Politechnika Wrocławska

Mechanical and Power Engineering Faculty
Full-time studies

Selected problems of thermal-flow processes

Exercise no. 6

Modelling of flow through a blade row

Wrocław 2020

TABLE OF CONTENTS

1. Introduction ... 2
2. Flow through a turbine stage ... 3
2.1. Geometry ... 3
2.2. Numerical mesh ... 20
2.3. Numerical model ... 28
2.4. Solver ... 42
2.5. Results ... 45
3. Results to be included in the report ... 48
4. Optional tasks (not required) ... 48
5. References ... 48

1. INTRODUCTION

The exercise will show how to model the flow of compressible fluid through a turbine stage. The stage will be modeled as a two-dimensional, without taking into account the walls of the steering wheel and the rotor, except for their blades. For further simplification, the stage will consist of a single steering wheel and rotor, and the remaining blades will be modeled using periodic boundary conditions. Air treated as ideal gas at a speed of 10 m/s and a temperature of 100 °C flows into the turbine stage. The air outflows to the environment at a pressure of 1 bar. The steering wheel (stator) is stationary while the rotor rotates at 1000 rpm. The diagram of the analyzed case is presented in Fig. 1.

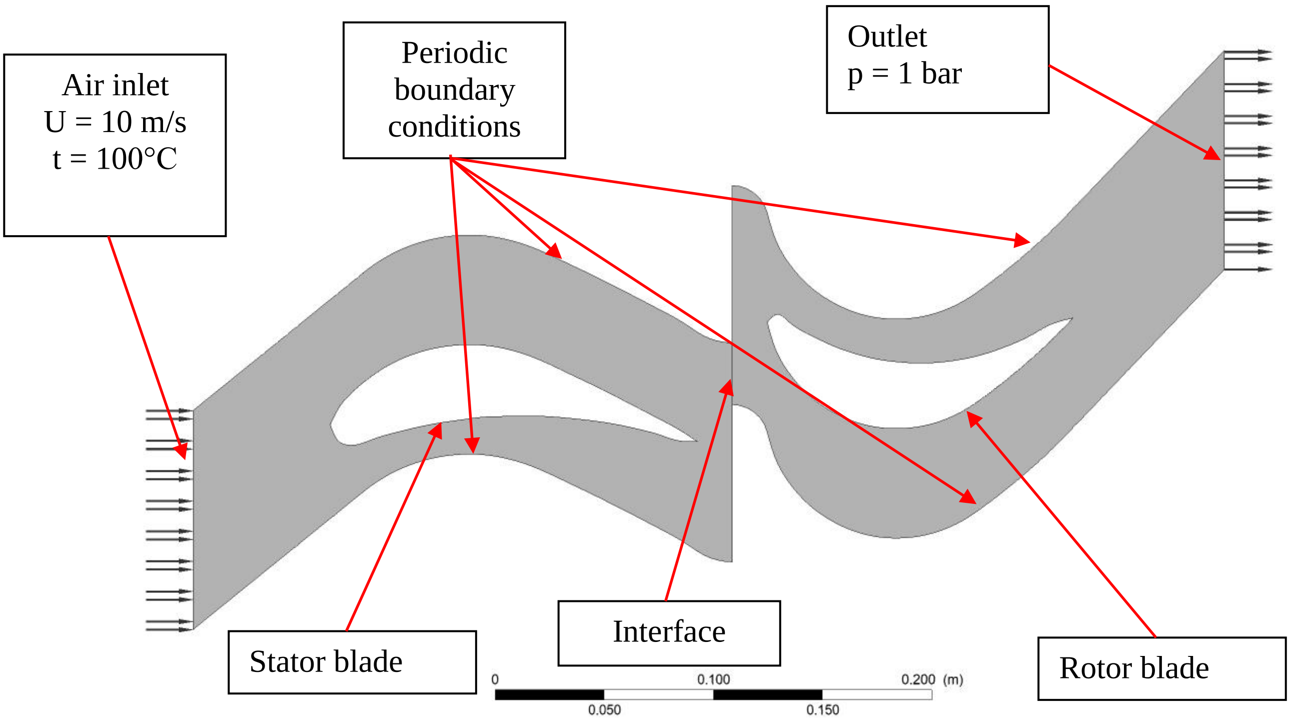


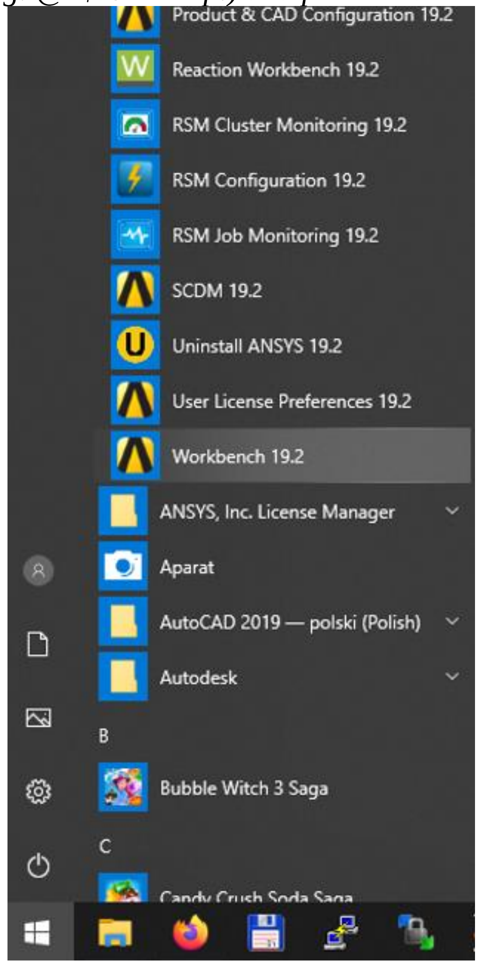
Fig. 1. Diagram of the issue of flow through a two-dimensional turbine stage

2. FLOW THROUGH A TURBINE STAGE

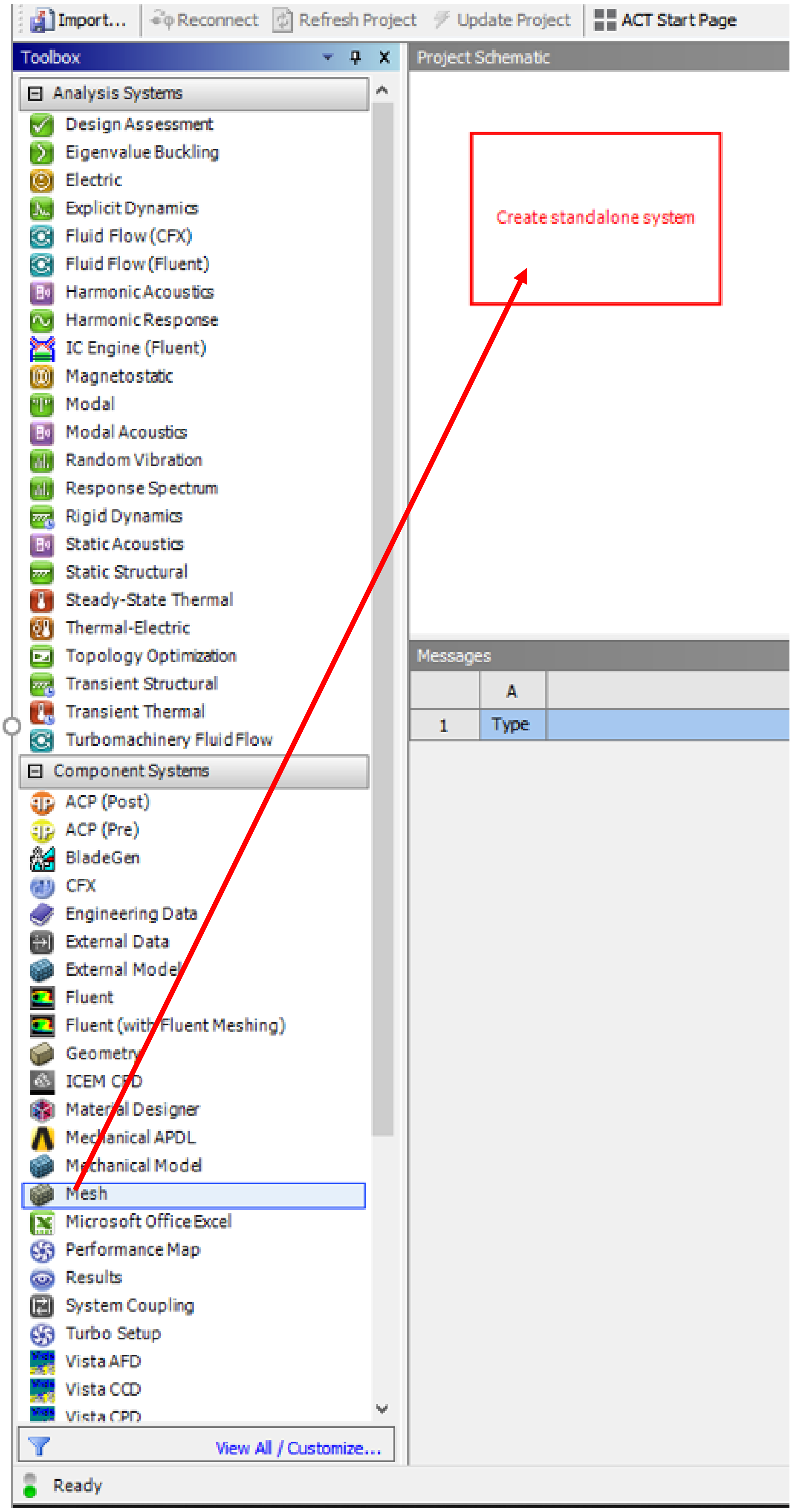
2.1. GEOMETRY

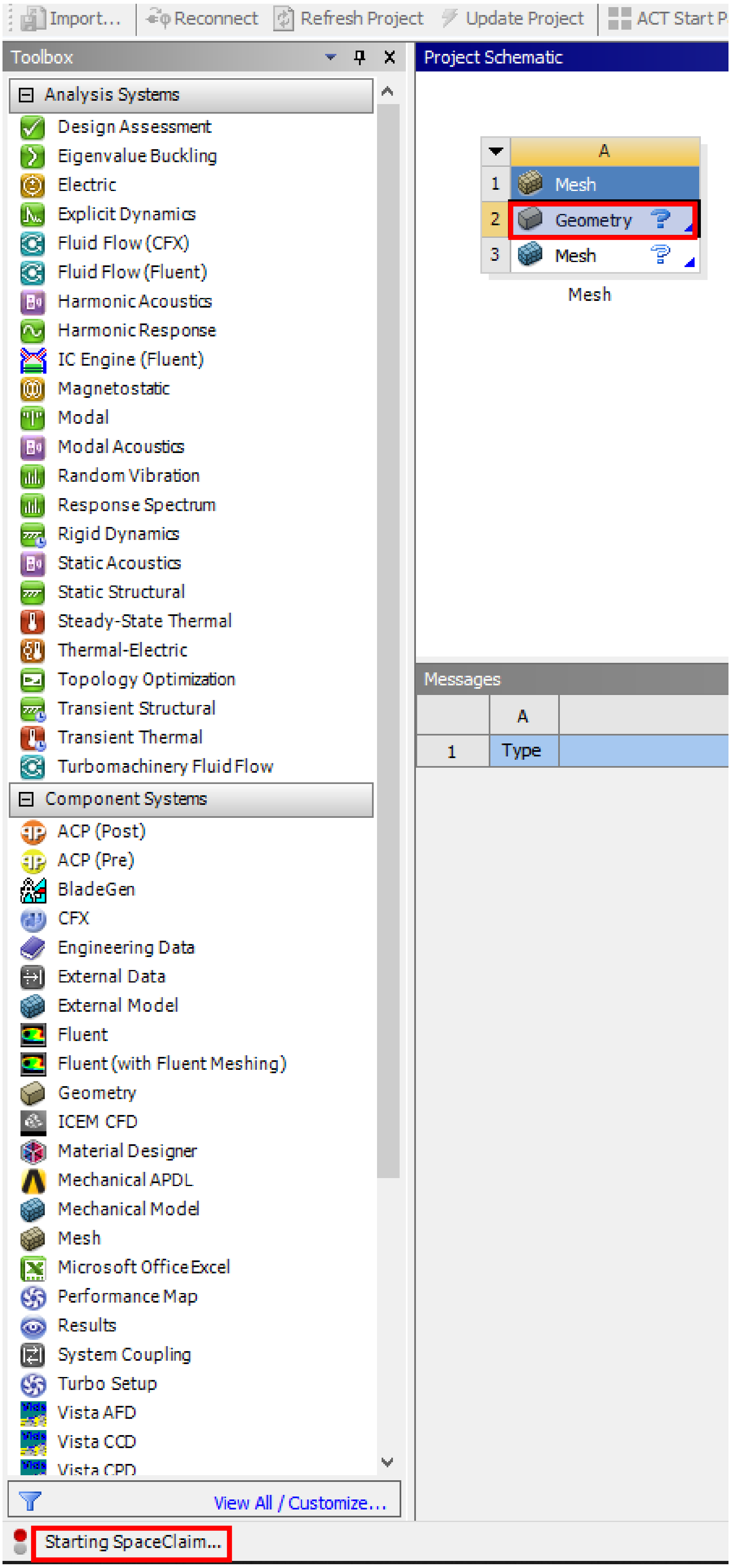
Do the following:

1) Open Ansys Workbench and save project as Ex6 in catalog Ex6 (*File->Save As*).
   RULE OF THUMB NO. 1: *Create a separate catalog for each project*
   RULE OF THUMB NO. 2: *In the names of directories do not use: spaces, special characters (e.g. @#$%^&* itp.) and polish marks*

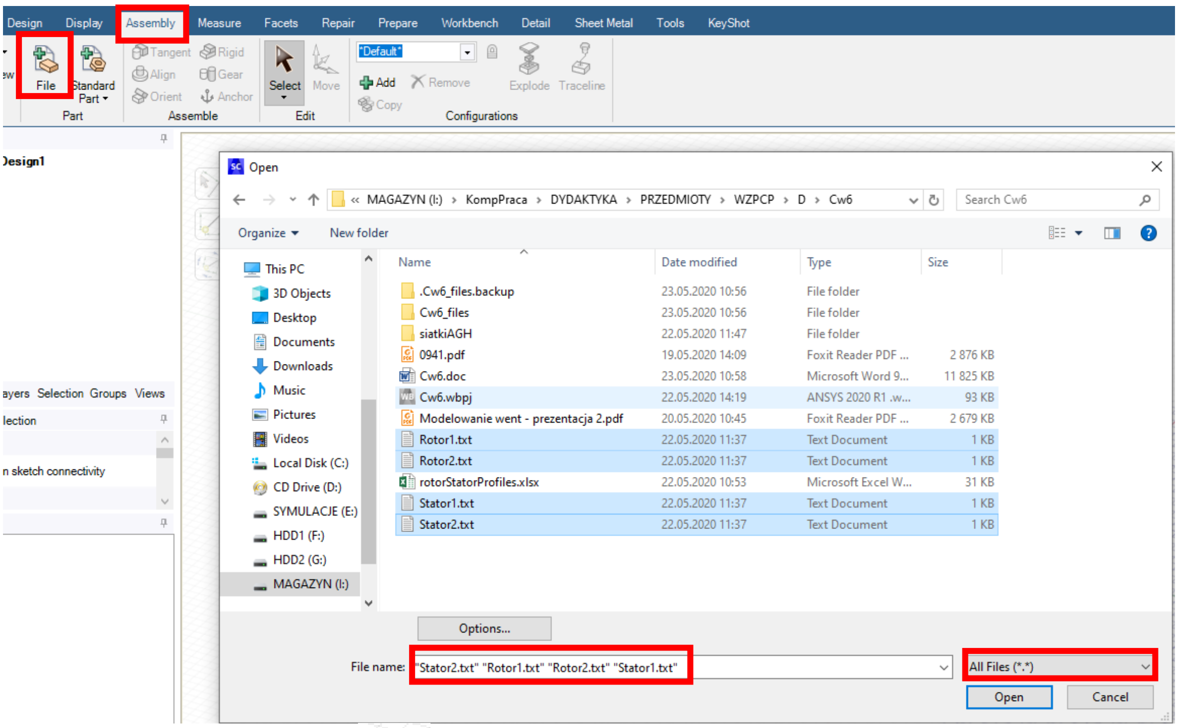


2) Select the *Mesh* module and open *Spaceclaim*. To do this, hold the left mouse button (LMP) on the *Mesh* module and drag it to the *Project Schematic* field. Then double-click LMP on *Geometry* to start the *Spaceclaim* program in which the geometry will be created. Note that in the lower left corner of the screen there is an inscription informing what program is running.

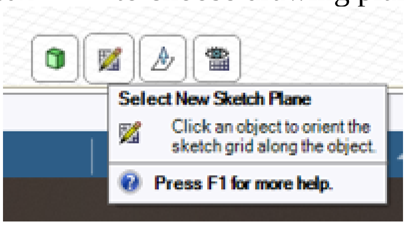

Import...
Reconnect
Refresh Project
Update Project
ACT Start Page
Toolbox
Project Schematic
Analysis Systems
Design Assessment
Eigenvalue Buckling
Electric
Explicit Dynamics
Fluid Flow (CFX)
Fluid Flow (Fluent)
Harmonic Acoustics
Harmonic Response
IC Engine (Fluent)
Magnetostatic
Modal
Modal Acoustics
Random Vibration
Response Spectrum
Rigid Dynamics
Static Acoustics
Static Structural
Steady-State Thermal
Thermal-Electric
Topology Optimization
Transient Structural
Transient Thermal
Turbomachinery Fluid Flow
Component Systems
ACP (Post)
ACP (Pre)
BladeGen
CFX
Engineering Data
External Data
External Model
Fluent
Fluent (with Fluent Meshing)
Geometry
ICEM CFD
Material Designer
Mechanical APDL
Mechanical Model
Mesh
Microsoft Office Excel
Performance Map
Results
System Coupling
Turbo Setup
Vista AFD
Vista CCD
Vista CPD
View All / Customize...
Create standalone system
Messages
A
1
Type
Ready


Import...
Reconnect
Refresh Project
Update Project
ACT Start P
Toolbox
Project Schematic
Analysis Systems
Design Assessment
Eigenvalue Buckling
Electric
Explicit Dynamics
Fluid Flow (CFX)
Fluid Flow (Fluent)
Harmonic Acoustics
Harmonic Response
IC Engine (Fluent)
Magnetostatic
Modal
Modal Acoustics
Random Vibration
Response Spectrum
Rigid Dynamics
Static Acoustics
Static Structural
Steady-State Thermal
Thermal-Electric
Topology Optimization
Transient Structural
Transient Thermal
Turbomachinery Fluid Flow
Component Systems
ACP (Post)
ACP (Pre)
BladeGen
CFX
Engineering Data
External Data
External Model
Fluent
Fluent (with Fluent Meshing)
Geometry
ICEM CFD
Material Designer
Mechanical APDL
Mechanical Model
Mesh
Microsoft Office Excel
Performance Map
Results
System Coupling
Turbo Setup
Vista AFD
Vista CCD
Vista CPD
View All / Customize...
Starting SpaceClaim...
A
1 Mesh
2 Geometry
3 Mesh
Mesh
Messages
A
1 Type

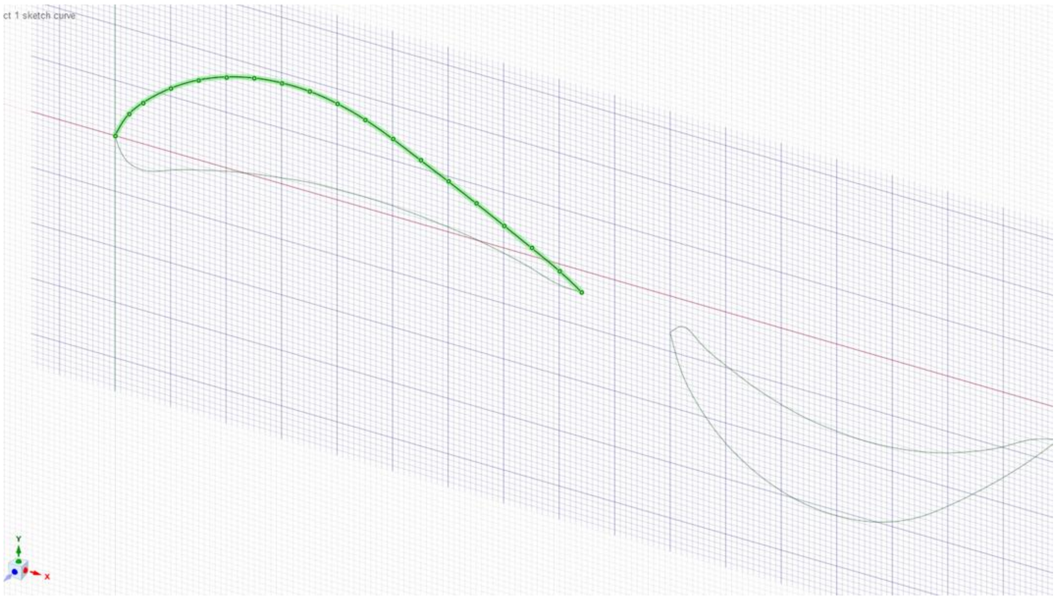
3) In *Assembly* tab select *File* icon, and choose files *Stator1.txt*, *Stator2.txt*, *Rotor1.txt* i *Rotor2.txt.*

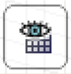


4) LMP on *Select New Sketch*  to choose drawing plane.

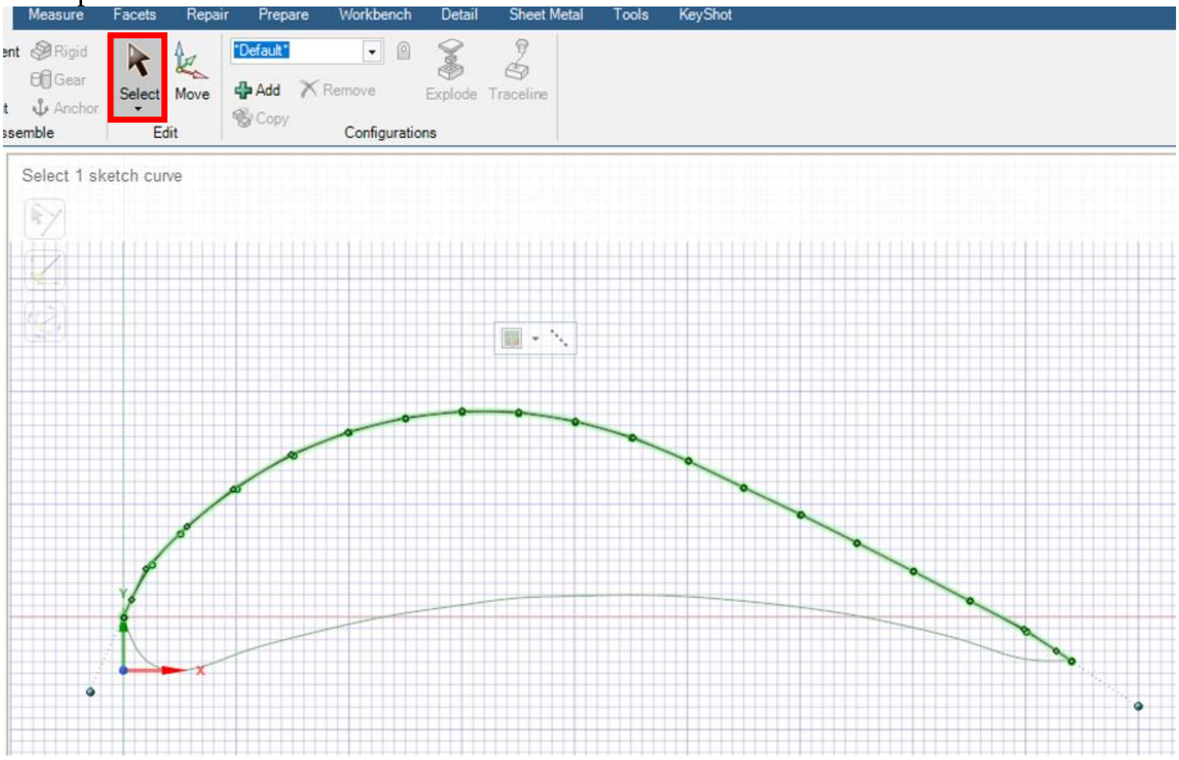


Select the X-Y plane by clicking the LMP of the stator or rotor profile as shown below.

