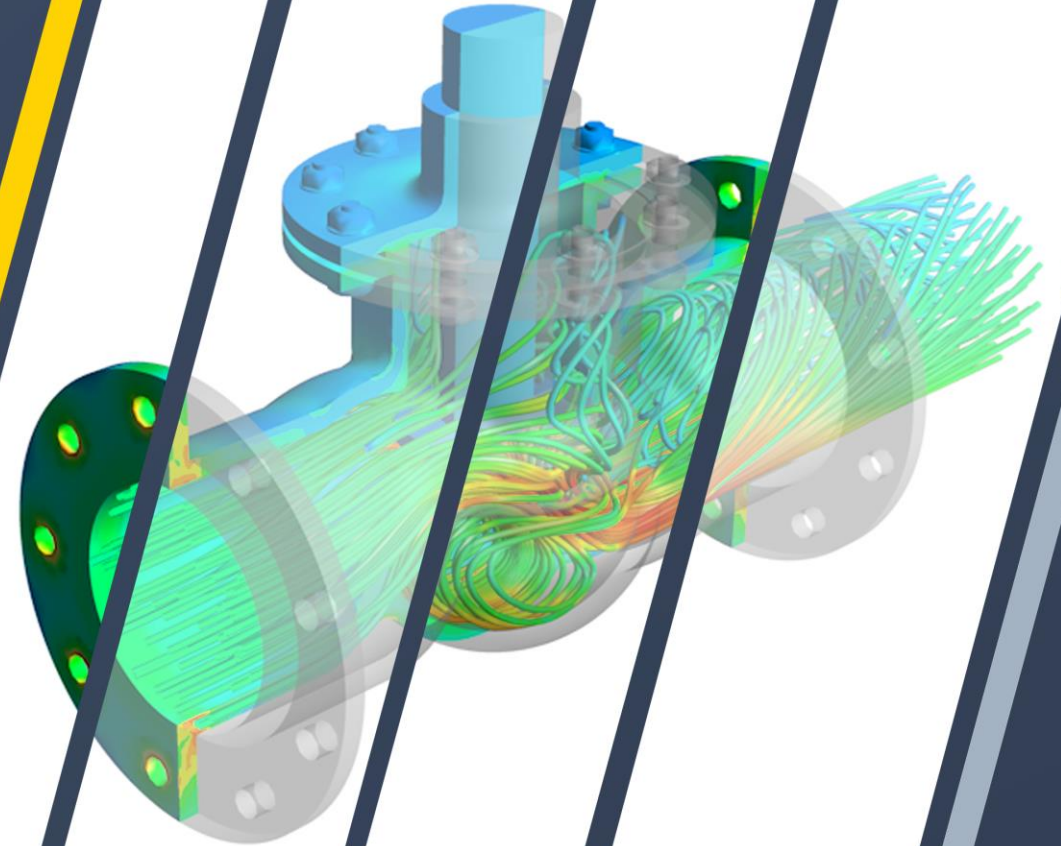




# Workshop 05.1: Time Transformation for modeling 1.5-stage

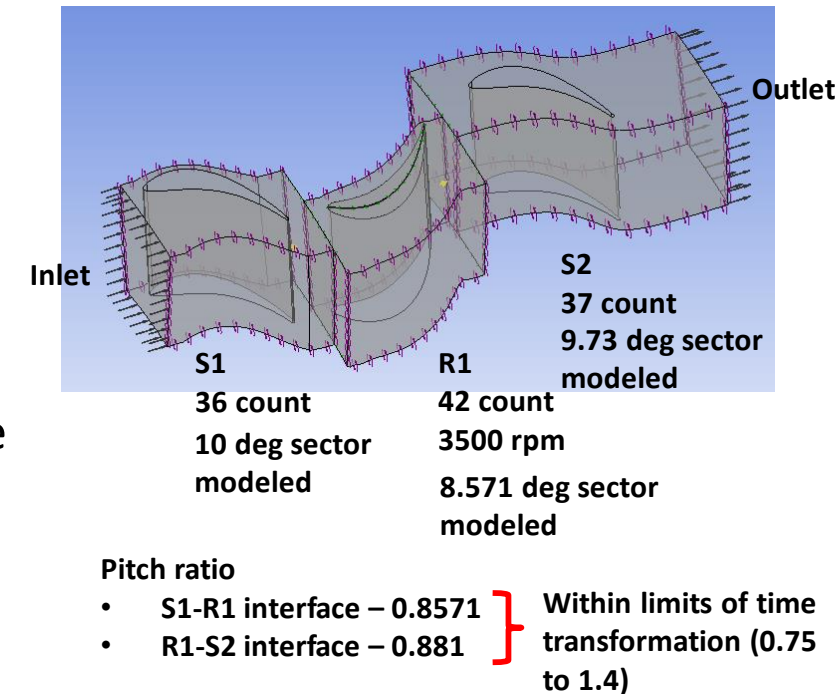
ANSYS CFX Rotating Machinery  
Modeling

Release 2019 R3



# Introduction

- This Workshop deals with 1.5 stage machine, operating at 3500 rpm
- The working fluid is Air Ideal Gas
- Aim is to illustrate the basic concepts of setting up, running, and monitoring a Transient blade row simulation using Time Transformation (TT) method in ANSYS CFX
- The geometry to be modeled consists of a single rotor blade passage and two stator blade passages (one from each stator row)
- Learning Aims:
  - Define a steady state simulation using Turbo machinery wizard
  - Perform steady state flow analysis for initializing TT simulation
  - Define & perform TBR analysis with TT method
  - Post process results in CFD Post



Step 1

Step 2

# TBR Methods in ANSYS CFX

## Focus of this tutorial

Profile  
Transformation  
(PT)

Small/Moderate  
Pitch

- Single Stage
- Multistage

**Time  
Transformation  
(TT)**

Small/Moderate  
Pitch

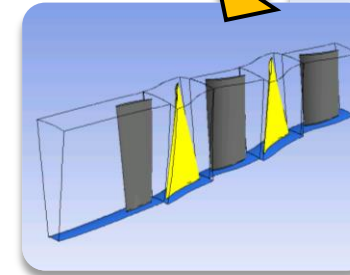
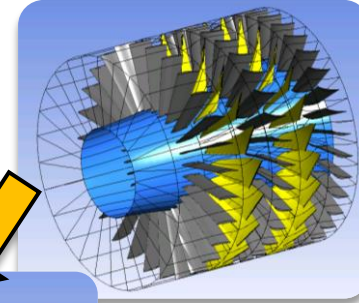
- Frozen gust
- Single Stage
- Multistage

Fourier  
Transformation  
(FT)

Large Pitch

- Frozen gust
- Fan Inlet Distortion
- Single Stage
- Blade Flutter

Full-wheel  
Model

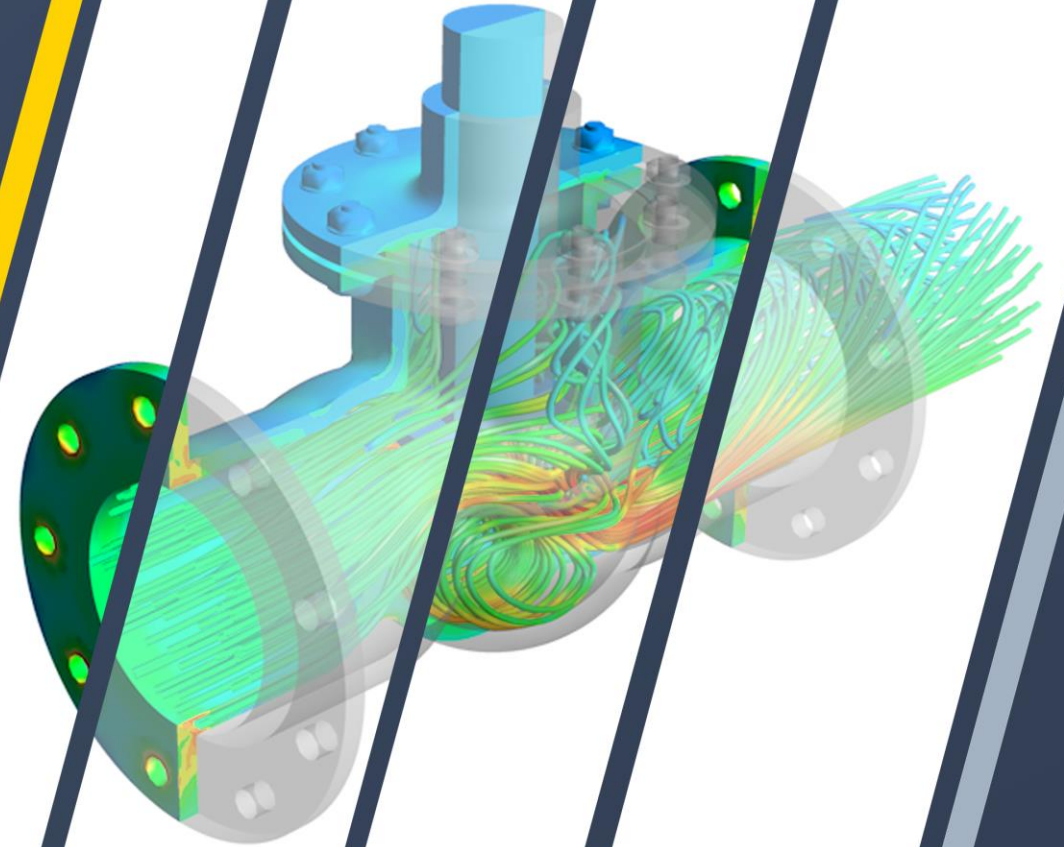


Reduced  
Model



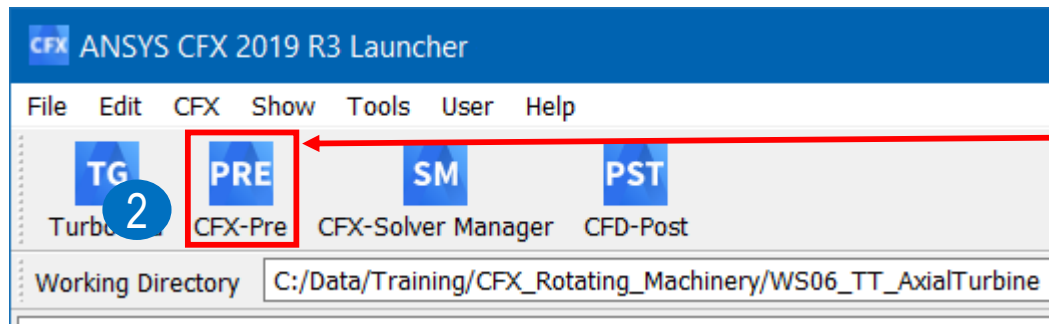
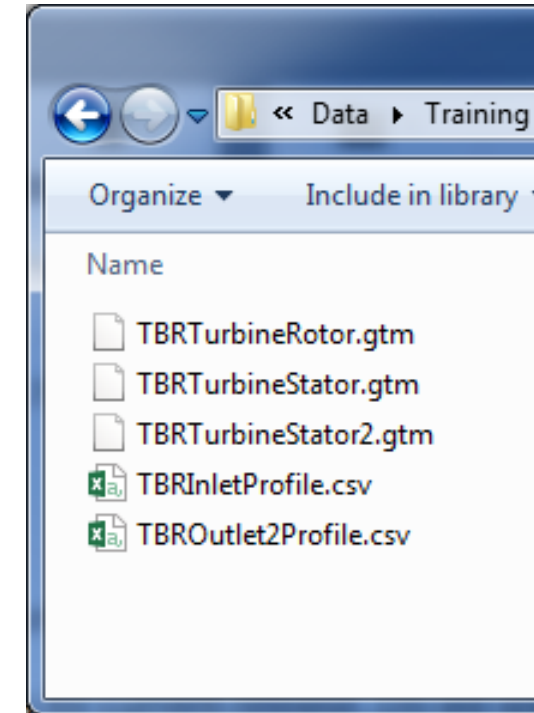
# Step 1

## Setting up a Steady State Simulation Using Turbo Wizard



# Setting up a working directory

- Copy mesh *.gtm* and profile BC *.csv* files provided with the workshop inputs in a working directory
- Start CFX-Launcher 1
  - *Start > All Programs > ANSYS 2019 R3 > CFX 2019 R3*
  - Browse to your working folder



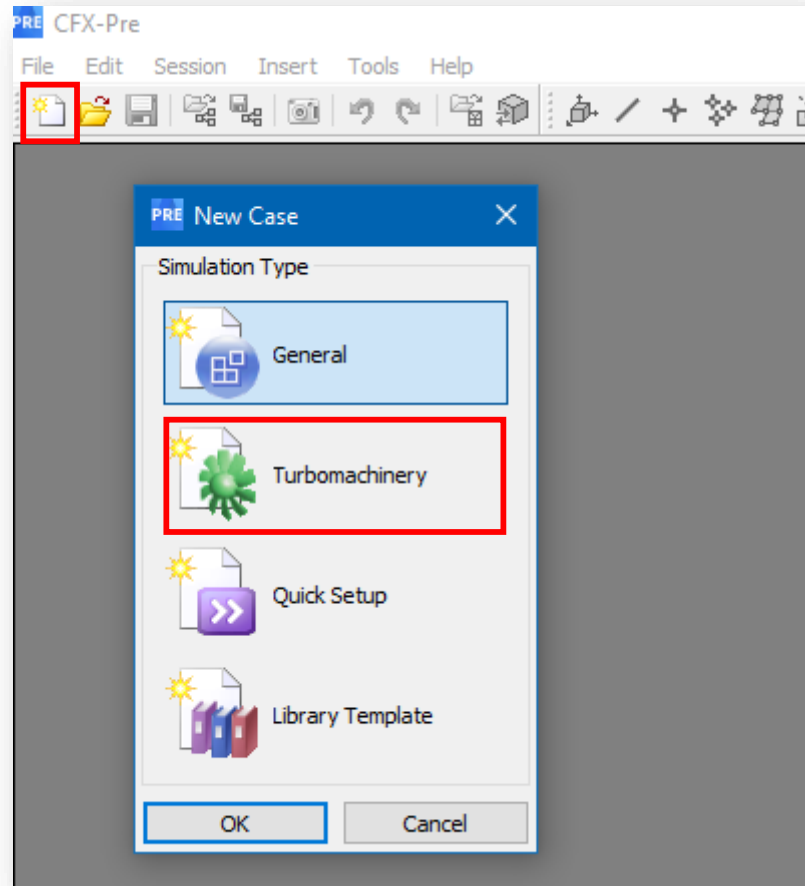
3

Click on *CFX-Pre* to open a new session

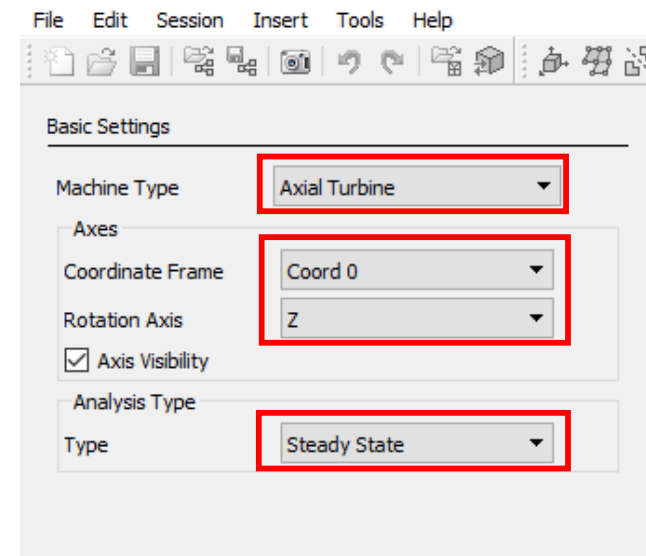
# Creating a New Case Using Turbomachinery Mode

Select *New Case*

Select  
*Turbomachinery*

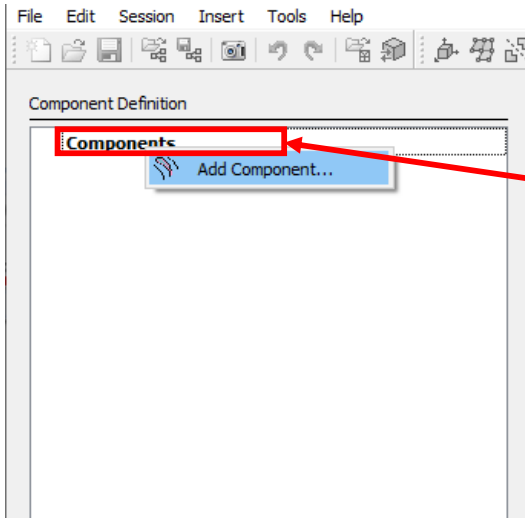


Select *Machine Type, Axis & Analysis Type*

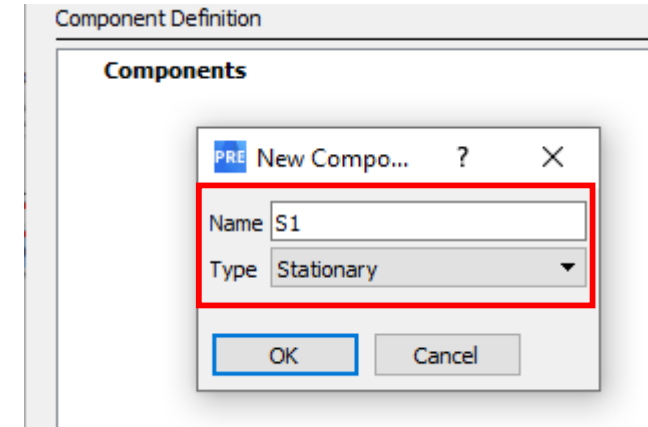


Click *Next* button at the bottom

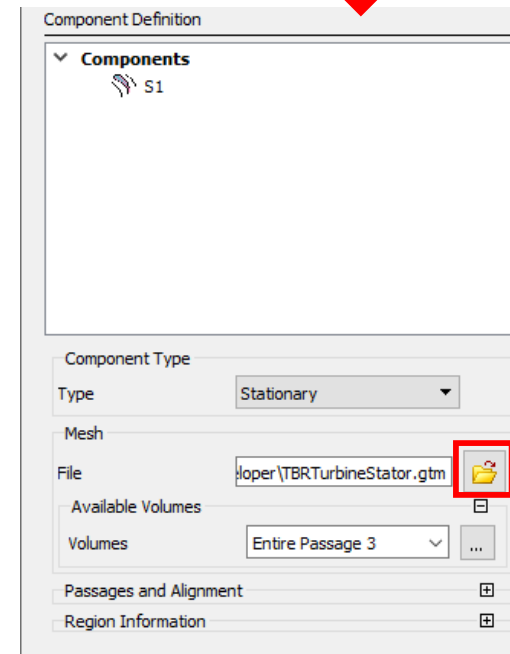
# Component Definition



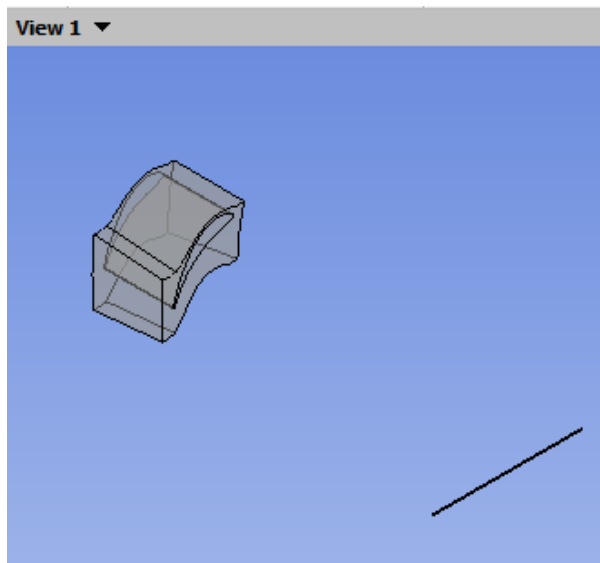
Right click on  
*Components*  
and select *Add  
Component...*



