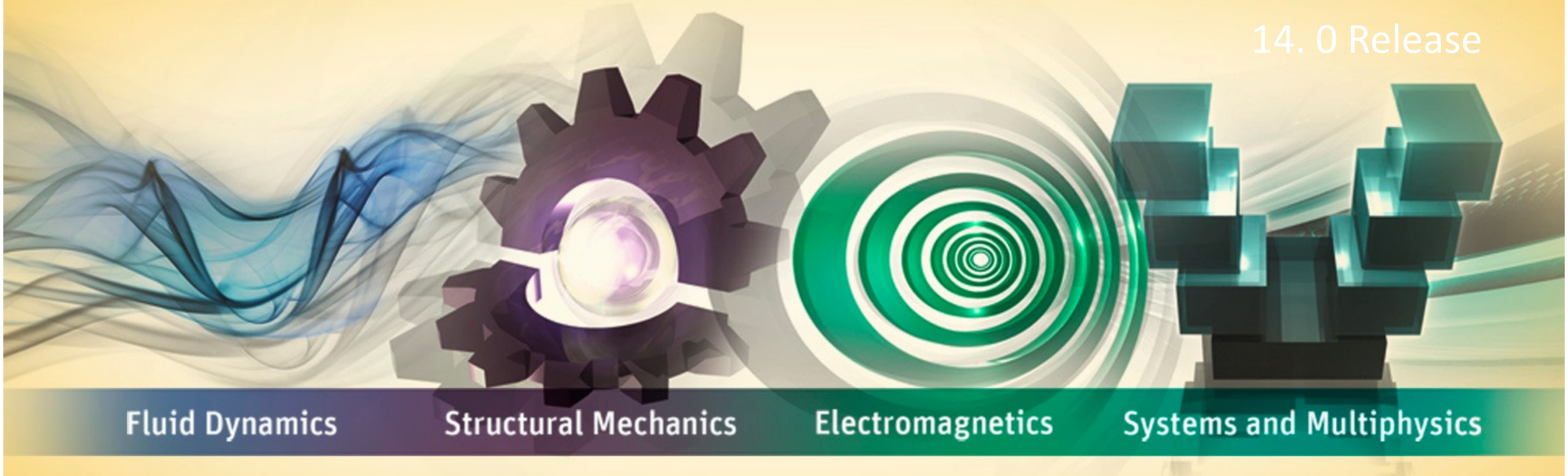
Lecture 8
Non-Conformal Interfaces & Moving Zones

14. 0 Release



Introduction to ANSYS FLUENT

Introduction

Lecture Theme:

Many CFD applications across industries involve systems or devices with moving parts. FLUENT offers many different models for rotating machinery, for arbitrary prescribed motion and for objects whose path is determined by the flow.

Learning Aims:

You will learn:

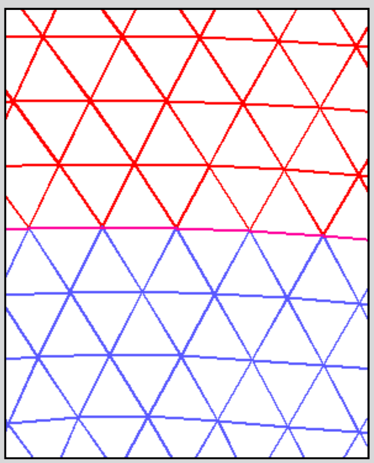
- How to define non-conformal interfaces and periodic boundary conditions
- The models available for rotating machinery such as the multiple reference frame and sliding mesh models
- The different dynamic meshing methods for arbitrary motion, including the coupled 6-DOF solver

Learning Objectives:

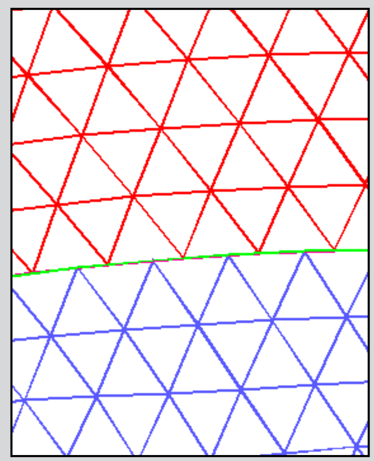
You will be able to use non-conformal meshes and will become familiar with FLUENT's models for systems with moving parts and when a particular model is applicable.

Overview

- **Across an interface between two cell zones, the nodes may or may not exactly align**
  - If the nodes match perfectly, this is a '**Conformal**' mesh
    - If using DesignModeler, combining bodies into a single part will give a conformal mesh
  - If the nodes do not match up, this is a '**Non Conformal**' mesh
    - FLUENT can interpolate across the interface, but this must be defined in the GUI.
    - If not, FLUENT will treat the interface as a wall, and no fluid can flow through.



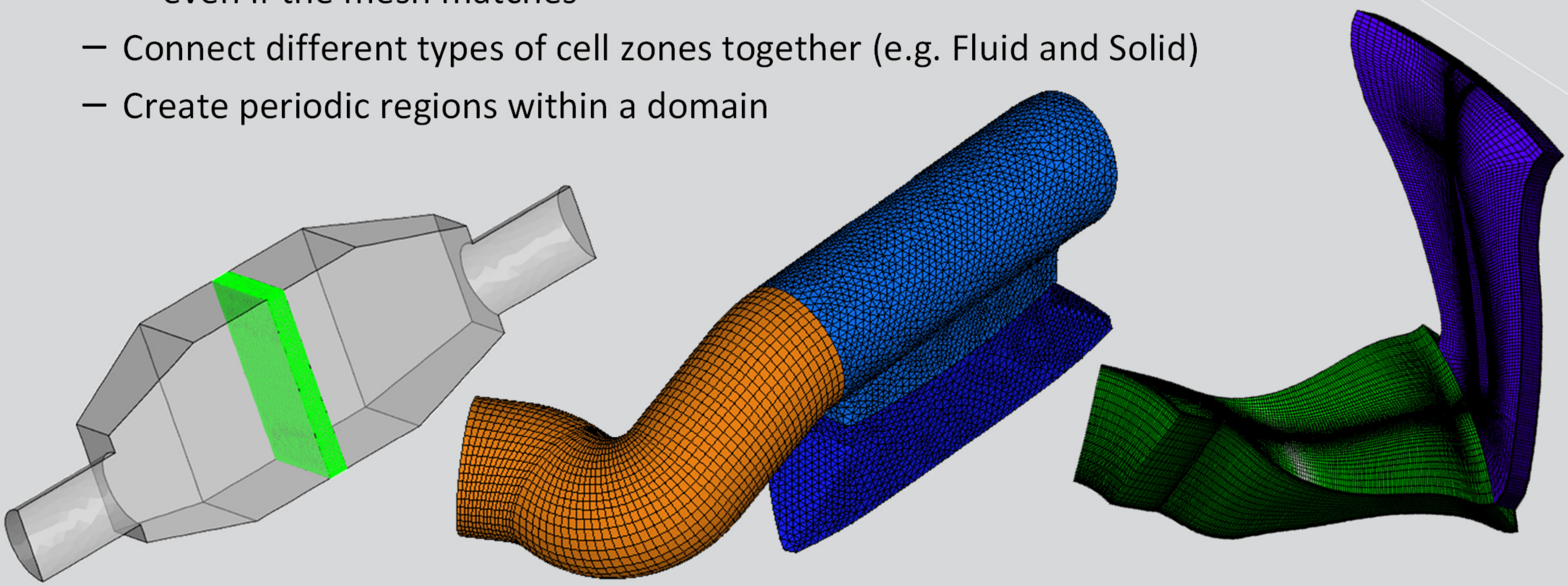
Conformal



Non-Conformal

Overview

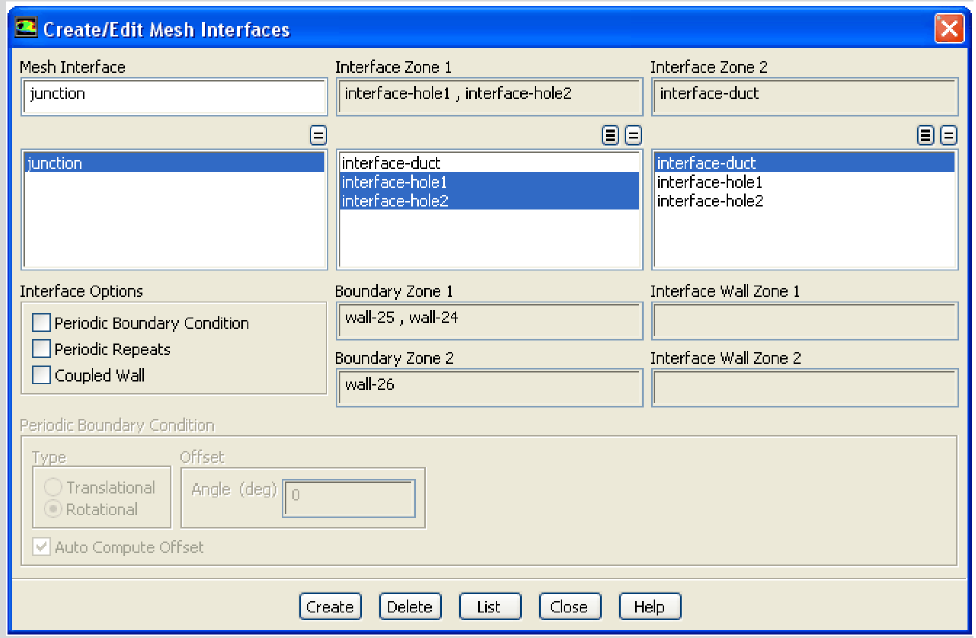
- **Non-conformal Interfaces can be used for:**
  - Connection of mismatched meshes (hex to tet for example)
    - a single mesh file may contain non-matching mesh regions and require non-conformal interfaces
  - Changes in reference frames between cell zones
    - even if the mesh matches
  - Connect different types of cell zones together (e.g. Fluid and Solid)
  - Create periodic regions within a domain