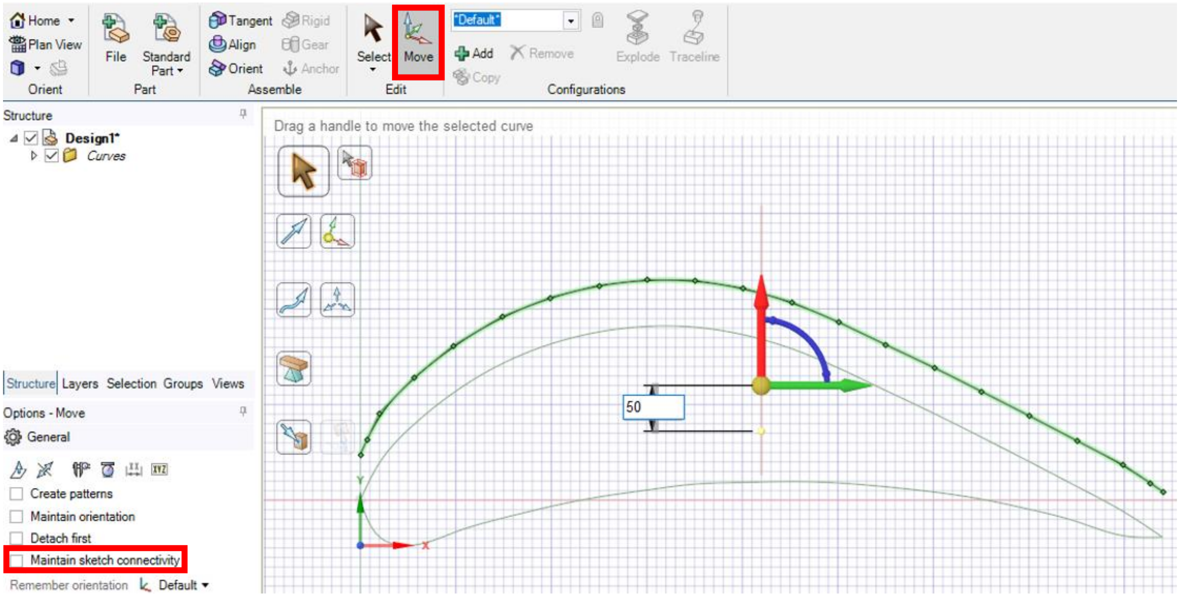


5) Click *Plan View*  to rotate drawing plane parallel to the screen (you can do it via *Shift + v* as well).

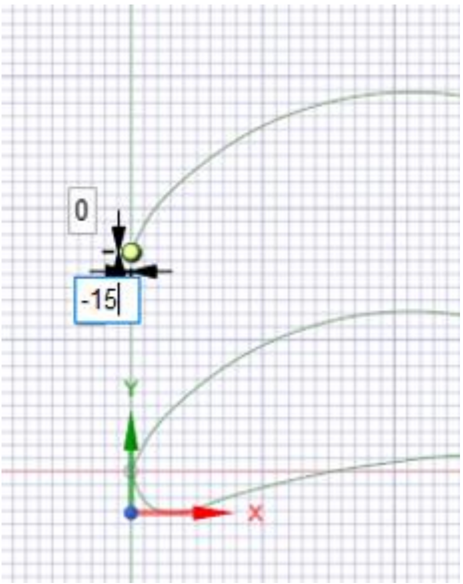
6) Choose *Select* and then LMP indicate the upper stator profile. Then *Ctrl + c* and *Ctrl + v* to copy and paste an identical profile. There are now two profiles in the same place.



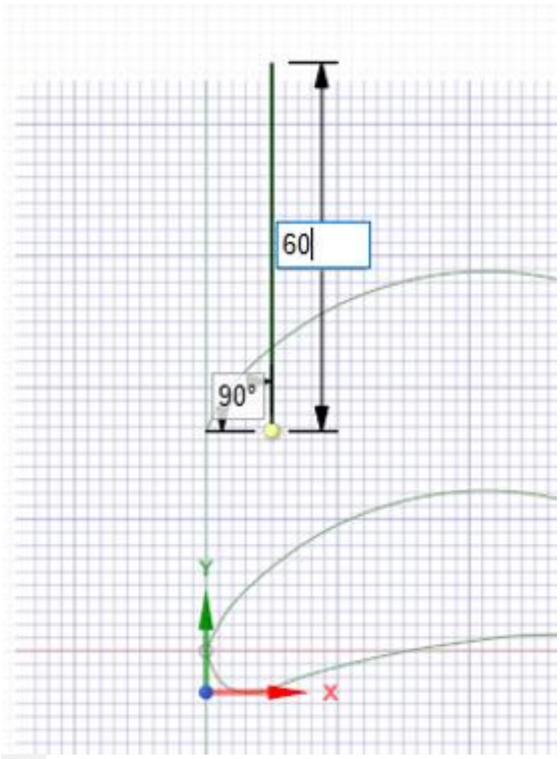
7) Select *Move*, LMP indicate the upper stator profile, uncheck the option on the left *Maintain sketch connectivity* and move the profile up 50 mm.



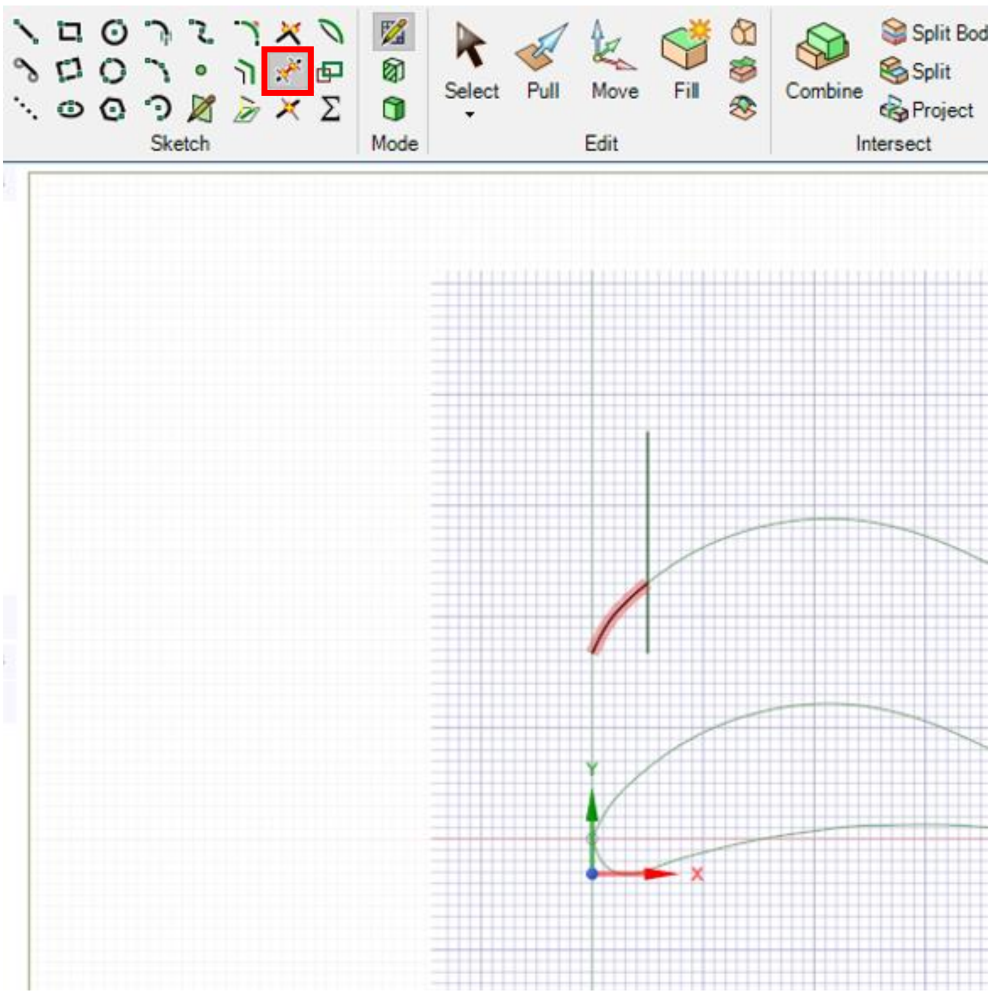
8) Move the cursor closer to the end of the shifted curve and press the *Shift* key, then use -15 mm for the horizontal dimension and 0 mm for the vertical dimension.



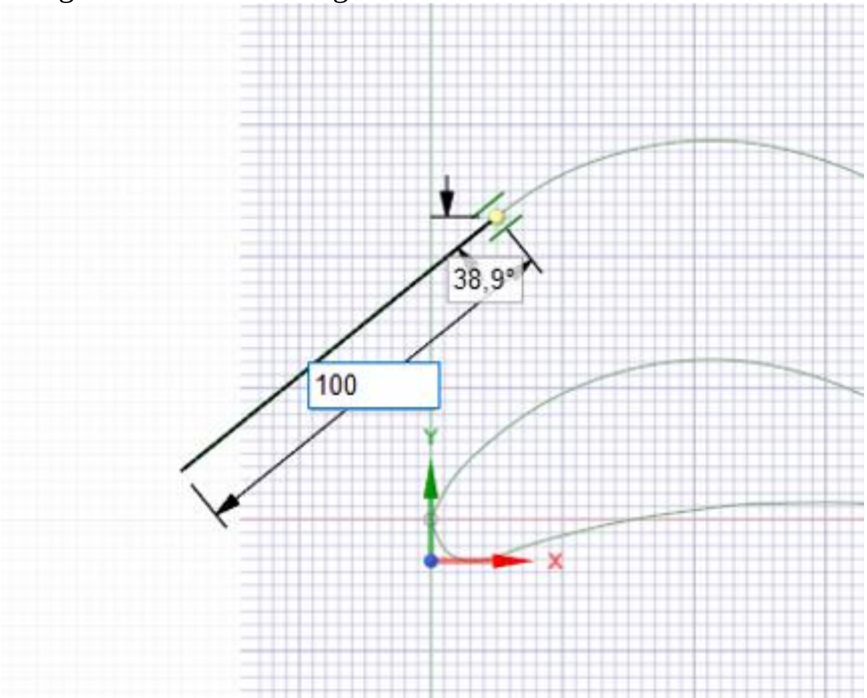
9) Draw a vertical line 60 mm long.



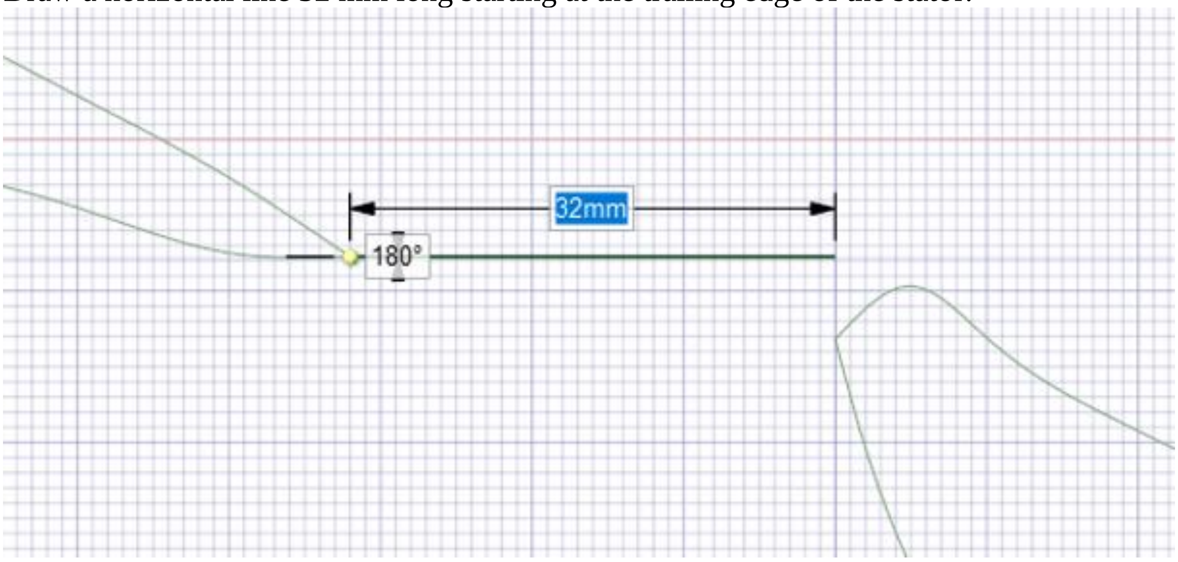
10) Select *Trim Away*  and remove the edge indicated in red and the unnecessary vertical line.



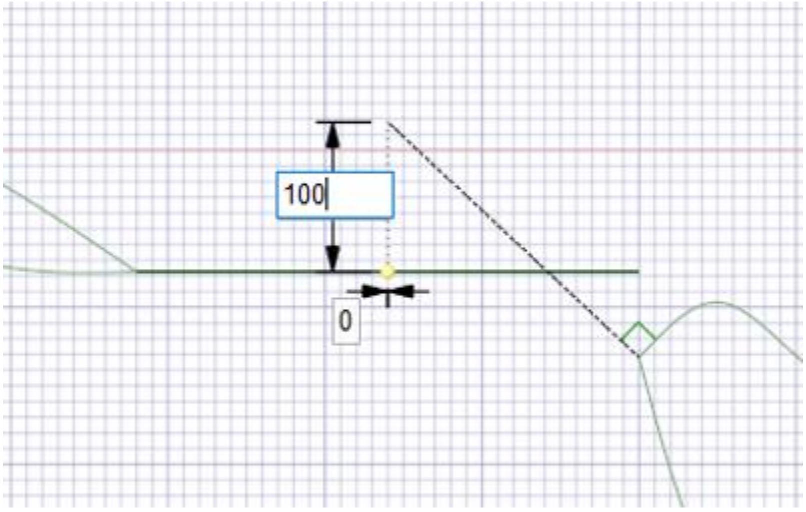
11) Draw a tangent line 100 mm long.

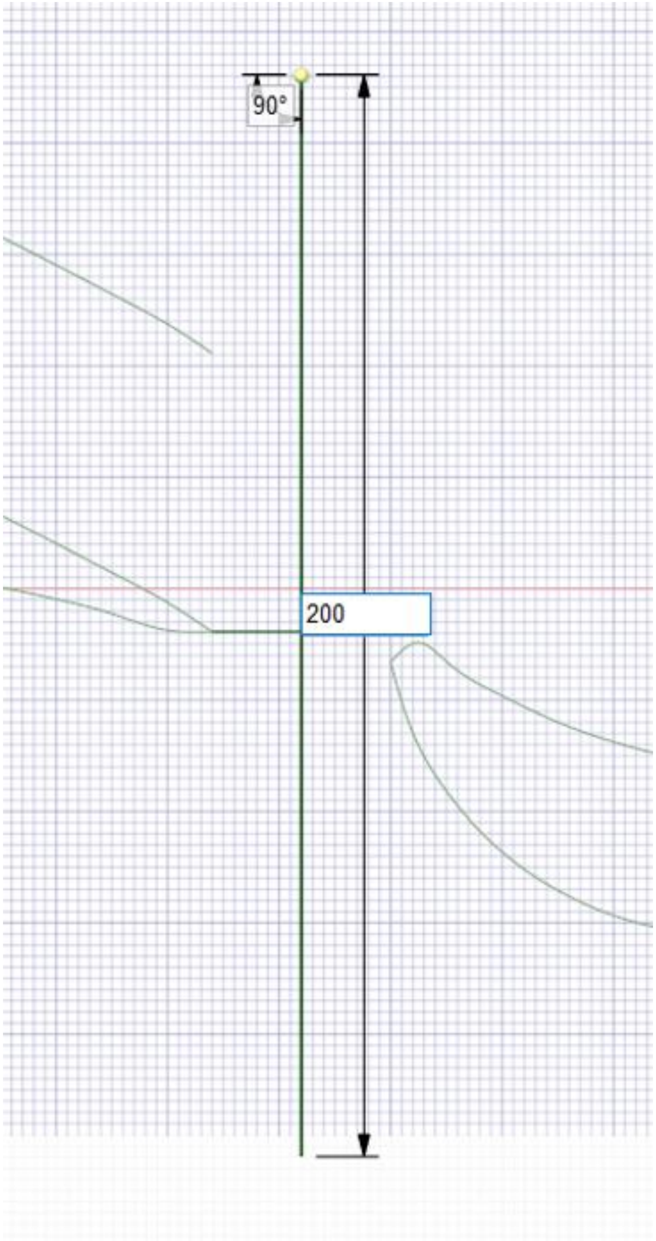


12) Draw a horizontal line 32 mm long starting at the trailing edge of the stator.

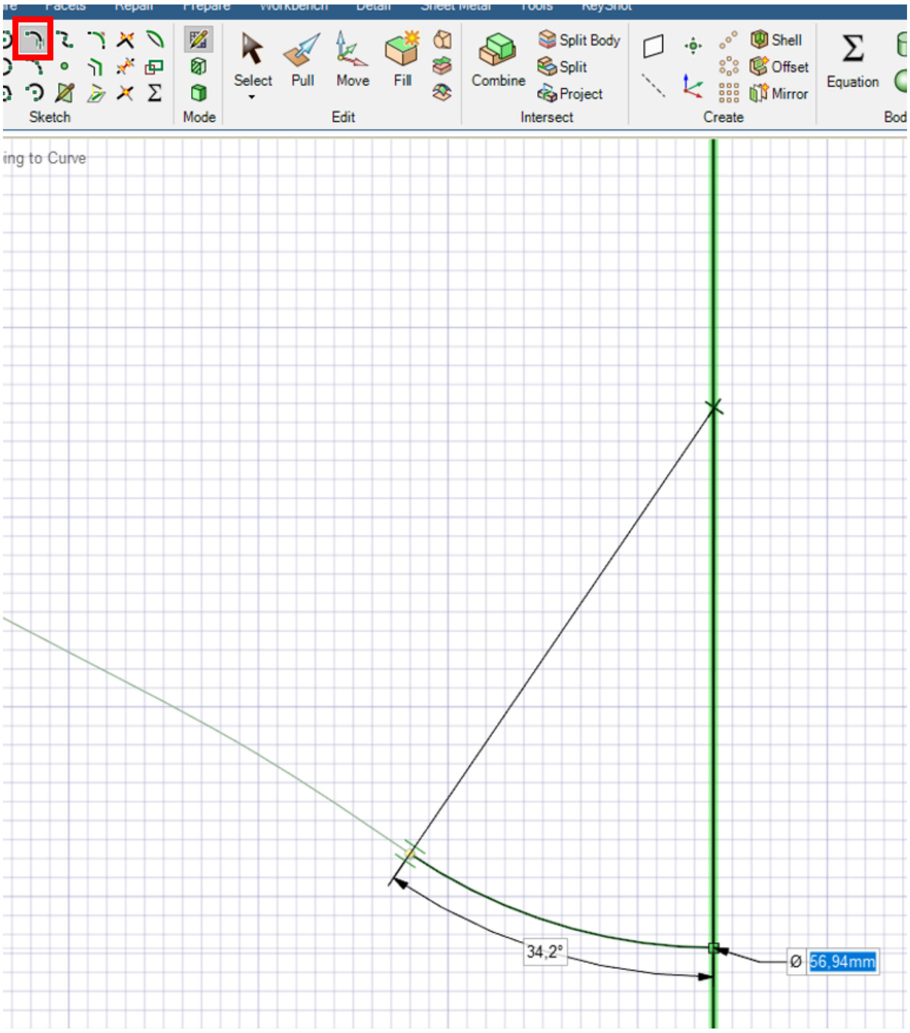


13) Move the cursor to the center of the drawn line (a green triangle will appear) and press *Shift*. For horizontal dimension, select 0 mm and for vertical 100 mm. Then draw a vertical line 200 mm long.

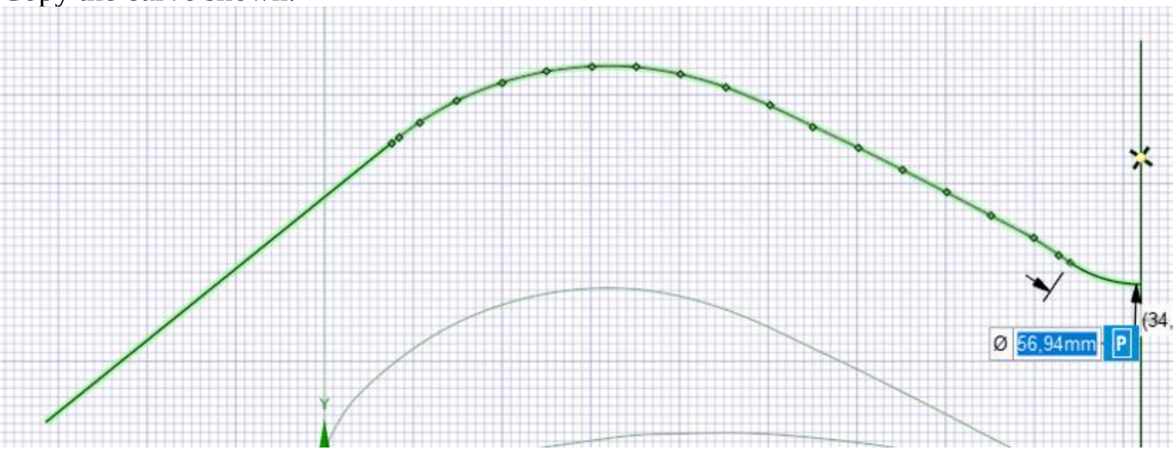




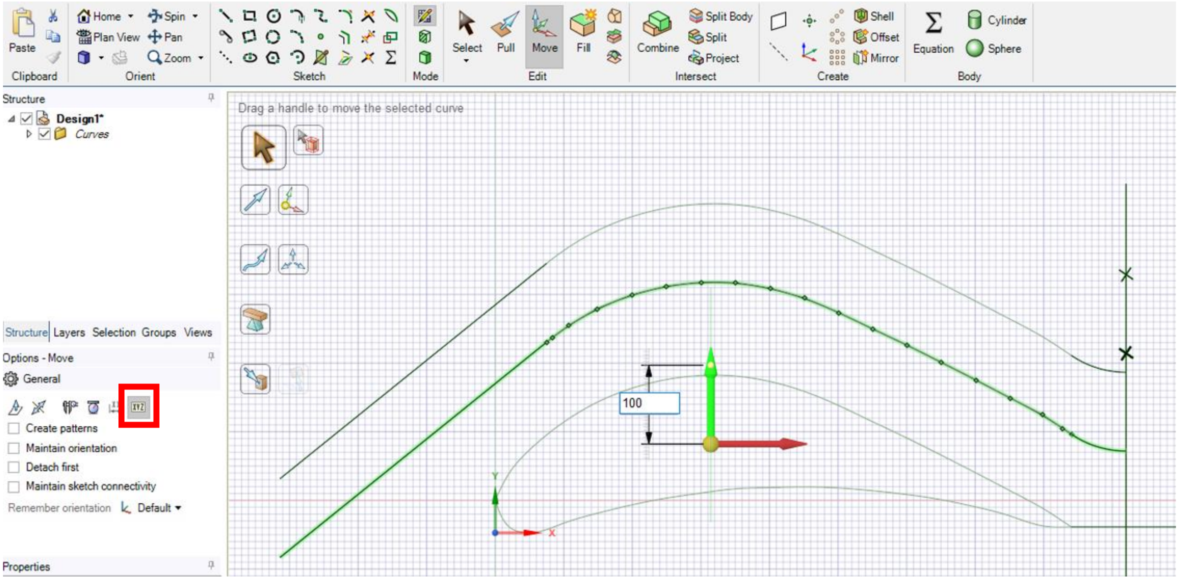
14) Draw a tangent arc to the copied stator line by selecting the command *Tangent Arc* and LMP end of the line.