Do selections as shown  
by red box and click *OK*



Browse to file and select  
*TBRTurbineStator.gtm*

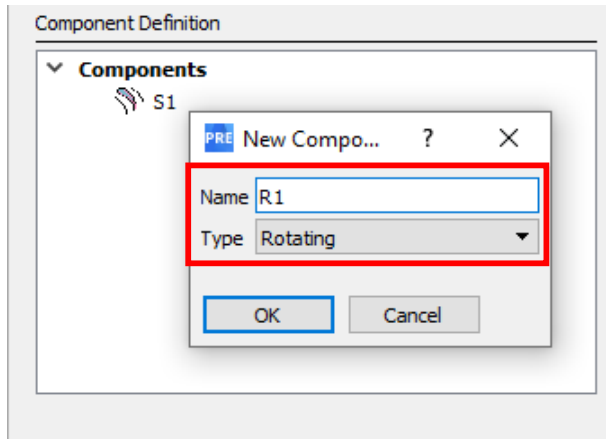


View after  
creating *S1*

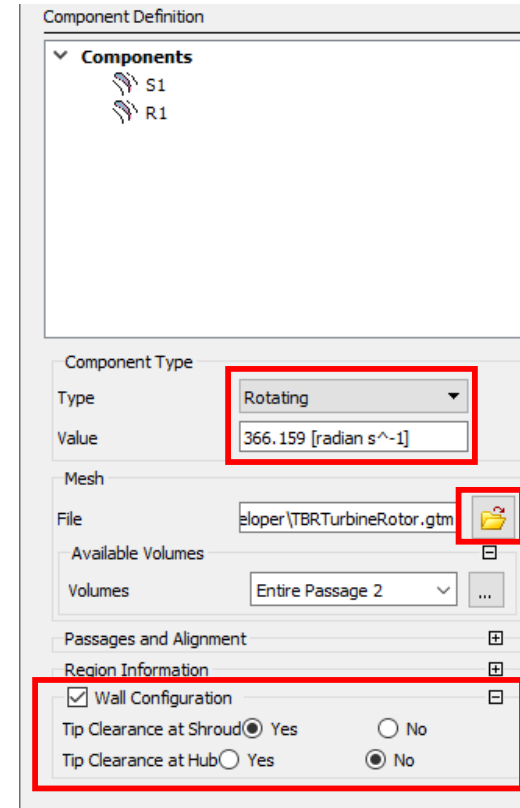
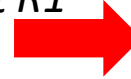




# Component Definition

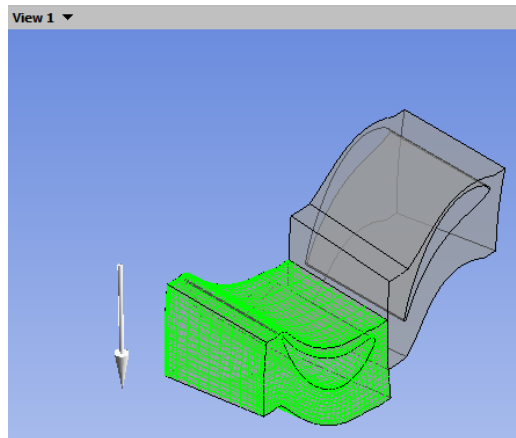


- Add a New Component *R1*
- Click *OK*



Do selections as shown by red box

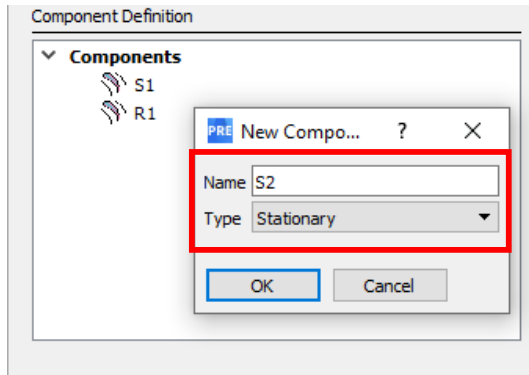
Browse to file and select *TBRTurbineRotor.gtm*



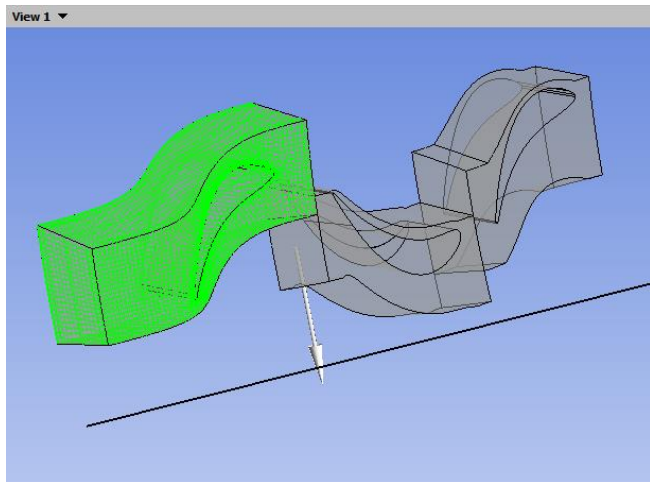
View after creating *S1*, *R1*



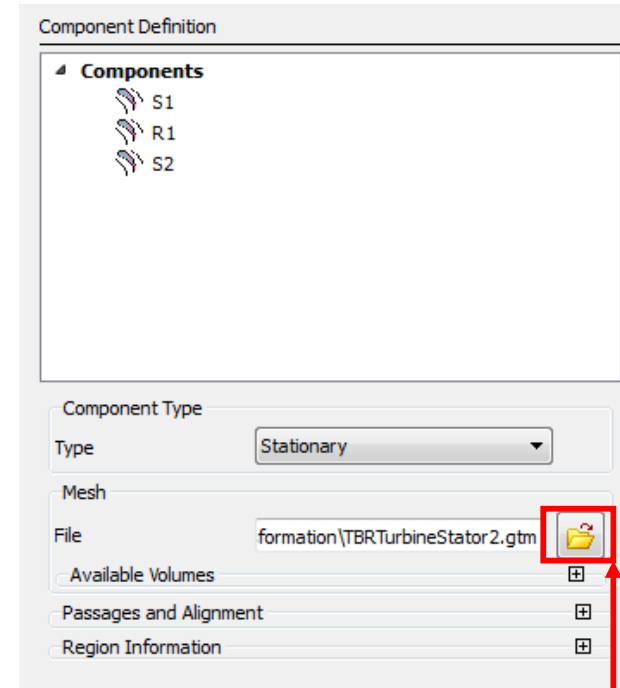
# Component Definition



- Add a new component S2
- Click *OK*



View after creating  
*S1, R1, S2*



Browse to file and select  
*TBRTurbineStator2.gtm*

Having defined all 3 components click *Next* at the bottom of *Component Definition* dialogue box

# Physics Definition

The screenshot shows the 'Physics Definition' dialog box with the following settings highlighted by red boxes:

- Fluid:** Air Ideal Gas
- Reference Pressure:** 0 [Pa]
- Heat Transfer:** Total Energy
- Turbulence:** Shear Stress Transport
- Inflow/Outflow Boundary Templates:** P-Total Inlet P-Static Outlet (selected)
- Inflow:**
  - P-Total:** 169000 [Pa]
  - T-Total:** 32.85 [C]
  - Flow Direction:** Normal to Boundary
- Outflow:**
  - P-Static:** 110000 [Pa]
- Interface:**
  - Default Type:** Stage (Mixing-Plane)

Do settings as shown  
by red boxes

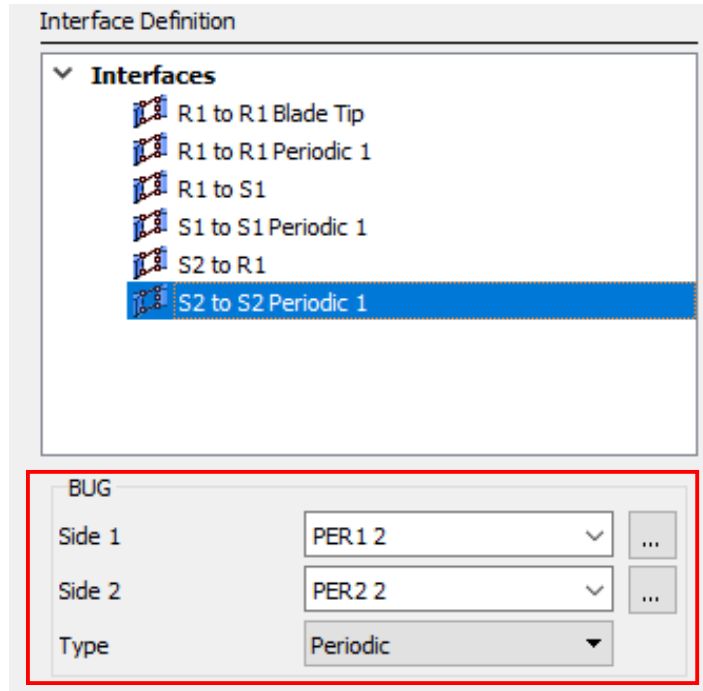
Inlet-Outlet Boundary  
Condition combination

Boundary Condition values

Use *Mixing-Plane*  
interface for steady  
state simulation

Having defined all conditions as shown click *Next* at  
the bottom of *Physics Definition* dialogue box

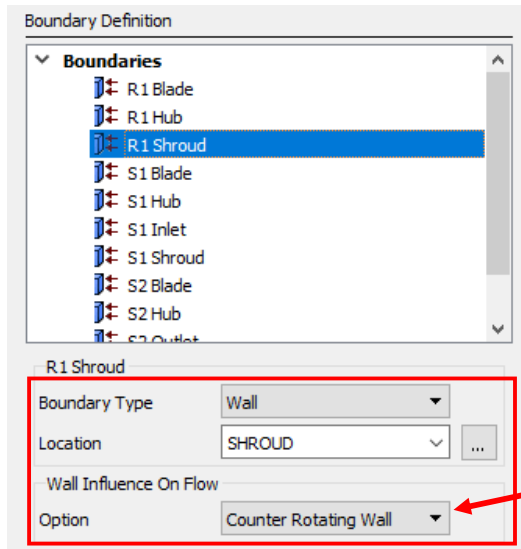
# Interface Definition



- Click on each interface in the list
- Visually examine if each interface is defined correctly (refer to both *Side 1* & *Side 2*)
- Verify interface *Type*
  - *Periodic* for periodic interfaces
  - *Mixing Plane* for *R1 to S1* and *S2 to R1*
  - *None* for *R1 to R1 Blade Tip* interface

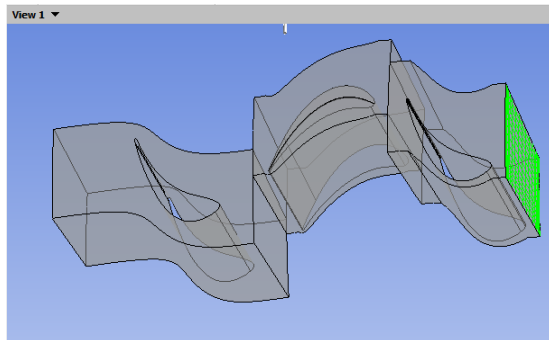
Having defined all Interfaces click *Next* at the bottom of *Interface Definition* dialogue box

# Boundary Definition

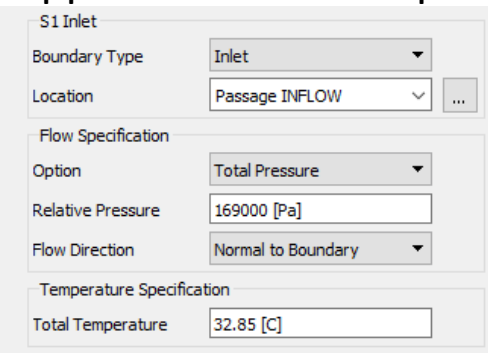


Represents a stationary wall in absolute frame

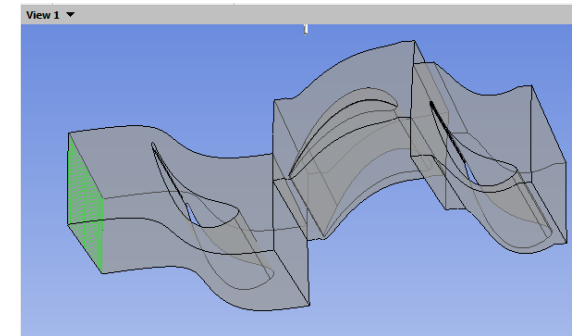
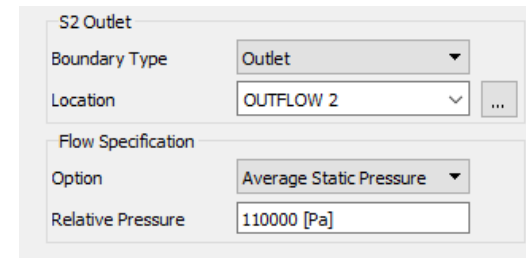
- Click on each *Boundary* in the list
- Visually examine if each *Boundary* is defined correctly (refer to *Location*)
- Verify *Boundary Type*
- Verify *Wall Influence On Flow*
  - should be *No Slip Wall* for all but *R1 Shroud*



Inlet boundary conditions supplied in earlier step



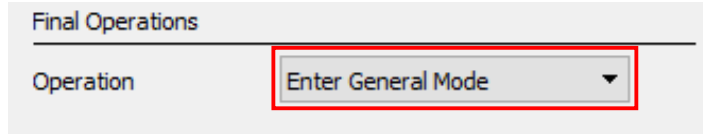
Outlet boundary conditions supplied in earlier step



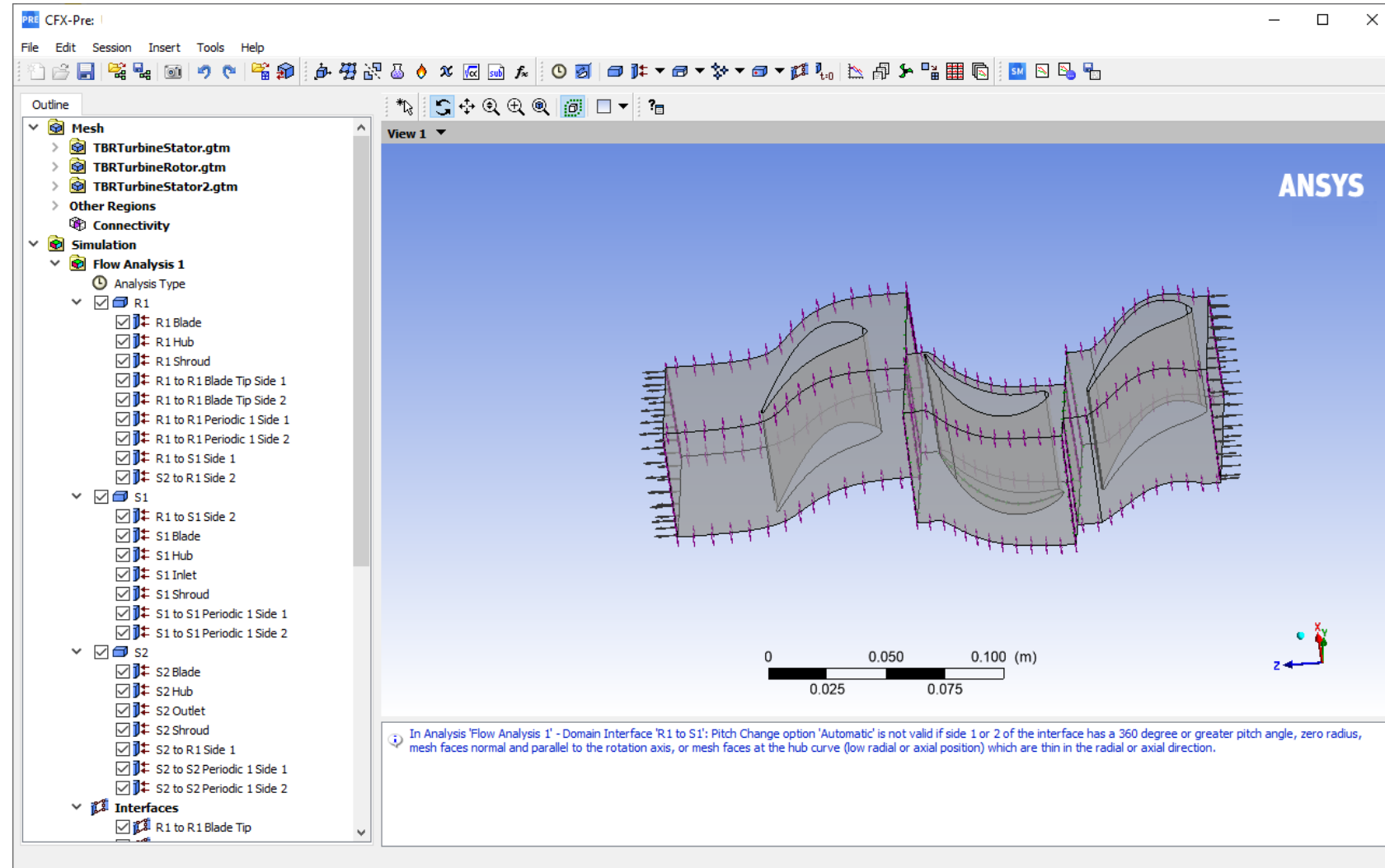
Click *Next* at the bottom of *Boundary Definition* dialogue box

# Final Operations

## View after exiting the turbo set up wizard



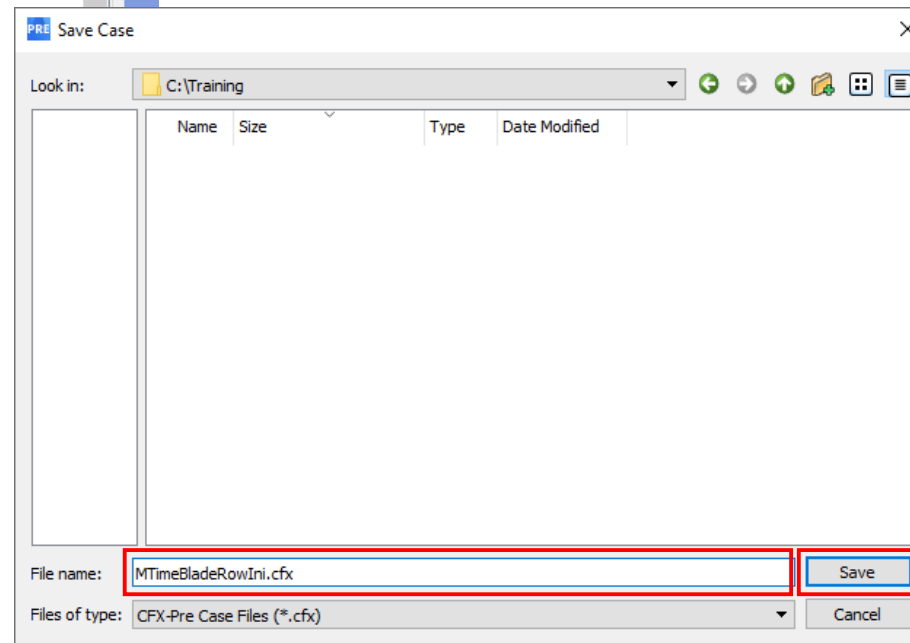
- Select *Enter General Mode*
- Click *Finish* at the bottom of *Final Operations* dialogue box



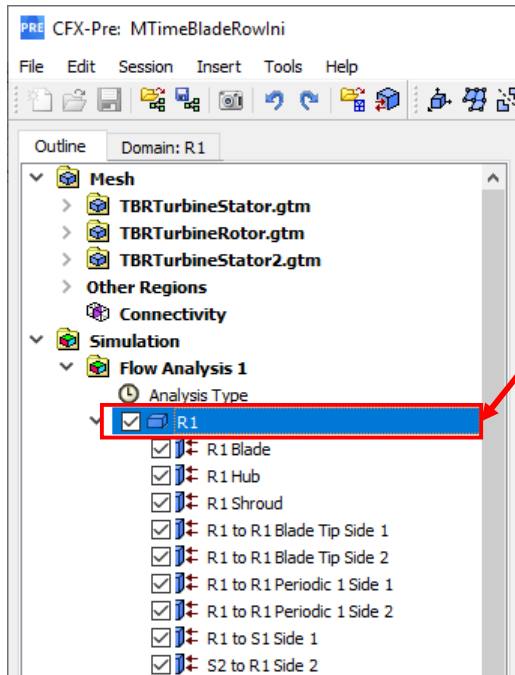
# Saving CFX file



- Click *File*
- *Save Case* → provide name: *MTimeBladeRowIni.cfx* and click *Save*



