Politechnika Wrocławska

Faculty of Mechanical and Power Engineering
Full-time studies

Selected problems of thermal-flow phenomena

Exercise no. 7

Modeling of flow around NACA2418 airfoil

Wrocław 2020

TABLE OF CONTENTS

1. INTRODUCTION

The exercise will show how to model compressible and supersonic fluid flow. The issue will be presented on the example of the flow around the NACA2418 airfoil. The airfoil with a chord length of 1 m is flowed in air at a temperature of 20 °C. The flow is carried out for three Reynolds numbers: 50,000, 500,000 and 5,000,000. The diagram of the analyzed case is presented in Fig. 1.

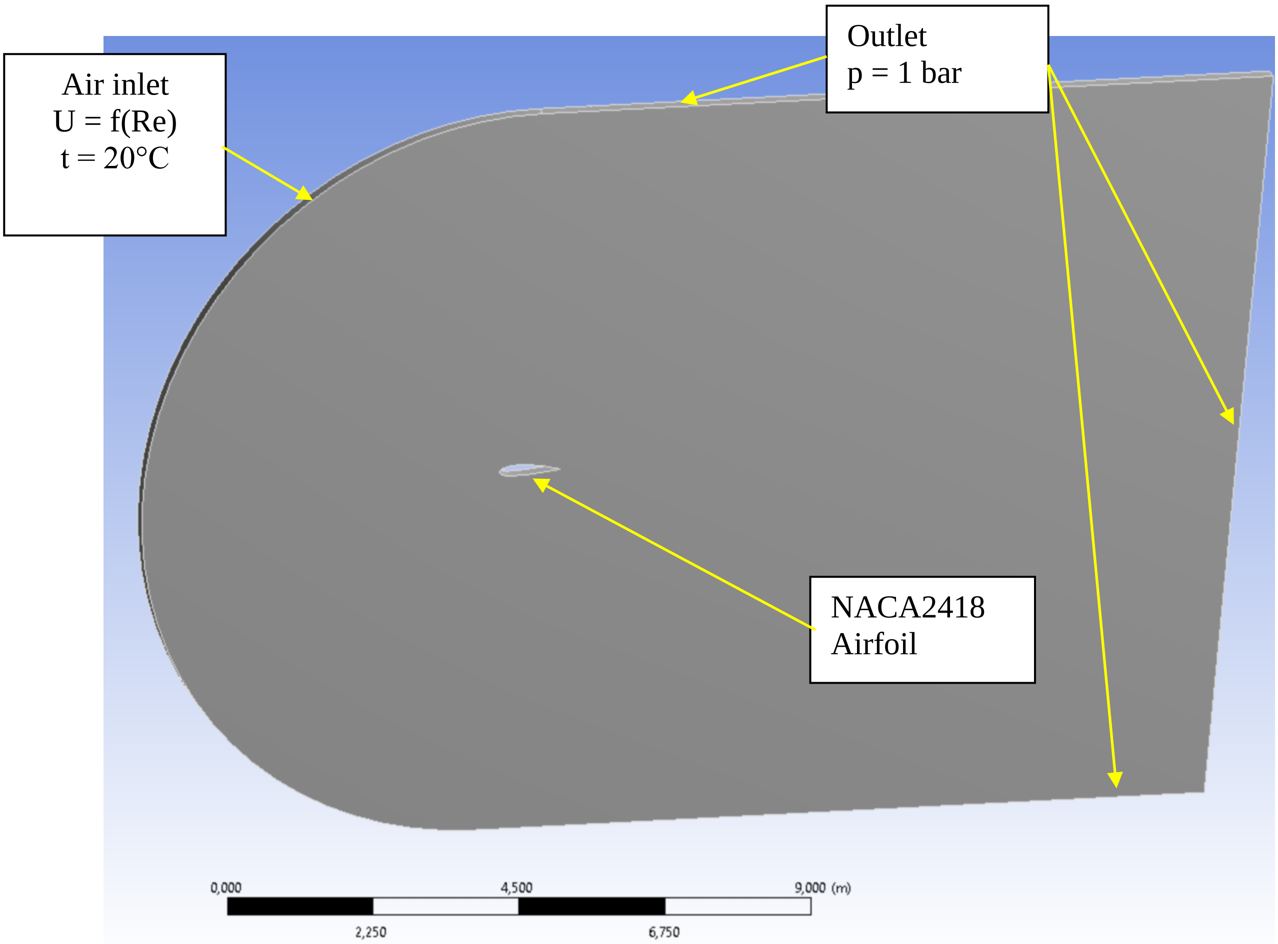


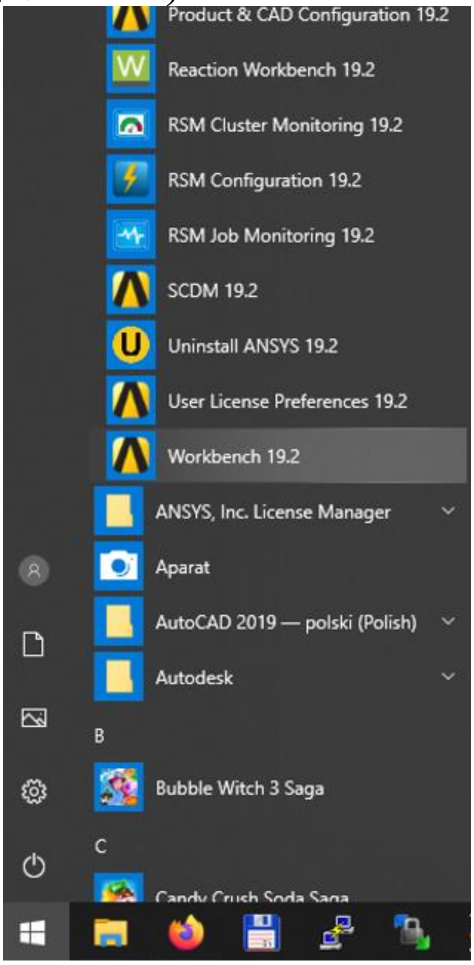
Fig. 1. Scheme of the flow around NACA2418 airfoil

2. FLOW AROUND NACA2418 AIRFOIL

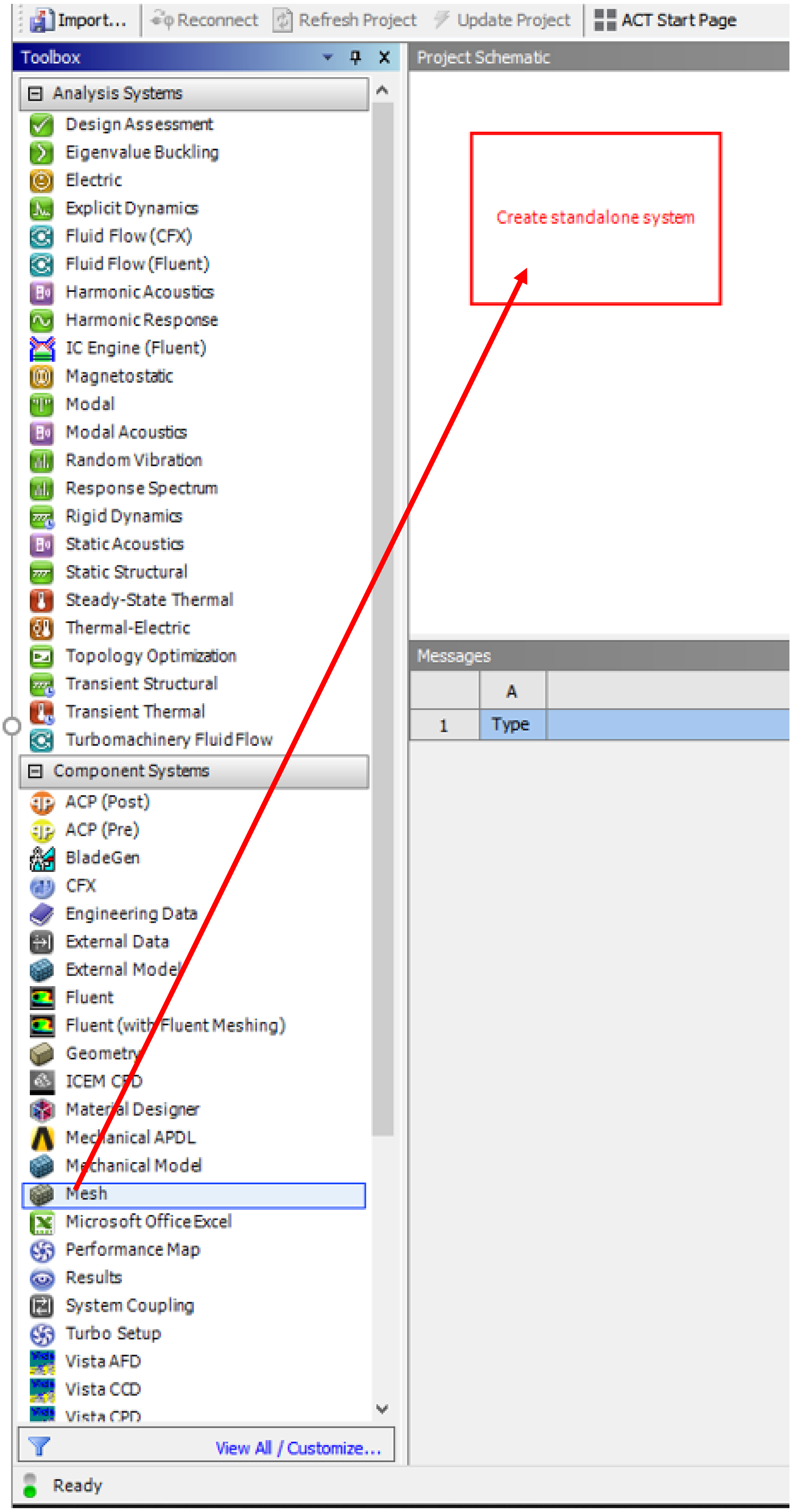
2.1. GEOMETRY

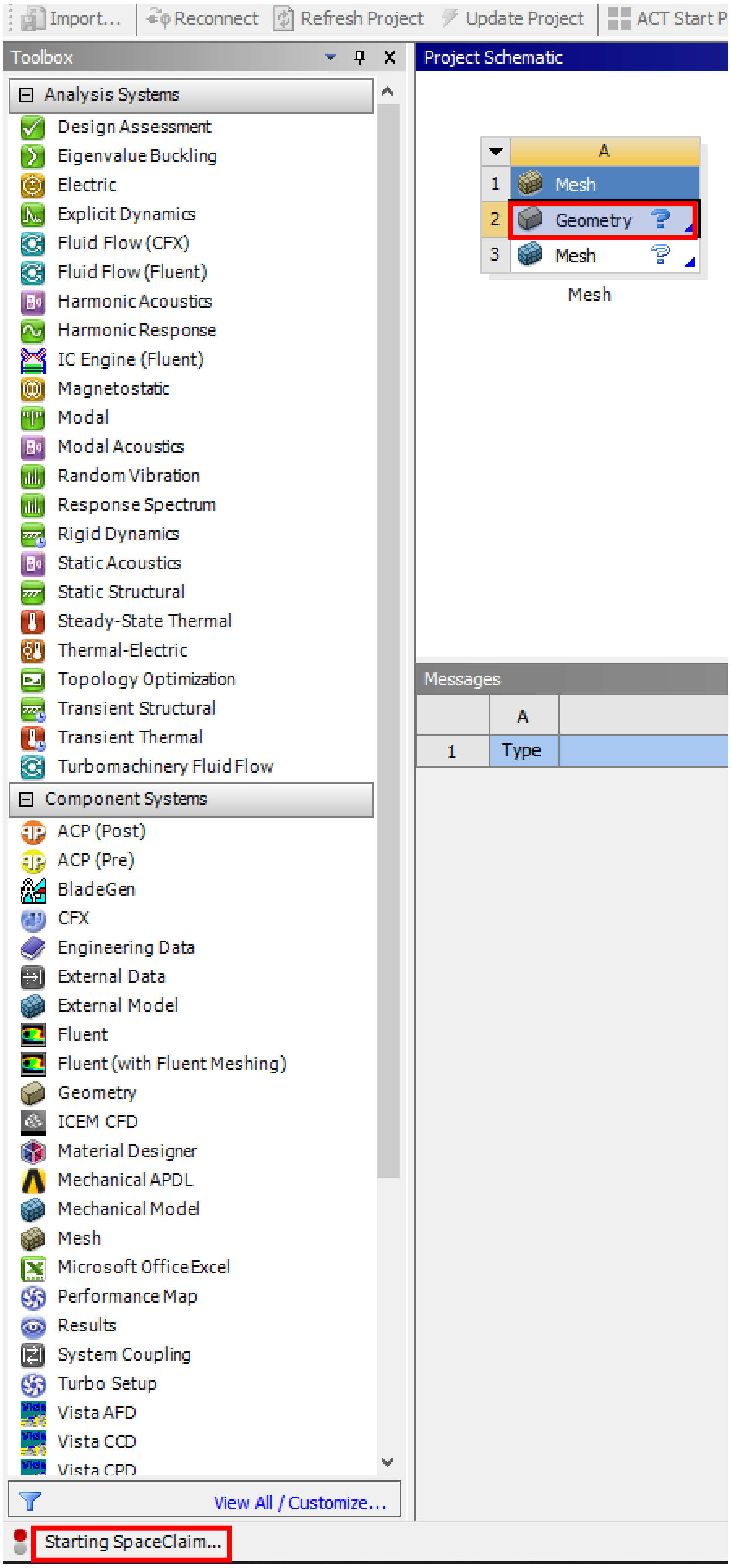
Do the following:

1) Open Ansys Workbench and save project as Ex7 in catalog Ex7 (*File->Save As*).
   RULE OF THUMB No. 1: *Create a separate catalog for each project*
   RULE OF THUMB No. 2: *In directory names, do not use: spaces, special characters (e.g. @ # $% ^ & * etc.) and Polish characters*

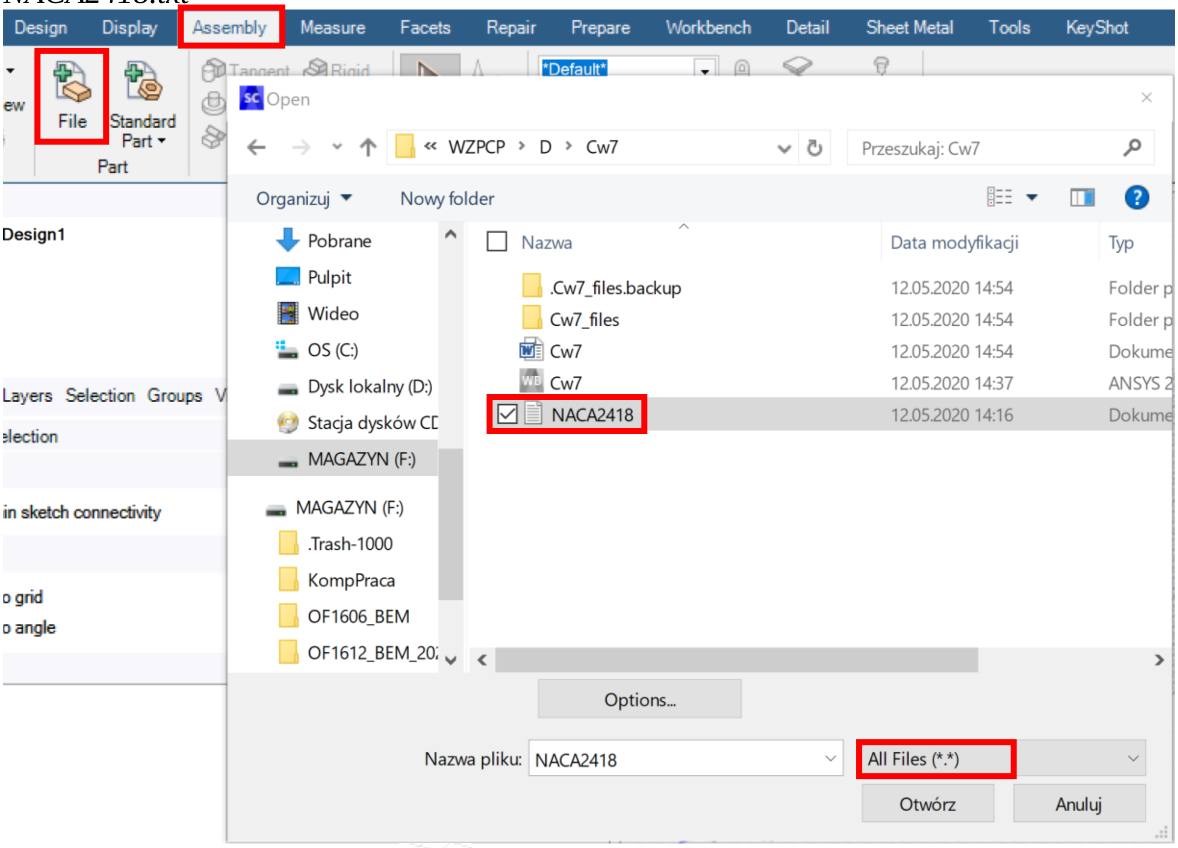


2) Select the *Mesh* module and open *Spaceclaim*. To do this, hold the left mouse button (LMB) on the *Mesh* module and drag it to the *Project Schematic* field. Then double-click LMB on *Geometry* to start the *Spaceclaim* program in which the geometry will be created. Note that in the lower left corner 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...
Ready
Create standalone system
Messages
A
1
Type

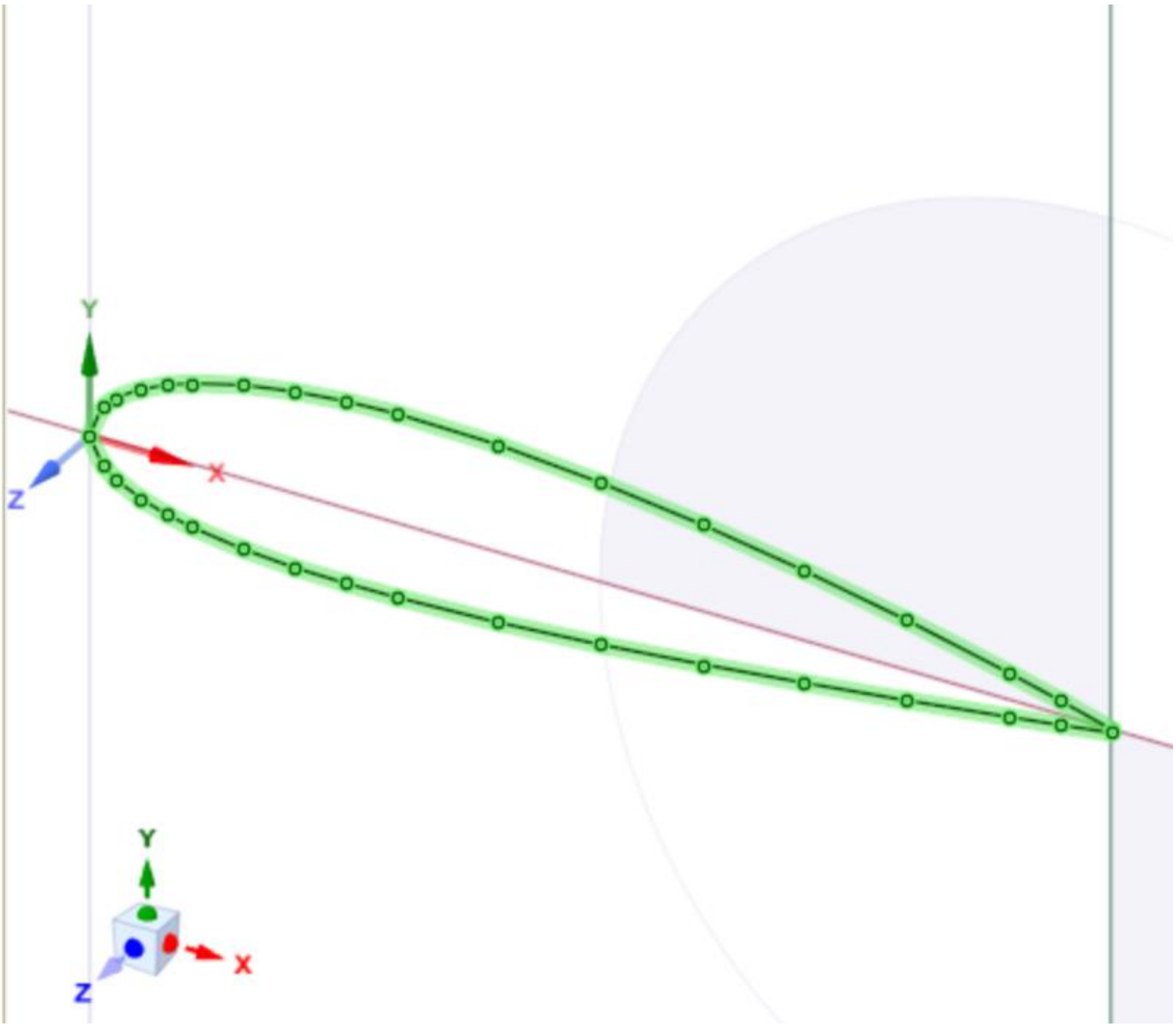

Import...
Reconnect
Refresh Project
Update Project
ACT Start P
Toolbox
Project Schematic
Analysis Systems
Design Assessment
Eigenvalue Buckling
Electric
Explicit Dynamics
Fluid Flow (CFX)
Fluid Flow (Fluent)
Harmonic Acoustics
Harmonic Response
IC Engine (Fluent)
Magnetostatic
Modal
Modal Acoustics
Random 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...
A
1 Mesh
2 Geometry
3 Mesh
Mesh
Messages
A
1 Type
Starting SpaceClaim...

