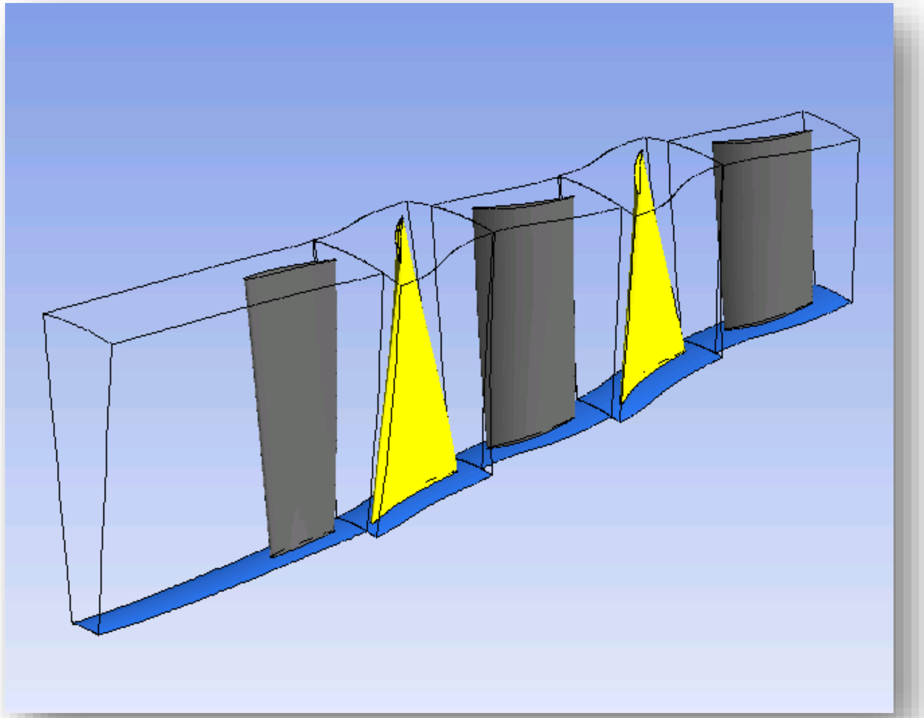
Lecture 04: Multi-Component Analysis - Steady

Release 2020 R2

Outline

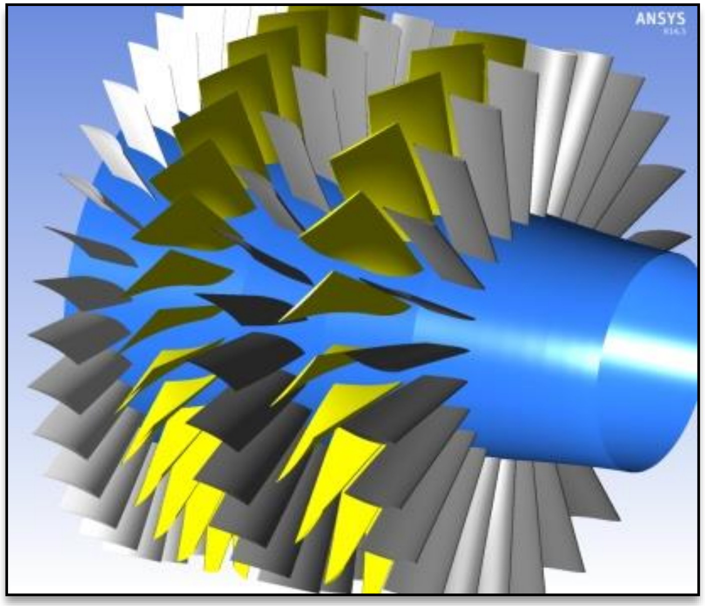
- Introduction to Multi-Component Analysis
- Multiple Reference Frames
- Mixing Plane Model
- Frozen Rotor Model
- New Fluent Turbo Models
- General Turbo Interfaces Placement Best Practices
- Summary



Ansys

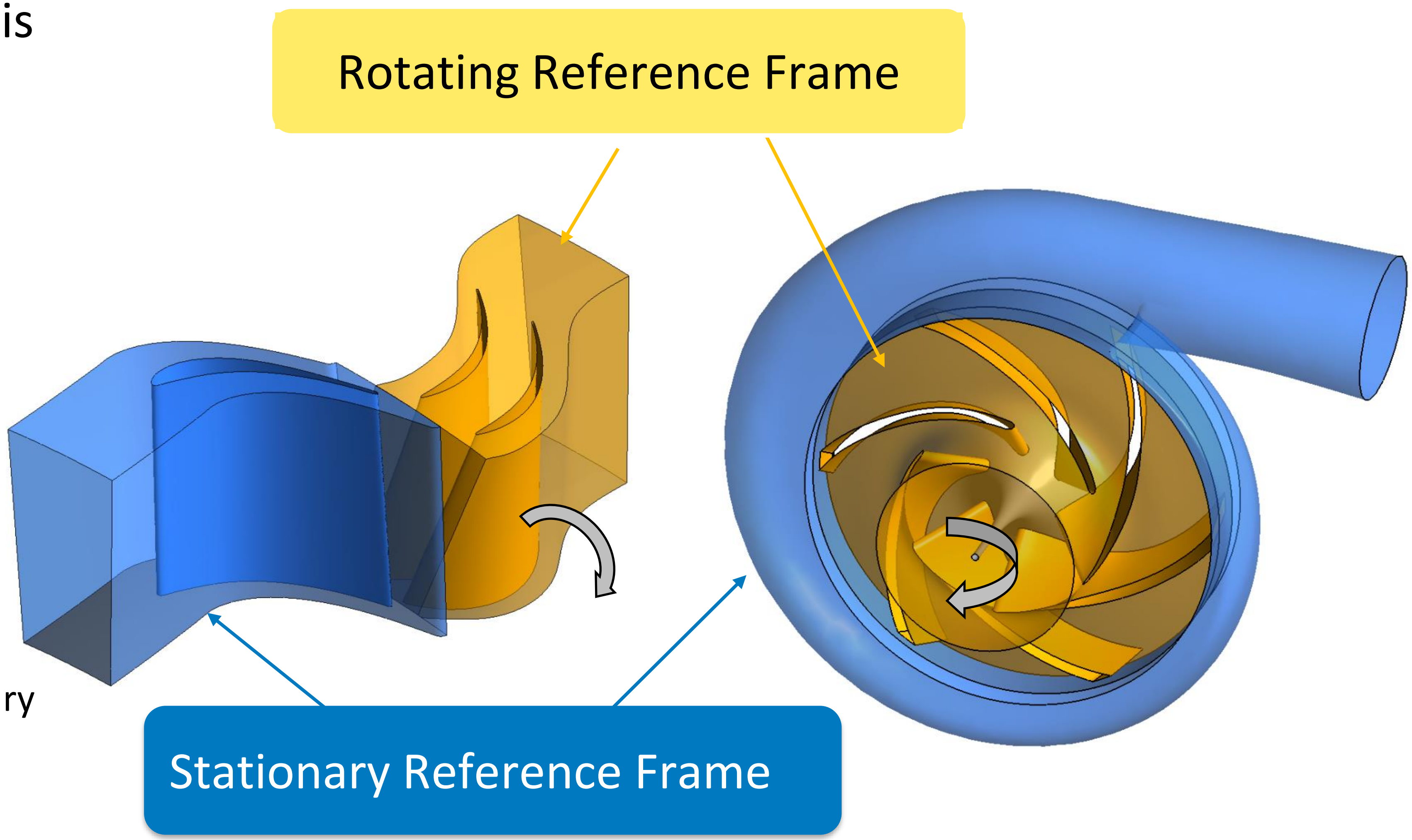
Introduction to Multi-Component Analysis

- Many rotating machines contain multiple components (Multi-stage machines)
  - E.g. a compressor stage consists of a rotor blade row to transfer energy to the fluid followed by a stator blade row to convert the swirl to a static pressure rise
- Some contain stationary surfaces which are not surfaces of revolution in respect to the axis of rotation
  - Example: The volute surfaces of a pump
- Single rotating component analysis could also be used for simplified single rotating parts
  - Example: pump impeller domain (see workshop 1 of this course)
- Multi-Component Analysis is most useful to examine component interactions
  - Match components to achieve desired performance over operating range



Introduction to Multi-Component Analysis (2)

- Multi-Component Analysis is best performed using the Multiple Reference Frames method:
- Flow domain is split to stationary (absolute) and rotating domains, meshed independently
  - Each domain in its respective frame
    - Rotating domains in Rotating Reference Frames
      Stationary domains in Stationary Reference Frames

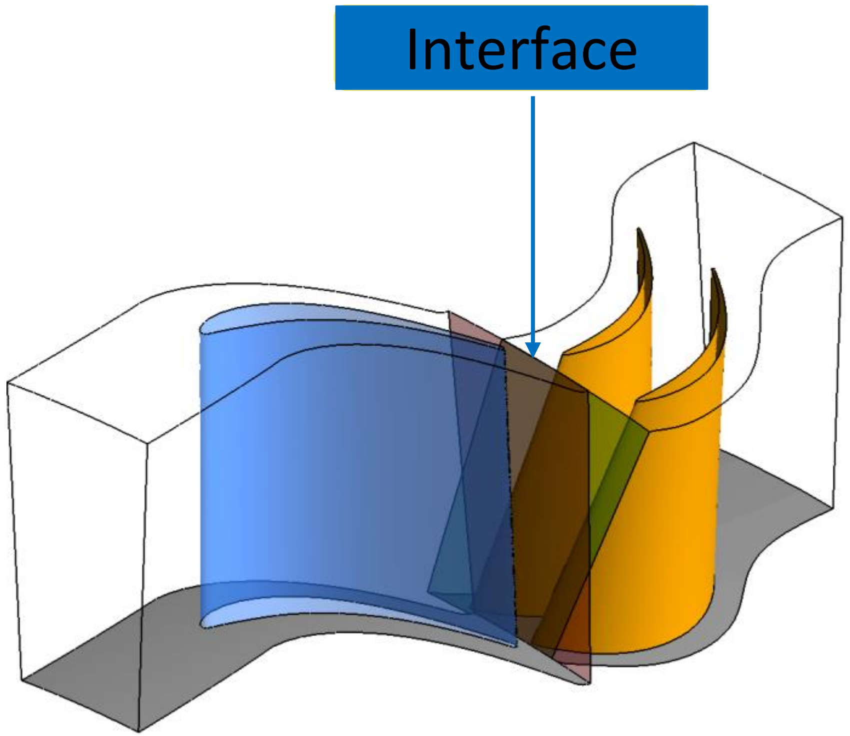


Introduction to Multi-Component Analysis (3)

- Multiple Reference Frames method simultaneously solves a series of rotating and stationary components
  - Two-way fully-coupled interactions
- Interfaces between rotating and stationary components handled by flow solver
  - Approaches available:
    - Steady state frame change methods
      - **Mixing Plane Model**
      - **Frozen Rotor Model**

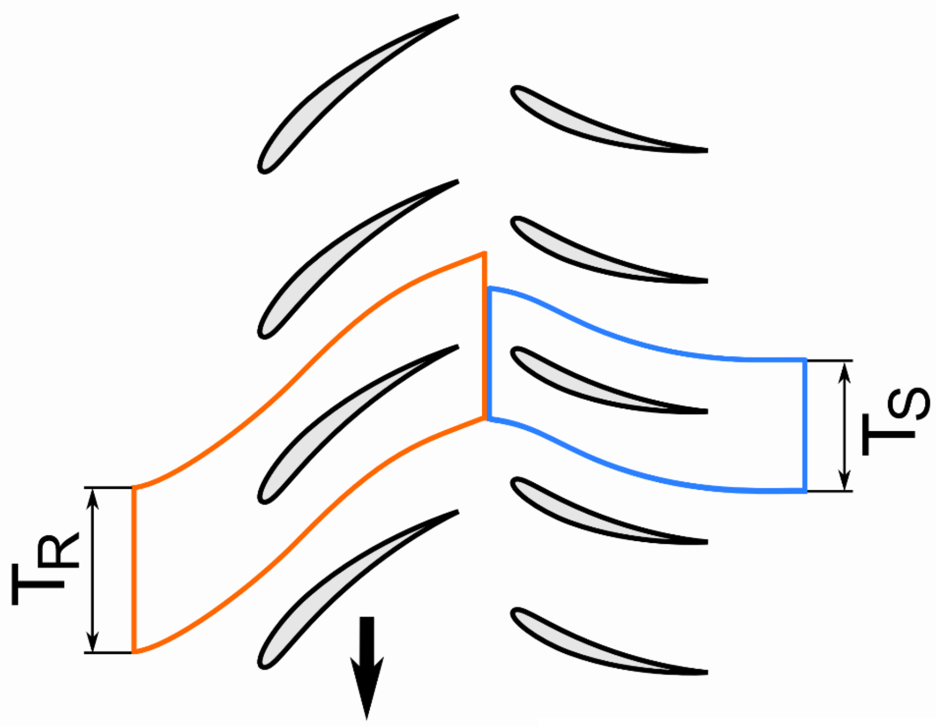
      Covered in this lecture
    - Transient full-annulus
    - Transient pitch change

