Riccardo Zamolo, Enrico Nobile
DIA - Dipartimento di Ingegneria e Architettura
Università degli Studi di Trieste

Esercitazioni di Termofluidodinamica Computazionale

External aerodynamics over Ahmed's body with ANSYS Fluent

April 2020

1 Introduction and problem definition

This tutorial document shows how to perform a full CFD analysis of a vehicle aerodynamics problem using ANSYS Workbench 2020 platform (hereafter "WB").

The geometry will be defined using ANSYS SpaceClaim 2020 (hereafter "SpaceClaim"), the mesh will be generated using ANSYS Meshing 2020 (hereafter "ANSYS Meshing") and the problem will be solved using ANSYS Fluent 2020 (hereafter "Fluent") with RANS (Reynolds-Averaged Navier-Stokes) formulation: a turbulence model will be employed to find a steady-state solution.

For the sake of simplicity, the specific vehicle for this study case is the Ahmed's body [1], whose geometry is reported in Figure 1, in the case of a rear slant angle of 25°.

Beside its basic geometry, the physics of the problem is not trivial:

> "Most of the drag of the body is due to pressure drag, which is generated at the rear end. The structure of the wake is very complex, with a separation zone and counter-rotating vortices coming off the slant side edges, whose strength is mainly determined by the base slant angle." [3] (See Figure 2).

Once the characteristics of the problem have been defined as in Table 1, where air at $T = 20°$ C is the working fluid, our main goal is to compare the computed drag coefficient C_D to experimental data:

$$C_D = \frac{2F_x}{\rho u_\infty^2 A_x} \tag{1}$$

where F_x is the computed x-component of the force acting on the body (drag force) and A_x is the projected body area along the x-axis.

Since we'll use a RANS (time-averaged) approach, we can take advantage of the symmetry of the problem around the x-y plane (see Figure 1) for both geometry and boundary conditions to simulate half the space around the Ahmed's body.

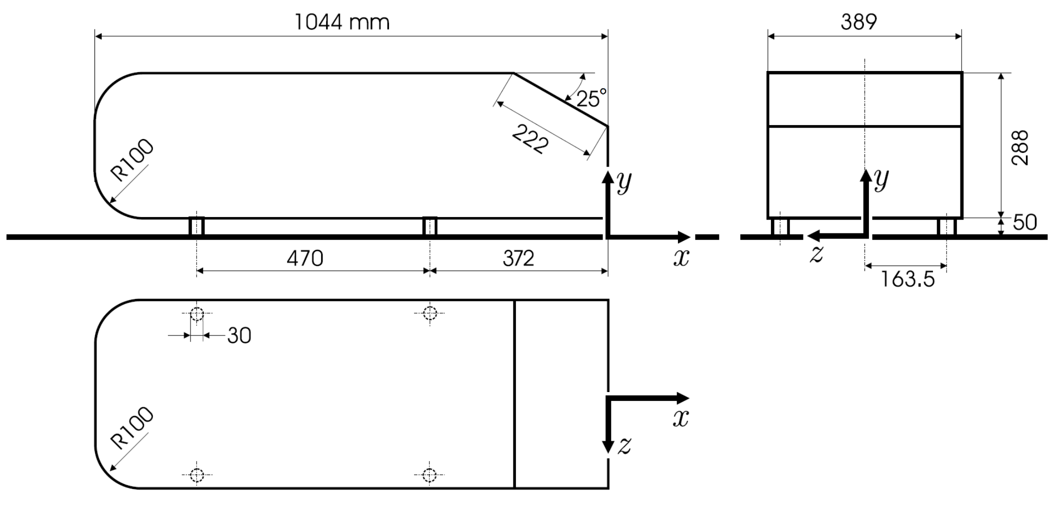


Figure 1: Ahmed's body with a rear vehicle slant angle of 25° (dimensions in mm).

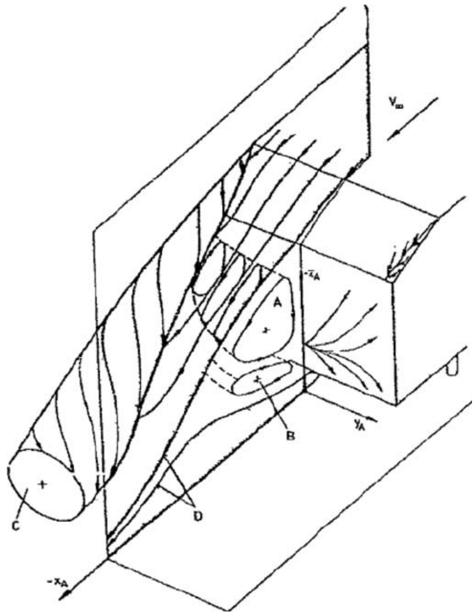


Figure 2: Rear separation zone (from [1]).

Characteristics of the problem.		
L	=	1044 mm
W	=	389 mm
H	=	288 mm
A_x	=	0.057516 m^2
u_∞	=	40 m/s
T	=	20° C
μ	=	1.789×10^{-5} kg/(m·s)
ρ	=	1.225 kg/m^3
Re	=	$\frac{\rho\, u_\infty\, L}{\mu} = 2.86 \times 10^6$

Table 1: Problem characteristics.

2 Workbench project

– Start WB and drag&drop a *Fluid Flow (Fluent)* component into the main window as in Figure 3; rename it *Ahmed_body* and save this new project: *File* → *Save as...*

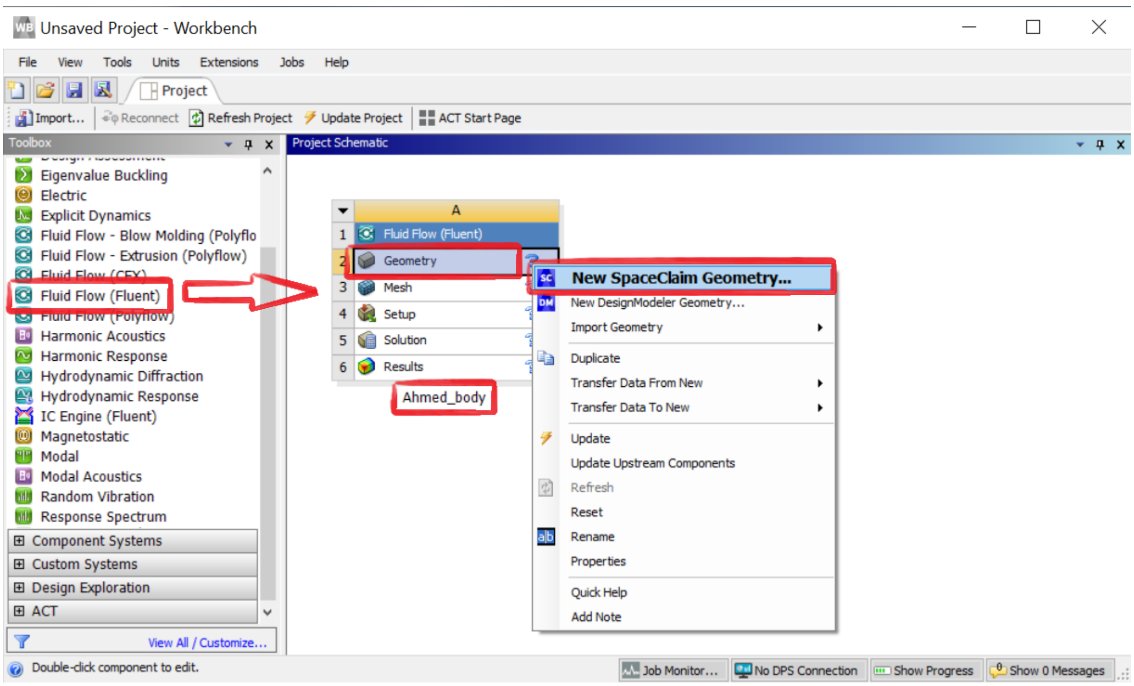


Figure 3: WB project (Fluid Flow with Fluent).

2.1 Geometry definition with SpaceClaim

– Right click on *Geometry* and click on *New SpaceClaim Geometry...* to define a new geometry with SpaceClaim (Figure 3).

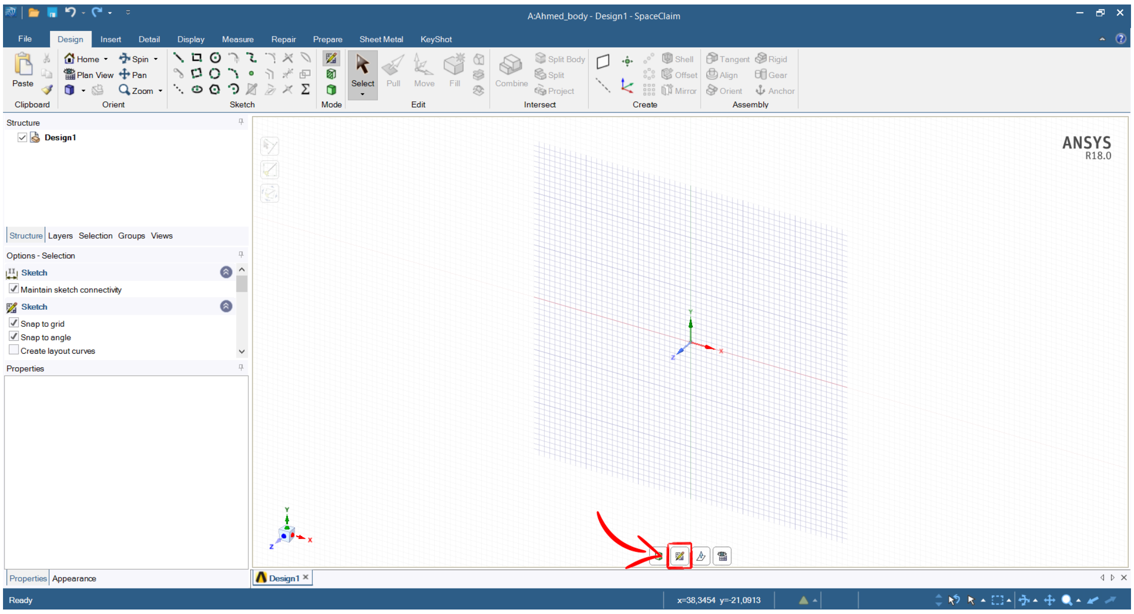


Figure 4: SpaceClaim.