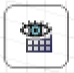
3) In *Assembly* tab select *File*, and then point to the file attached to the exercise *NACA2418.txt*



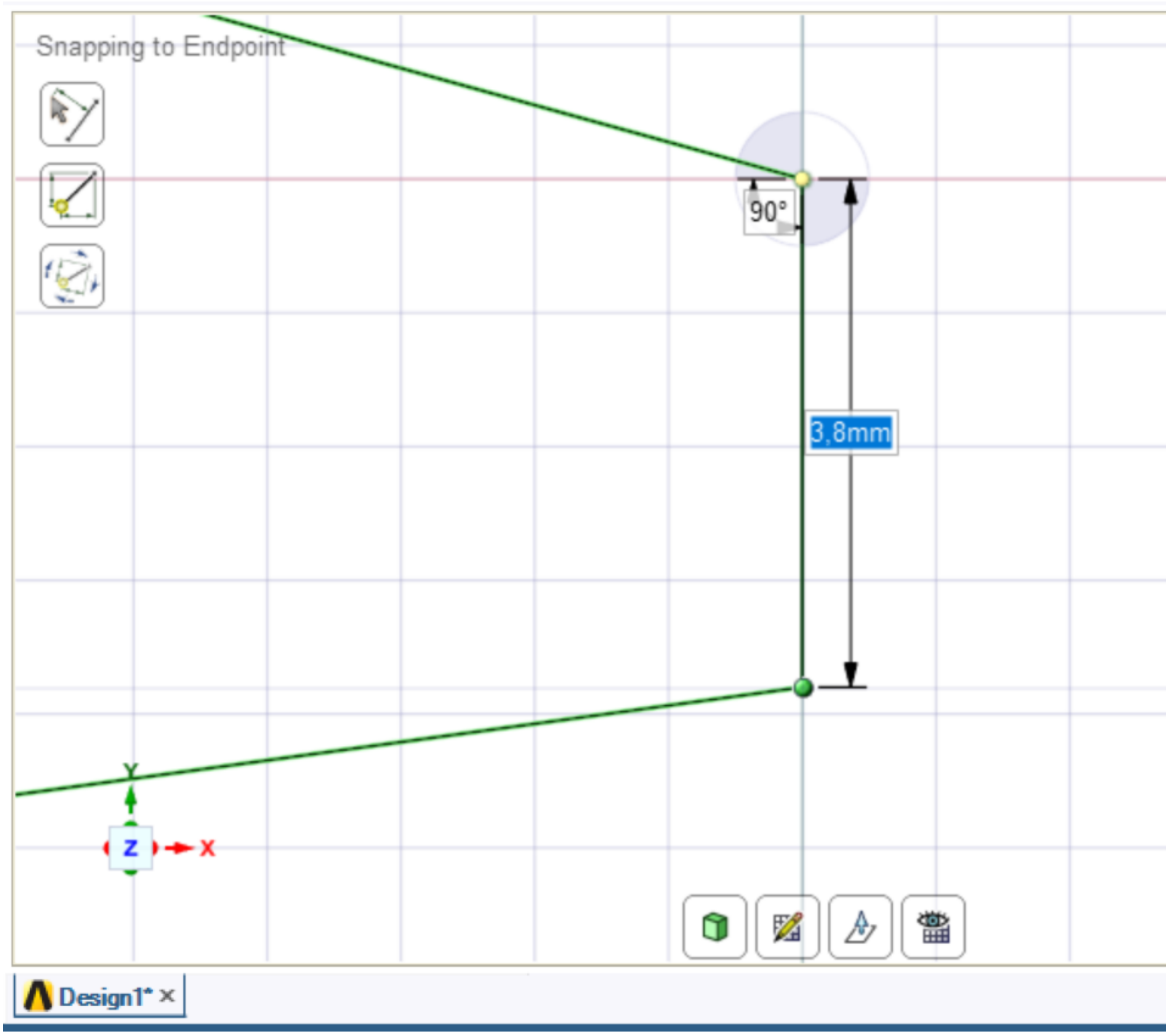
4) Click LMB *Select New Sketch*  to choose sketching plane.

Select the X-Y plane as shown below.



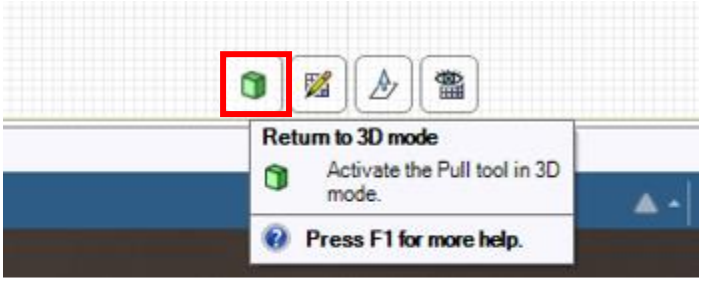
5) Click *Plan View*  to rotate the drawing plane parallel to the screen (you can also do this by pressing *Shift* + *v*).

6) In the panel at the top of the screen, select the line drawing icon  and draw a line near the trailing edge of the profile to close it.

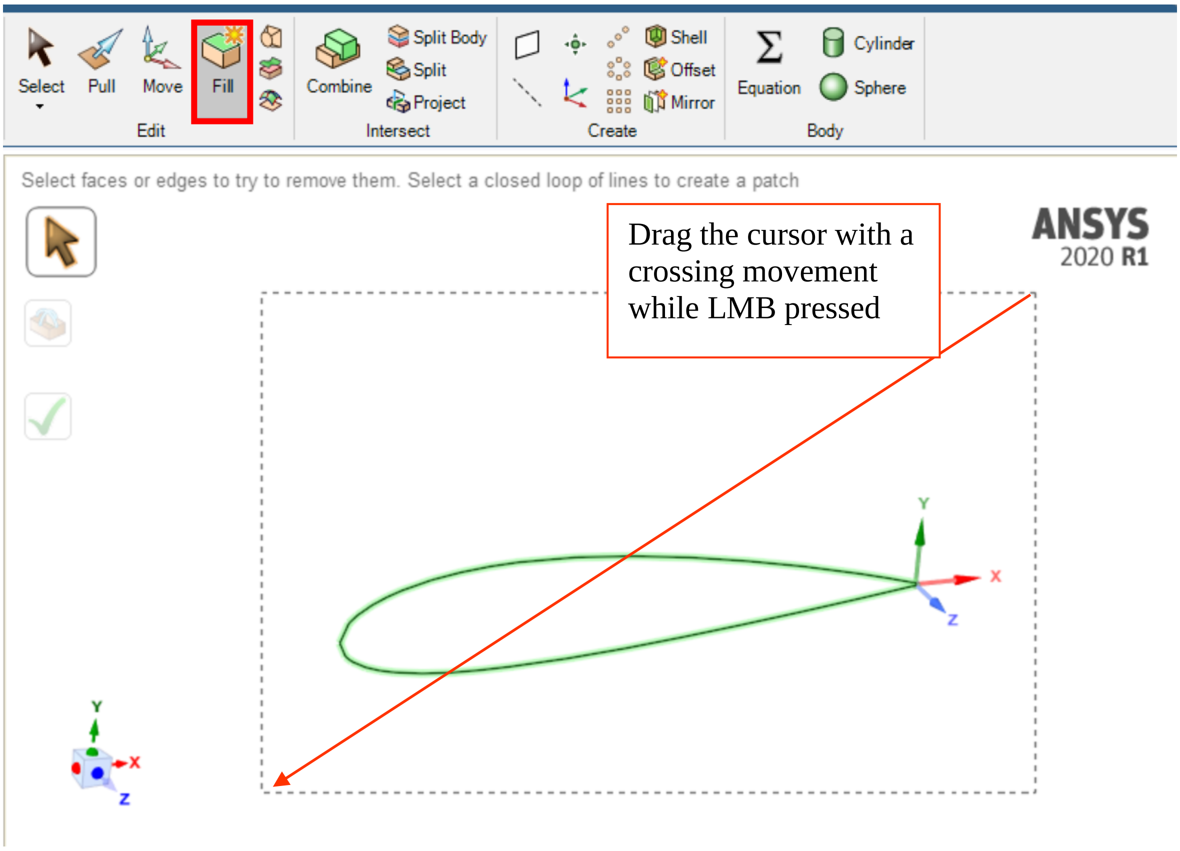


Remember that if you fail at any time, you can click the undo icon  (located in the upper left corner of the screen) or *Ctrl* + *z*.

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

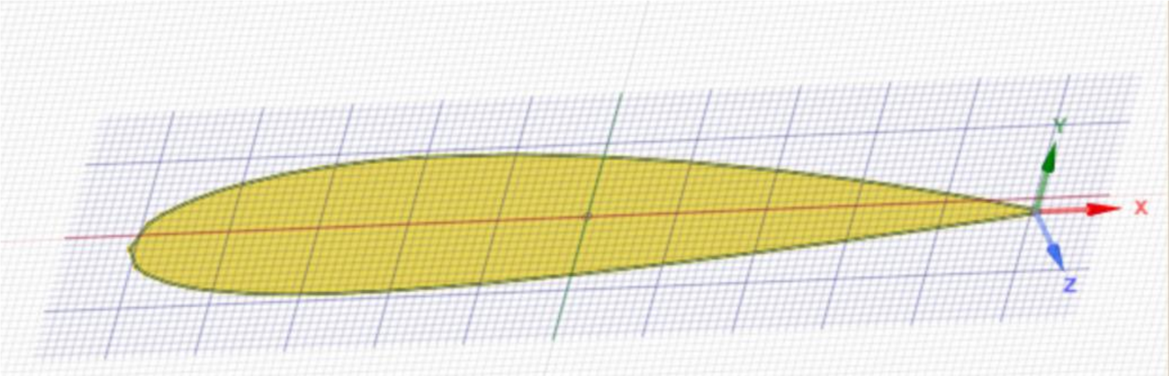


8) In *Design* tab, select *Fill*, and then select the entire profile by dragging LMB

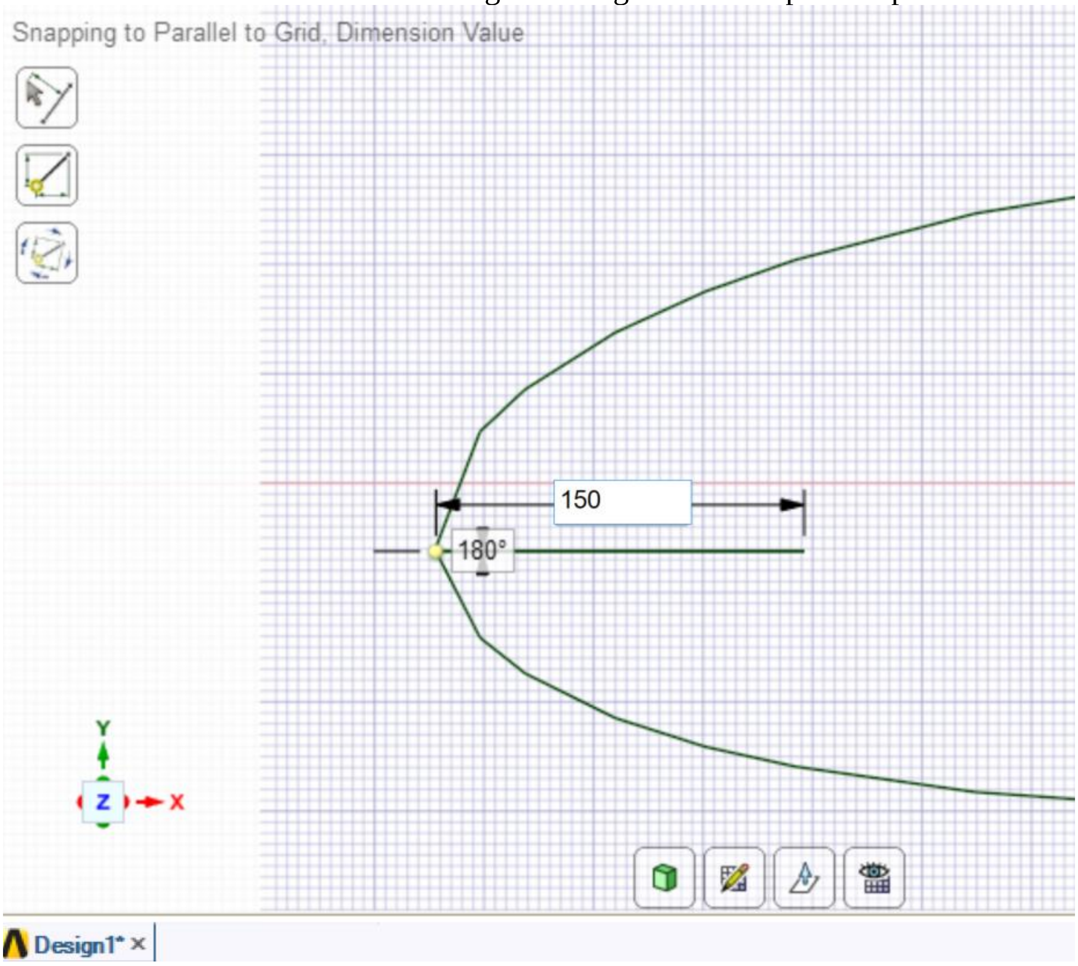


9) Select *Complete*

10) Select the plane to draw the sketch on one of the side surfaces of the profile

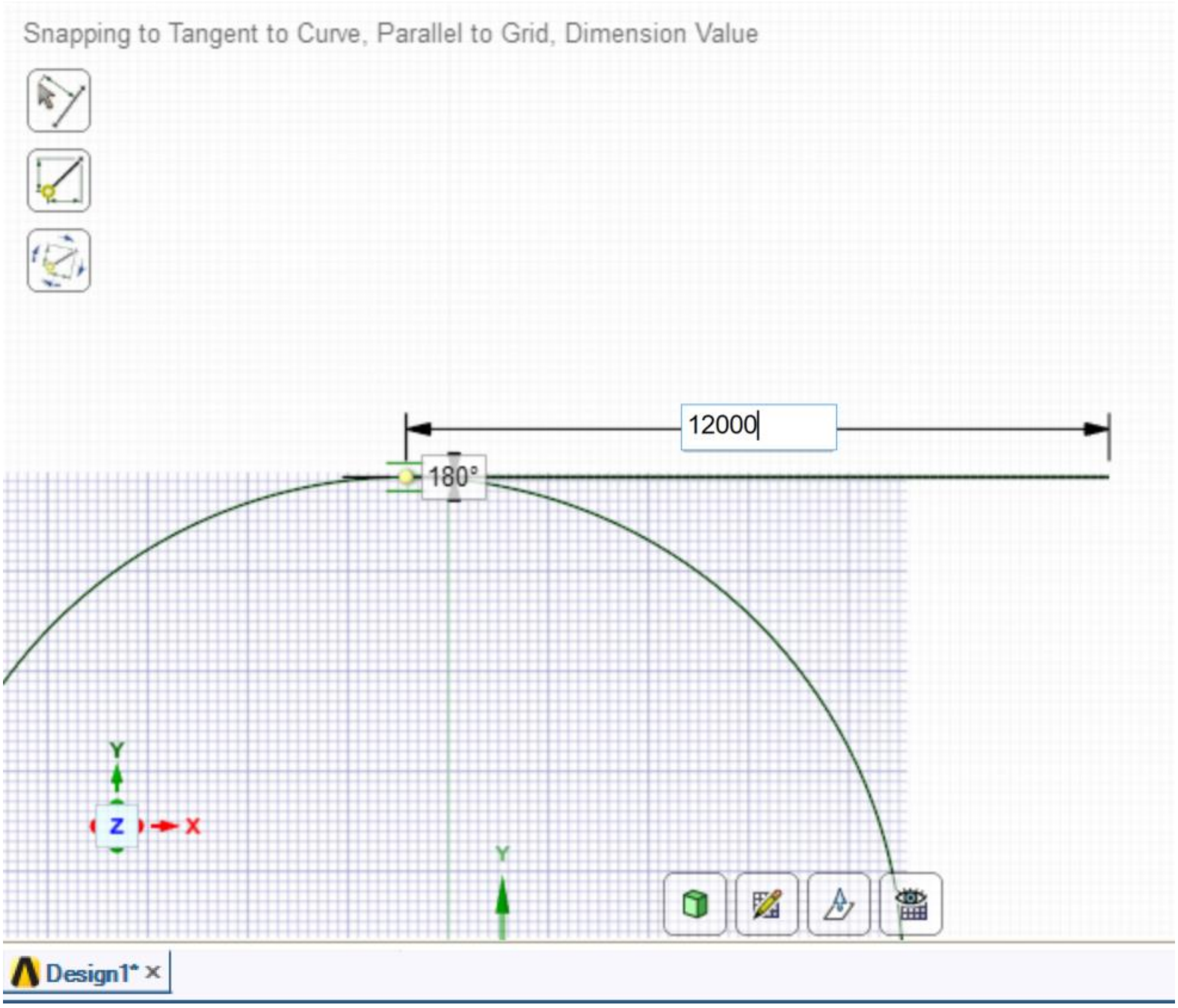


11) Draw a horizontal line 150 mm in length coming out of the tip of the profile

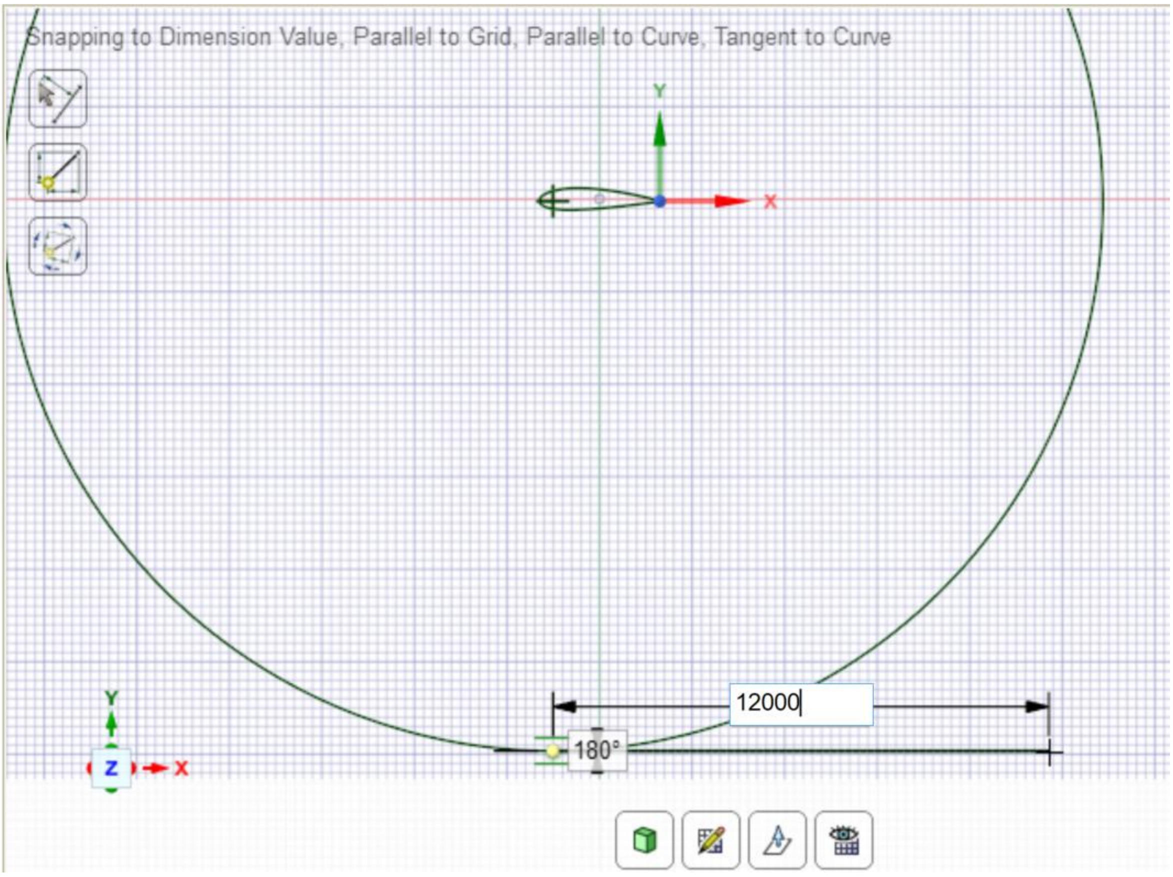


12) Draw a circle with a diameter of 12000 mm hooked at the right end of the line