    Covered in following lecture
- General combination of components:
  - Any combination of Rotating Reference Frames (RRF) and Stationary domains in Stationary Reference Frames (SRF)
  - RRF to RRF: different rotational speeds
  - RRF to RRF: counter-rotating



Pitch Change Illustration

- Taking advantage of rotational periodicity in Multiple Reference Frames method is especially desirable
  - Solve for one passage per component, where possible
    - Reduce computation cost
  - Different blade counts in rotor and stator result in non-matching pitches between rotor and stator single-passage domains
    - In the example on the right, the rotor blade pitch T_R is larger than that one of the stator vane T_S
  - To account for non-matching pitches, a so-called pitch change method is needed
  - The mixing plane approach inherently comprises a treatment for unequal pitches, through the mixing of the flow in circumferential direction
  - For the frozen rotor method a pitch change method has been implemented in Fluent

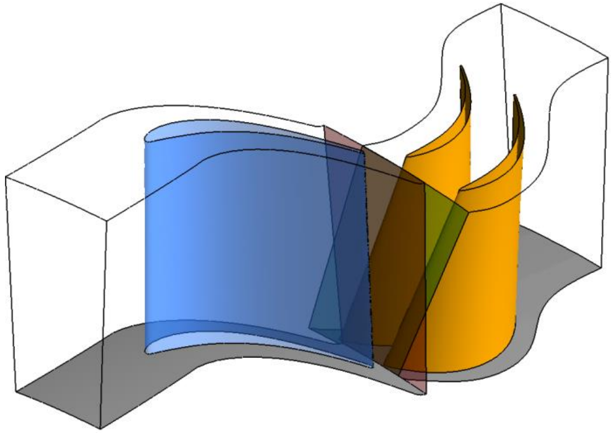


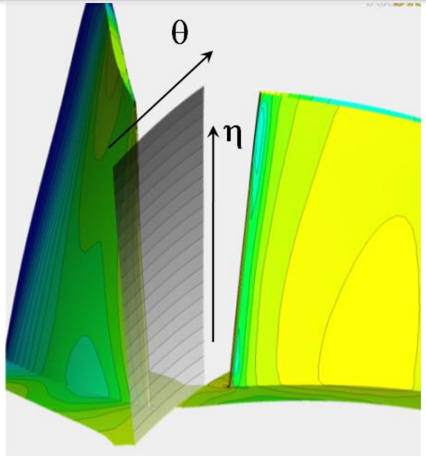
Mixing Plane Model (MP)

Ansys

Mixing Plane Model

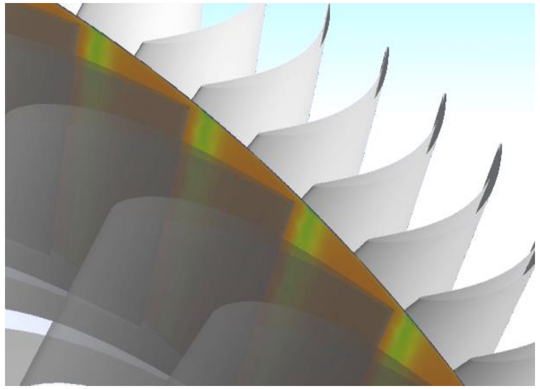
Blade frame change via conservative “mixing” process in circumferential direction



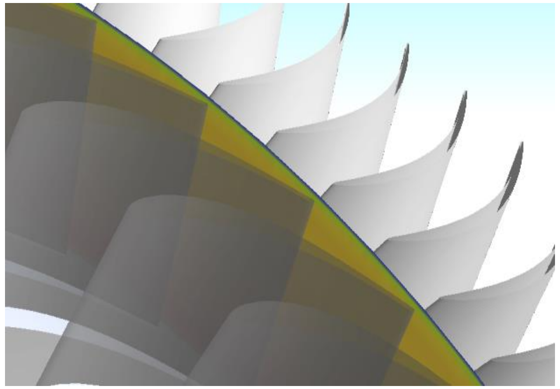


Upstream interface side

Downstream interface side

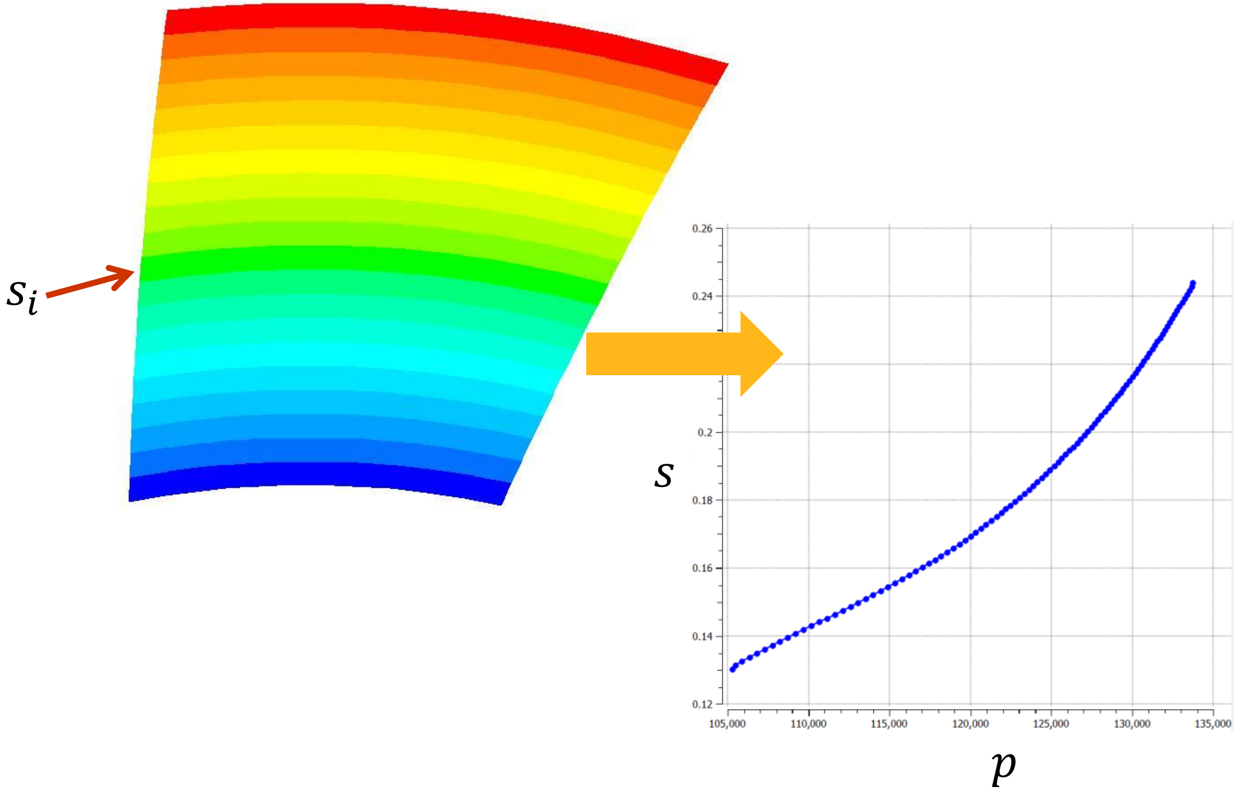


Mixing



Mixing Plane Interpolation

- The mixing plane model uses a circumferential averaging technique
  - Any flow variable distribution $\phi(s,\theta)$ on the upstream side of the interface is transformed to an averaged spanwise profile → used in the downstream side of the interface
  - The same transformation is performed for all necessary variables on the downstream side and the resulting downstream averaged profiles are used in the upstream side of the interface
  - The spanwise direction s can be any coordinate in the direction from hub to shroud
  - Variation in the spanwise direction is preserved
  - Blade pitch change via conservative “mixing” process in circumferential direction

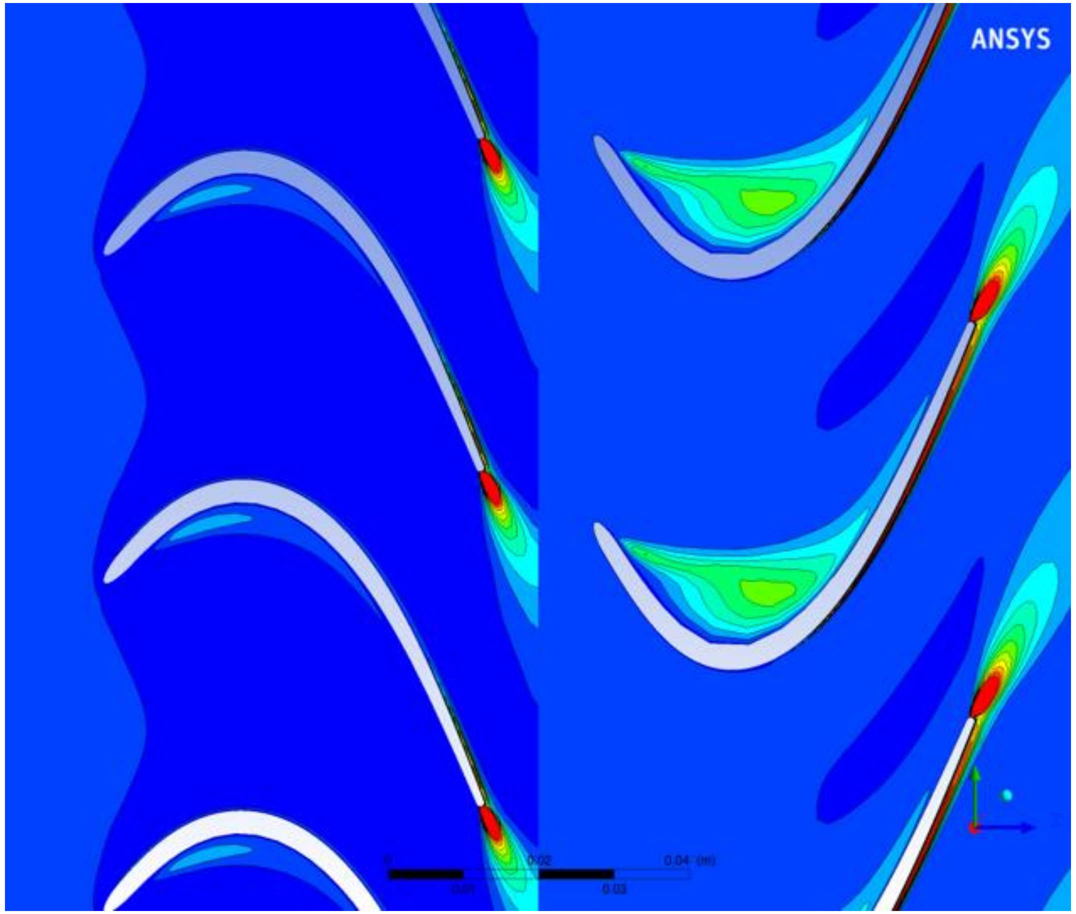


$$\bar{\varphi}(s) = \frac{1}{\Delta\theta_p}\int_{\Delta\theta_p} \varphi(s,\theta)d\theta \approx \bar{\varphi}(s_i) = \frac{\sum_j \varphi_{i,j} A_{i,j}}{\sum_j A_{i,j}}$$