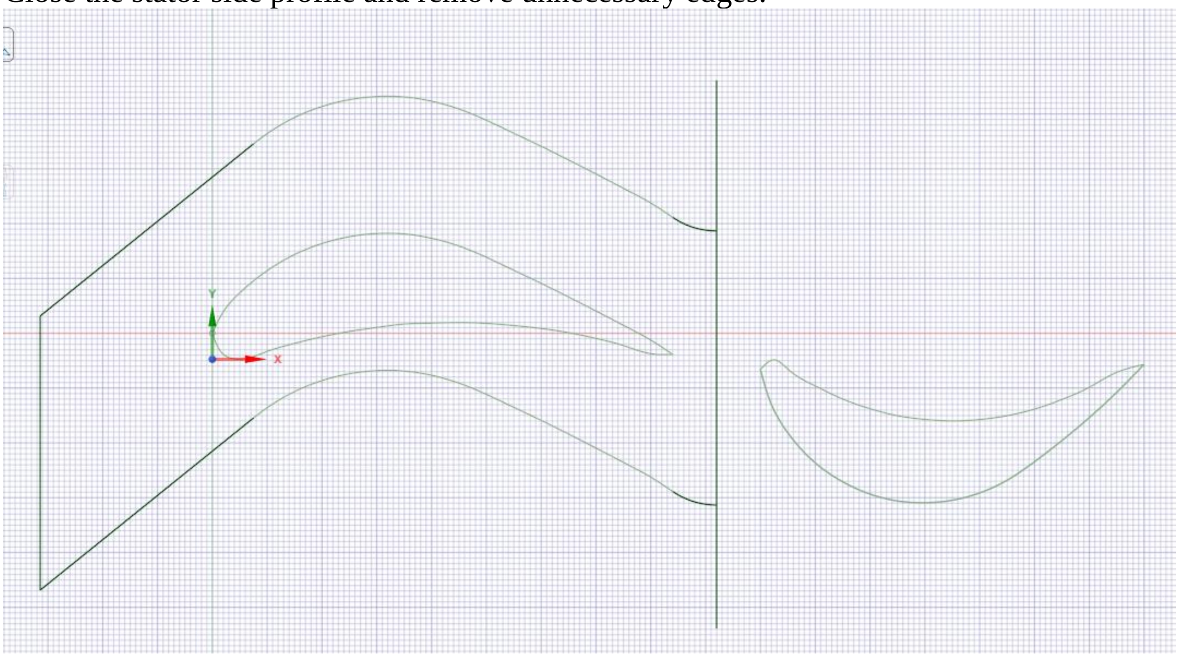
15) Copy the curve shown.



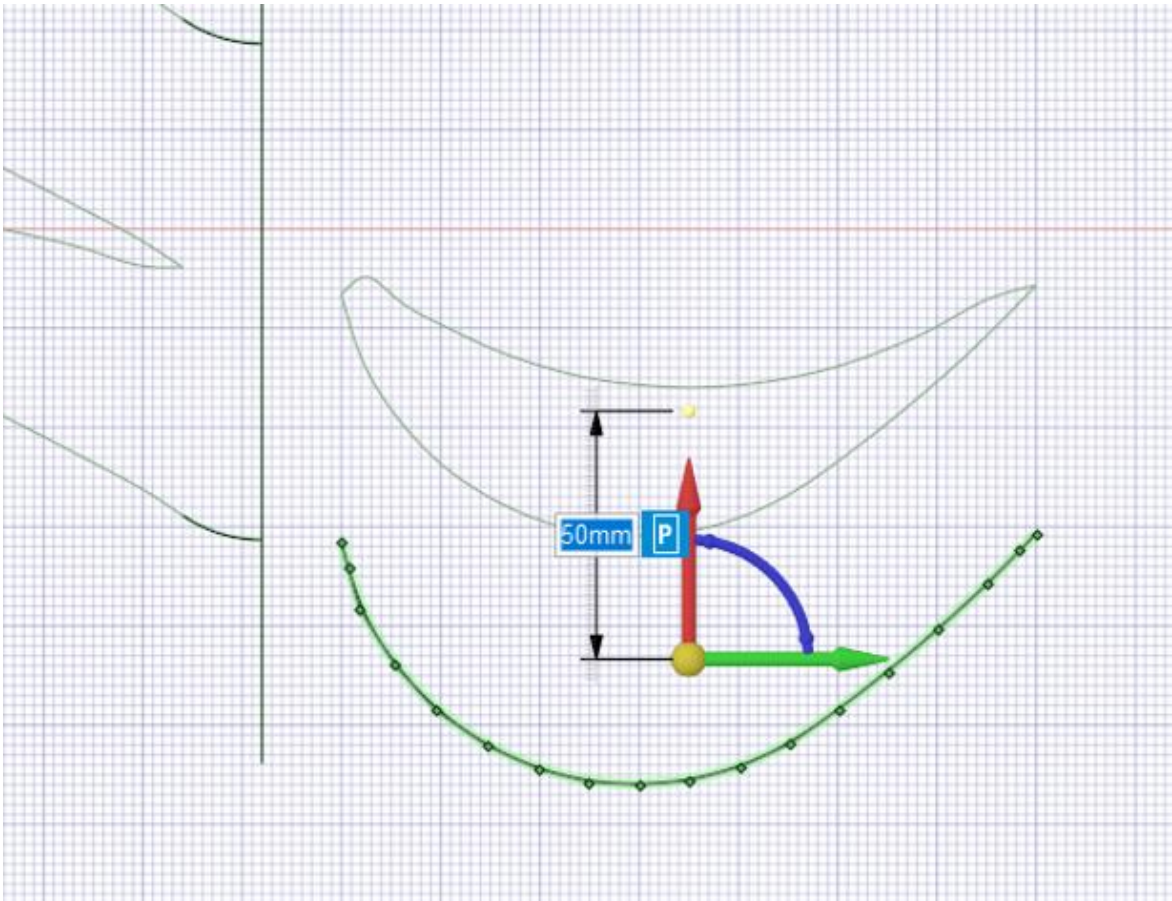
16) Then move each curve downwards by 100 mm. To do this, select the *Move* command and select *Cartesian Coordinates* on the left.



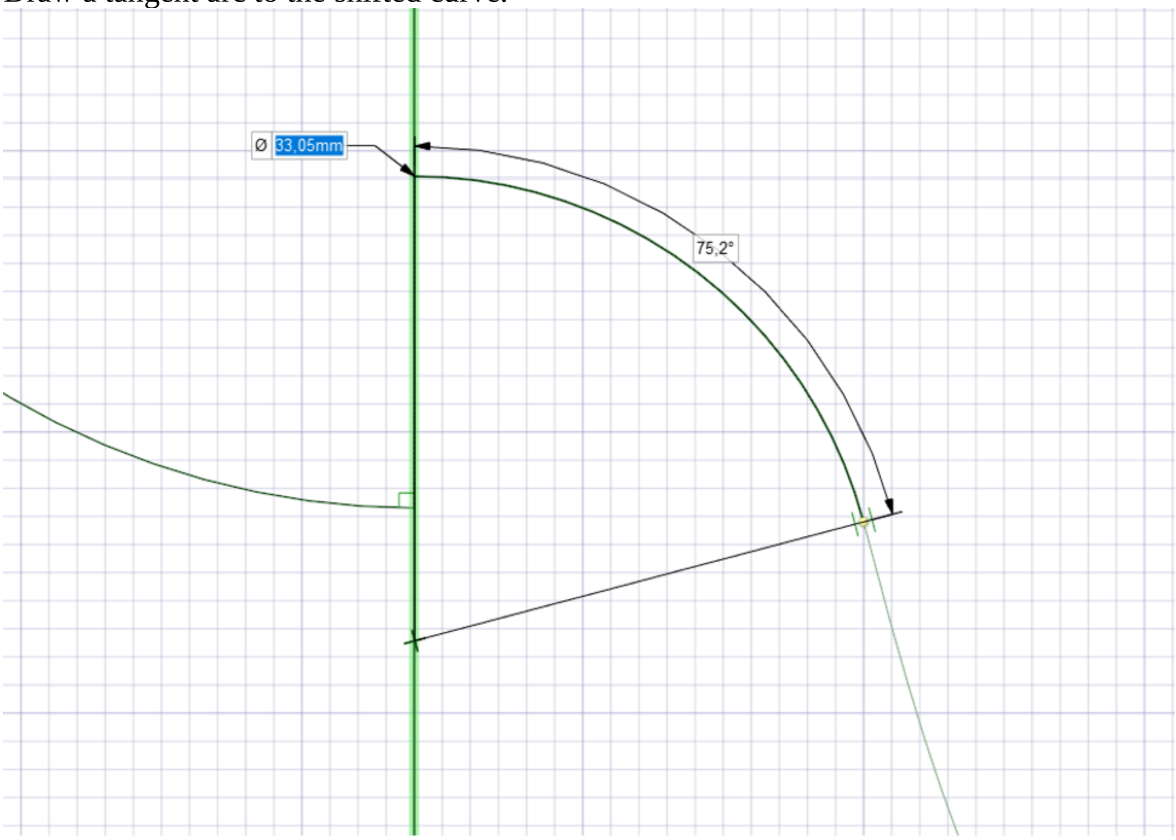
17) Close the stator side profile and remove unnecessary edges.



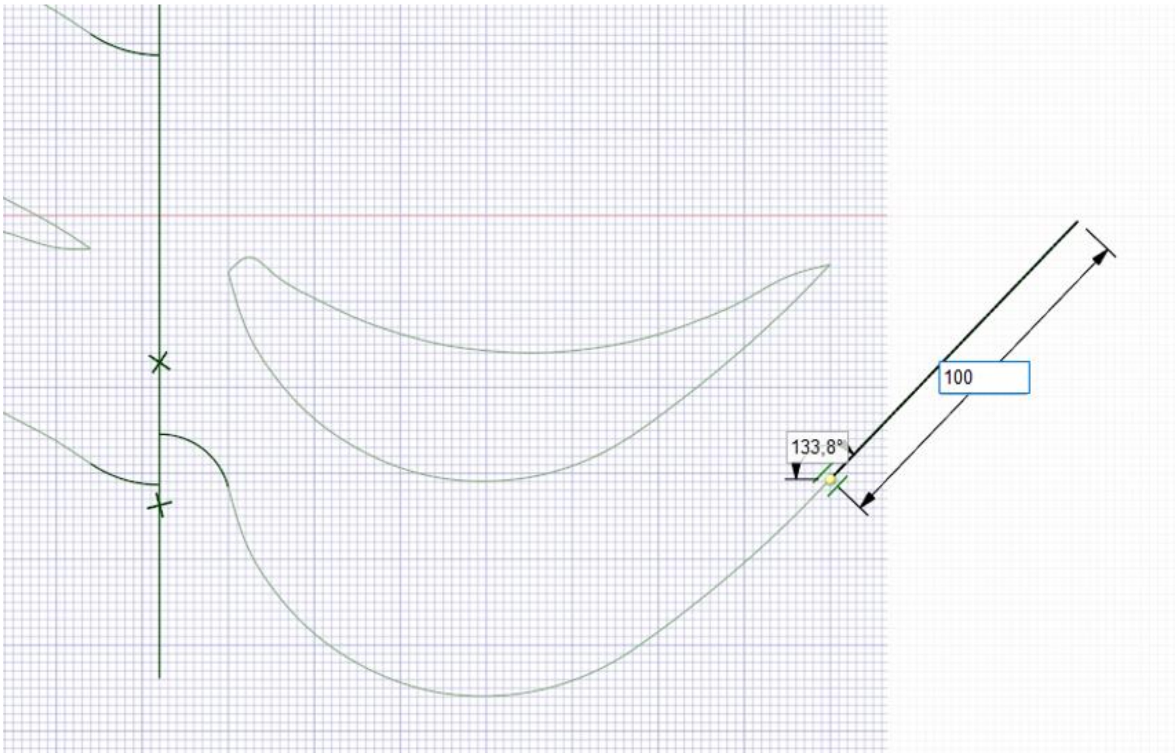
18) Copy and move the bottom rotor curve by 50 mm.



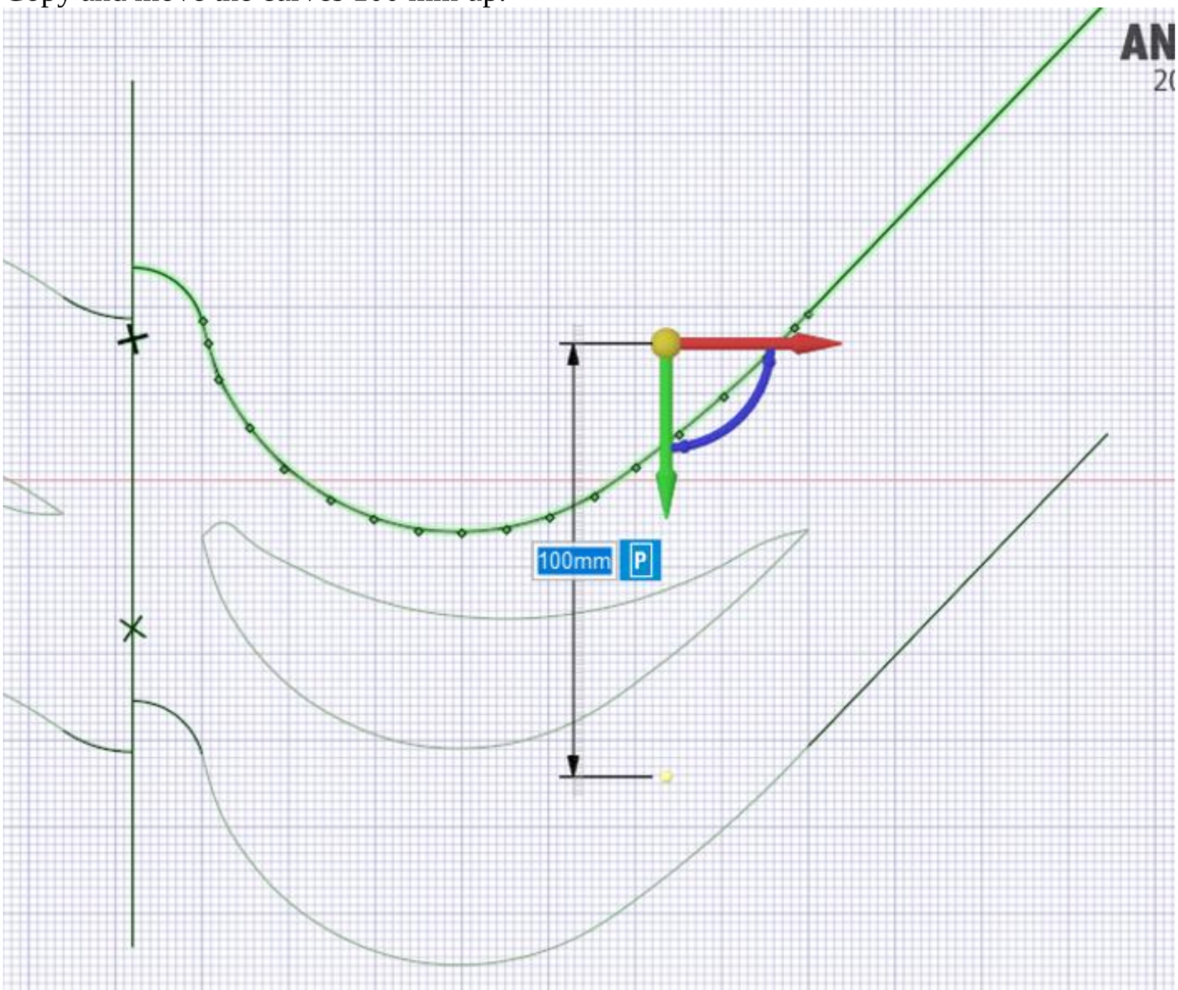
19) Draw a tangent arc to the shifted curve.



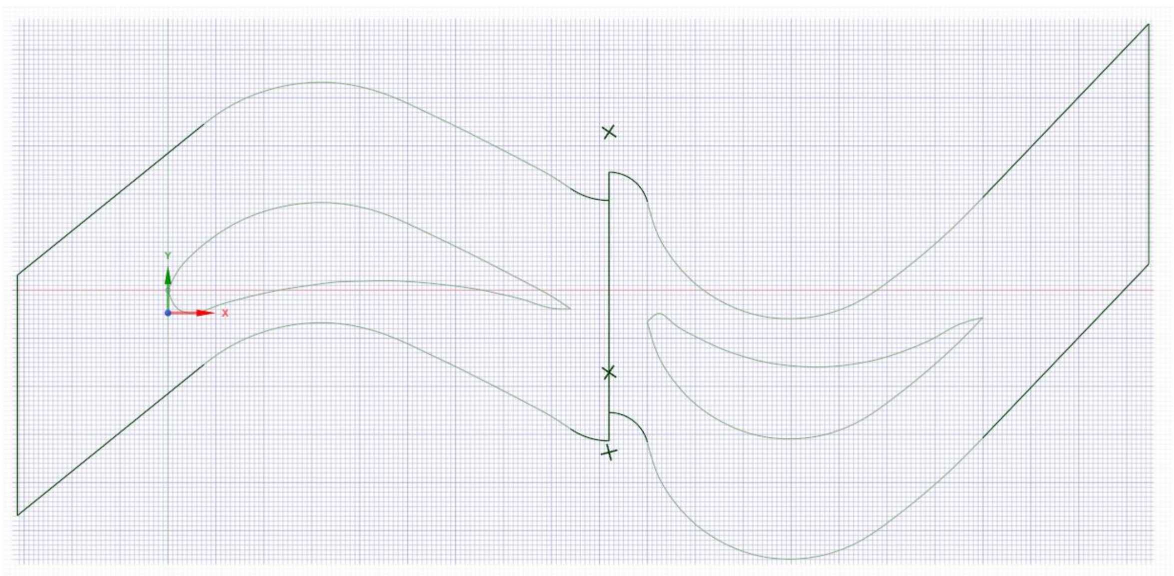
20) Draw a tangent line 100 mm long.



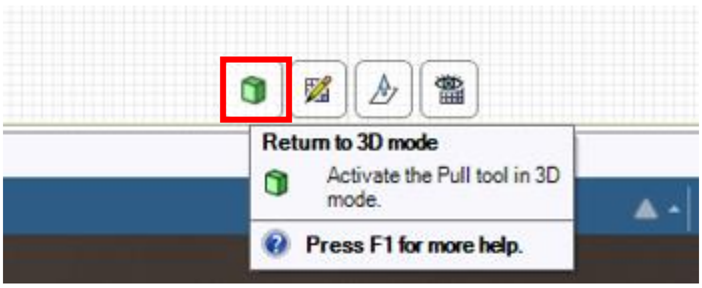
21) Copy and move the curves 100 mm up.



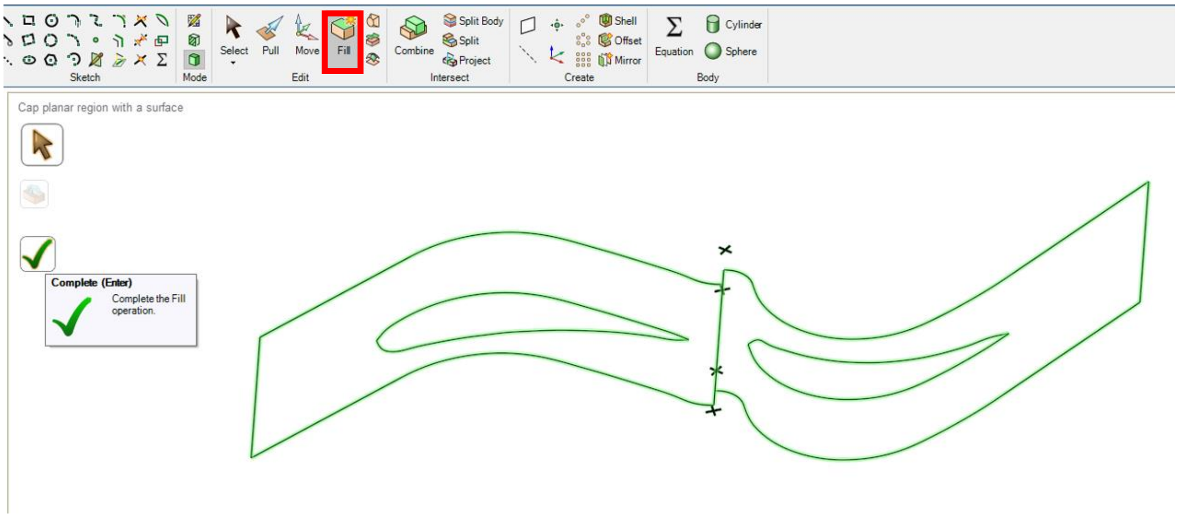
22) Close the profile and remove unnecessary edges. If it turns out that you need to remove the entire vertical line, you need to zoom in near the junction of the arch with the vertical line and draw the missing piece of the arch.



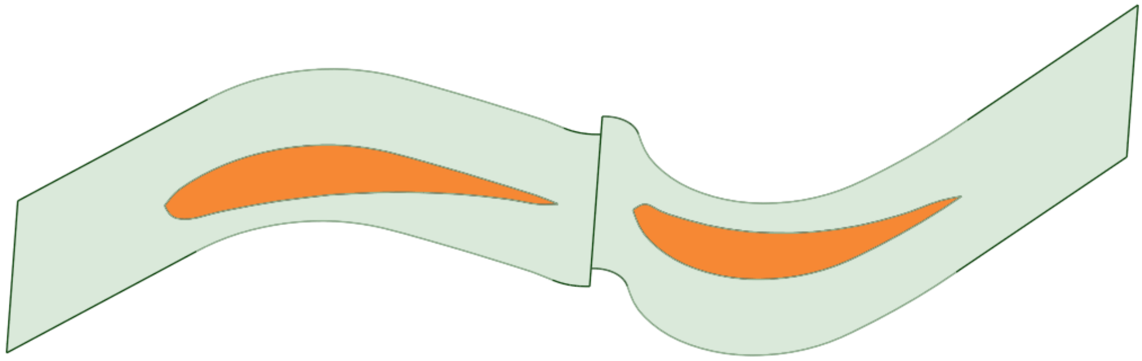
23) To exit the line drawing command, press Esc and LMP, click the *Return to 3D mode* icon at the bottom of the screen.