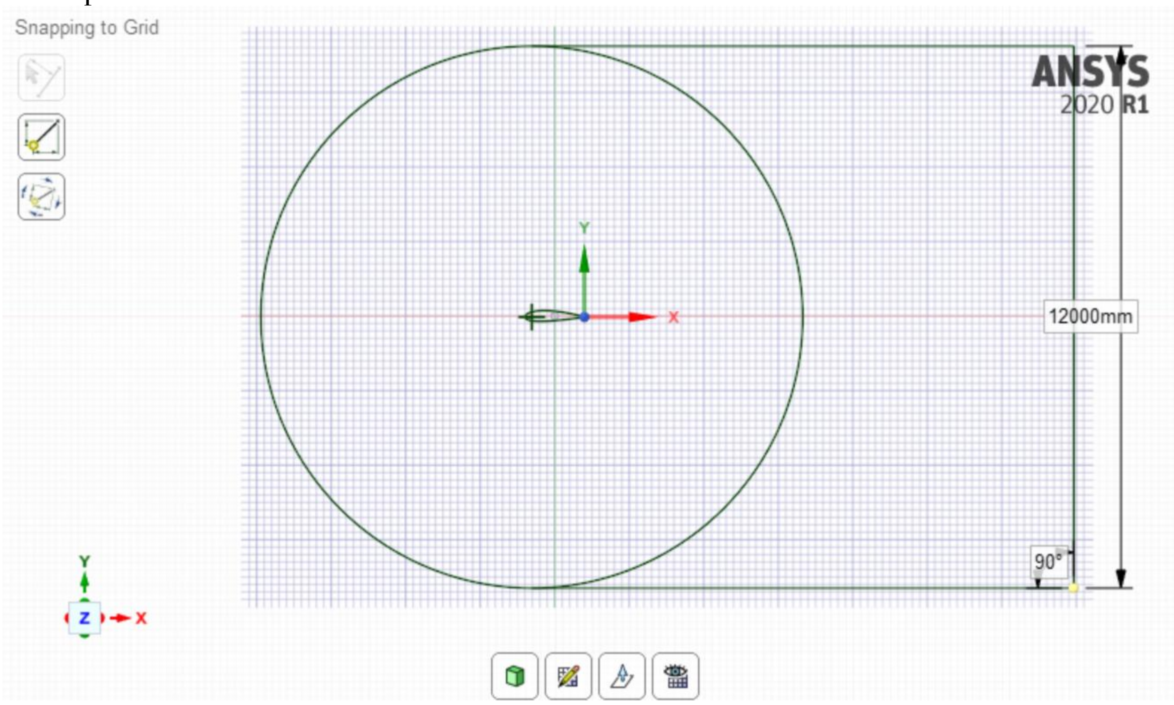
13) Draw a line 12000 mm long above the profile and tangent to the circle



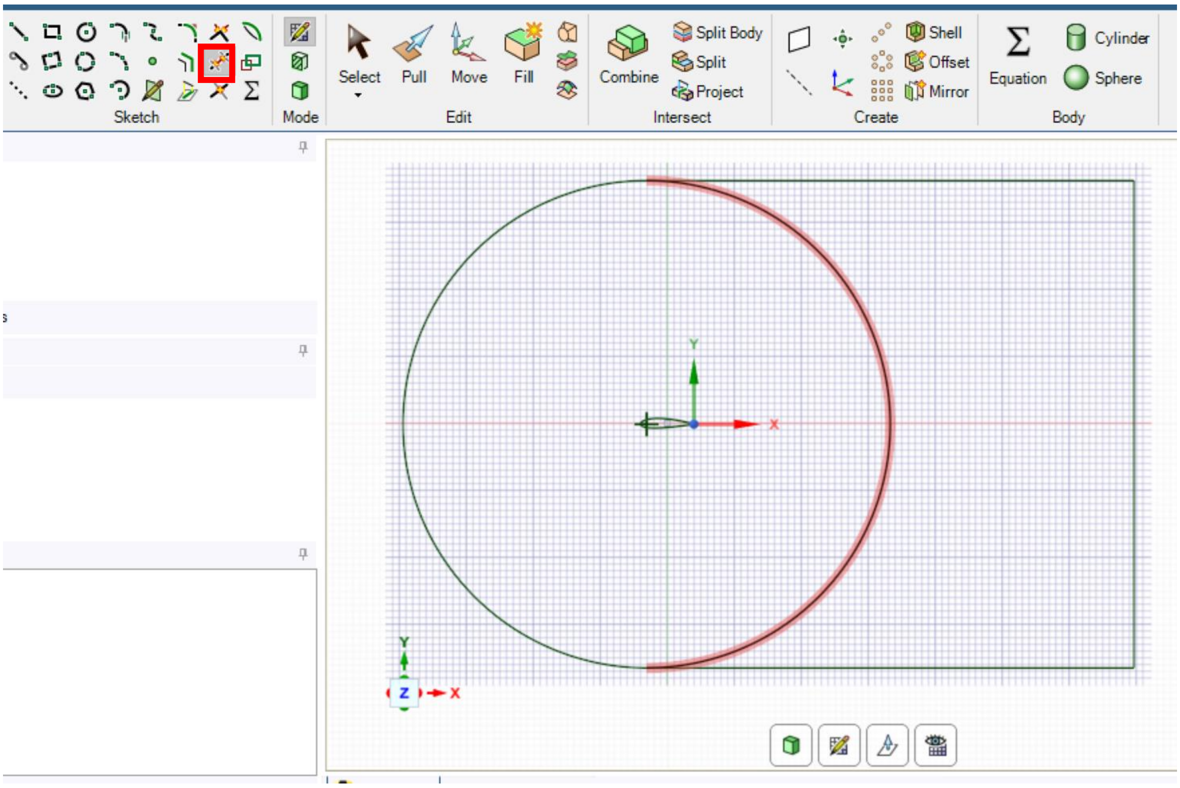
14) Create a similar line under the profile



15) Close profile

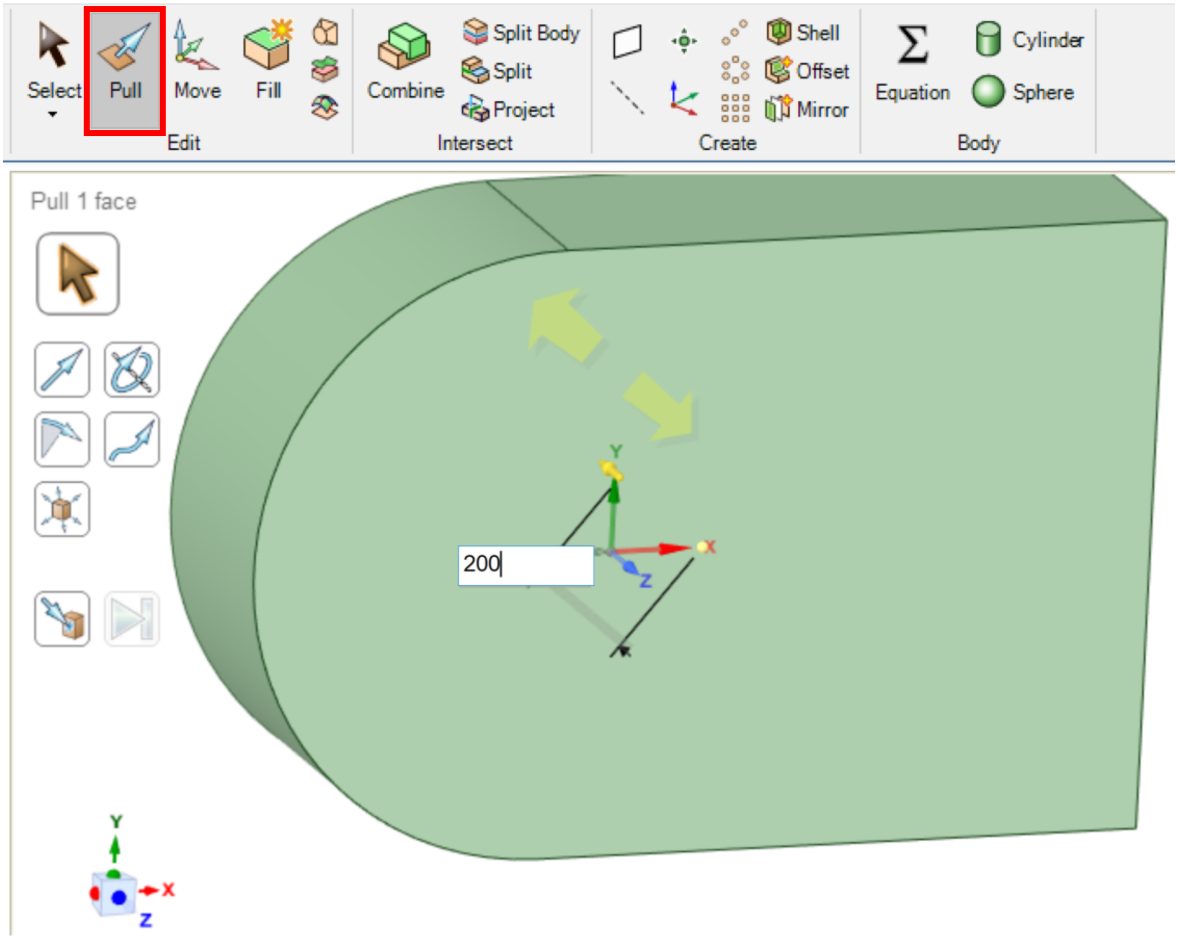


16) Select the *Trim away* icon and remove the red circle marked by pointing to it LMB

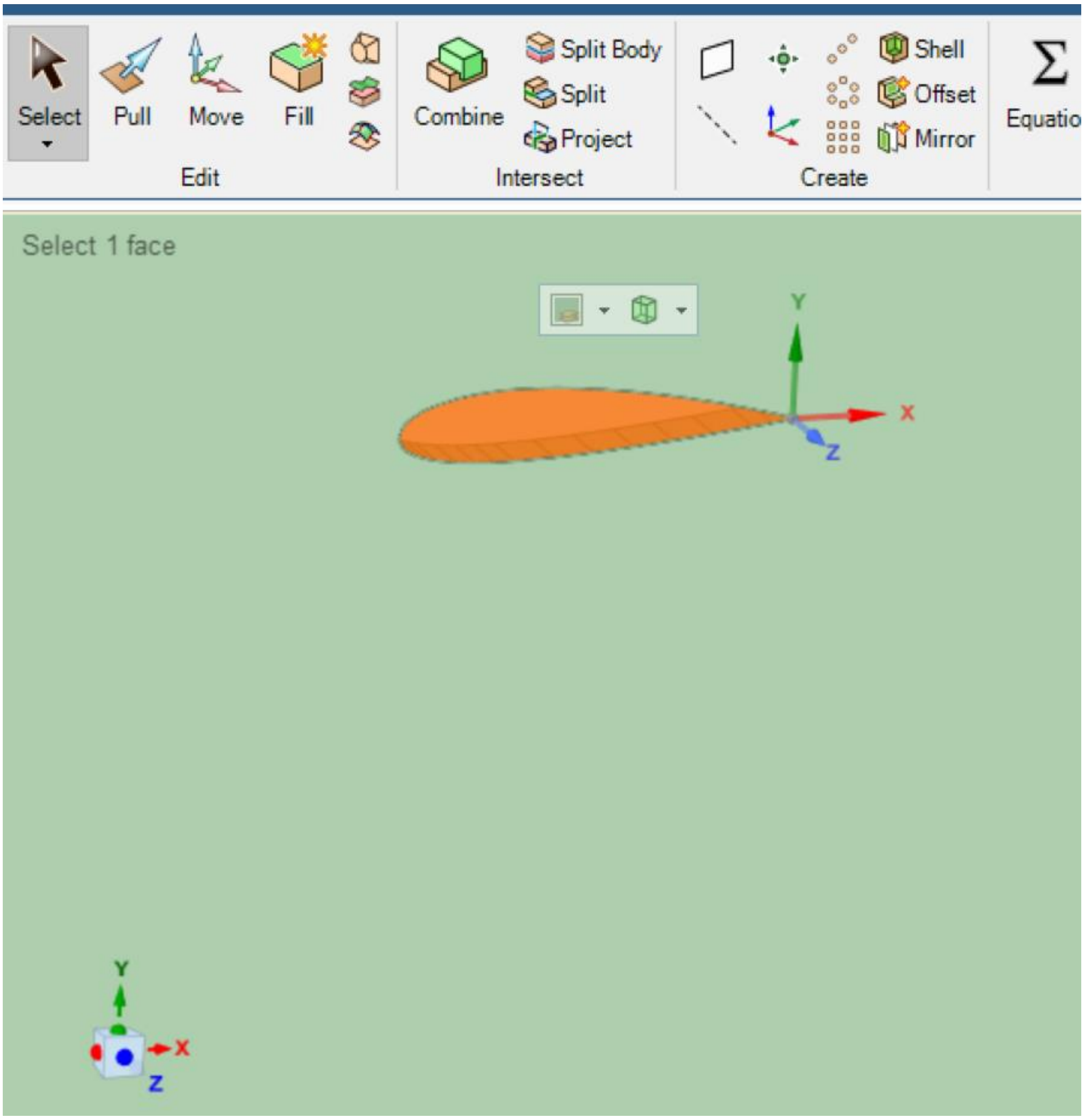


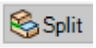
17) Point to LMB the line you no longer need inside the profile and press the *Delete* key

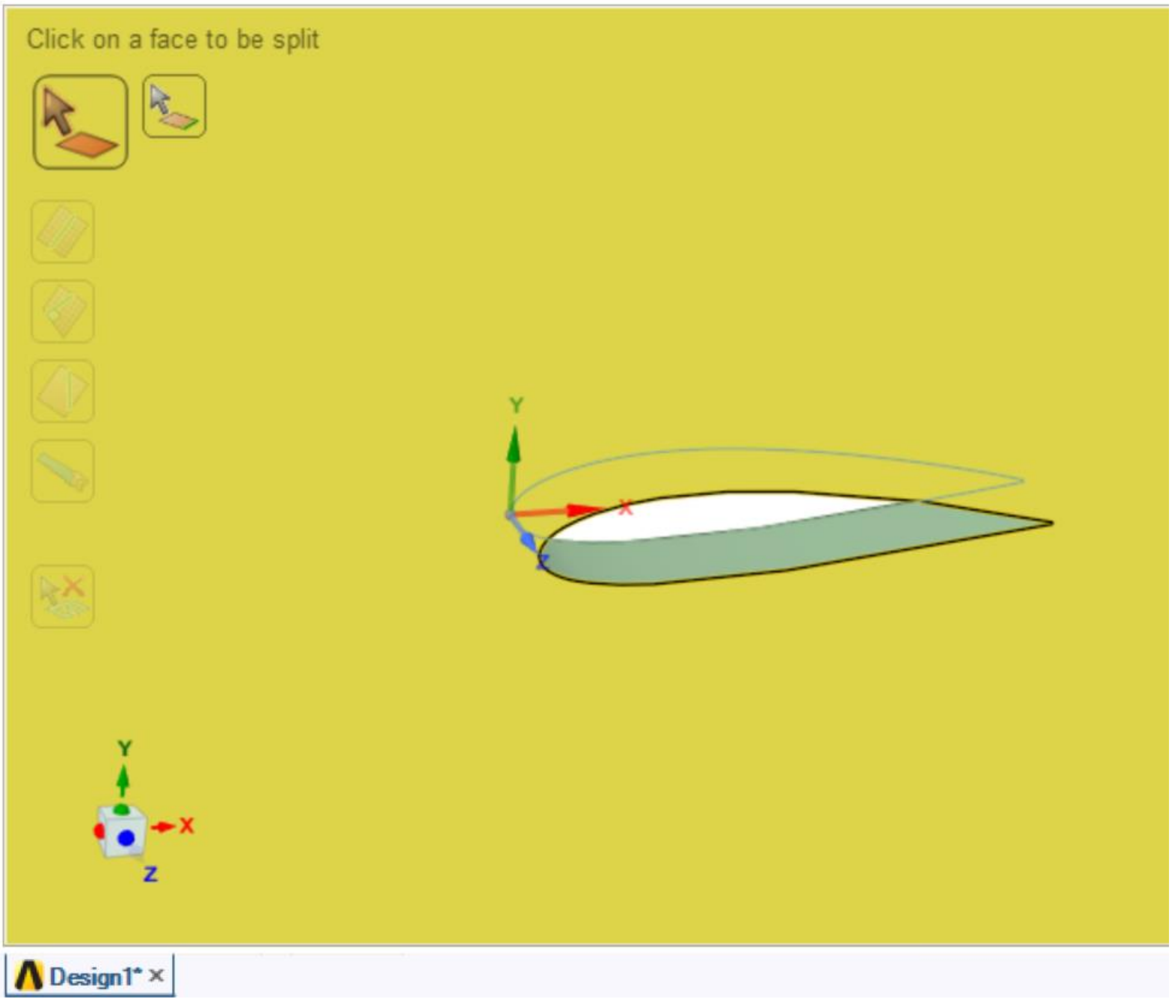
18) Return to the 3D view and select the *Pull* command, and then pull out the created profile by 200 mm.



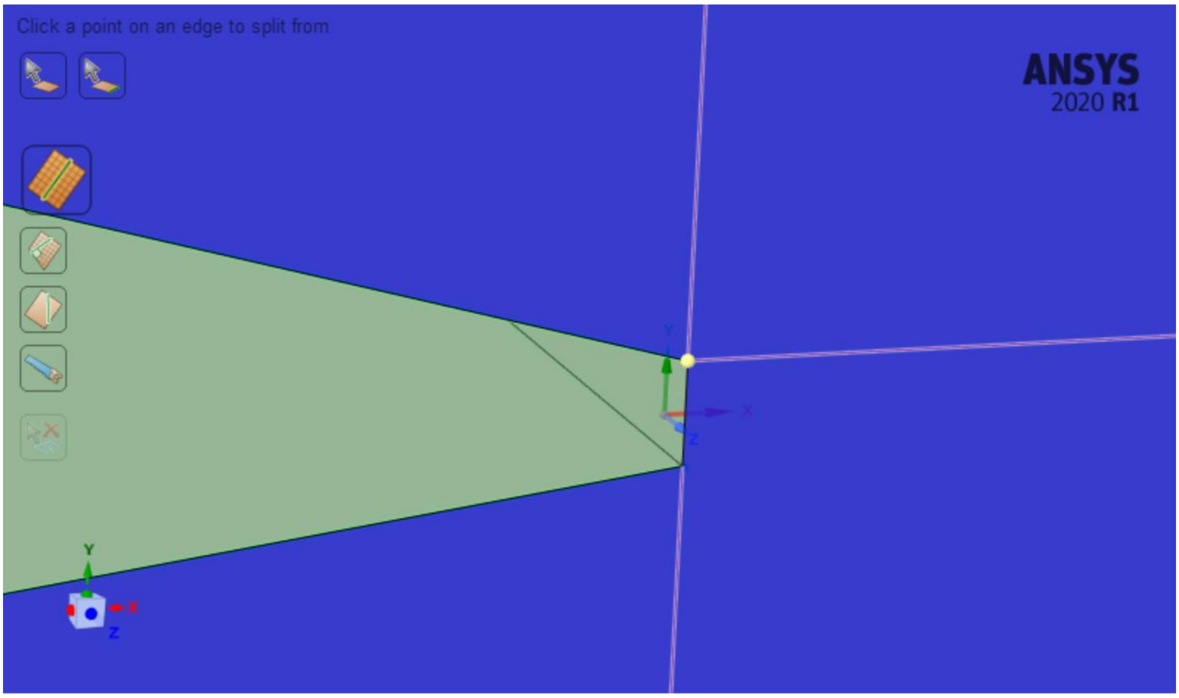
19) LMB point to the profile and press *Delete* to delete it.