– Click on *Sketch mode* icon and move the mouse in the main window to position the sketch on the $x - y$ plane (or click on the z axis), then click on *Plan view* icon Plan View to have an orthogonal plan view (Figure 4).
– Using *Line*, define the simplified 2D profile of the Ahmed's body reported in Figure 5:

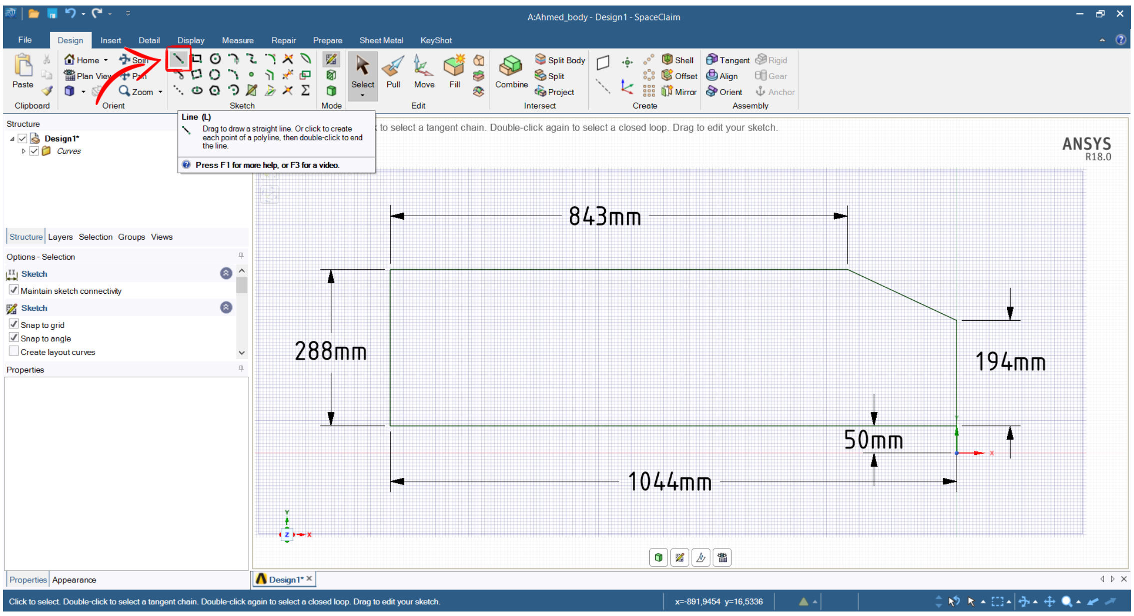


Figure 5: Ahmed's body starting profile.

– Click on *View* icon and select the *Trimetric* view (Figure 6):

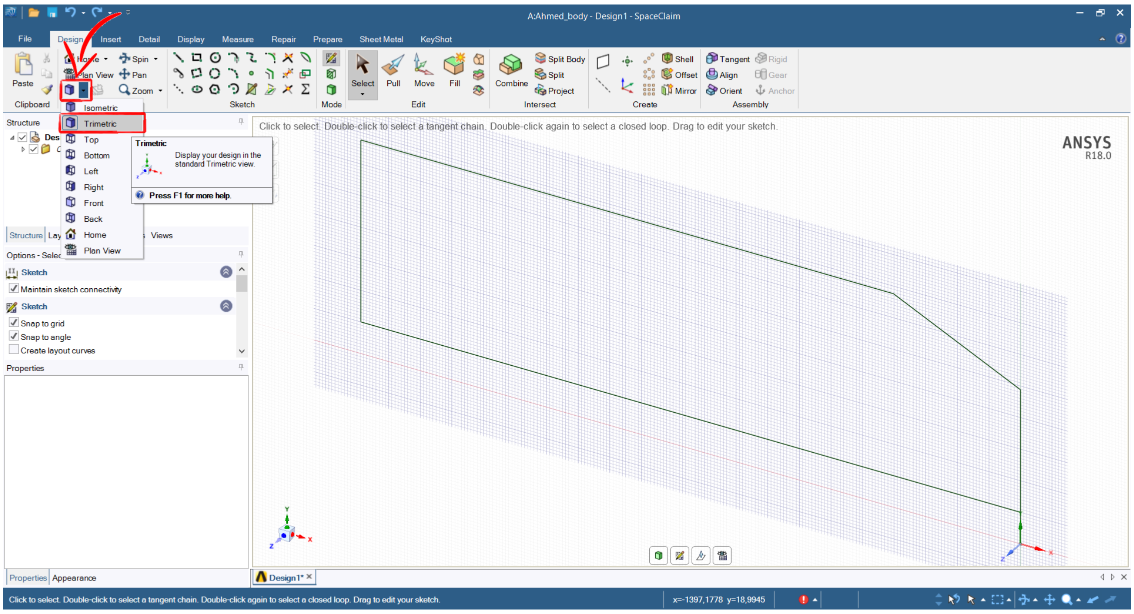


Figure 6: Trimetric view.

– Now we can extrude the profile using the *Pull* command: click on the profile and insert 194,5mm by keyboard to pull the profile along the $+z$ direction (Figure 7):

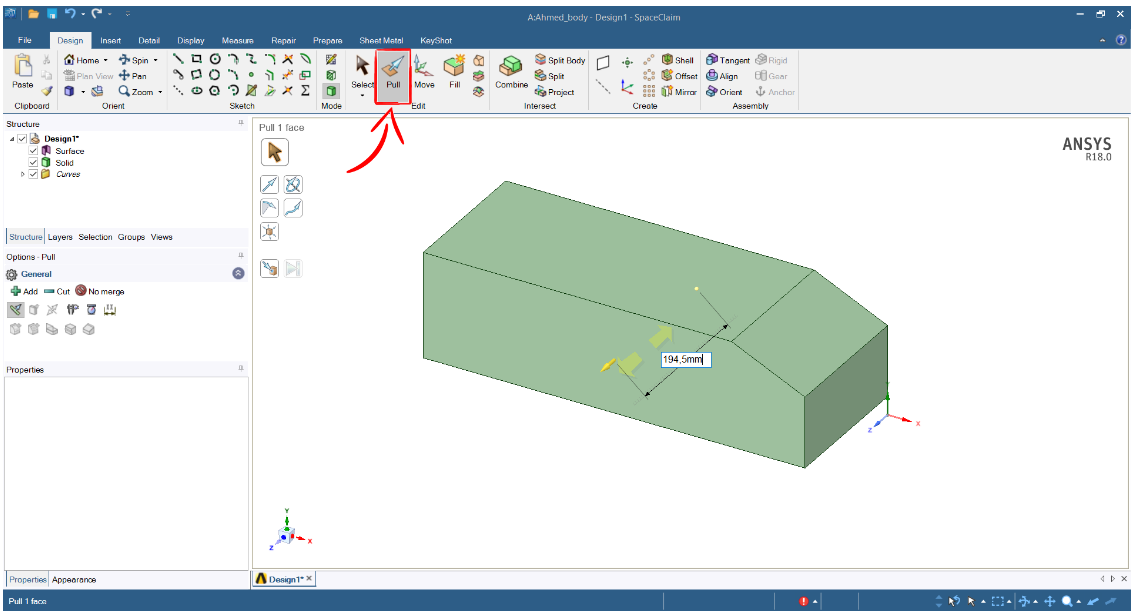


Figure 7: Pull command to extrude the profile.

– Use middle mouse button to rotate the view, mouse wheel to zoom the view, Ctrl + middle mouse button to move the view.
– To create the double roundings on the vehicle front, select *Pull* command and click on one of the front edges, then click on *Round* icon (Figure 8):

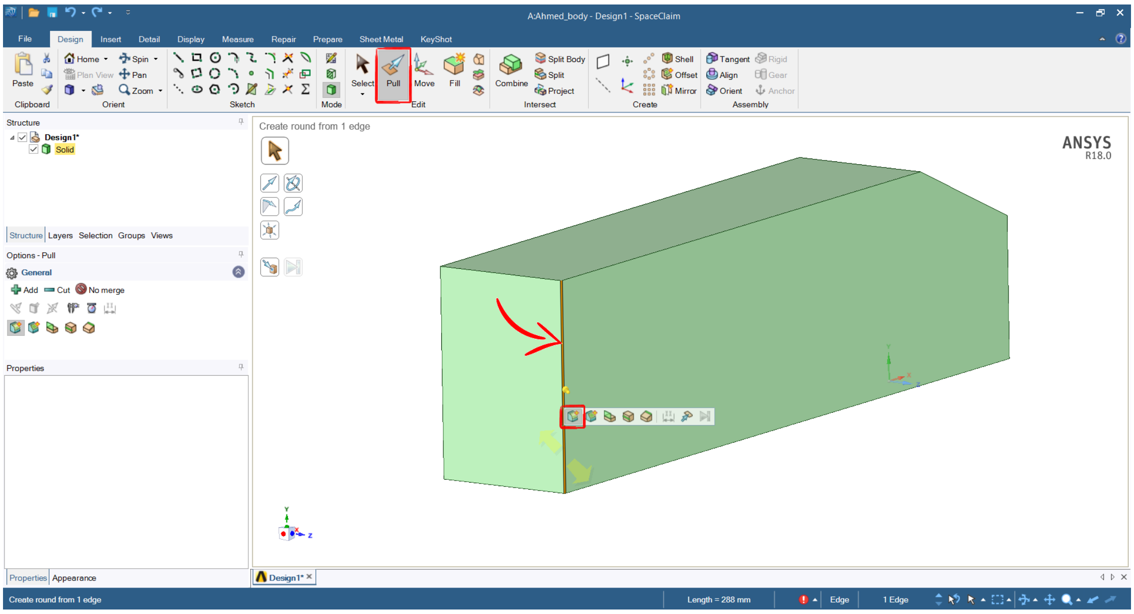


Figure 8: Roundings.