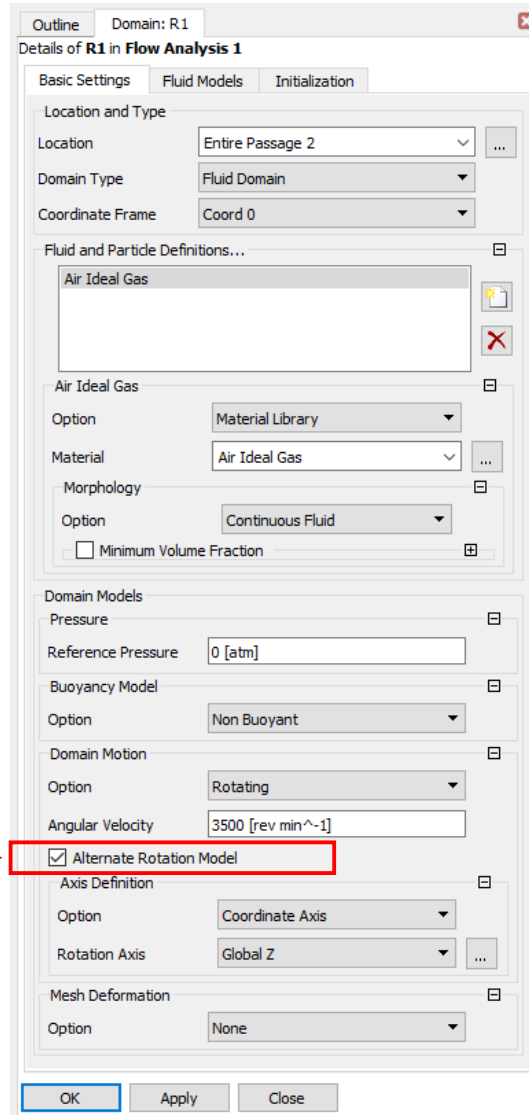
# Additional Domain Settings



Double click on  
R1 in *Outline*

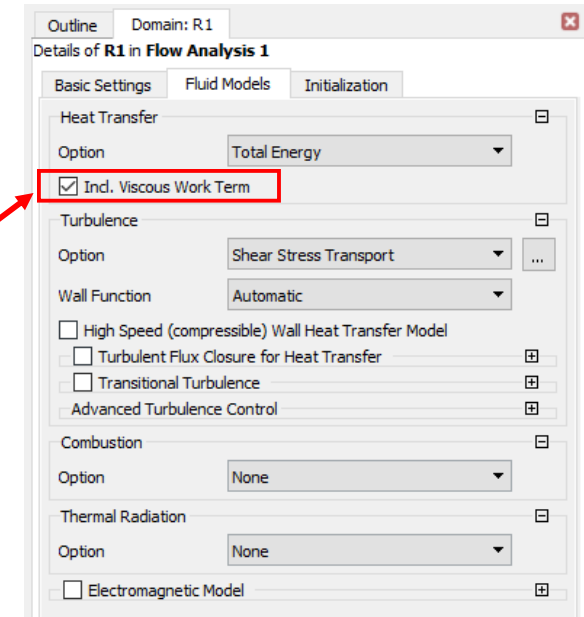
Ensure that  
this box is  
checked

## Basic setting tab



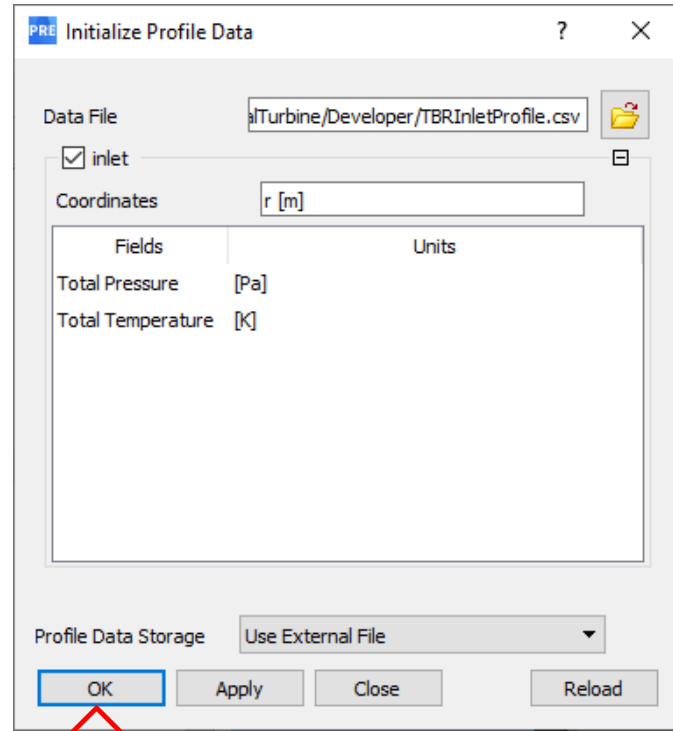
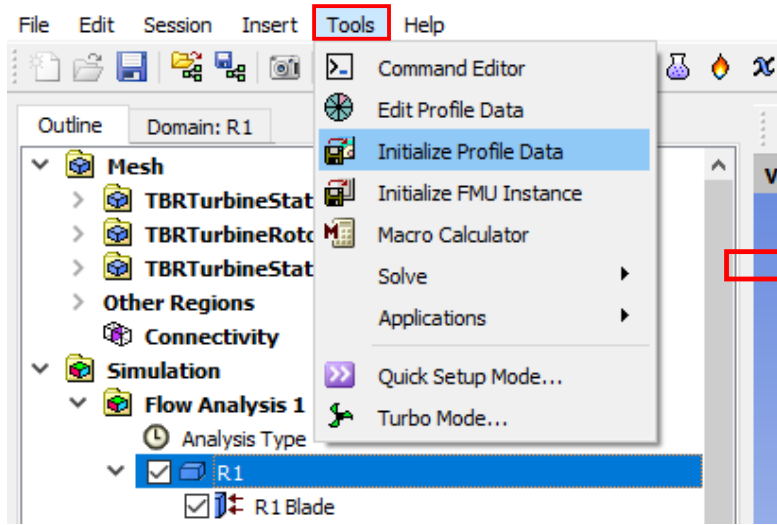
## Fluid models tab

Ensure  
that this  
box is  
checked





# Reading Profile Data



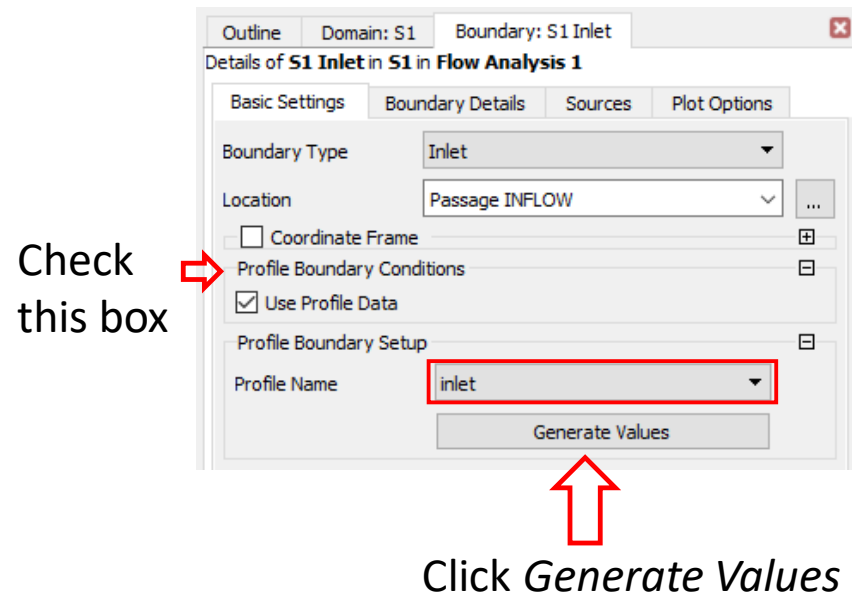
Click OK

- *Tools>Intialize Profile Data*
- Select file *TBRInletProfile.csv* which has inlet profile data
- In same way *Intialize Profile Data* using *TBROutlet2Profile.csv*

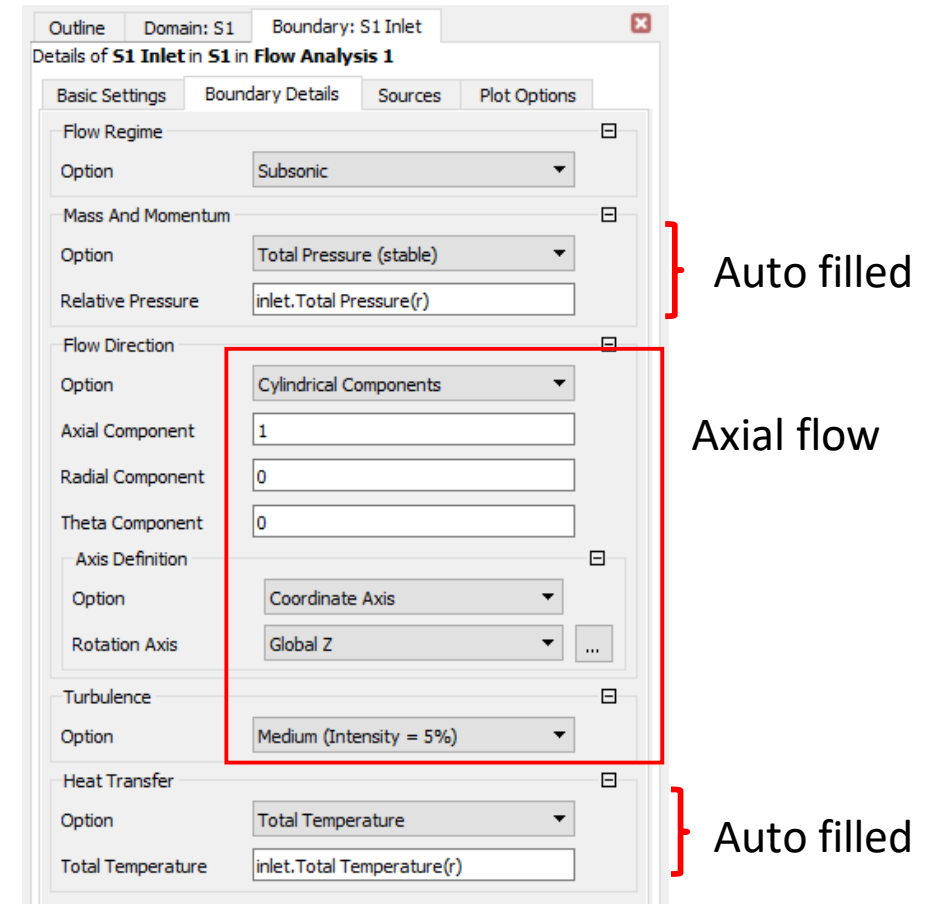
# Modifying Inlet Boundary

- In *Outline* → double click on *S1 Inlet*
- Do selections as shown by red boxes and red arrows

*Basic Settings* tab



*Boundary Details* tab



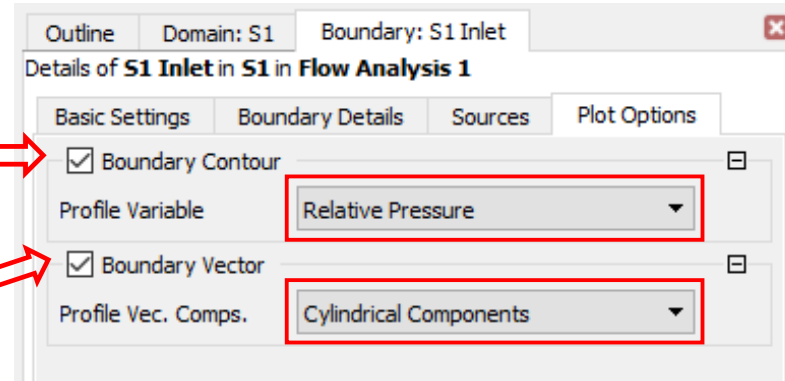
Click *Apply* after selections

# Visualizing Profile Data on S1 Inlet

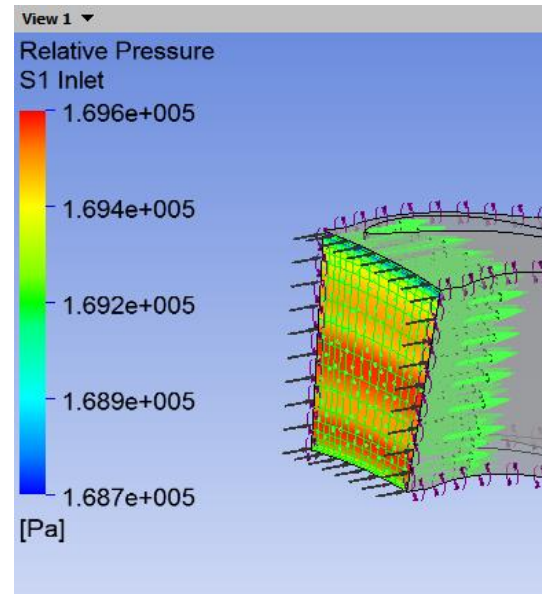
*Plot Options tab*

Check this box to  
view contours

Check this box to  
view vectors



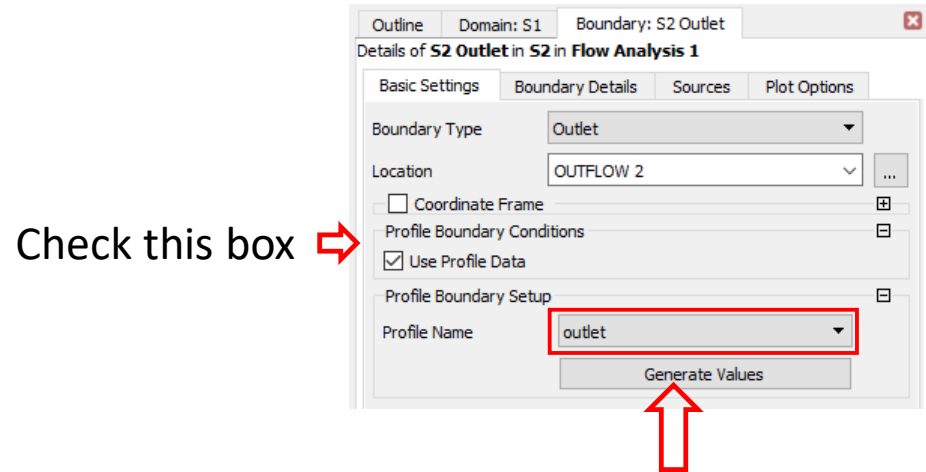
Plot of Pressure  
contours & Cylindrical  
velocity components



# Modifying Outlet Boundary

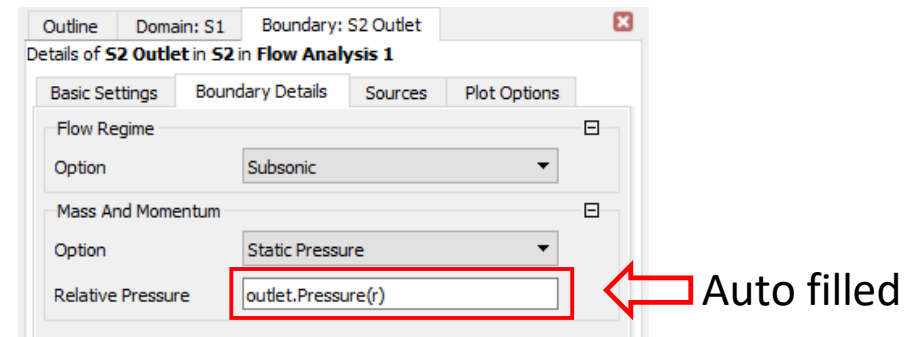
- In *Outline* double click on *S2 Outlet*
- Do selections as shown by red boxes and red arrows

*Basic Settings tab*



Click *Generate Values*

*Boundary Details tab*

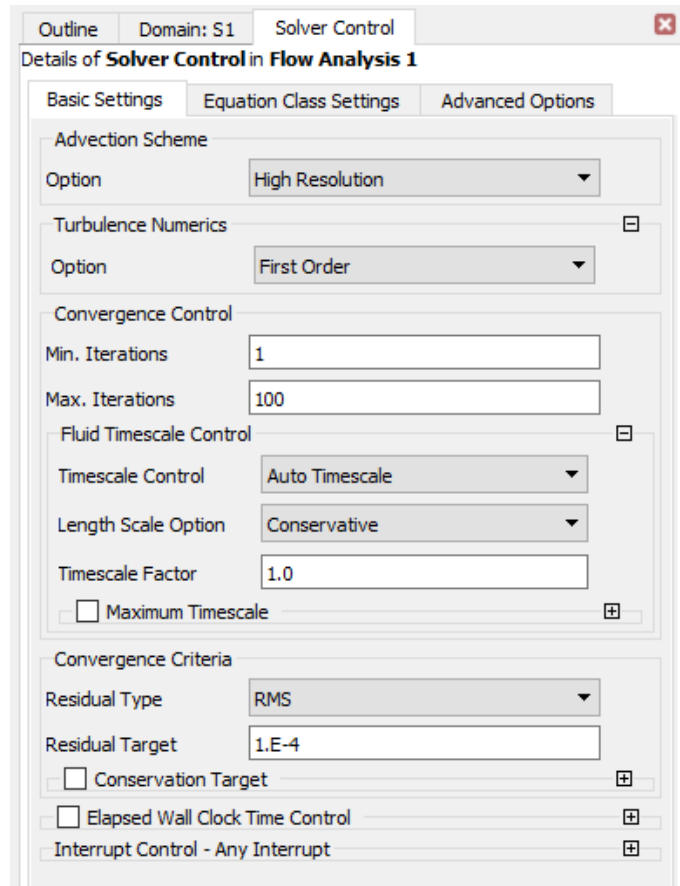


Click *Apply* after selection

One can see pressure contour plot on S2 Outlet boundary by going to *Plot Options* tab of this boundary condition

# Solver Settings

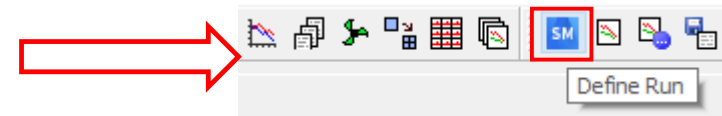
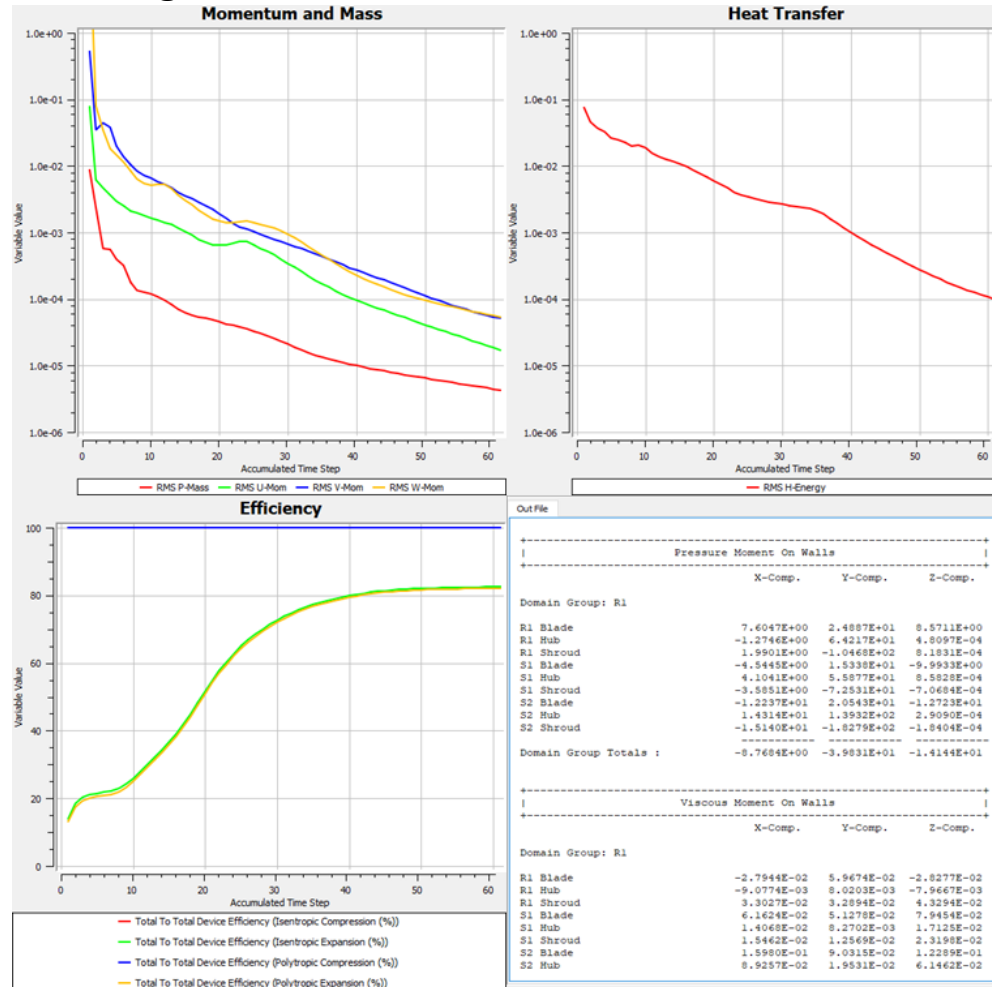
In *Outline* double click on *Solver Control* & review the solver settings