j = index of cell faces whose centroids are close to s within band i

Mixing Plane Model Contours Example

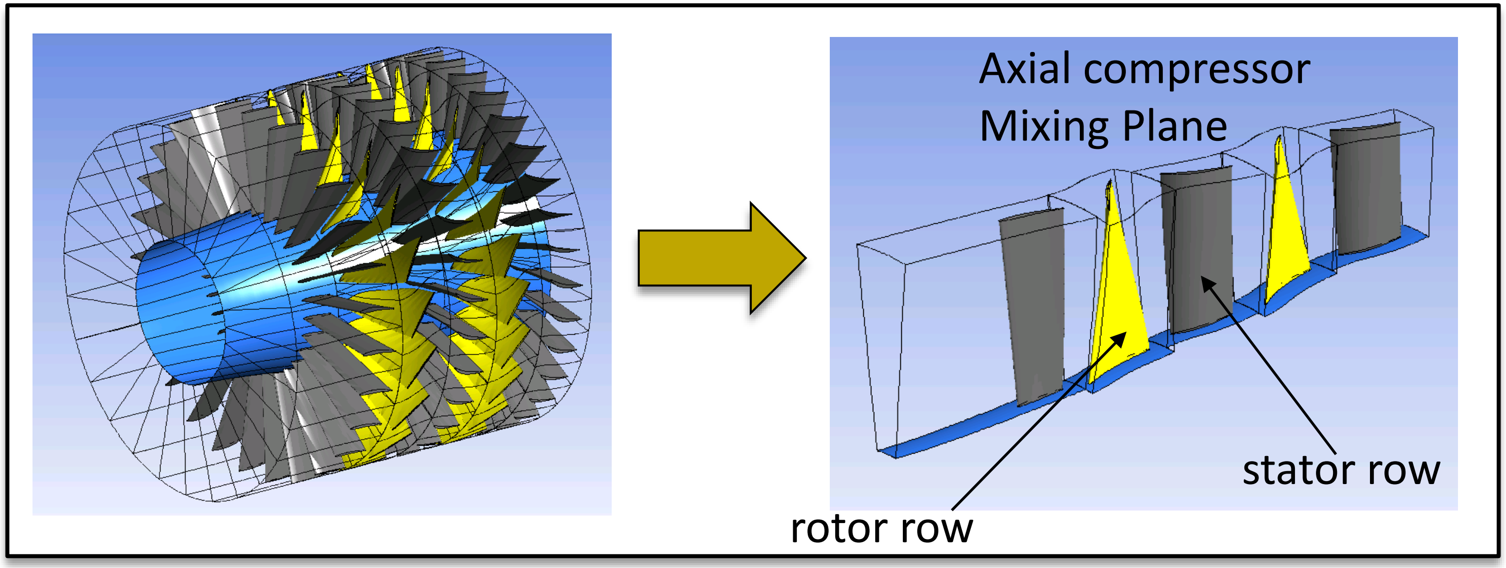
No transient interaction between blade rows (wakes, vortices)



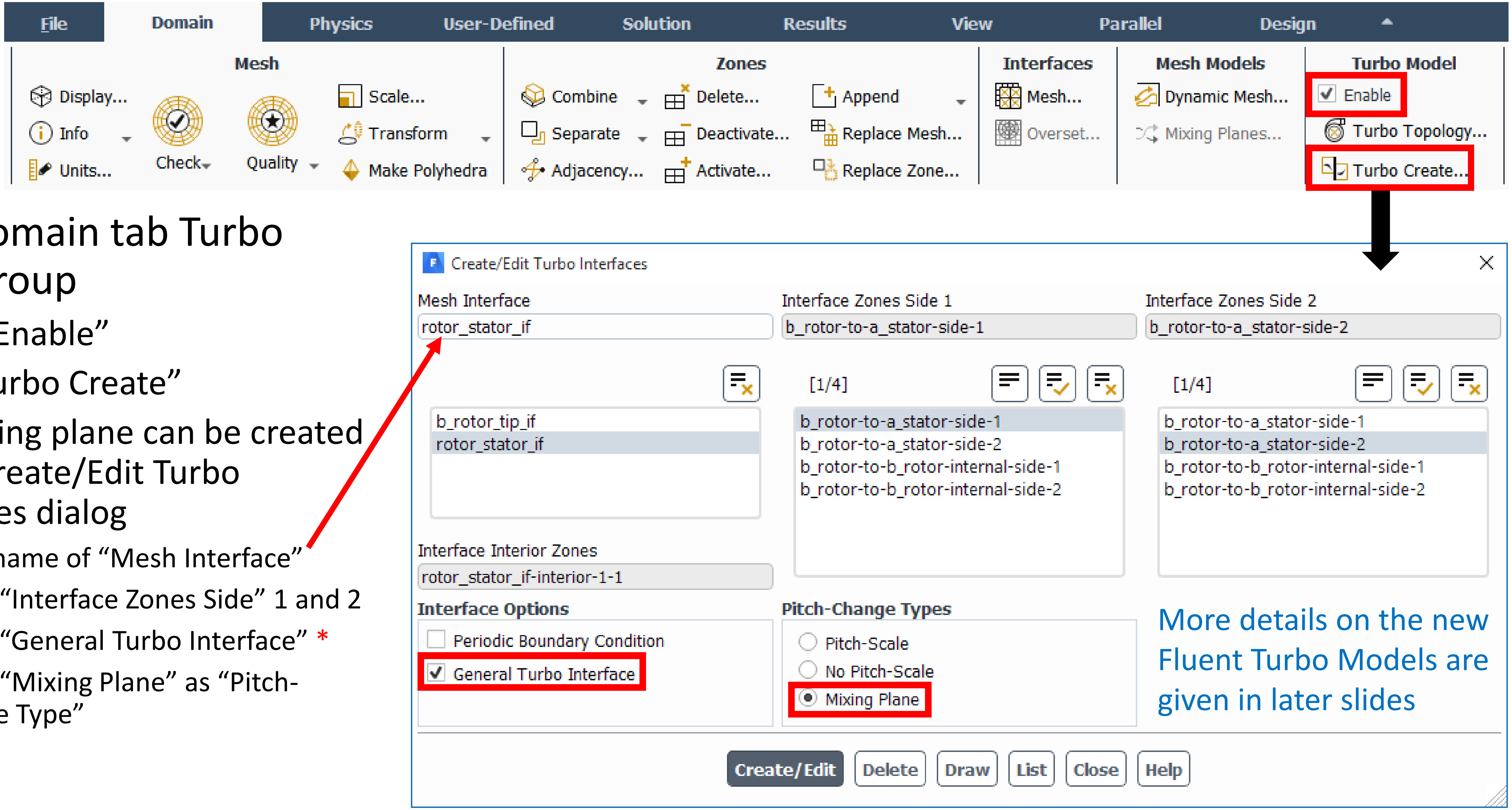
Ansys

Mixing Plane Characteristics

- Usable when circumferential variation at interface is small or not important
  - Axial compressor, axial turbine, some radial impeller with vaned diffuser…
- Simulation is run as steady state
- Only one passage in each component is modeled using rotational periodic boundary conditions
- For each rotating component, its cell-zone is set to rotational frame motion in the same way as for a single rotating component (shown in Lecture 02 of this course)
- Mixing losses occur at so-called General Turbo Interface (GTI) Mixing Plane interfaces (see next slide) → Since mixing processes have inherent losses by physics!



Mixing Plane Model Setup in Fluent



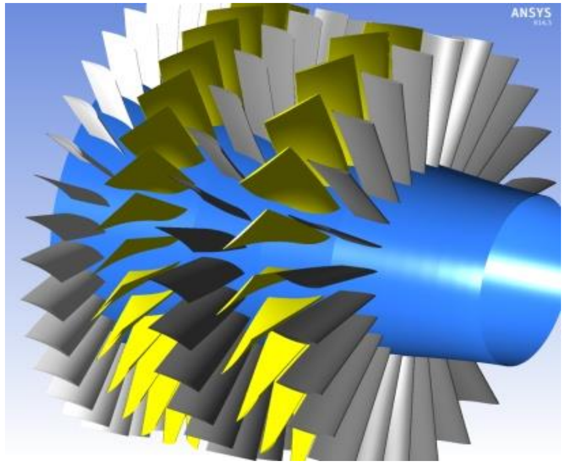
- In the Domain tab Turbo Model group
  - Check “Enable”
  - Click “Turbo Create”
  - The mixing plane can be created in the Create/Edit Turbo Interfaces dialog
    - Enter name of “Mesh Interface”
    - Select “Interface Zones Side” 1 and 2
    - Select “General Turbo Interface” *
    - Select “Mixing Plane” as “Pitch-Change Type”

More details on the new Fluent Turbo Models are given in later slides

* Important Notice: The GTI interface must be non-conformal

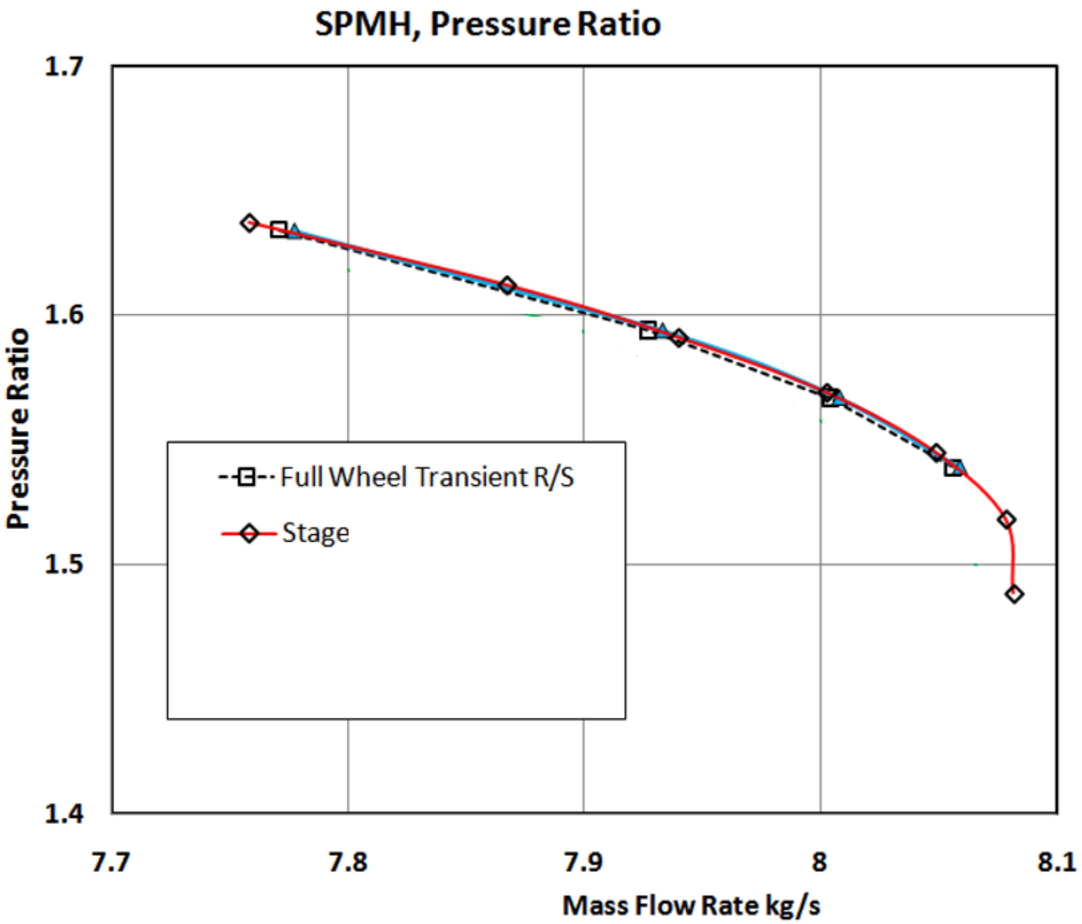
Mixing Plane Example – 2.5 Stage Axial Compressor

Hannover compressor with modified blade counts



Design rotational speed	17100 rpm
Mass flow rate	7.82 kg/s
Total pressure ratio	2.7
Isentropic efficiency	89.8%
Inlet total pressure	60 kPa
Inlet Mach number	0.5

Ref. 1/3 wheel	11.6 mil nodes
TT, PT, mixing-plane	1.3 mil nodes



Mixing Plane Model Limitations

- Interpolation process can introduce errors at the mixing plane interface
- Mixing plane can handle some backflow, but you should avoid using the mixing plane if a large amount of backflow is present (leads to poor convergence)*
  - Robustness issues with bidirectional flow within a circumferential band
  - Backflow direction, total temperature, scalars will be computed from downstream profiles
- Mixing process resulting from the averaging will introduce additional (usually small) loss to the stage calculation
- Wake effects on downstream blade rows and shock wave interactions will not be predicted
  - Modeling these effects requires a transient rotor stator model
- Errors in the mixing plane model increase as the spacing between the stages decreases (stronger interaction)

* Backflow is allowed in the meridional but not in the blade to blade view; in the latter case, one has both inflow and outflow in the same band and that's problematic for the averaging. If this is the case (blade to blade backflow) change position of mixing plane or do a transient simulation

