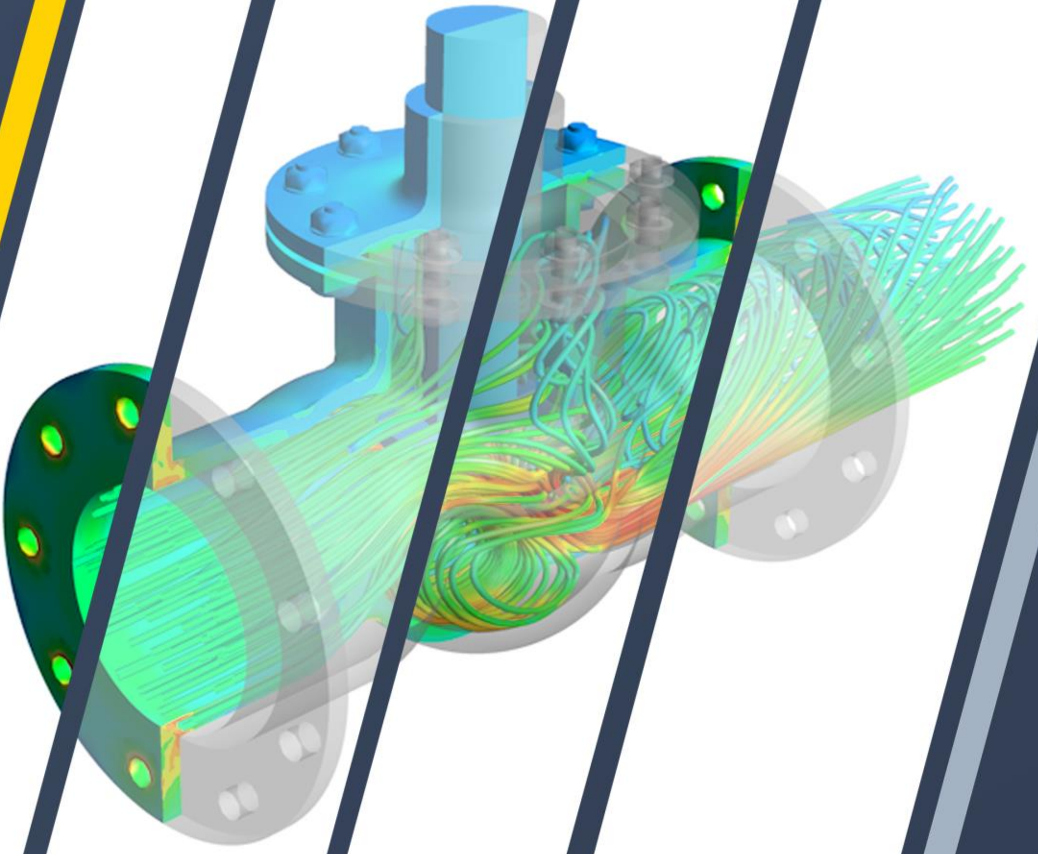
Workshop 04.2: Axial Fan Stage (MFR)

ANSYS CFX Rotating Machinery Modeling

Release 2019 R3



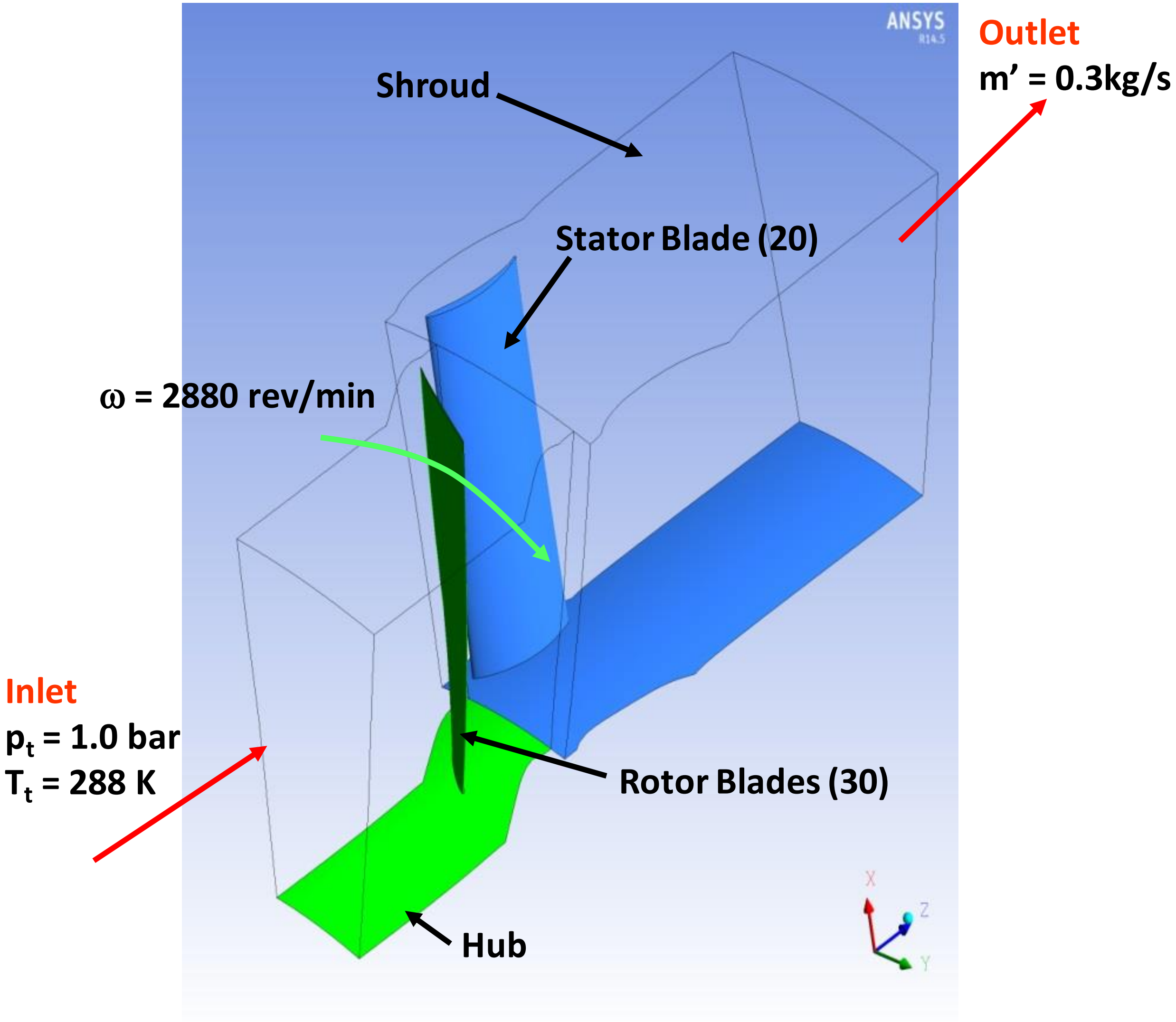
Introduction

Workshop Description:

- This Workshop deals with an Axial Fan stage operating at 2880 rpm. The working fluid is Air Ideal Gas and the flow is assumed to be steady.
- Due to rotational periodicity a single blade passage will be modeled.

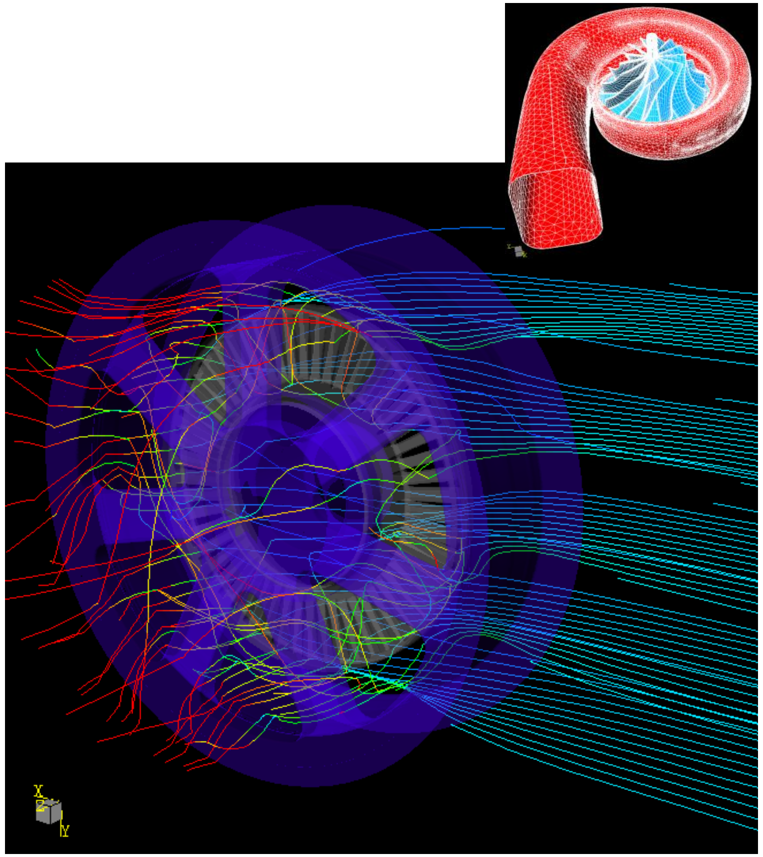
Learning Aims:

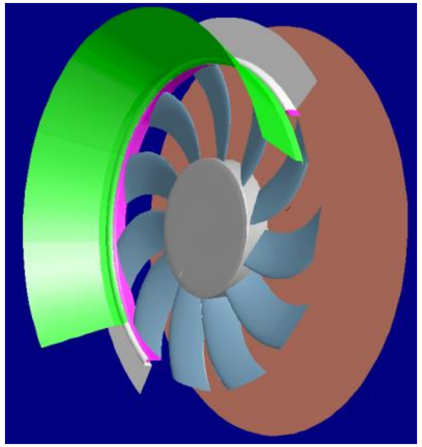
- Working outside Workbench.
- Working with rotating domains.
- Basic setup using multiple frames of reference.
- Turbo-specific post-processing.

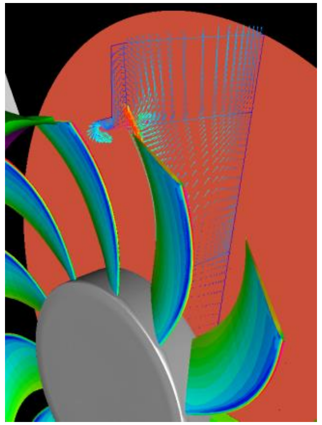


Multiple Frames of Reference

- Axial, radial & mixed turbo machines, mixing vessels
- Diverse component installations
  - – Gaps, tip clearance, …
- Available models:
  - – Stage: steady calculation, circumferential averaging
  - – Frozen Rotor: steady calculation, frozen blade position
  - – Transient Rotor Stator (TRS): transient calculation, rotating interface mapping, rotor-stator interaction
  - – Transient Blade Row methods:
    - ➢ Time Transform - Inlet Disturbance, Stage TRS
    - ➢ Fourier Transform - Inlet Disturbance, Stage TRS & Blade Flutter
    - ➢ Profile Transform – TRS
    - ➢ Harmonic Analysis







Pre-processing Goals

- Using stand-alone CFX mode (outside Workbench).
- Launch CFX-Pre from CFX-Launcher.
- Define stationary and rotating domain.
- Define boundary conditions.
- Define rotor stator interface.
- Set up monitor points using simple expressions.

- **Launch the CFX Solver Manager.**
- **Check solution process.**

- **Launch CFD-Post.**
- **Initialize turbo post-processing.**
- **Define turbo surface.**
- **Use turbo coordinates for contour plots.**

Running Outside vs Inside Workbench

Outside	Inside
Less computational overhead	Simplifies the workflow
Simpler directory / file structure	Easy to link to other *Analysis Systems* and *Components*
But no direct association between geometry, mesh, setup and results files	Easier to update a project when a change is made
No built-in automation for parametric studies	Needed for optimisation using DesignExploration

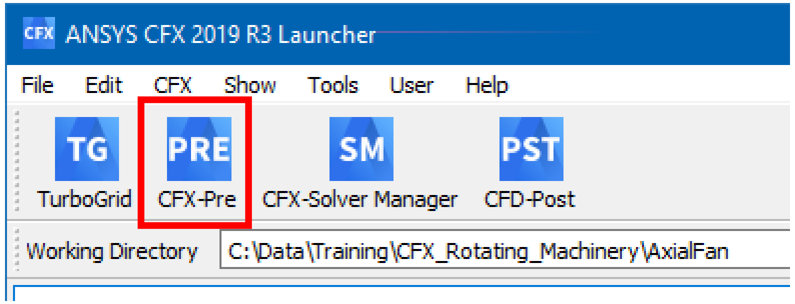
Start CFX-Pre

Copy the CFX mesh files , *Rotor.gtm* and *Stator.gtm* provided with the workshop inputs in a working folder of your choice.

