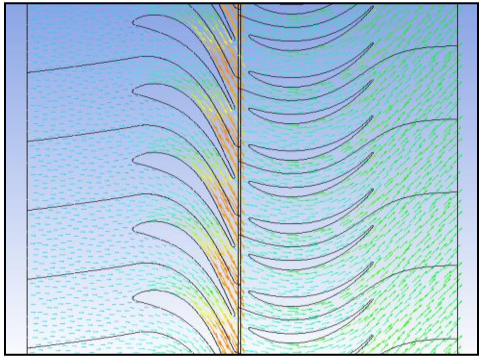
Frozen Rotor Frame Change Model

- **Frozen Rotor: Components have a fixed relative position, but the appropriate frame transformation and pitch change is made**
  - Steady-state solution



rotation

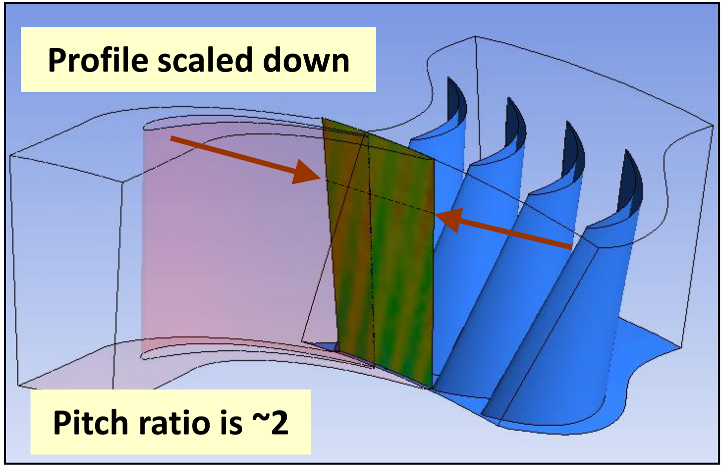




Pitch change is accounted for

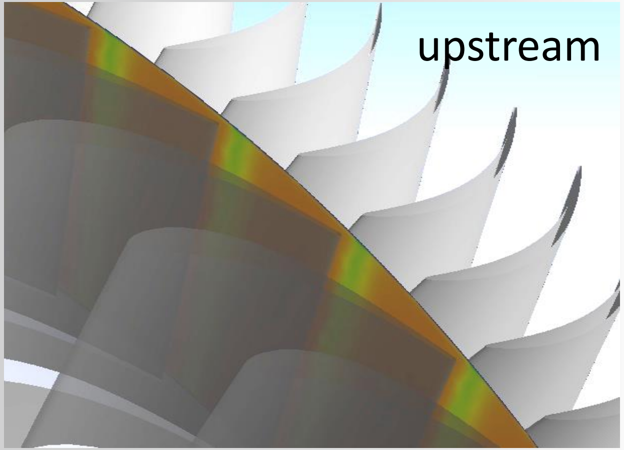
Frozen Rotor Frame Change Model

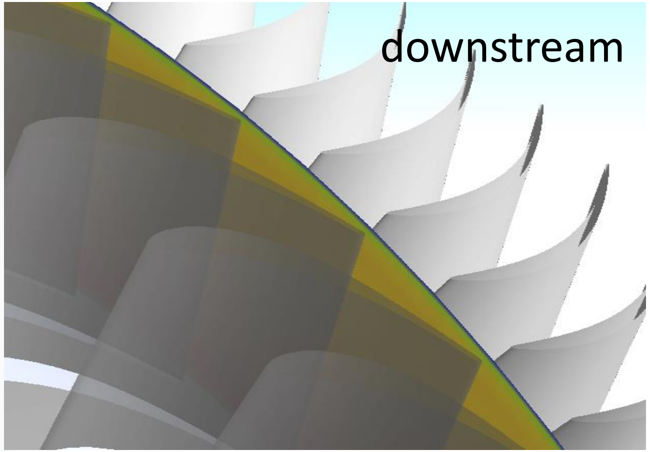
- **Frozen Rotor Usage:**
  - The quasi-steady approximation involved becomes small when the through flow speed is large relative to the machine speed at the interface
  - This frame change model requires the least amount of computational effort
  - Transient effects at the frame change interface are not modeled
  - Pitch ratio should be close to one to minimize scaling of profiles
    - A discussion on pitch change will follow