ANSYS

Inserting Non-conformal Interfaces

- **To create a non-conformal interface:**

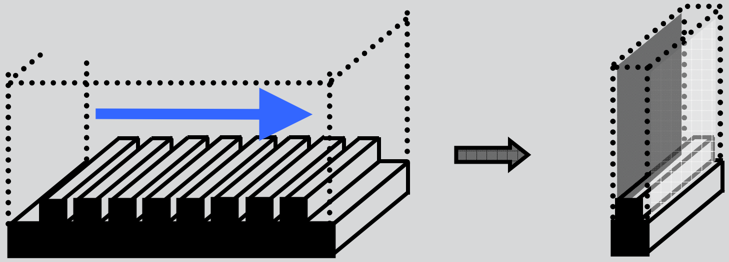


  - **Step 1: Define/Boundary Conditions**
    - Change the type of each pair of zones that comprises the non-conformal boundary to **interface**
  - **Step 2: Define/Mesh Interfaces**
    - Enter a name for the interface in the Mesh Interface text-entry box
    - Specify the two interface zones that comprise the mesh interface by selecting one or more zones in the Interface Zone 1 list and one or more zones in the Interface Zone 2 list
      - If one of your interface zones is much smaller than the other, you should specify the smaller zone as Interface Zone 1 to improve the accuracy of the intersection calculation
    - Enable the desired Interface Options if appropriate

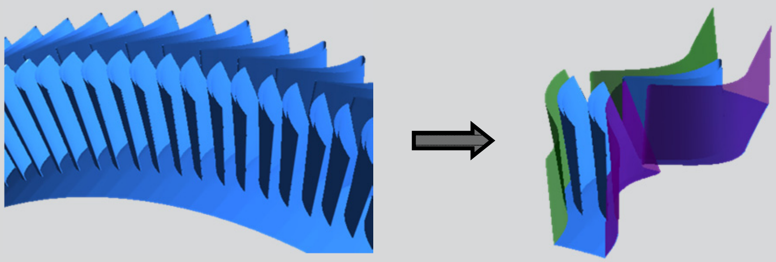
Periodic Boundary Condition

- **To create a Periodic boundary condition**
  - Enable the Periodic Boundary Condition option in the Mesh interfaces panel
  - Select either Translational or Rotational as the periodic boundary condition Type
    - Retain the enabled default setting of Auto Compute Offset if you want ANSYS FLUENT to automatically compute the offset
  - Mesh can be non-conformal

- **Translational Periodicity**
  - Simulates geometries that have translational periodicity
  - Allows for either the mass flow rate or the pressure change across the interface to be specified
  - The quantity not specified will be part of the solution

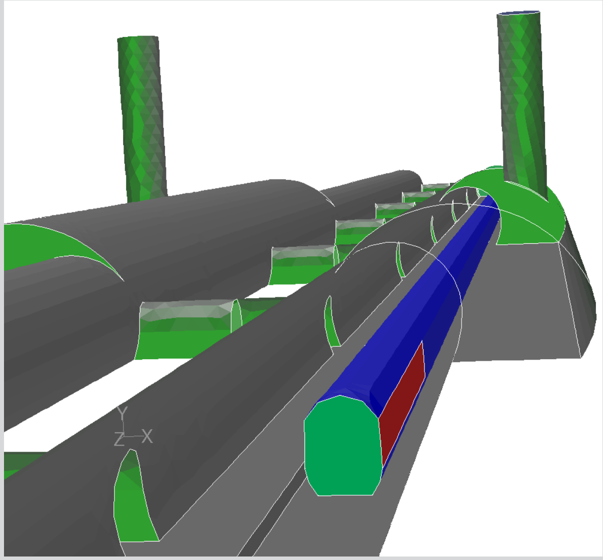


- **Rotational Periodicity**
  - Simulates rotationally periodic geometries
  - Before proceeding, you have to correctly enter the rotational axis for the corresponding cell zone in the BC panel

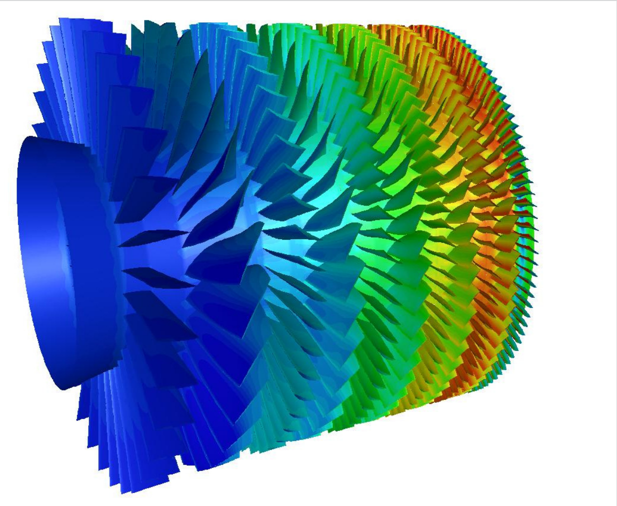


Introduction to Moving Zones

- **Many flow problems involve domains which contain translating or rotating components**

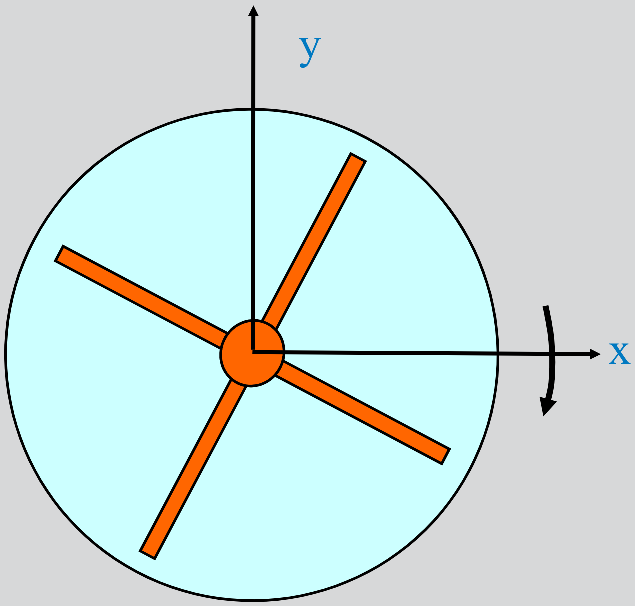


- **Two types of motion are possible: translational and rotational**
- **There are two basic modeling approaches for moving domains:**
  - [Moving Reference Frames]
    - Frame of reference is attached to the moving domain
    - Governing equations are modified to account for moving frame
  - [Moving / Deforming Domains]
    - Domain position and shape are tracked with respect to a stationary reference frame
    - Solutions are inherently unsteady

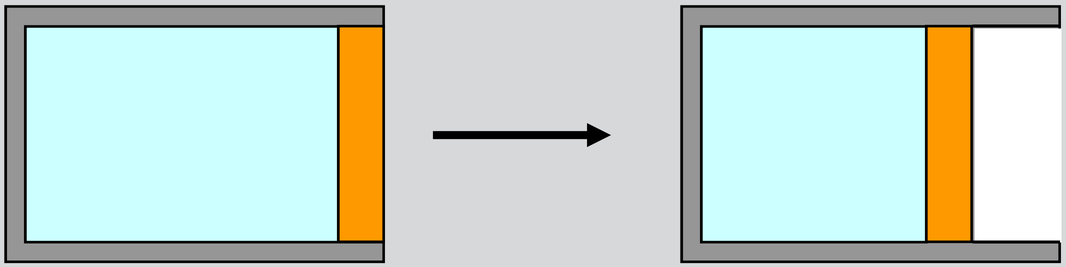


Moving Reference Frames vs. Mesh Motion

- **Moving Reference Frame**
  - Domain moves with coordinate system
  - To follow the motion of the body, topology of the mesh does not need to be updated
    - ➔ Rotation / Translation of the Moving domain



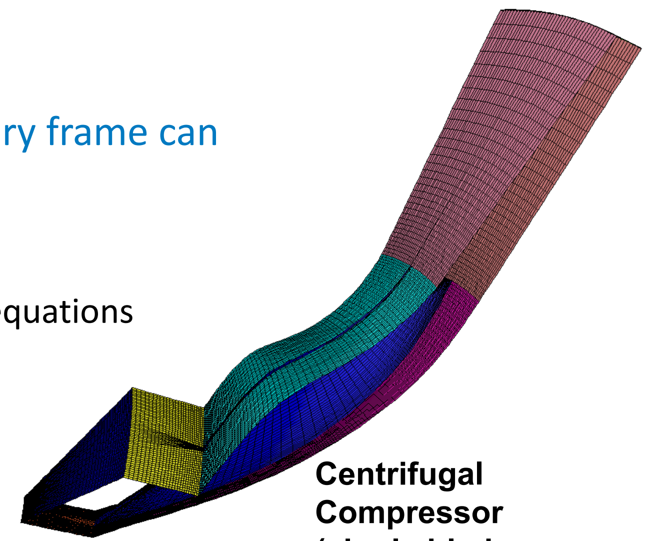
- **Mesh Motion**
  - Domain changes shape as a function of time
  - To follow the motion of the body, topology of the mesh need to be updated
    - ➔ Smoothing / Remeshing of the domain



Rotating Equipment

- **Why use a rotating reference frame?**
  - A flow field which is unsteady when viewed in a stationary frame can become steady when viewed in a rotating frame
  - Steady-state problems are easier to solve...