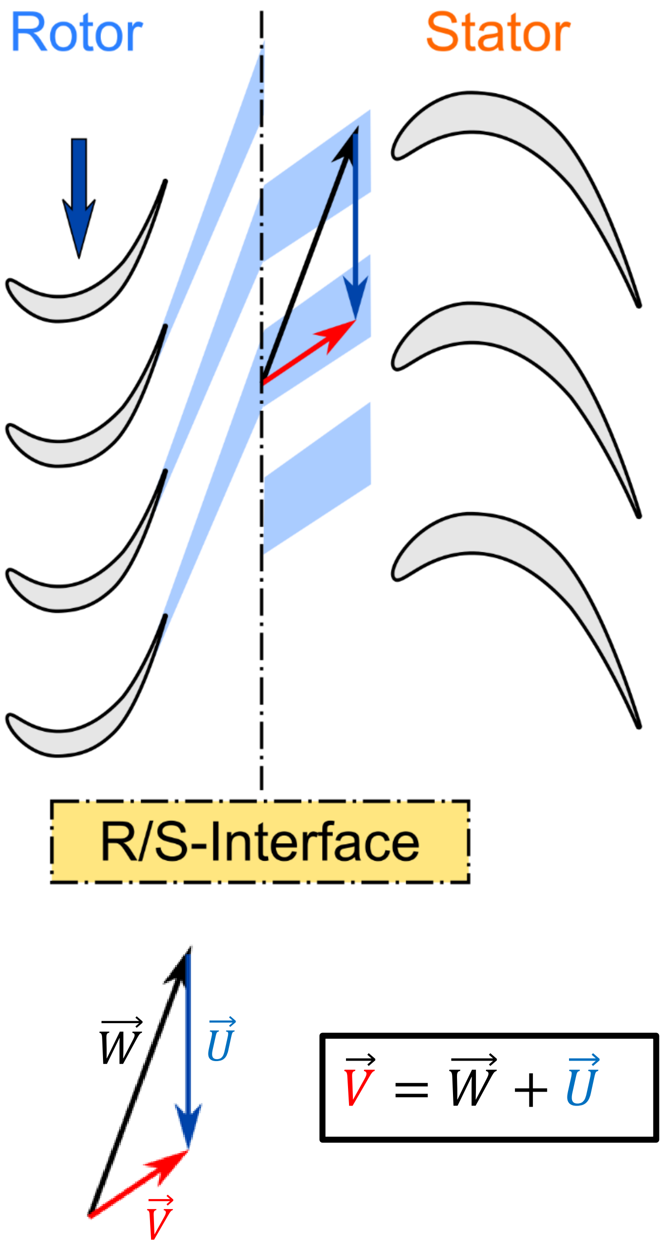
Frozen Rotor Model (FR)

Frozen Rotor Model

Frame change at interface w/o averaging

Sensitive to relative component position (so called clocking effect)

Blade pitch change via conservative interface flux scaling

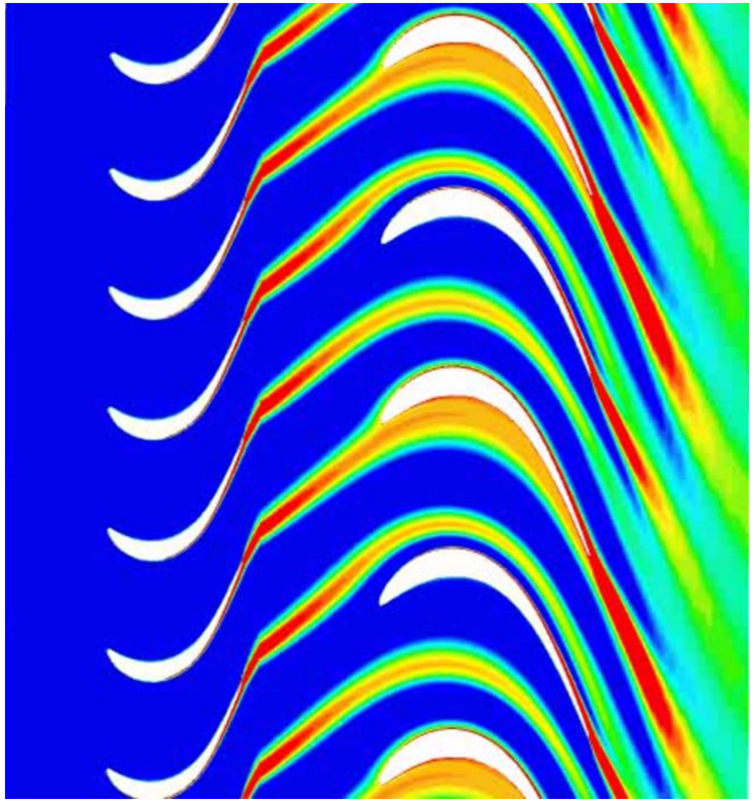


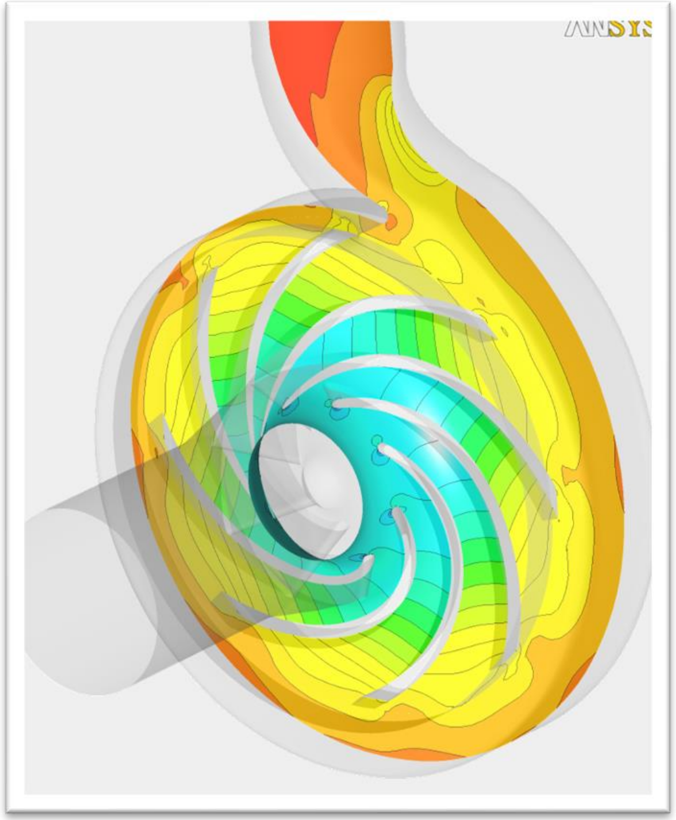
Frozen Rotor Model

- At the interfaces, appropriate transformations of the velocity vectors and velocity gradients are performed, and local fluxes of mass, momentum, energy, and other scalars are determined
- No account is taken for the relative motion of one domain with respect to the other!
  - Meshes do not move with time
  - Result is sensitive to relative position of components
    - Try different rotor angular positions
- More robust and requires less intensive computing resources than Mixing Plane Model
- Mostly used for radial machines

Frozen Rotor Model Contours Examples

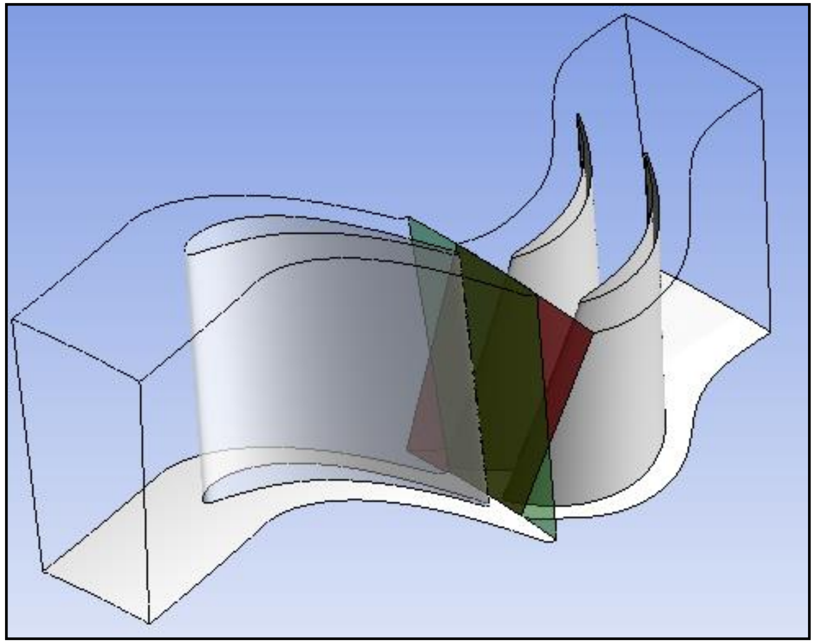
Blade wakes are fixed to a specific angular position



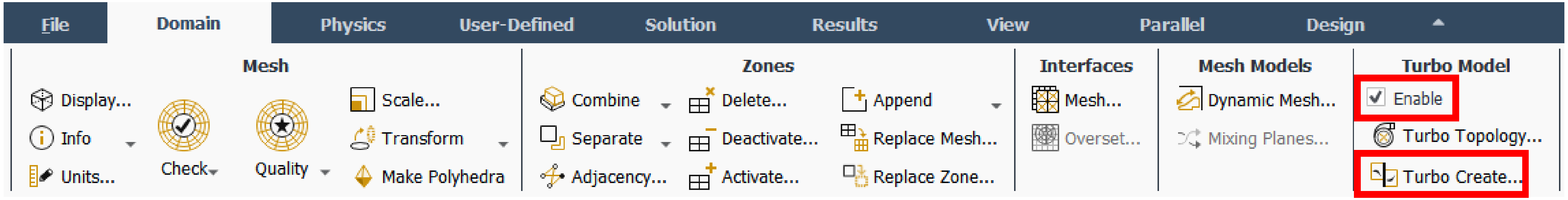


Frozen Rotor Model Characteristics

- Used when circumferential variation at interface is important, i.e. non-axisymmetric flow domains
  - impeller/volute, turbine/draft tube, propeller/ship, etc.
- Simulation is run as steady state
- For each rotating component, its cell-zone is set to rotational frame motion in the same way as for a single rotating component (shown in Lecture 02 of this course)
- Pitch change across Frozen interface is permitted
  - Flow is conservatively scaled to account for pitch change
- Errors can be significant when pitch ratio is large
  - Maintain pitch ratio near 1 to minimize errors
    - Example: Upstream blade count = 60, downstream blade count = 113
      - Pitch change of (360/113): (360/60) = 0.53
    - Use 1 upstream passage 2 downstream passages
      - Pitch change of 2*(360/113):(360/60) = 1.06