Stage Frame Change Model

- **Stage: Performs a circumferential averaging of the fluxes through bands on the interface**
  - Accounts for time-averaged interactions, but not transient interactions
  - Steady-state solution





Stage Frame Change Model

- **Stage Usage:**
  - Allows steady-state predictions to be obtained for multi-stage machines
  - Incurs a one-time mixing loss equivalent to assuming that the physical mixing supplied by the relative motion between components is sufficiently large to cause any upstream velocity profile to mix out prior to entering the downstream machine component
  - The Stage model requires more computational effort than the Frozen Rotor model to solve but not as much as the Transient Rotor-Stator model

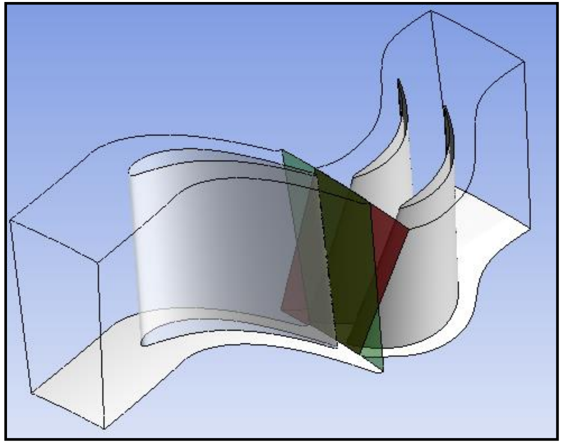
Transient Rotor Stator Frame Change Model

- **Transient Rotor-Stator:**
  - Predicts the true transient interaction of the flow between a stator and rotor passage
  - The transient relative motion between the components on each side of the interface is simulated
  - The principle disadvantage of this method is that the computer resources required may be large
  - Initialise with converged solution from a Frozen Rotor or Stage model to minimise the number of timesteps needed to establish the flow

ANSYS

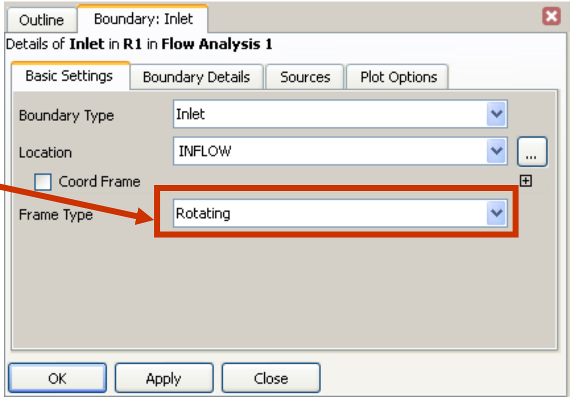
Pitch Change

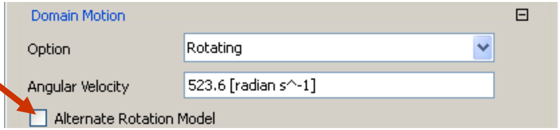
- **When the full 360º of rotating and stationary components are modeled the two sides of the interface completely overlap**
  - However, often rotational periodicity is used to reduce the problem size
- **Example: A rotating component has 113 blades and is connected to a downstream stator containing 60 vanes**
  - Using rotationally periodicity, a single rotor and stator blade could be modeled
    - This would lead to pitch change at the interface of (360/113): (360/60) = 0.53:1
  - Alternatively two rotor blades could be modeled with a single stator vane, as shown to the right, giving a pitch ratio of 2*(360/113):(360/60) = 1.06:1