Start CFX-Launcher

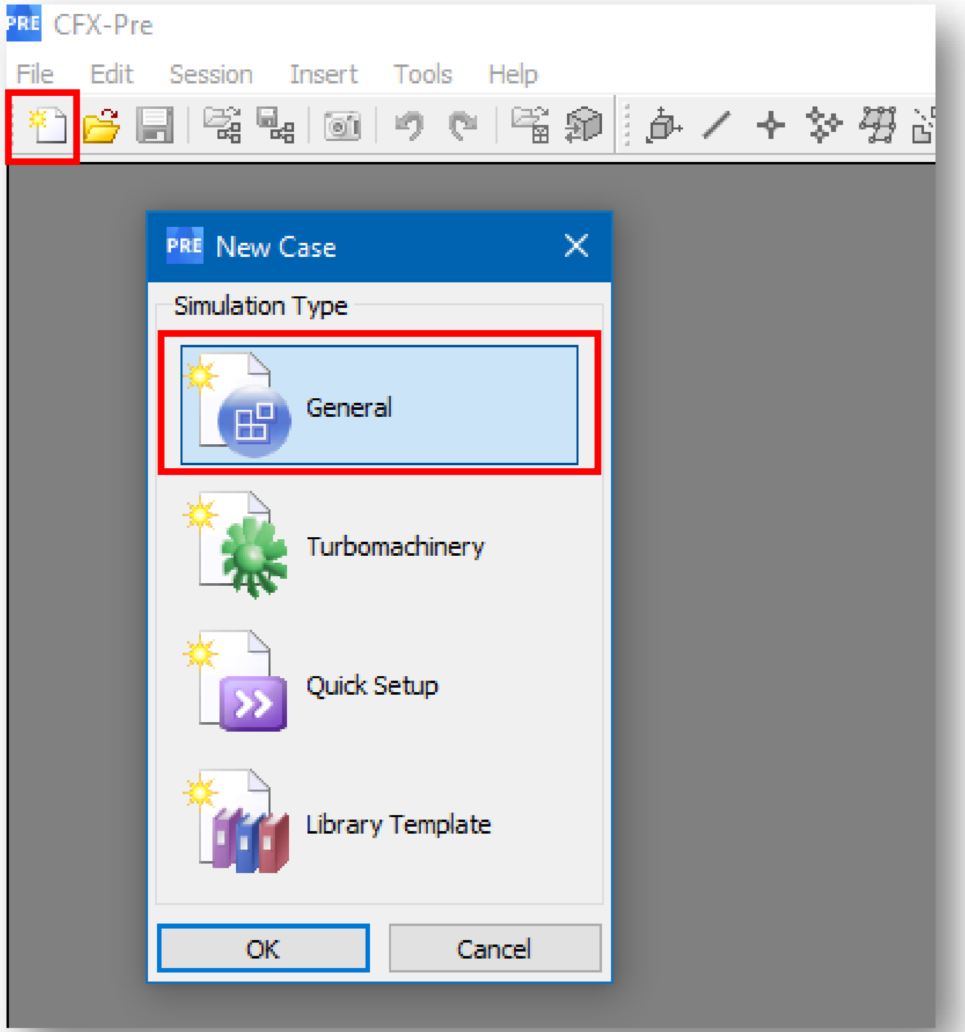
- *Start > All Programs > ANSYS 2019 R3 > CFX 2019 R3*
- Browse to your working folder.

Start CFX-Pre from CFX-Launcher.



Choose *New Case* and then *General*

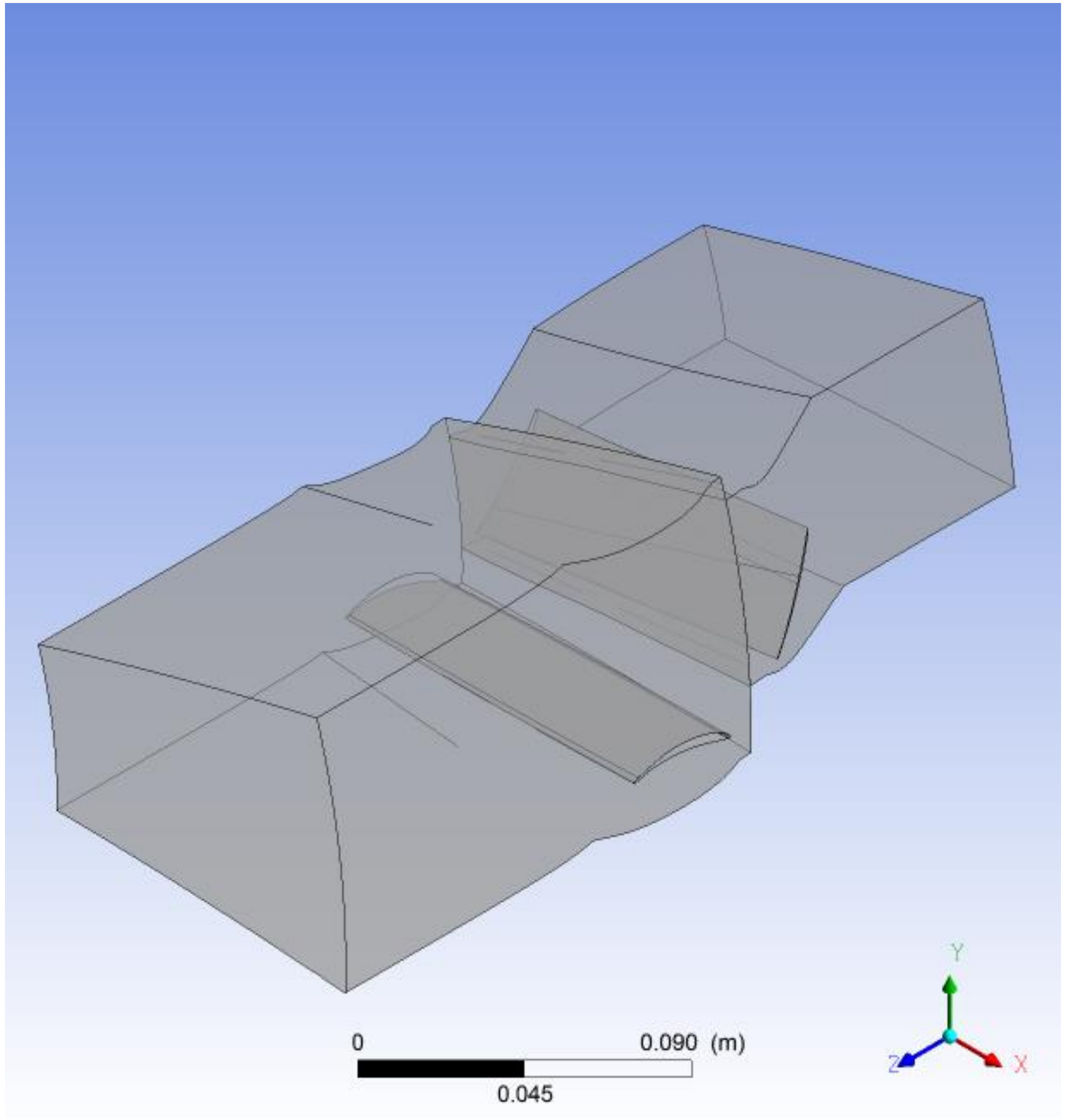
- The Turbomachinery mode could be used for an automated set-up. We will use General to understand the fundamentals.



Import Mesh

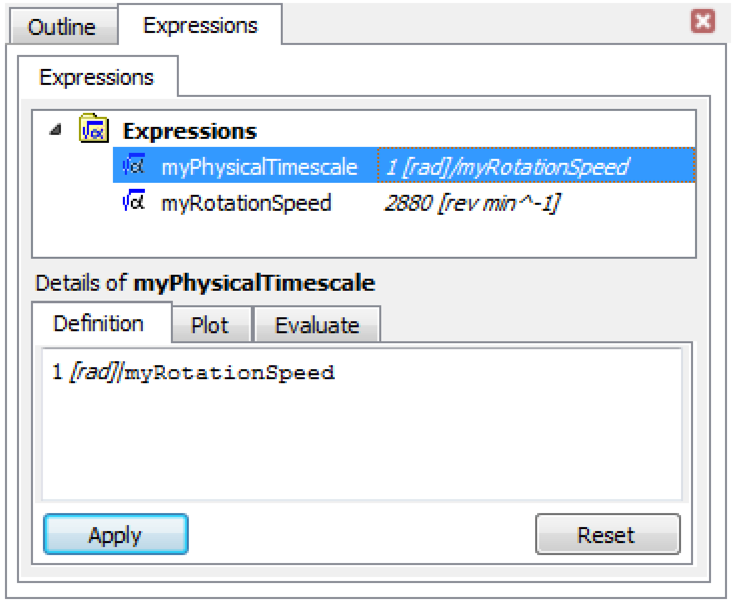
- RMB on *Mesh* in the *Outline* and *Import Mesh > CFX Mesh*
- First import the mesh, *Rotor.gtm*
- Then import the mesh, *Stator.gtm*
  - The order is important!
  - These meshes are in the *CFX-Mesh* format.
    - ➢ They were created in ANSYS TurboGrid and exported to this format.

- *File > Save Case As*: *AxialFanStage.cfx*



Define Expressions

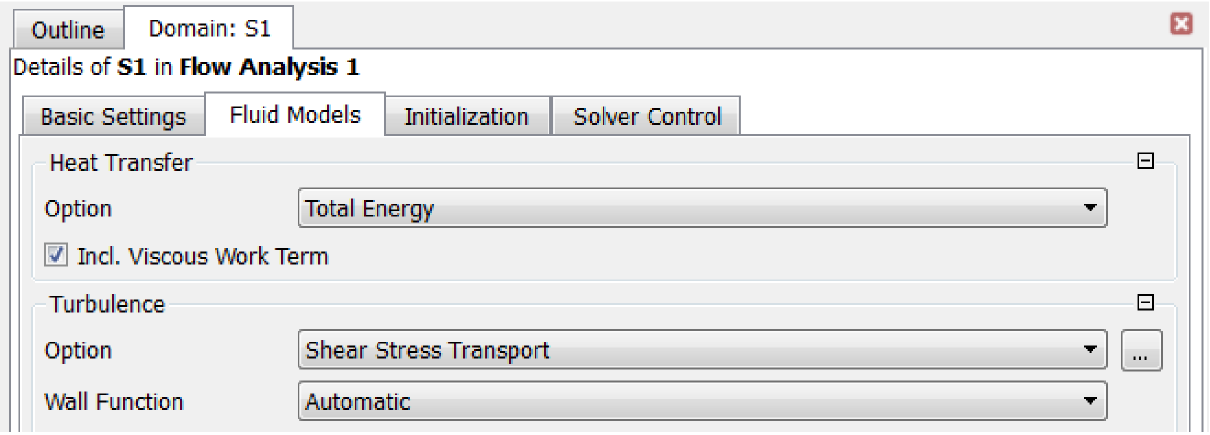
- Create expressions that will be used later to define the speed of rotation and the physical timescale for convergence control.
  - `myRotationSpeed        = 2880 [rev min^-1]`
  - `myPhysicalTimescale    = 1 [rad]/myRotationSpeed`



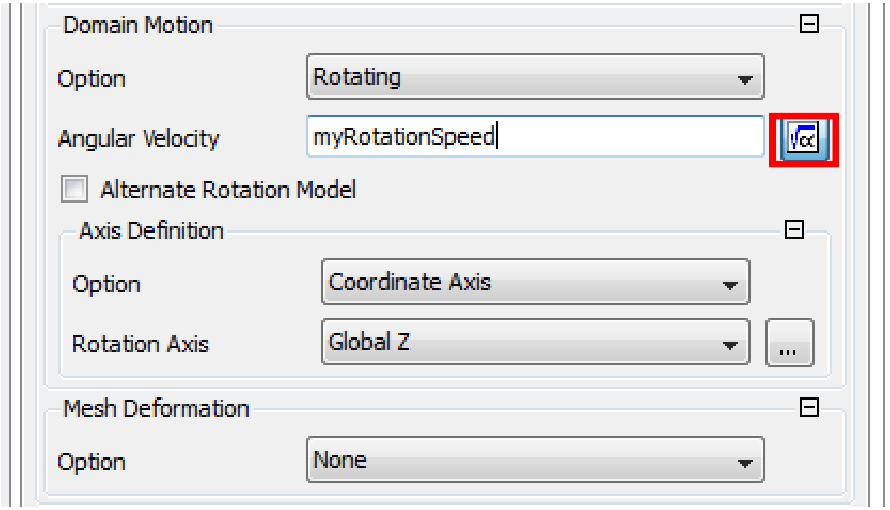
- For rotating machinery, the reciprocal of the rotational speed is often used to define the timescale.