    ➔ Additional acceleration terms are added to the momentum equations
    - Simpler BCs
    - Low computational cost
    - Easier to post-process and analyze



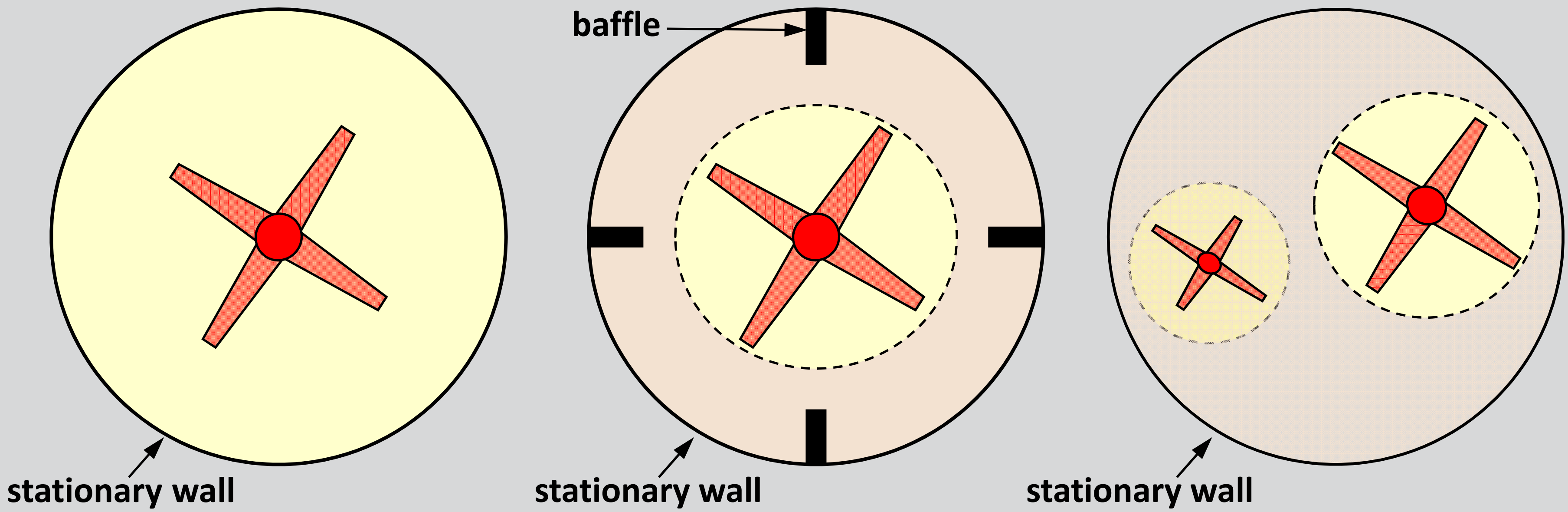
Centrifugal Compressor (single blade passage)

- **Limitation:**
  - You may still have unsteadiness in the rotating frame due to turbulence, circumferentially non-uniform variations in flow, separation, etc.
    - Example: vortex shedding from fan blade trailing edge

- Rotationally-periodic boundaries can be employed for efficiency (reduced domain size)

Single vs. Multiple Reference Frame Modeling

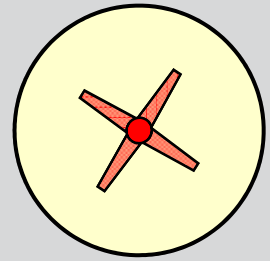
- **When domains rotate at different rates or when stationary walls do not form surfaces of revolution Multiple Reference Frames (MRF) are needed**

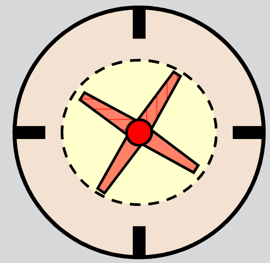


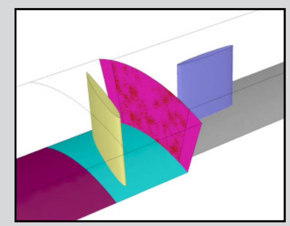
SRF is sufficient   MRF is necessary   MRF is necessary

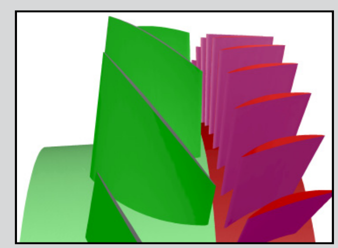
Different Approaches:

- **Overview of Modeling Approaches:**
  - **Single Reference Frame (SRF)**
    - Entire computational domain is referred to a moving reference frame
      → **steady-state**
  - **Multiple Reference Frame (MRF)**
    - Selected regions of the domain are referred to moving reference frames
    - Interaction effects are ignored
      → **steady-state**
  - **Mixing Plane (MPM)**
    - Influence of neighboring regions accounted for through use of a mixing plane model at rotating/stationary domain interfaces
    - Circumferential non-uniformities in the flow are ignored
      → **steady-state**
  - **Sliding Mesh (SMM)**
    - Motion of specific regions accounted for by a mesh motion algorithm
    - Flow variables interpolated across a sliding interface
    - → **Unsteady problem** - can capture all interaction effects with complete fidelity, but more computationally expensive than SRF, MRF, or MPM









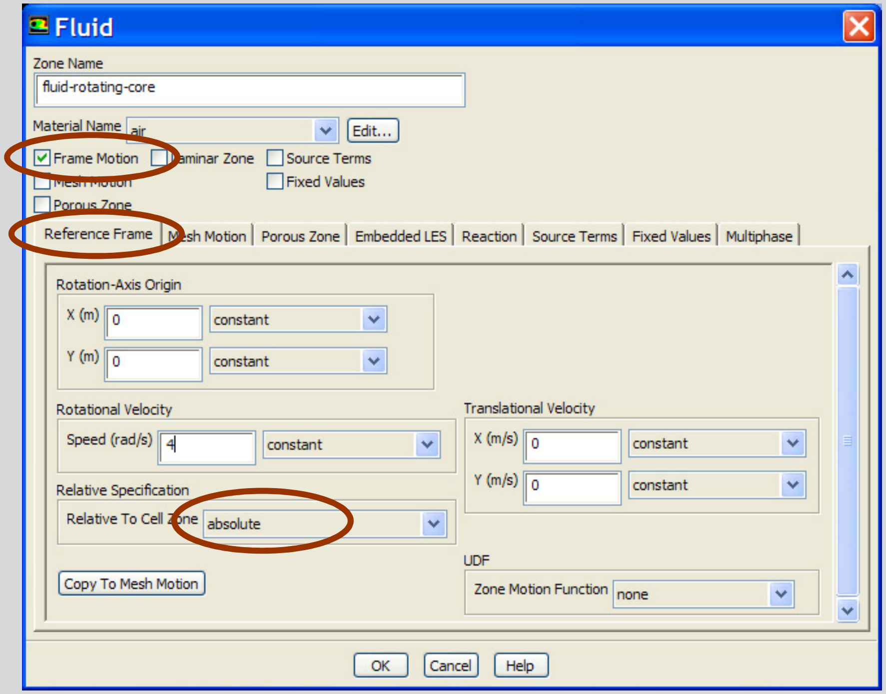
ANSYS

Defining a MRF Zone

- The simplest approach to set up and solve is using a **Moving Reference Frame**
  - Solution is steady state
  - Mesh never actually moves, local accelerations applied to each grid cell
  - This is applicable if there is a 'steady-state' solution to the problem, so:
    - Exact relative positions of moving and stationary (rotor / stator) parts does not matter
    - No vortex shedding or other transient phenomena

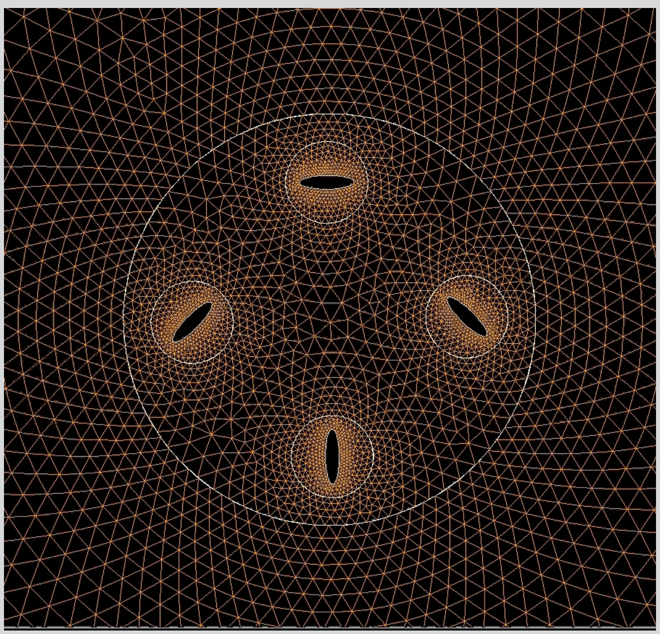
- For each **cell zone**, enable "Frame Motion" and set the details of the motion.