Setup Guidelines

- Boundary Conditions – are input quantities are relative to the Stationary frame, or the Rotating frame
- Walls which are tangential to the direction of rotation in a rotating frame of reference can be set to be stationary in the absolute frame of reference by assigning a Wall Velocity which is counter-rotating
- Use the Alternate Rotation Model if the bulk of the flow is axial (in the stationary frame) and would therefore have a high relative velocity when considered in the rotating frame
  - E.g. the flow approaching a fan will be generally axial, with little swirl
  - Used to avoid “false swirl”

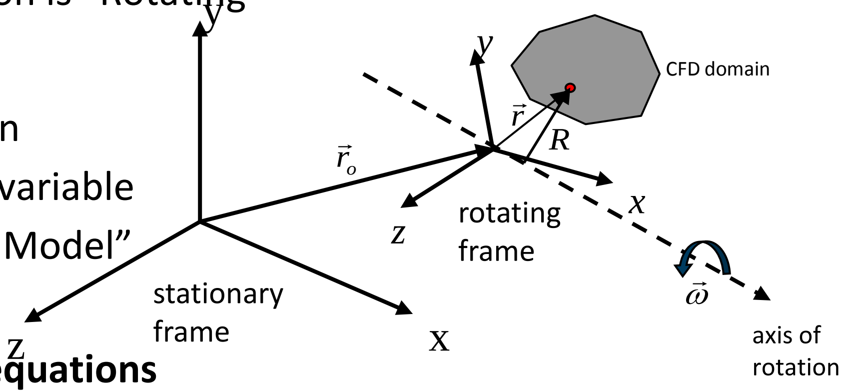




Navier-Stokes Equations: Rotating Reference Frames

Equations can be solved in absolute or rotating (relative) reference frame

- **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
  - Default solution method when Domain Motion is "Rotating"
- **Absolute Velocity Formulation**
  - Derived from the relative velocity formulation
  - Uses the *absolute velocity* as the dependent variable
  - Can be used by enabling "Alternate Rotation Model" in CFX-Pre setup
- **Rotational source terms appear in momentum equations**



The Velocity Triangle

The relationship between the absolute and relative velocities is given by

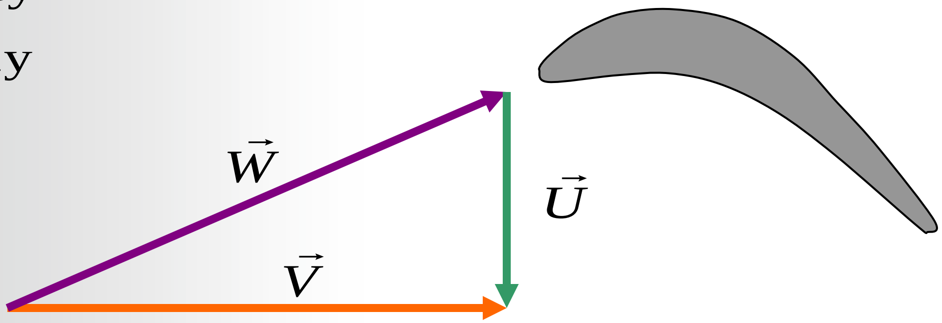
$$\vec{V} = \vec{W} + \vec{U}$$

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

$$\vec{U} \equiv \varpi \times \vec{r}$$

$$\vec{V} = \text{Absolute Velocity}$$

$$\vec{W} = \text{Relative Velocity}$$



Comparison of Formulations

- Relative Velocity Formulation: x-momentum equation

$$\frac{\partial \rho w_x}{\partial t} + \nabla \cdot \rho \vec{W} w_x = -\frac{\partial p}{\partial x} + \nabla \cdot \vec{\tau}_{vrx} - \rho\left(2\vec{\omega}\times\vec{W} + \vec{\omega}\times\vec{\omega}\times\vec{r}\right)\cdot \hat{\imath}$$