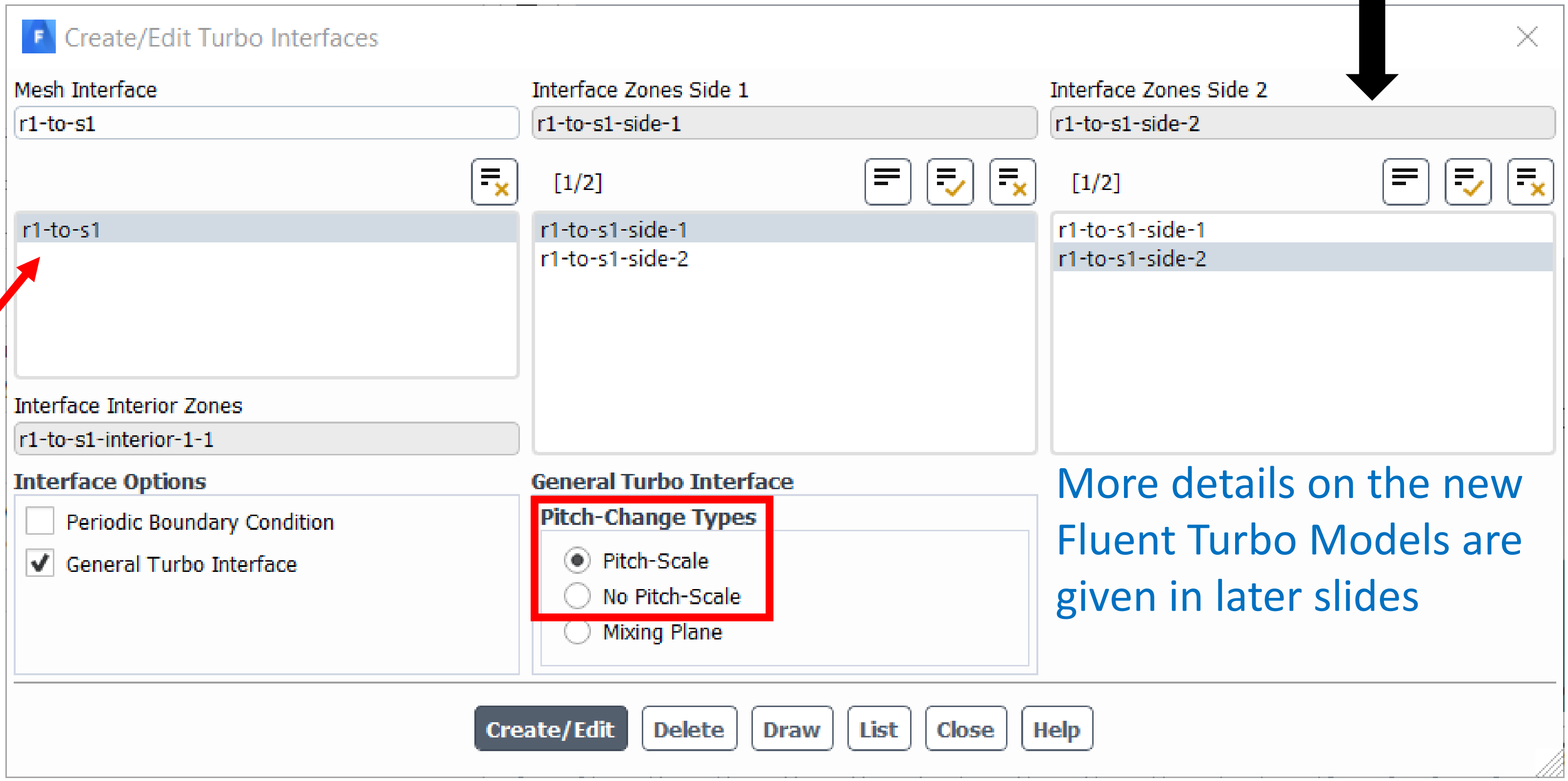


Frozen Rotor Model Setup in Fluent



- In the Domain tab Turbo Model group
  - Check “Enable”
  - Click “Turbo Create”
  - The mixing plane can be created in the Create/Edit Turbo Interfaces dialog
    - Enter name of “Mesh Interface”
    - Select “Interface Zones Side” 1 and 2
    - Select “General Turbo Interface”*
    - Depending on pitch ratio, as “Pitch-Change Type” select
      - “Pitch-Scale” (PS) or
      - “No Pitch-Scale” (NPS)

    PS and NPS are illustrated in slide 30

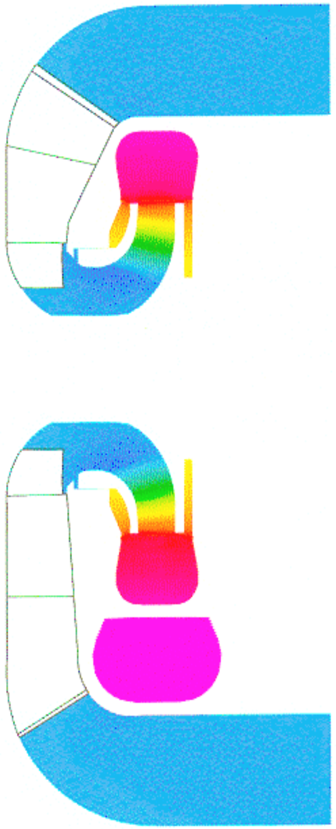


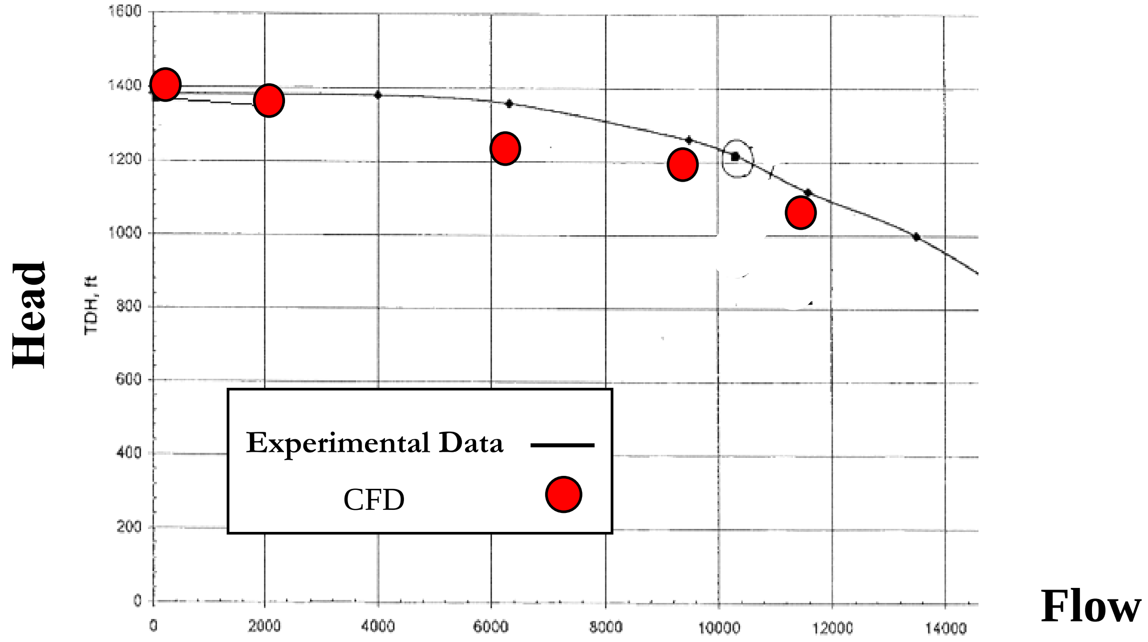
More details on the new Fluent Turbo Models are given in later slides

* Important Notice: the GTI interface must be non-conformal

Ansys

Frozen Rotor Example Calculation





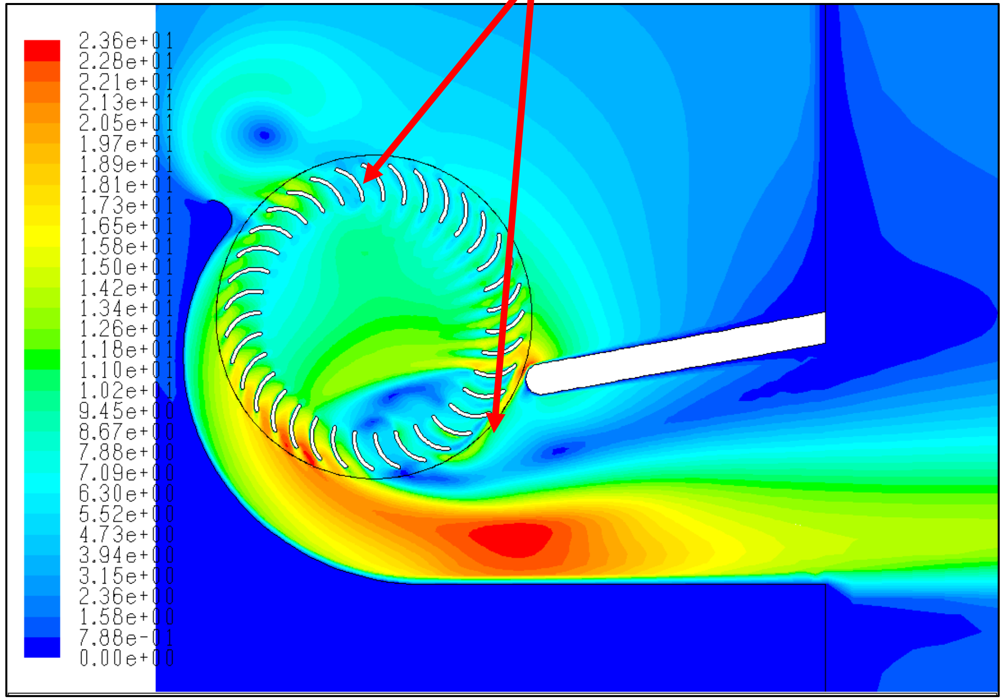
Ansys

Frozen Rotor Model Limitations

- Frozen Rotor model (FR) ignores the relative motions of the fluid zones with respect to each other, and thus do not account for fluid dynamic interaction between components
- Results may be dependent on the “frozen” position of the components
  - More pronounced when blade count is small
  - Try different rotor angular positions
- FR can produce misleading results in cases where the flow passes across the rotating domain (flow enters and leaves the outer boundary of the rotating domain), or there are significant flow reversals across the interface
  - Example: crossflow fans (illustrated in next slide)
- For cases where the FR approximation is not reasonable, you should use transient rotor stator instead

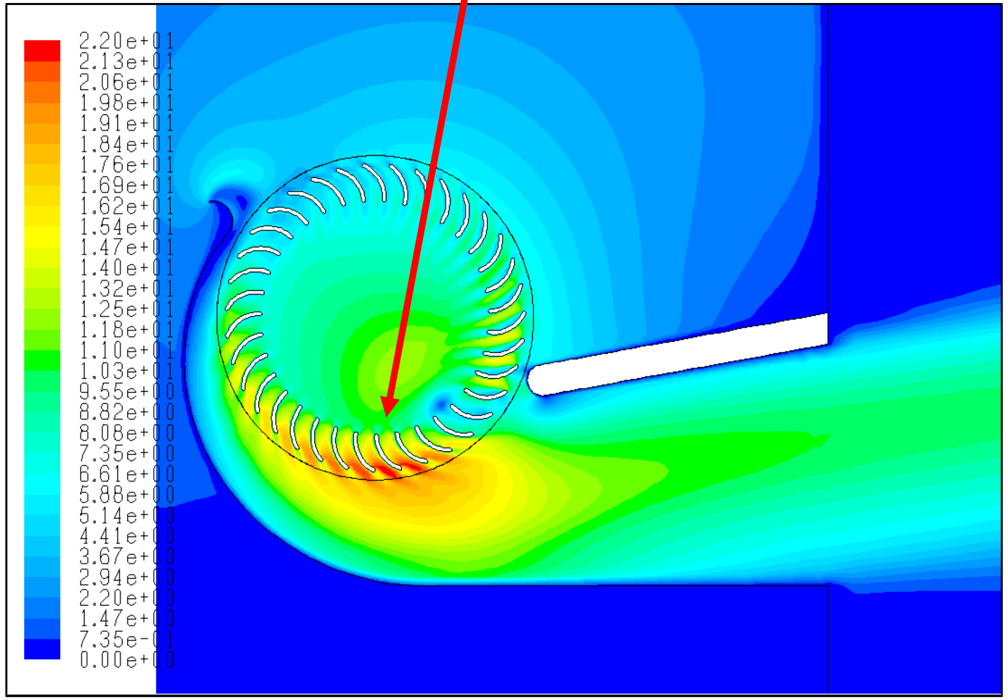
Crossflow Fan Example: Frozen Rotor vs Unsteady Sliding Mesh

Unphysical flow features



Frozen Rotor

Physical outlet vortex



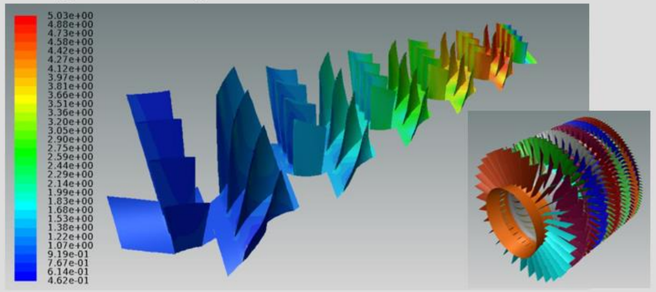
Sliding Mesh
(instantaneous solution)

New Fluent Turbo Models and Multi-Component Analysis Setup

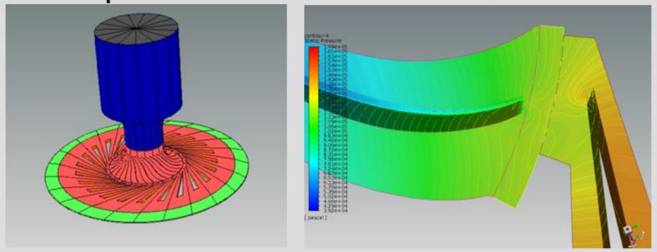
General Turbo Interfaces (GTI)