Create Domain: Stator (S1)

- Create a stationary Domain called *S1*.
  - Located on *Passage 2*.
  - Fluid is *Air Ideal Gas* with *Ref. Pressure* of *1[bar]*.
  - Select *SST Turbulence Model* and *Total Energy* for *Heat Transfer*.
    - ➢Check *Incl. viscous Work Term*

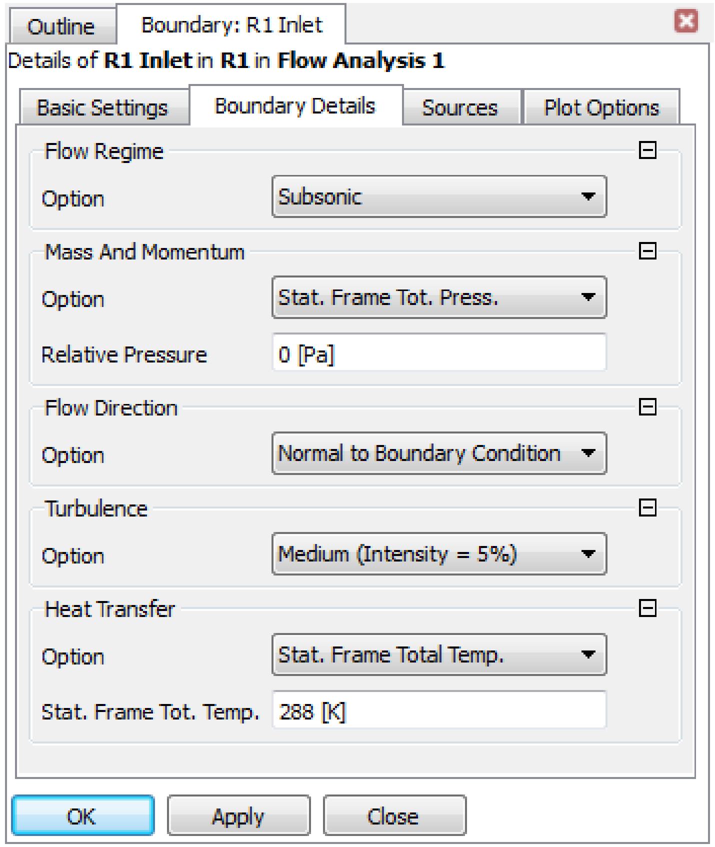


- Create a rotating Domain called *R1*.
  - Located on *Passage*.
  - Change the *Option* for *Domain Motion* to *Rotating*.
  - Define the *Angular Velocity* using the expression *myRotationSpeed* and select *Global Z* as the *Rotation Axis*.



Create Boundary: Inlet (in R1)

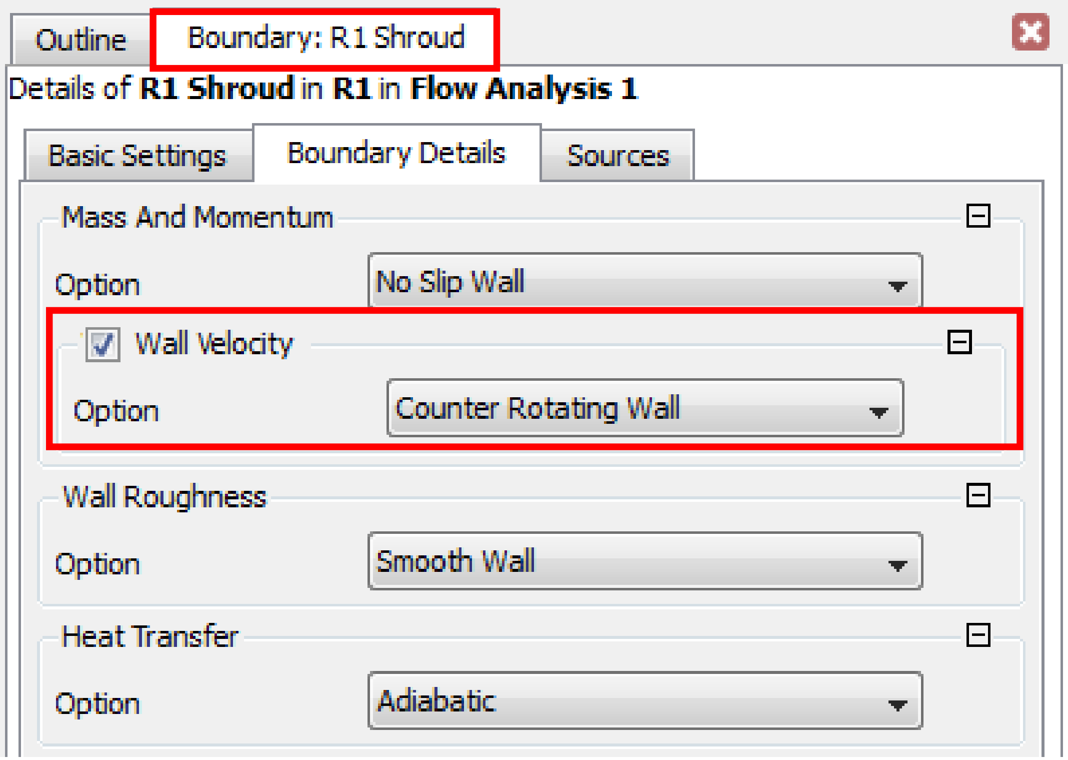
- In *Domain R1*, create an *Inlet* boundary called *R1 Inlet*.
  - Located on *INFLOW*.
  - *Frame Type* is *Stationary* so that conditions set are relative to the absolute/stationary reference frame.
  - For *Mass And Momentum* select:
    - *Stat. Frame Tot. Press. Option*.
    - *Relative Pressure of 0 [Pa]*.
    - *Normal to Boundary Condition* for the *Flow Direction*.
  - For *Turbulence* choose the *Medium (Intensity = 5%) Option*.
  - *Stat. Frame Total Temperature = 288 [K]*.



Create Boundary: Blade, Hub & Shroud (in R1)

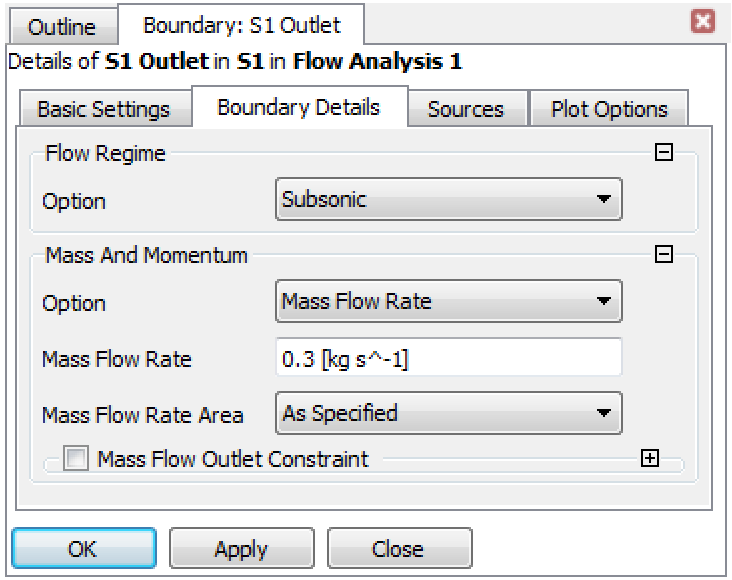
The *Blade* and the *Hub* rotate with the domain, but the *Shroud* is stationary.

- Create a wall boundary called *R1 Blade* in *Domain R1*.
  - Located on *BLADE*.
  - *Frame Type is Rotating*.
- Similarly define a wall boundary called *R1 Hub*.
  - Location = *HUB*
- Create a wall boundary called *R1 Shroud*.
  - Location = *SHROUD*
  - Check the box for *Wall Velocity*.
  - Set the *Option* to *Counter Rotating Wall* (Counter Rotating in rotating system = stationary in global system).

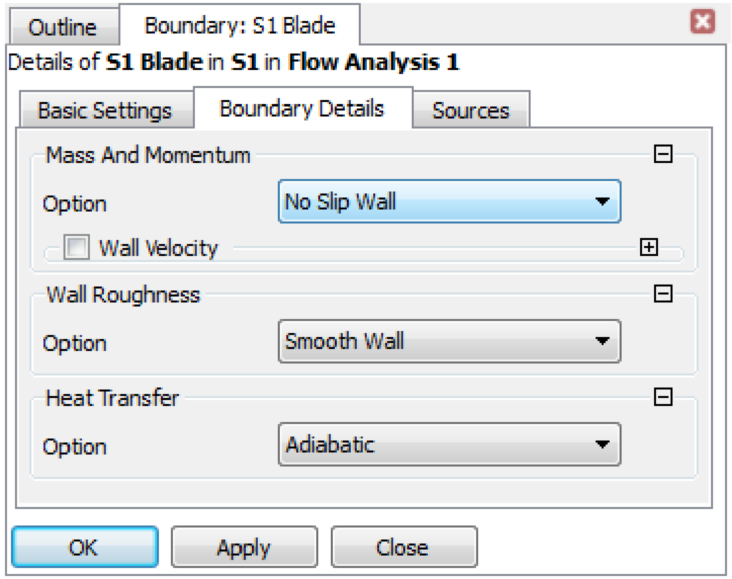


Create Boundary: Outlet and Walls (in S1)

- Create an *Outlet* boundary in *Domain S1* called *S1 Outlet*.
  - Located on *OUTFLOW 2*.
  - *Mass Flow Rate = 0.3 [kg/s]*.
  - *Mass Flow Rate Area = As Specified*



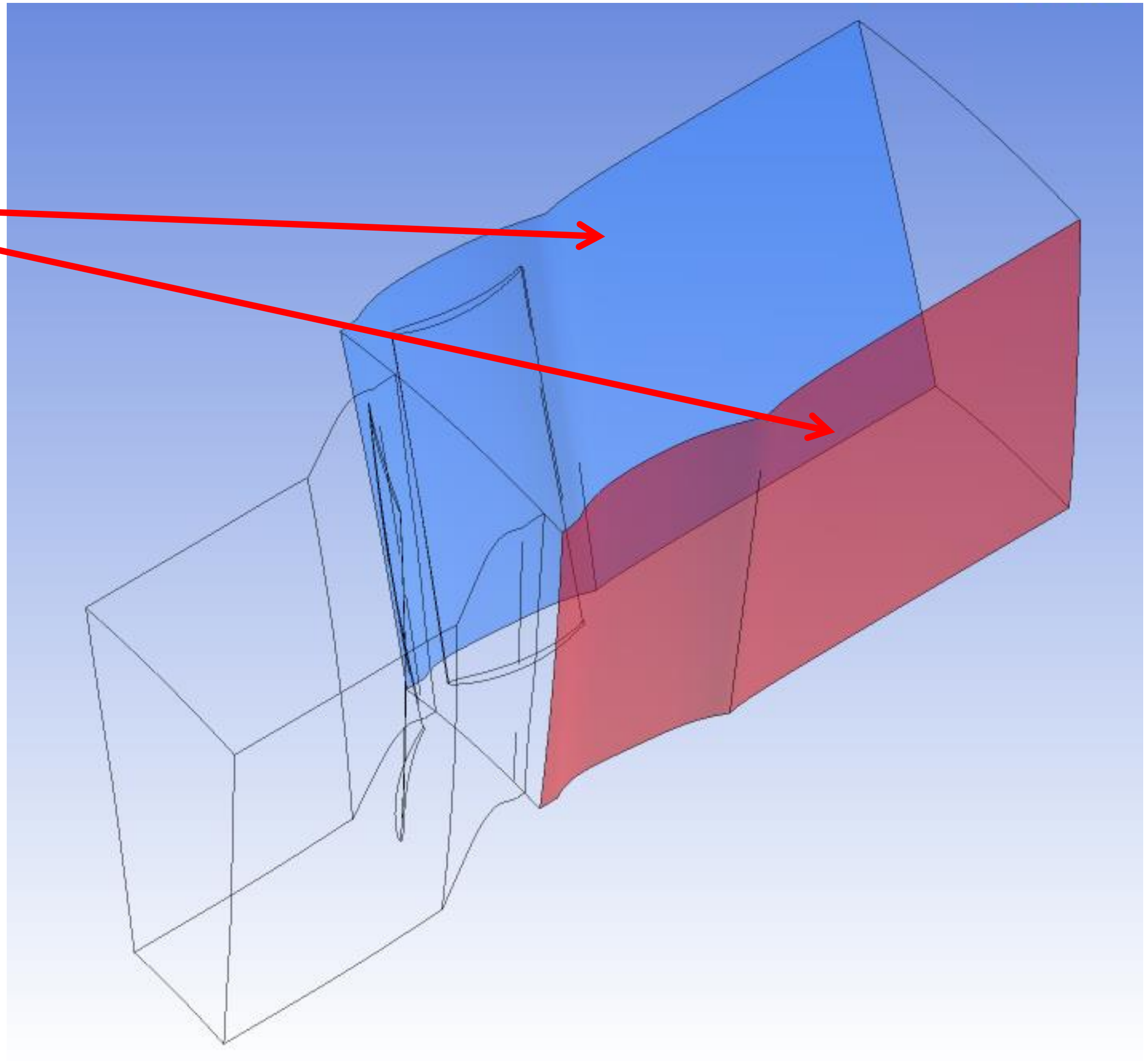
- Create the following adiabatic, no-slip wall boundaries in S1.
  - *S1 Hub* located on *HUB 2*.
  - *S1 Shroud* located on *SHROUD 2*.
  - *S1 Blade* located on *BLADE 2*.