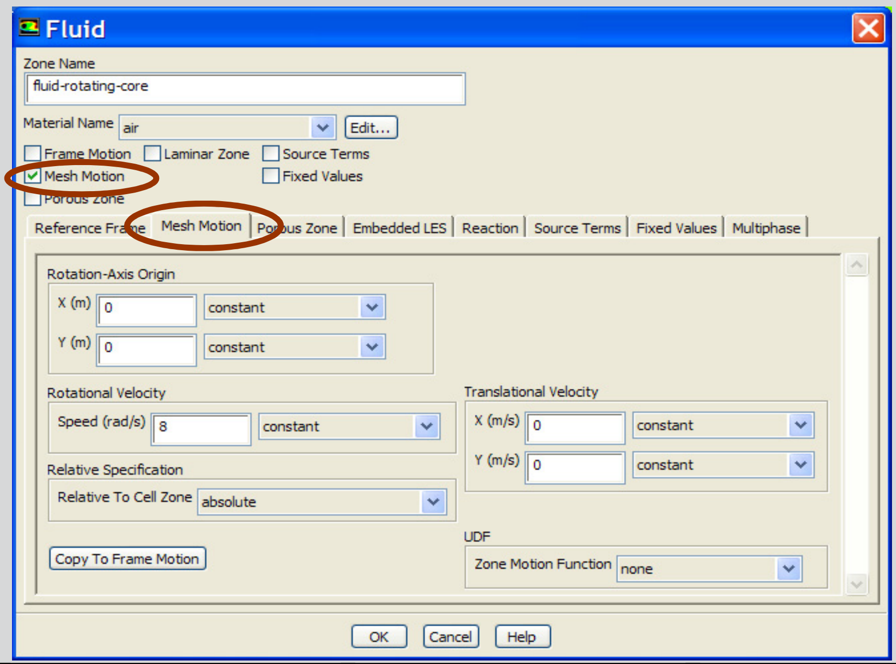
This motion can be defined relative to another zone, it doesn't have to be set to absolute coordinates



Defining a Sliding Mesh problem

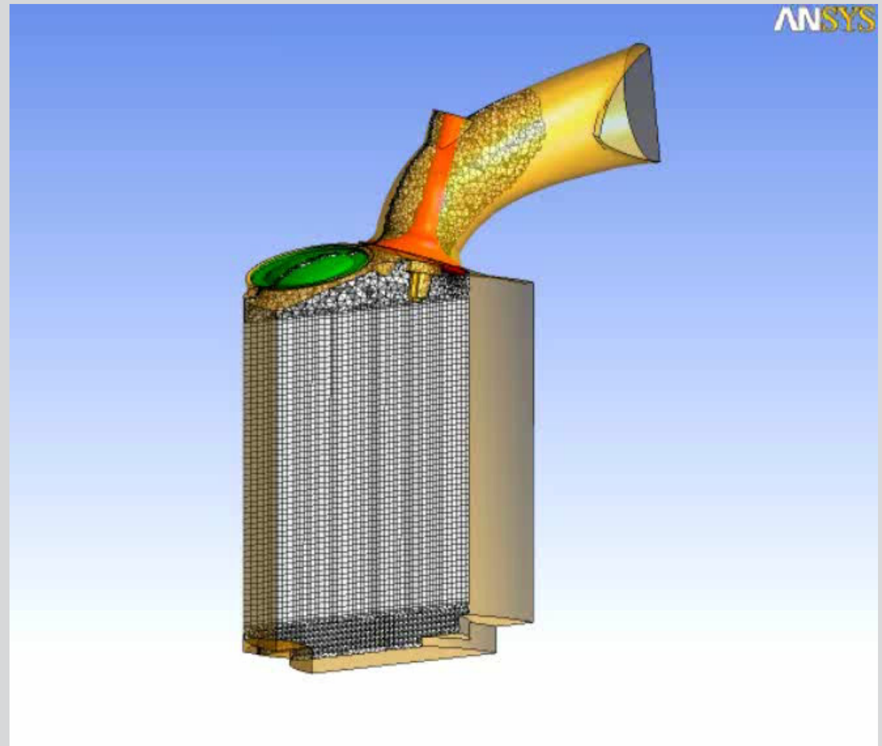
- In other problems, one must actually move the mesh components. The solution is therefore transient.
- Define the linear or rotational motion of each zone to use a sliding zone by setting "**Mesh Motion**"
  - Remember that in this case the different cell zones are actually moved at each timestep.
  - Make sure the model is always saved before testing the motion!
  - Other useful tips about running a transient simulation (like generating images on-the-fly will be given in a later lecture.





Mesh Deformation

- **Mesh Deformation can be applied in simulations where boundaries or objects are moved**
  - The solver calculates nodal displacements of these regions and adjusts the surrounding mesh to accommodate them



- **Examples of deforming meshes include**
  - Automotive piston moving inside a cylinder
  - A flap moving on an airplane wing
  - A valve opening and closing
  - An artery expanding and contracting
  - …

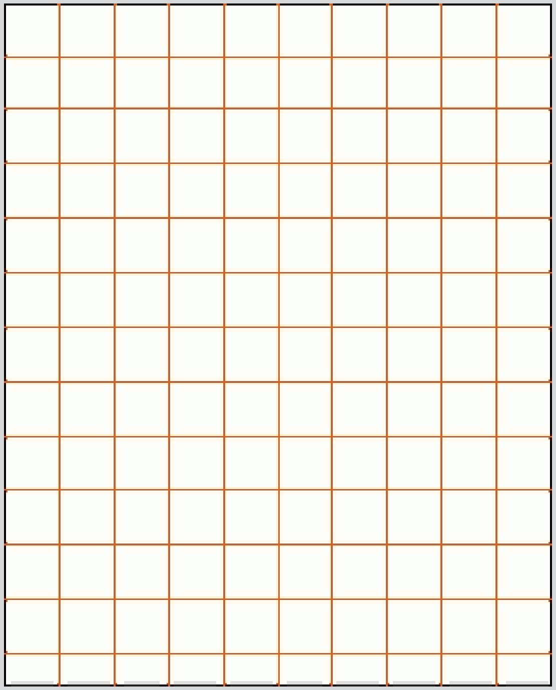
Dynamic Mesh (DM) Methods

- **Internal node positions are automatically calculated based on user specified boundary/object motion, cell type, and meshing schemes**

- **Basic Schemes**
  - Spring analogy (smoothing)
  - Local remeshing
  - Layering

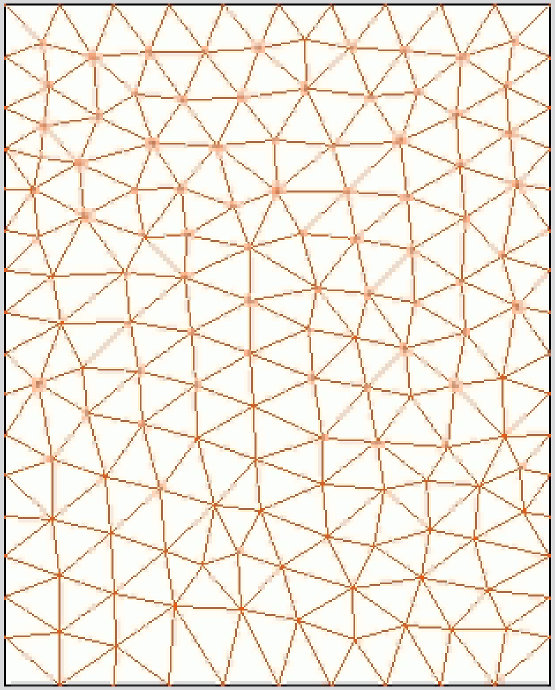
- **Other Methods**
  - 2.5 D
  - User defined mesh motion
  - In-cylinder motion (RPM, stroke length, crank angle, …)
  - Prescribed motion via profiles or UDF
  - Coupled motion based on hydrodynamic forces from the flow solution, via FLUENT’s six-degree-of-freedom (6DOF) solver

Dynamic Mesh Methods



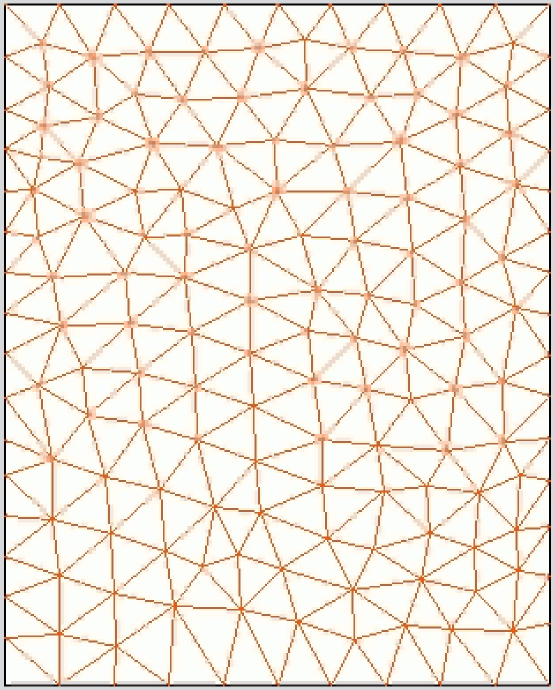
Layering

Layers of cells are generated and collapsed as they are overrun by the moving boundary. Layering is appropriate for quad/hex/prism meshes with linear or rotational motion and can tolerate small or large boundary deflections