Coriolis acceleration   Centripetal acceleration

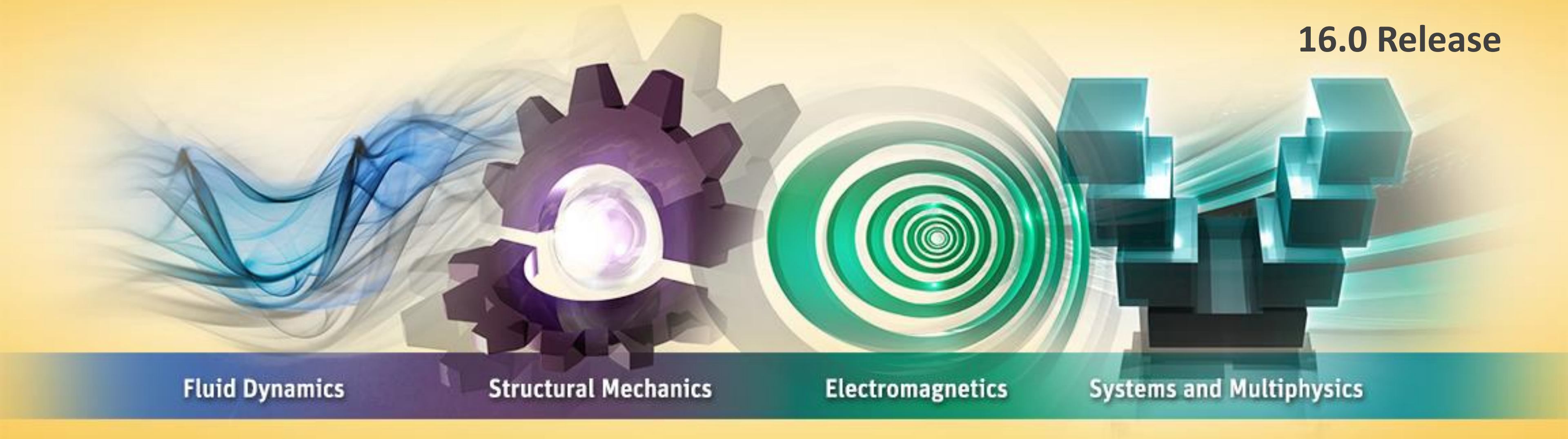
- Absolute Velocity Formulation: x-momentum equation

$$\frac{\partial \rho v_x}{\partial t} + \nabla \cdot \rho \vec{W} v_x = -\frac{\partial p}{\partial x} + \nabla \cdot \vec{\tau}_{vx} - \rho\left(\vec{\omega}\times\vec{V}\right)\cdot \hat{\imath}$$

Coriolis + Centripetal accelerations

Appendix: Moving Solid Zones

16.0 Release



Introduction to ANSYS CFX

Realize Your Product Promise®

Motion in Solid Domains

In most simulations it is <u>not</u> necessary to specify any motion in solid domains

Consider a rotating blade simulation in which the blade is included as a solid domain and heat transfer is solved through the blade

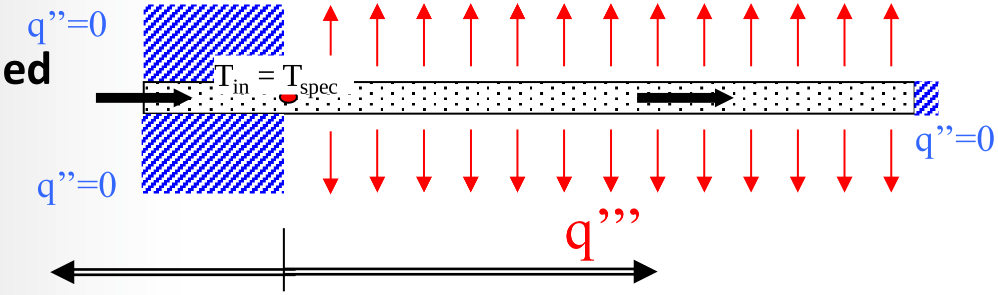
- Even though the fluid domain is solved in a rotating frame of reference, the mesh is not actually rotated in the solver. Therefore it will always line up with the solid
- The solid domain does not need to be placed in a rotating frame of reference since the heat transfer solution has no Coriolis or centripetal terms