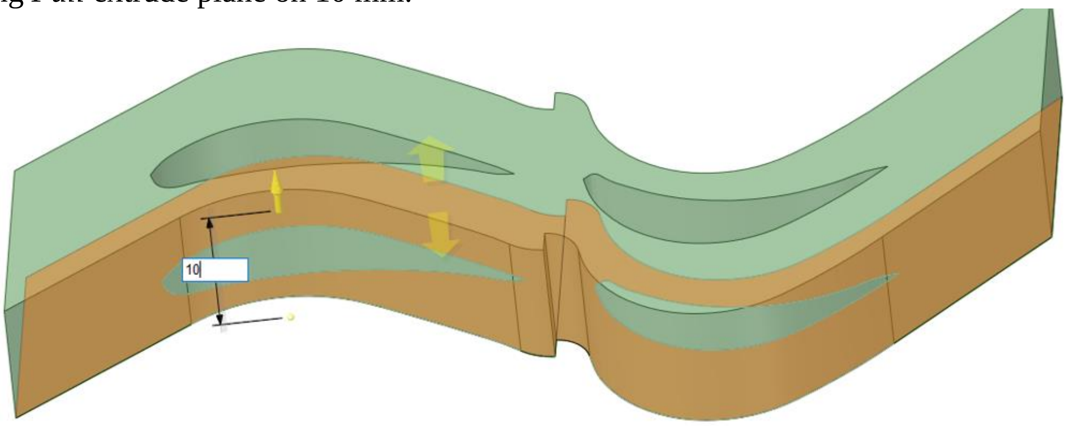
24) Choose *Fill* and then *Ctrl + a* to select all curves. Then confirm *Complete.*



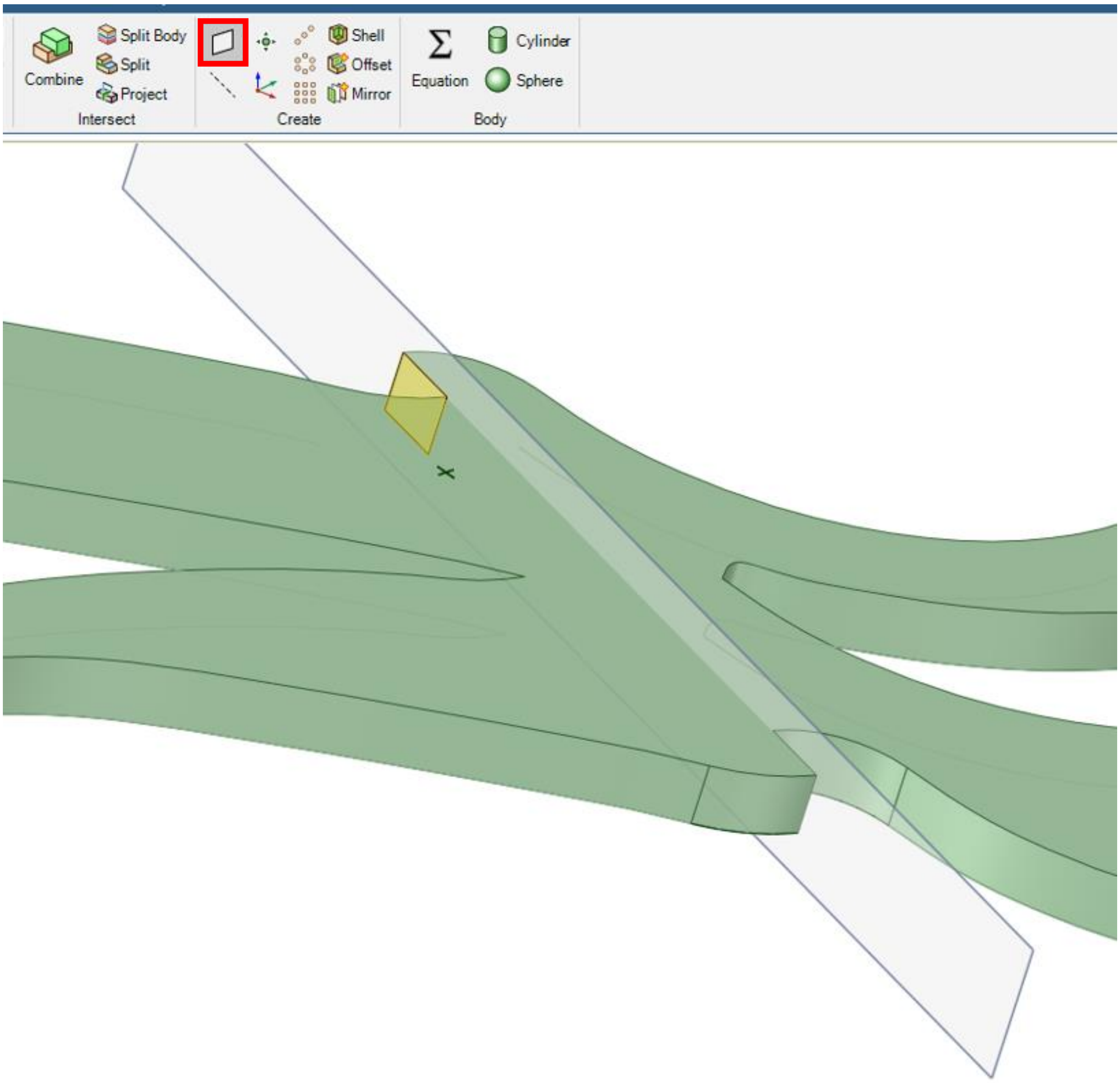
25) Select LMP stator and rotor blades and remove them with the *Delete* key.



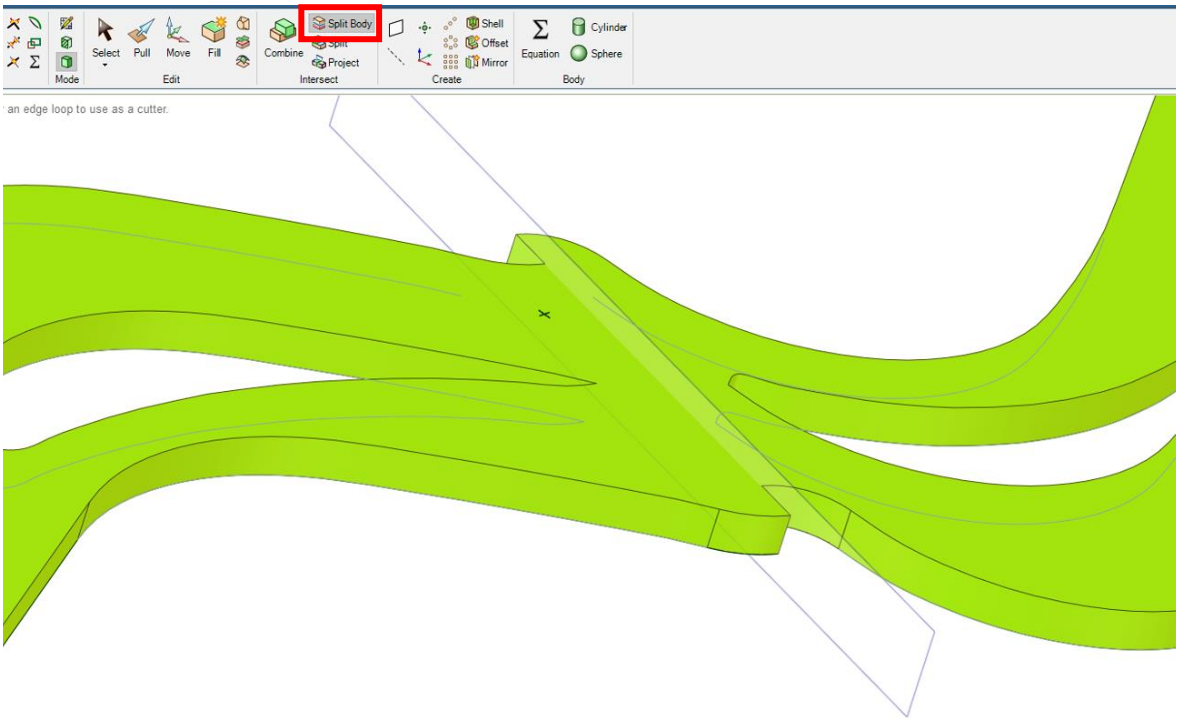
26) Using *Pull* extrude plane on 10 mm.



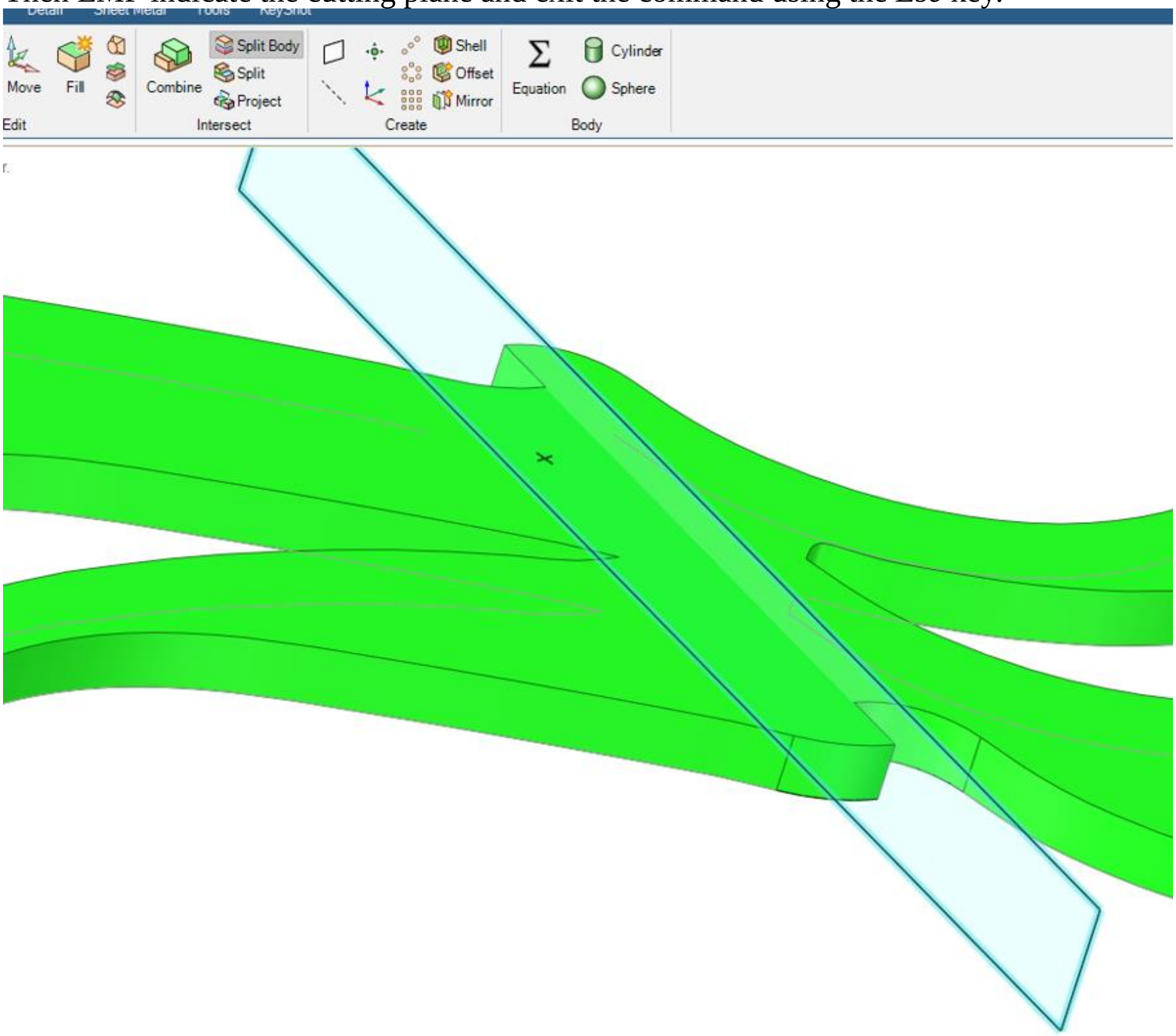
27) Create a plane as shown below.



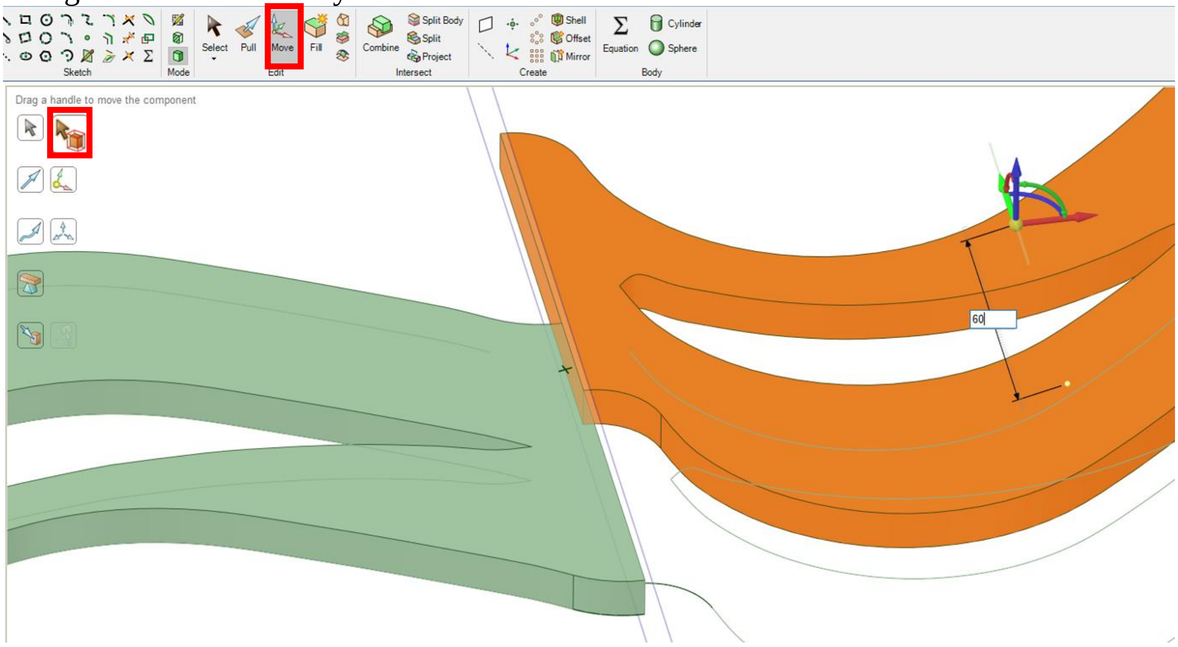
28) Select *Split Body* and select the body.



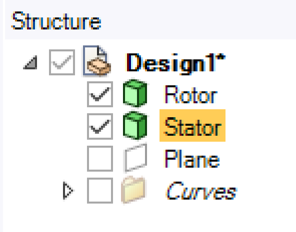
29) Then LMP indicate the cutting plane and exit the command using the *Esc* key.



30) Using *Move* move rotor by 60 mm.



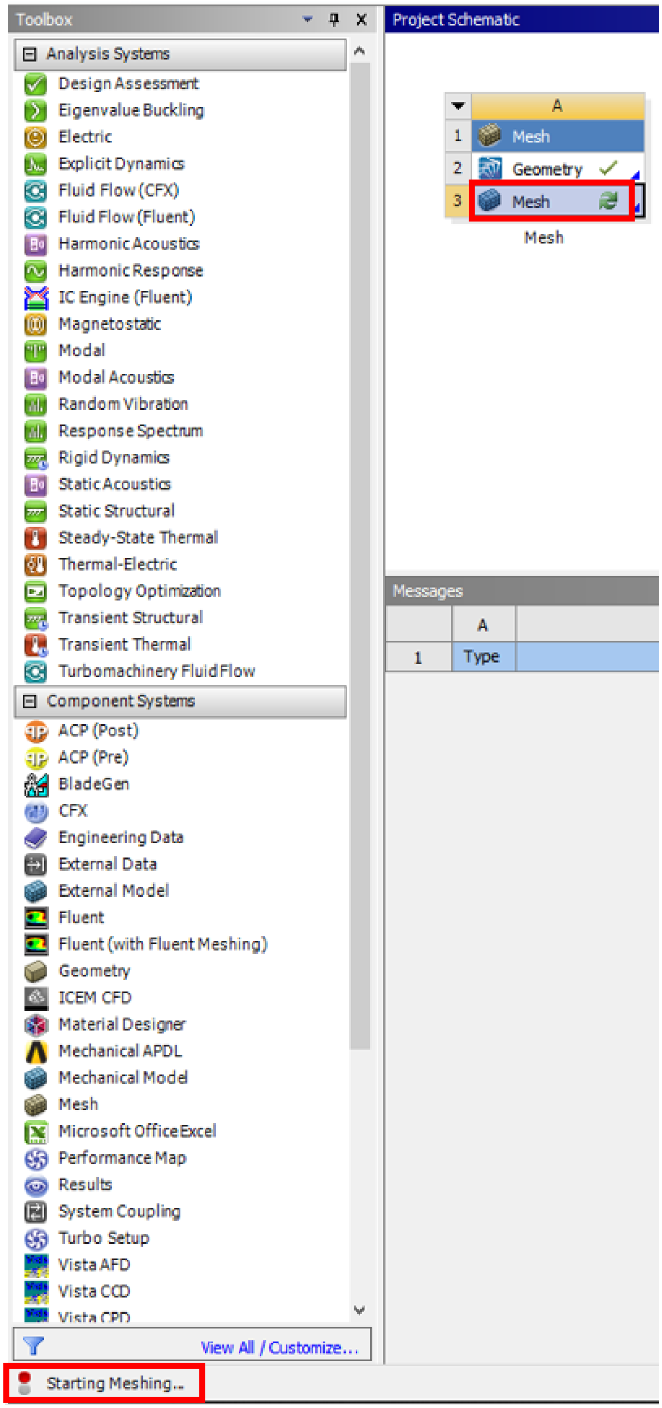
31) Change the block names accordingly as *Stator* and *Rotor*.



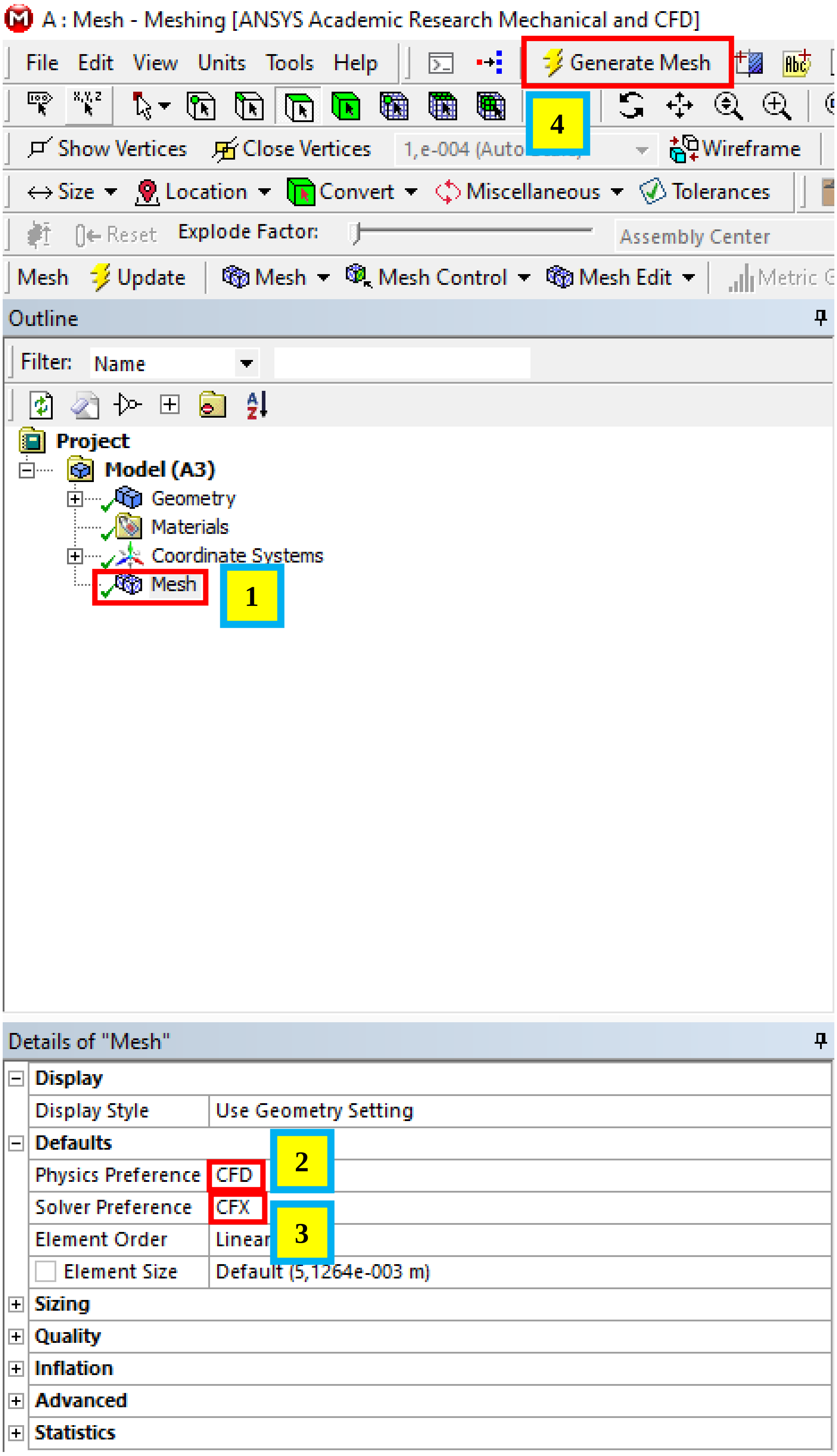
32) Close *Spaceclaim* and save project in *Workbench* with the use of *Ctrl + s*.