- **Interface-based** rotor/stator interaction models
  - General Turbo Interfaces (**GTI**) to connect rotor-rows to stator-rows
  - The bases of all (current and future) blade row models - steady & transient
  - Handles pitch-change - simulation on reduced geometry
  - Maintain implicit discretization - robustness & speed
  - Allow for switching between turbo models on the fly - consistent workflow
    - GTI must be non-conformal for this to work
  - The interfaces can handle most of the turbo configurations - wide applicability
    - Axial, Radial, Mixed
    - Small to very large pitch-ratio
    - Sector-to-sector , Sector-to-360 and 360-to-360
    - Secondary flow path: injections, bleeds & cavities

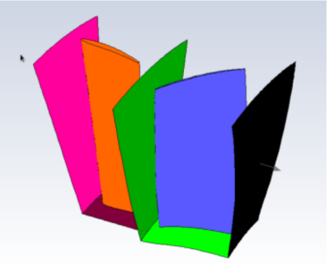
5 ½ Stage Axial Compressor



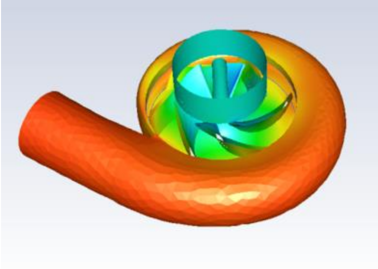
Radial Compressor



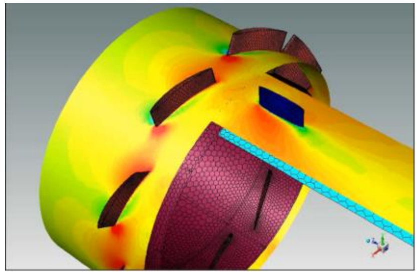
Sector-to-sector



360-to-360

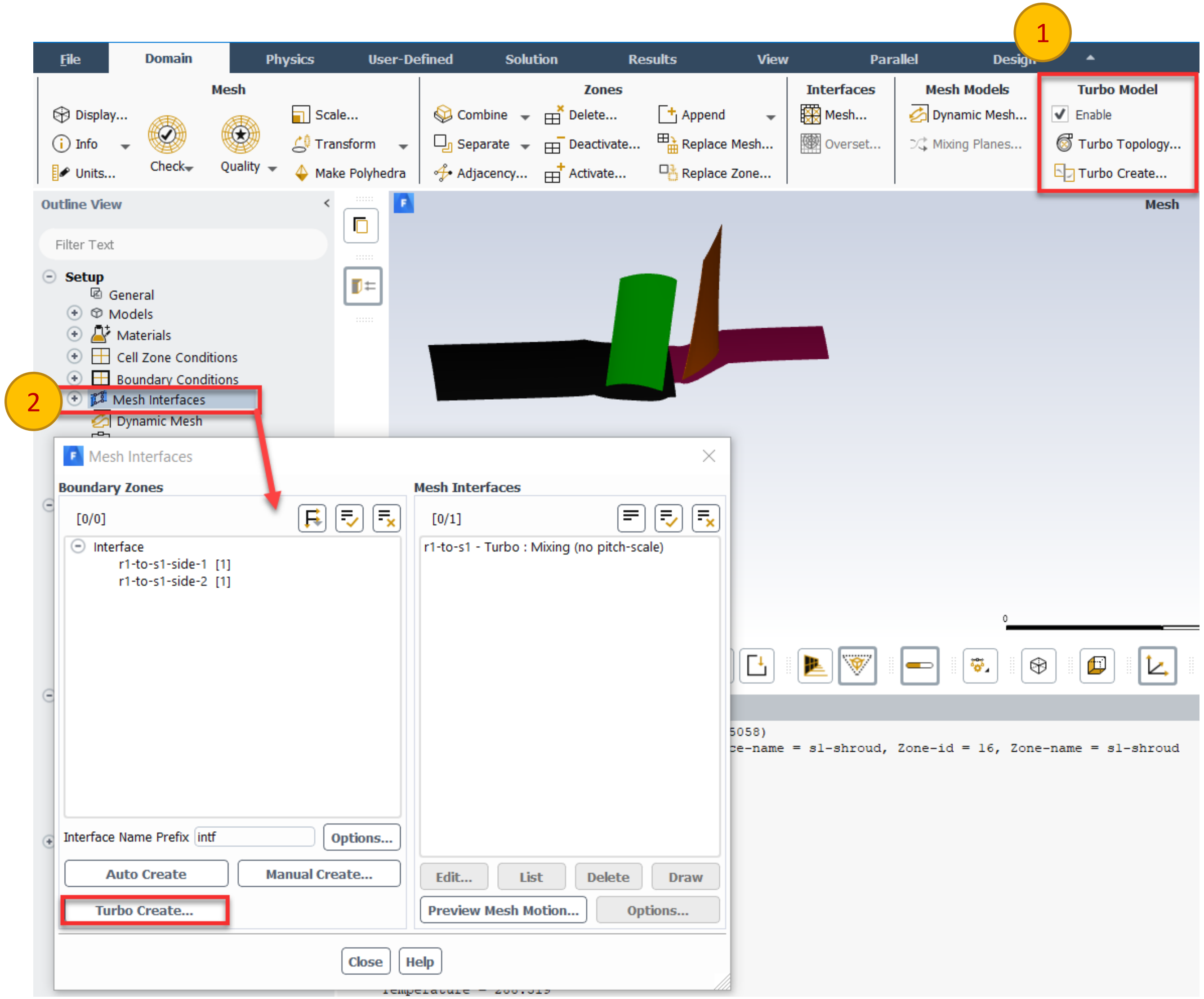


Sector-to-360



General Turbo Interface Creation

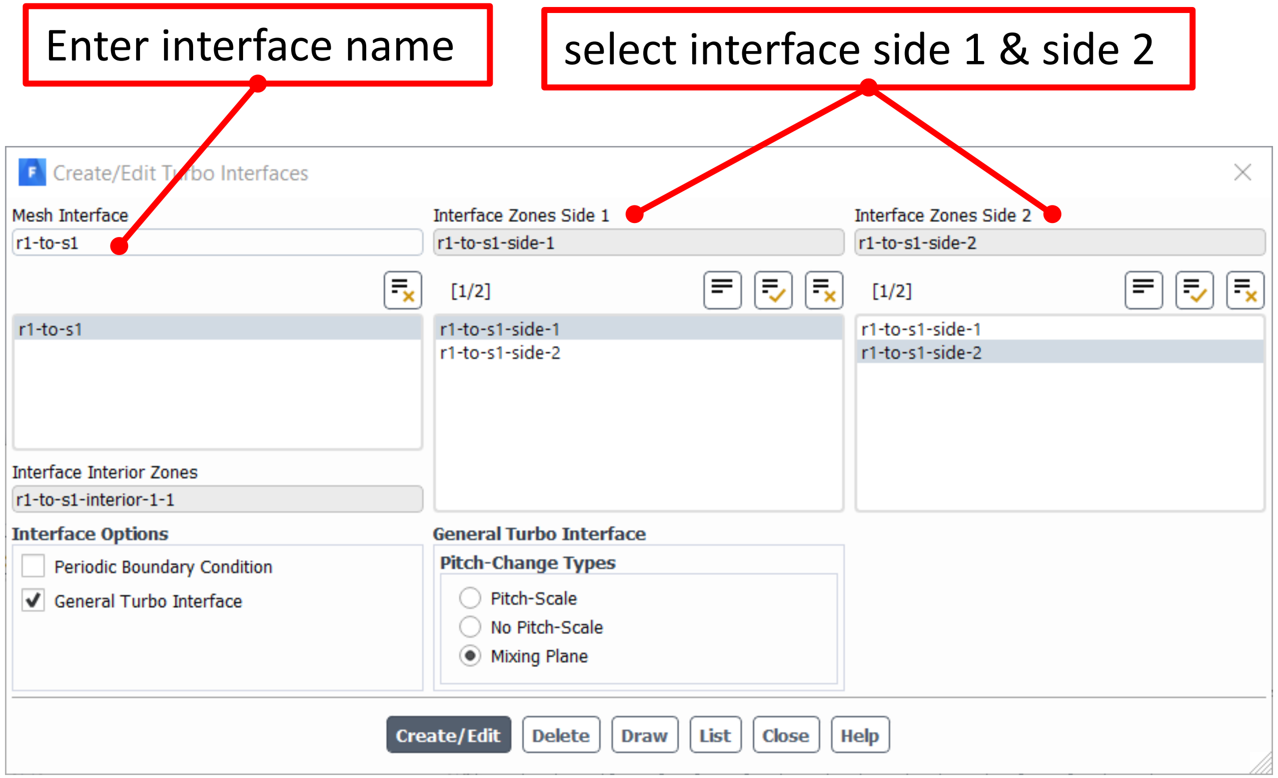
- General Turbo Interfaces are created using the new Turbo Create… panel
- Turbo Create… panel
  - One place to create/Edit turbo interface or switch turbo interface method
  - Accessed from Ribbon (1) or from under Mesh Interfaces (2)



Ansys

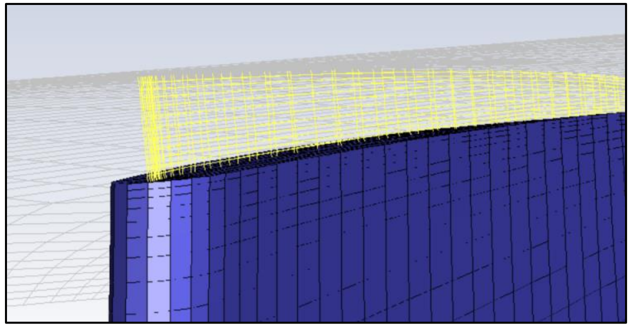
Creating Interfaces for Turbo Simulations

- Create essential interfaces for turbo simulations:
  - Tip Gaps
  - Non-conformal Periodic Interfaces
  - General Turbo Interfaces (GTI)
    - Selecting the Pitch-Change model

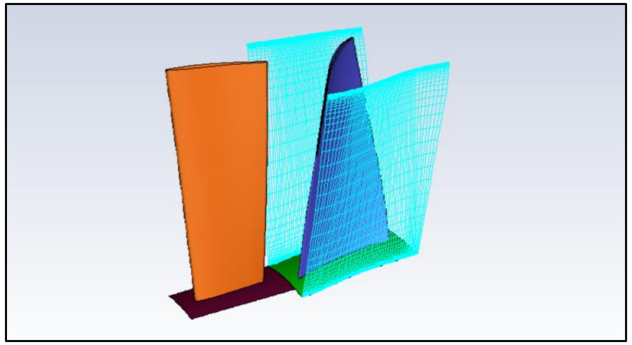


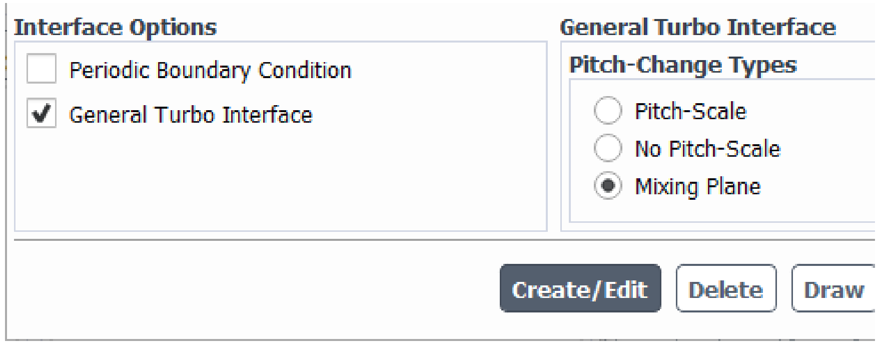
Interface Types from Turbo Create… Panel

Tip Gap: (For example at the tip of a rotor blade near a stationary shroud)
Do not select any interface option, then click on [create/edit]

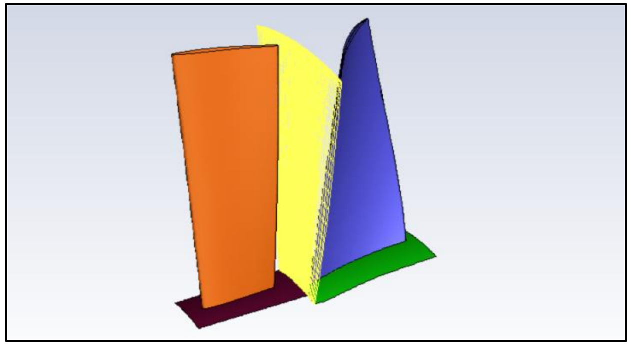


Non-conformal Periodic Interfaces:
Select Periodic Boundary Conditions,
Then click [create/edit].
Conformal Periodic Interfaces are set in boundary conditions as was shown in single rotating component, Lecture 02, also shown in next slide