– Specify 100mm by keyboard as rounding radius (Figure 9):

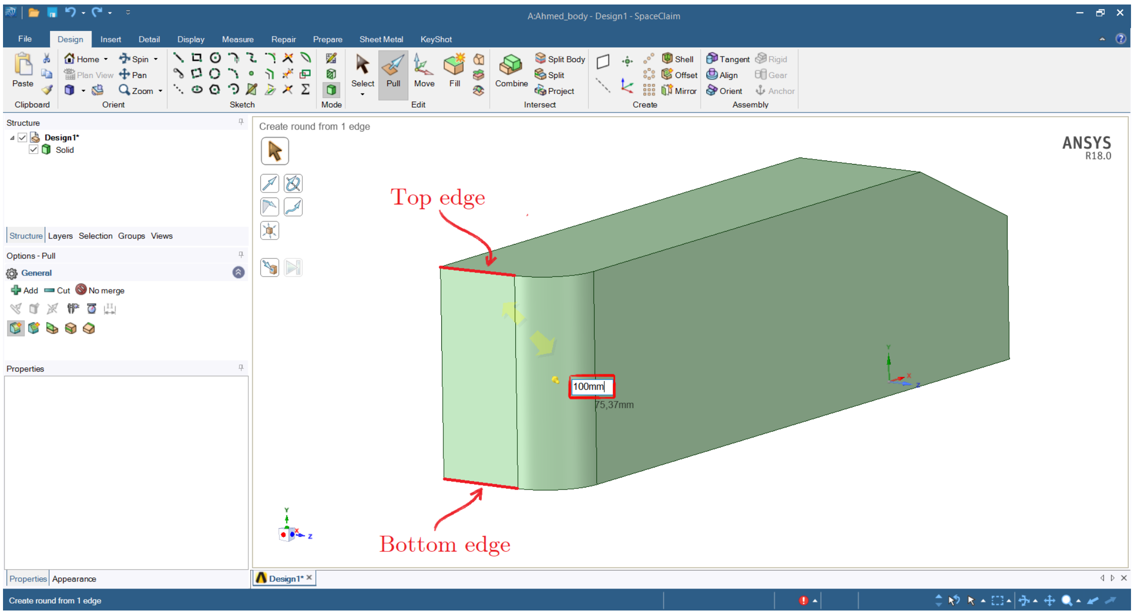


Figure 9: Roundings.

– Repeat this rounding operation for top and bottom edges.

– Select the *Sketch mode* icon and select the bottom plane of Ahmed's body (Figure 10), then click *Plan View*.

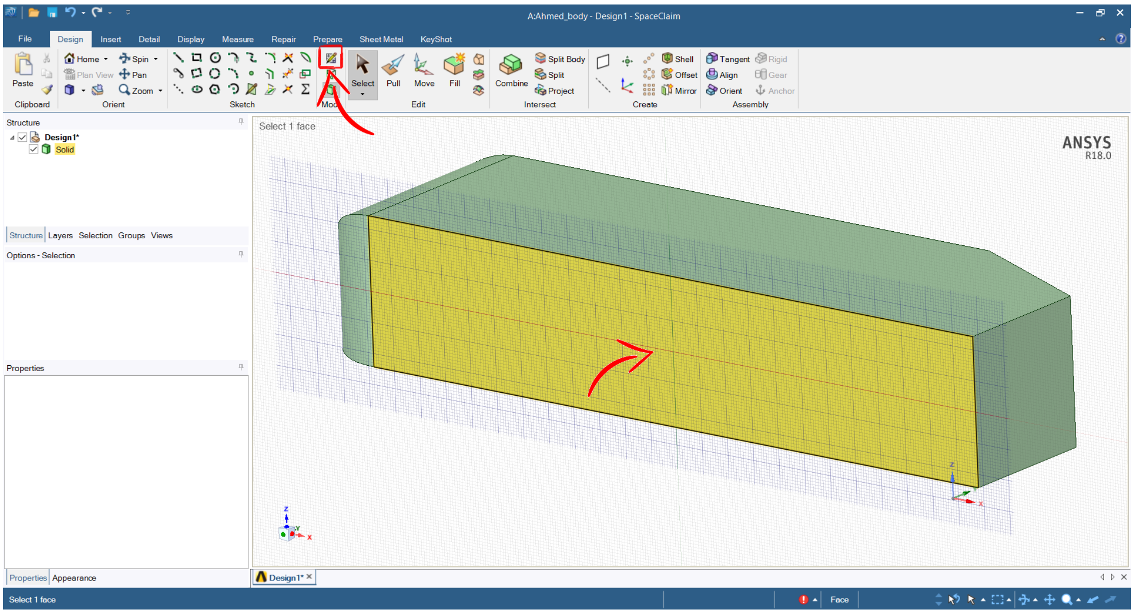


Figure 10: Sketch plane definition.

– Use *Construction line* and *Offset Curve* to draw the construction geometry of Figure 11 that will be required to define the supporting feet of Ahmed's body:

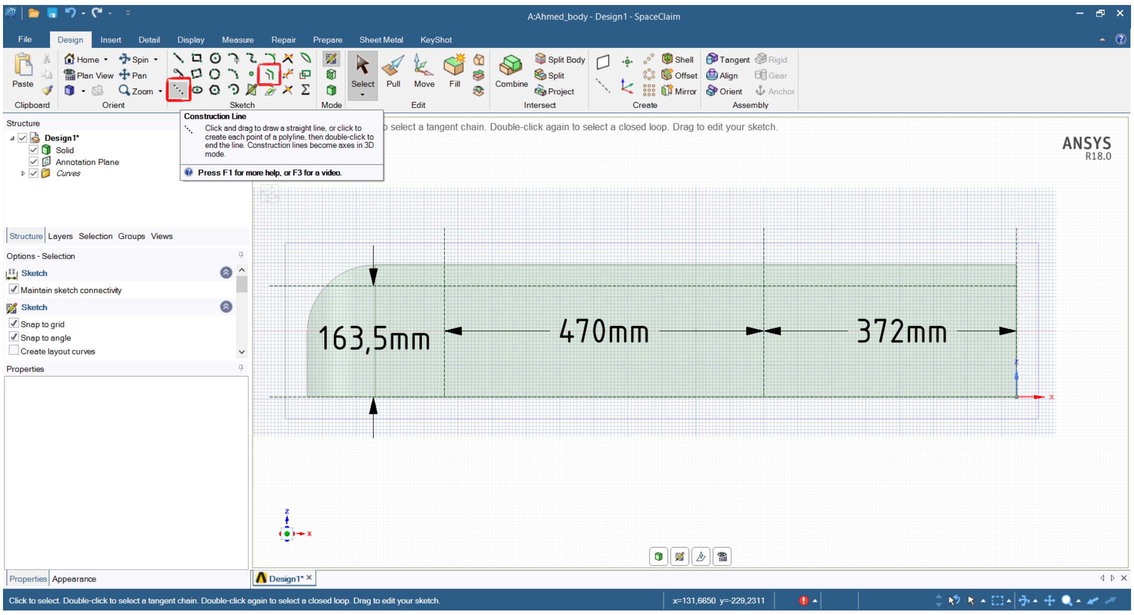


Figure 11: Bottom construction geometry.

– Using *Circle*, draw the base of cylindrical feet of Ahmed's body, specifying 30mm by keyboard as diameter (Figure 12):

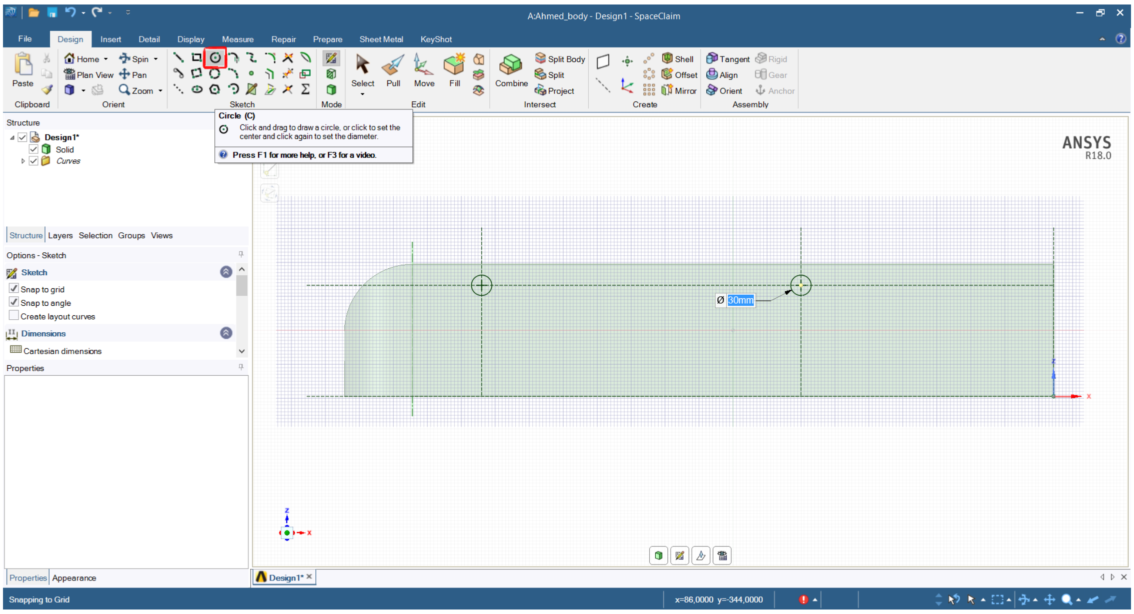


Figure 12: Supporting feet geometry.

– Use *Pull* to extrude both circles with an extruding length of 50mm, defined by keyboard (Figure 13):

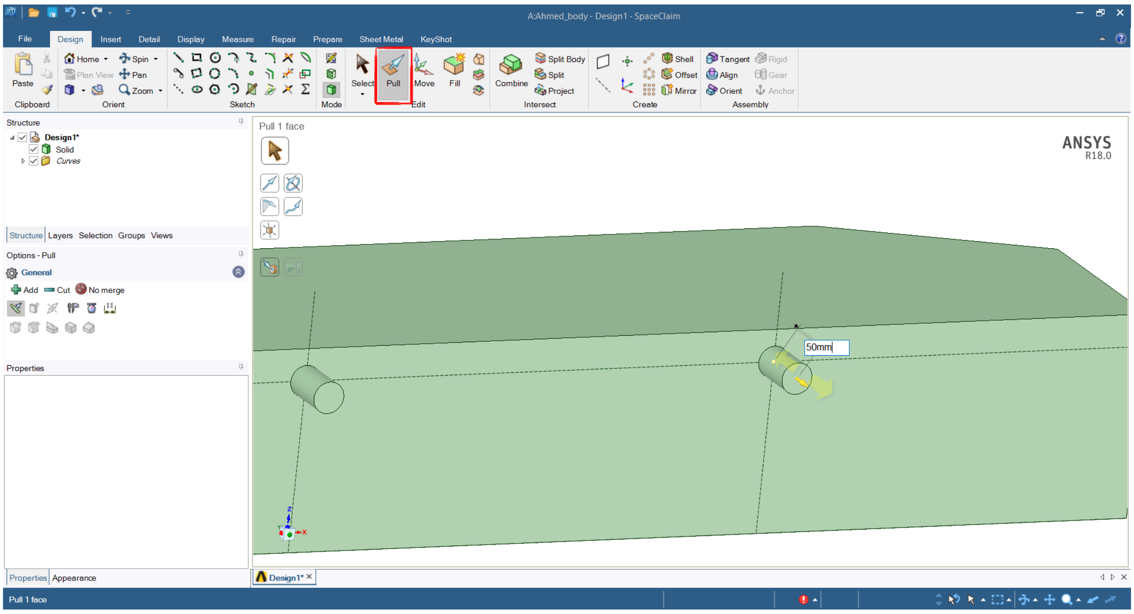


Figure 13: Supporting feet.