2.2. NUMERICAL MESH

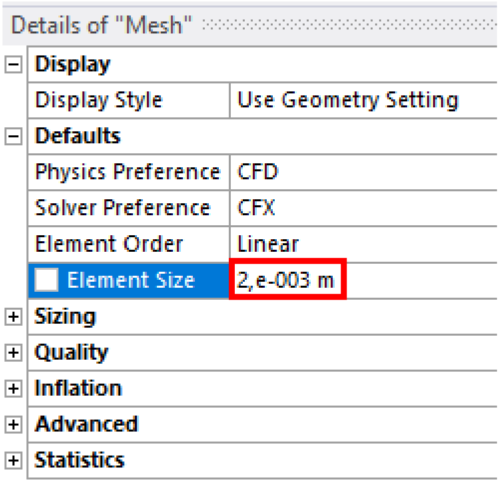
1) Open *Ansys Meshing* by double-click LMP *Mesh*



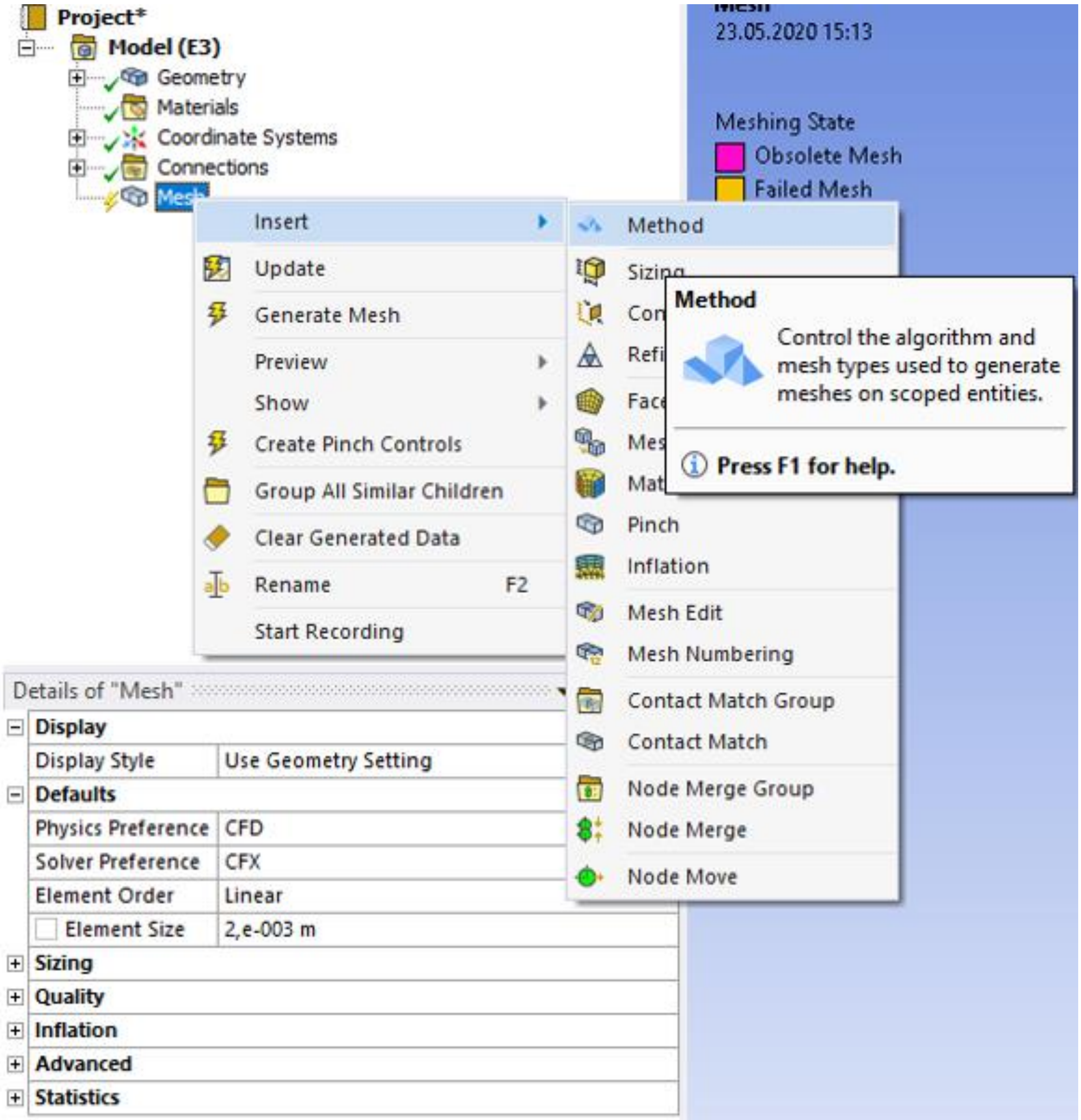
2) In *Ansys Meshing*: 1) click *Mesh*, 2) change *Physisc Preference* into *CFD*, 3) change *Solver Preference* into *CFX*, 4) click LMP *Generate Mesh*



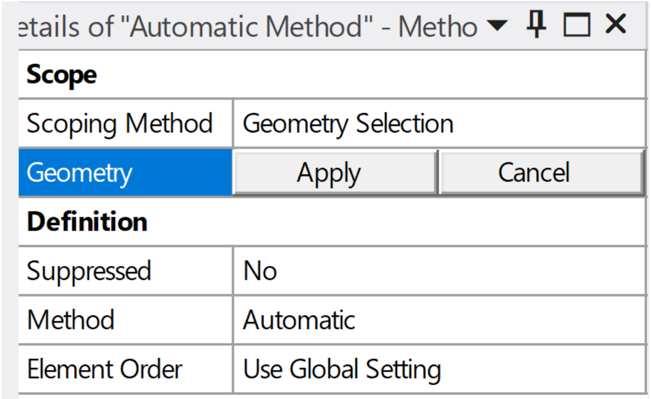
3) In *Details of Mesh* change *Element Size* into 2e-3 m



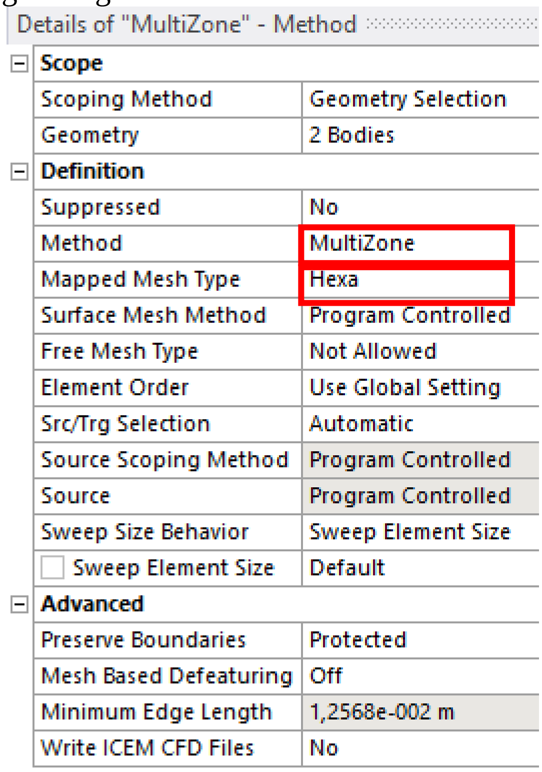
4) Click RMB on *Mesh* and select *Insert->Method*



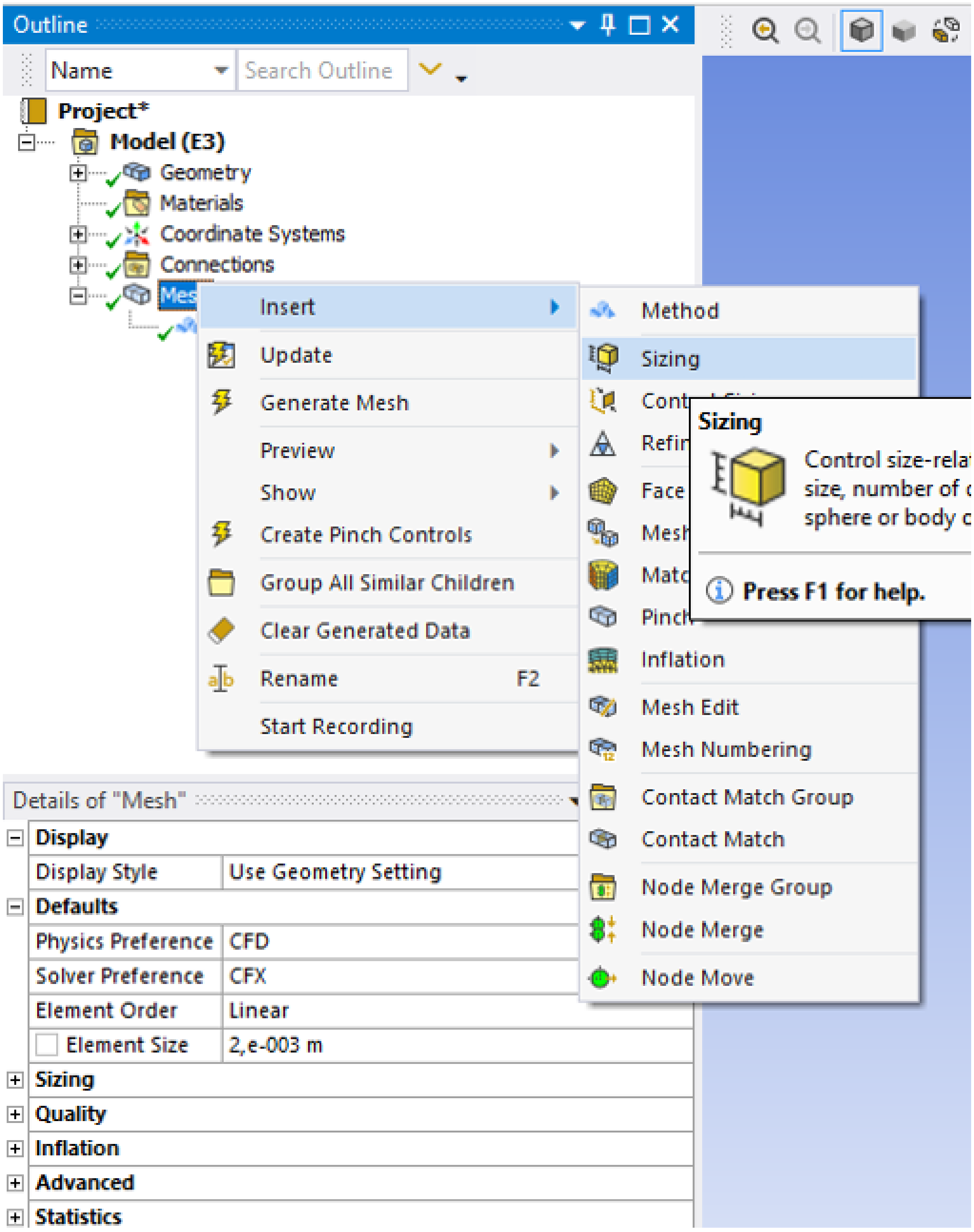
5) LMP indicate both bodies and confirm *Apply* in *Geometry*



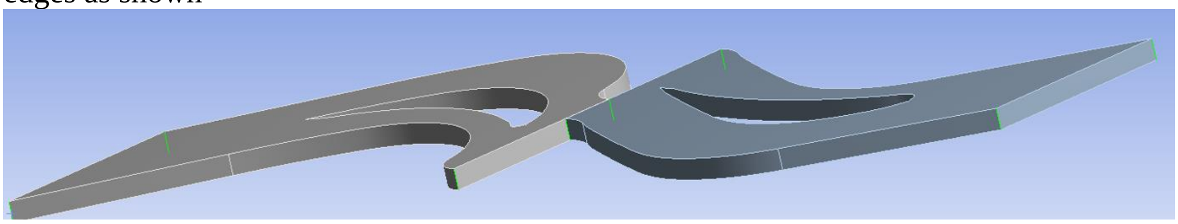
6) Apply the following settings



7) Click RMB on *Mesh* and select *Insert->Sizing*



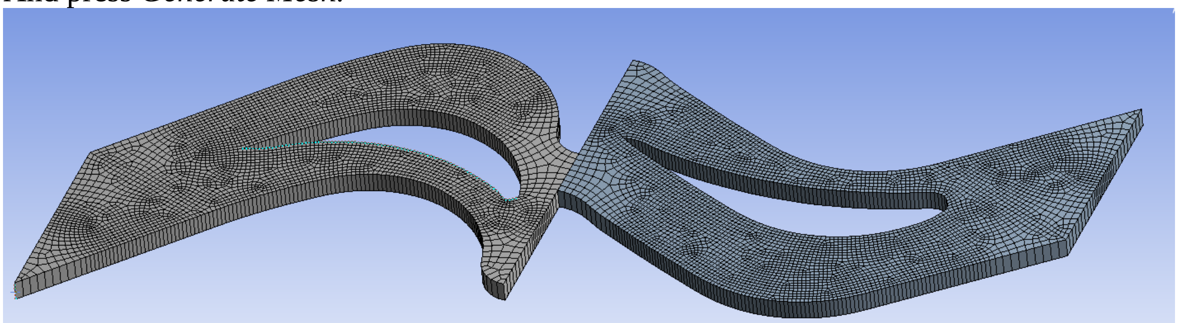
8) Change the selection filter to edges (shortcut *Ctrl* + *E*) and indicate 8 green edges as shown



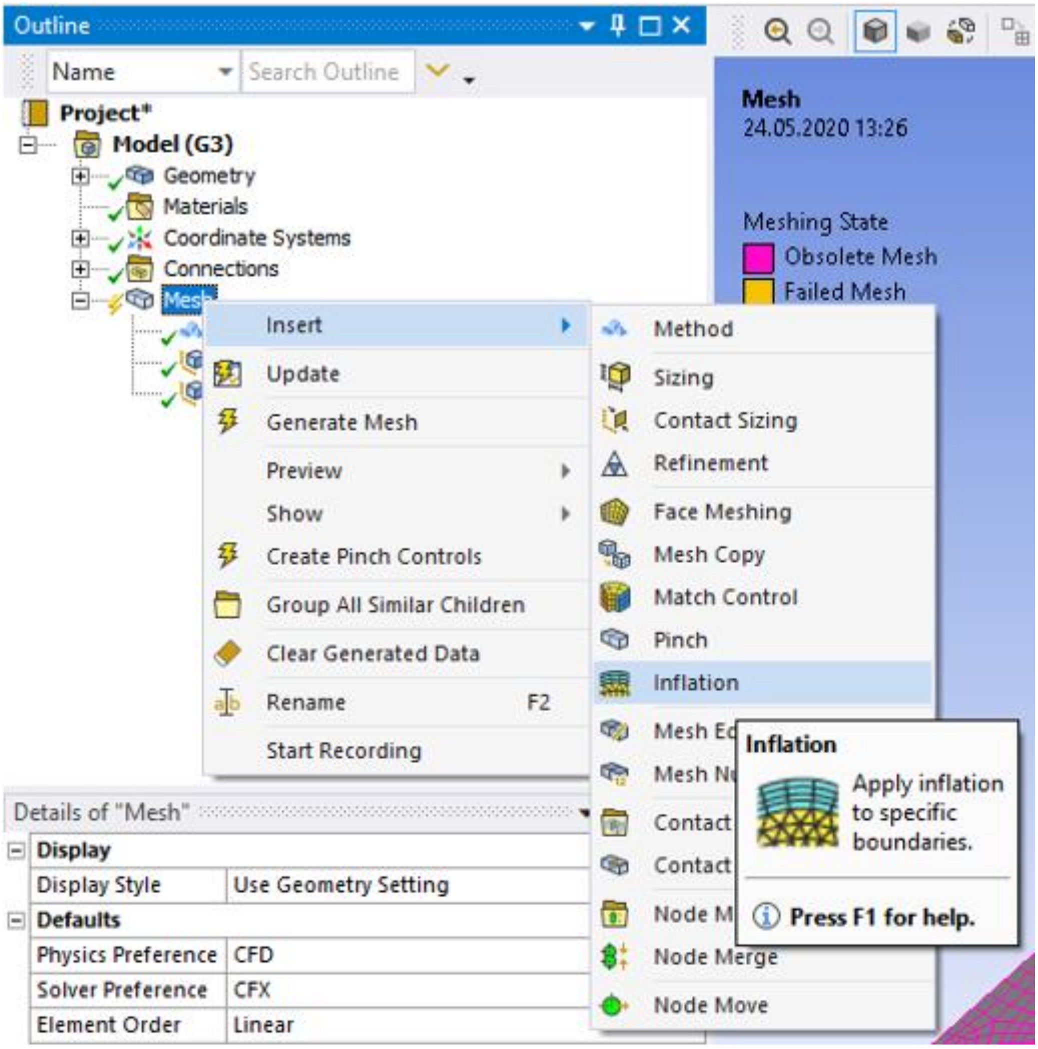
9) Apply the following settings

Details of "Edge Sizing" - Sizing	
Scope	
Scoping Method	Geometry Selection
Geometry	8 Edges
Definition	
Suppressed	No
Type	Number of Divisions
Number of Divisions	1
Advanced	
Behavior	Hard
Capture Curvature	No
Capture Proximity	No
Bias Type	No Bias

And press *Generate Mesh.*

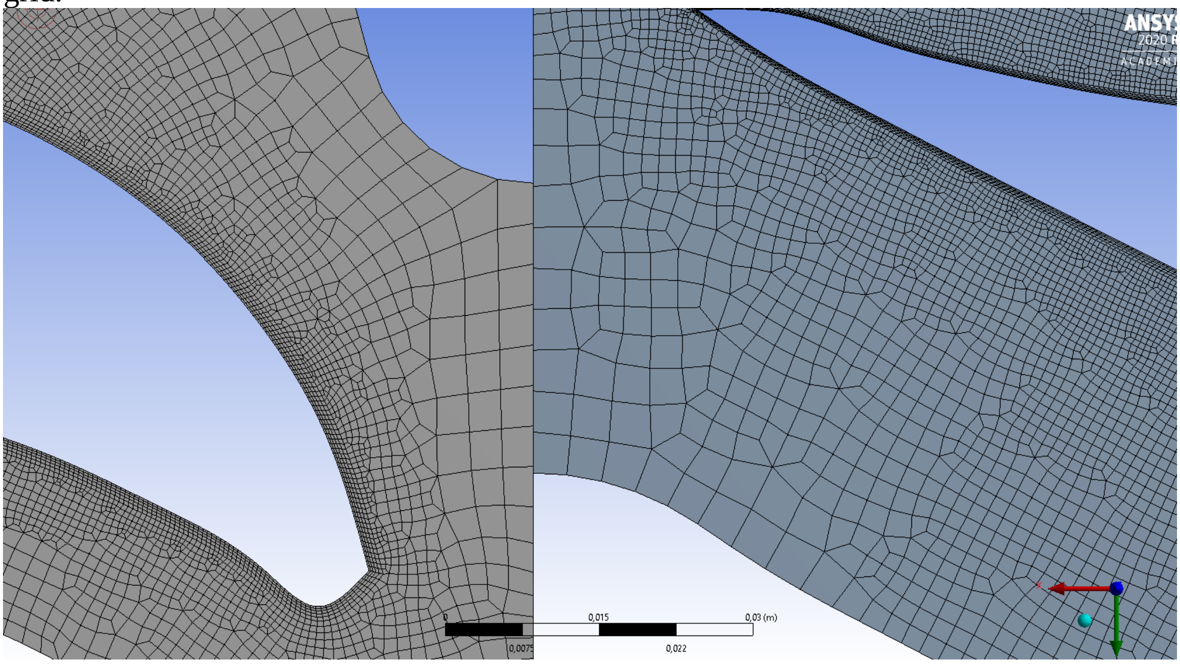


10) Insert *Sizing* (face) on the surface of both blades as 5e-4 m.
11) Insert *Inflation.*



12) Select both bodies and confirm *Geometry->Apply*.

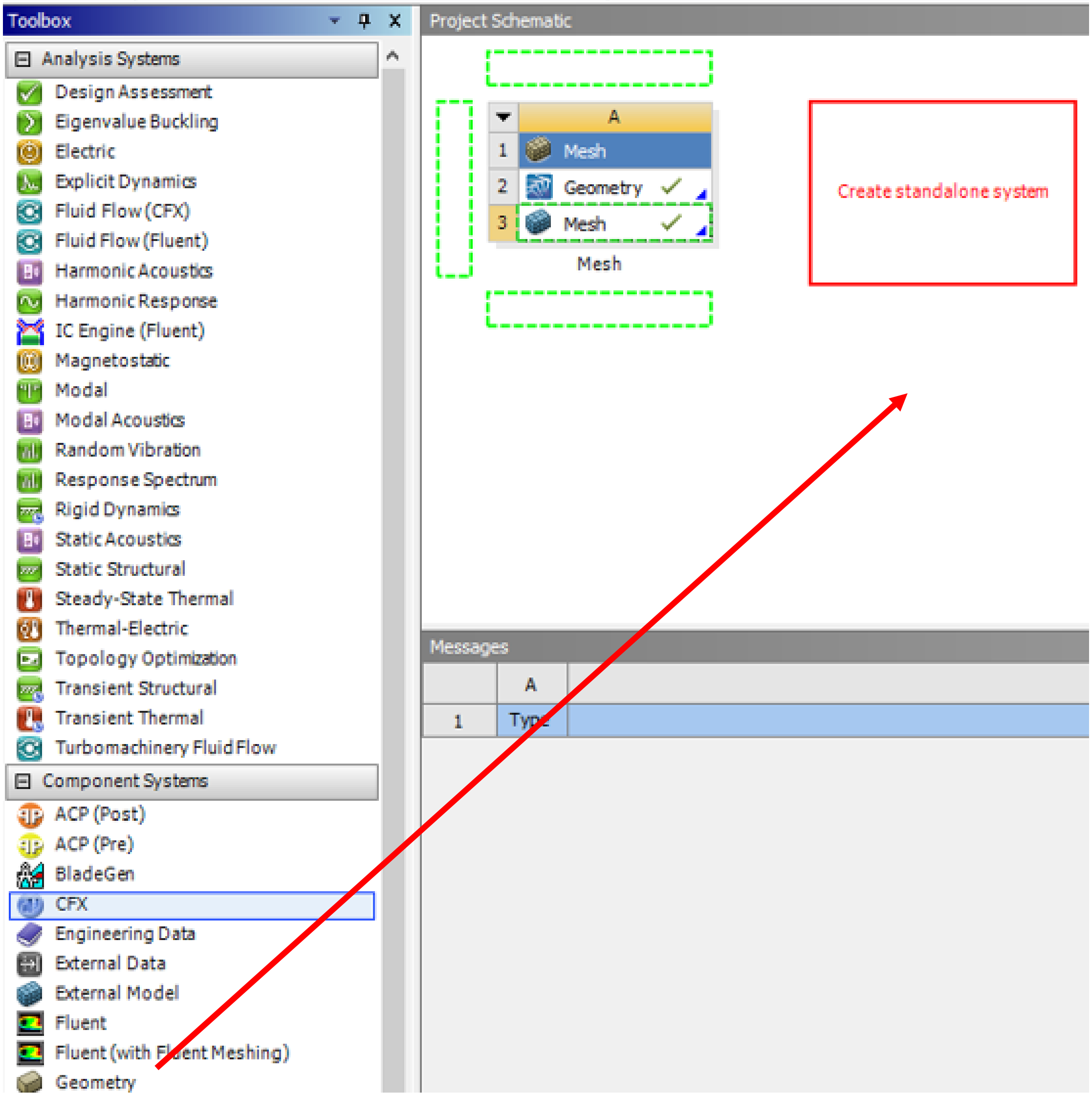
13) In *Boundary* select 4 blade surfaces and approve *Apply*. Generate a numerical grid.