ANSYS

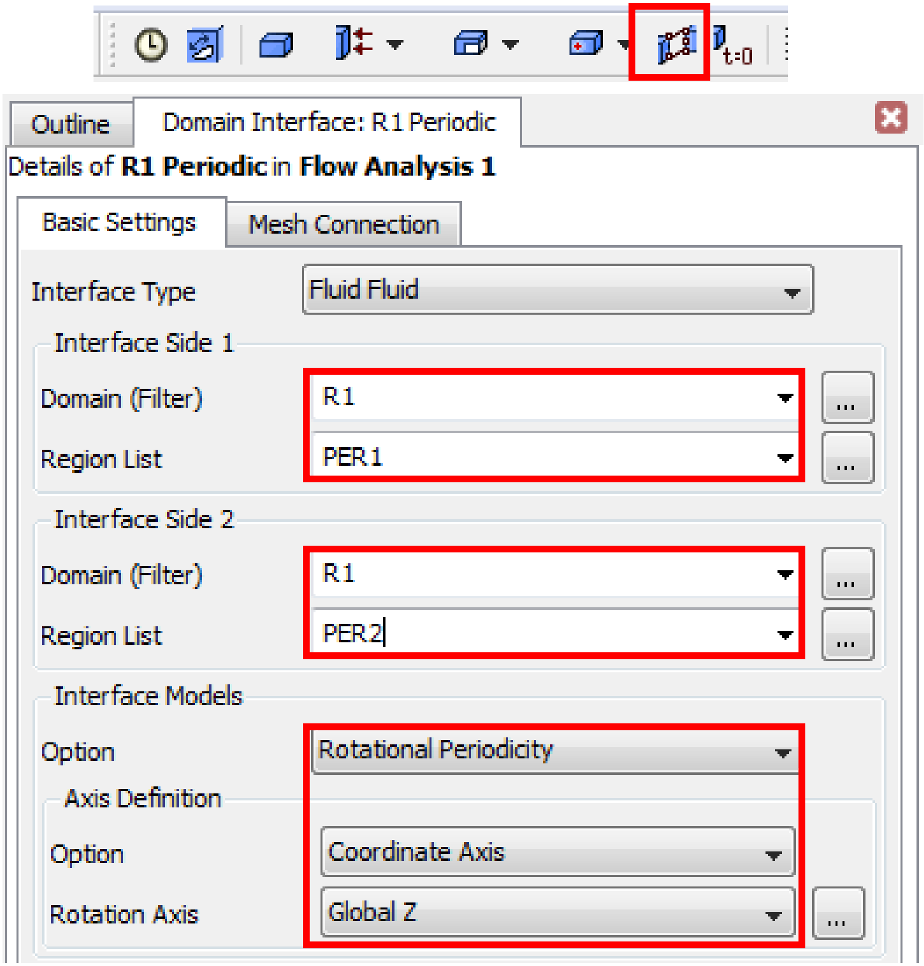
Periodic Interfaces

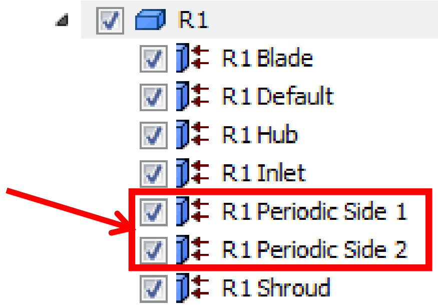
Periodic Interfaces are defined to connect the two sides of each domain.



Periodic Interfaces

- Create a *Domain Interface* named *R1 Periodic*
  – *Interface Type = Fluid Fluid*
  – *Domain (Filter): R1* (for both sides)
  – *Region List*:
    ➢ *PER1* (Side 1)
    ➢ *PER2* (Side 2)
  – *Interface Model: Rotational Periodicity*
  – *Rotation Axis: Global Z*
- Similarly create a periodic interface named *S1 Periodic*
  – This time set *the Domain (Filter)* to *S1* (for both sides)
  – *Region List:*
    ➢ *PER1 2* (Side 1)
    ➢ *PER2 2* (Side 2)
  – *Interface Model: Rotational Periodicity*
  – *Rotation Axis: Global Z*

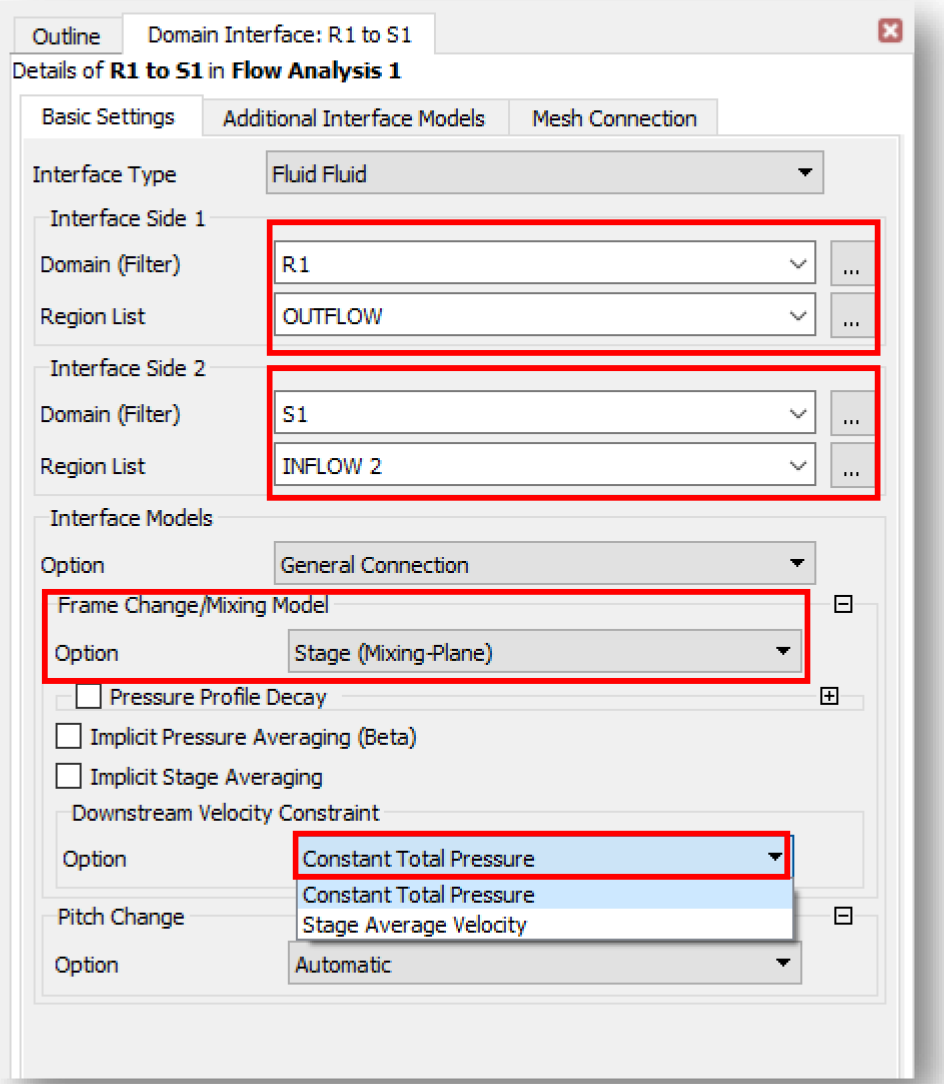




NOTE: Each side of the interface is a boundary in the domain.

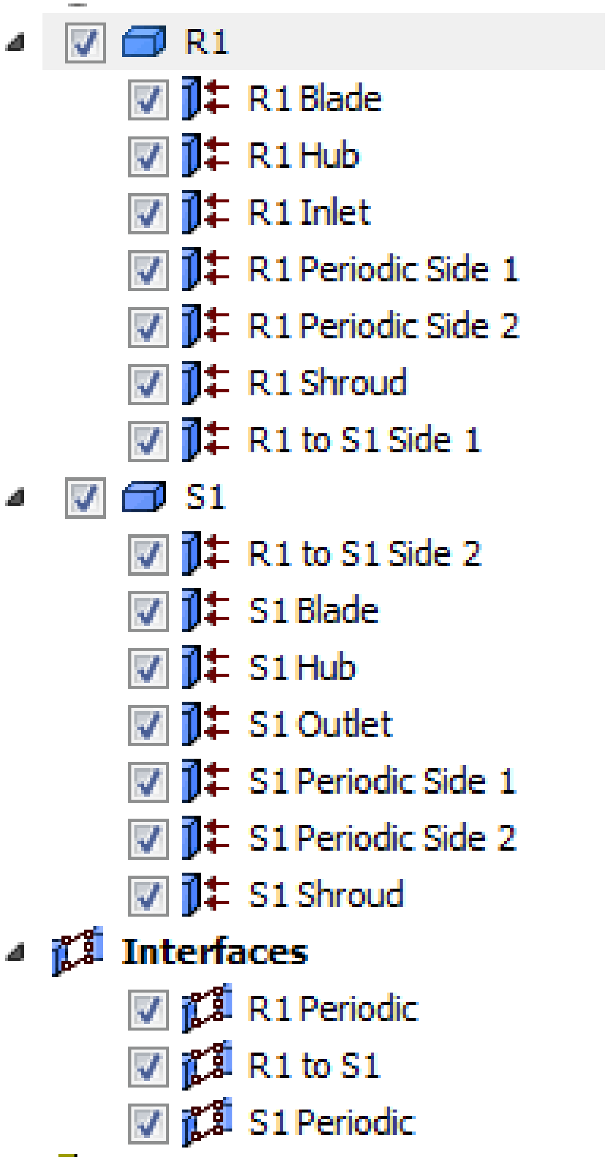
Stage Interface: Rotor to Stator

- Create a *Domain Interface* named *R1 to S1*
  - *Domain (Filter):*
    - *R1* (Side 1)
    - *S1* (Side 2)
  - *Region List:*
    - *OUTFLOW* (Side 1)
    - *INFLOW 2* (Side 2)
  - *Frame Change/Mixing Model: Stage*
  - *Downstream Velocity Constraint: Constant Total Pressure*