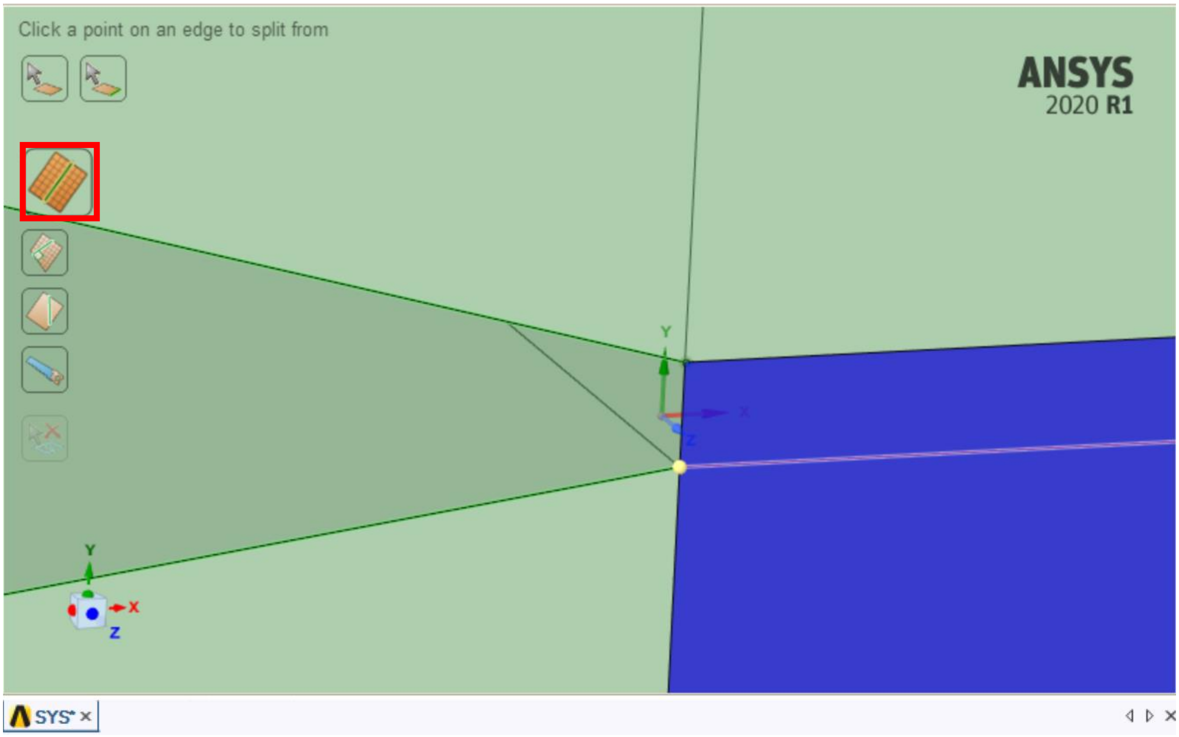
20) In the exercise, a hexagonal mesh will be created. To create it, you need lines that will designate individual grid "blocks". We will use the icon to create them *Split*  in *Design* tab. After selecting it, indicate a large, flat surface marked in yellow



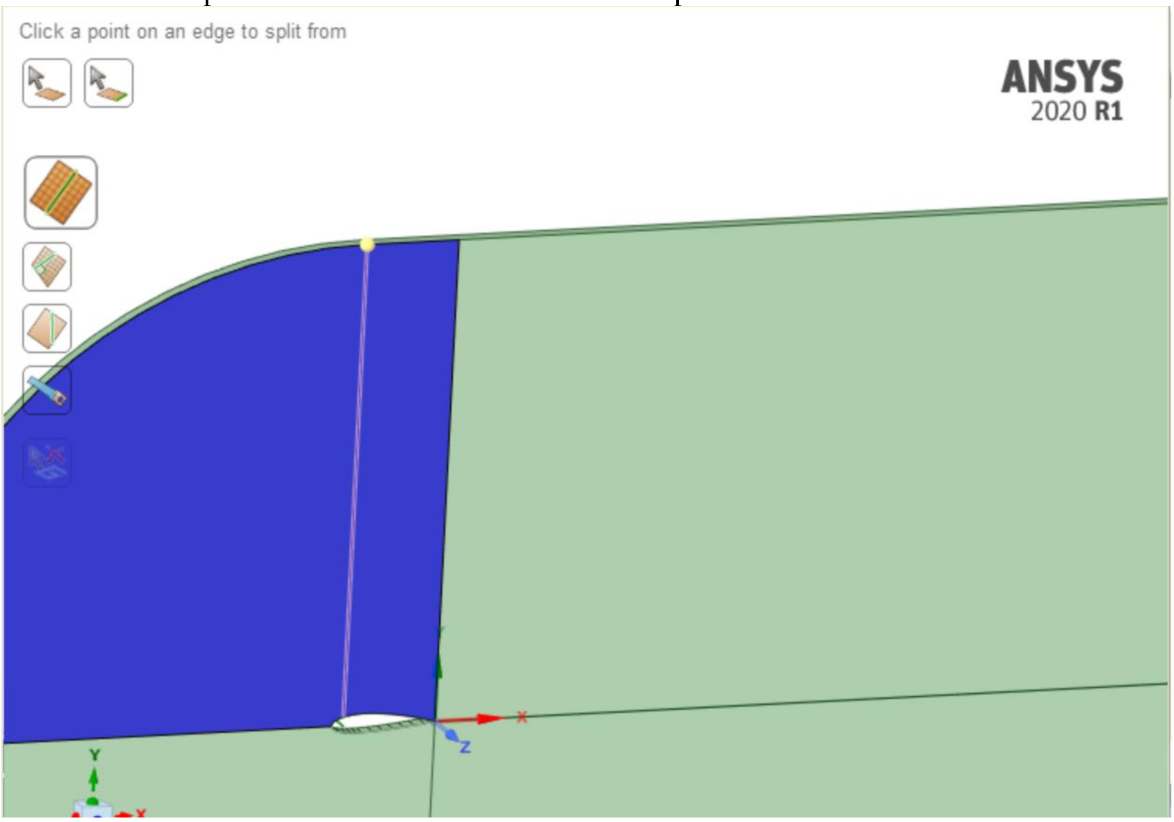
21) Move the cursor to the trailing edge and select the top vertex - an intersection is created



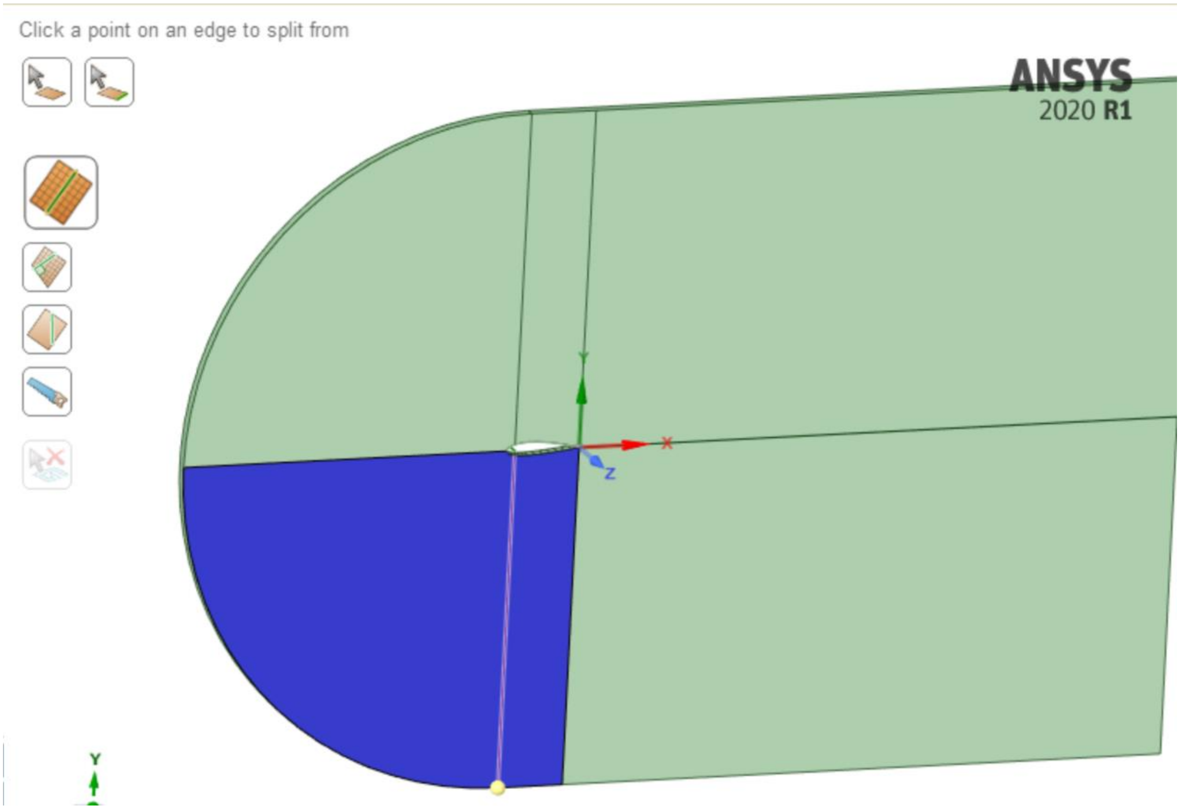
22) Do the same with the bottom vertex - if there is no blue background with a visible intersecting line, make sure that the option *Select UV cutter point* is selected



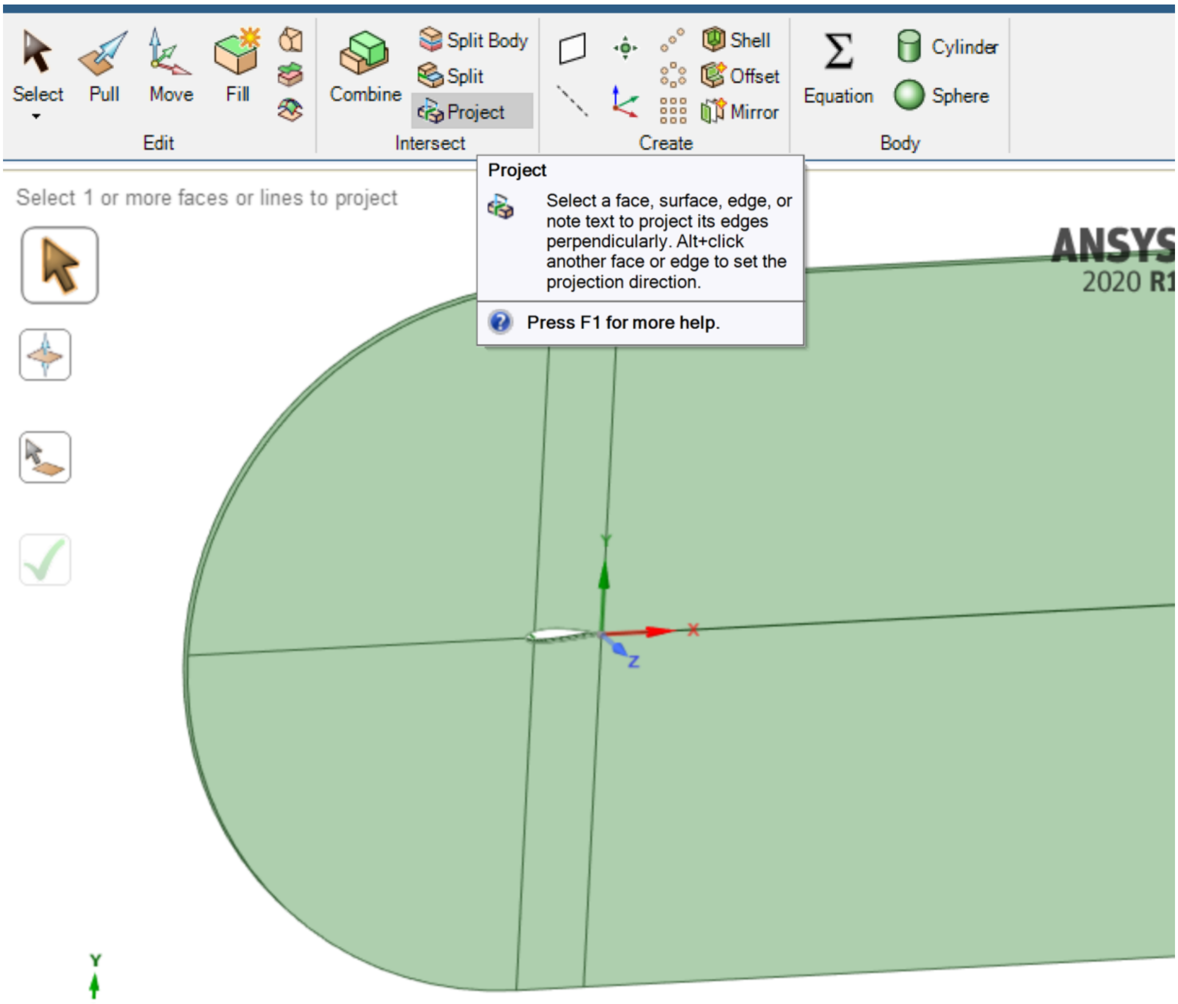
23) Then select the plane below and the vertex from the part of the circle



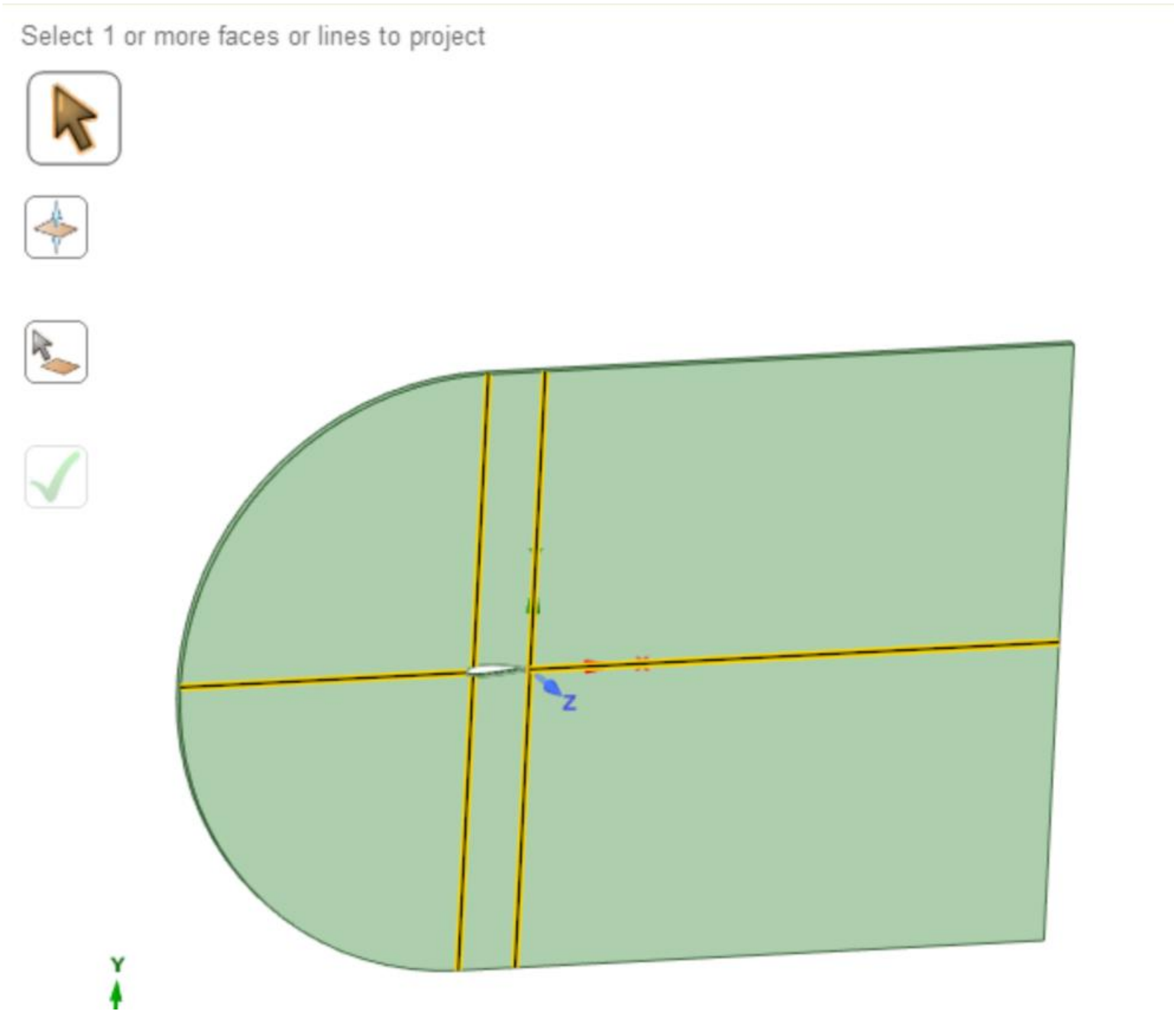
24) Similarly, create an intersection under the profile



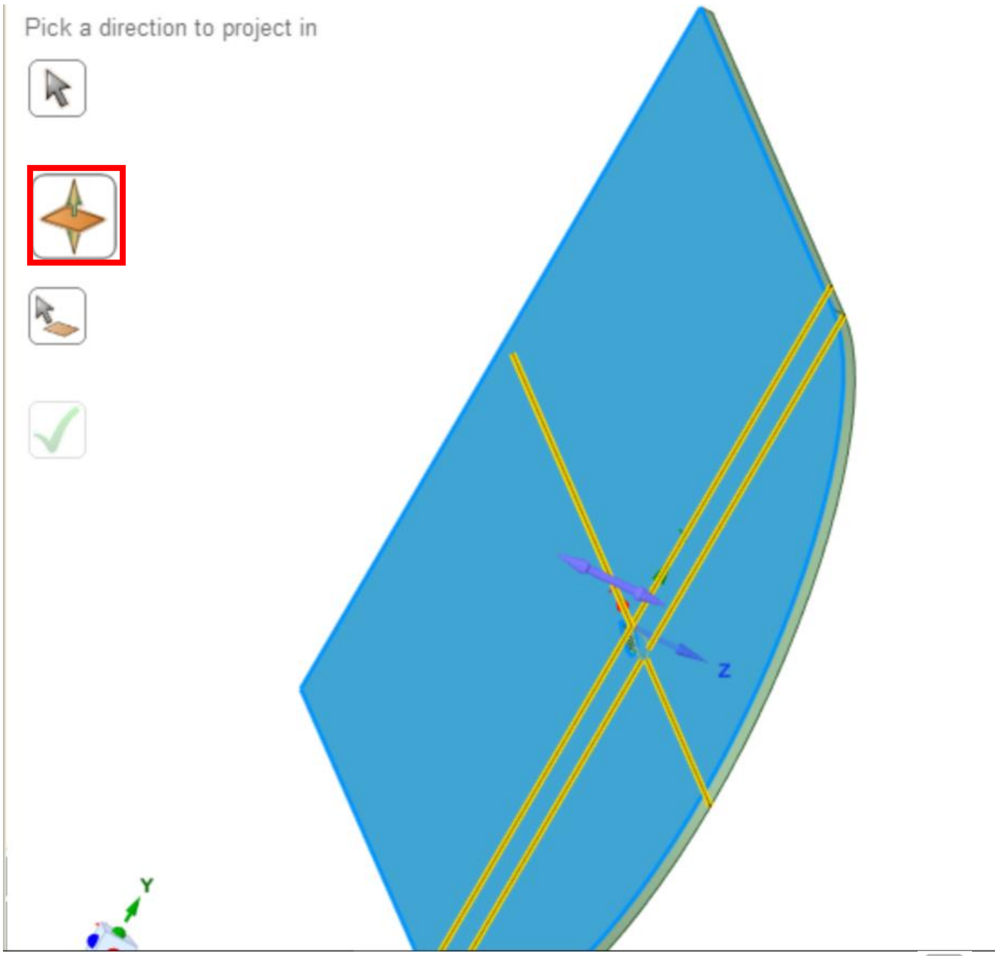
25) Similar cutting edges should be made on the other side of the geometry. This can be done by the previous method or the following method using the command *Project*



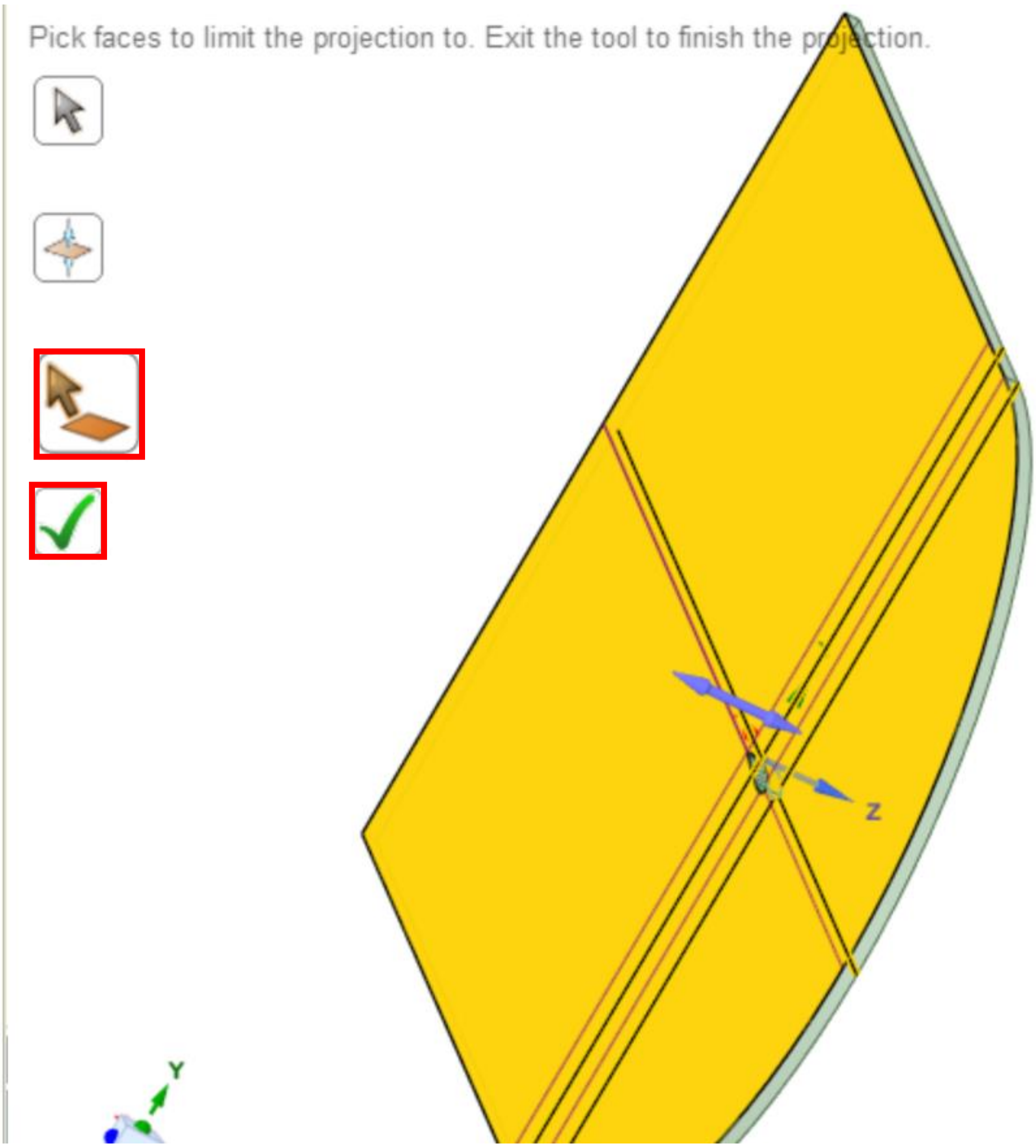
26) While holding down the *Ctrl* key select previously created edges