Local Remeshing

In local remeshing, as cells become skewed due to moving boundaries, cells are collapsed and the skewed region is remeshed. Local remeshing is appropriate for tri/tet meshes with large range of boundary motion

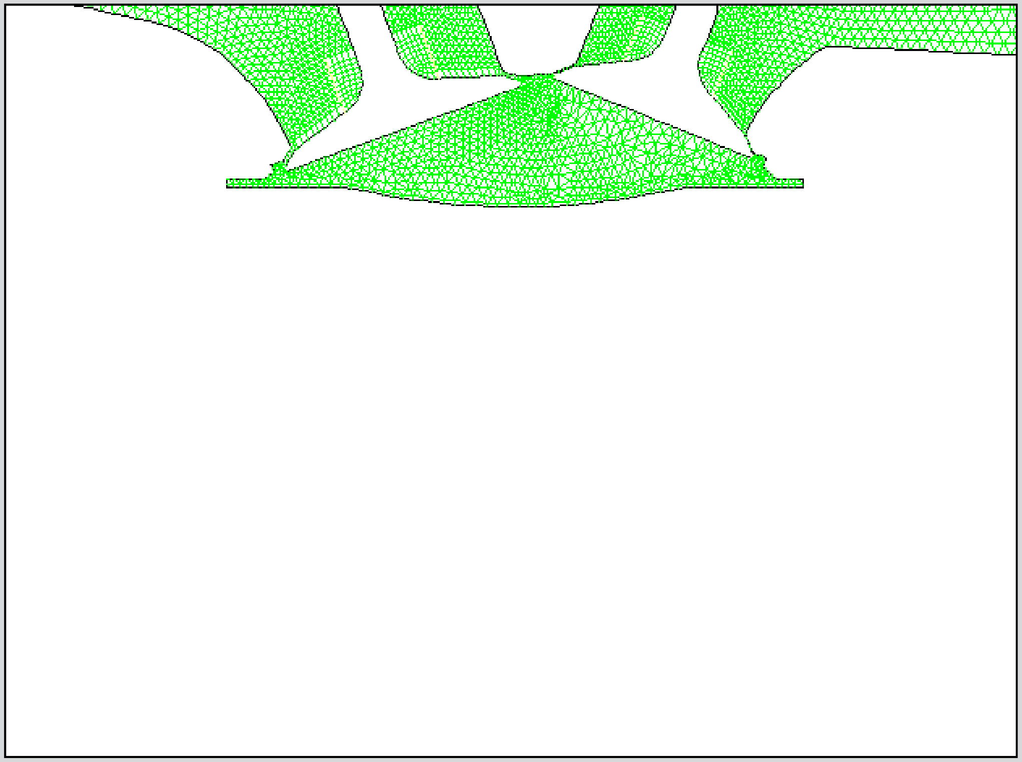


Spring Analogy

Spring analogy is useful when there are small boundary deformations. The connectivity and cell count is unchanged during motion. Spring analogy is appropriate for tri/tet meshes with small deformations

The Dynamic Mesh (DM) Model

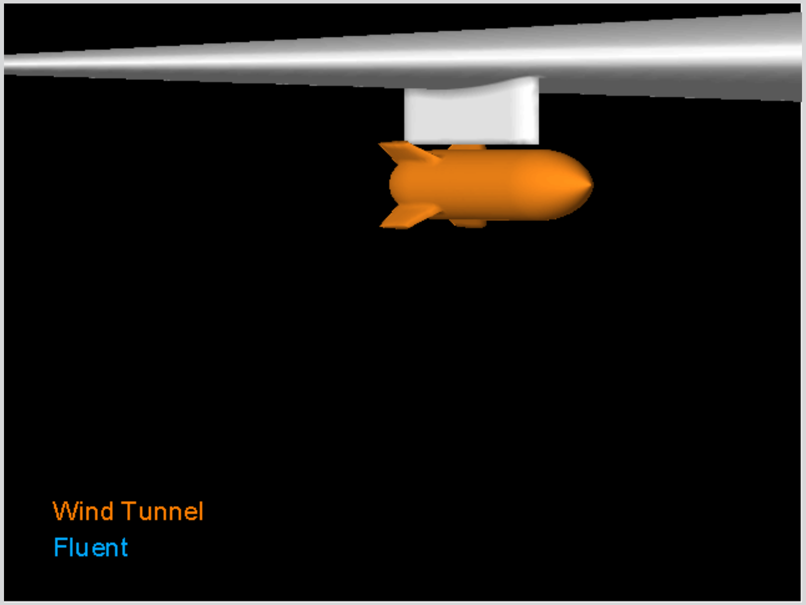
- **Combination of approaches:**
  - Initial mesh needs proper decomposition
  - Layering:
    - Valve travel region
    - Lower cylinder region
  - Remeshing:
    - Upper cylinder region
  - Non-conformal interface between zones



- **An advanced training course on Dynamic Mesh is available**

6 DOF Coupled Motion

- **Objects move as a result of aerodynamic forces and moments acting together with other forces, such as the gravity force, thrust forces, or ejector forces**
  – In such cases, the motion and the flow field are thus coupled, and we call this coupled motion



- **FLUENT provides the 6DOF Model**
  – The trajectory of an object is computed based on the aerodynamic forces/moments, gravitational force, and ejector forces
  – The 6-DOF UDF is fully parallelized

Summary

- **Five different approaches may be used to model flows over moving parts**
  - Single (Rotating) Reference Frame Model
  - Multiple Reference Frame Model
  - Mixing Plane Model
  - Sliding Mesh Model
  - Dynamic Mesh Model (Consider our Advanced Trainings)
- **The first three methods are primarily steady-state approaches while sliding mesh is inherently unsteady**
- **Enabling these models involves in part, changing the stationary fluid zones to either Moving Reference Frame or Moving Mesh**
- **Most physical models are compatible with moving reference frames or moving meshes (e.g. multiphase, combustion, heat transfer, etc.)**

Appendix

Motion in Solid Zones

- **In most simulations it is not necessary to specify any motion in solid zones**
- **Solid zone motion should be used when the convection of energy needs to be considered**
  - For example, a hot jet impinging on a rotating disk. To prevent a hot spot from forming the convection of energy in the solid needs to be included
- **In this case the heat transfer equation has a convective term**

Motion in Solid Zones

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

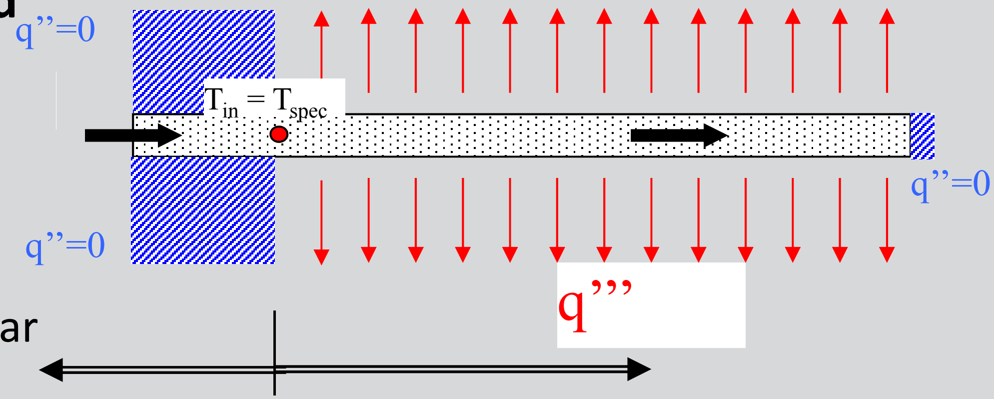
$$\frac{\partial(\rho h)}{\partial t} + \nabla \cdot (\rho U_S h) = \nabla \cdot (\lambda \nabla T) + S_E$$

Solid Velocity

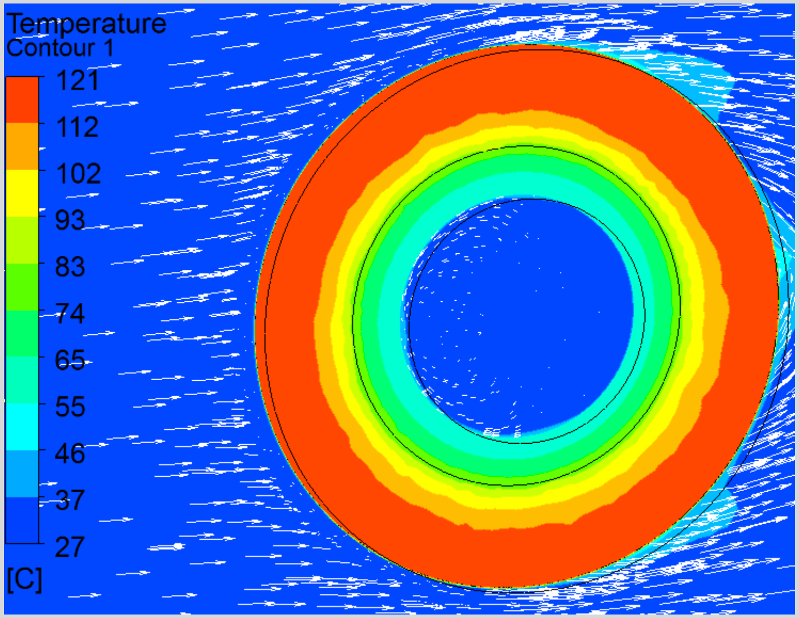
- **Note that the solid is never physically moved when using this approach, there is only an additional advection term added to the energy equation**