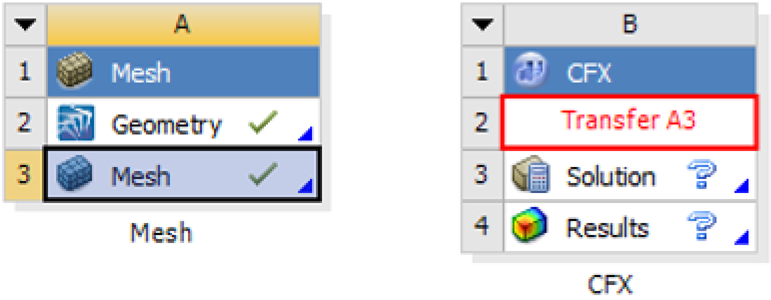
14) The last step is to name the bodies and surfaces. Give the following names using *Create Named Selection* (if you do not remember how to use *Create Named Selection*, check the previous instructions):
    a. Stator body – *Stator_domain*
    b. Rotor body – *Rotor_domain*
    c. Stator inlet – *inlet*
    d. Rotor outlet – *outlet*
    e. Stator flat surfaces (lower and upper) – *statorSym*
    f. Rotor flat surfaces (lower and upper) – *rotorSym*
    g. Contact surface between stator and rotor – *statorRotorInterface*
    h. Contact surface between rotor and stator – *rotorStatorInterface*
    i. Right lateral surface of the stator body – *statorPeriodic1*
    j. Left lateral surface of the stator body – *statorPeriodic2*
    k. Right lateral surface of the rotor body – *rotorPeriodic1*
    l. Left lateral surface of the rotor body – *rotorPeriodic2*
    m. Stator blade surface – *statorBlade*
    n. Rotor blade surface – *rotorBlade*
15) Close *Ansys Meshing* and save project in *Workbench.*

2.3. Numerical model

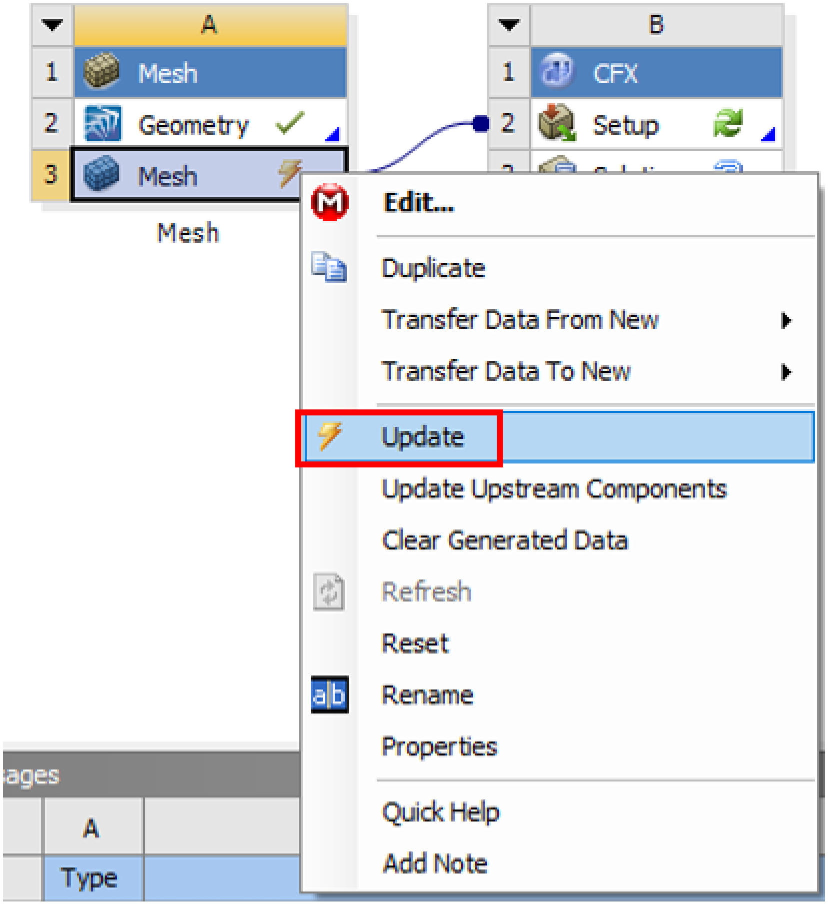
1) Insert *CFX*



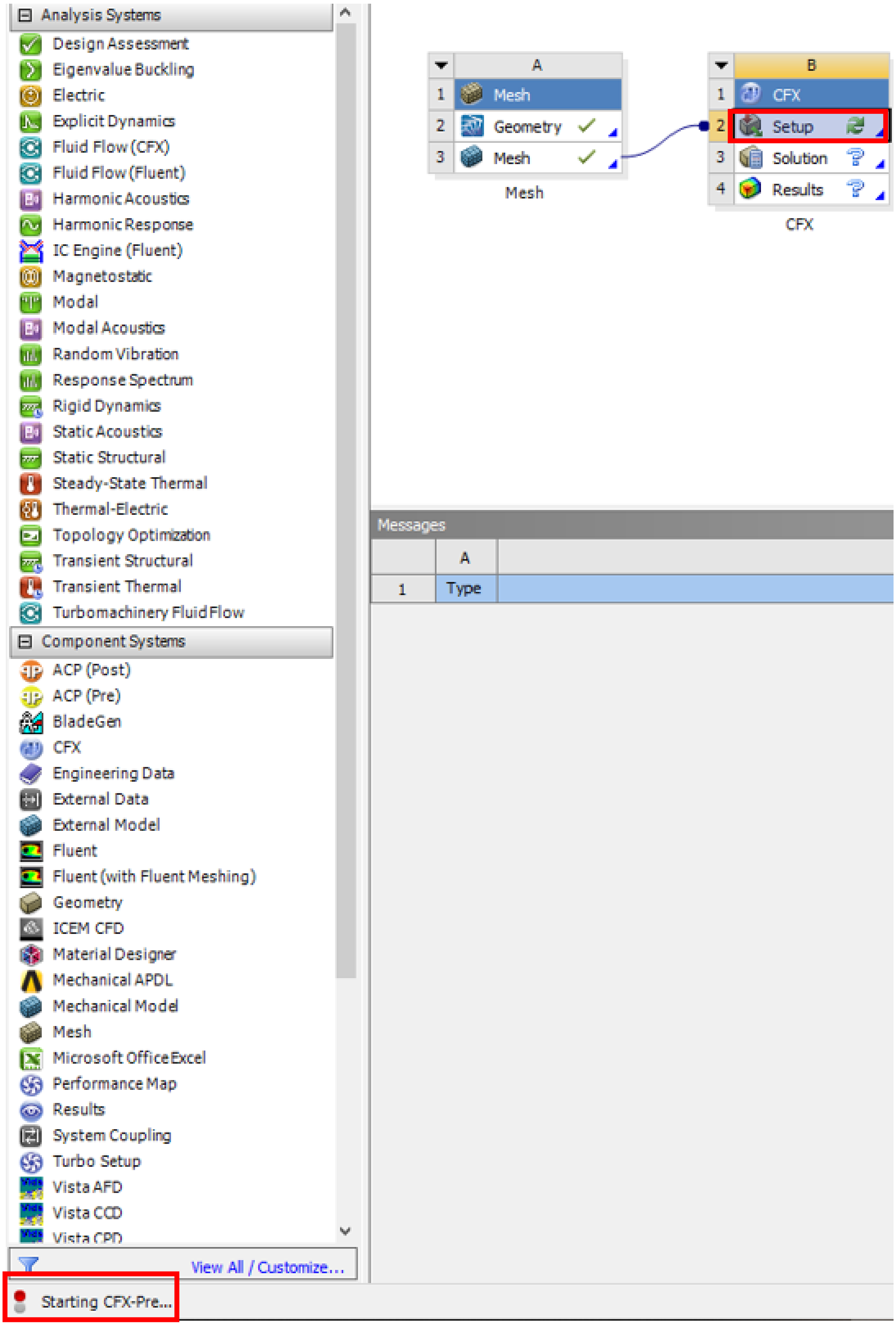
To connect the *Mesh* module with *CFX*, grab the LMP *Mesh* (lower one) and drag it to the *Setup* until the *Transfer A3* box appears, and then release the LMP - the connection has been created



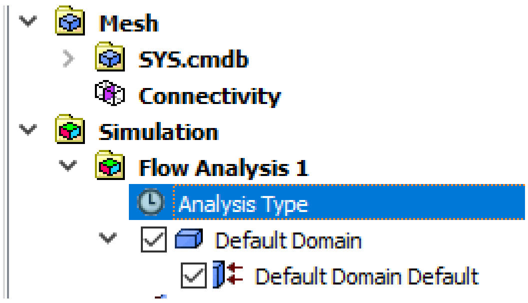
Click RMB on *Mesh* and select *Update*



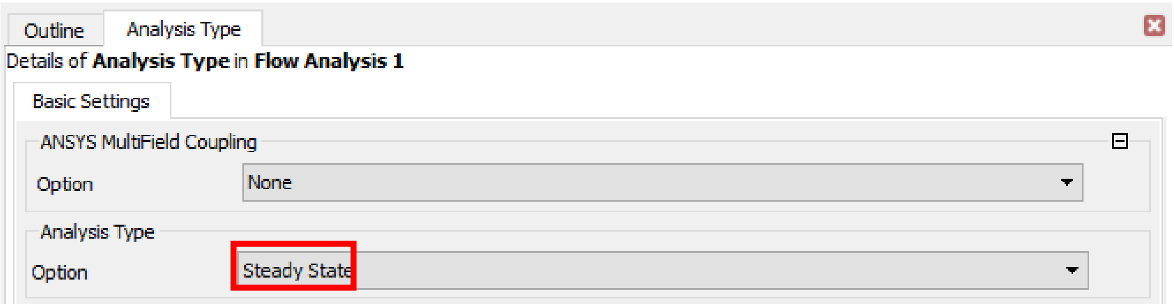
Double-click *Setup* to run *Ansys CFX*



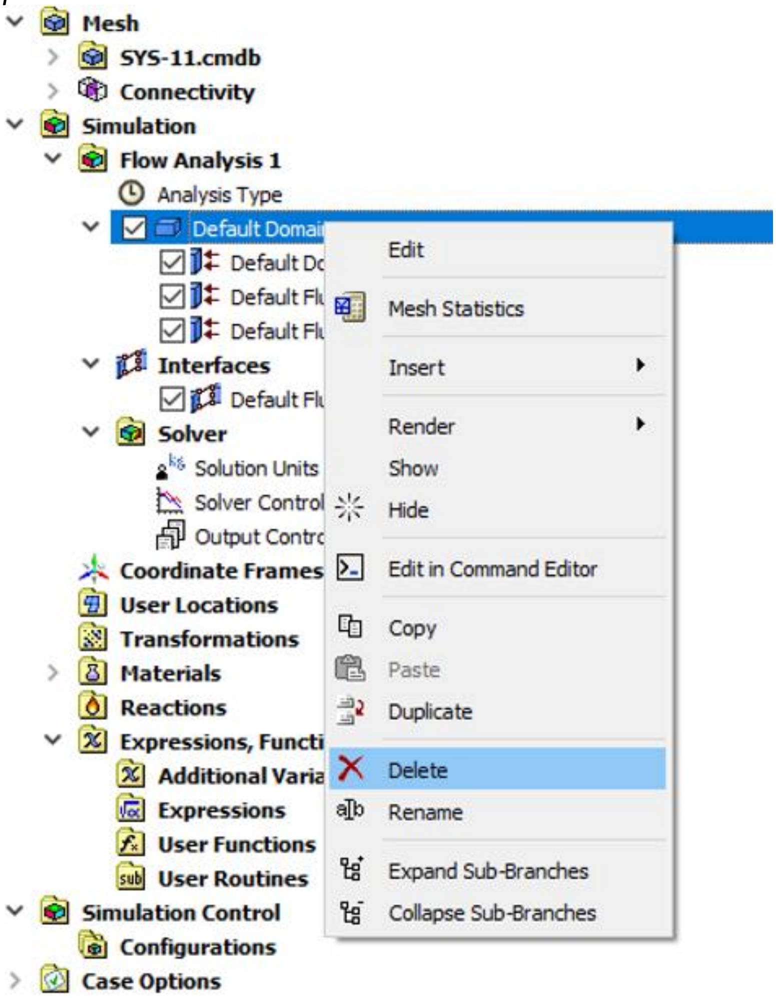
2) Open *Analysis Type* by double-click LMP



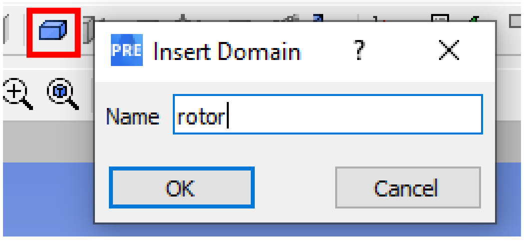
Apply the following settings and confirm *OK*.



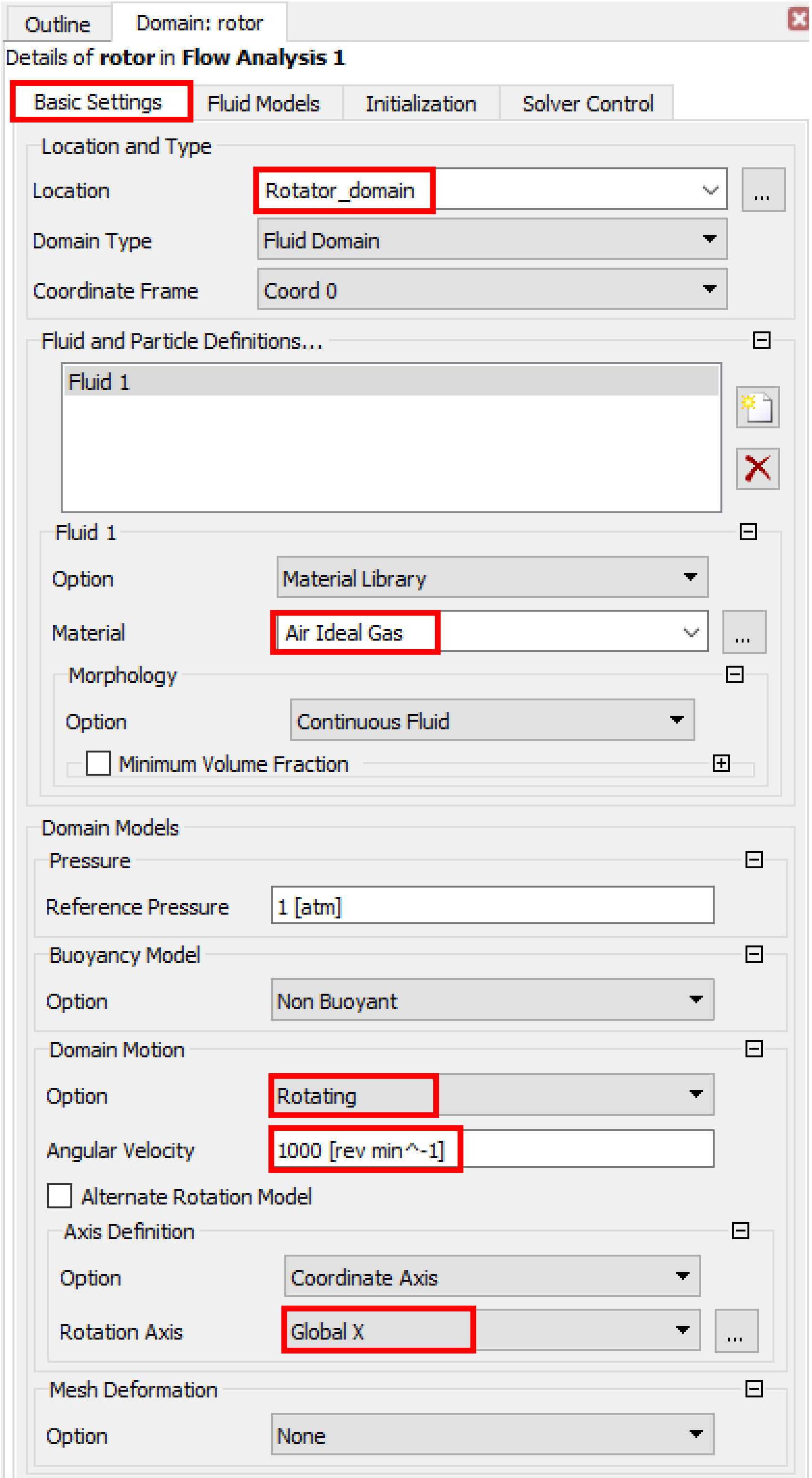
3) Delete *Default Domain*

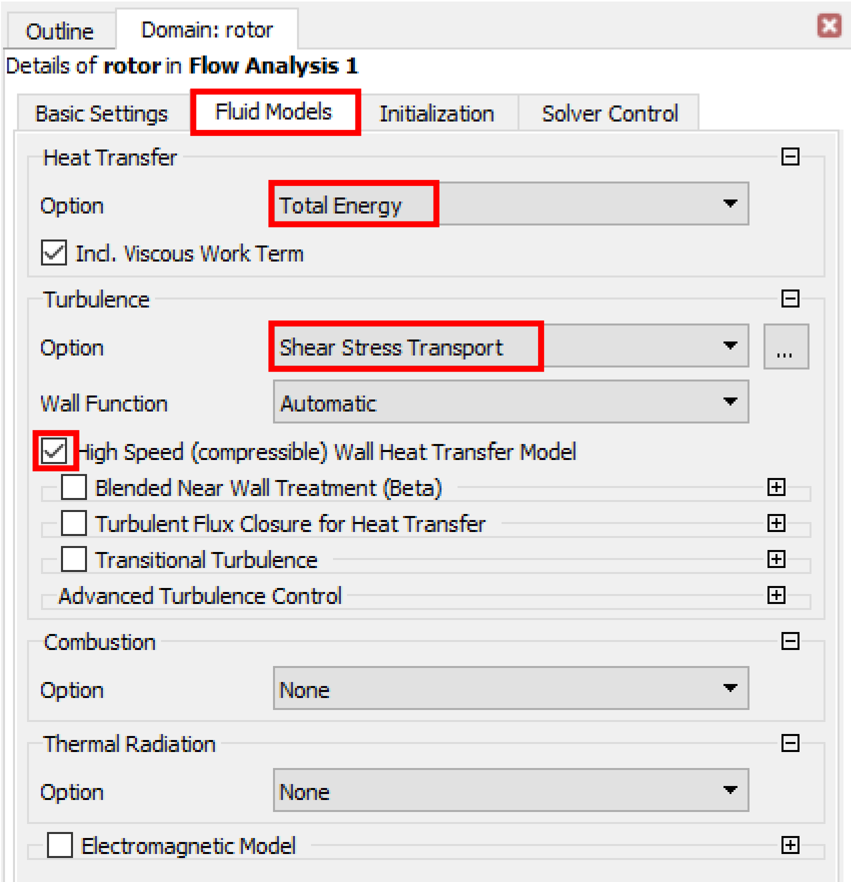


4) Create *rotor* domain

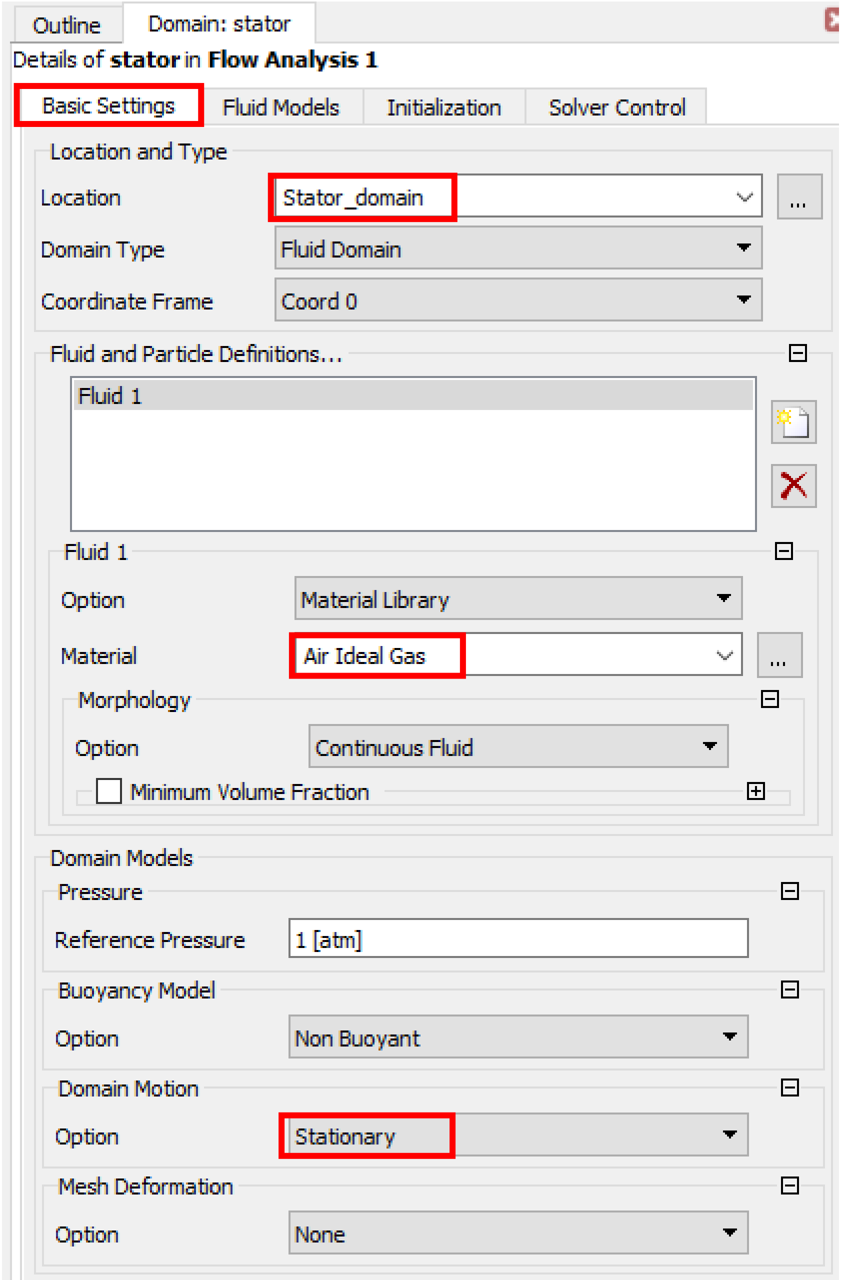


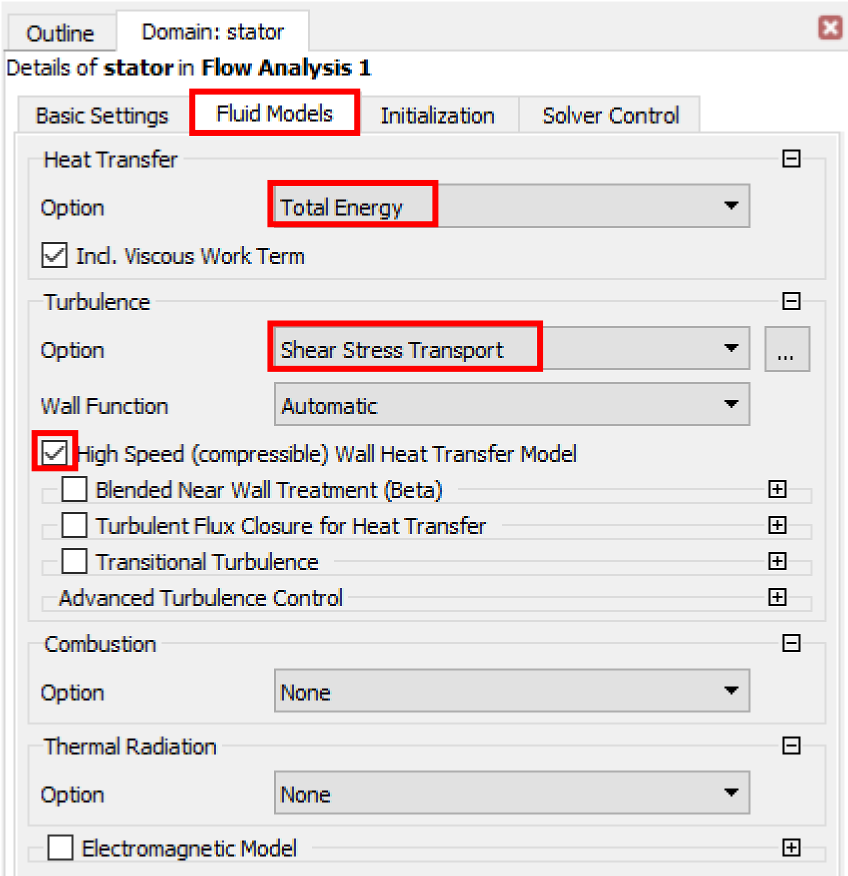
and apply below settings


Outline
Domain: rotor
Details of rotor in Flow Analysis 1
Basic Settings
Fluid Models
Initialization
Solver Control
Location and Type
Location
Rotator_domain
Domain Type
Fluid Domain
Coordinate Frame
Coord 0
Fluid and Particle Definitions...
Fluid 1
Fluid 1
Option
Material Library
Material
Air Ideal Gas
Morphology
Option
Continuous Fluid
Minimum Volume Fraction
Domain Models
Pressure
Reference Pressure
1 [atm]
Buoyancy Model
Option
Non Buoyant
Domain Motion
Option
Rotating
Angular Velocity
1000 [rev min^-1]
Alternate Rotation Model
Axis Definition
Option
Coordinate Axis
Rotation Axis
Global X
Mesh Deformation
Option
None