Check Boundaries/Interfaces

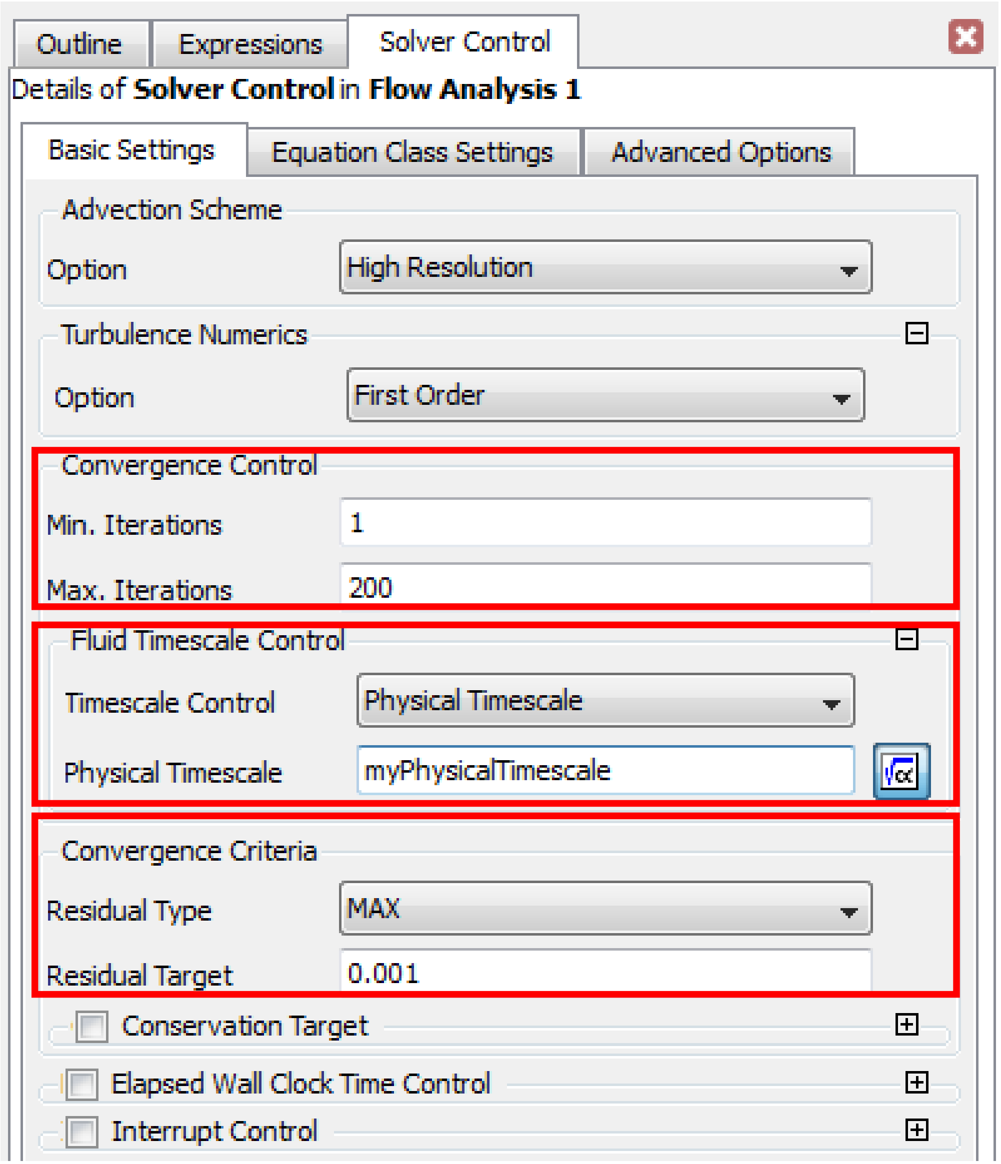
Check the outline tree for the boundaries and interfaces defined.



Solver Control

Edit Solver Control:

- Increase *Max Iterations* to *200*.
- *Physical Timescale: myPhysicalTimescale*.
- Change *Residual Type* to *MAX*.
- *Residual Target = 0.001*



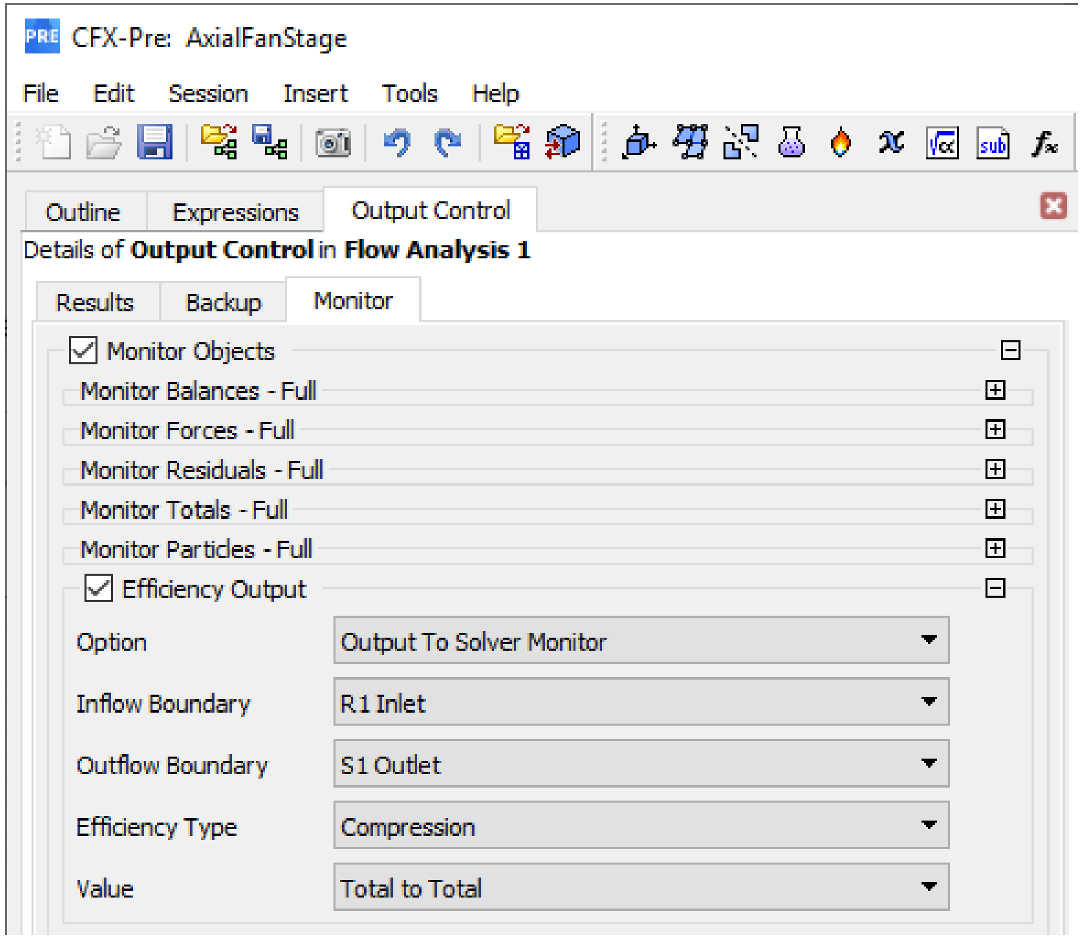
Output Control

In the *Outline* Double-click on *Output Control*.

Select *Monitor Tab* and enable *Monitor Objects*.

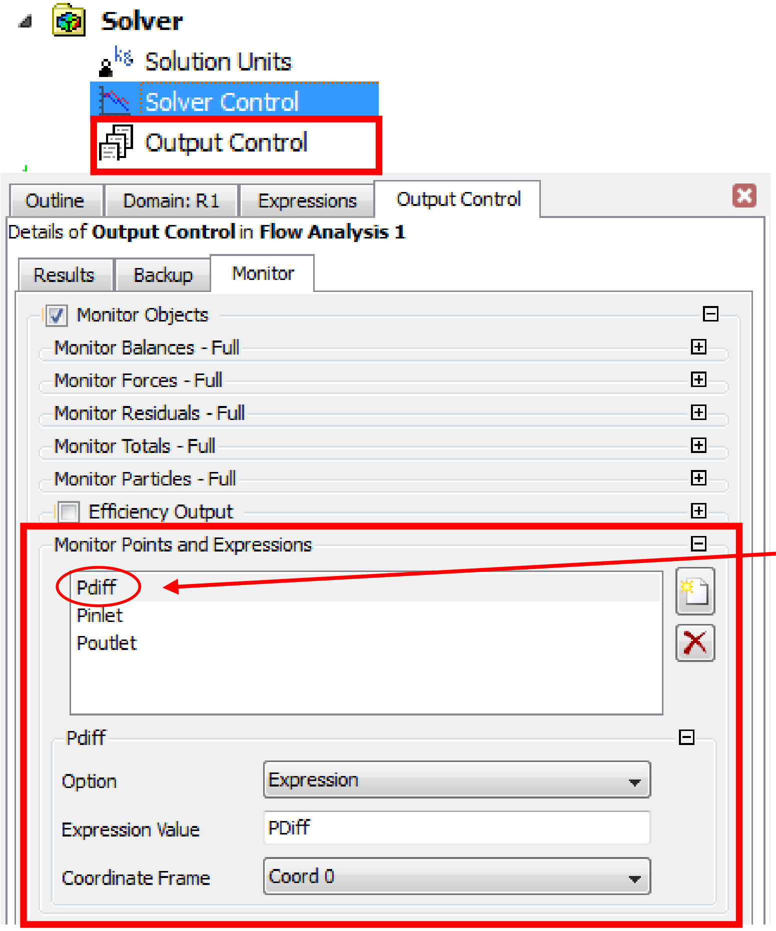
- Enable *Efficiency Output*.
- *Efficiency Type : Compression*.
- *Value: Total to Total*.

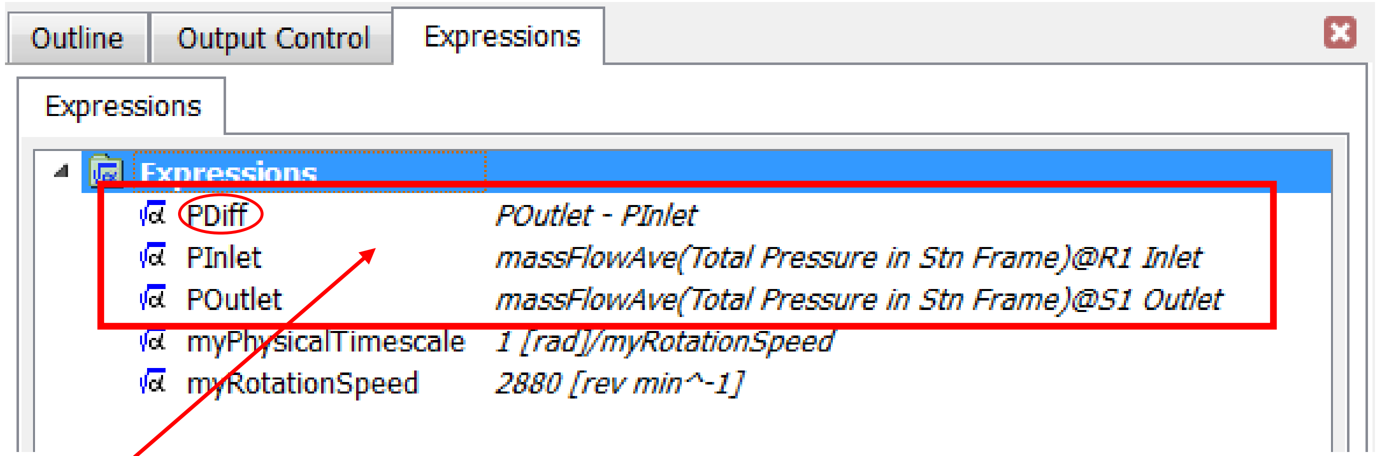
- NOTE: The efficiency calculates both the Isentropic and Polytropic efficiencies.



Output Control

- Define expressions for monitoring during the run.
- *Expression PDiff* gives the pressure rise developed by the fan.