Motion in Solid Zones

- **Solid zone 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

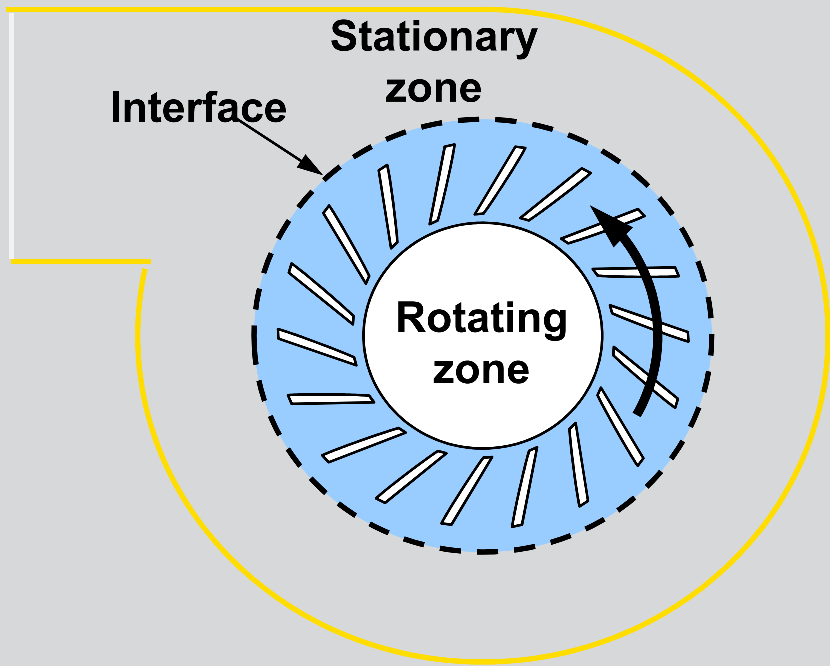


  - **Rotational Motion**
    - For example, a brake rotor which is heated by brake pads



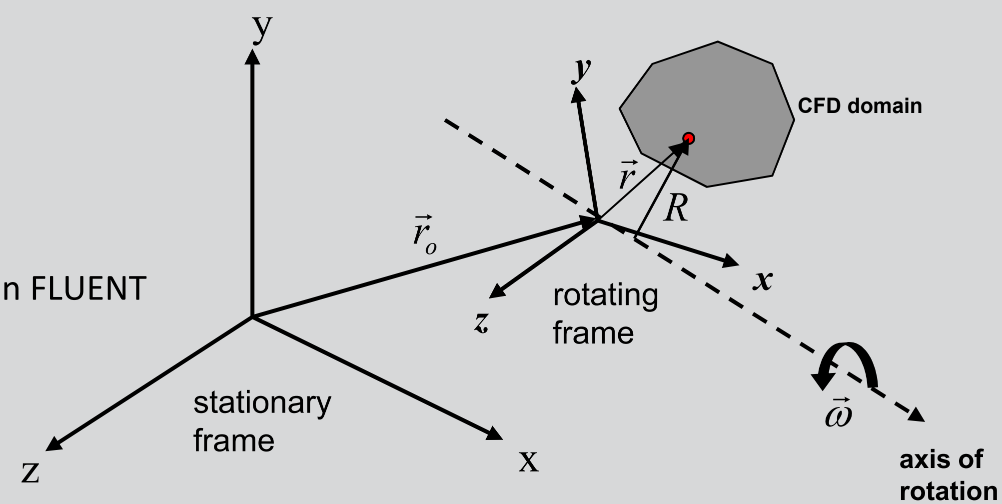
Moving Reference Frame Modeling

Systems like these can be solved by dividing the domain into multiple fluid zones
– some zones will be rotating, others stationary



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
    - Can be selected under the General tab in Problem Setup
  - **Absolute Velocity Formulation**
    - Derived from the relative velocity formulation
    - Uses the *absolute velocity* as the dependent variable
    - Default formulation for rotating zones in FLUENT
  - **Rotational source terms appear in momentum equations**



The Velocity Triangle

- The relationship between the absolute and relative velocities is given by

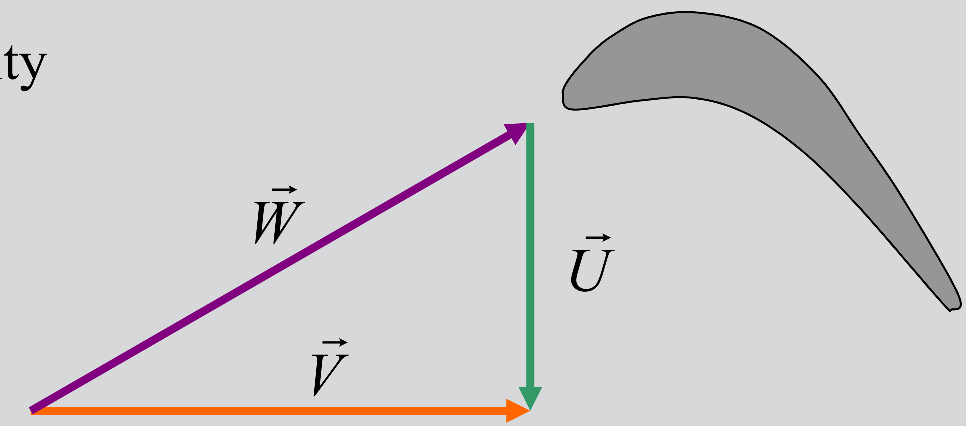
$$\vec{V} = \vec{W} + \vec{U}$$

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

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

$\vec{V}$ = Absolute Velocity

$\vec{W}$ = 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

Introduction to the MRF Model

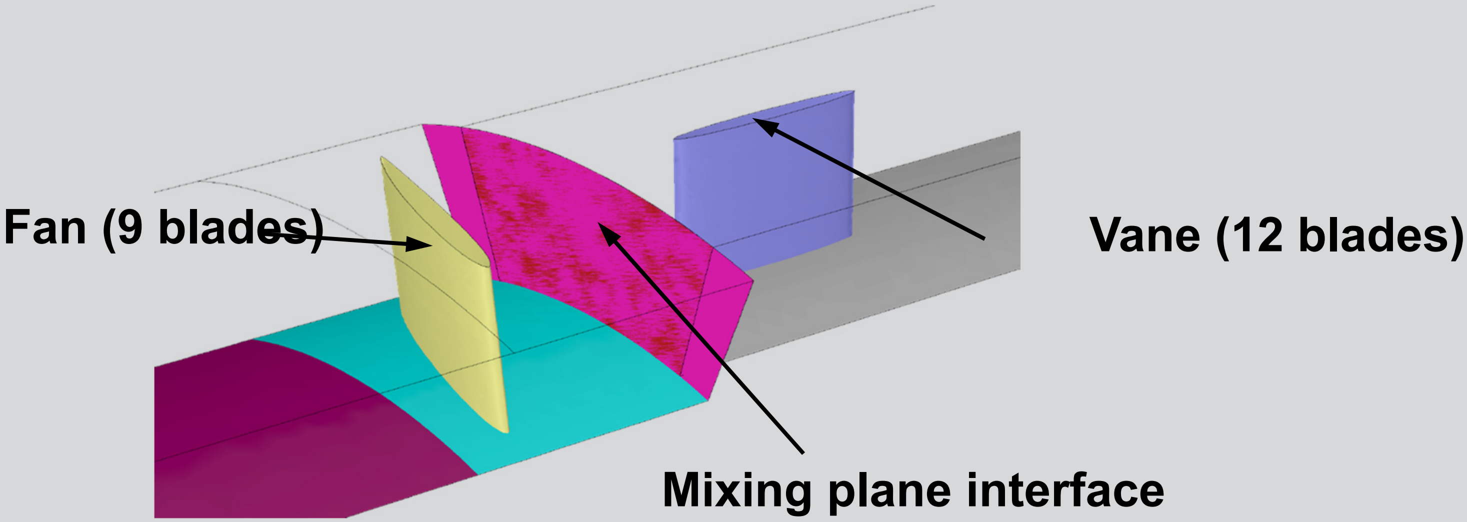
- **The domain is subdivided into stationary and rotating fluid zones**
  – More than one rotating zone is permitted
  – Zones can rotate at different speeds
- **Governing equations are solved in each fluid zone**
  – SRF equations used in rotating zones
  – At the interfaces between the rotating and stationary zones, appropriate transformations of the velocity vector and velocity gradients are performed to compute fluxes of mass, momentum, energy, and other scalars
  – Flow is assumed to be steady in each zone (clearly an approximation)
- **MRF ignores the relative motions of the zones with respect to each other**
  – Does not account for fluid dynamic interaction between stationary and rotating components
  – For this reason MRF is often referred to as the “frozen rotor” approach
- **Ideally, the flow at the MRF interfaces should be relatively uniform or “mixed out”**

The Mixing Plane Model (MPM)

The MPM is a technique which permits steady-state solutions for multistage axial and centrifugal turbomachines where upstream and downstream periodic domains do not match at the connection