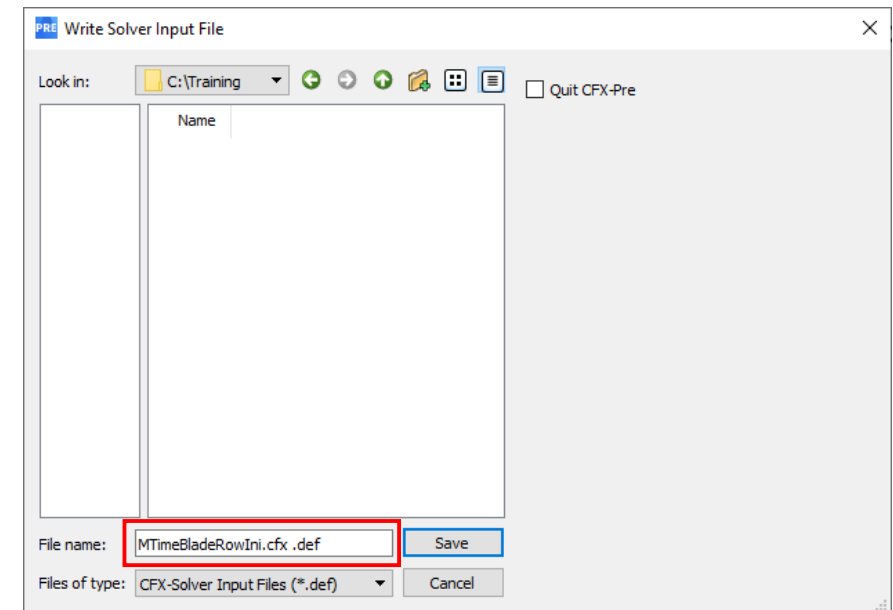
Click on *Apply* to save any changes

# Obtaining a Steady State Solution

- All steps for setting-up a steady state simulation are complete
- Run steady state simulation. You will use these results for initializing the transient TT simulation



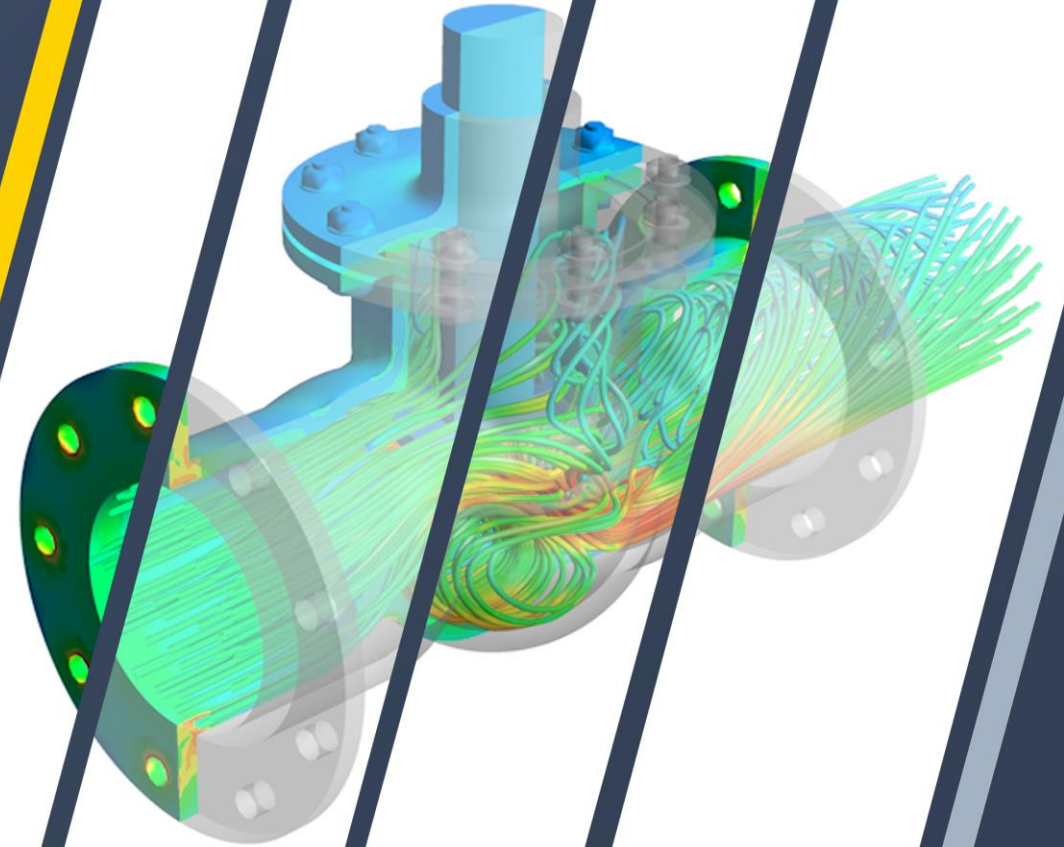
Click on *Define Run*



Accept Default name and click *Save*



## Step 2 Setting up a Transient Blade Row Simulation Using TT method



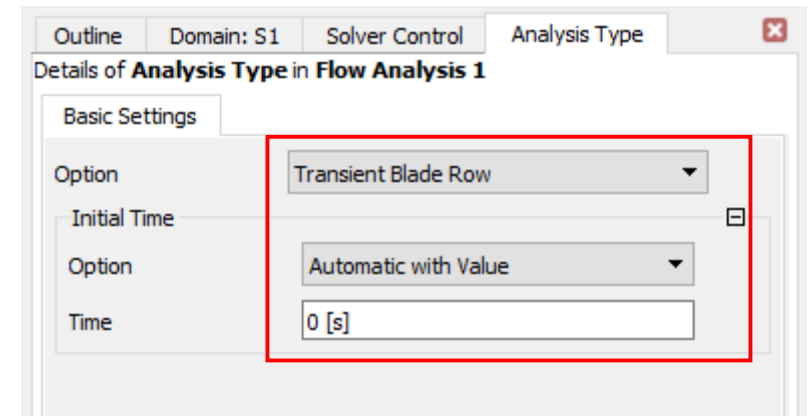


# Preparing TT Simulation

- Create a duplicate of the steady state CFX file in the working directory
- Rename the file as *MTimeBladeRow.cfx*
- Open above file in *CFX-Pre* for setting up a TBR simulation using the Time Transformation (TT) method

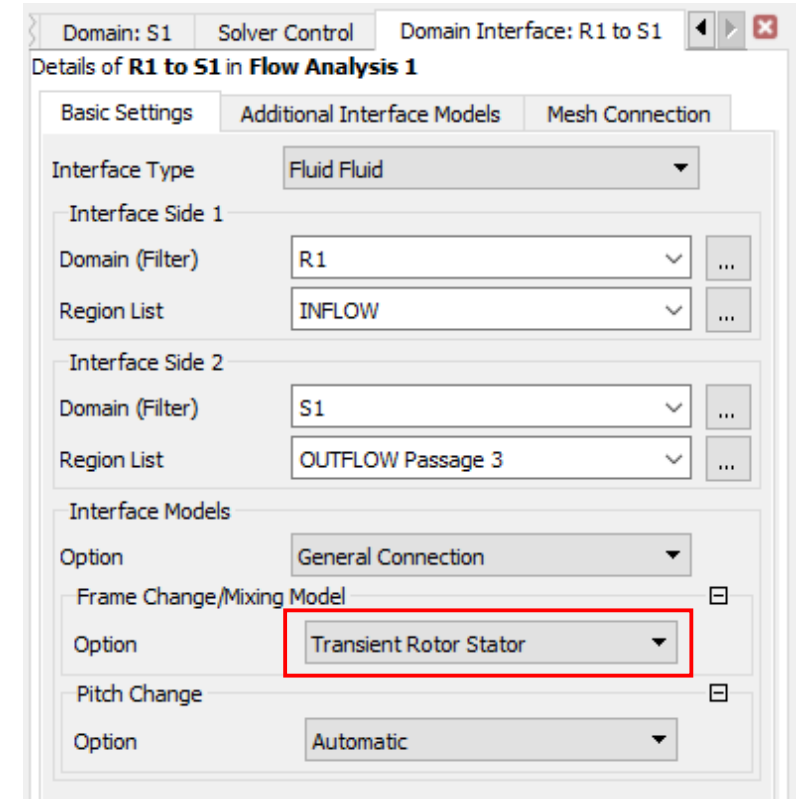
# Modifying Analysis Type

- Double click on *Analysis Type*
- Do the selection as shown to the right by red box



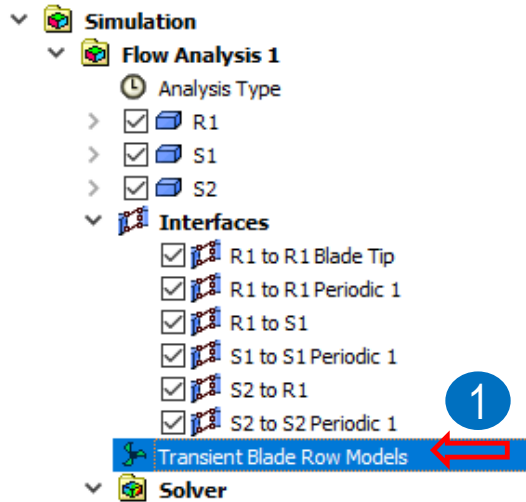
# Modifying R1-S1 & S2-R1 Interface

- Double click on *R1 to S1* interface and modify as shown
- Same modification needs to be done for *S2 to R1* interface

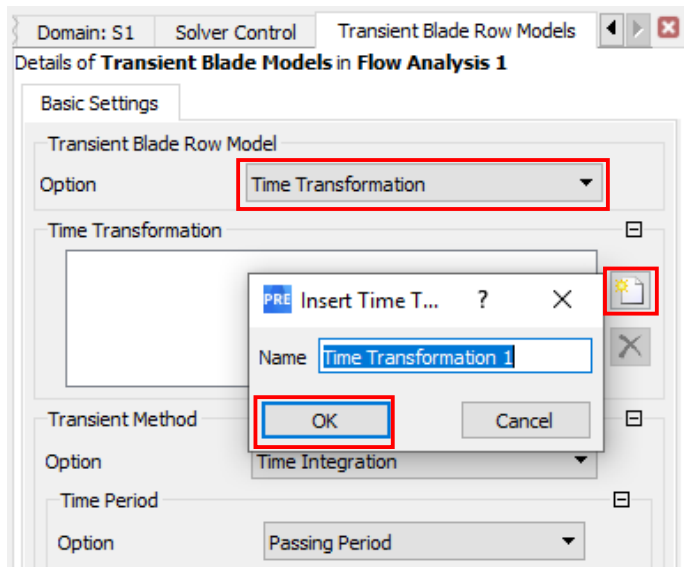


Click *Apply* after selection

# Setting up Time Transformation



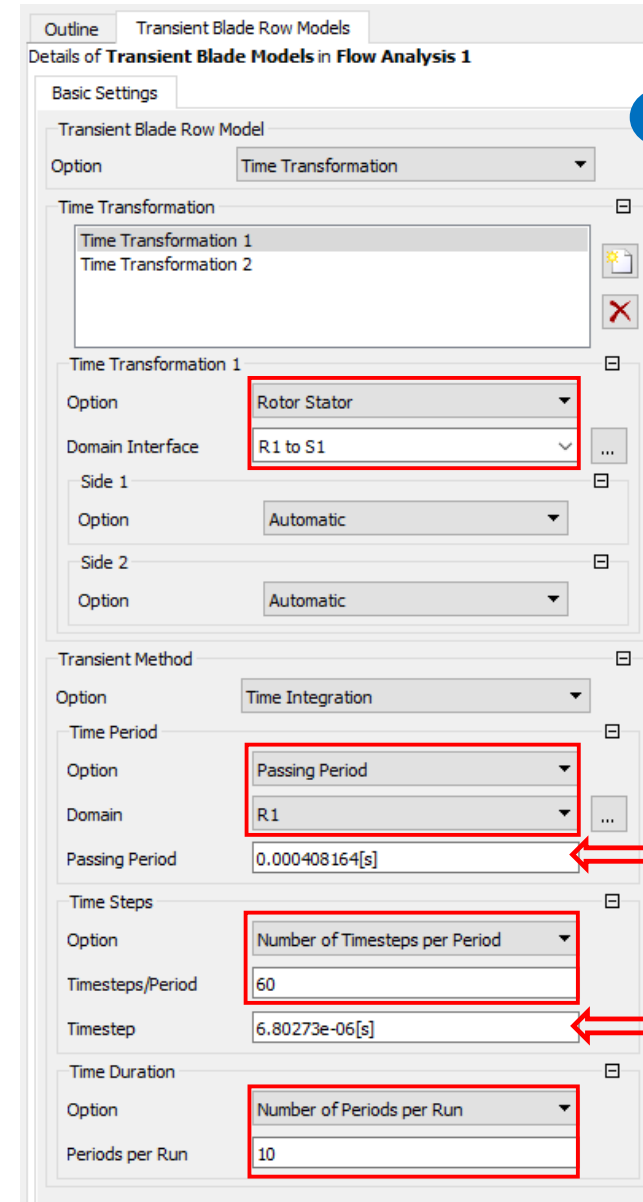
Double click



Select *Time Transformation*

Click to add new

Accept default name & Click *OK*



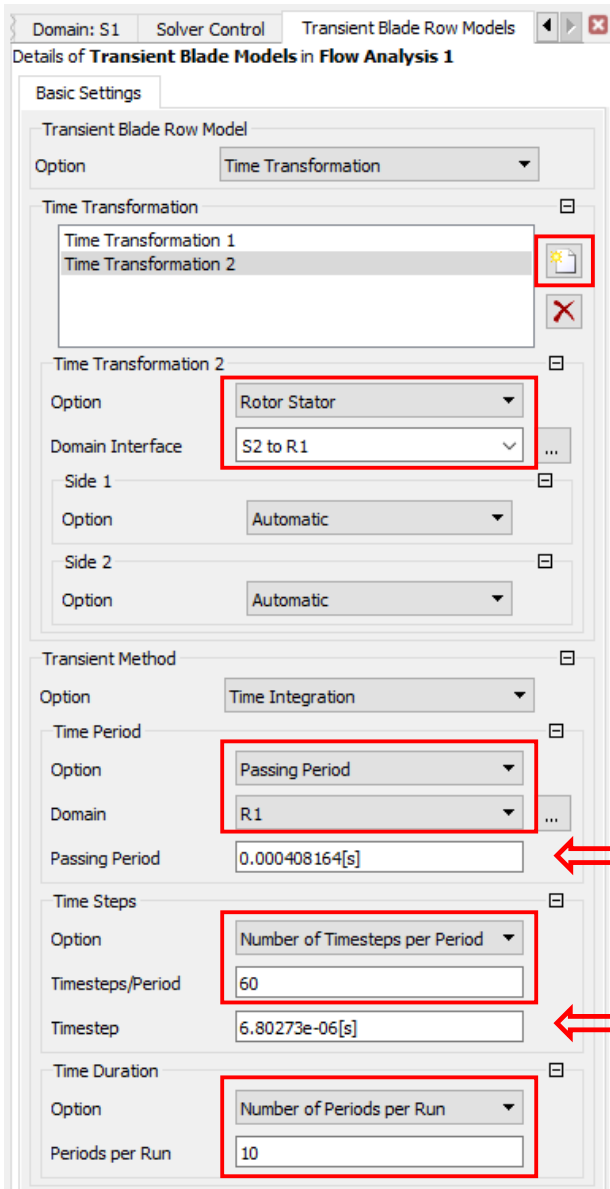
Values in red boxes are selected or provided

Auto calculated

Auto calculated

Click *Apply* after entering inputs

# Setting up Time Transformation



Add one more *Time Transformation*

Auto calculated

Auto calculated

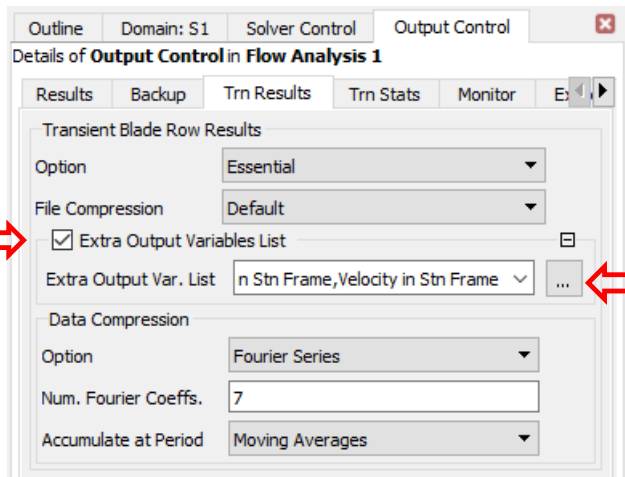
$$\text{Passing period} = 2 * \pi / (\text{Number of Blades} * \text{Angular Velocity})$$

$$\text{Time step} = \text{Passing Period} / \text{Number of Time steps per Period}$$

Click *Apply* after entering inputs

# Setting Output Control

*Transient Results tab*

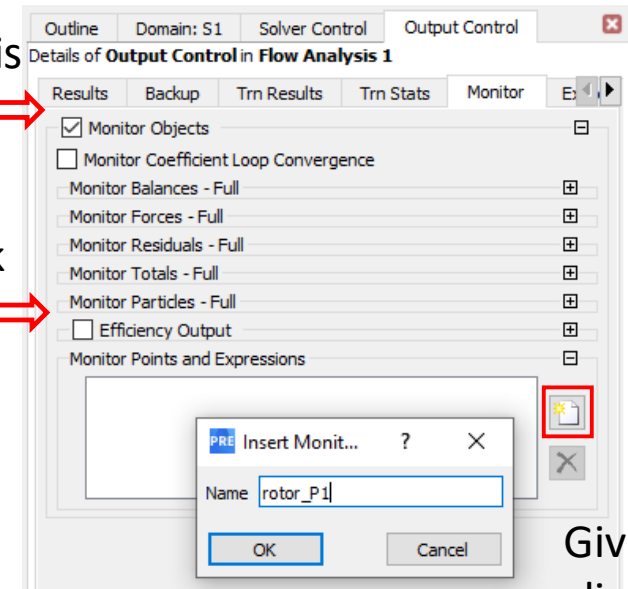


Added two *Extra Output Variables*:  
*Mach Number in Stn Frame, Velocity in Stn Frame*

*Monitor tab*

Check this box

Un-check this box



Click to add  
*Monitor Point*

Give name &  
click OK

Refer to next slide for filling in  
inputs for *Monitor Point*  
*rotor\_P1*

# Setting Output control

Monitor Points and Expressions

rotor\_P1

Option: Cylindrical Coordinates

Output Variables List: 'e,Total Temperature,Velocity

Position Axial Comp.: 0.211 [m]

Position Radial Comp.: 0.2755 [m]

Position Theta Comp.: 182 [degree]

Coordinate Frame: Coord 0

Monitor Location Control

Interpolation Type: Nearest Node

☐ Domain Name

☐ Frame Type

Position Update Frequency

Option: Initial Mesh Only

Monitor Points and Expressions

rotor\_P1  
stator\_P1

Option: Cylindrical Coordinates

Output Variables List: 'e,Total Temperature,Velocity

Position Axial Comp.: 0.202 [m]

Position Radial Comp.: 0.2755 [m]

Position Theta Comp.: 178 [degree]

Coordinate Frame: Coord 0

Monitor Location Control

Interpolation Type: Nearest Node

☐ Domain Name

☐ Frame Type

Position Update Frequency

Option: Initial Mesh Only

Monitor Points and Expressions

rotor\_P1  
rotor\_P2  
stator\_P1

Option: Cylindrical Coordinates

Output Variables List: 'e,Total Temperature,Velocity

Position Axial Comp.: 0.27 [m]

Position Radial Comp.: 0.2755 [m]

Position Theta Comp.: 176 [degree]

Coordinate Frame: Coord 0

Monitor Location Control

Interpolation Type: Nearest Node

☐ Domain Name

☐ Frame Type

Position Update Frequency

Option: Initial Mesh Only

Monitor Points and Expressions

rotor\_P1  
rotor\_P2  
stator\_P1  
stator\_P2

Option: Cylindrical Coordinates

Output Variables List: 'e,Total Temperature,Velocity

Position Axial Comp.: 0.28 [m]

Position Radial Comp.: 0.2755 [m]

Position Theta Comp.: 174 [degree]