Now the geometry of the Ahmed's body is completed, but we need to define an external enclosure to simulate the external aerodynamics around the body. This enclosure starts $2L$

before the body and extends $5L$ behind it; its height is 1.4m and its width is $2.5W$ (these dimensions are recommended for the 10th joint ERCOFTAC Workshop on Refined Turbulence Modelling).
– Click on *Prepare* tab and then click on *Enclosure* icon; select *Box* as Enclosure type and untick *Symmetric dimensions* (Figure 14):

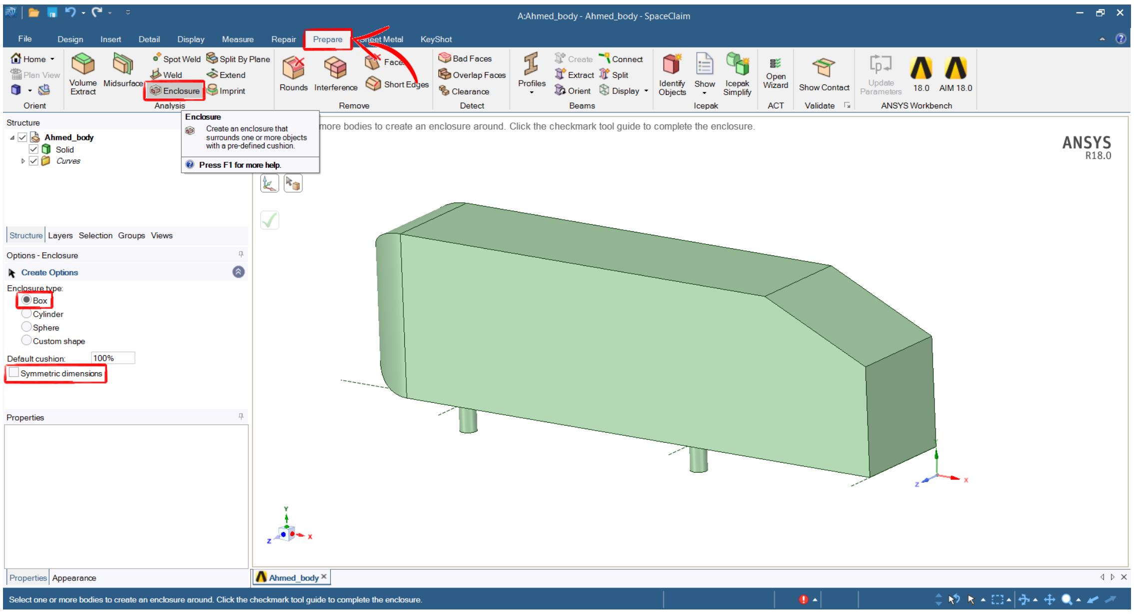


Figure 14: Enclosure options.

– Select the body and insert the dimensions reported in Figure 15, then click on  icon:

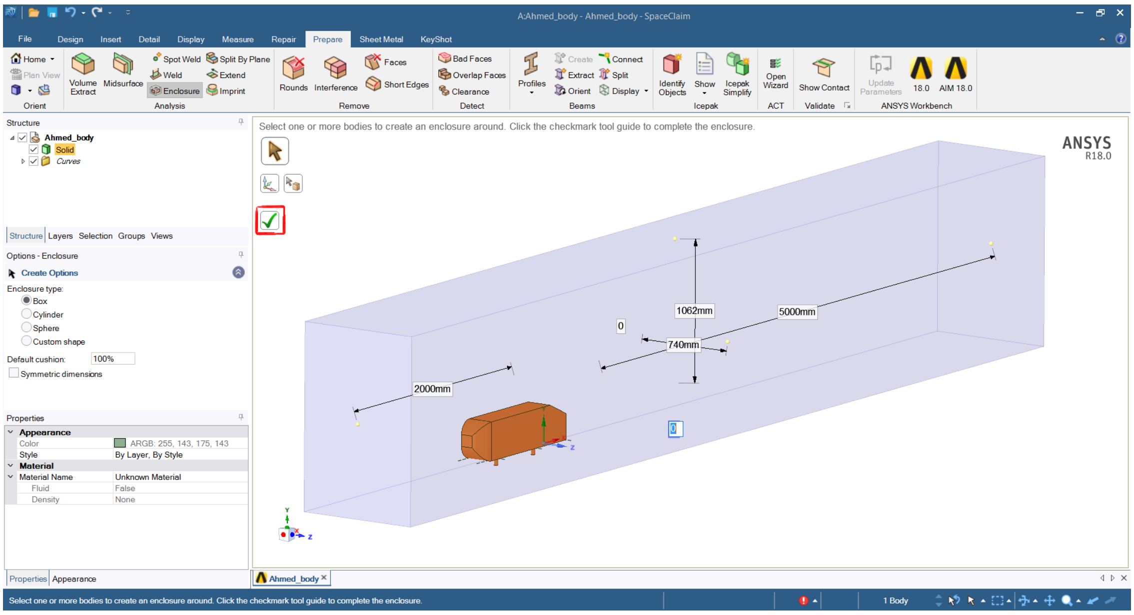


Figure 15: Enclosure dimensions.

– In the *Structure* tab, right click on *Solid* (the Ahmed's body) → *Suppress for Physics* since we don't need to mesh it nor to perform calculations inside it (as it would in the case of a two-way fluid-structure coupling) (Figure 16):

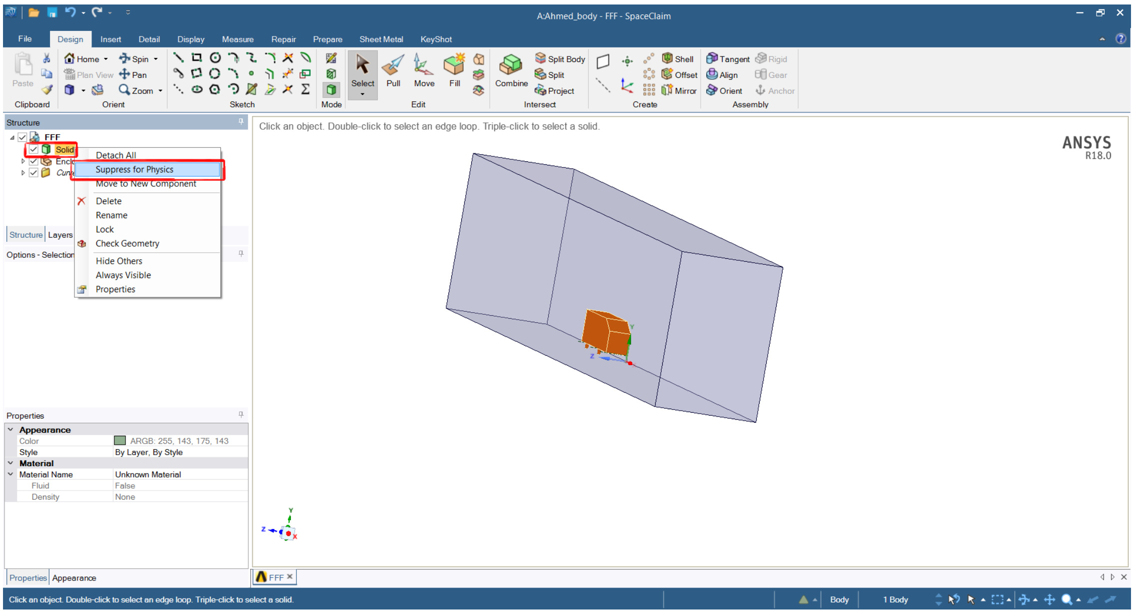


Figure 16: Ahmed's body volume is not required for our CFD analysis.

Since we want to increase the mesh resolution around the body, we need to define a *Body of influence*: click on *Sketch* icon under *Design* tab and select the symmetry plane of the Ahmed's body (Figure 17):

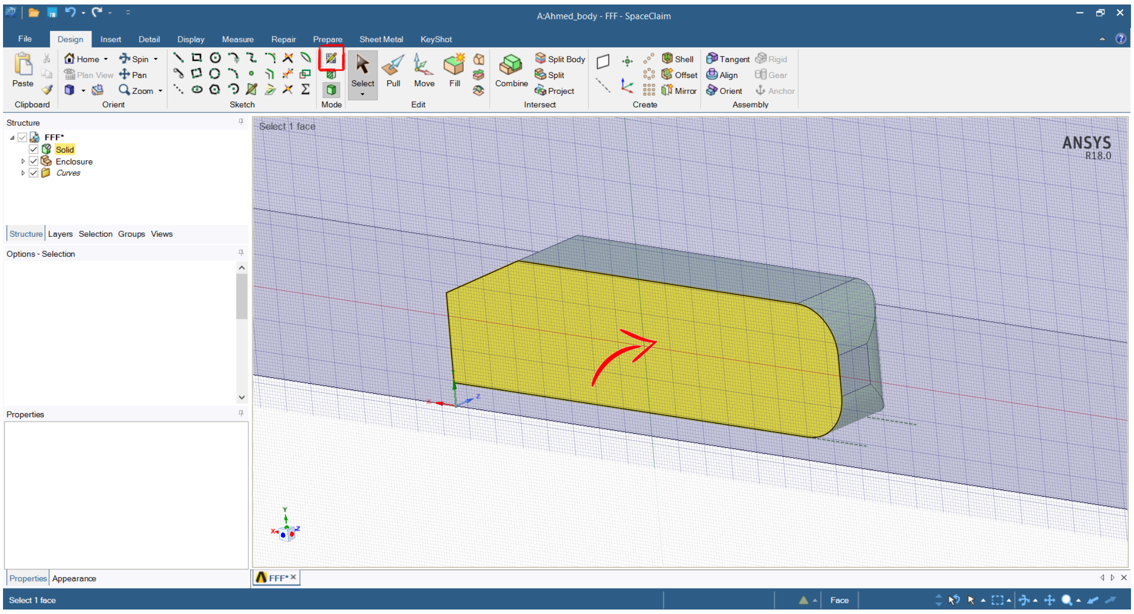


Figure 17: Plane selection for *Body of influence* definition.