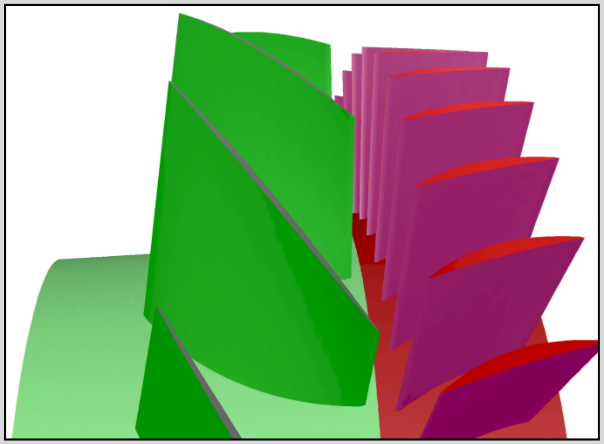
Advantage:

- MPM requires only a single blade passage per blade row regardless of the number of blades, because of circumferential averaging non-uniformities in the flow at the mixing plane interface
- MPM can handle different numbers of blades at both sides of mixing plane

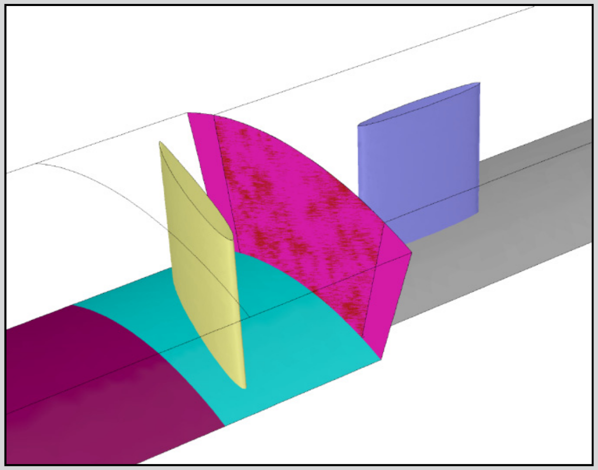


MPM vs. MRF

- **MRF can be used only if we have equal periodic angles for each row**



- **For multistage turbomachinery problems**
  - The stage boundary conditions are often known (e.g. inlet total pressure and temperature and stage outlet static pressure) but not the inter-stage conditions
  - Blade counts will generally not be the same from one row to the next

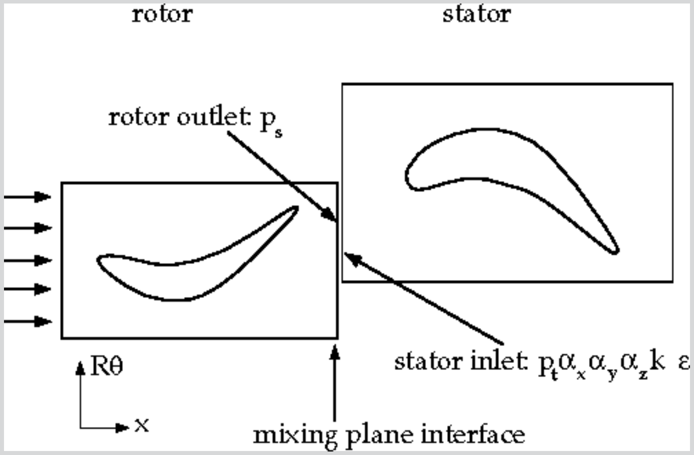


- **The MPM requires only a single blade passage per blade row regardless of the number of blades**
  - This is accomplished by mixing out (averaging) the circumferential non-uniformities in the flow at the inter-stage (mixing plane) interface

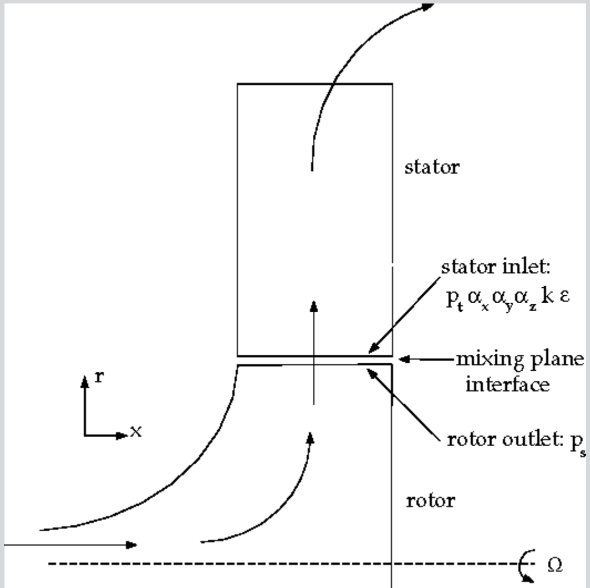
Axial vs. Radial Mixing Planes

SRF solutions are obtained in each domain, with the domains linked by passing boundary conditions from one zone to another

- The inlet/outlet boundaries must be assigned BC types in one of the following combinations:
  - Pressure outlet / Pressure inlet
  - Pressure outlet / Velocity inlet
  - Pressure outlet / Mass flow inlet



Axial machines

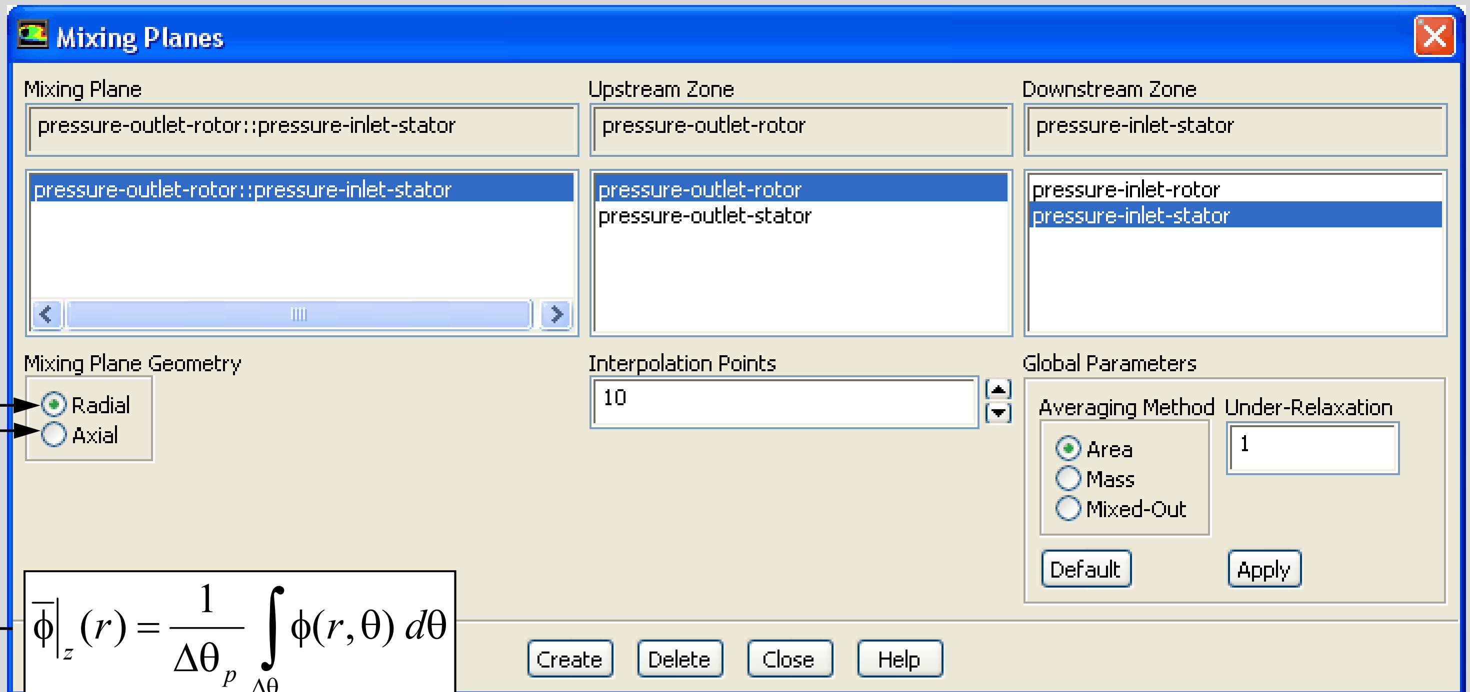


Radial machines

Mixing Plane – Set Up

Mixing Plane Model

- GUI: Define → Mixing Planes



$$\bar{\phi}\big|_z(r) = \frac{1}{\Delta\theta_p}\int_{\Delta\theta_p}\phi(r,\theta)\,d\theta$$