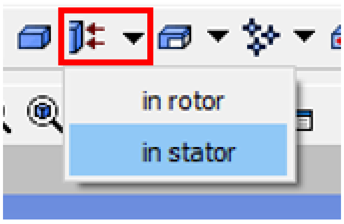


5) Similarly, create a domain *stator* and apply below settings

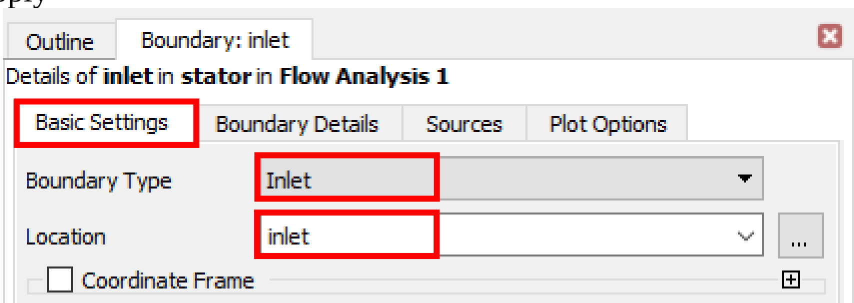

Outline
Domain: stator
Details of stator in Flow Analysis 1
Basic Settings
Fluid Models
Initialization
Solver Control
Location and Type
Location
Stator_domain
Domain Type
Fluid Domain
Coordinate Frame
Coord 0
Fluid and Particle Definitions...
Fluid 1
Fluid 1
Option
Material Library
Material
Air Ideal Gas
Morphology
Option
Continuous Fluid
Minimum Volume Fraction
Domain Models
Pressure
Reference Pressure
1 [atm]
Buoyancy Model
Option
Non Buoyant
Domain Motion
Option
Stationary
Mesh Deformation
Option
None

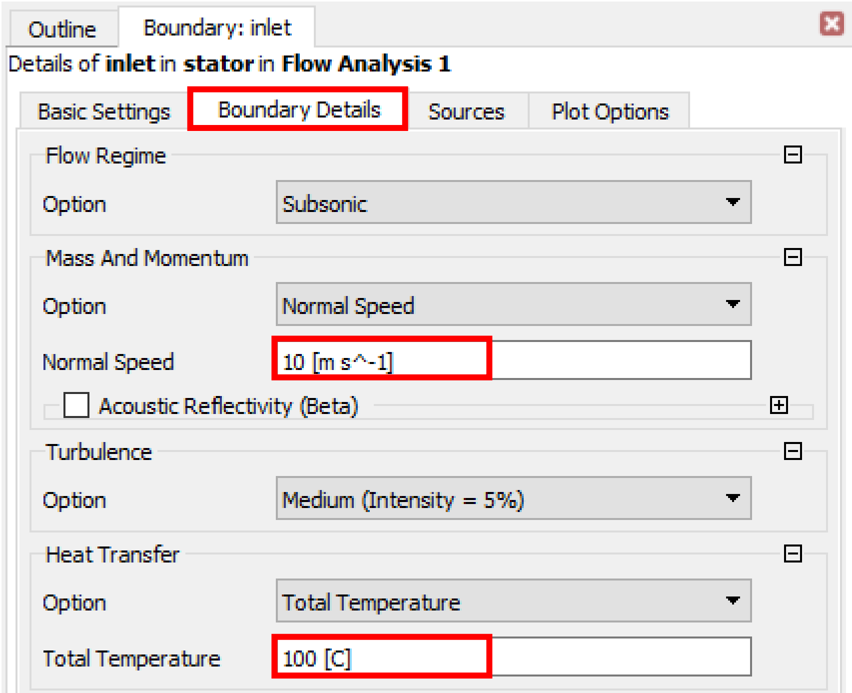


6) In *stator* domain

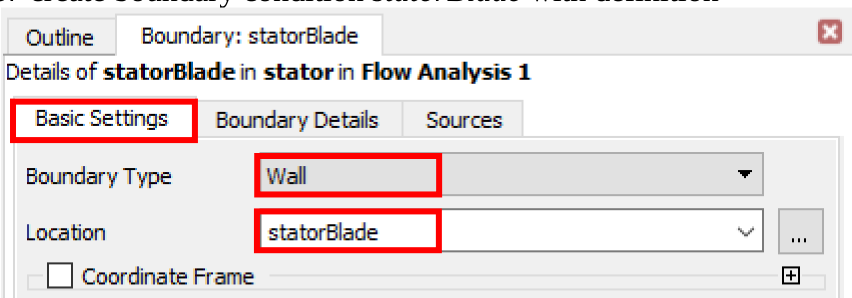


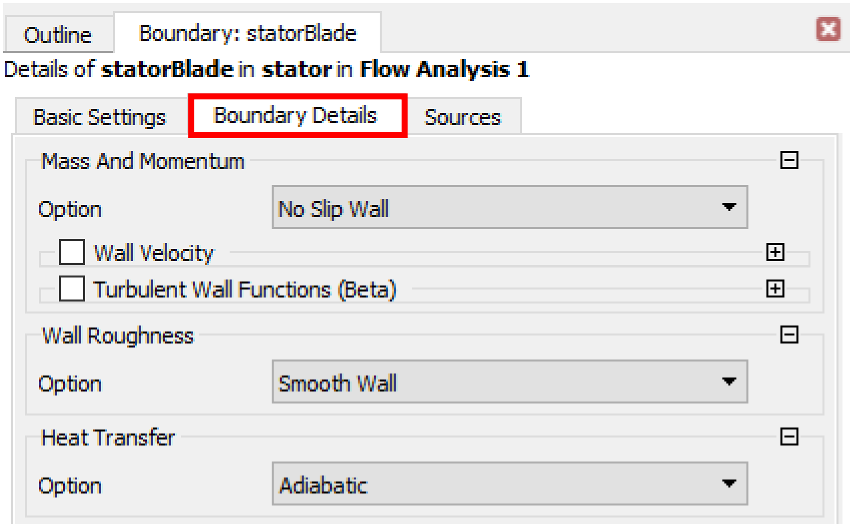
And apply



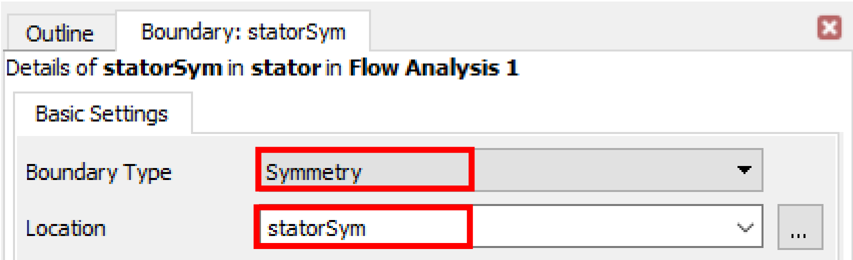


7) In *stator* create boundary condition *statorBlade* with definition

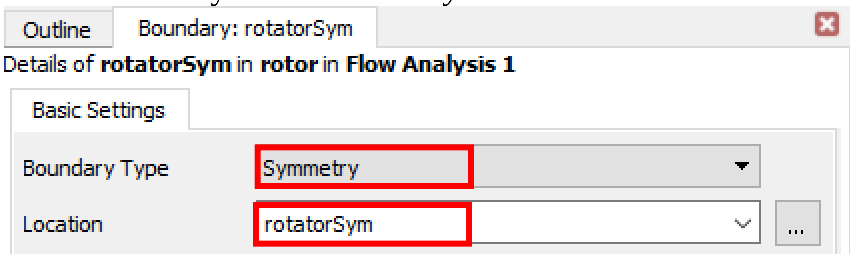




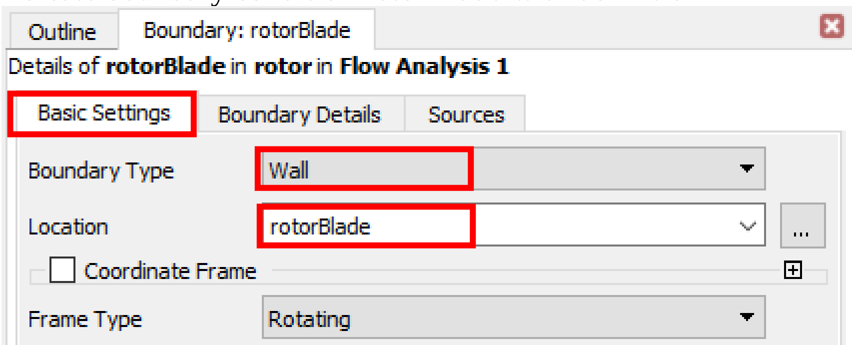
8) In *stator* create boundary condition *statorSym* with definition

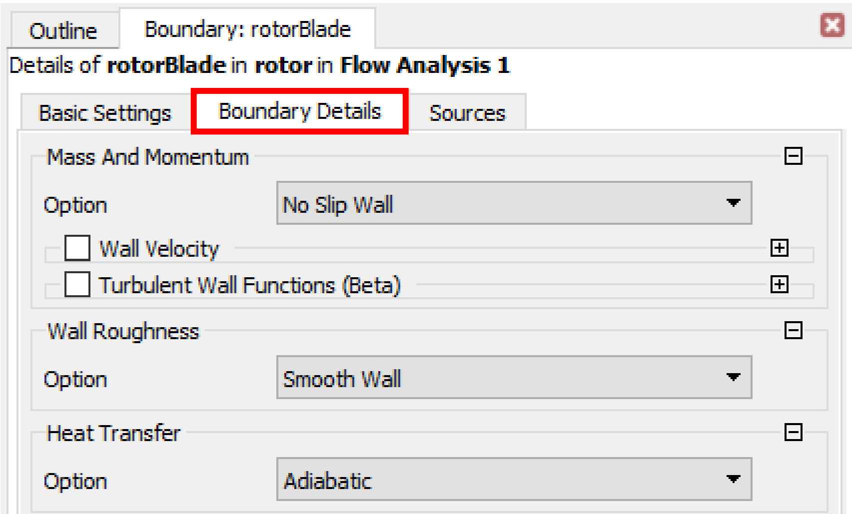


9) In *rotor* create boundary condition *rotorSym* with definition

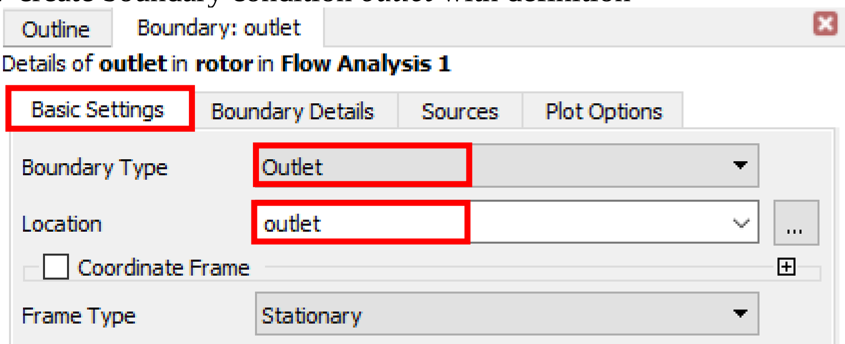


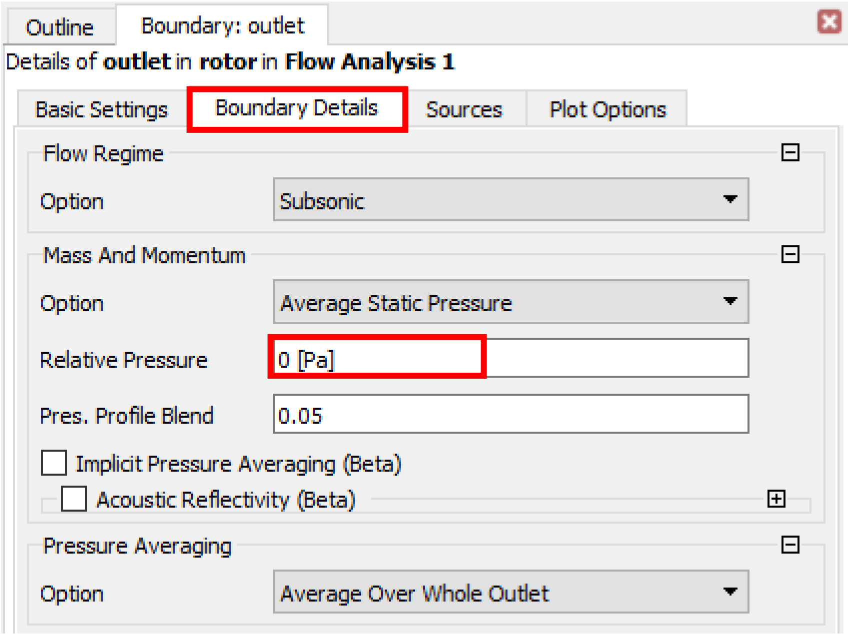
10) In *rotor* create boundary condition *rotorBlade* with definition



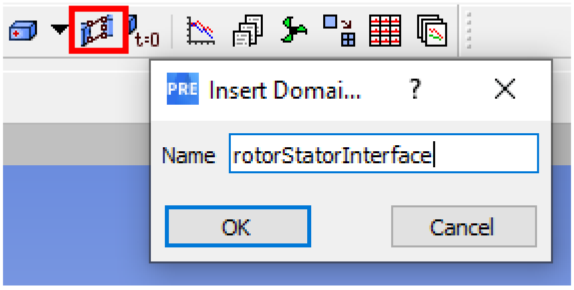


11) In *rotor* create boundary condition *outlet* with definition

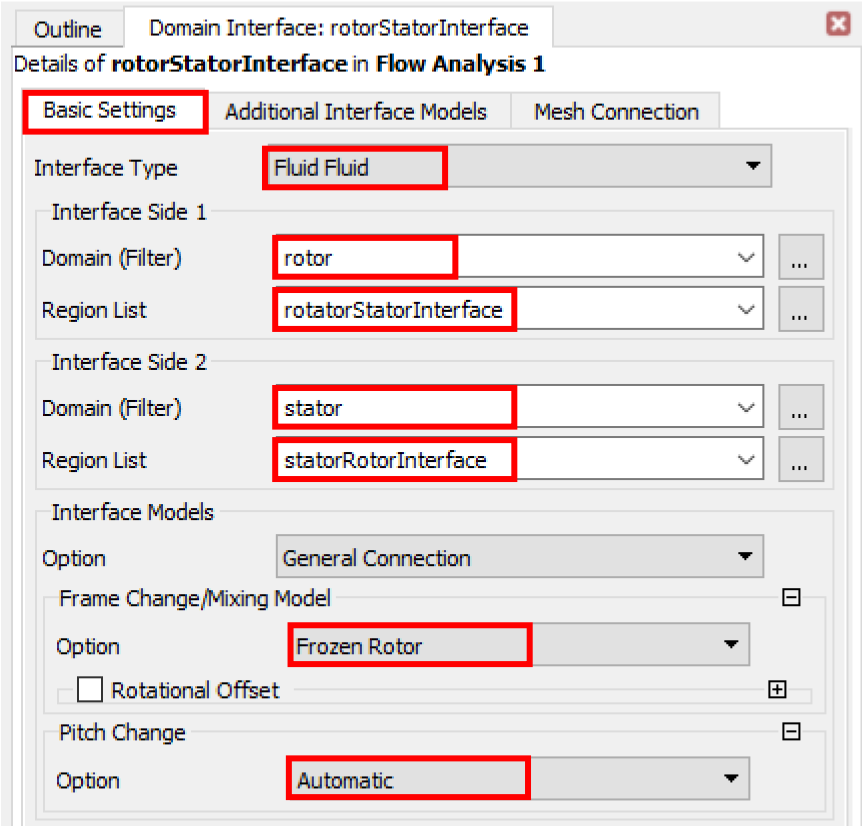




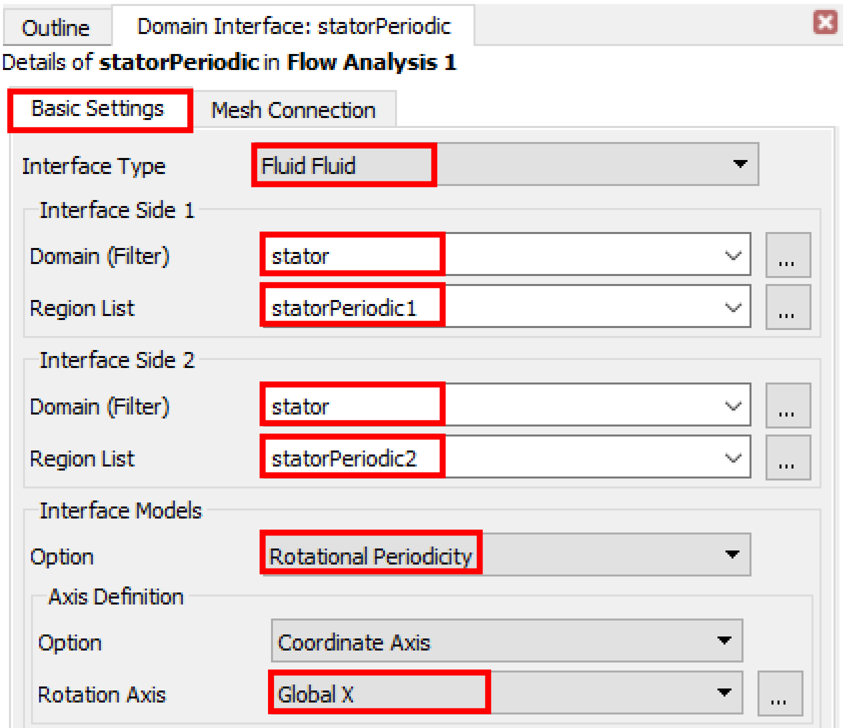
12) Create *interface* named *rotorStatorInterface*

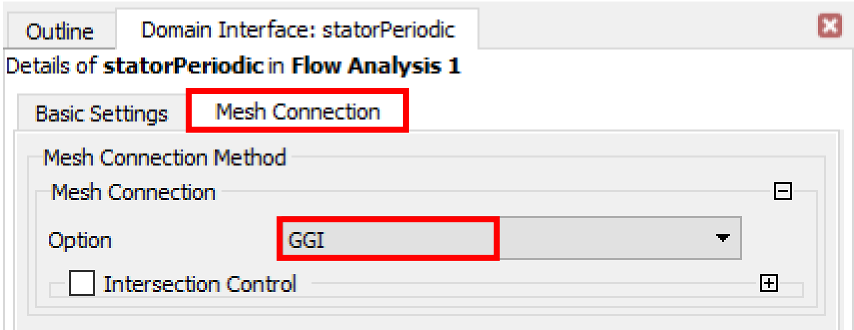


And apply the following

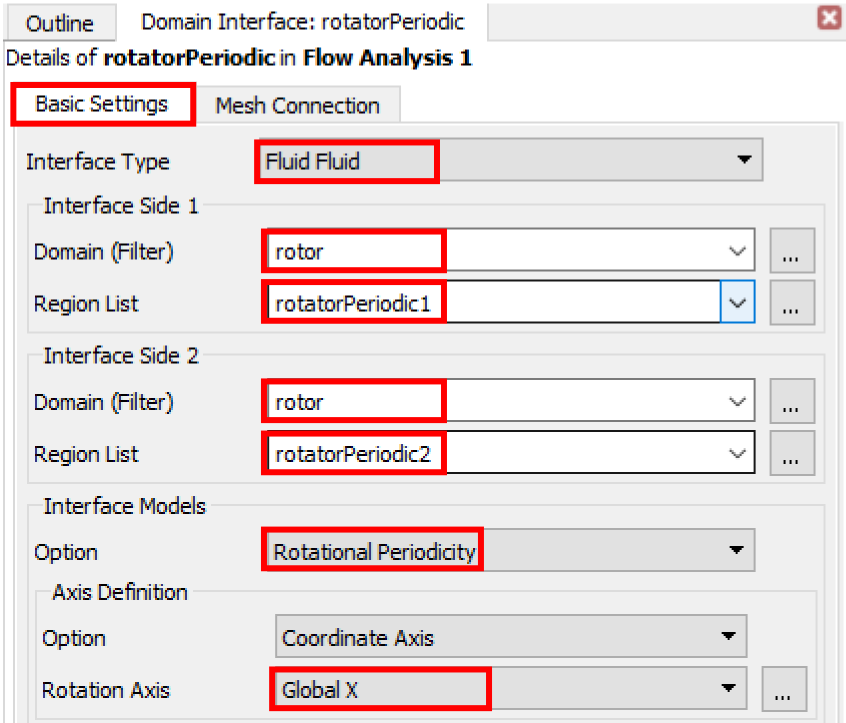


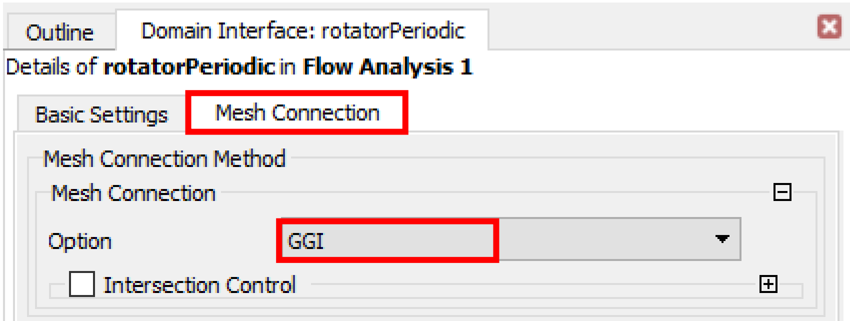
13) Create periodic boundary conditions in the domain *stator* using *interface* named *statorPeriodic*