Coordinate Frame: Coord 0

Monitor Location Control

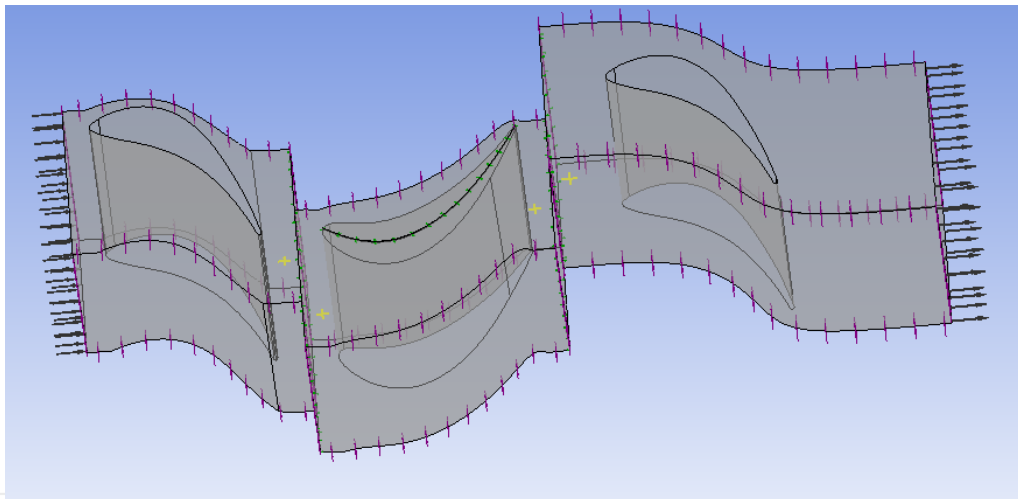
Interpolation Type: Nearest Node

☐ Domain Name

☐ Frame Type

Position Update Frequency

Option: Initial Mesh Only

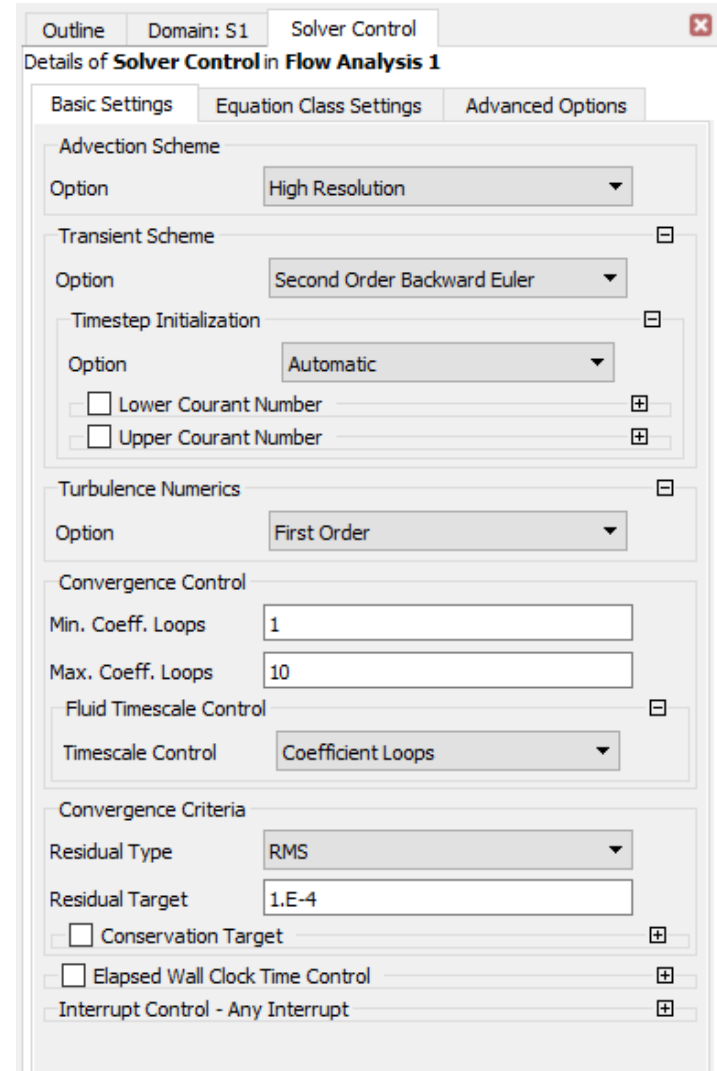


- Four *Monitor Points* are created as shown above
- *Output variable List* is: *Pressure, Temperature, Total Pressure, Total Temperature, Velocity*
- These *Monitor Points* can be seen as yellow cross symbols in the adjacent picture



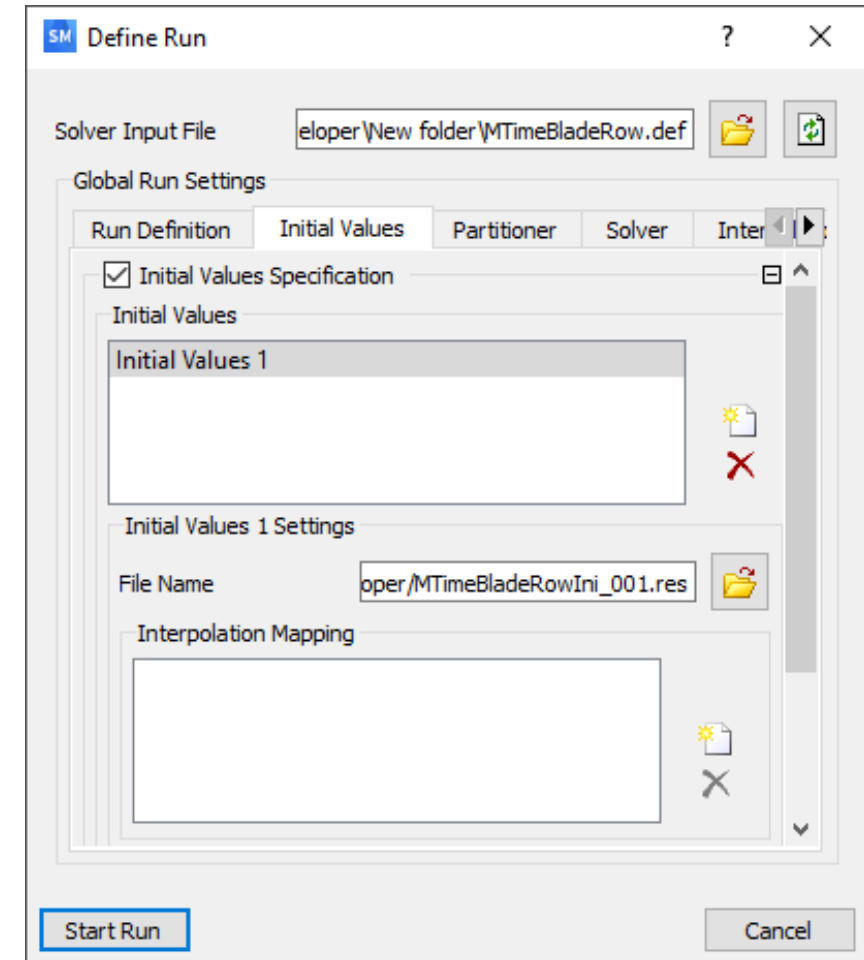
# Solver Settings

- In *Outline*, double click on *Solver Control* & verify settings as shown



# Saving the Files and Running the Simulation

- Save *.cfx* and *.def* files
- Use steady state results for initializing this TBR simulation
- Do a serial run as the global mesh has approximately 55000 nodes only



# Saving the Files and Running the Simulation

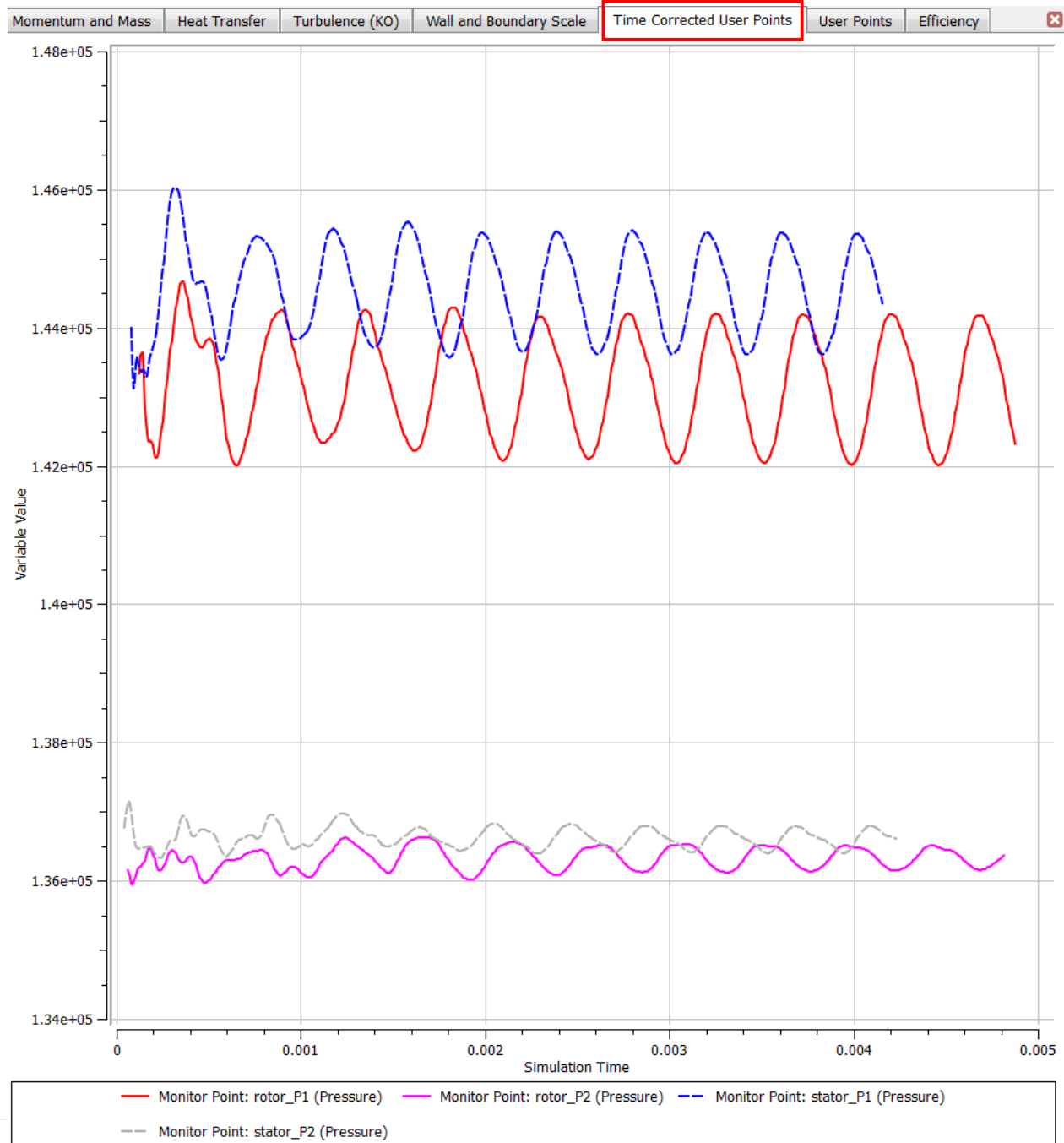
- Extra Information in *.out* file for TBR run
  - Before the simulation begins, the *Transient Blade Row Post-processing Information* summary in the *.out* file will display the time step range over which the solver will accumulate the Fourier coefficients of the Time Transformation solution
  - Similarly, the *Time Transformation Stability* summary displays whether the rotor-stator pitch ratio is within the acceptable range

Transient Blade Row Post-processing Information				
Number of Fourier coefficients = 7				
Domain Name	Disturbance period	Fourier coefficient accumulation time step range		
		Start	End	
R1	4.5045E-04	100	100	
	4.6296E-04	100	100	*
S1	3.9683E-04	100	100	*
S2	3.9683E-04	100	100	*
*: The Fourier coefficients were initialized from the solution that was provided as initial condition.				

Time Transformation stability limits			
Pitch ratio computed using stationary/rotating component pitches			
Disturbance name	Pitch ratio		
	Minimum	Maximum	Current
Time Transformation 2	0.79	1.27	1.14 Good
Time Transformation 1	0.79	1.27	1.17 Good

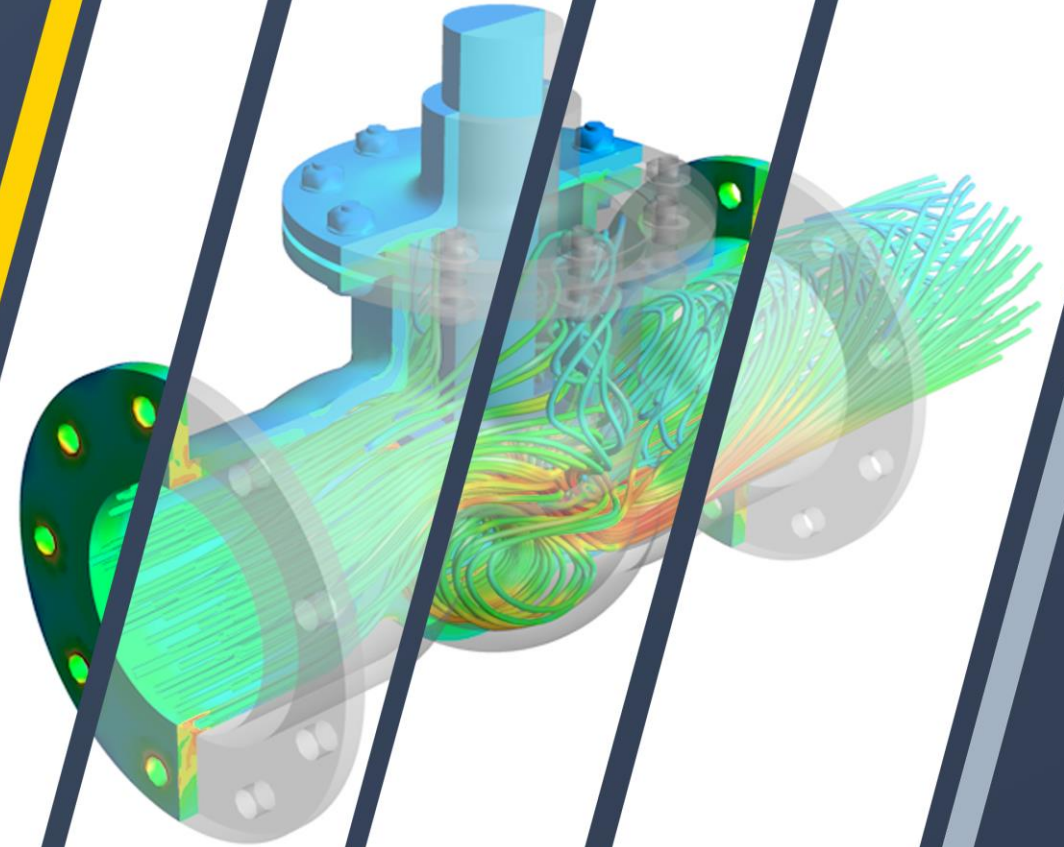
# Monitoring During the Run

- After the *CFX-Solver Manager* has run for a short time, you can track the monitor points you created in *CFX-Pre* by clicking the *Time Corrected User Points* tab at the top of the graphical interface of *CFX-Solver Manager*
- As the run is concluded in approximately 3 hours, the output and result files of this workshop are provided with the workshop inputs
  - *MTimeBladeRow\_002.out*
  - *MTimeBladeRow\_002.res*