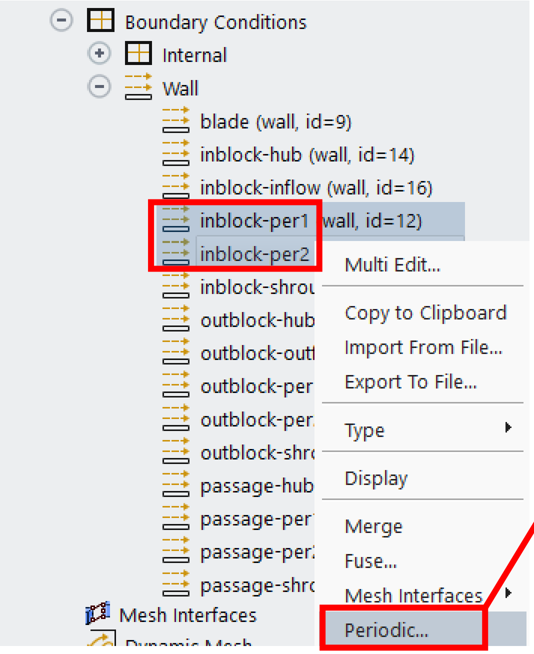


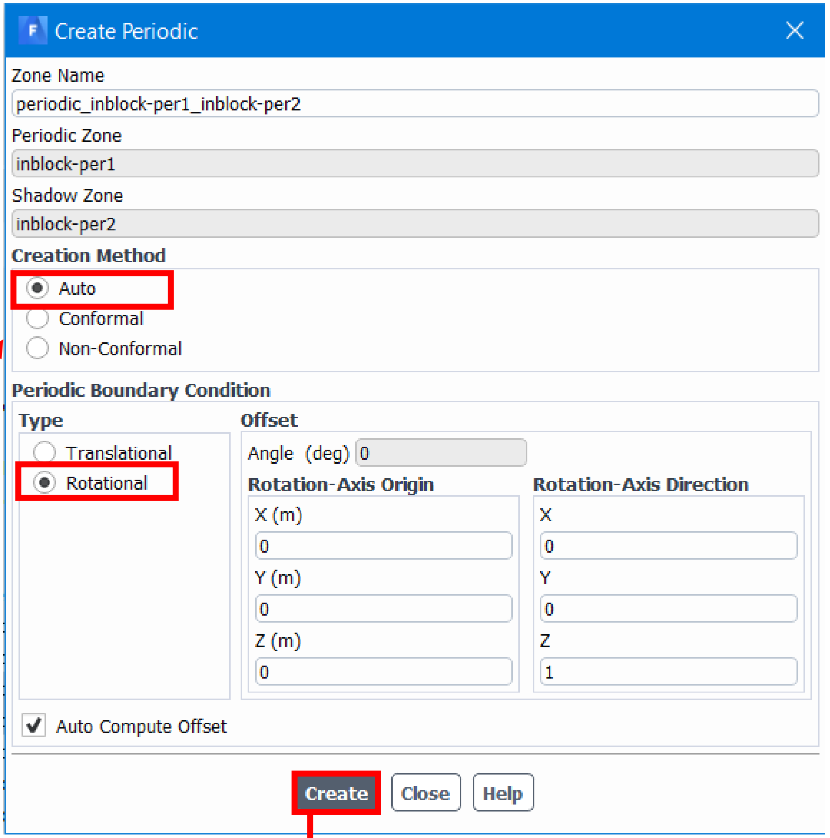
General Turbo Interfaces:
Select General Turbo Interfaces,
Select Pitch-change method,
Then click on [create/edit], see slide 30 for details

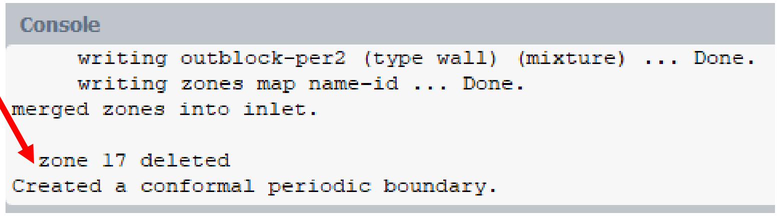


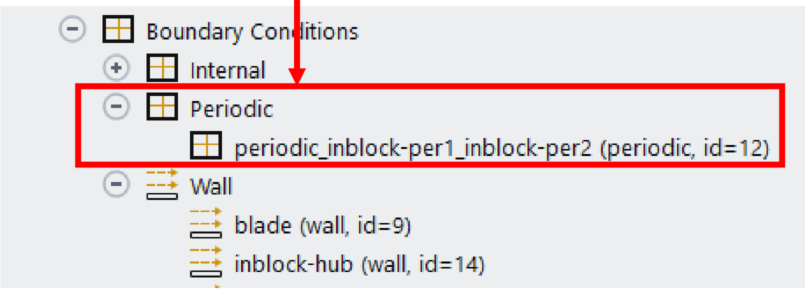
Creating Periodic Interfaces (same as for single rotating component)

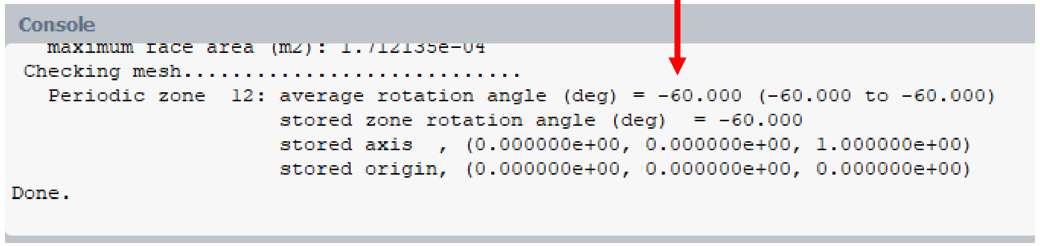
- Select the two sides of the rotational periodic boundaries under Boundary Conditions
  - Depending on their name they might initially reside under Wall boundaries
- RMB> Periodic
- Select Rotational
  - Rotation-Axis is automatically set to the cell-zone rotation axis previously set
- Check in console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











Use of General Turbo Interfaces

- Mixing-Plane (MP)

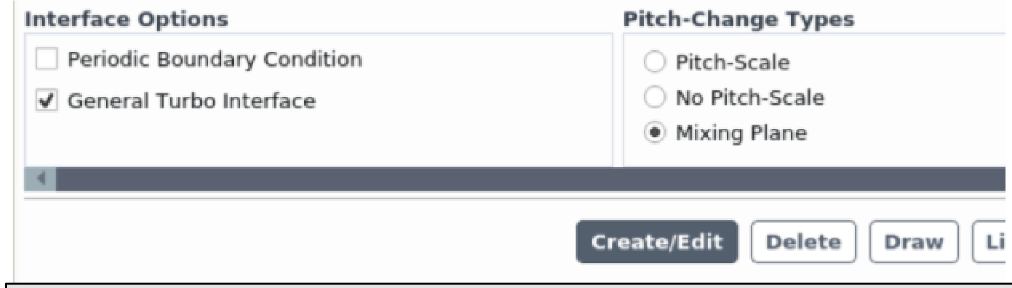
- Frozen Rotor (FR)
- Transient Rotor/Stator (TRS)
  (Sliding Interface)

With Pitch-Change

Pitch-Scale (PS)

No-Pitch-Scale (NPS)

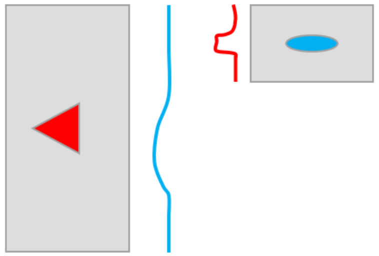
- Pitch-Scale is not available for
  - pitch-ratio > 2 or
  - Connecting 360 to 360 interfaces
- Use No-Pitch-Scale in such cases



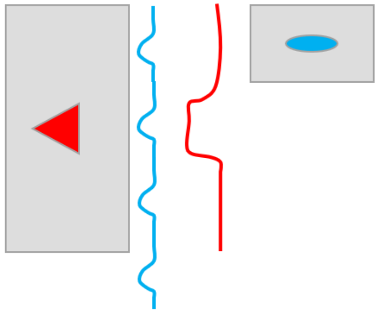
In FLUENT you do not explicitly select Frozen Rotor or Transient Rotor/Stator interface. They are implicitly defined by <u>the Fluid zone motion specification</u>.

PS or NPS can be used as FR or TRS
Depending on rotating zone motion specification

- **FR** : if Rotor fluid zone is specified as ***Frame Motion***
- **TRS**: if Rotor fluid zone is specified as ***Mesh Motion***



Pitch-Scale
Reasonable approximation for small pitch-ratio.
Unrealistic approximation as pitch-ratio get larger.



No-Pitch-Scale
More reasonable for large pitch-ratio

Similar Model to CFX Profile-Transformation

Ansys

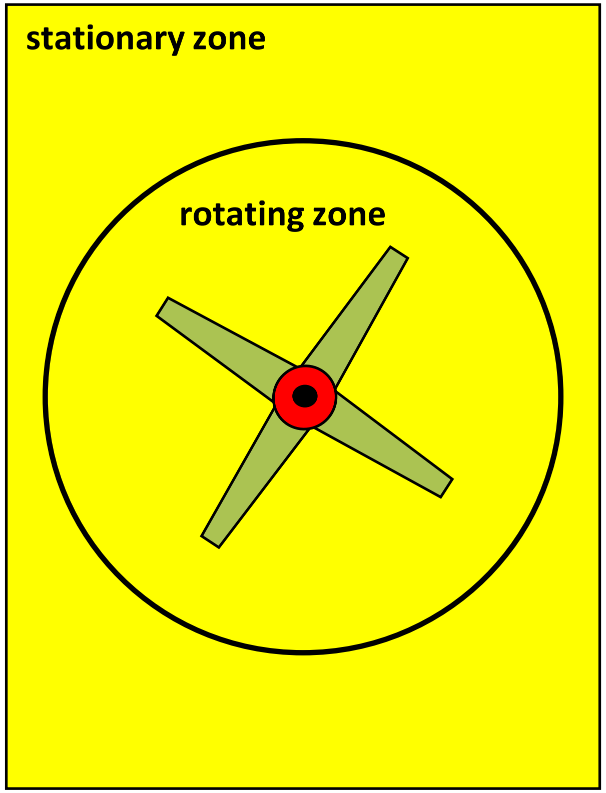
Multi-component Fluent Turbo Setup

- Follow proper sequence of general setup before you setup the General Turbo Interfaces (GTI)
  - Make sure axis defined properly in Fluid Zone for rotors and stators
    - For stators, the axis of rotation is used for defining the rotational periodic boundaries
  - Make sure periodic boundaries (conformal or non-conformal) are created before you create GTI
  - Make sure tip gap interfaces are created before you create GTI
- All settings for any rotating component and all general physics and solver settings should follow the rules and best practices given in lecture 02 “Single Rotating Component Analysis”
  - Operating conditions, gravity, turbulence model selection, energy equation, rotation axis, periodic interfaces, boundary conditions, solver settings, initialization procedure, time scale factor, report plots, …
  - Important Note: Currently, FMG initialization is not compatible with all General Turbo Interfaces and should be avoided

General Turbo Interfaces Placement Best Practices

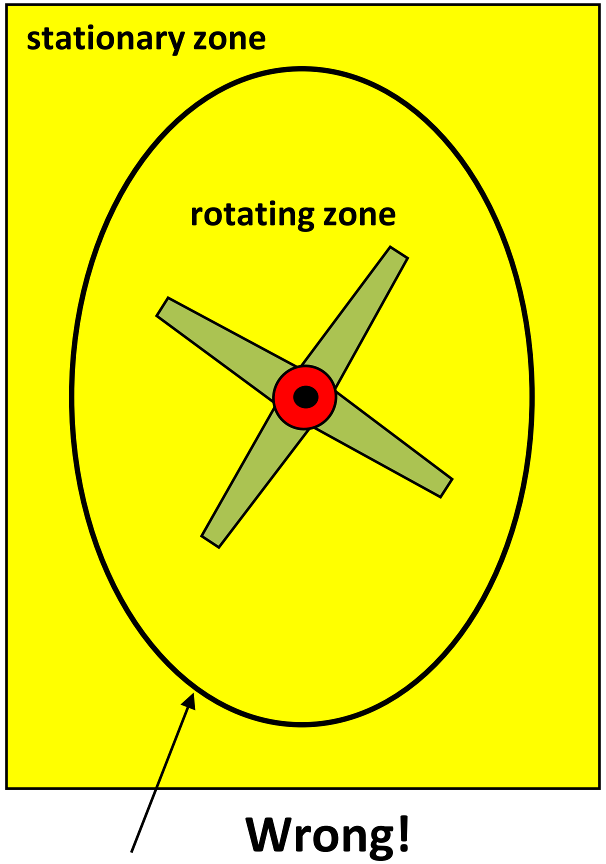
Ansys

Geometric Requirement



Correct

Make sure interface is a surface
or revolution about rotational axis

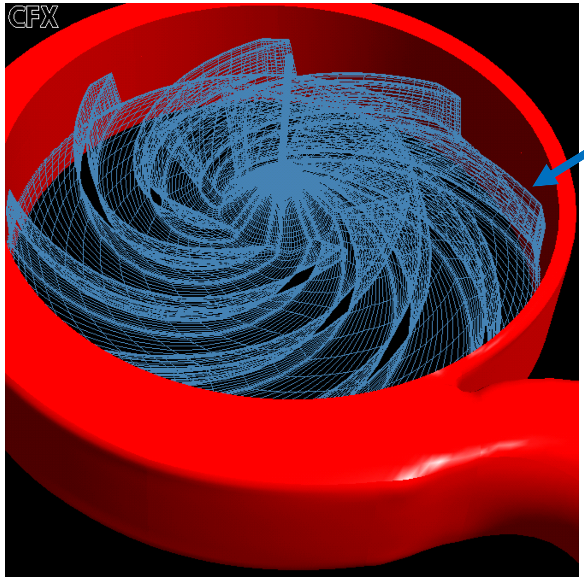


Wrong!