27) Select the projection direction by pointing to the flat surface



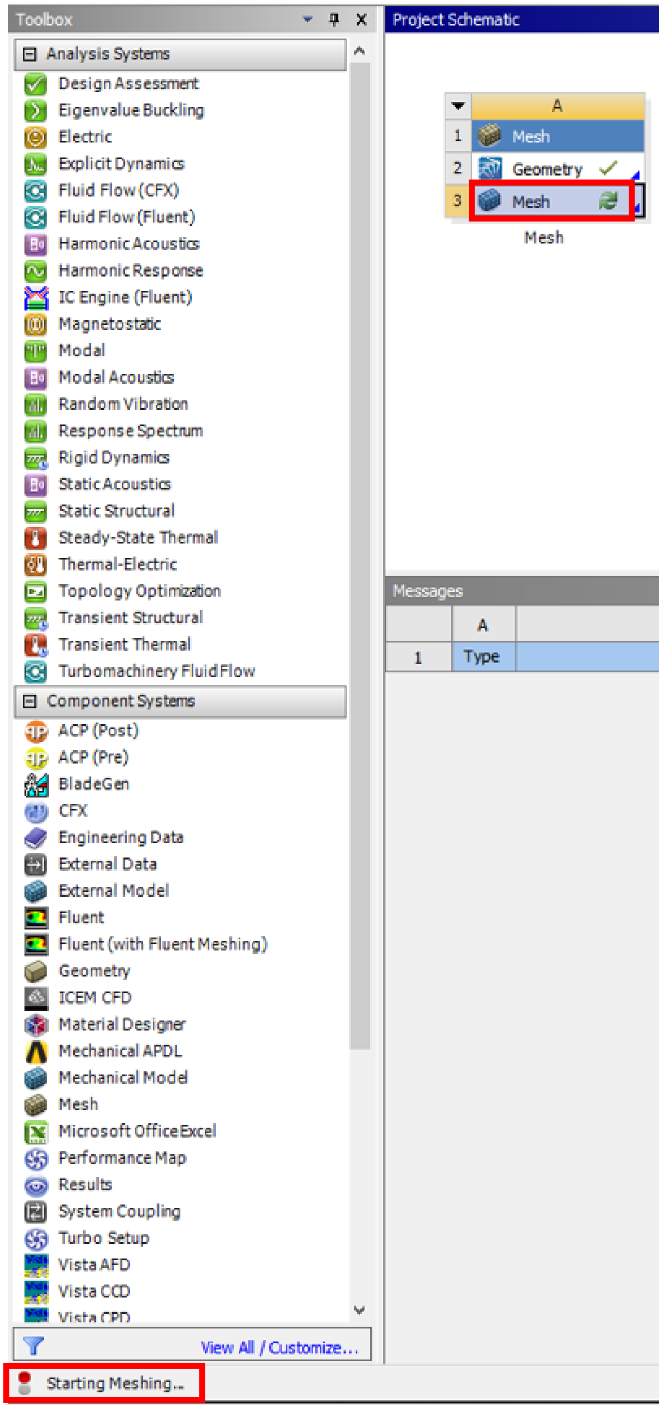
28) Select a large flat surface as the projection plane and confirm *Complete* 



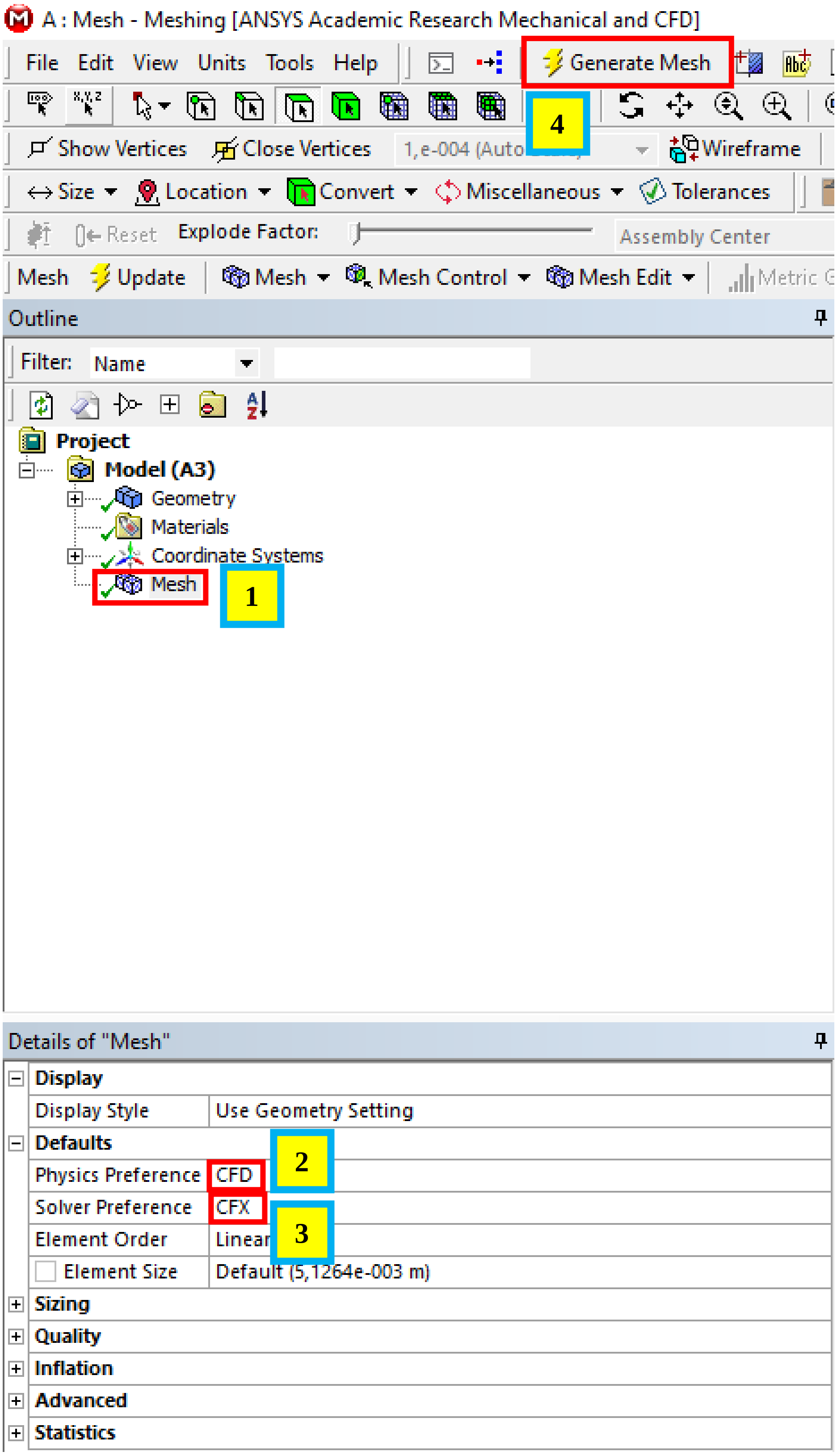
29) Close *Spaceclaim* and save project in *Workbench* using *Ctrl + s*

2.2. NUMERICAL MESH

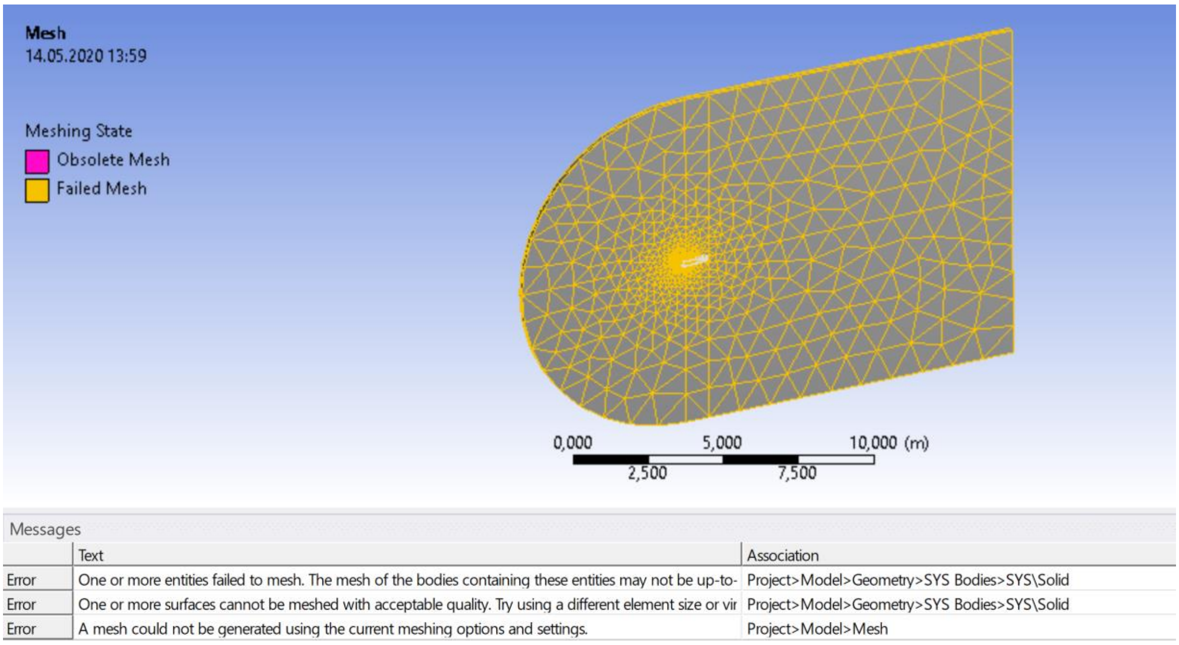
1) Open *Ansys Meshing* by double-click LMB *Mesh*



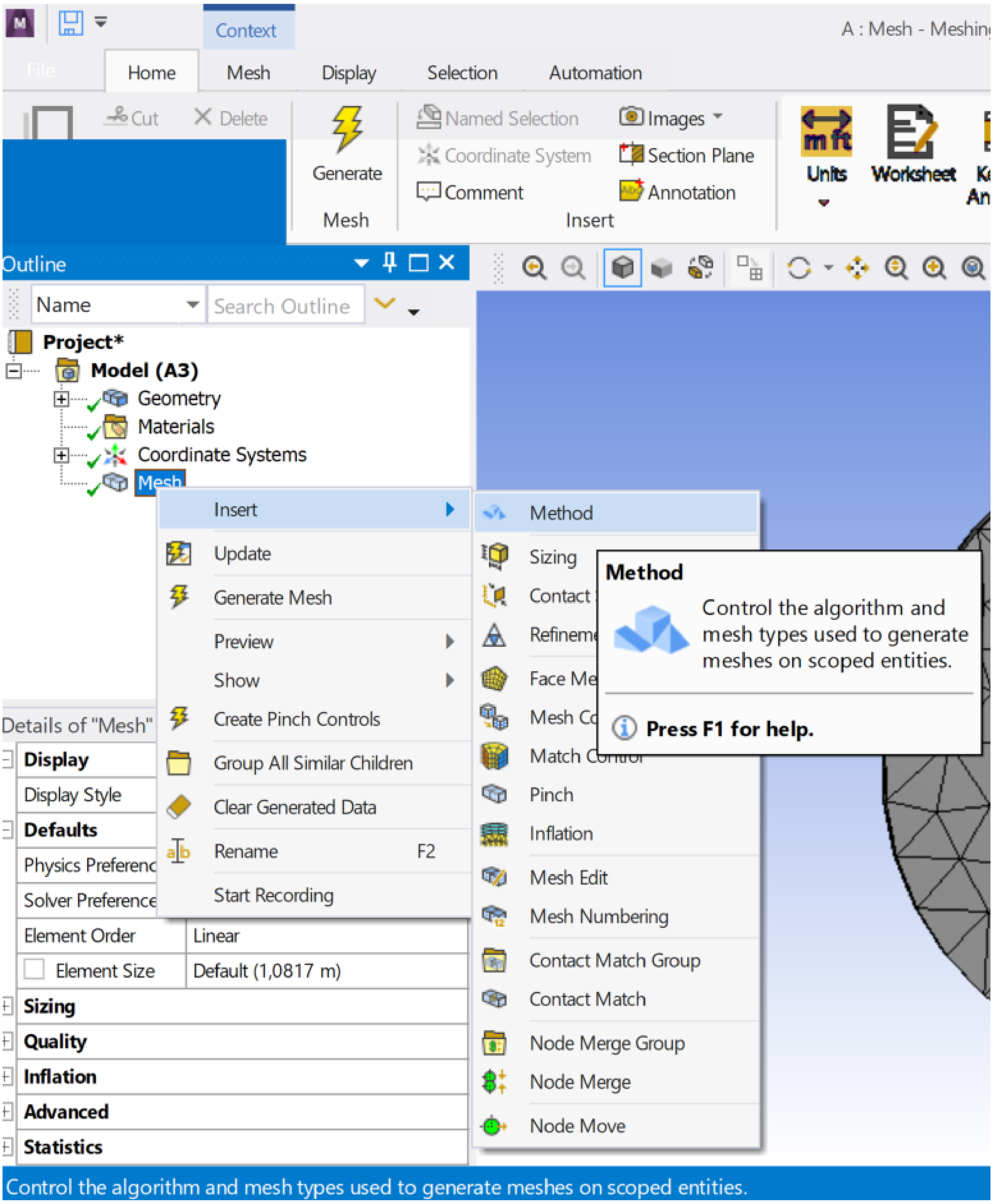
2) In *Ansys Meshing*: 1) click *Mesh*, 2) change *Physisc Preference* into *CFD*, 3) change *Solver Preference* into *CFX*, 4) click LMB *Generate Mesh*



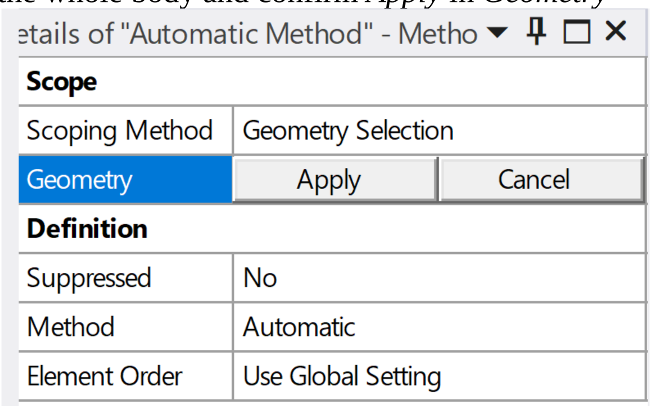
The default mesh is not valid. The mesh should be edited.



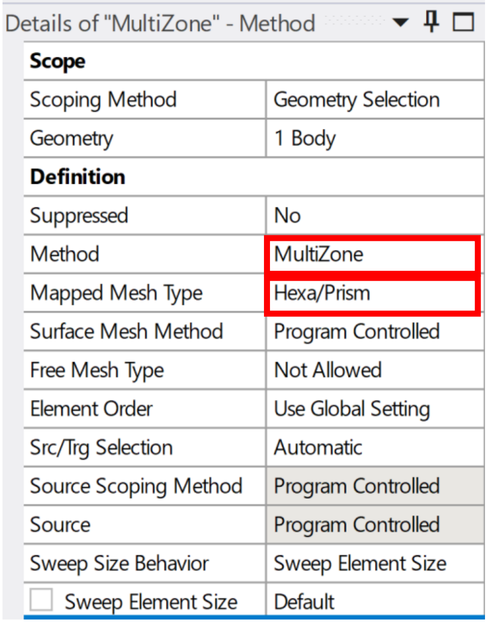
3) Click right mouse button (RMB) on *Mesh* and select *Insert->Method*



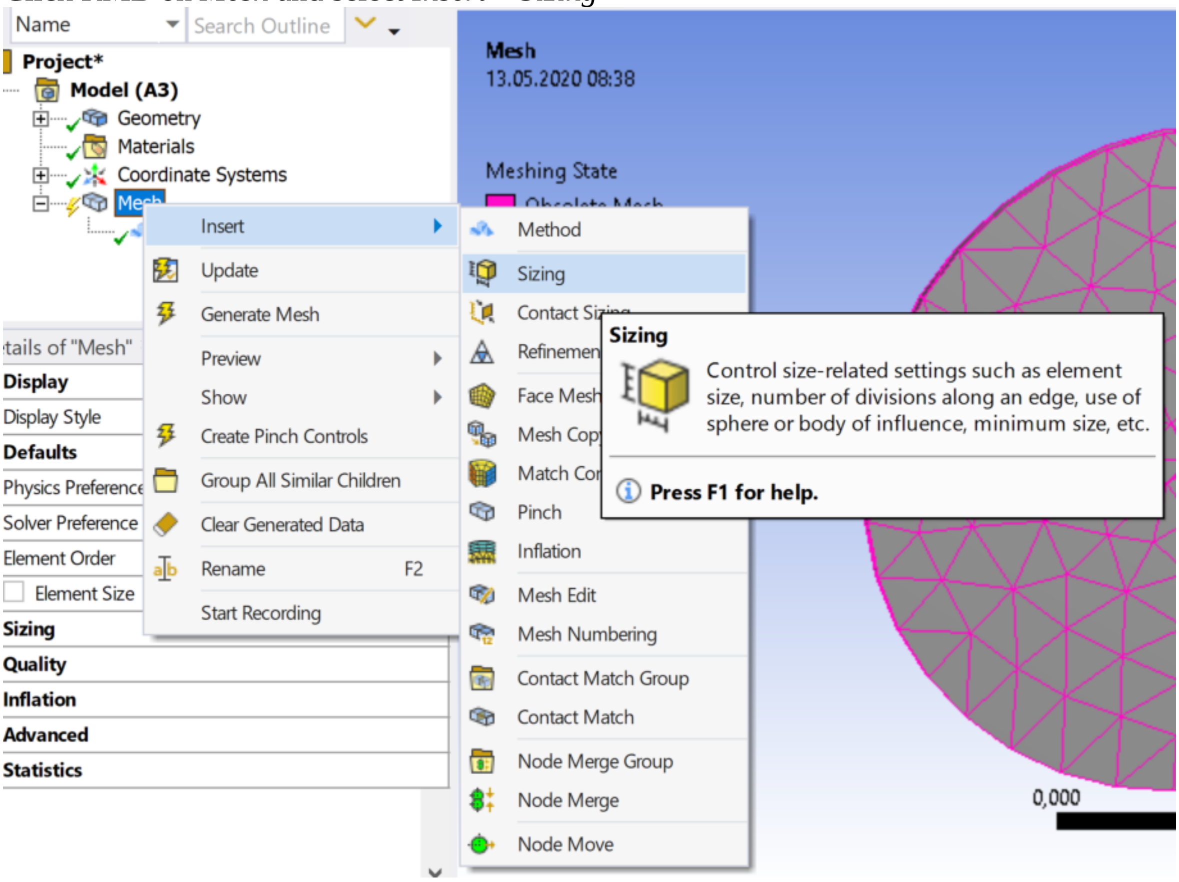
4) LMB indicate the whole body and confirm *Apply* in *Geometry*