– Click on *Plan View* and define the rectangular geometry of Figure 18 (the x length of the body is 1044mm, the width of the rectangle is 2544mm):

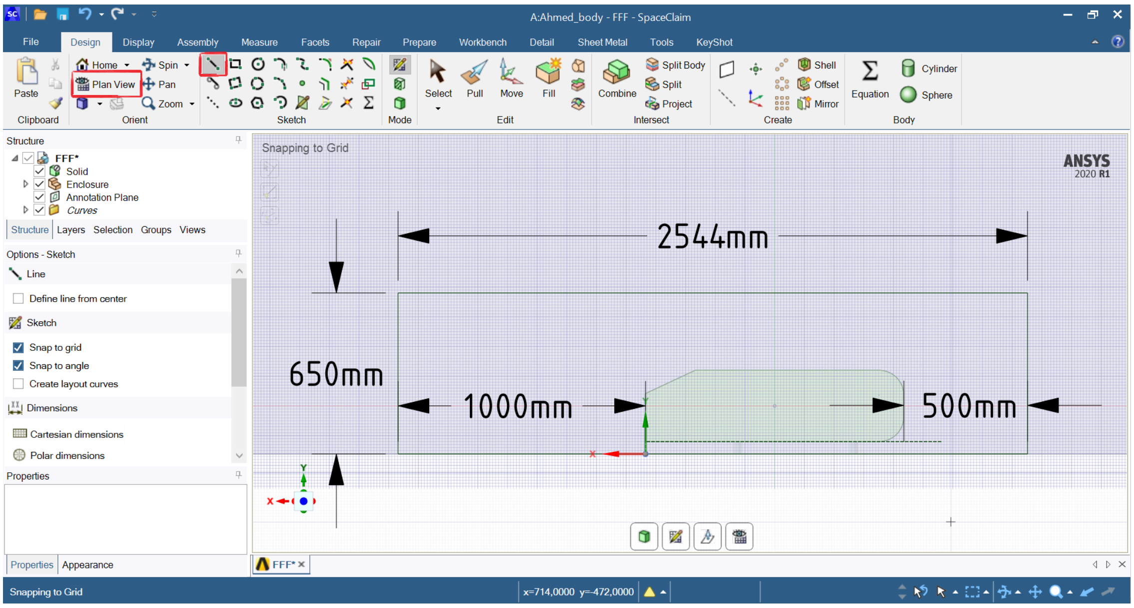


Figure 18: Body of influence definition.

– In the *Structure* tab, untick *Solid* and *Enclosure* (Figure 19):

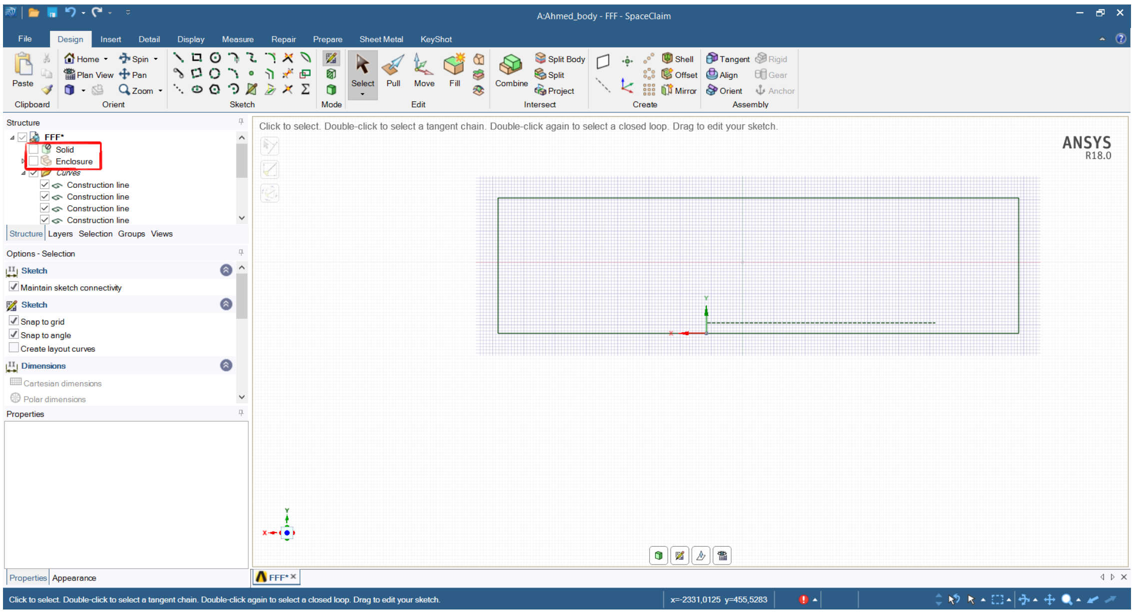


Figure 19: Body of influence definition.

– Click on *Pull* to extrude the rectangular profile in the $+z$ direction for 400mm (Figure 20):

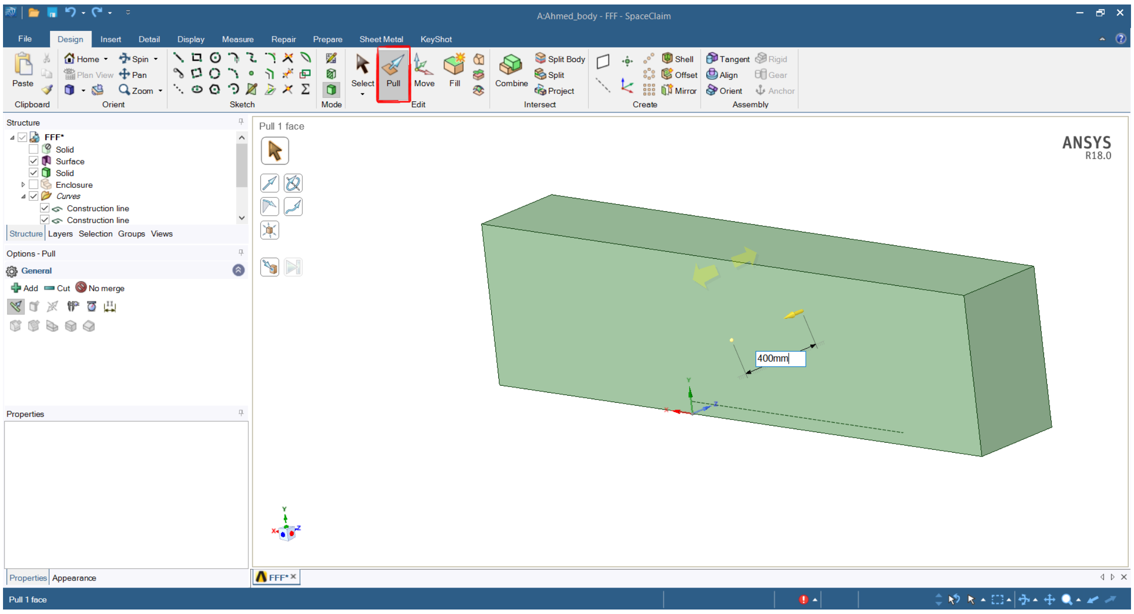


Figure 20: Body of influence.

– Rename the new solid as 'Size_box' (Figure 21):

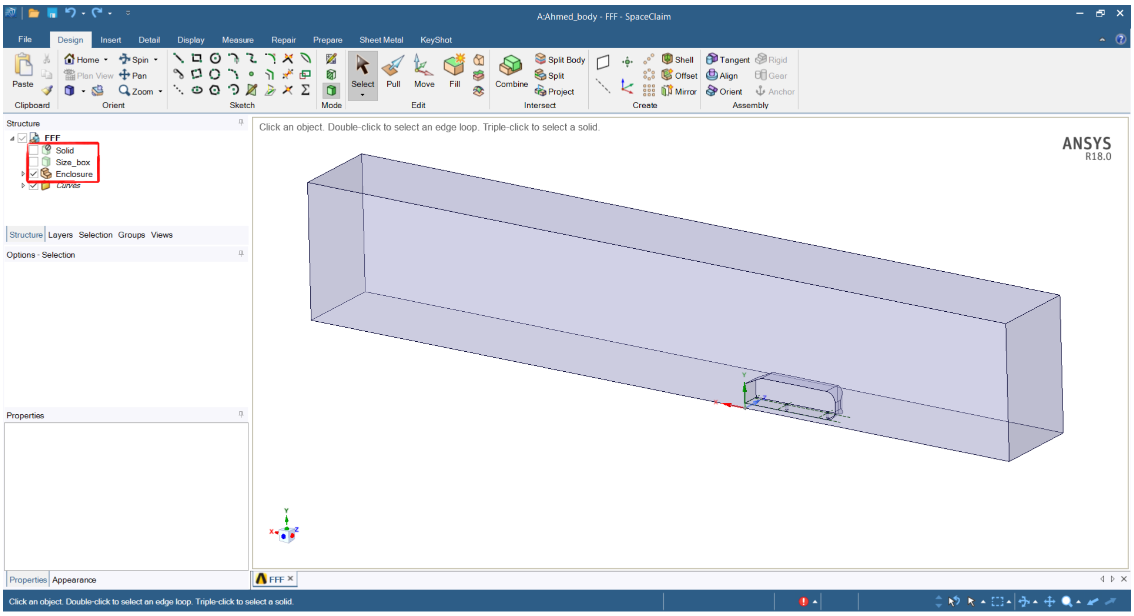


Figure 21: Ahmed's body enclosure.

– *File* → *Save Project* and close SpaceClaim.
Now that every geometric entity has been defined, we can start meshing the enclosure using ANSYS Meshing.

2.2 Meshing the enclosure with ANSYS Meshing

In WB, right click on *Mesh* → *Edit* to mesh the enclosure in ANSYS Meshing (Figure 22):

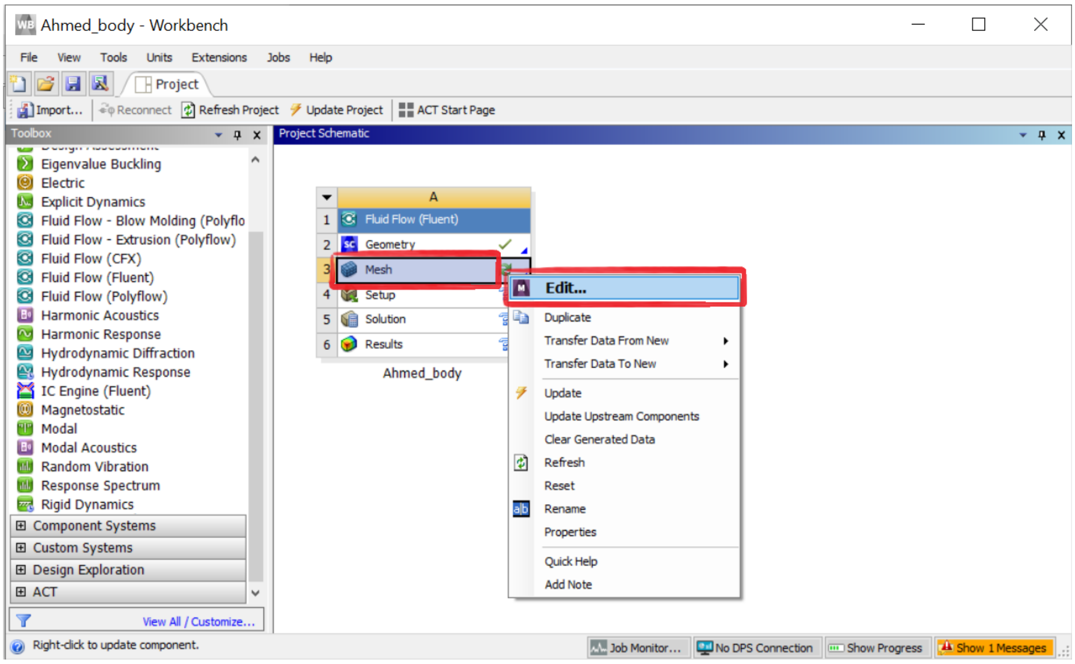


Figure 22: Starting a new Meshing session from WB.