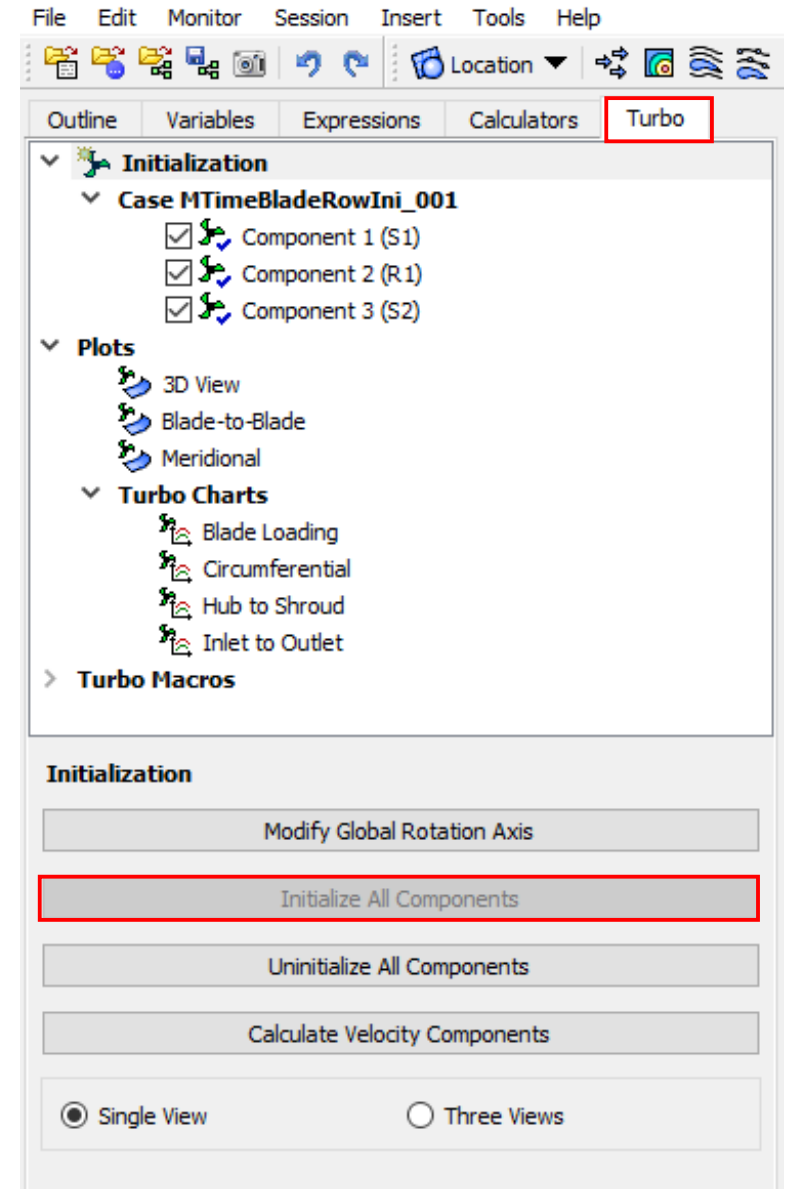


# Post Processing in CFD-Post



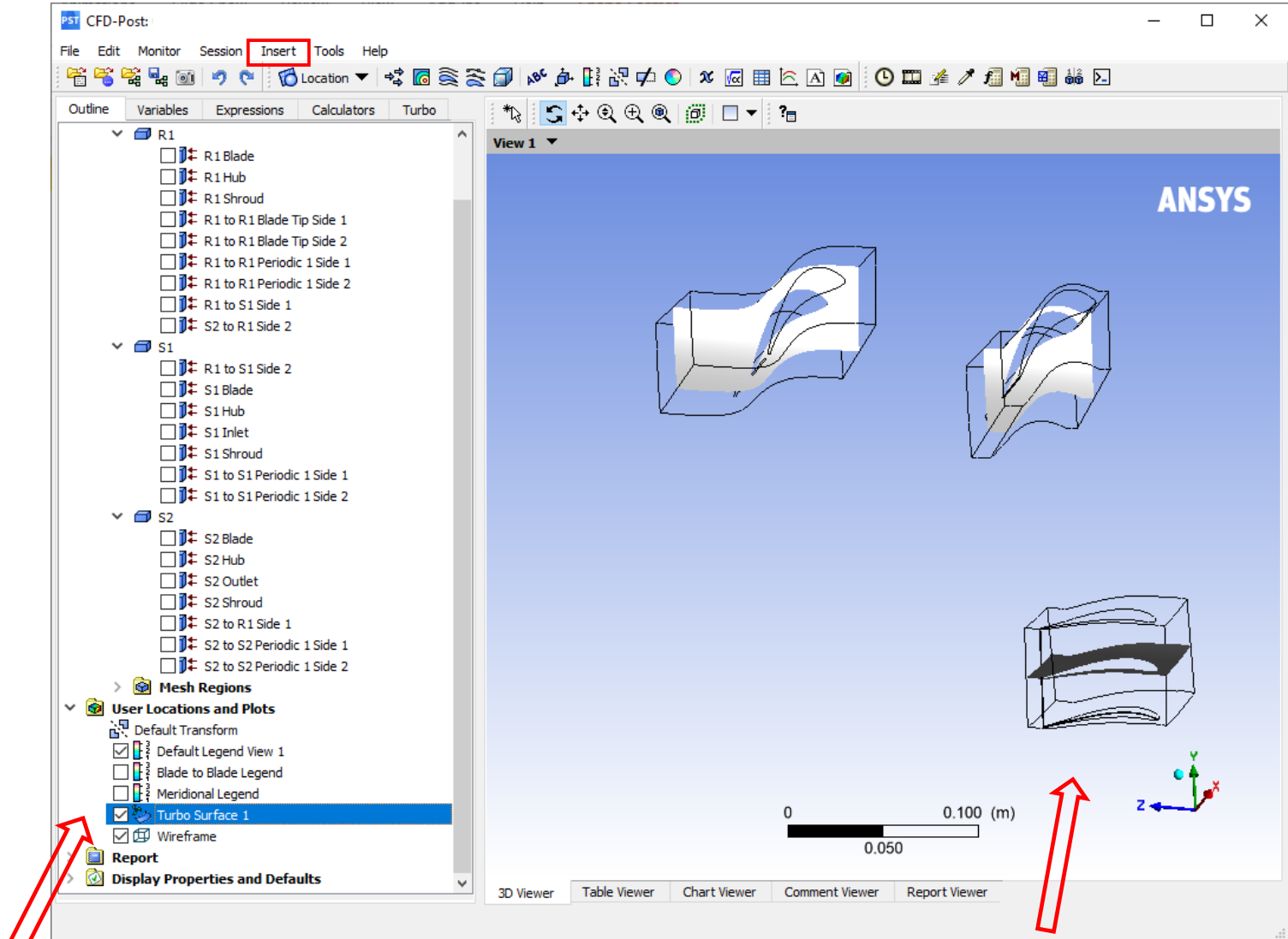
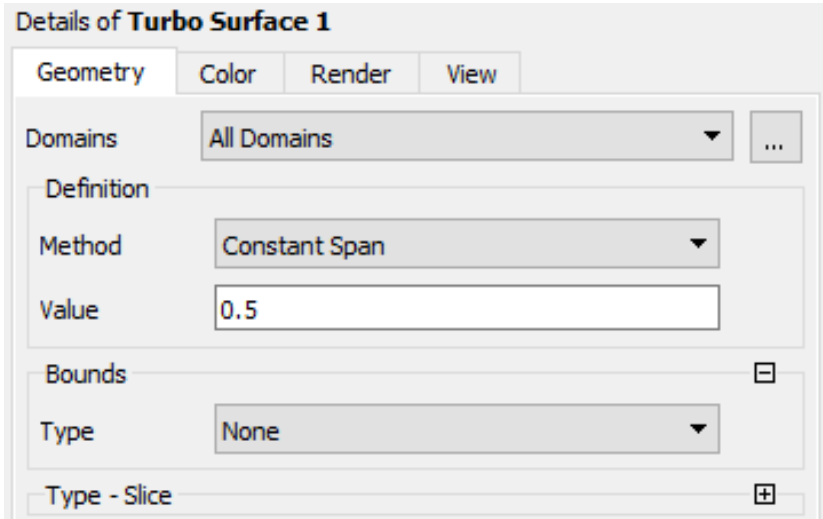
# Turbo Initialization

- Open result file in *CFD-Post*. Load all 3 domains and click *OK* in the *Transient Blade Row Post-processing* warning dialog box
- Select the *Turbo* tab
- A dialog box will ask if you want to auto-initialize all turbo components. Click *Yes*



# Creating a Turbo Surface

- *Insert>Location>Turbo Surface*
- Chose constant span and provide value as 0.5
- Click *OK*



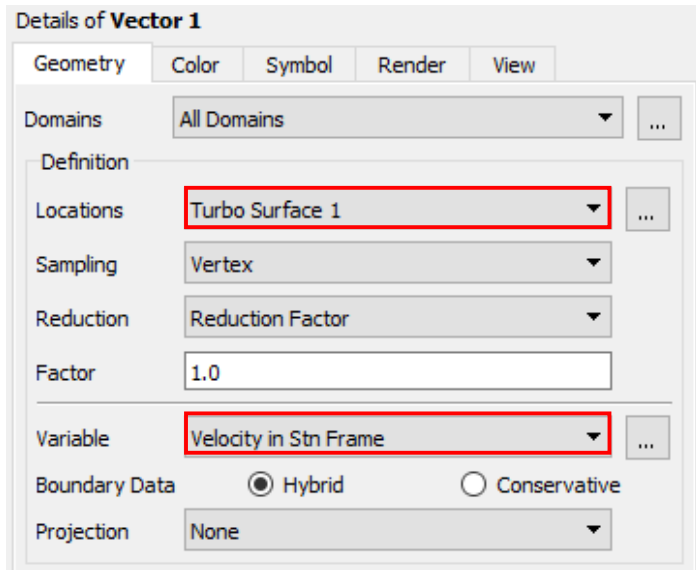
Uncheck this box for switching off from view

Rotor is rotated by 10 periods of run time and hence not aligned with stator

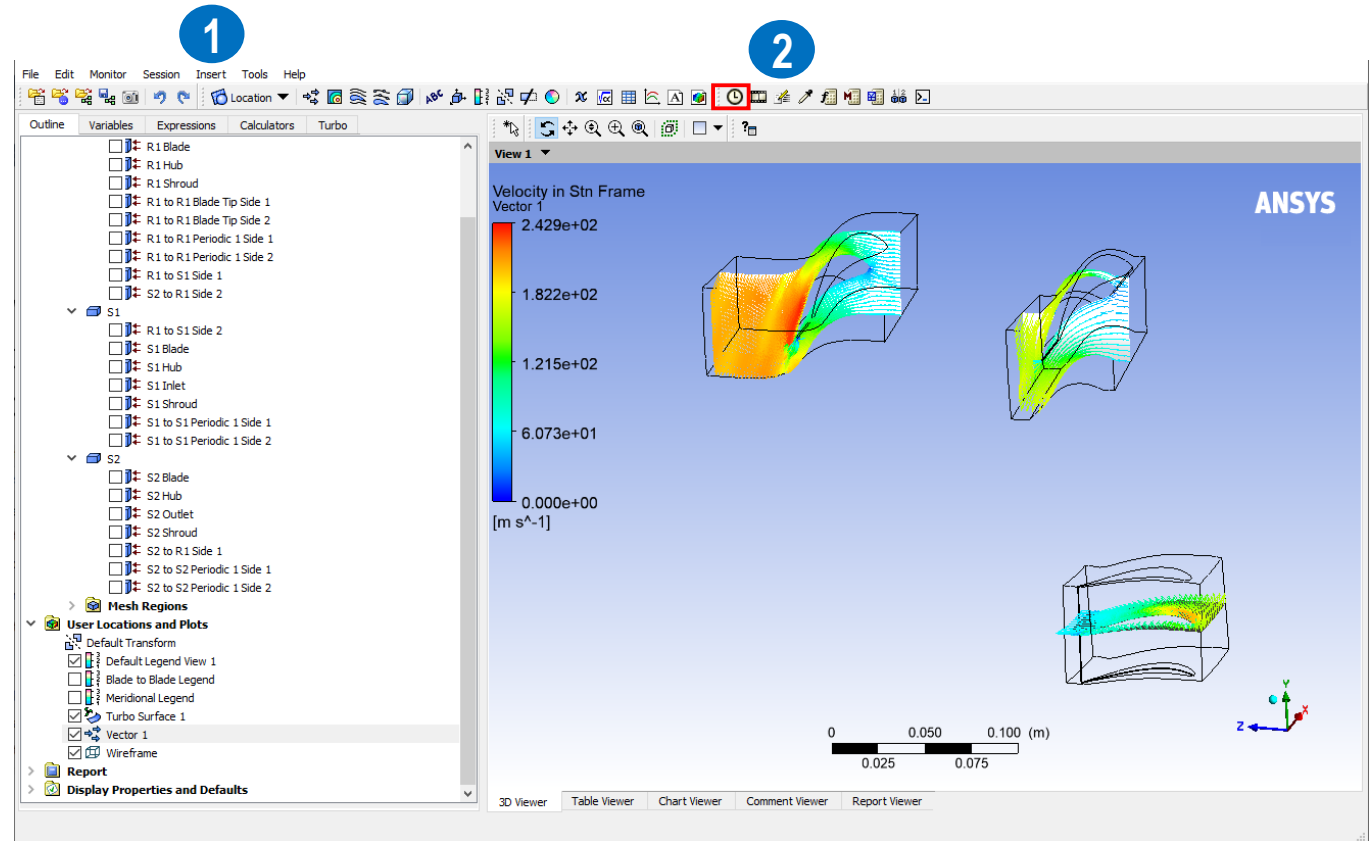


# Creating a Vector Plot

- *Insert>Vector* 1
- Do selections as shown below
- Click *Apply*



Switch off *Vector 1* by unchecking box next to it in the *Outline* at the end



Timestep Selector

MTimeBladeRow\_001

Loaded Timestep: 100 - Final

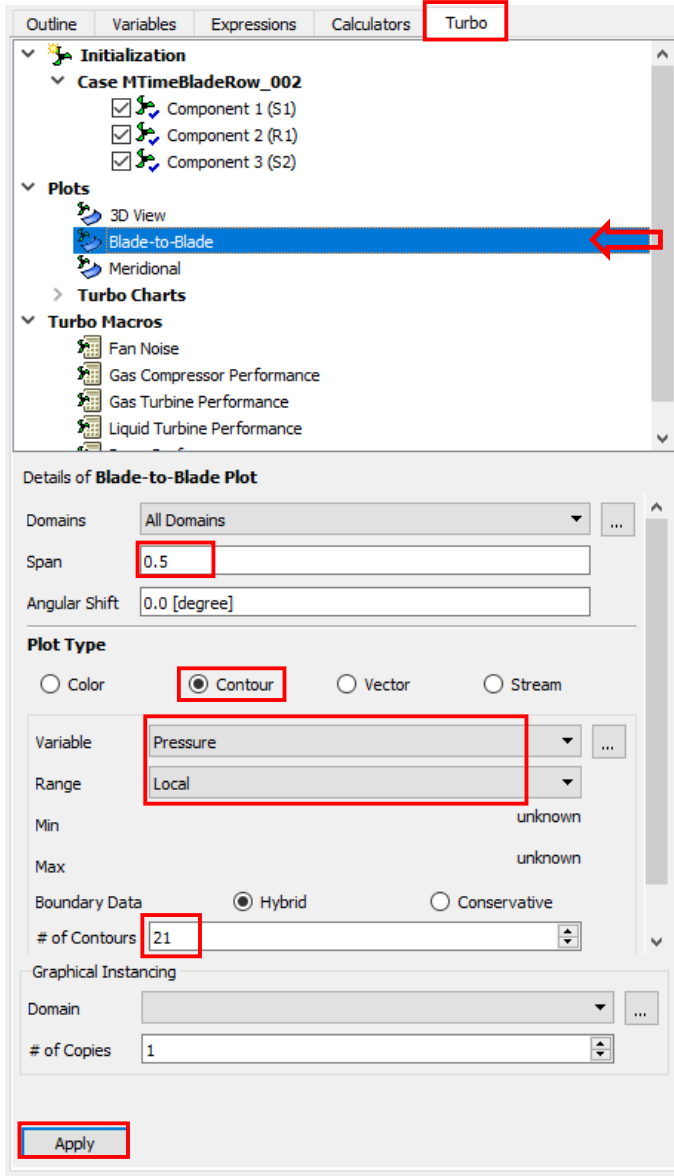
#	Step	Solver Step	Time [s]	Phase
1	0	61	0	0
2	1	62	6.80274e-6	0.000396825
3	2	63	1.36055e-5	0.000793651
4	3	64	2.04082e-5	0.00119048
5	4	65	2.7211e-5	0.0015873
6	5	66	3.40137e-5	0.00198413
7	6	67	4.08164e-5	0.00238095
8	7	68	4.76192e-5	0.00277778
9	8	69	5.44219e-5	0.0031746
10	9	70	6.12246e-5	0.00357143
11	10	71	6.80274e-5	0.00396825

Timestep Sampling: Simulation Timestep

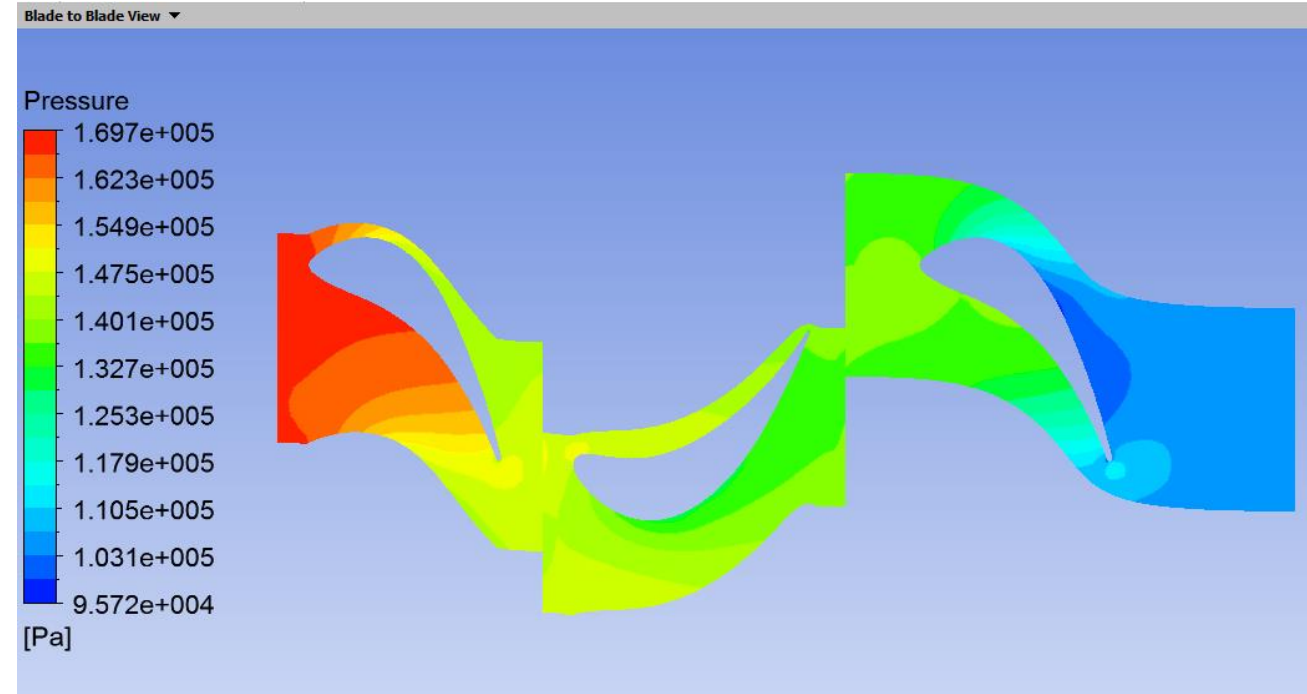
Apply Reset Close

- Click on *Timestep Selector*
- Click on 1<sup>st</sup> timestep and *Apply*
- Now rotor will move back to its starting position

# Blade to Blade Plot, Turbo Tab



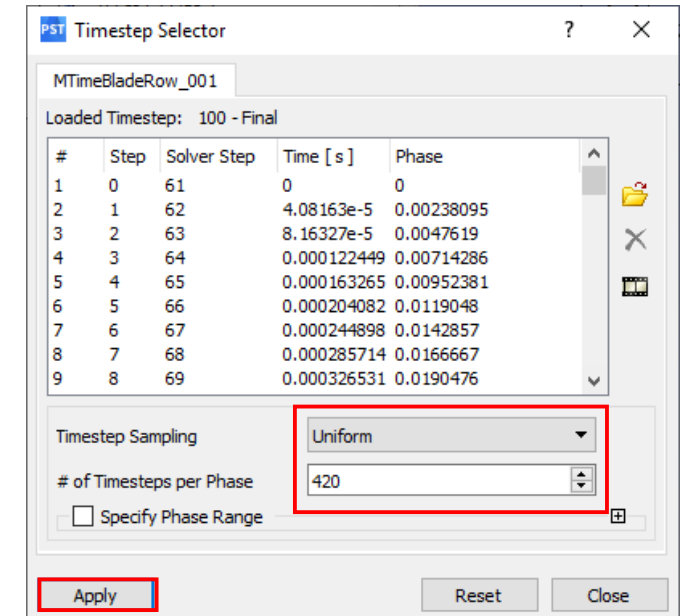
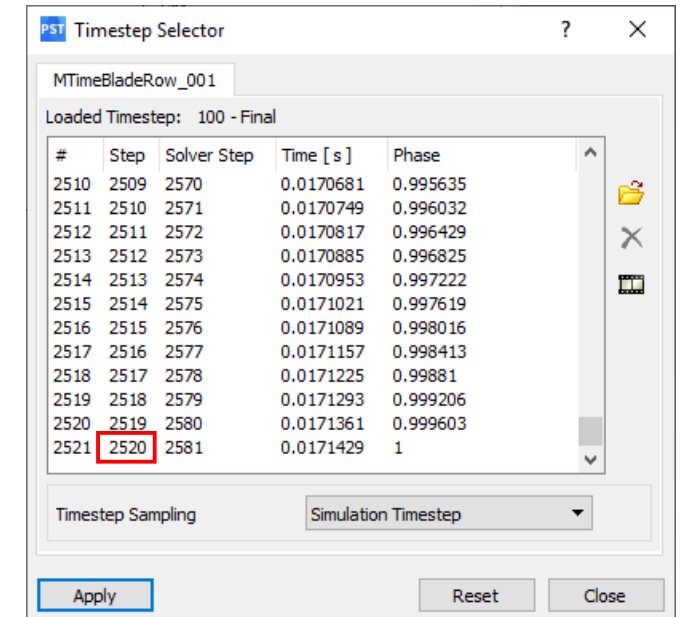
Double click



Pressure contour for selected time step is shown

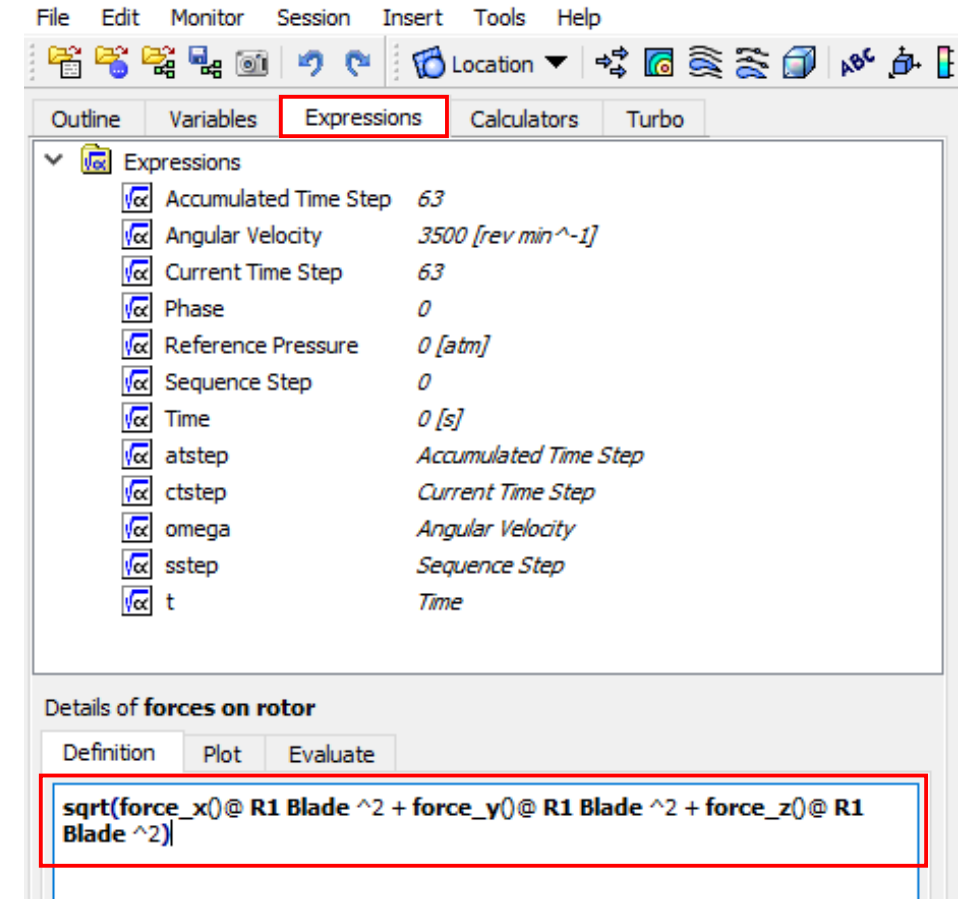
# Time Steps in Result File

- For a transient blade row case, CFD-Post automatically reconstructs variables for the flow solution for the last time step. Intermediate time steps for time instances in the common period are located in the *Timestep Selector*.
- We used 60 time steps per rotor blade passing period and there are 42 rotor blade passing periods in a common period. Therefore, the total number of intermediate time steps in the common period is  $60 \times 42 = 2520$ .
- For this case, the solver has reconstructed results over one common period (2520 time steps). You will reduce the total number of time steps to 420 to speed up the generation of the time chart.