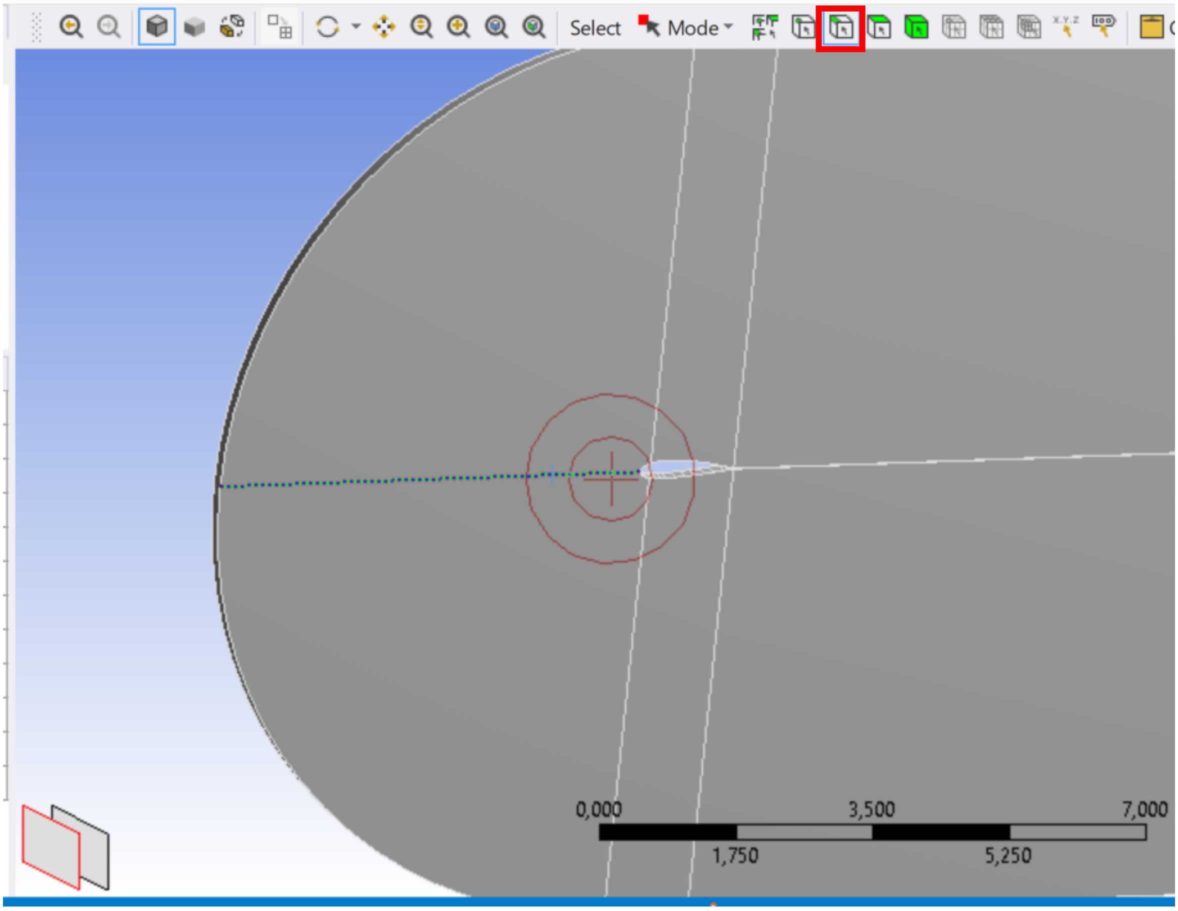
5) Apply the following settings



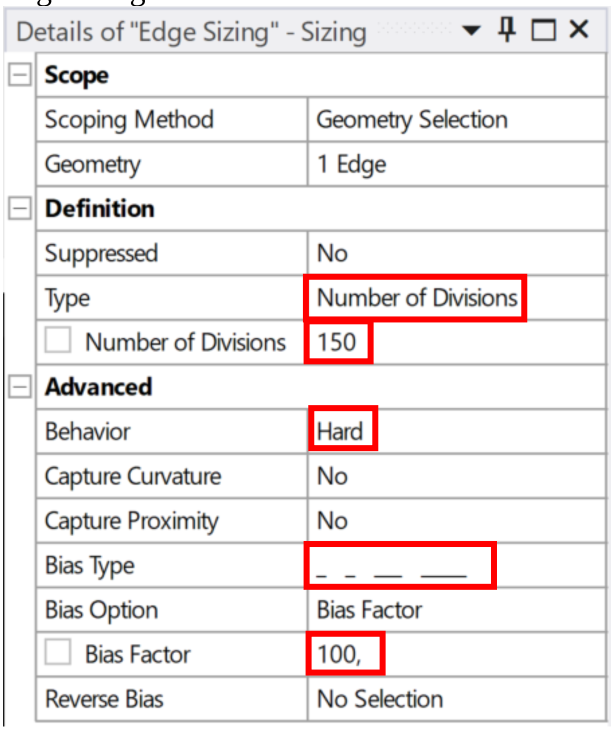
6) Click RMB on *Mesh* and select *Insert->Sizing*



7) Change the selection filter to edges (shortcut *Ctrl + E*) and point to the edge as shown below



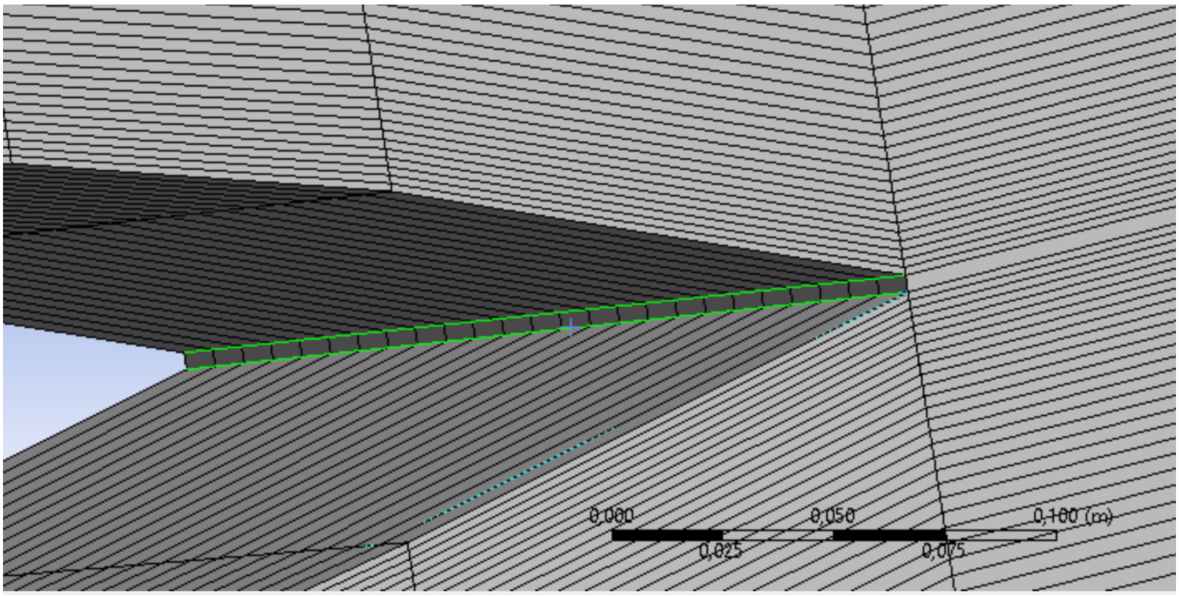
8) Apply the following settings



Details of "Edge Sizing" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge
Definition	
Suppressed	No
Type	Number of Divisions
Number of Divisions	150
Advanced	
Behavior	Hard
Capture Curvature	No
Capture Proximity	No
Bias Type	_ _ — ——
Bias Option	Bias Factor
Bias Factor	100,
Reverse Bias	No Selection

9) Similarly, insert *Sizing* for the edges at the trailing edge shown below

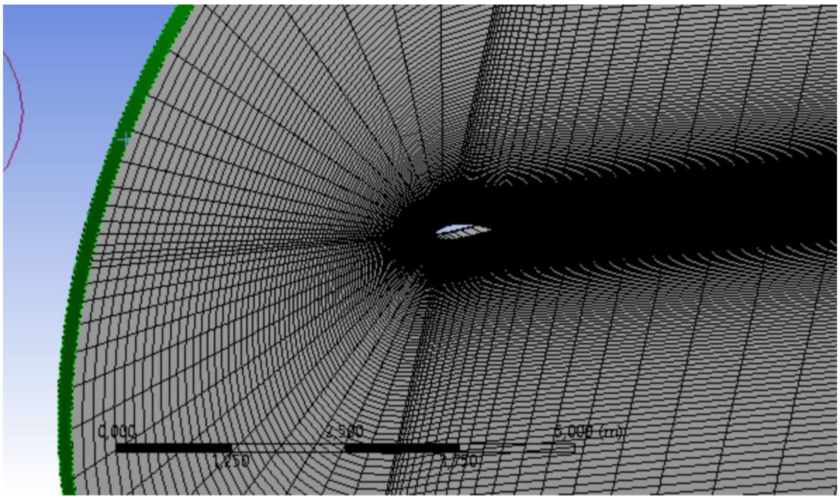


10) Apply the following settings

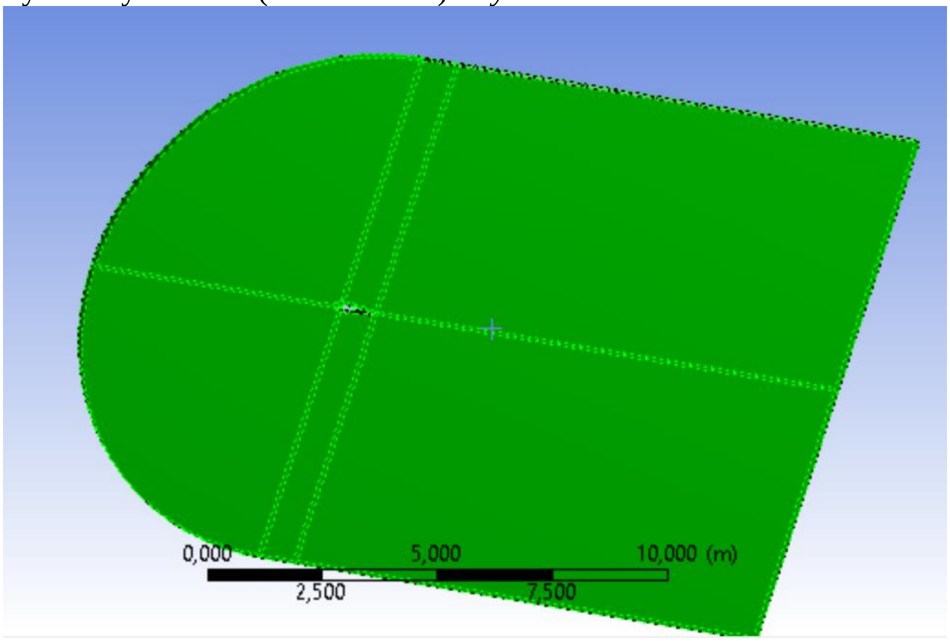
Scope	
Scoping Method	Geometry Selection
Geometry	2 Edges
Definition	
Suppressed	No
Type	Number of Divisions
Number of Divisions	1
Advanced	
Behavior	Hard
Capture Curvature	No
Capture Proximity	No
Bias Type	No Bias

And press *Generate Mesh.*

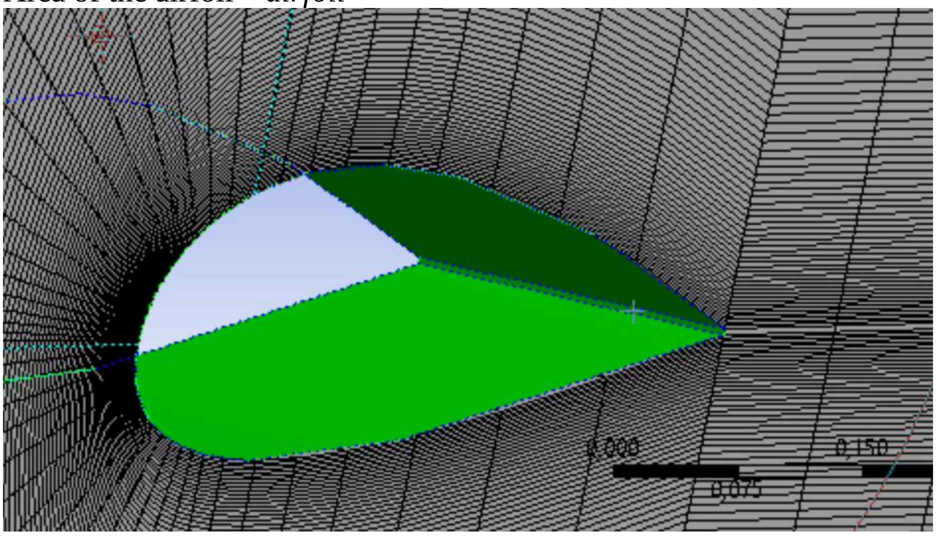
11) The last step is to name the volumes and surfaces. Give the following names using *Create Named Selection* (if you do not remember how to use *Create Named Selection*, check the previous instructions):
    a. Whole volume – *Fluid_domain*
    b. Inlet surface (cylindrical surface) – *inlet*



c. Symmetry surfaces (on both sides) – *sym*



d. Area of the airfoil – *airfoil*

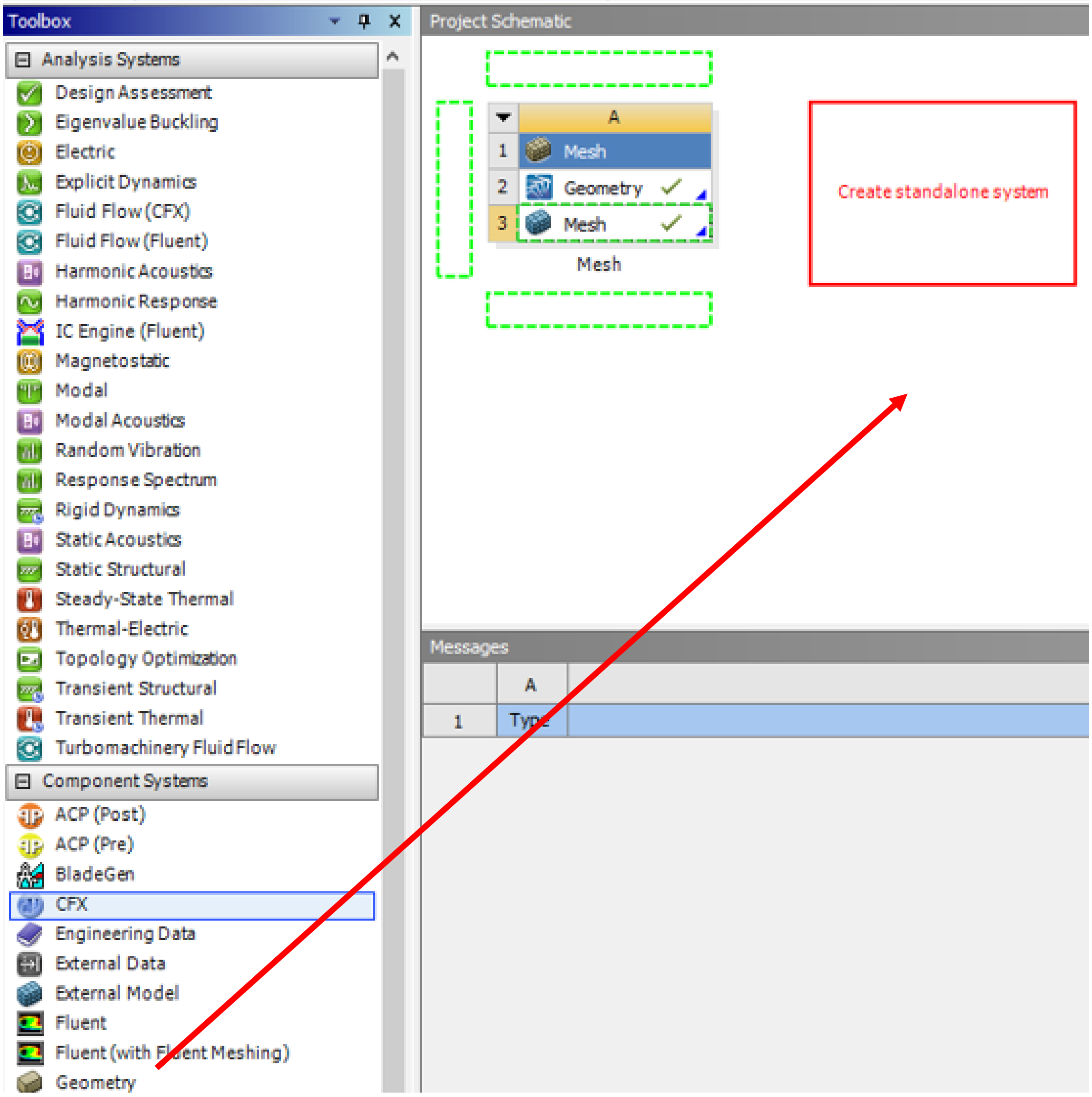


e. Outlet surface (other surfaces) – *outlet*

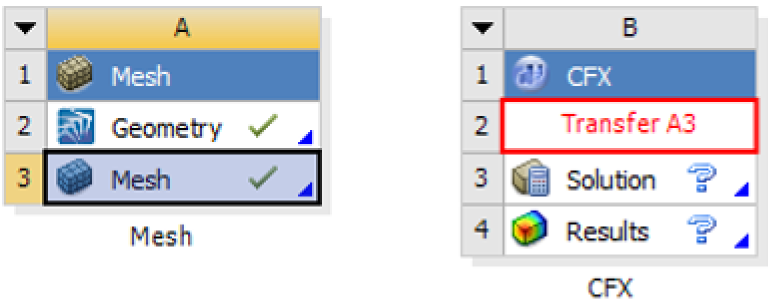
12) Close *Ansys Meshing* and save project in *Workbench.*

2.3. NUMERICAL MODEL

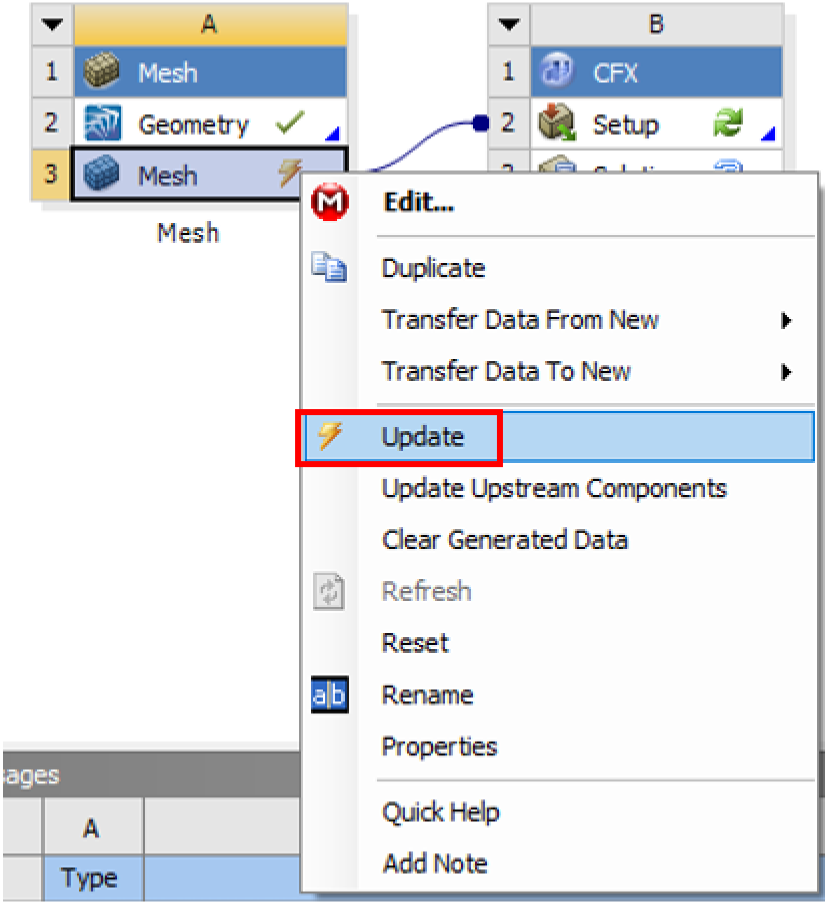
1) Insert *CFX*



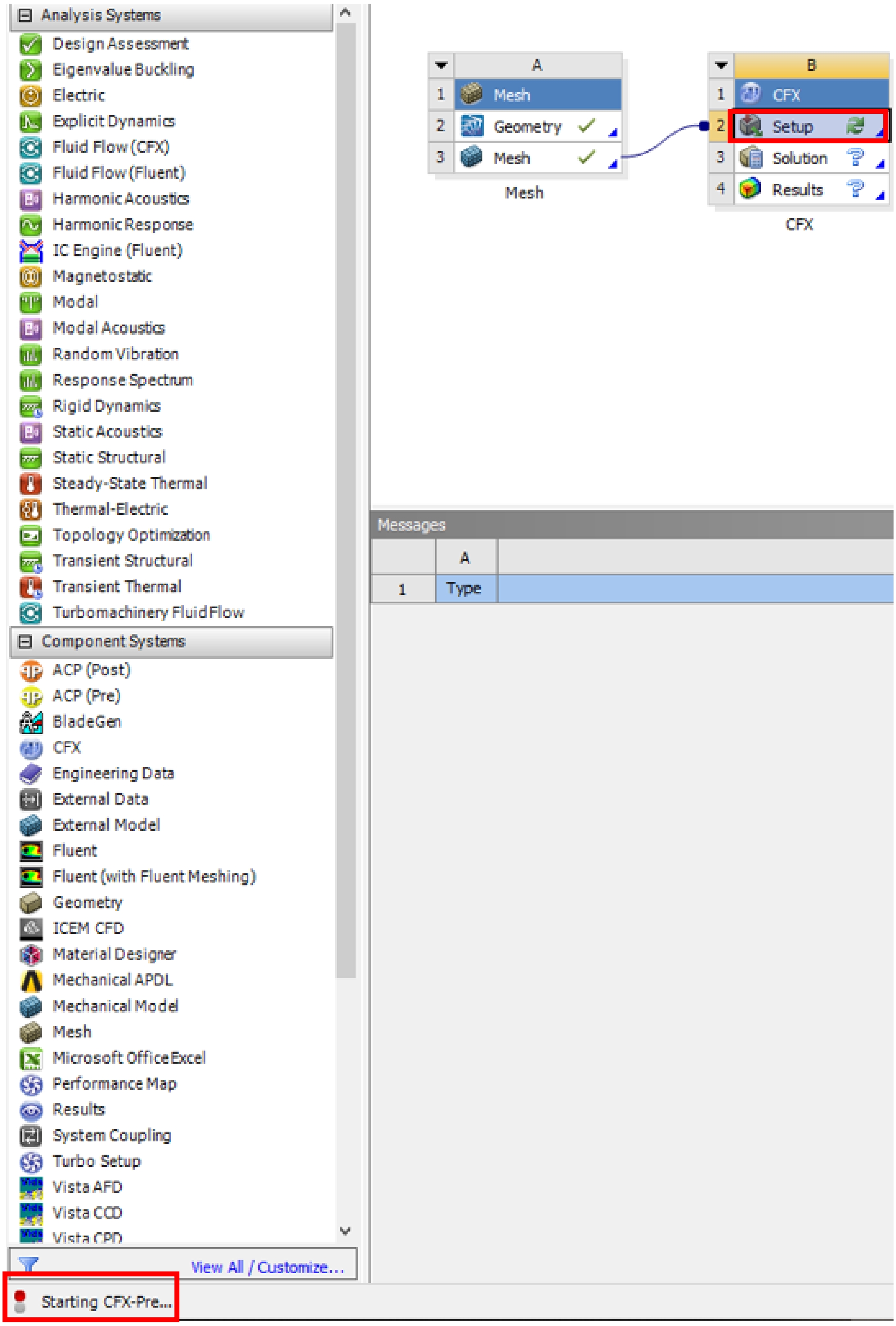
To connect the *Mesh* module with CFX, grab the LMB *Mesh* (below) and drag it to *Setup* until the *Transfer A3* box appears, and then release the LMB - the connection has been created