— Under *Outline* tab → *Project* → *Model* → *Geometry* right click on *FFF\Size_box* and select *Hide Body* (Figure 23):

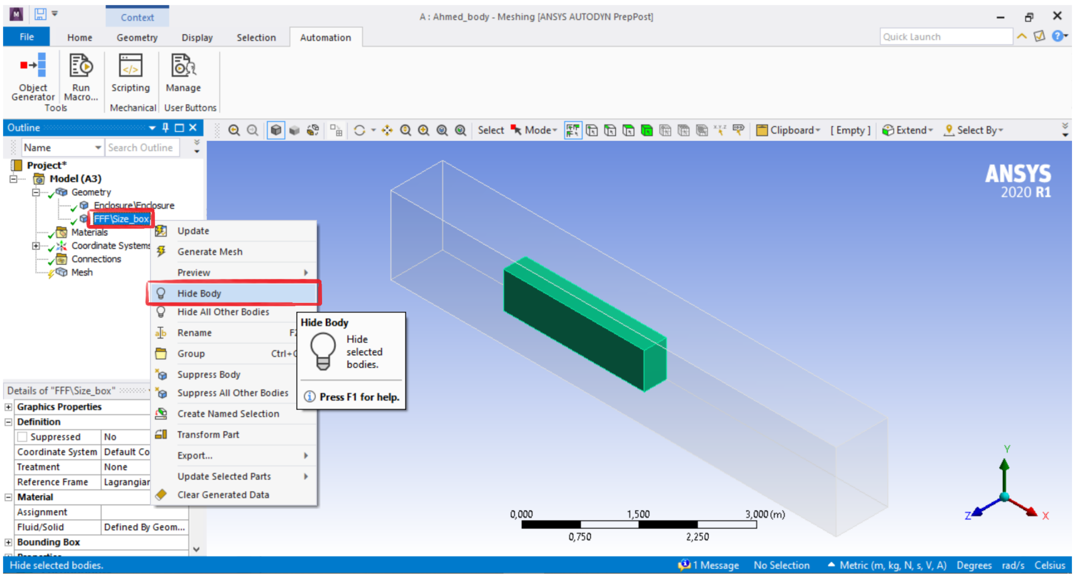


Figure 23: Hiding the body of influence.

– Click the *Face* icon and select the inflow face (left click), then right click on it → *Create Named Selection* and specify the name *Inflow* (Figure 24):

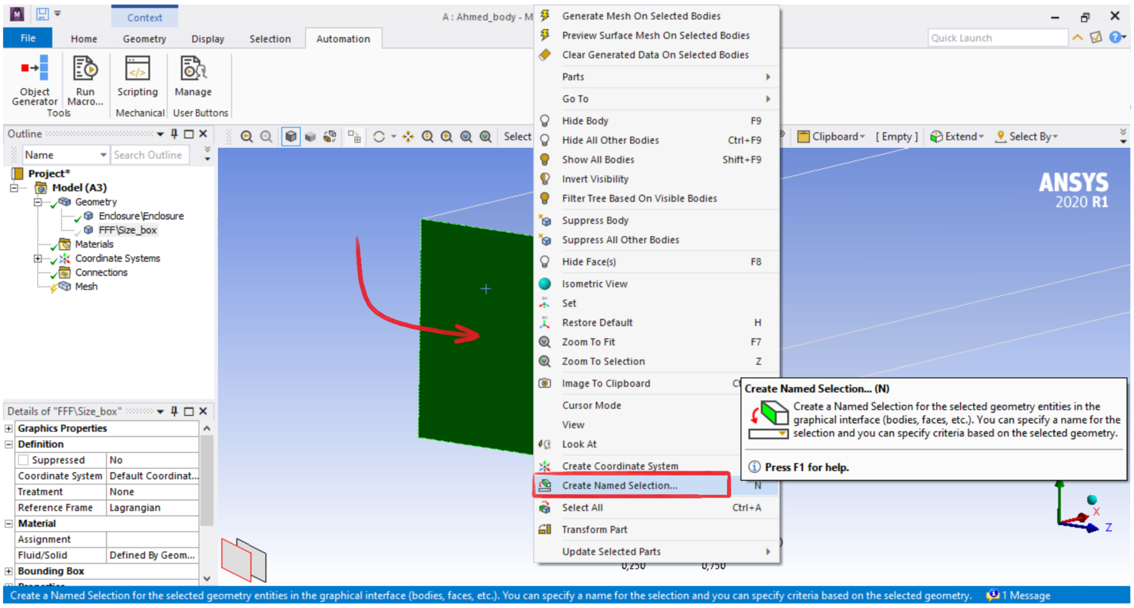


Figure 24: Named selection.

– Repeat this operation for all the faces of the enclosure (Top, Ground, Side, Symmetry, Outflow, Ahmed_surface). For the surface of the Ahmed's body: click on the z axis for a side view, click on *Select Mode* → *Box Select* to easily select all its patches (Figure 25):

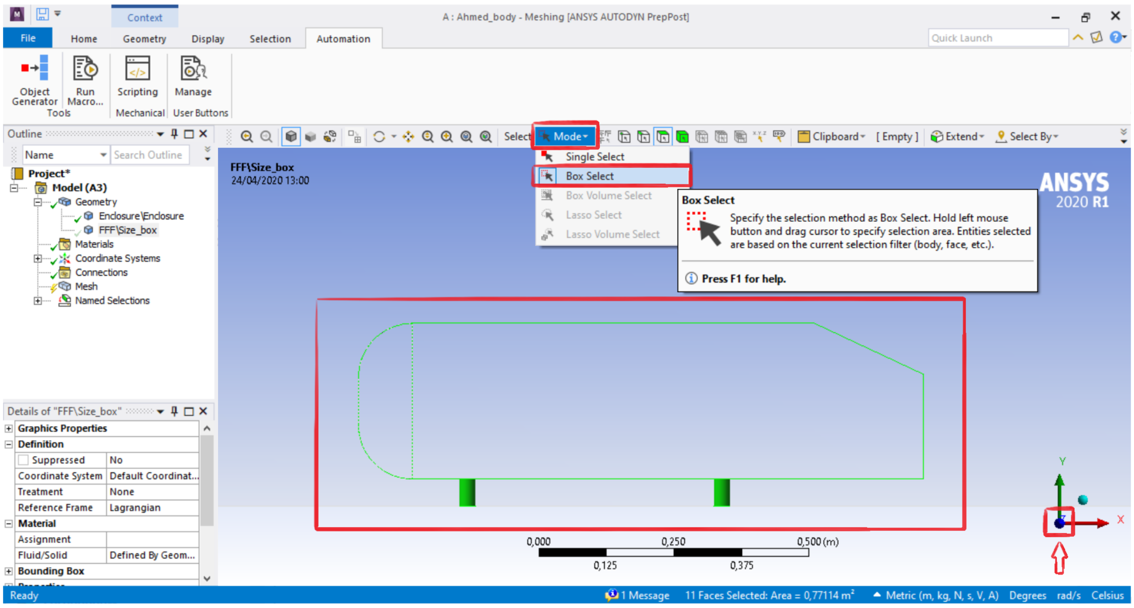


Figure 25: Ahmed's surface selection.

— Under *Outline* tab → *Project* → *Model* → *Mesh*, in the *Details of "Mesh"* tab, specify all the marked properties as in Figure 26:

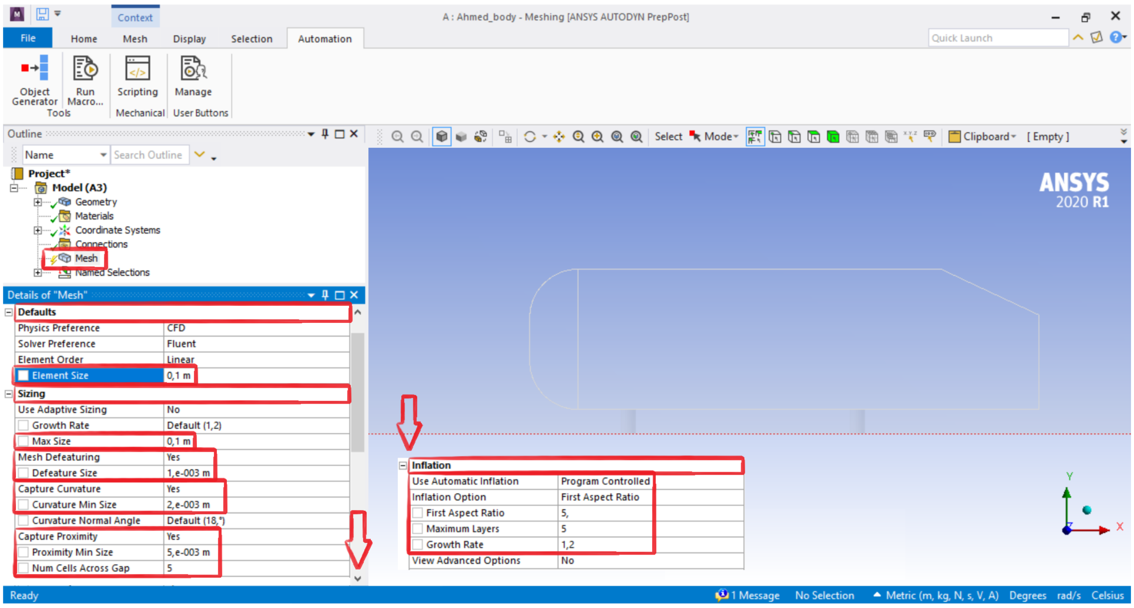


Figure 26: Mesh Sizing and Inflation properties.