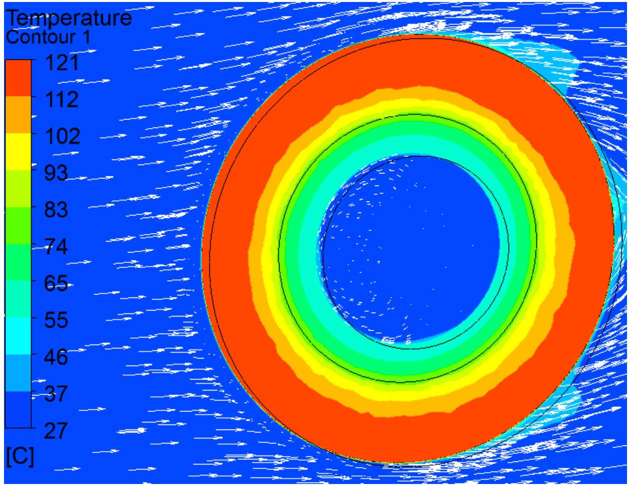
Solid domain motion should be used when the advection of energy needs to be considered

- For example, a hot jet impinging on a rotating disk. To prevent a hot spot from forming, the advection of energy in the solid needs to be included

Motion in Solid Domains

- **Solid Domain Motion can be classified into two areas:**
  - **Translational Motion**
    - For example, a process where a solid moves continuously in a linear direction while cooling
    - The solid must extend completely through the domain
  - **Rotational Motion**
    - For example, a brake rotor which is heated by brake pads





Motion in Solid Domains

In Solid Domains, the conservation of the energy equation can account for heat transport due to motion of the solid, conduction and volumetric heat sources

$$\frac{\partial(\rho h)}{\partial t} + \nabla \cdot (\rho U_S h) = \nabla \cdot (\lambda \nabla T) + S_E$$

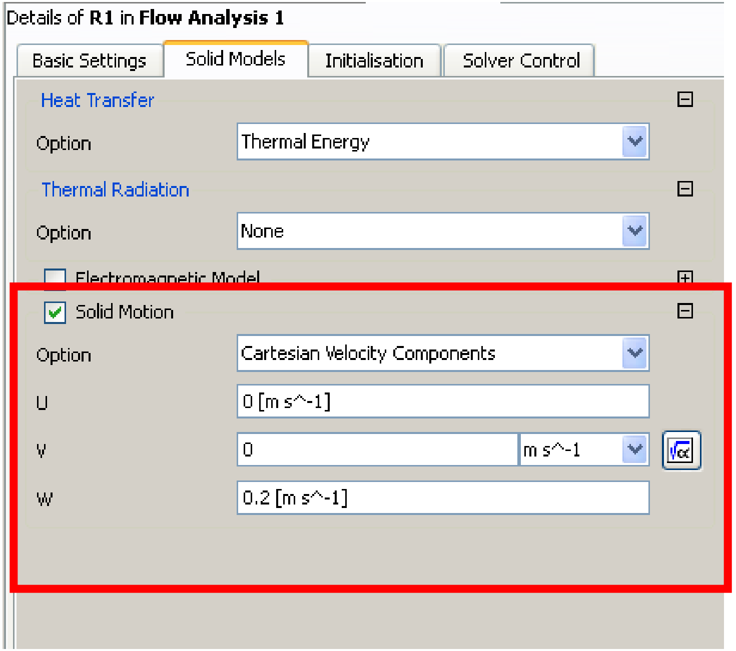
Solid Velocity

Note that the solid is never physically moved when using this approach, there is only an additional advection term added to the energy equation