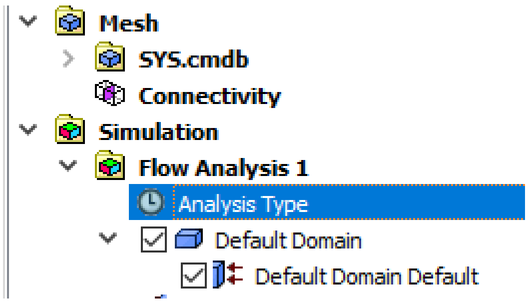
Click RMB on *Mesh* and choose *Update*



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

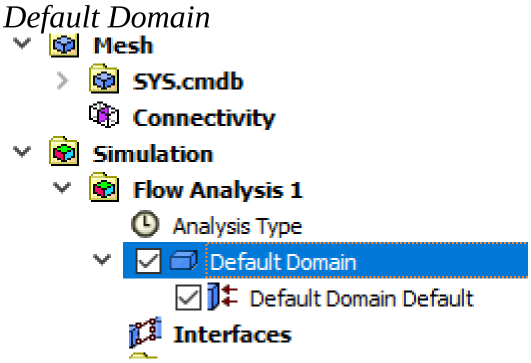


2) Open *Analysis Type* by double-click LMB

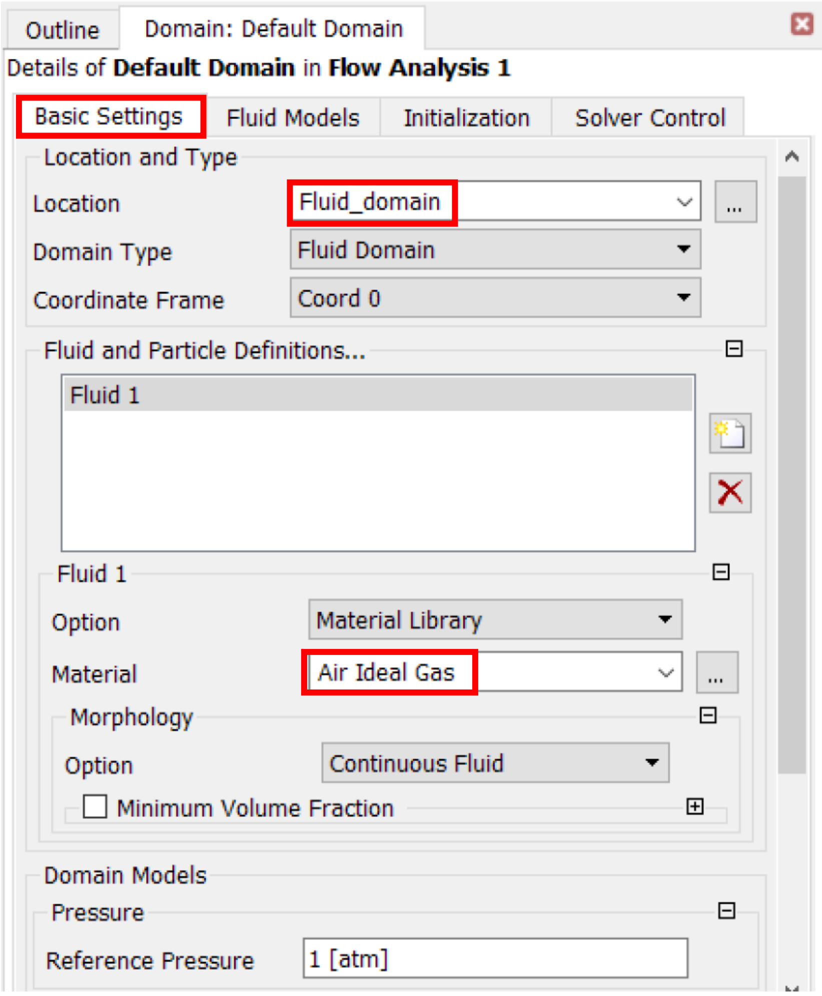


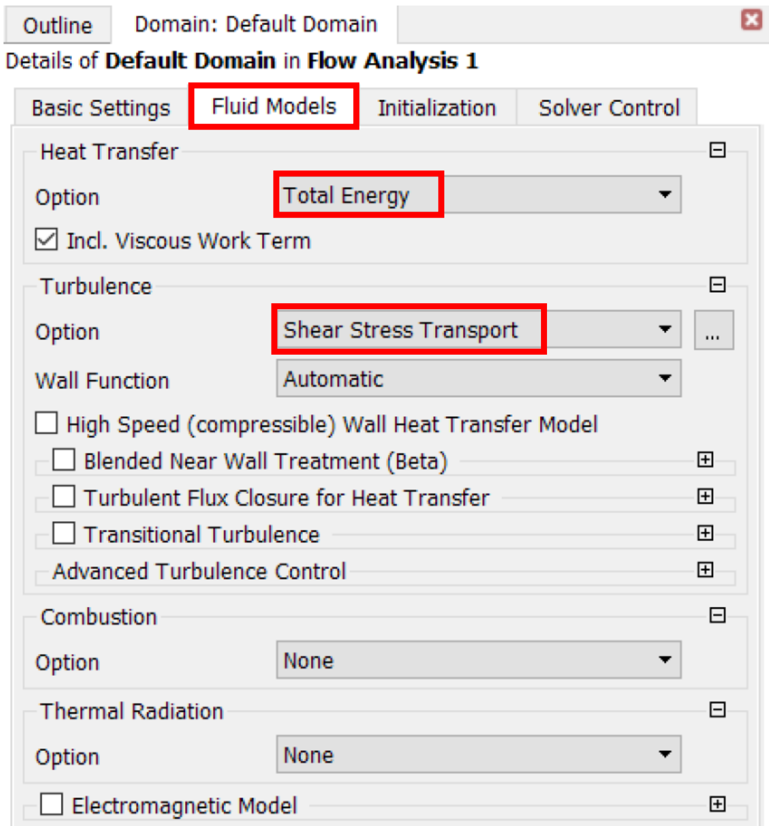
Apply the following settings and confirm *OK*.

3) Double-click LMB *Default Domain*

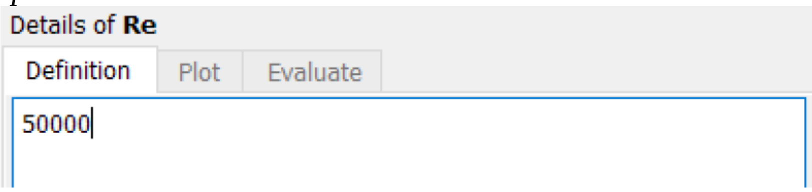


Apply the following settings

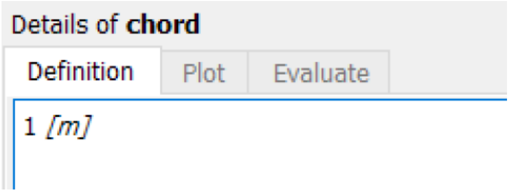

Outline
Domain: Default Domain
Details of Default Domain in Flow Analysis 1
Basic Settings
Fluid Models
Initialization
Solver Control
Location and Type
Location
Fluid_domain
Domain Type
Fluid Domain
Coordinate Frame
Coord 0
Fluid and Particle Definitions...
Fluid 1
Fluid 1
Option
Material Library
Material
Air Ideal Gas
Morphology
Option
Continuous Fluid
Minimum Volume Fraction
Domain Models
Pressure
Reference Pressure
1 [atm]


Outline
Domain: Default Domain
Details of Default Domain in Flow Analysis 1
Basic Settings
Fluid Models
Initialization
Solver Control
Heat Transfer
Option
Total Energy
Incl. Viscous Work Term
Turbulence
Option
Shear Stress Transport
Wall Function
Automatic
High Speed (compressible) Wall Heat Transfer Model
Blended Near Wall Treatment (Beta)
Turbulent Flux Closure for Heat Transfer
Transitional Turbulence
Advanced Turbulence Control
Combustion
Option
None
Thermal Radiation
Option
None
Electromagnetic Model

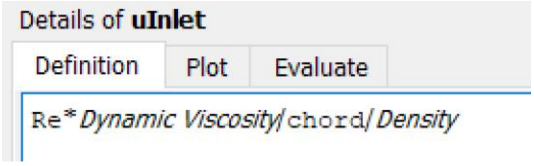
4) Create *expression* named *Re* with a definition



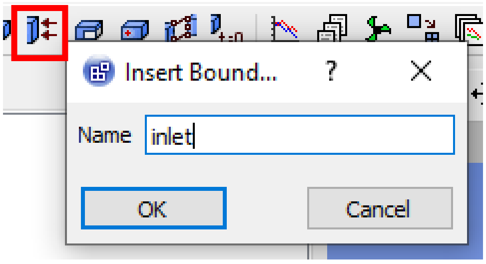
5) Create *expression* named *chord* with a definition



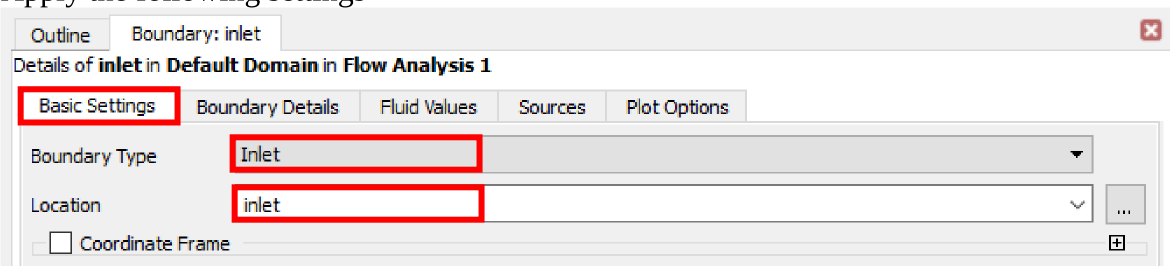
6) Create *expression* named *uInlet* with a definition