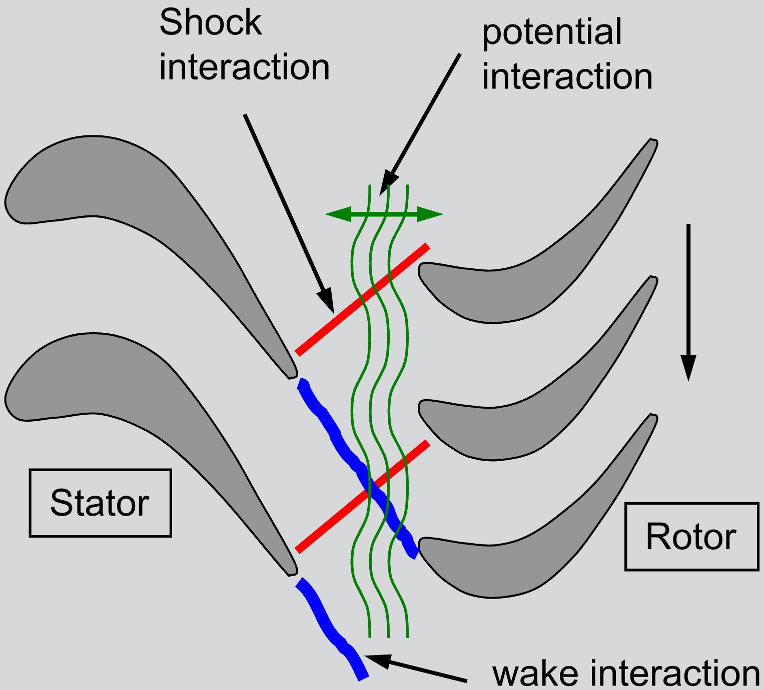
$$\bar{\phi}\big|_r(z) = \frac{1}{\Delta\theta_p}\int_{\Delta\theta_p}\phi(z,\theta)\,d\theta$$

Mixing Plane Geometry determines method of circumferential averaging
- Choose Radial for axial flow machines
- Choose Axial for radial flow machines

The Sliding Mesh Model (SMM)

The relative motion of stationary and rotating components in a turbo-machine will give rise to <u>unsteady interactions</u>

- **These interactions are generally classified as follows:**
  - Potential interactions (pressure wave interactions)
  - Wake interactions
  - Shock interactions



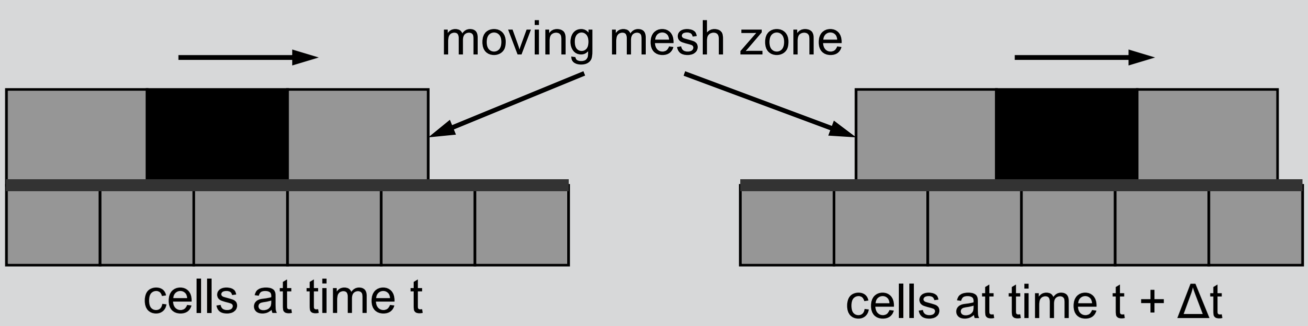
Both MRF and MPM neglect unsteady interaction entirely and thus are limited to flows where these effects are weak

If unsteady interaction can not be neglected, we can employ the Sliding Mesh model (SMM) to account for the relative motion between the stationary and rotating components

How the Sliding Mesh Model Works

Like the MRF model, the domain is divided into moving and stationary zones, separated by <u>non-conformal interfaces</u>

Unlike the MRF model, each moving zone's mesh will be updated as a function of time, thus making the mathematical problem inherently <u>unsteady.</u>



Another difference with MRF is that the governing equations have a new moving mesh form, and are solved in the <u>stationary reference frame for absolute quantities</u>

- Moving reference frame formulation is NOT used here (i.e. no additional accelerations acting as sources terms in the momentum equations)
- Equations are a special case of the general moving/deforming mesh formulation
  - Assumes rigid mesh motion and sliding, non-conformal interfaces

N-S Equations: Sliding Mesh

$$\frac{d}{dt}\int_V \rho dV + \oint_S \rho(\vec{V} - \vec{U}) \cdot = 0$$ **(continuity)**

$$\frac{d}{dt}\int_V \rho v_x dV + \oint_S \left[\rho\left(\vec{V} - \vec{U}\right)v_x + p\hat{i}\right] \cdot d\vec{S} = \oint_S \vec{\tau}_{vx} \cdot d\vec{S}$$ **(x momentum)**

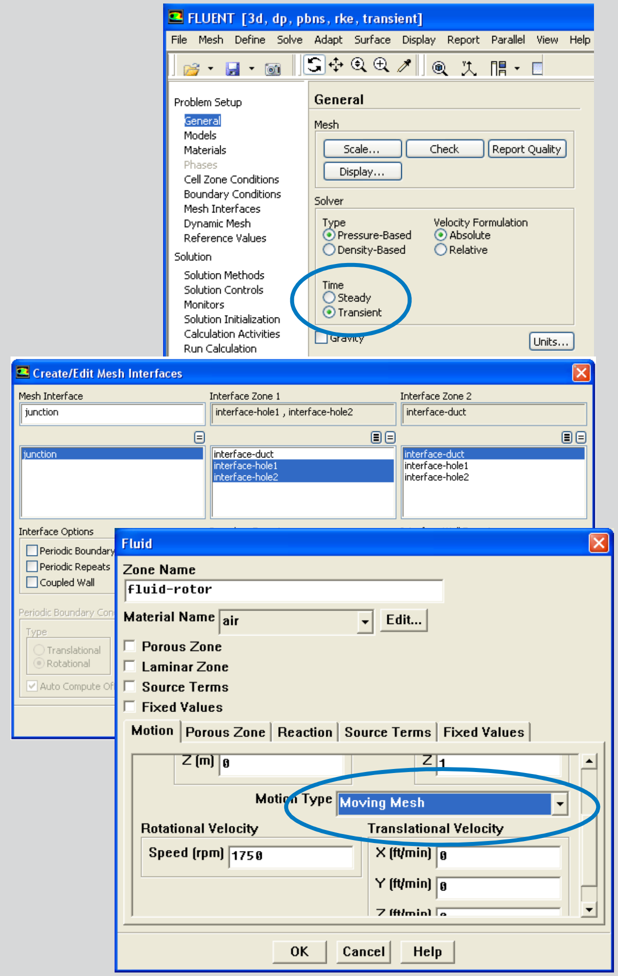
$$\frac{d}{dt}\int_V \rho v_y dV + \oint_S \left[\rho\left(\vec{V} - \vec{U}\right)v_y + p\hat{j}\right] \cdot d\vec{S} = \oint_S \vec{\tau}_{vy} \cdot d\vec{S}$$ **(y momentum)**

$$\frac{d}{dt}\int_V \rho v_z dV + \oint_S \left[\rho\left(\vec{V} - \vec{U}\right)v_z + p\hat{k}\right] = \oint_S \vec{\tau}_{vz} \cdot d\vec{S}$$ **(z momentum)**

$$\frac{d}{dt}\int_V \rho e_t + \oint_S \rho\left(\vec{V} - \vec{U}\right)\left(e_t + \frac{p}{\rho}\right) \cdot d\vec{S} = \oint_S \left(\vec{\tau}_{vx} v_x + \vec{\tau}_{vy} v_y + \vec{\tau}_{vz} v_z - \vec{q}\right) \cdot d\vec{S}$$ **(energy)**

SMM - Setup

- **Enable the <u>unsteady solver</u>**
- **Define sliding zones as Interface BC types**
- **For each interface zone pair, create a non-conformal interface**
  - Enable Periodic option if sliding/rotating motion is periodic.
  - Enable Coupled for conjugate heat transfer
- **For moving zones, select Moving Mesh as Motion Type in Fluid BC panel**
- **Other BCs are same as SRF, MRF models**

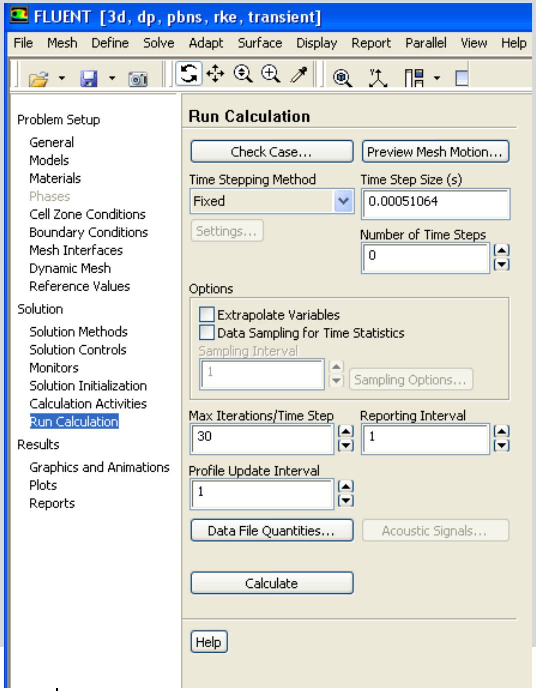


Solving SMM Problems

- **Choose appropriate Time Step Size and Max Iterations per Time Step to ensure good convergence with each time step**
  – Time Step Size should be no larger than the time it takes for a moving cell to advance past a stationary point:

$$\Delta t \approx \frac{\Delta s}{\omega R}$$

Δs = Average cell size
ωR = Translational speed



- **Advance the solution until the flow becomes time-periodic (pressures, velocities, etc., oscillate with a repeating time variation).**
  – Usually requires several revolutions of the grid.
  – Good initial conditions can reduce the time needed to achieve time-periodicity`