Attention: The monitor name must differ from the expression name! In the this example the expression PDiff is used for monitor point Pdiff

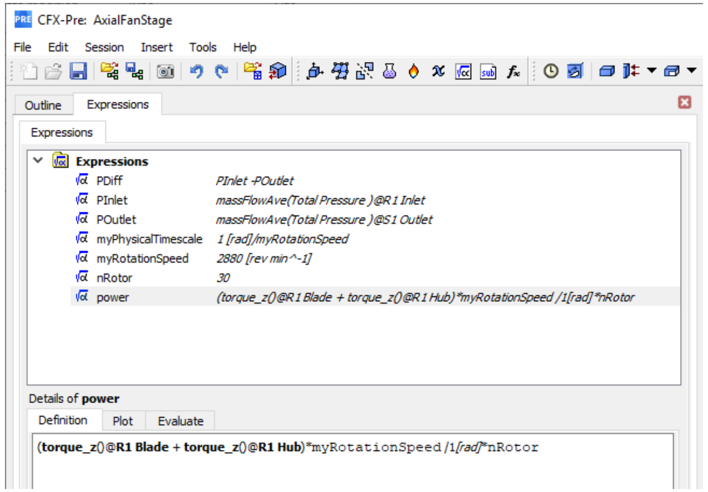
Output Control

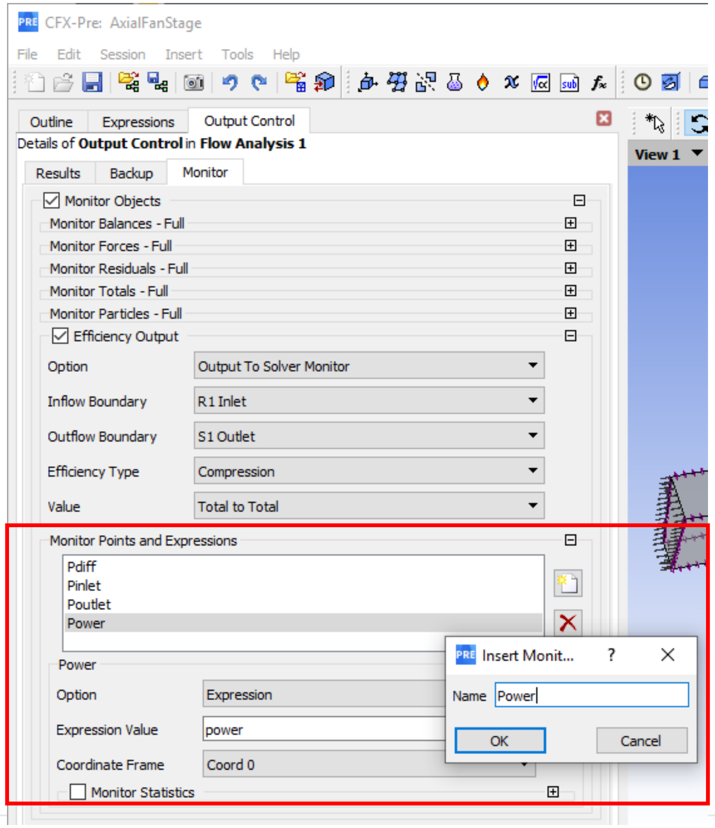
Create the following expressions to calculate Power consumed by the fan:

- Expression for number of rotor blades
  - *Name: nRotor*
  - *Value: 30*
- Expression for power
  - *Name: power*
  - *Value*: *`(torque_z()@R1 Blade + torque_z()@R1 Hub)*myRotationSpeed/1 [rad] *nRotor`*

Define a *Monitor Point* for Power

- *Name: Power*
- *Expression Value: power*





Obtaining a Solution

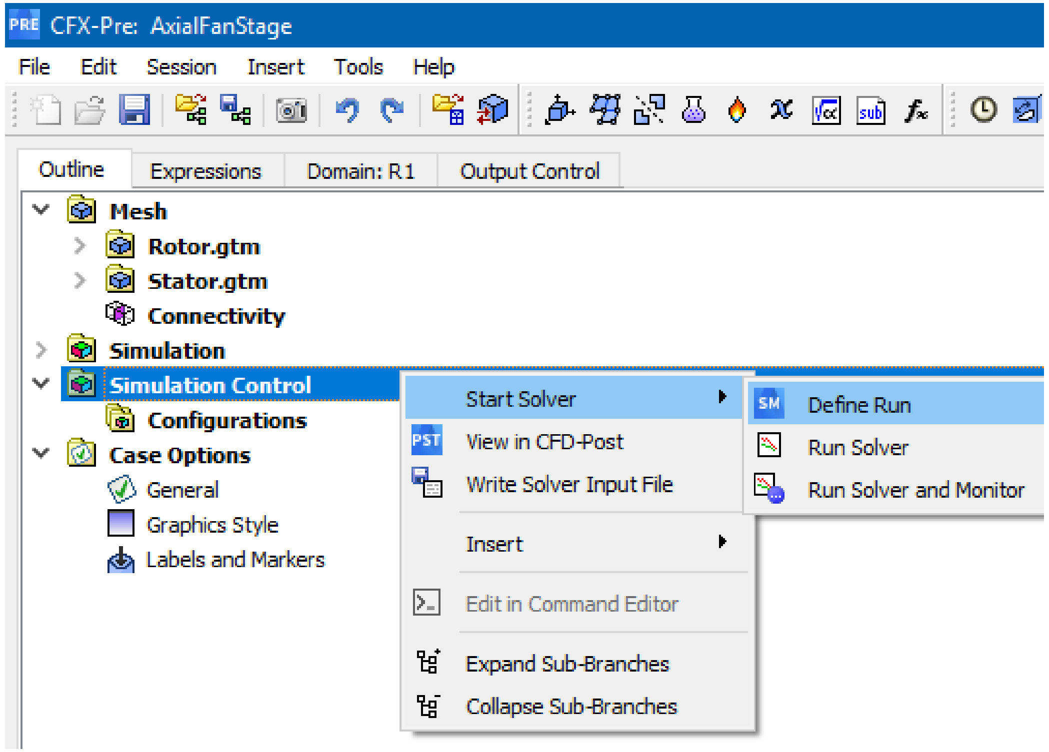
- Using stand-alone CFX mode (outside Workbench).
- Launch CFX-Pre from CFX-Launcher.
- Define stationary and rotating domain.
- Define boundary conditions.
- Define rotor stator interface.
- Set up monitor points using simple expressions.

- Launch the CFX Solver Manager.
- Check solution process.

- Launch CFD-Post.
- Initialize turbo post-processing.
- Define turbo surface.
- Use turbo coordinates for contour plots.

Write the definition file

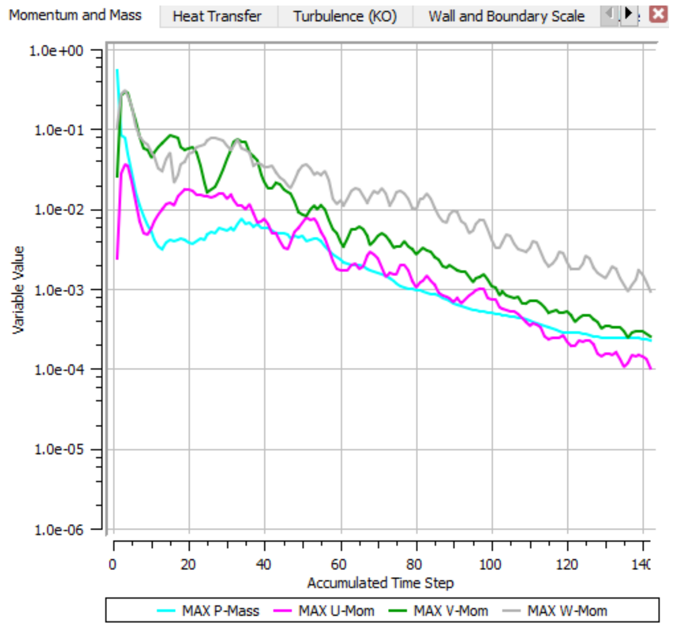
- *File > Save Case*
- Right-click on Simulation Control and select *Start Solver > Define Run*.
- Save the definition (def) file to *AxialFanStage.def* when prompted.
- Close CFX-Pre.
- The CFX-Solver Manager will open automatically.
- The Solver Input File (definition file) for the run will already be specified.
- Click on Start Run.

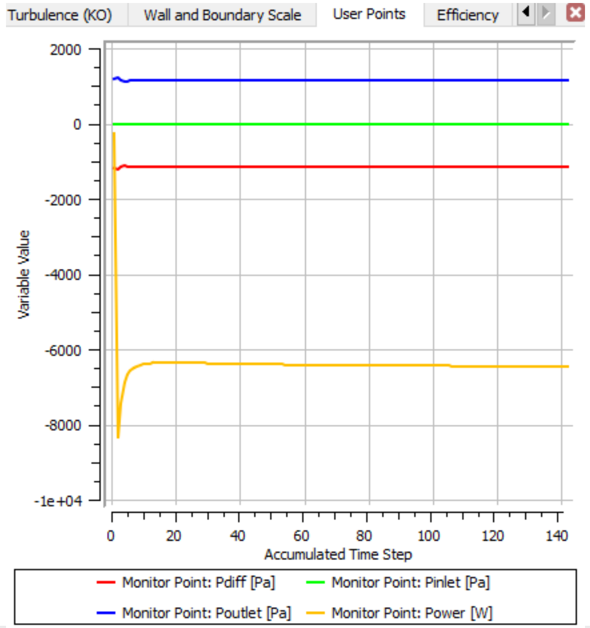


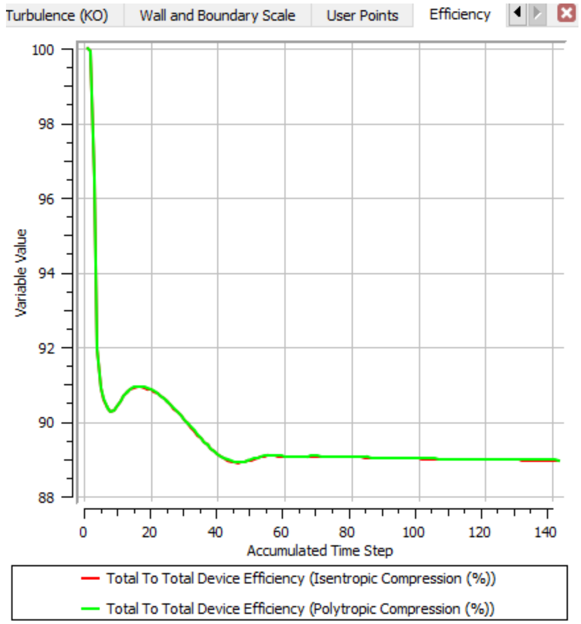
Solver

- The serial run is concluded in approximately 6 minutes
  - The MAX residuals are below the target of 1e-03
  - All monitor points reach constant values
    - ➢ The pressure rise developed by the fan is about 1159 Pa
    - ➢ The power consumed is about 6446 W
    - ➢ Isentropic and Polytropic Compression Efficiency is about 89%

Hint: The above values can be obtained by clicking on the appropriate monitor curves







Post-processing Goals

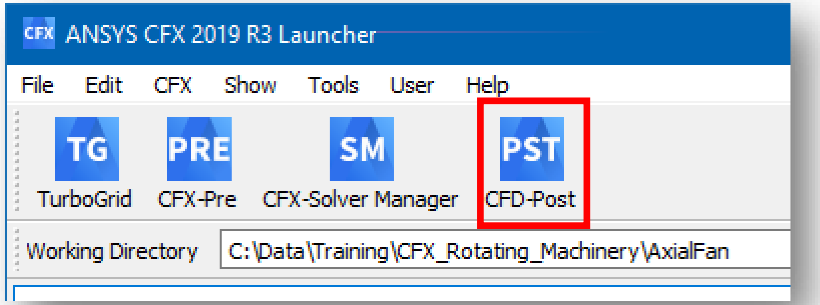
- Using stand-alone CFX mode (outside Workbench).
- Launch CFX-Pre from CFX-Launcher.
- Define stationary and rotating domain.
- Define boundary conditions.
- Define rotor stator interface.
- Set up monitor points using simple expressions.

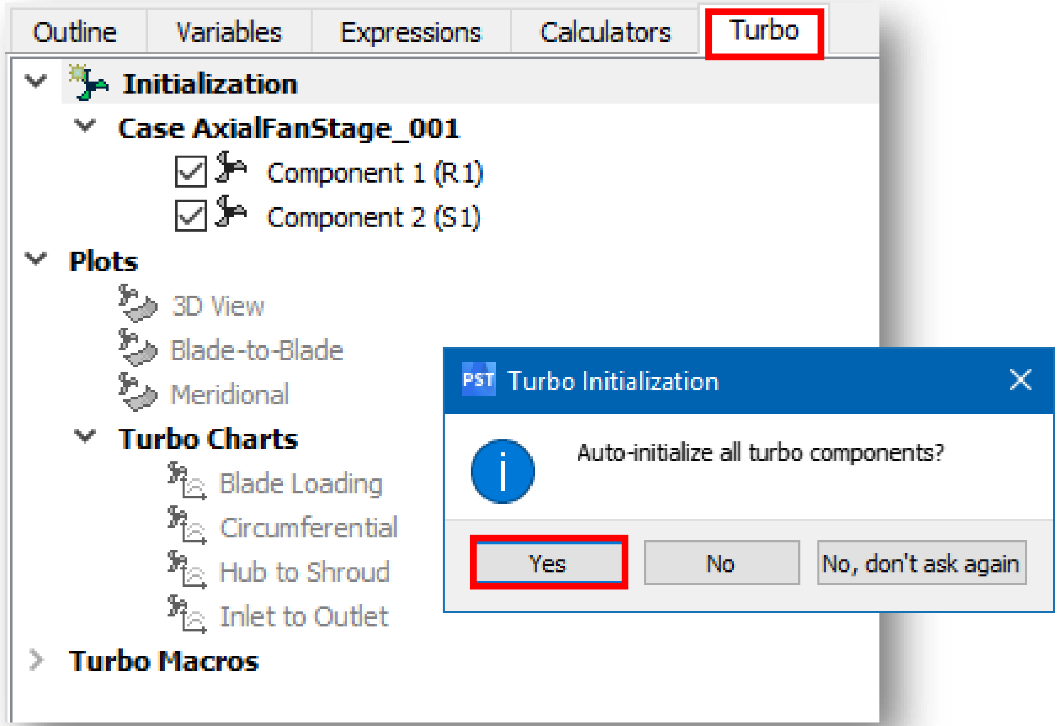
- Launch the CFX Solver Manager.
- Check solution process.