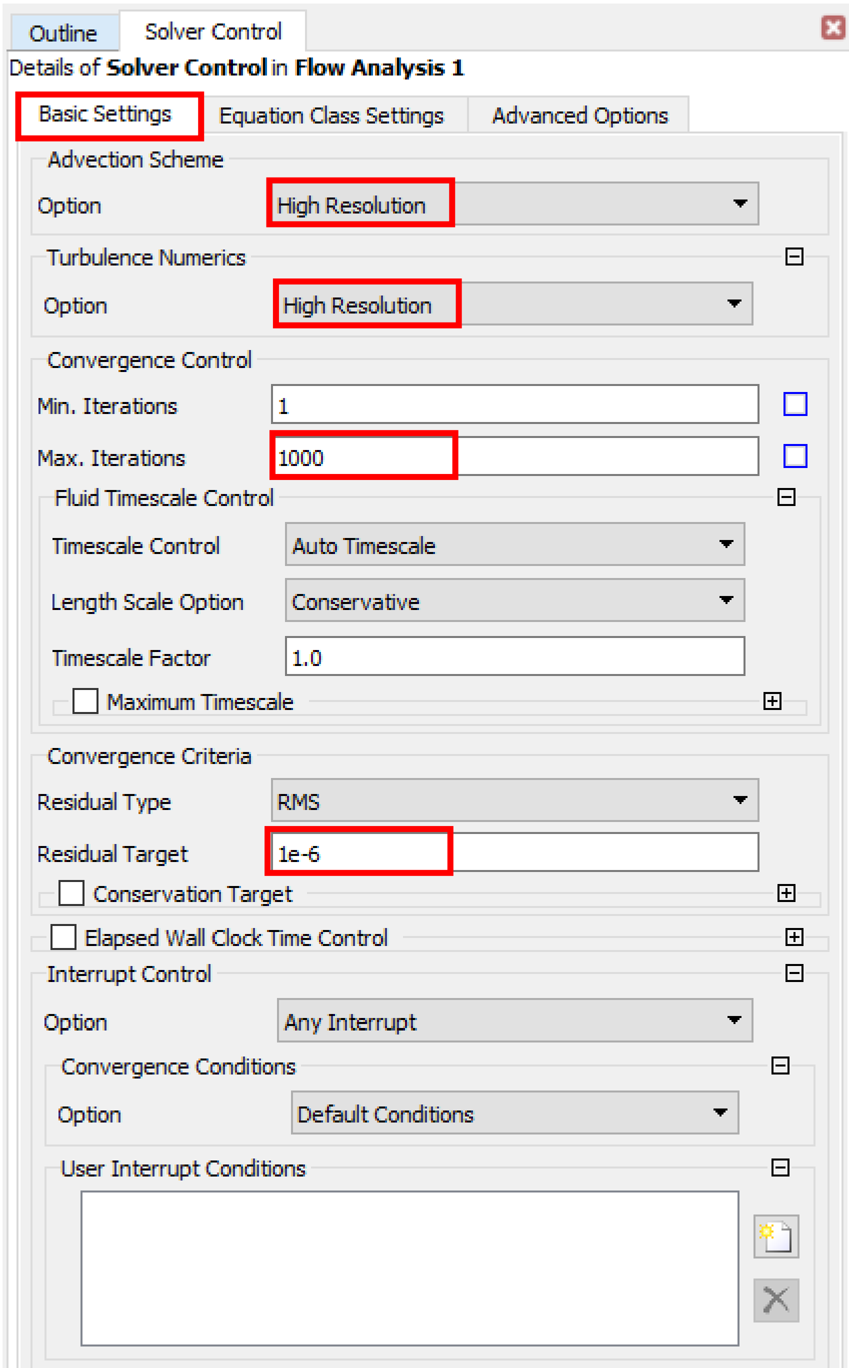


14) Create periodic boundary conditions in the domain *rotator* using *interface* named *rotorPeriodic*





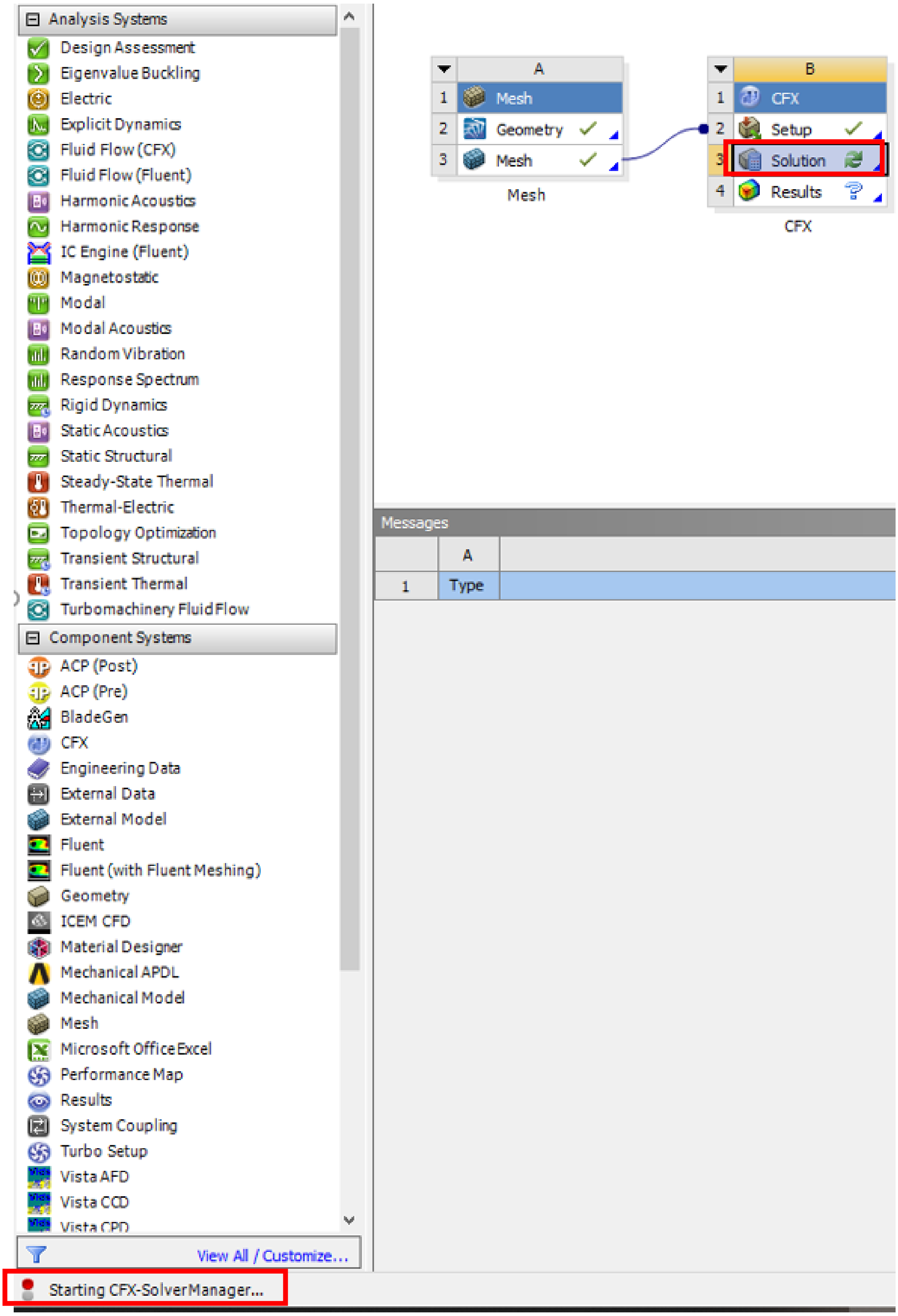
15) Create a monitoring point containing *expression* with the following definition: torque()@rotorBlade
16) Open *Solver Control* and apply



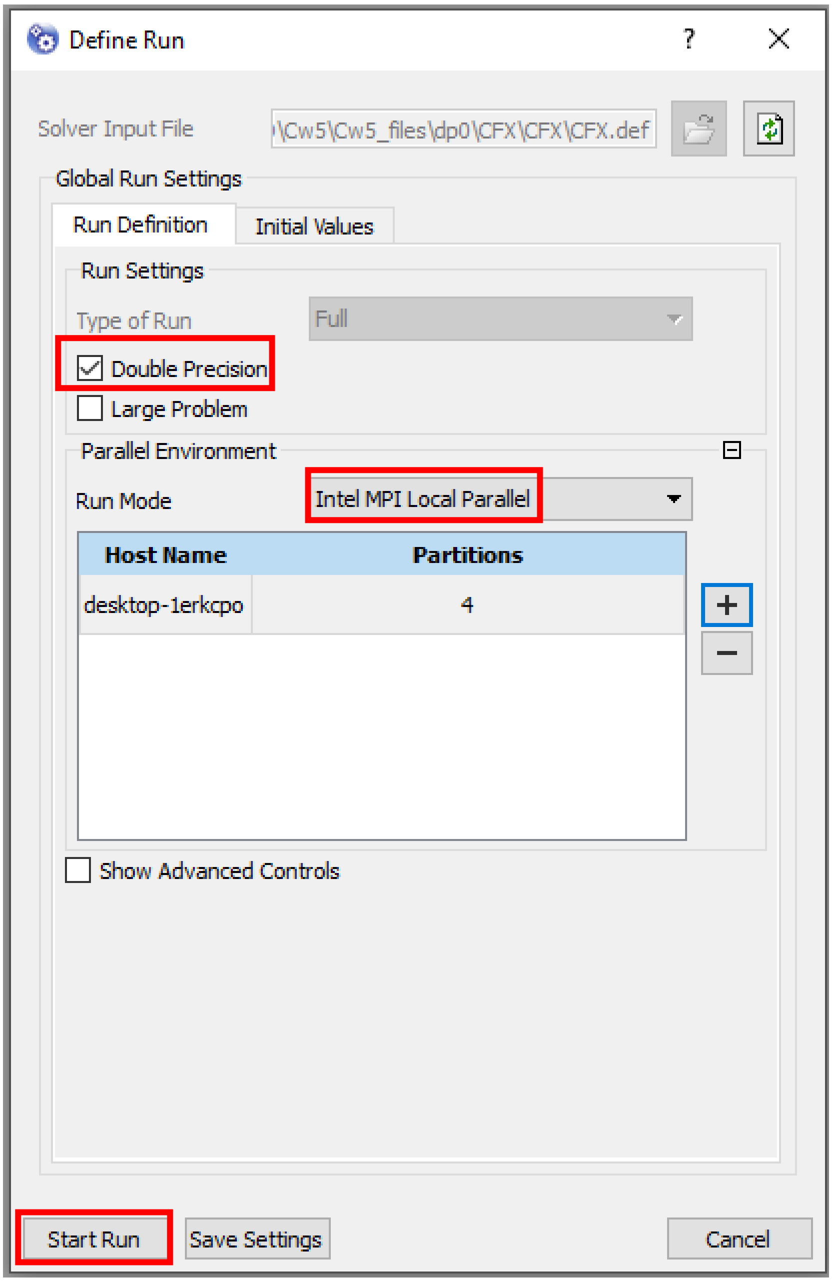
17) Close *Ansys CFX*.

2.4. Solver

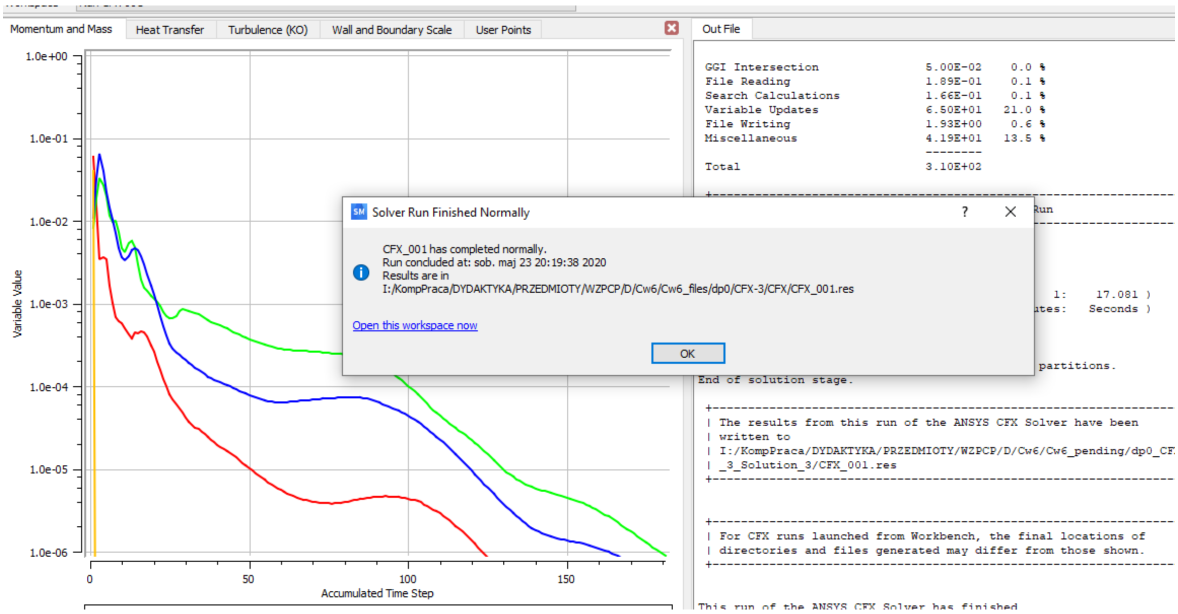
1) Double-click *Solution* to run *Ansys CFX Solver Manager*



2) Apply the following settings and press *Start Run*. The program will perform calculations. Wait a few moments for the message to complete the calculations.



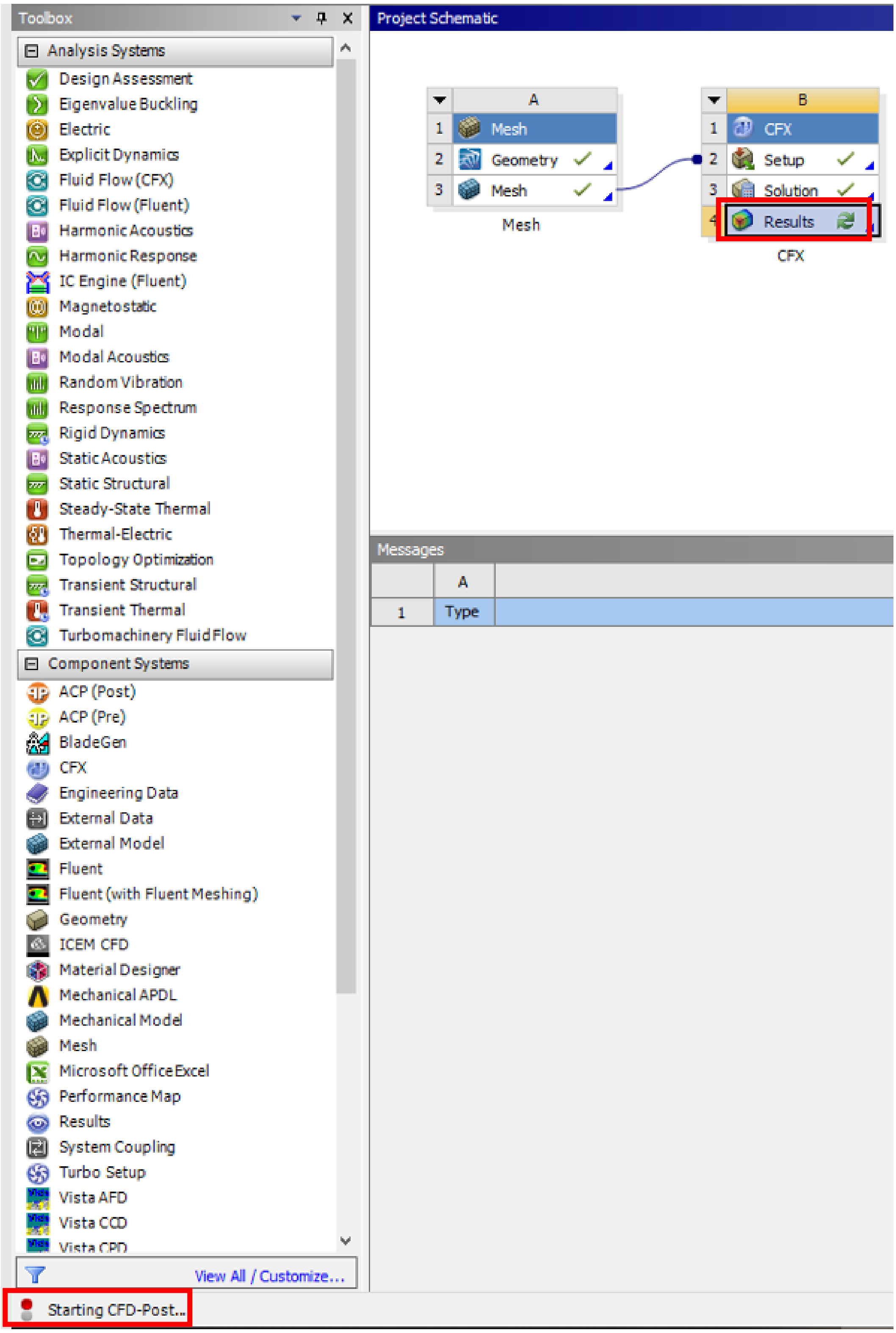
3) Calculations take about 5 minutes. Convergence is achieved after about 150 iterations. Watch the individual tabs as they change. Pay particular attention to the *User Point* tab, where the torque is shown on the rotor blade profile. Steady state will be reached when the curve stabilizes, which will occur after about 50 iterations. Additional iterations should be performed until all residuals have stabilized. To stop, press the *Stop* button at the top of the screen and confirm *Yes*.



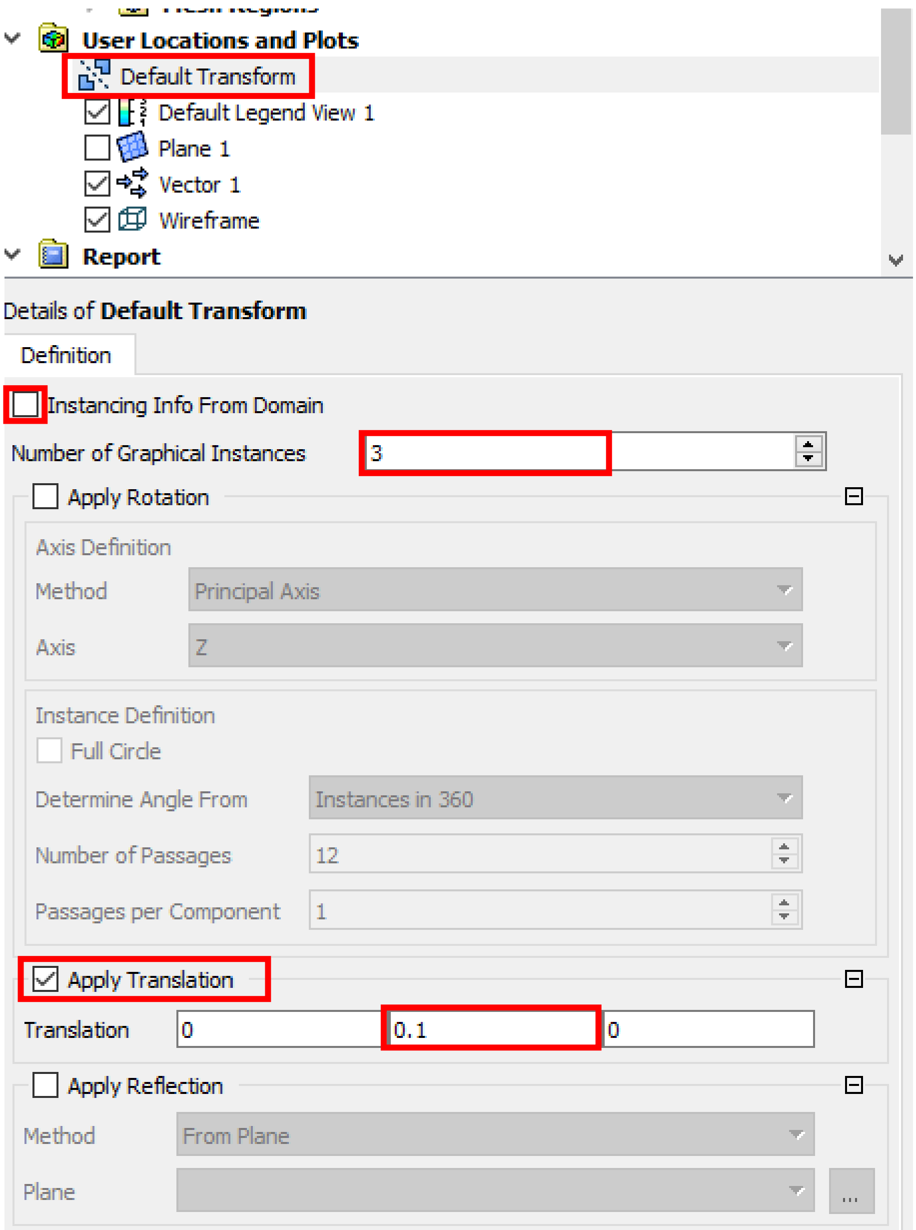
4) After completing the calculations, the program will display a confirmation message.
5) Confirm *OK* and close *Ansys CFX Solver Manager*. Save project in *Workbench.*

2.5. RESULTS

1) Double-click LMP *Results* to run *Ansys CFD Post* and see the results.



2) For better observation of the results use *Default Transform*



3) Create a plane perpendicular to the profile (parallel to the *sym* surface) and show the contours
   a. *Pressure*
   b. *Absolute Pressure*
   c. *Total Pressure*
   d. Relative velocity – *Velocity*
   e. Absolute velocity – *Velocity in Stn Frame*
   f. *Mach Number*
   g. *Density*
   h. *Temperature*
   i. *Total temperature*
   j. *Turbulence Kinetic Energy*
   k. *Turbulence Eddy Dissipation*
   l. *Yplus*
4) Using *areaAve* function calculate in a table:
   a. Average inlet speed value
   b. Value of the mass flow at the inlet and outlet

c. Value of average temperature on the surface of the blades
d. Average value of *Yplus* variable on the surface of the blades

5) Show the contours of the *Yplus* variable on the surface of the blades
6) Display the streamlines on the previously created plane
7) * Calculate the power generated in the turbine stage

3. RESULTS TO BE INCLUDED IN THE REPORT

1) Contours in a plane parallel to the surface *sym*:
    a. *Pressure*
    b. *Absolute Pressure*
    c. *Total Pressure*
    d. Relative velocity – *Velocity*
    e. Absolute velocity – *Velocity in Stn Frame*
    f. *Mach Number*
    g. *Density*
    h. *Temperature*
    i. *Total temperature*
    j. *Turbulence Kinetic Energy*
    k. *Turbulence Eddy Dissipation*
    l. *Yplus*
2) In table show the results of:
    a. Average inlet speed value
    b. Value of the mass flow at the inlet and outlet
    c. Value of average temperature on the surface of the blades
    d. Average value of *Yplus* variable on the surface of the blades
3) The contours of the *Yplus* variable on the surface of the blades
4) Streamlines
5) Answer the question: What is *Yplus*? Based on the analysis of the *Yplus* variable value, whether the SST turbulence model used was selected correctly?

4. OPTIONAL TASKS (NOT REQUIRED)

1. Perform calculations using a different turbulence model and compare the results.
2. Return to the *Meshing* module and perform calculations for: a) thinner numerical grid, b) denser numerical grid (especially on the surface of the blades). Check how it affects the calculations and the value of the *Yplus* variable and the torque.

5. REFERENCES

[1] I.H. Johnston, Dianne Smart, An experiment in turbine blade profile design, Ministry of technology Aeronautical Research Council, London, 1967.