— Since we want an extruded mesh with prisms in the boundary layers, we must define their locations; under *Outline* tab → *Project* → *Model* → *Named Selections*, click on *Ground* and set *Include* under *Program Controlled Inflation* in the *Details of "Ground"* tab (Figure 27):

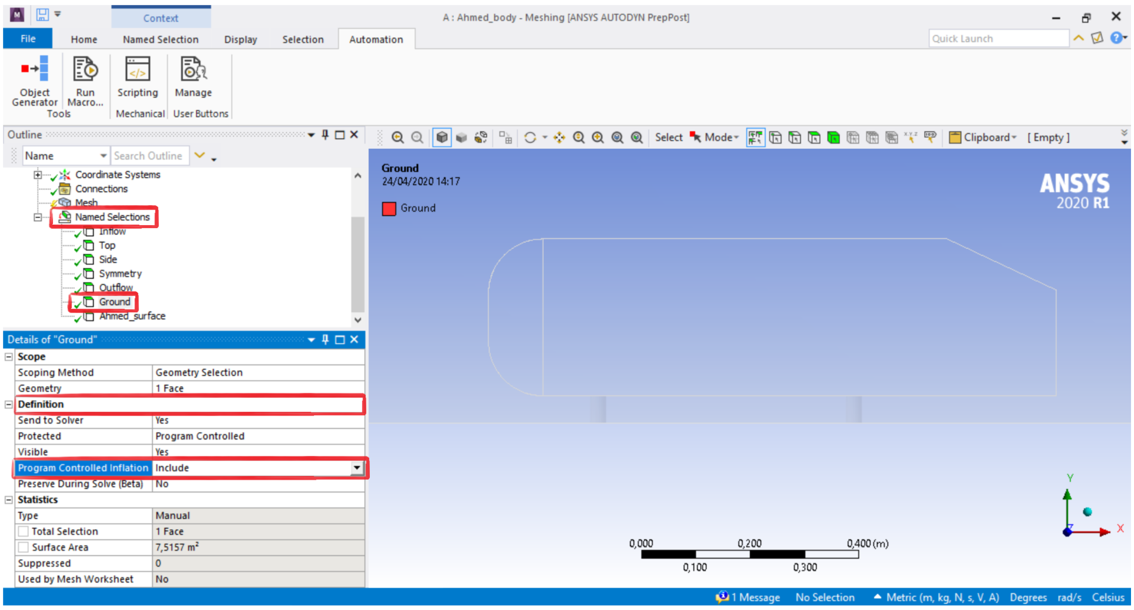


Figure 27: Inflation locations.

— Repeat the same operations for the *Ahmed_surface* Named Selection.

— *Outline* → *Project* → *Model*, right click on *Mesh* → *Insert* → *Sizing* (Figure 28):

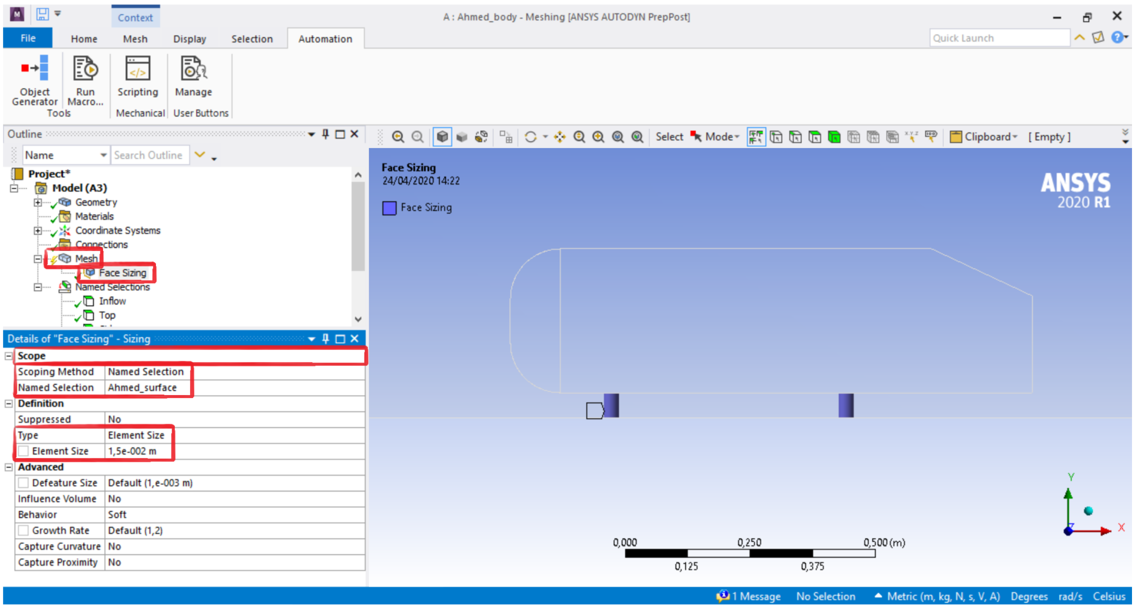
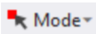


Figure 28: Mesh sizings.

— Under *Outline* tab → *Project* → *Model* → *Geometry*, right click on *FFF\Size_box* → *Show Body*; click the *Single Select* icon  and *Body* icon ;

— Repeat the insertion of a new sizing: in *Details of "Sizing" - Sizing* tab, for *Scoping Method* select *Geometry Selection*, for *Geometry* select the whole enclosure, for *Type* select *Body of Influence*, for *Bodies of Influence* select the size box and for *Element Size* insert 0,030m (Figure 29).

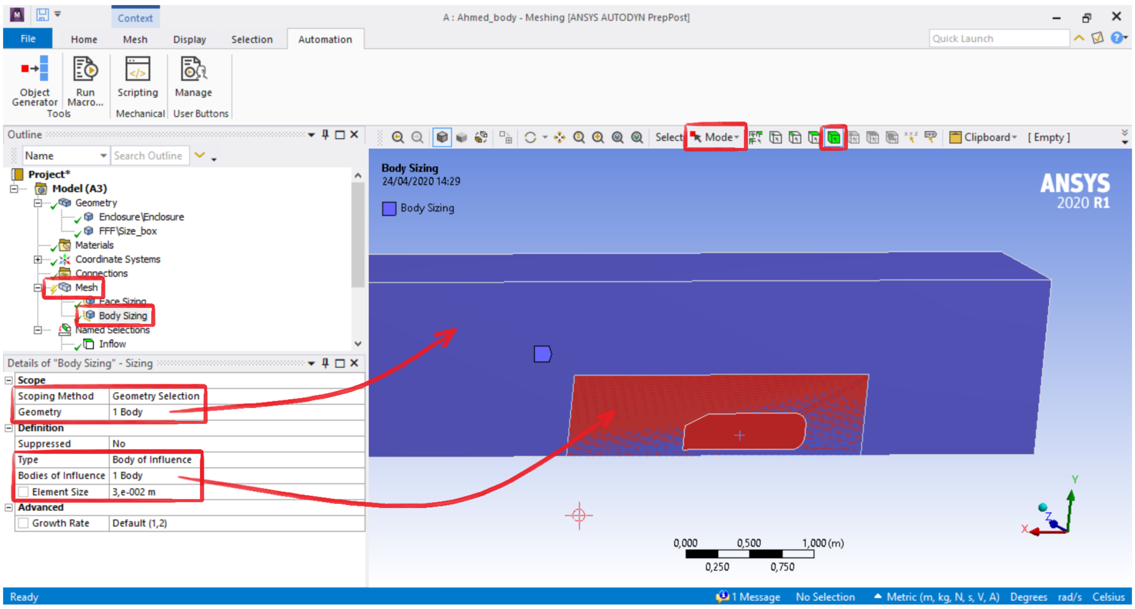


Figure 29: Mesh sizings.

– Right click on *Mesh* → *Generate Mesh* to create the mesh (Figure 30):

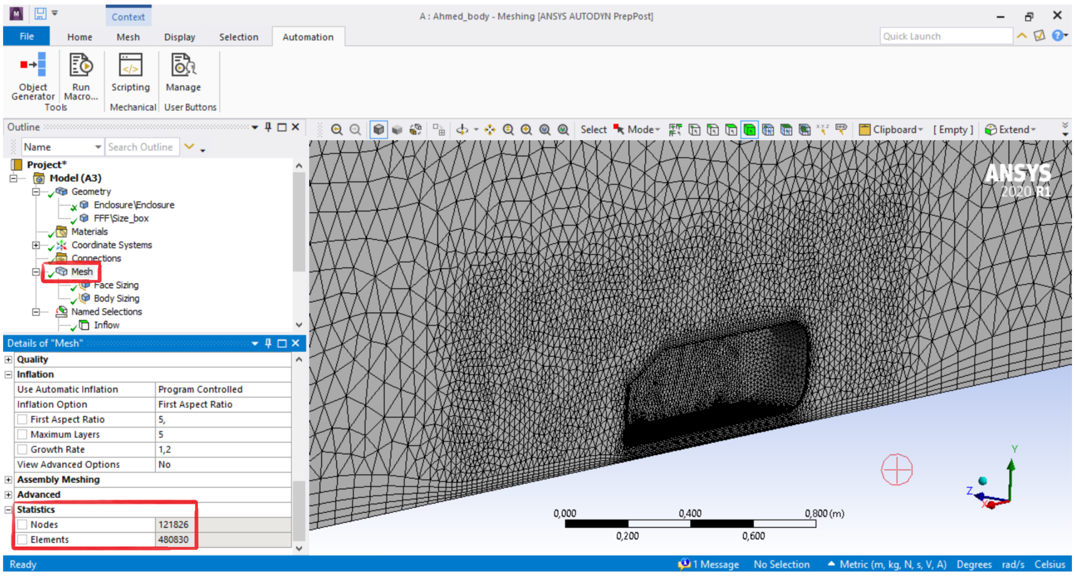


Figure 30: Mesh detail around the body.