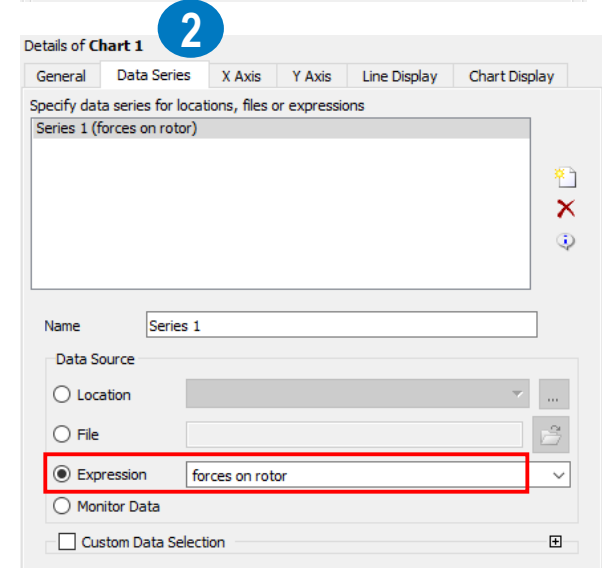
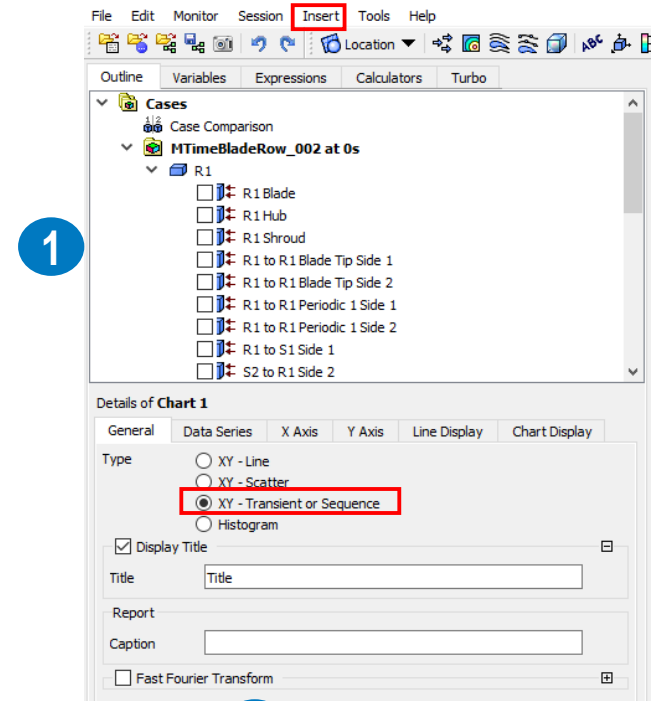
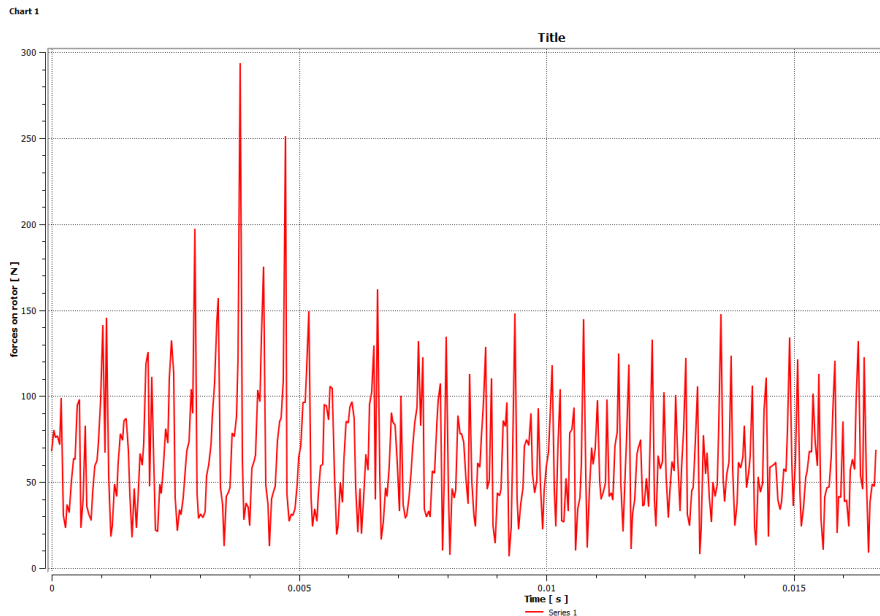
# Creating Expression for Force on Blade

- *Insert>Expression*
- Give name *forces on rotor*
- Enter value as  $\sqrt{\text{force}_x()@ R1 \text{ Blade}^2 + \text{force}_y()@ R1 \text{ Blade}^2 + \text{force}_z()@ R1 \text{ Blade}^2}$
- Click *Apply*



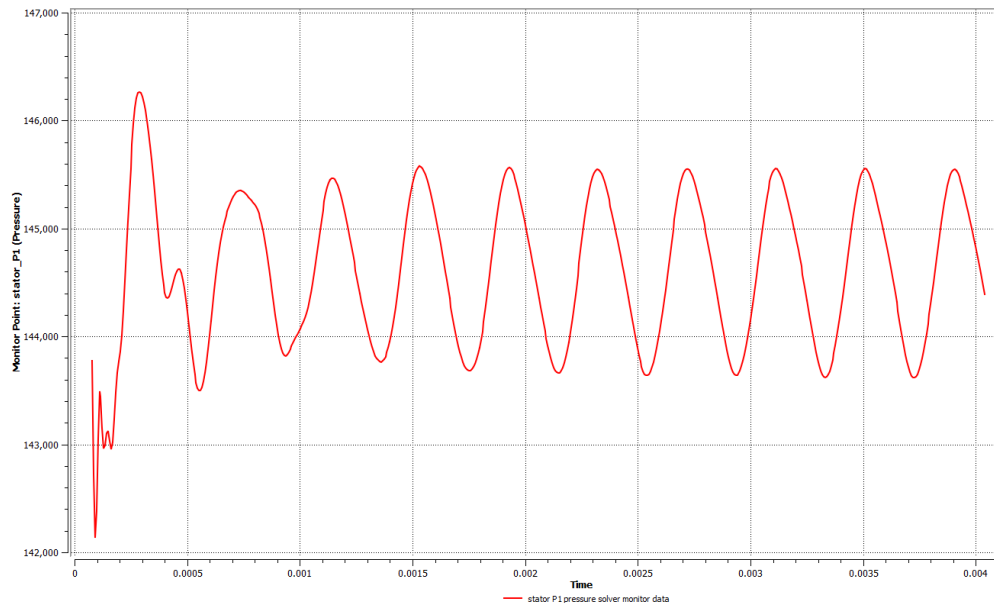
# Chart Creation

- *Insert>Chart* 1
- Select type as *XY – Transient or Sequence*
- *Data Series* tab 2
  - Select *Data Source* as *Expression* and chose expression created in previous slide
- Click *Apply* to generate the chart (shown below)



# Chart Creation – Monitor data

- In the same way
  - *Insert>Chart*
  - Select type as *XY – Transient or Sequence*
- *Data Series* tab (provide a meaningful data series *Name*)
  - Select *Data Source* as *Monitor Data*
  - Select axis variables
    - *X Axis* → *Time*
    - *Y Axis* → *Monitor Point: stator\_P1 (Pressure)*



Details of **Chart 2**

General **Data Series** X Axis Y Axis Line Display Chart Display

Specify data series for locations, files or expressions

stator P1 pressure solver monitor data

Name **stator P1 pressure solver monitor data**

Data Source

☐ Location ☐ File ☐ Expression ☒ **Monitor Data**

Monitor Variable Selection

Source ☒ Current Cases ☐ File

X Axis

Variable **Time**

☐ Take absolute value of data points

Y Axis

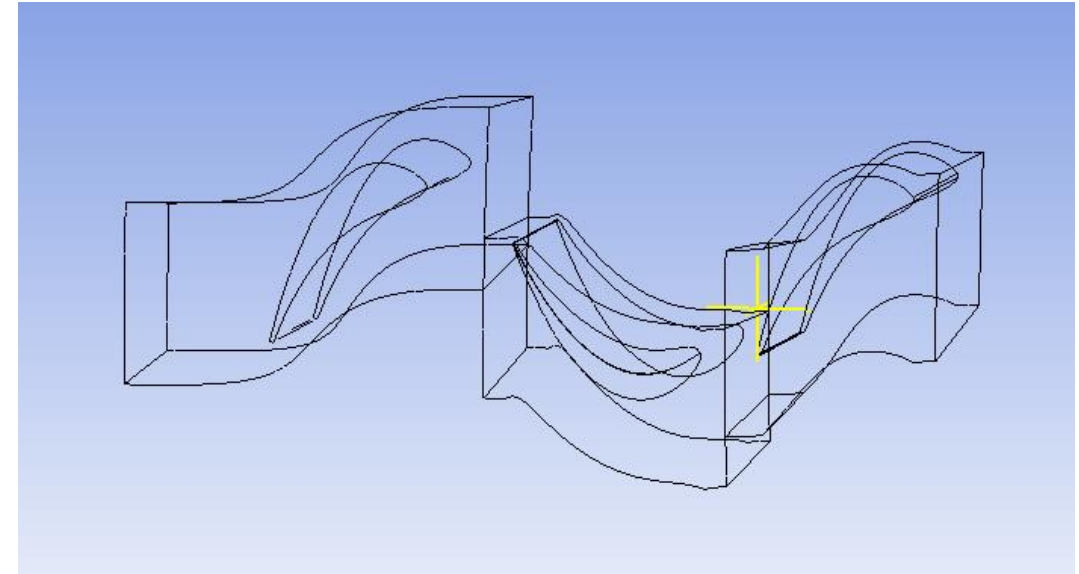
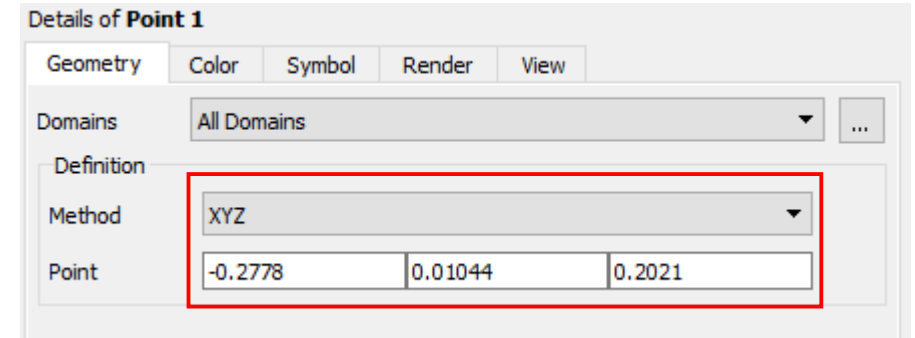
Variable **Monitor Point: stator\_P1 (Pressure)**

☐ Take absolute value of data points

# Creating a Point

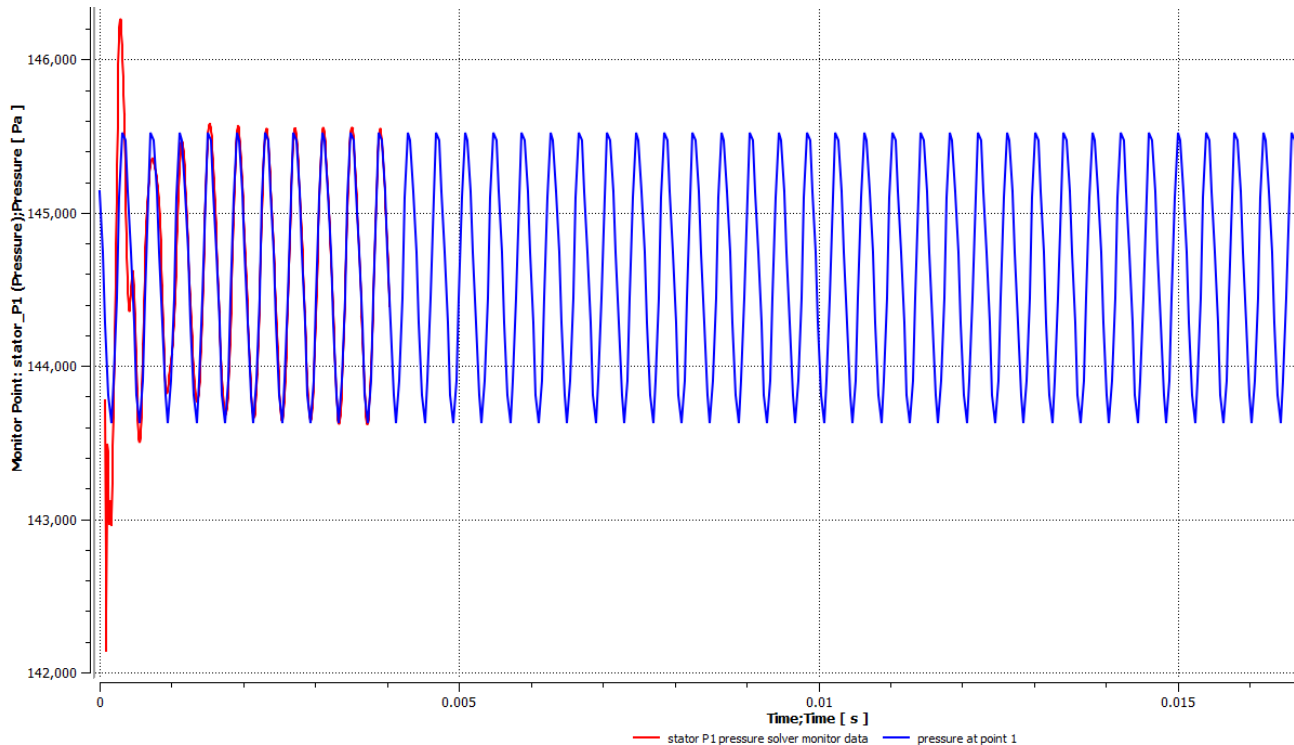
- *Insert>Location>Point*
- Accept default name
- Enter XYZ coordinates as shown (corresponding to *Monitor Point stator\_P1* created earlier in *CFX-Pre*)
- Click *Apply*
- Observe location of *Point 1* in *3D Viewer* window

Intention of adding this point is to compare pressure signal reconstructed from Fourier Coefficients at this point with solver monitor stator\_P1



# Chart Creation – at Point 1

- Double click on *Chart 2* in the *Outline*
- Go to *Data Series* tab
- Do selections as shown by red boxes



*Stator\_P1* Pressure data (*Solver Monitor*) & *pressure point 1* (*CFD-Post* reconstructed from Fourier series) compare well

Details of **Chart 2**

General **Data Series** X Axis Y Axis Line Display Char

Specify data series for locations, files or expressions

stator P1 pressure solver monitor data  
pressure at point 1 (Point 1)

Name **pressure at point 1**

Data Source  
☒ Location **Point 1**  
☐ File  
☐ Expression Time  
☐ Monitor Data  
☐ Custom Data Selection

Add a new series

Provide name

Location → Point 1

Details of **Chart 2**

General Data Series **X Axis** Y Axis Line Display Char

Data Selection  
Expression **Time**

Details of **Chart 2**

General Data Series X Axis **Y Axis** Line Display Char

Data Selection  
Variable **Pressure**

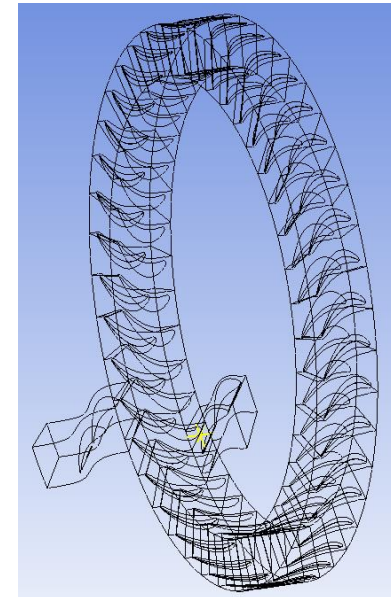
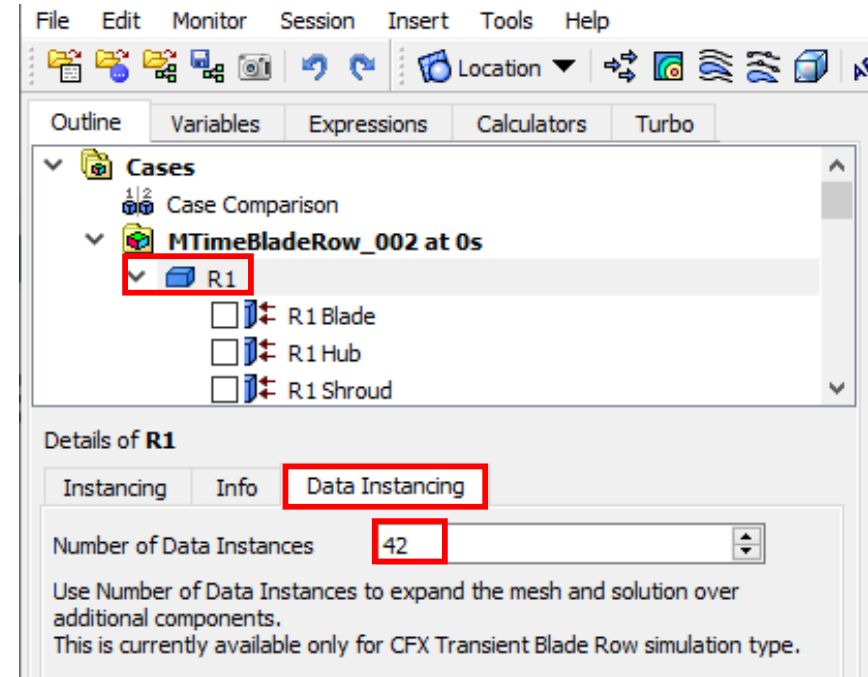


# Data Instancing Transformations

- Double click on *R1* in the *Outline*
- Go to *Data Instancing* tab
- Enter *Number of Data Instances* as 42
- Click on *Apply*

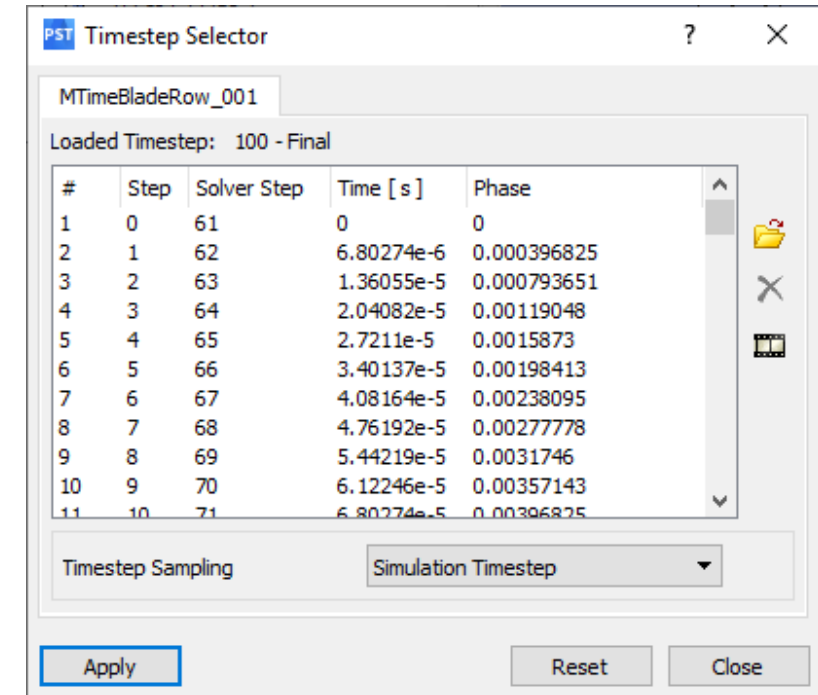
*CFD-Post* will create additional mesh nodes proportional to the number of extra passages created, and populate them with solution variables correctly updated to their corresponding position in time and space