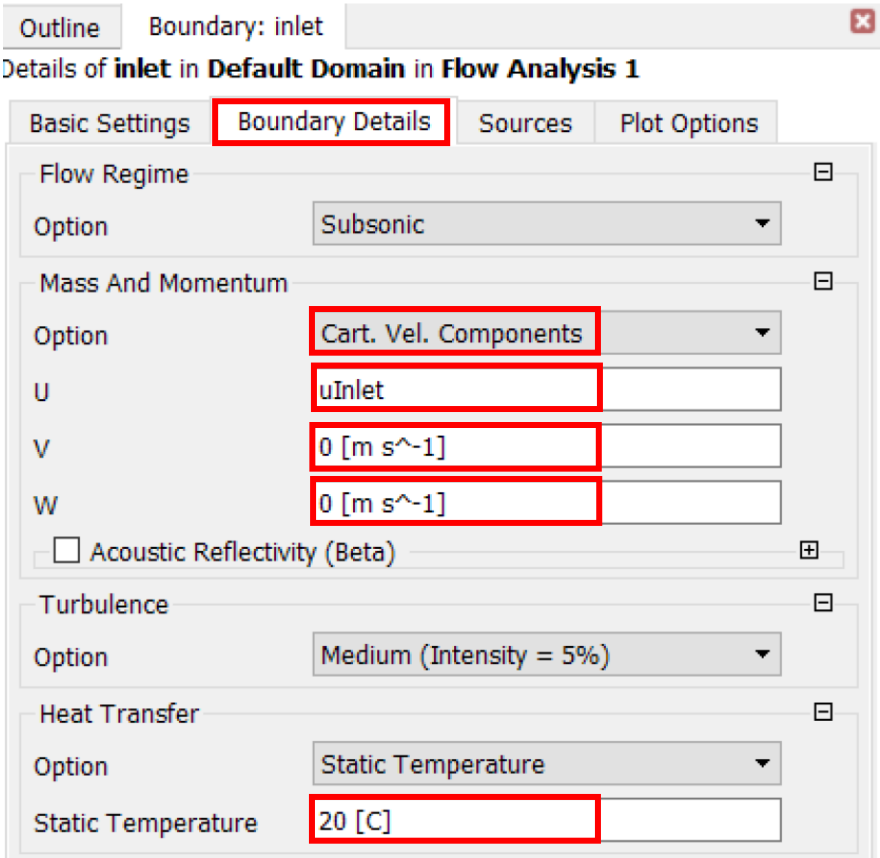


7) Create boundary condition *inlet*

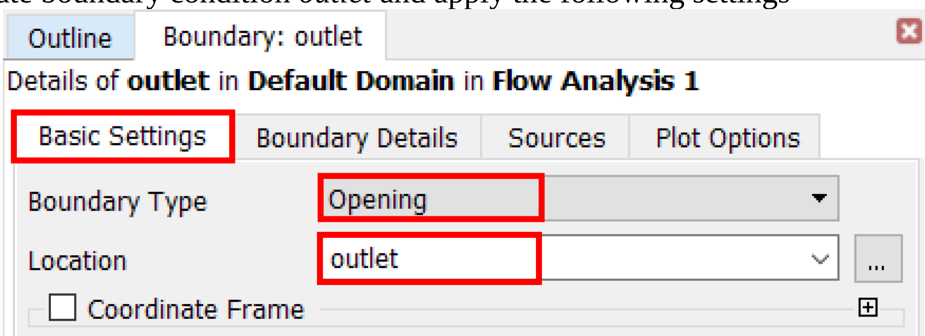


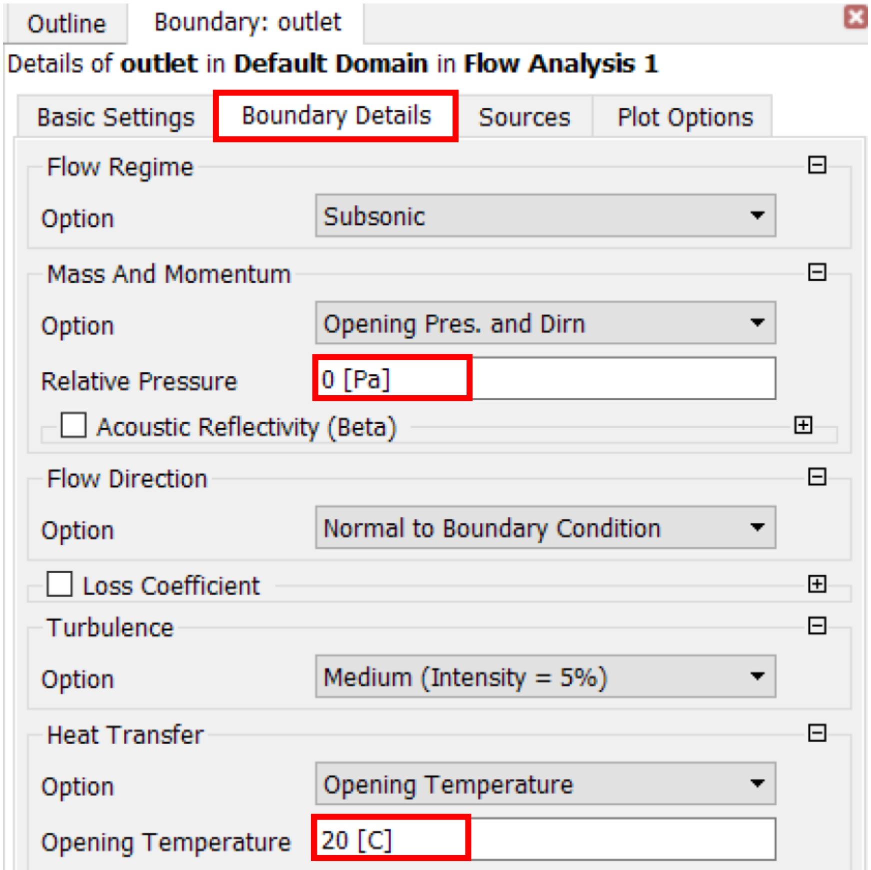
8) Apply the following settings



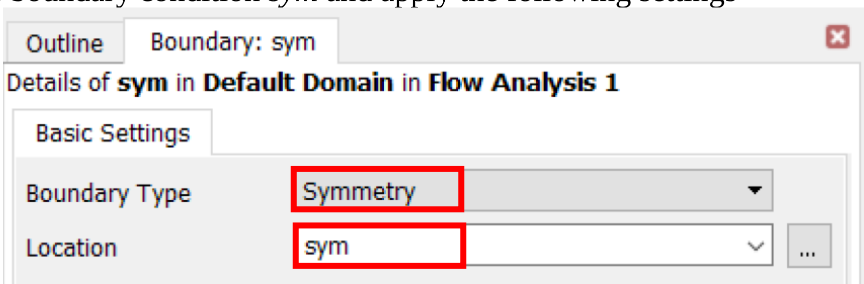


9) Create boundary condition *outlet* and apply the following settings

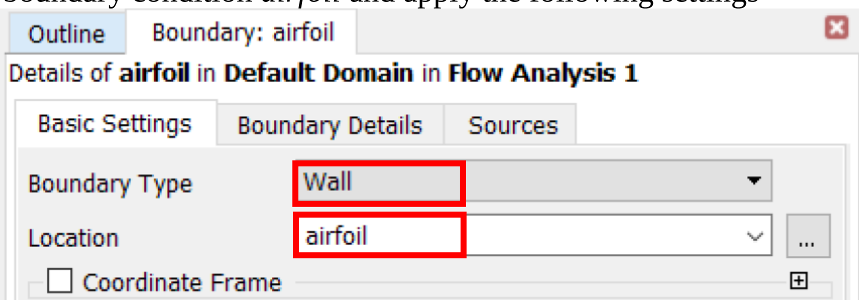


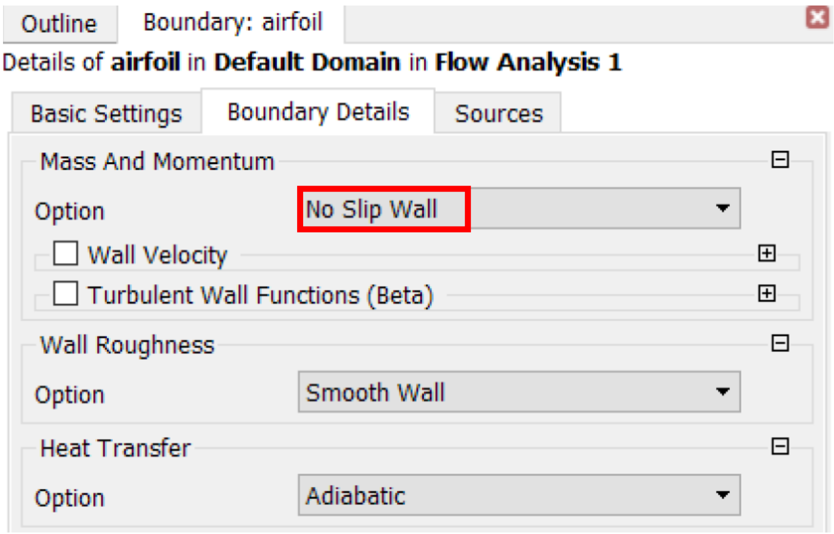


10) Create boundary condition *sym* and apply the following settings

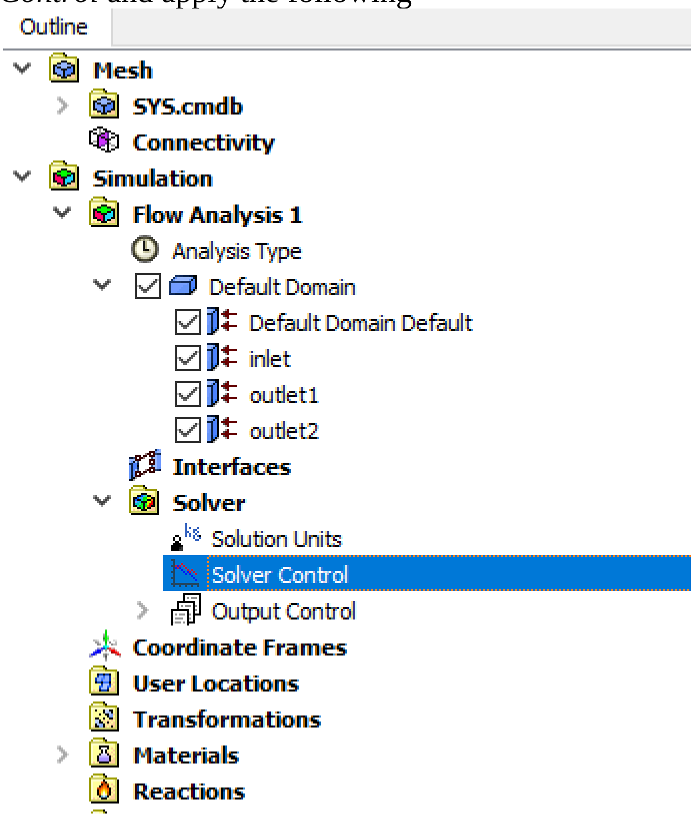


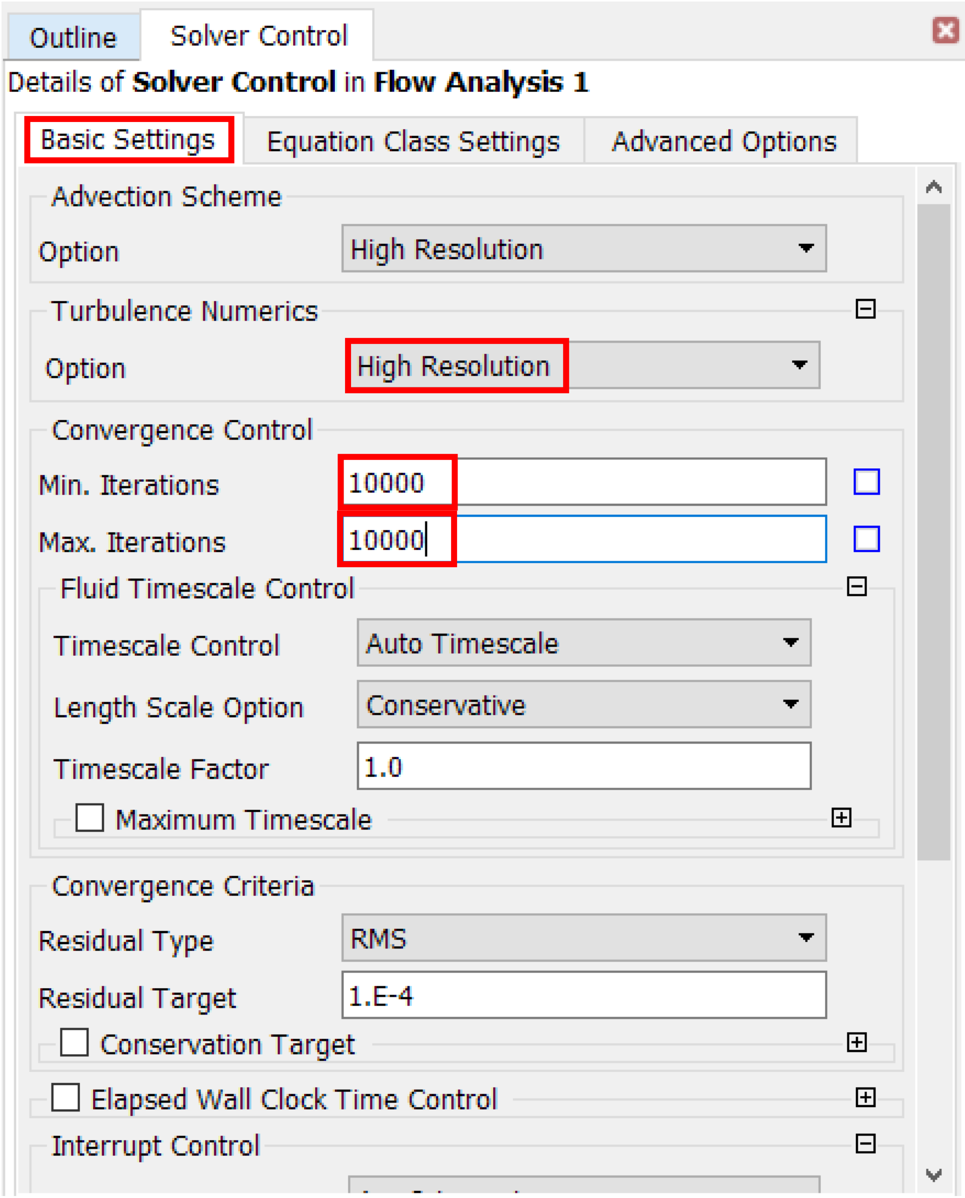
11) Create boundary condition *airfoil* and apply the following settings





12) Create a monitor point containing *expression* with a definition: areaInt(Wall Shear)@airfoil
13) Open *Solver Control* and apply the following

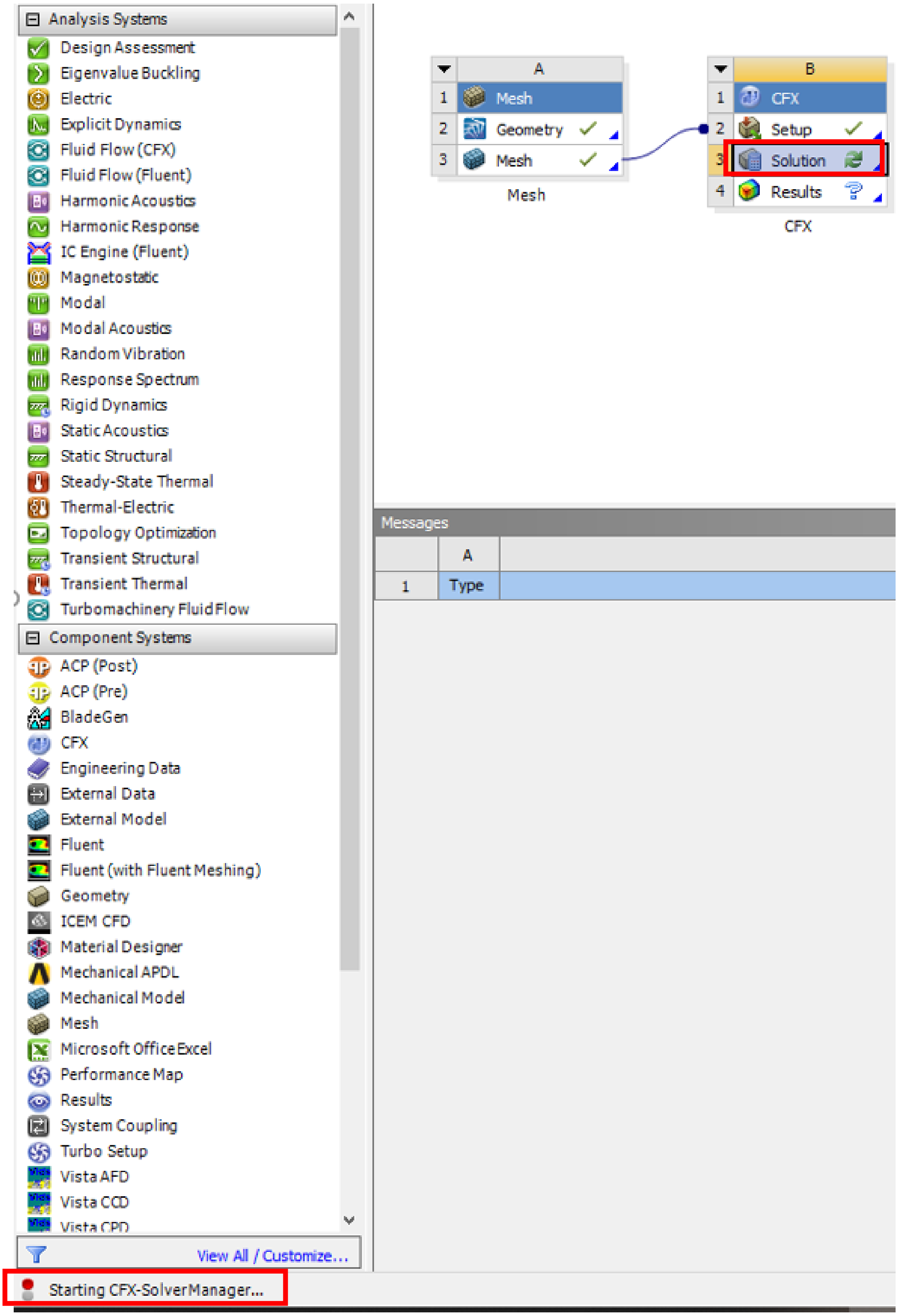




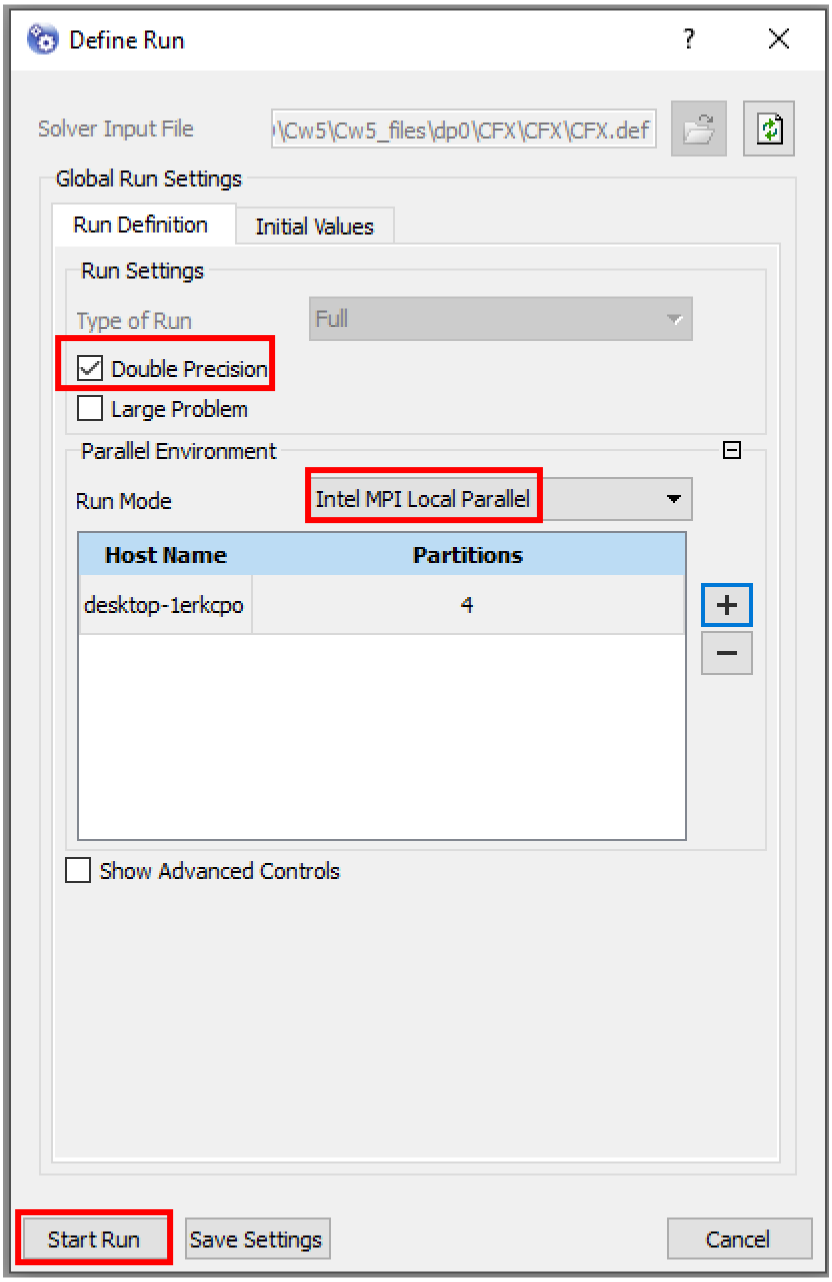
14) Close *Ansys CFX*.

2.4. Calculations

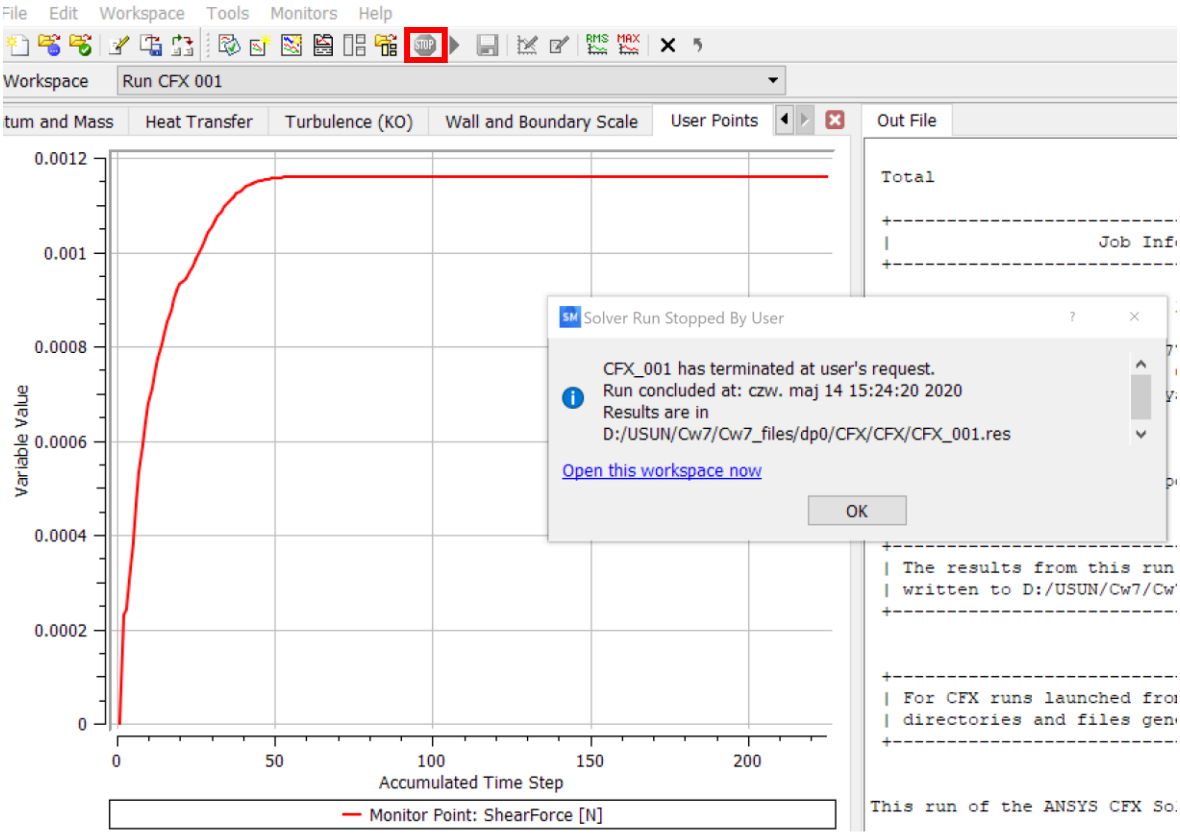
1) Double-click *Solution* to run *Ansys CFX Solver Manager*



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



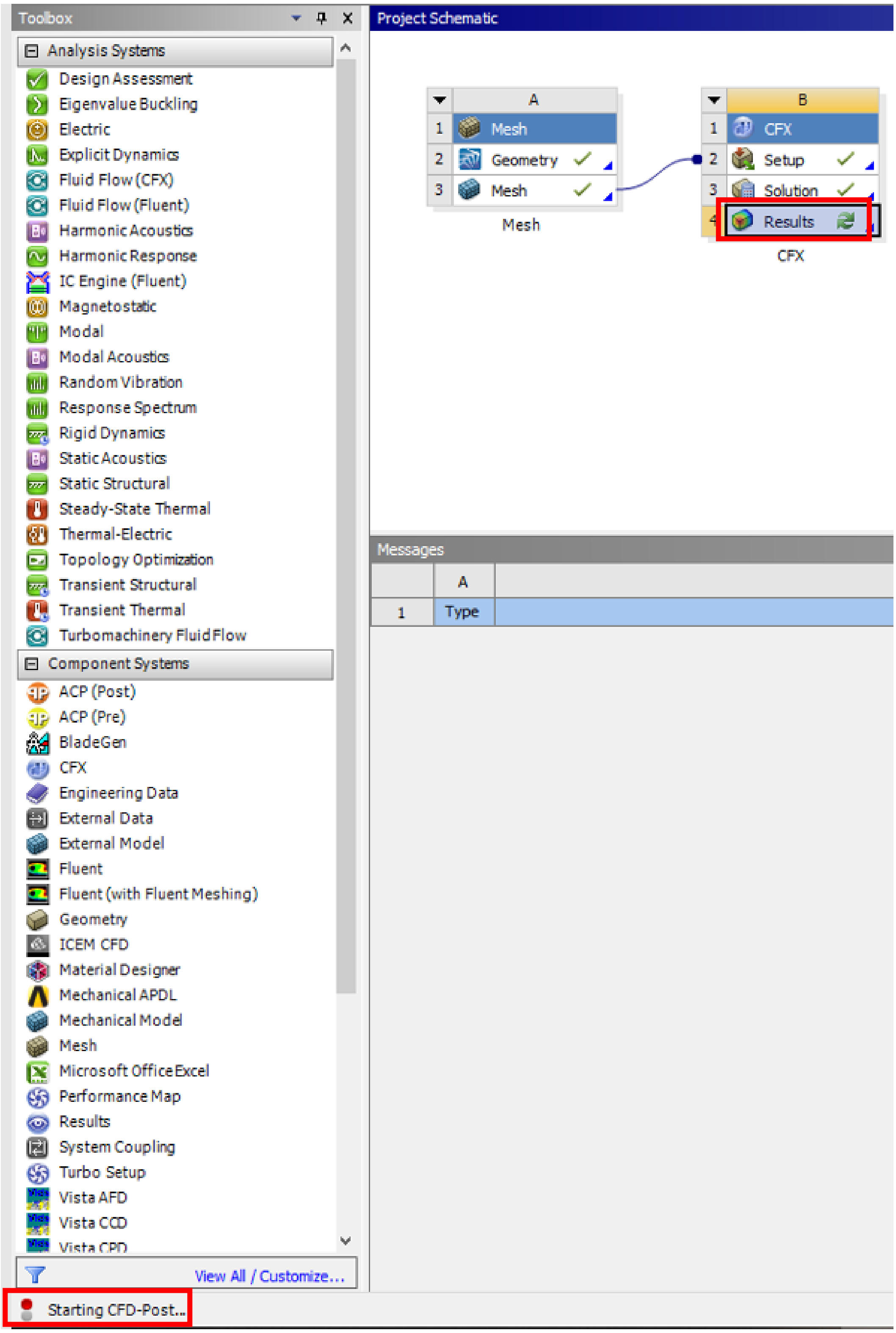
3) Calculations take about 2 minutes. Watch the individual bookmark tabs as they change. Pay special attention to the *User Point* tab, where the shear force on the profile is shown. Steady state will be reached when the curve stabilizes, which will occur after about 100 iterations. Additional iterations should be performed until all residuals have stabilized. To terminate calculations, 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 LMB *Results* to run *Ansys CFD Post* and see the results.



2) Create a plane perpendicular to the profile (parallel to the *sym* surface) and show the contours of

   a. *Pressure*
   b. *Absolute Pressure*
   c. *Total Pressure*
   d. *Velocity*
   e. *Mach Number*
   f. *Density*

   g. *Temperature*
   h. *Total temperature*
   i. *Turbulence Kinetic Energy*
   j. *Turbulence Eddy Dissipation*
3) Using *areaAve* calculate in a table:
   a. Inlet velocity value
   b. Inlet mass flow value
   c. The value of the average surface temperature at *airfoil*
   d. Average value of the variable *Yplus* at *airfoil*
4) Show contours of *Yplus* at *airfoil* surface
5) Display the power lines on the previously created plane
6) *Calculate drag and lift coefficients for the tested airfoil
7) Return to *Ansys CFX Pre* and change *expression Re* into 500 000 and repeat the calculations and points 2)-6). Next the same for *Re* = 5 000 000.

3. RESULTS TO BE INCLUDED IN THE REPORT

For *Re* = 50 000, 500 000 and 5 000 000 (Note: if possible, it is best to combine the results for comparison; e.g. Fig. 1. Pressure distribution in the plane perpendicular to the airfoil, a) Re = 50,000, b) Re = 500,000, c) Re = 5,000,000):

1) Contours in a plane parallel to the surface *sym*:
   a. *Pressure*
   b. *Absolute Pressure*
   c. *Total Pressure*
   d. *Velocity*
   e. *Mach Number*
   f. *Density*
   g. *Temperature*
   h. *Total temperature*
   i. *Turbulence Kinetic Energy*
   j. *Turbulence Eddy Dissipation*
2) Put the results in the table:
   a. Inlet velocity value
   b. Inlet mass flow value
   c. The value of the average surface temperature at *airfoil*
   d. Average value of the variable *Yplus* at *airfoil*
3) Photograph of the *Yplus* variable at *airfoil* surface
4) Streamlines (photo).
5) Answer the question: Based on the analysis of the *Yplus* variable value, whether the SST turbulence model used was selected correctly?

4. OPTIONAL TASKS

1. Perform calculations using a different turbulence model and compare the results.
2. Return to the *Meshing* module and compact the mesh (especially on the airfoil surface). Increase the number of elements also on other edges. Check how it affects the calculations and the value of the *Yplus* variable.

5. REFERENCES

[1] https://www.youtube.com/watch?v=xkEvS5eafCg&t=6s Access: 15.05.2020