Interface is not a surface
or revolution

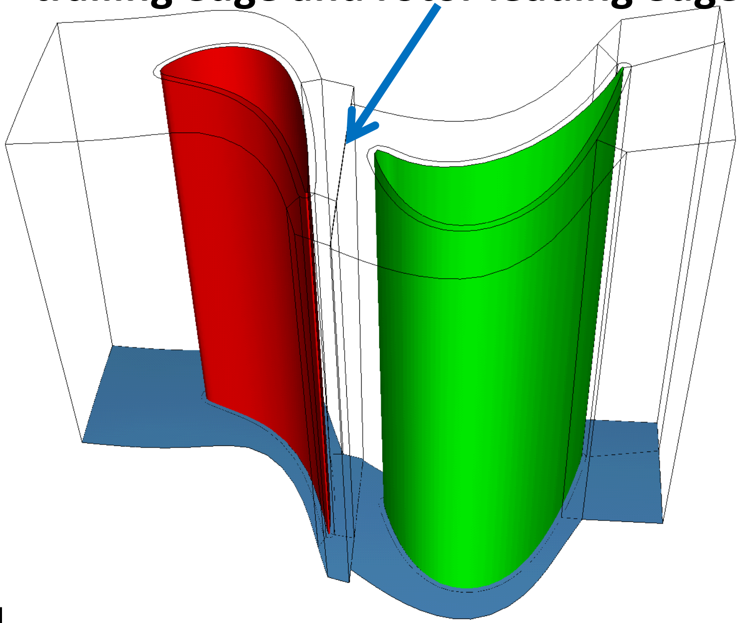
Interface Placement – General Practices

- For interfaces contained within an enclosed system (e.g. blower within a casing), choose a position midway between the outer most point on the moving zone and the closest stationary wall (e.g. tip of a pump rotor)
- For interfaces embedded in a large domain (e.g. axial fan in a room), choose a location where flow conditions are relatively uniform at the interface
  - Usually close to the rotor/impeller
  - If R*omega is supersonic in the domain, if possible, move the interface to a smaller radius



Interface midway between rotor tip and casing

Interface midway between vane trailing edge and rotor leading edge

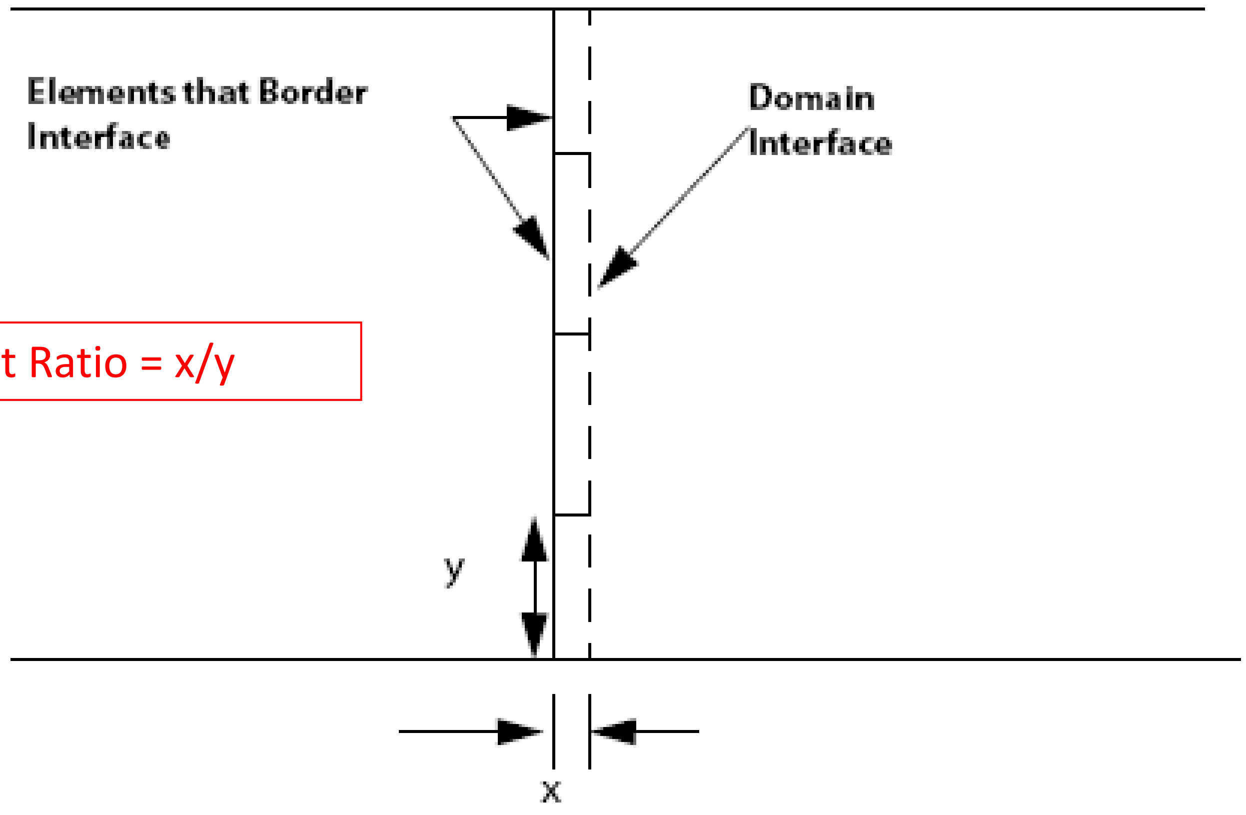


Ansys

Aspect Ratio Considerations

- To minimize numerical errors, element aspect ratio at the interface should be between 0.1 and 10

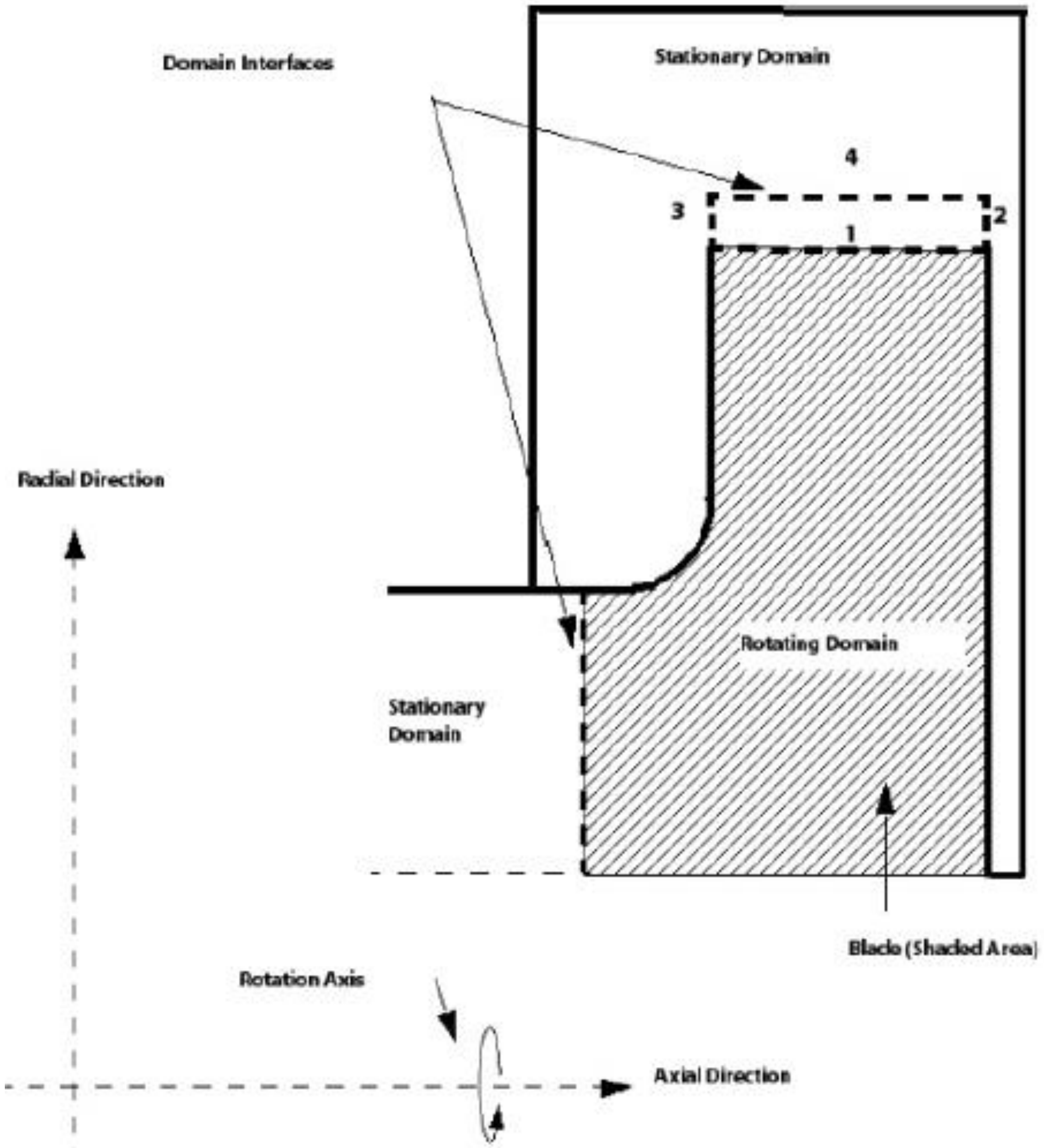
Aspect Ratio = x/y



Ansys

Interfaces Near Blade Tip

- Although it is convenient to place a domain interface at the blade edge (1), this can result in convergence difficulties
- A better arrangement is to extend the rotating domain away from the blade edge Domain Interfaces can then be created at (2), (3), and (4)
  - Make sure (4) is not too far, to avoid supersonic relative Mach numbers



Using Multiple Domain Interfaces

- It is possible to have a combination of General Turbo Interface models within a single problem definition
  - For example, in a stator/rotor/stator configuration, the first Fluid-Fluid interface could be a Mixing Plane or Frozen Rotor interface, and the second interface could be a Transient Rotor-Stator interface.
    - This might make sense if you are only interested in accounting for the transient interactions between the rotor and the downstream stator row.

Summary

- Multi-Component Analysis is best performed using Multiple Reference Frames
- Two steady-state frame change models available:
  - Mixing plane model
    - Works best for problems where circumferential averaging between blade rows is a reasonable approximation
      - Axial compressor, axial turbine, etc.
    - Different pitch angles can be used, one blade row per passage is all that is needed
  - Frozen rotor model
    - Works best for cases with asymmetry in the flow
      - Pump in a volute, Fan in a volute, etc.
    - Target pitch ratio as close to 1 as possible, or use the no-pitch-scale option

End of presentation