- **Launch CFD-Post.**
- **Initialize turbo post-processing.**
- **Define turbo surface.**
- **Use turbo coordinates for contour plots.**

CFD-Post

- Launch CFD-Post from CFX-Launcher.
- Load results.
- Select the Turbo tab.
- Auto-initialize turbo components.
  - Based on naming conventions
    - ➢ Blade
    - ➢ Hub
    - ➢ Shroud
    - ➢ Inlet
    - ➢ Outlet

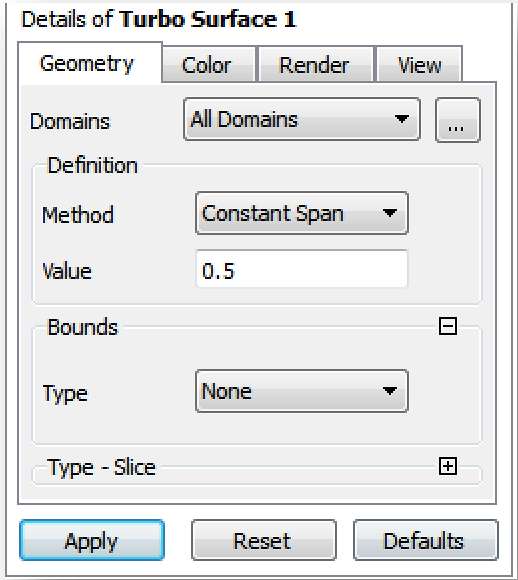


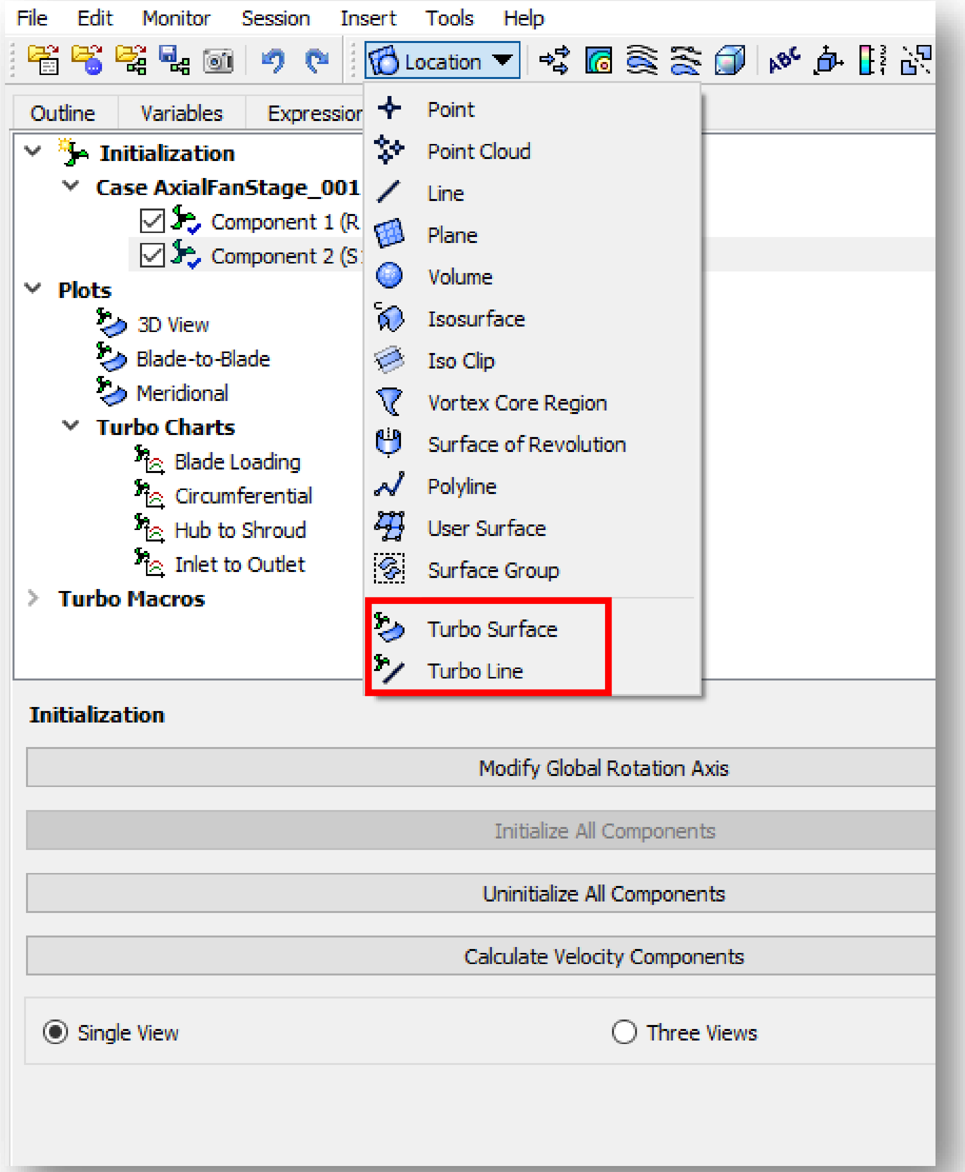


CFD-Post

- After initialization, turbo-specific *Locations* are available in CFD-Post
  - – *Turbo Surface*
  - – *Turbo Line*

- Create a *Turbo Surface*
  - – Constant Span = 0.5



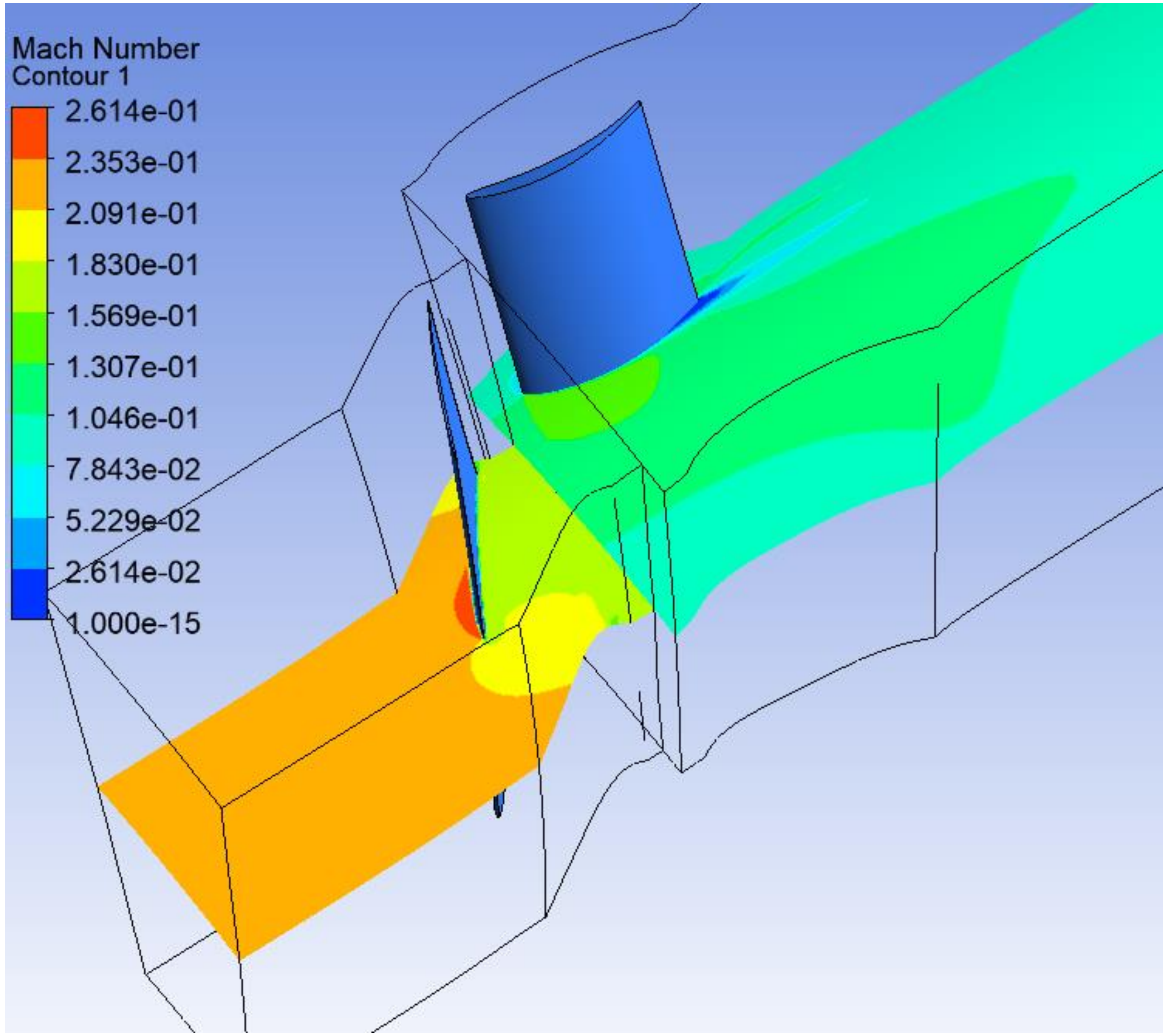


CFD-Post

Define Contour Plots on the Turbo Surface for:

- *Mach Number*
- *Total Pressure*
- *Total Pressure in Stn Frame*

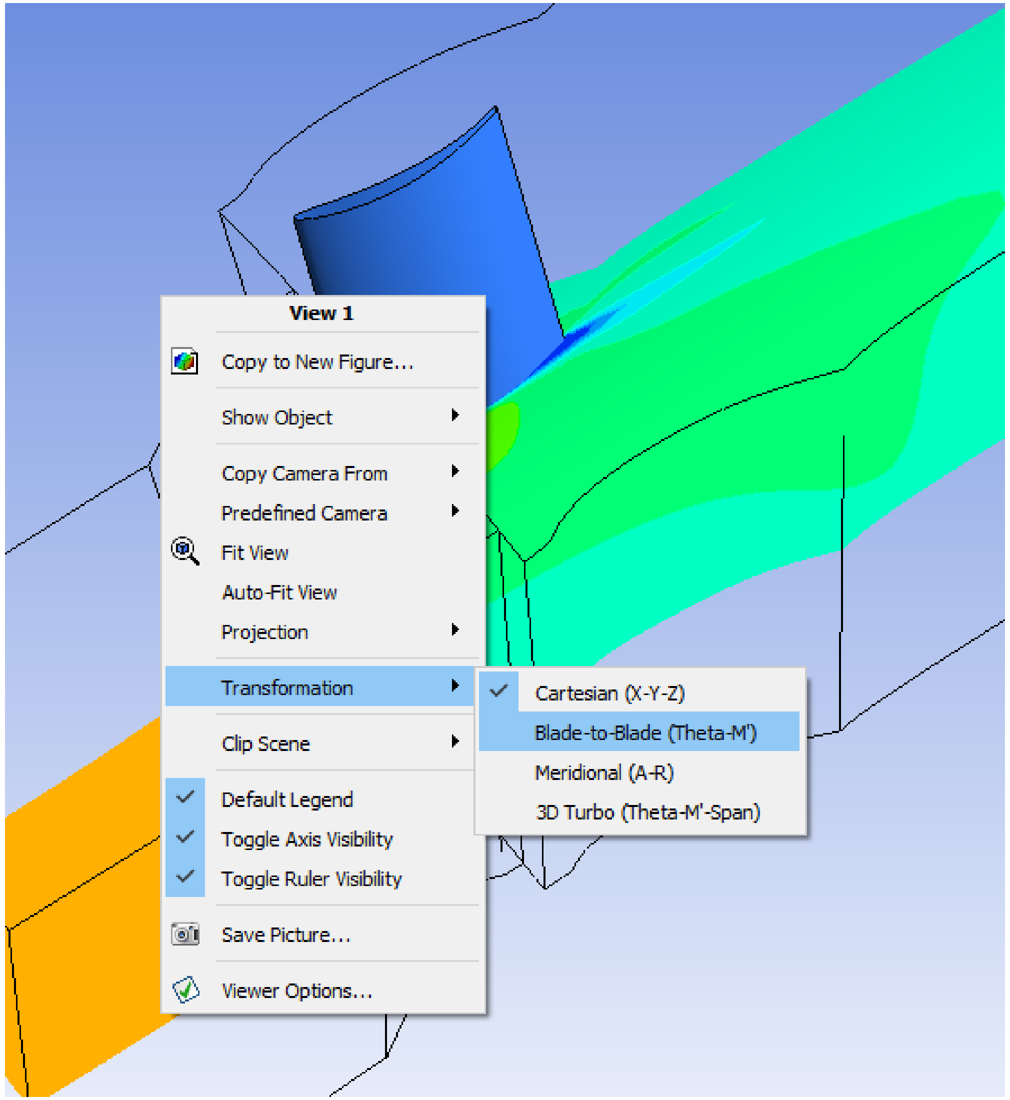
When there are rotating frames, extra variables are created, e.g. Total Pressure in Stn Frame, Velocity in Stn Frame. These are referred to the stationary rather than the rotating frame and so are useful for visualization.