Translating Solid Domains

Enable Solid Motion

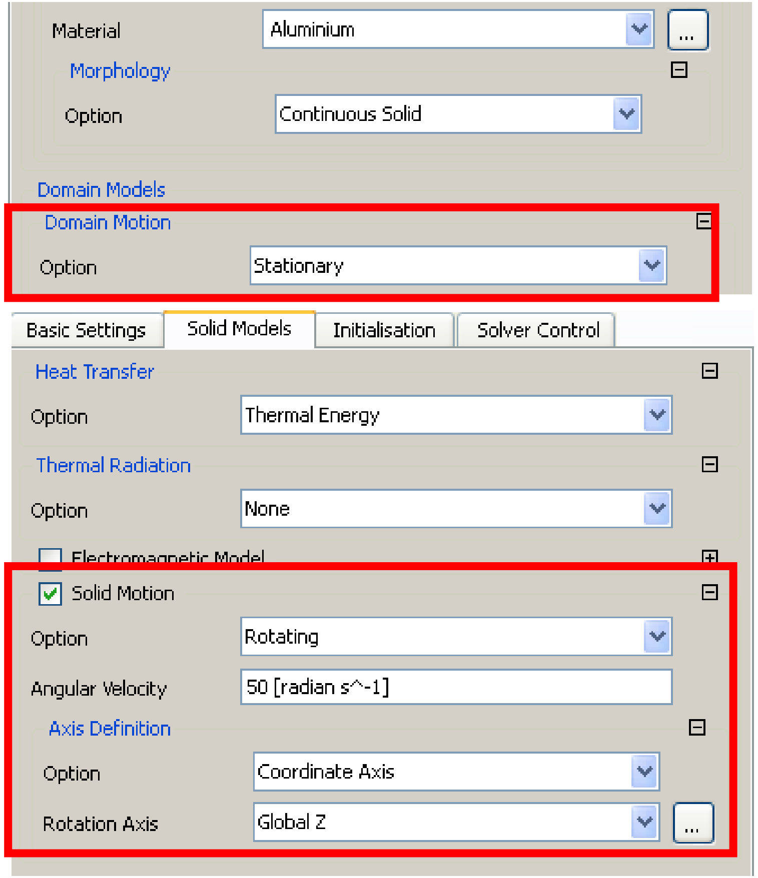
Specify Solid Motion in terms of Cartesian Velocity Components



Rotating Solid Domains

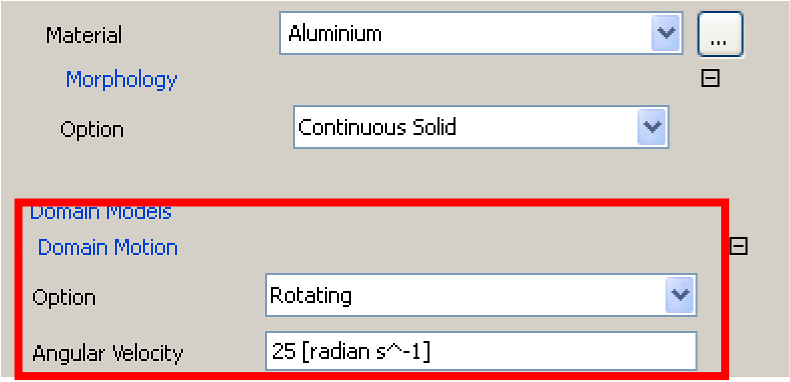
Rotating Solid Domains can be set up in one of two ways:

- **Solid Motion Method**
  - On Basic Settings tab, set the Domain Motion to Stationary
  - On the Solid Models tab, set Solid Motion to Rotating
  - The solid mesh does not physically rotate, but the model accounts for the rotational motion of the solid energy



Rotating Solid Domains

- **Rotating Domain Method**
  - Set the Domain Motion to “Rotating”
  - Do not set any Solid Motion
- To account for rotational motion of solid energy or a heat source which rotates with the solid, a transient simulation is required and a Transient Rotor Stator interface must be used



Note: “Solid Motion” as applied on the “Solid Models” tab is calculated relative to the Domain Motion, so is not normally used if “Domain Motion” is “Rotating”