- Repeat this for *S1* → 36 data instances
- Repeat this for *S2* → 37 data instances
- You can view contours created the on entire 360 degree mesh



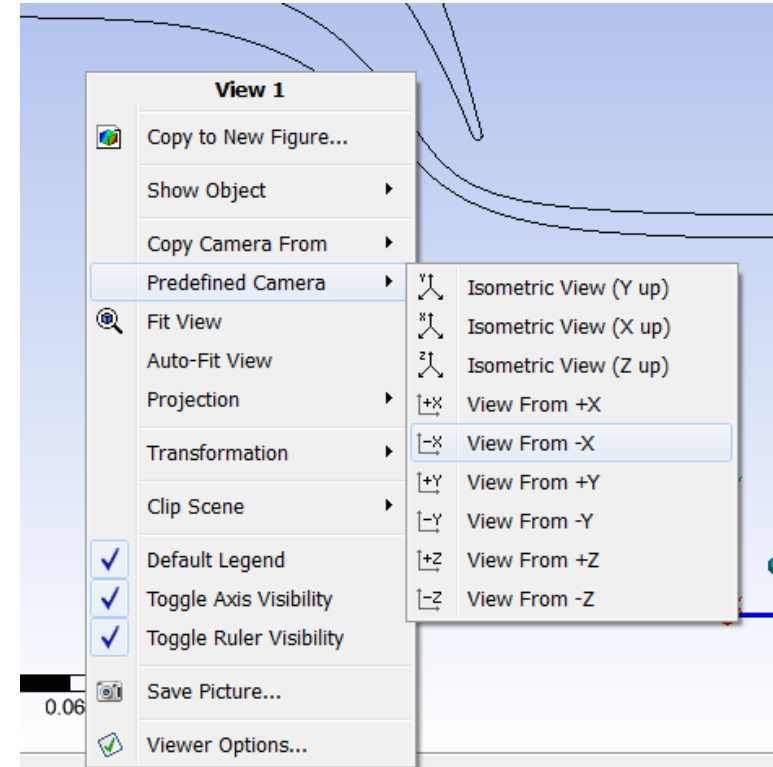
# Animating Movement of Rotor Relative to Stator

- With the *Timestep Selector* set to time step 0, you will make an animation showing the relative motion starting from this time step and lasting for one stator blade passing period



# Animating Movement of Rotor Relative to Stator

- Select the *3D Viewer* tab
- Position the geometry for the animation by right-clicking on a blank area in the viewer and selecting *Predefined Camera > View From -X*

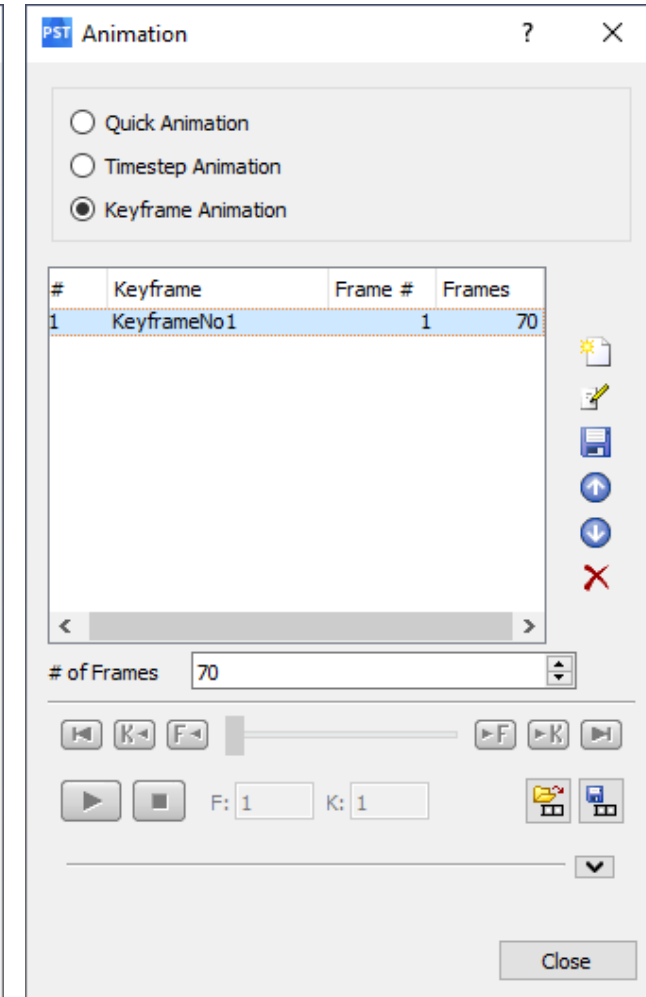
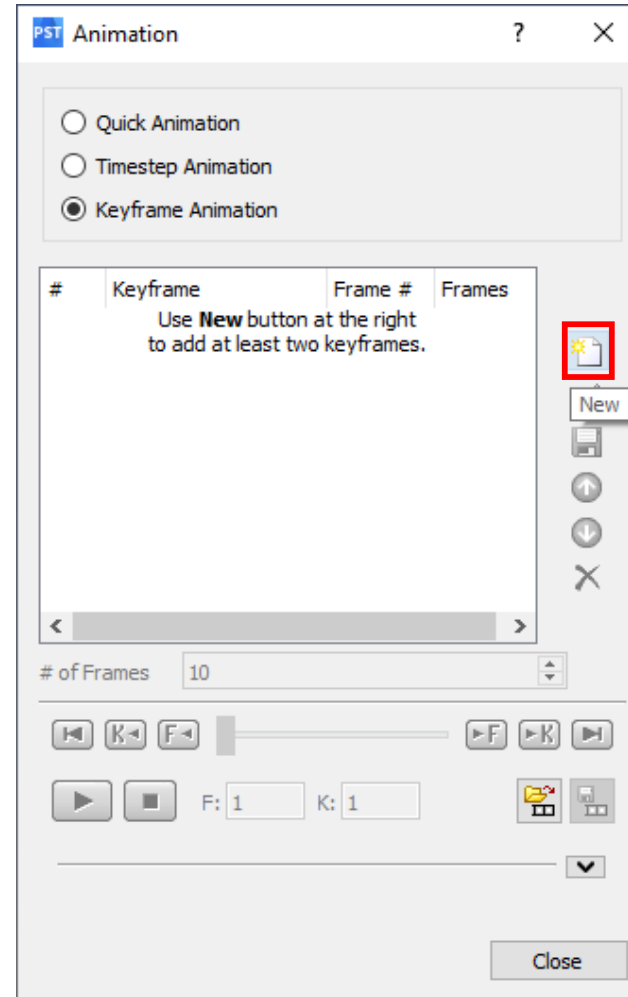


# Animating Movement of Rotor Relative to Stator



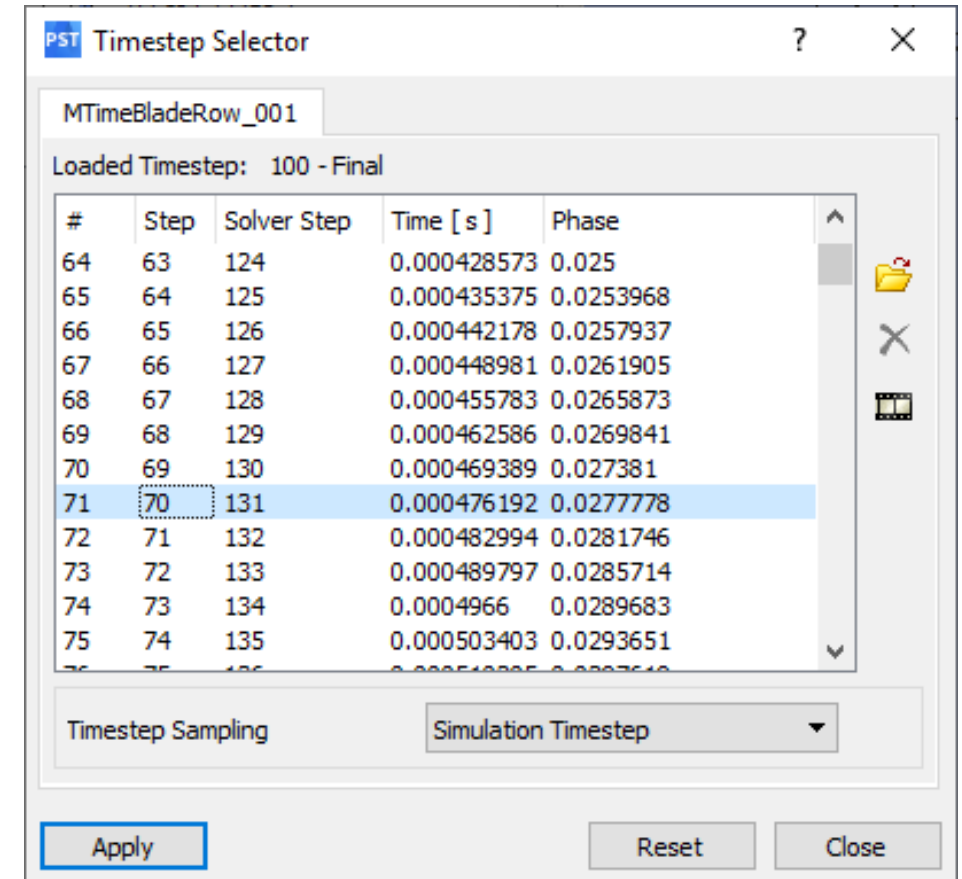
- Click *Animation*
- In the *Animation* dialog box, select *Keyframe Animation*
- Click *New* to create *KeyframeNo1*
- Select *KeyframeNo1*, then set # of *Frames* to 70, then press *Enter* while inside the # of *Frames* box

Be sure to press Enter and confirm that the new number appears in the list before continuing. This will place 70 intermediate frames between the keyframes, for a total of 72 frames.



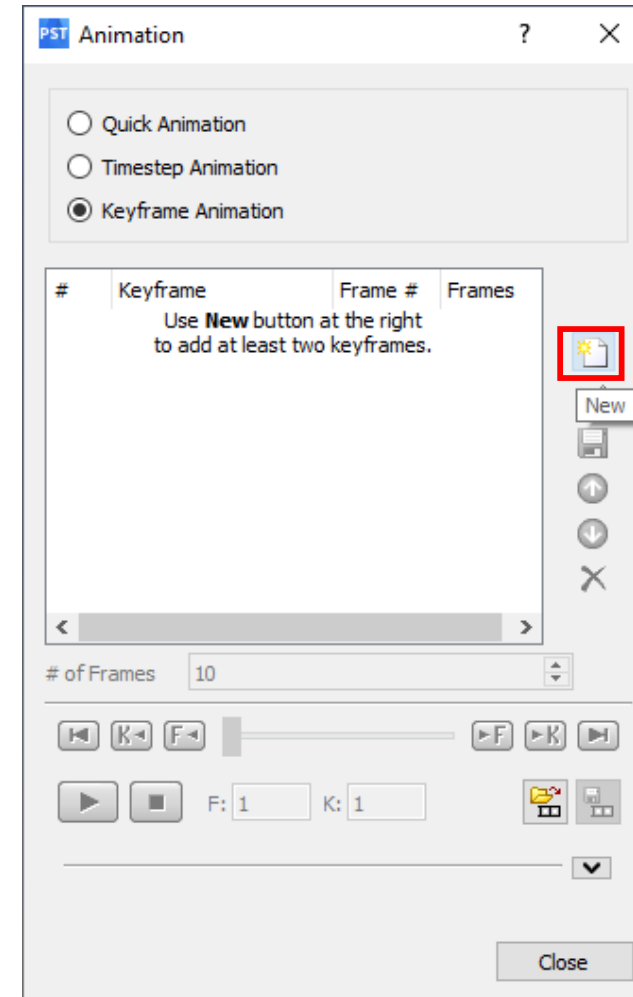
# Animating Movement of Rotor Relative to Stator

- Use the *Timestep Selector* to load time step 70 and then close this dialog box



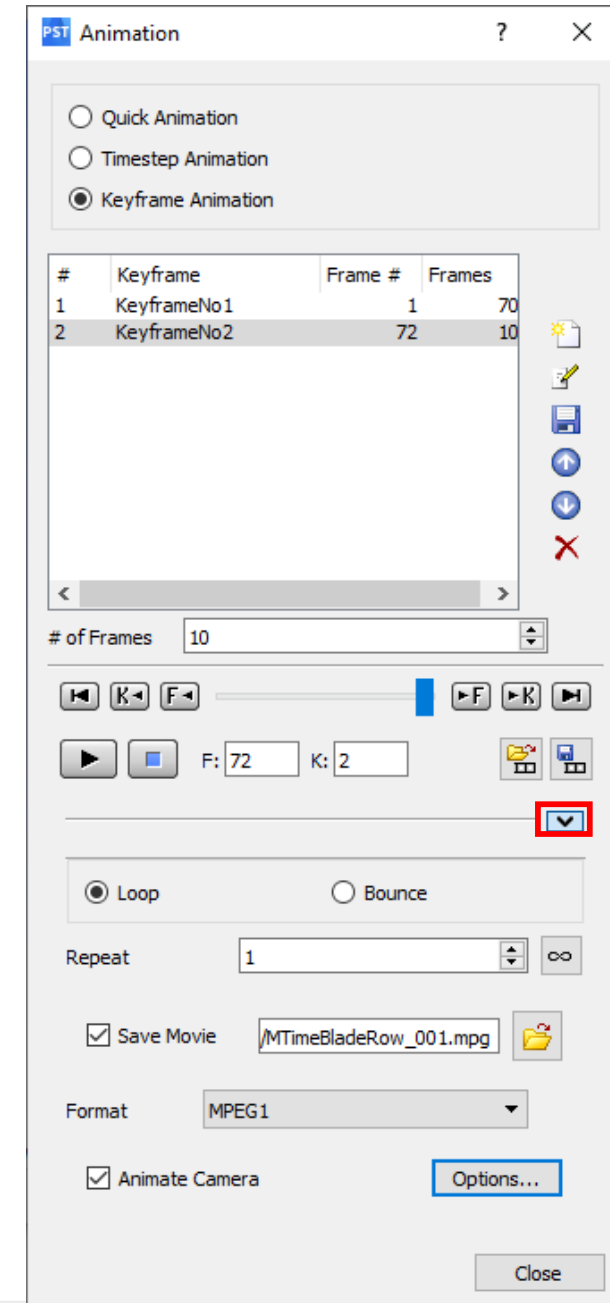
# Animating Movement of Rotor Relative to Stator

- In the *Animation* dialog box, click *New* to create *KeyframeNo2*



# Animating Movement of Rotor Relative to Stator

- Click *More Animation Options* to expand the Animation dialog box
- Select *Save Movie*
- Specify a filename for the movie
- Set Format to *MPEG1*

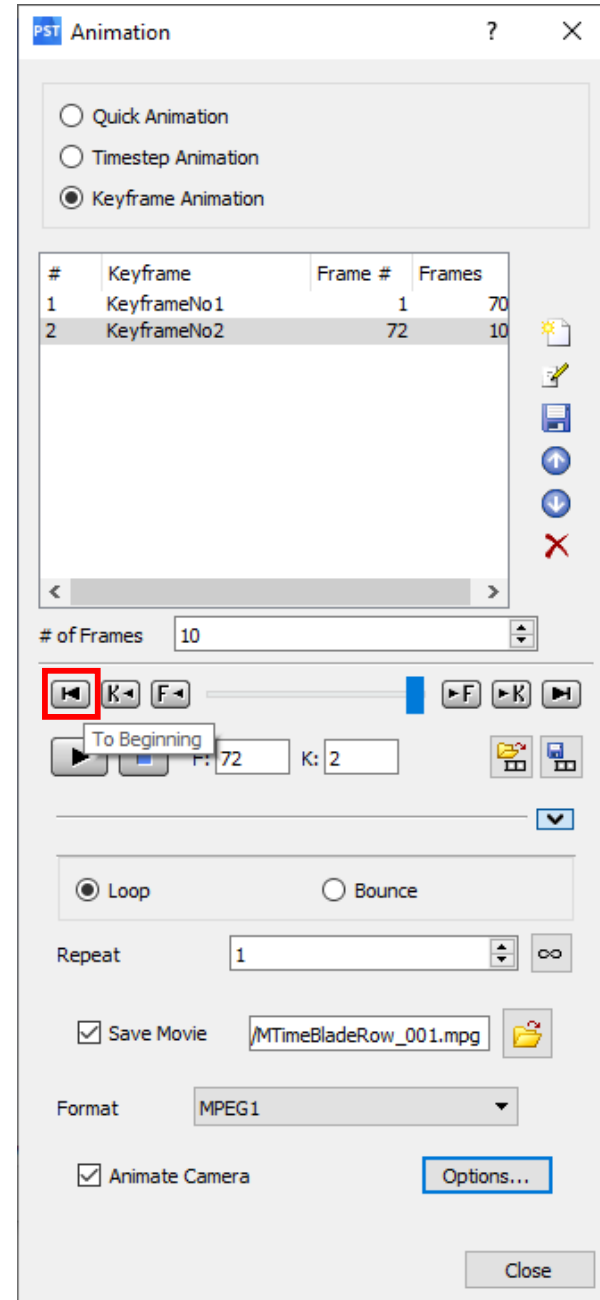


*More  
Animation  
Options*

# Animating Movement of Rotor Relative to Stator

- Click *To Beginning* to rewind the active keyframe to *KeyframeNo1*

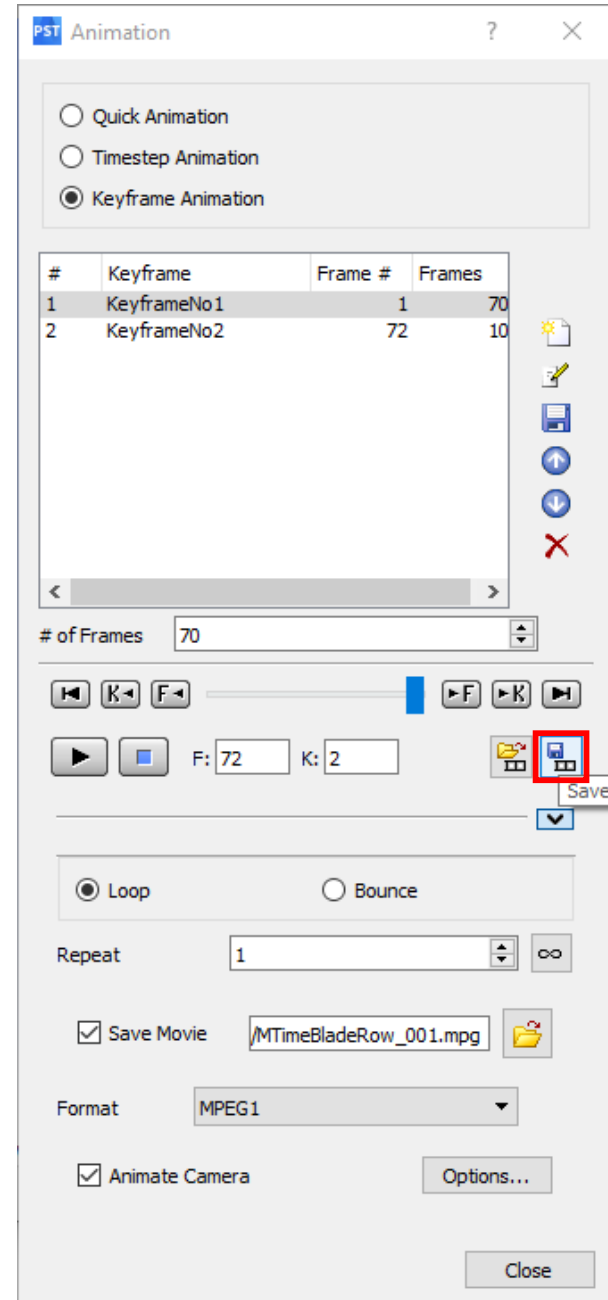
The active keyframe is indicated by the value appearing in the F: field in the middle of the Animation dialog box. In this case it will be 1. Wait for CFD-Post to finish loading the objects for this frame before proceeding.





# Animating Movement of Rotor Relative to Stator

- Click *Save Animation State* and save the animation to a file. This will enable you to quickly restore the animation settings in case you want to make changes. Animations are not restored by loading ordinary state files (those with the .cst extension).



# Animating Movement of Rotor Relative to Stator

- Click *Play* the animation

It takes a while for the animation to be completed. To view the movie file, you will need to use a media player that supports the MPEG format. From the animation and plots, you can see that the flow is continuous across the interface. This is because CFD-Post is capable of interpolating the flow field variables to the correct time and position using the computed Fourier coefficients.

- When you have finished, close the *Animation* dialog box and then close *CFD-Post*