CFD-Post

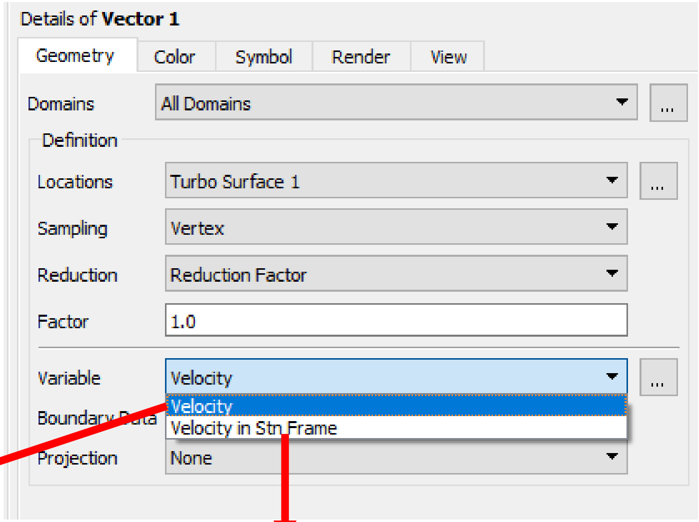
Change transformation.

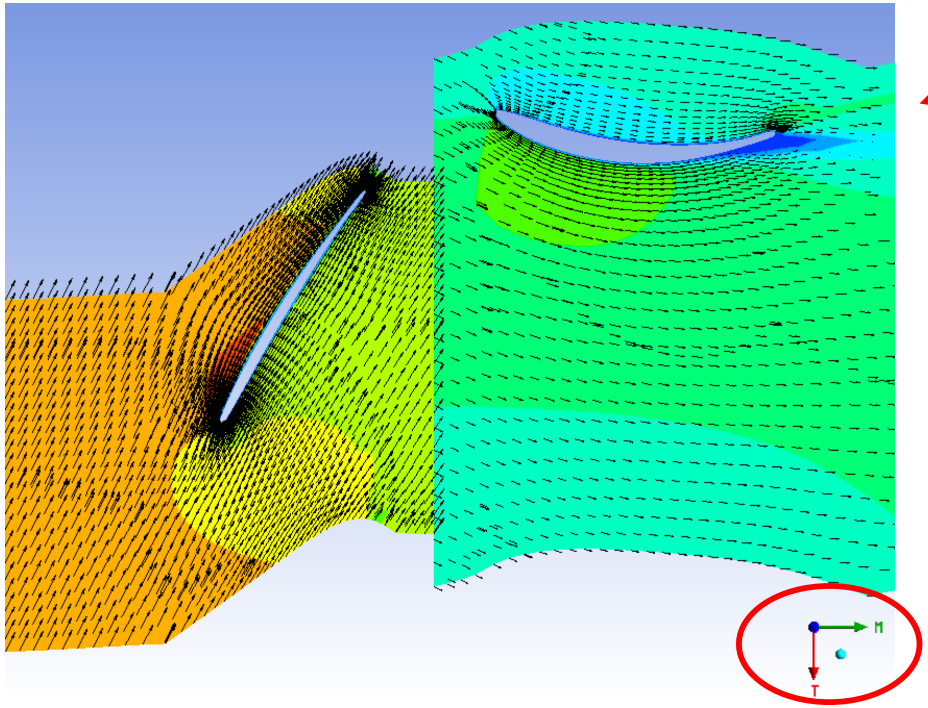
- RMB in empty space in the graphics window
- *Blade-to-Blade (Theta-M').*

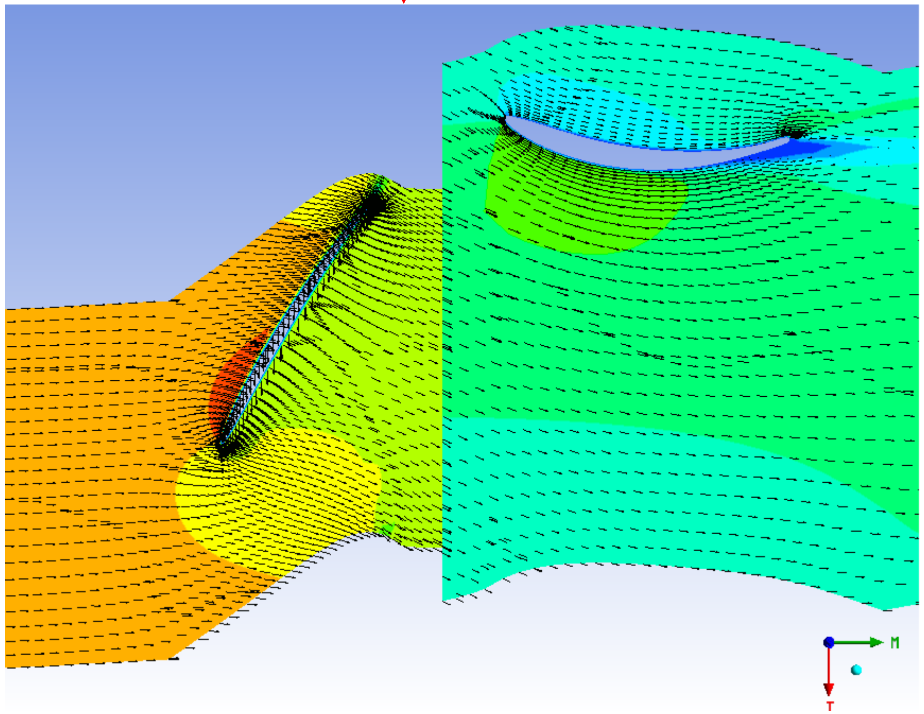


CFD-Post

- In Blade-to-Blade-view note the change to the coordinate system.
- Generate Vectors on Turbo-surface.
  - Note difference between vectors for Velocity (bottom left) and those for Velocity in Stn Frame (bottom right).



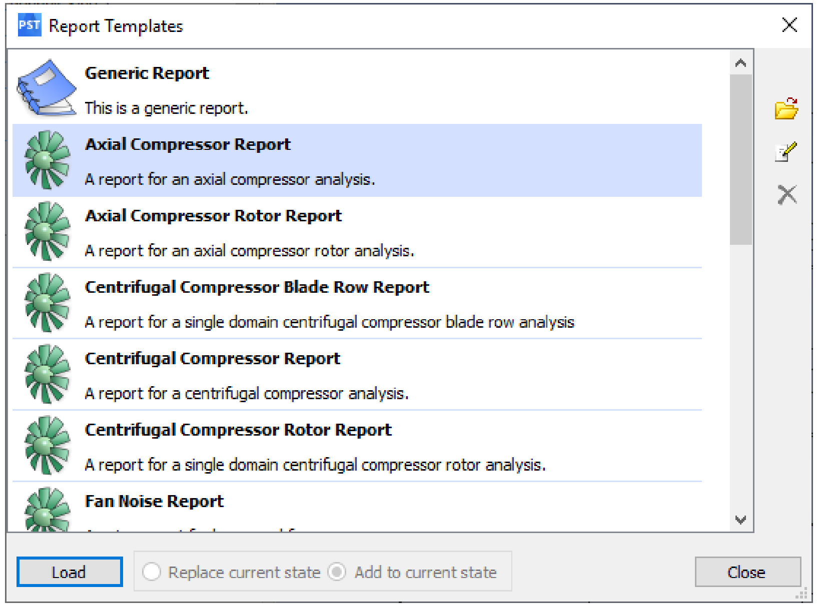


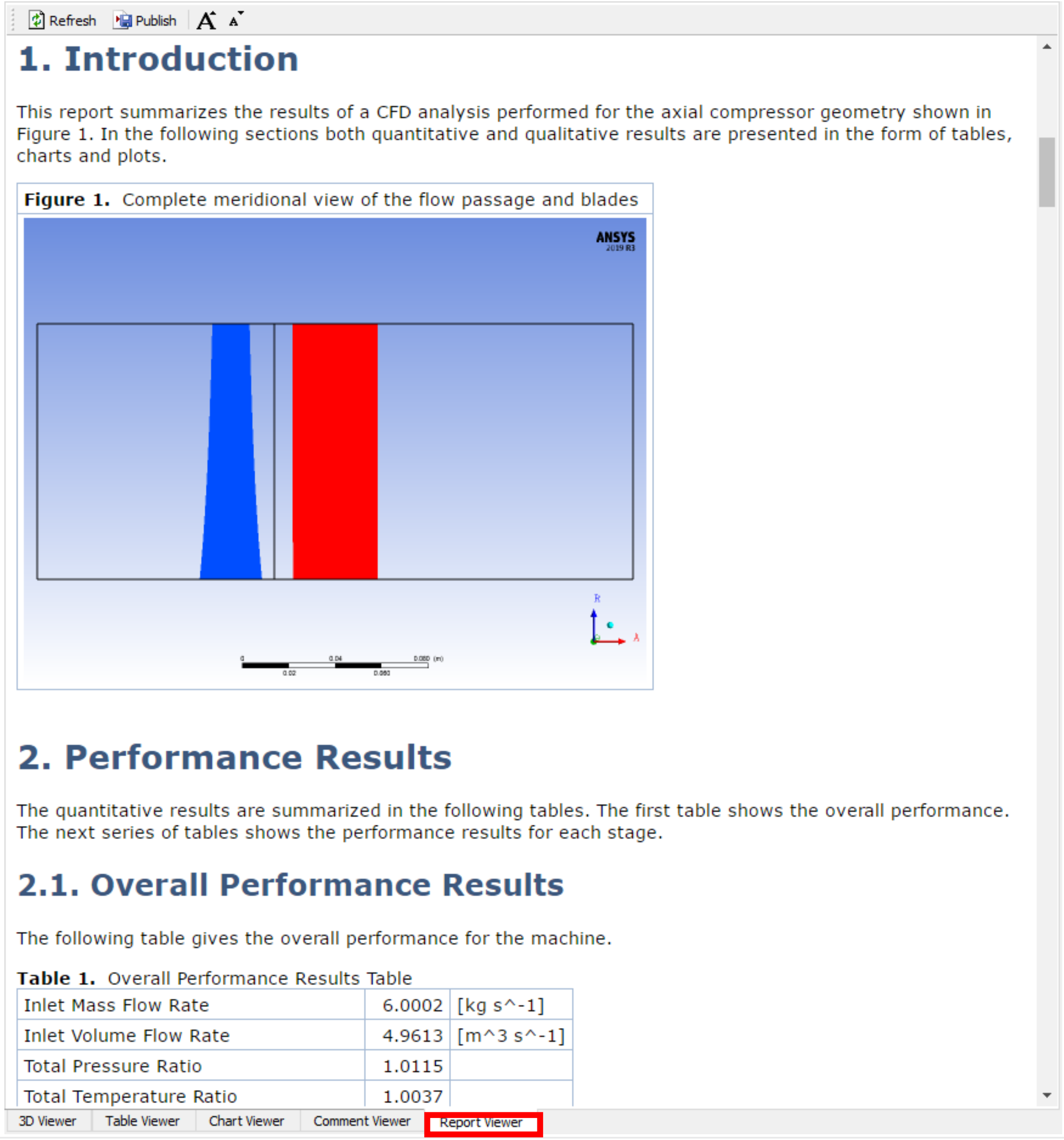


CFD Post

Axial Compressor Report

- CFD-Post provides a report template to present the performance of this type of device.
- Click on *File > Report > Report Templates*, select *Axial Compressor Report* and then *Load*.
- When the report has been generated go to the Report Viewer tab and look through the contents of the report.





Summary

- This workshop has shown a basic setup using multiple frames of reference.
- An interface dealing with the frame change has been defined (stage).
- Post-processing in turbo-specific coordinate system has been defined.
- Transformation of coordinates for visualisation has been set.