– *Outline* → *Project* → *Model* → *Geometry*, click on *Mesh* to display the mesh; click on the top *Mesh* tab → *Insert* → *Section Plane* to insert a section plane for exploring the volume mesh (Figure 31). Click on *Show Whole Elements* icon to display whole mesh elements and not their projection on the section plane:

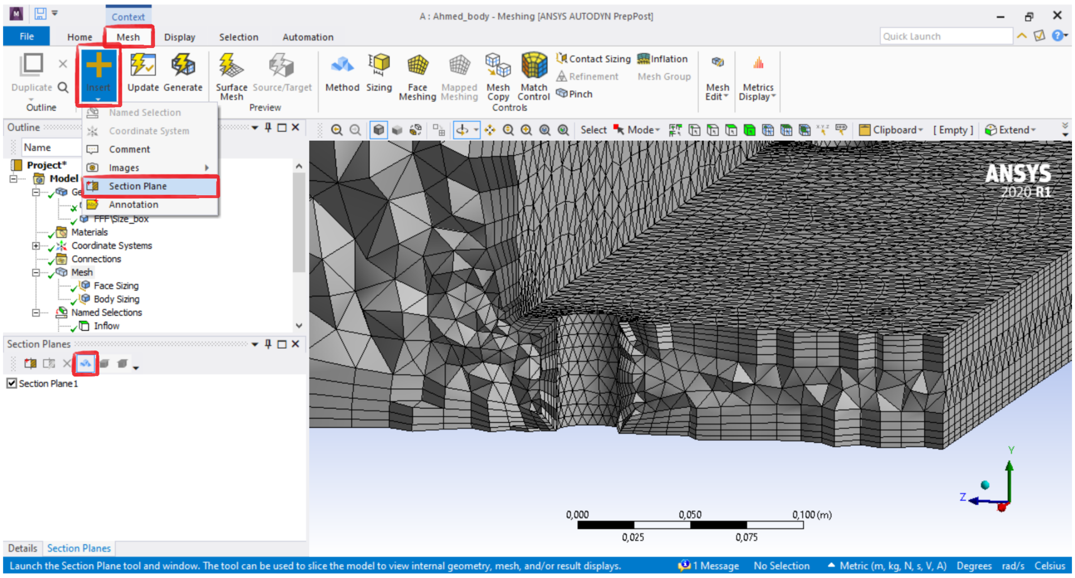


Figure 31: Mesh prism layers around the body surface and the ground.

The meshing phase is now complete: the mesh is composed by 380k tetrahedrons (*Tet4*, 4 nodes) for the volume mesh and 100k triangular prisms (*Wed6*, 6 nodes) in the inflation layers near the walls.
– *File* → *Save Project...* to save the mesh into the WB Project and close ANSYS Meshing.

2.3 Setup the problem in Fluent

In WB, right click on *Setup* → *Edit* to launch Fluent (Figure 32):

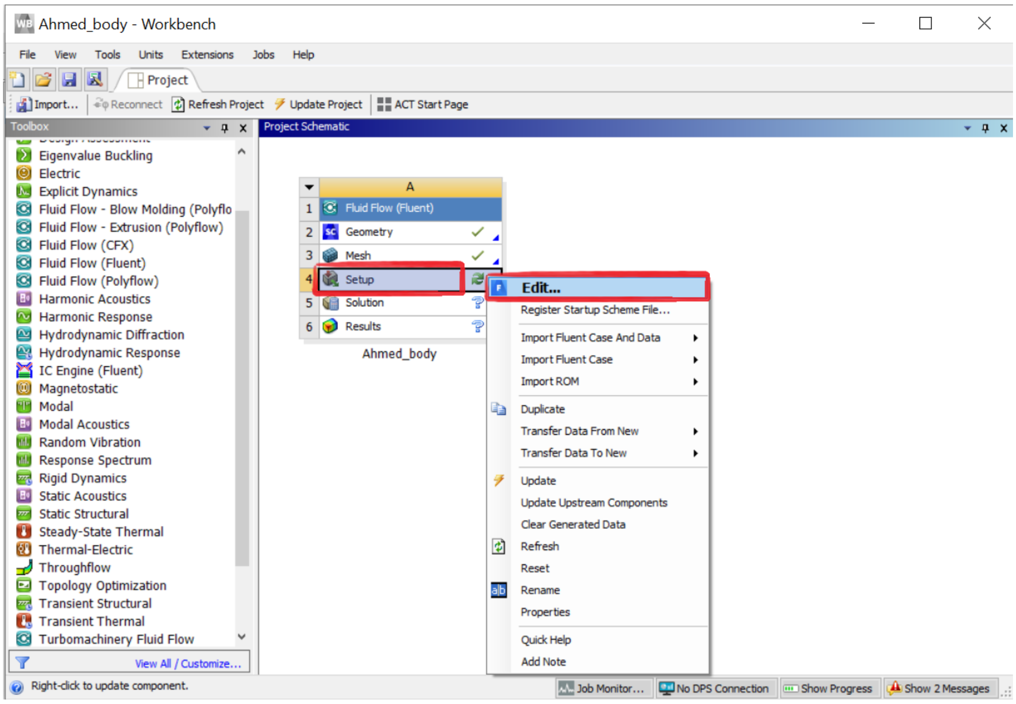


Figure 32: Starting Fluent from WB.

– In the *Fluent Laucher* window, untick *Double Precision* Option (Figure 33):

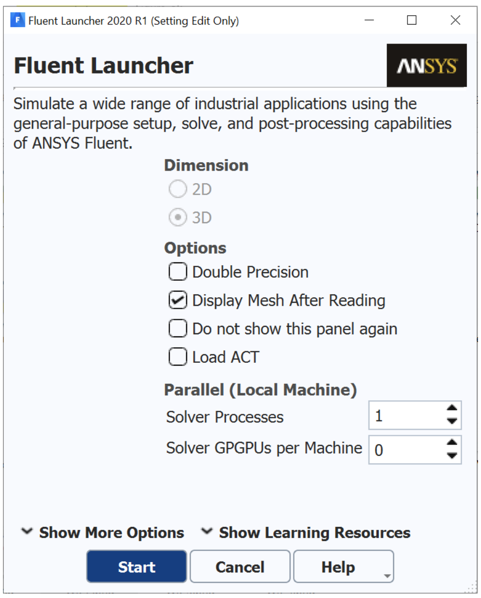


Figure 33: Fluent settings.

— Under the *Tree* tab → *Setup* → *General*, click on *Check* and *Report Quality* to check the mesh quality and set the marked settings as in Figure 34:

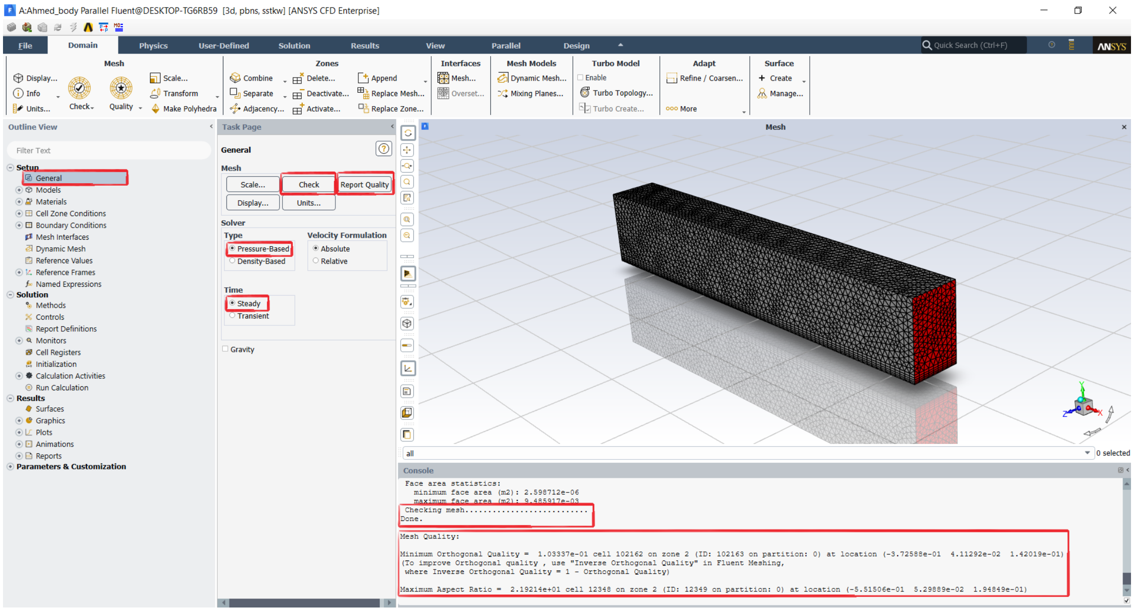


Figure 34: Fluent general Settings.

— Under *Setup* → *Models* double click on *Viscous - SST k-omega* to specify a *Realizable k-ε* turbulence model with *Non-Equilibrium Wall Functions* as in Figure 35:

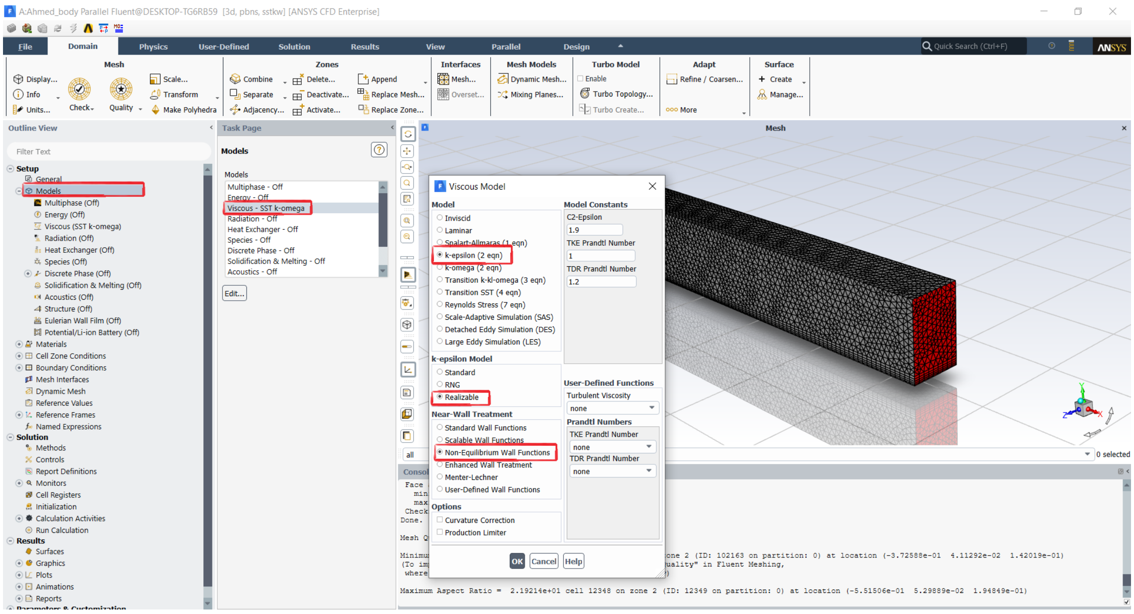


Figure 35: Turbulence model.

– Under *Setup* → *Materials* → *Fluid*, click on *air* to check the correct values of Density and (Dynamic) Viscosity as in Figure 36:

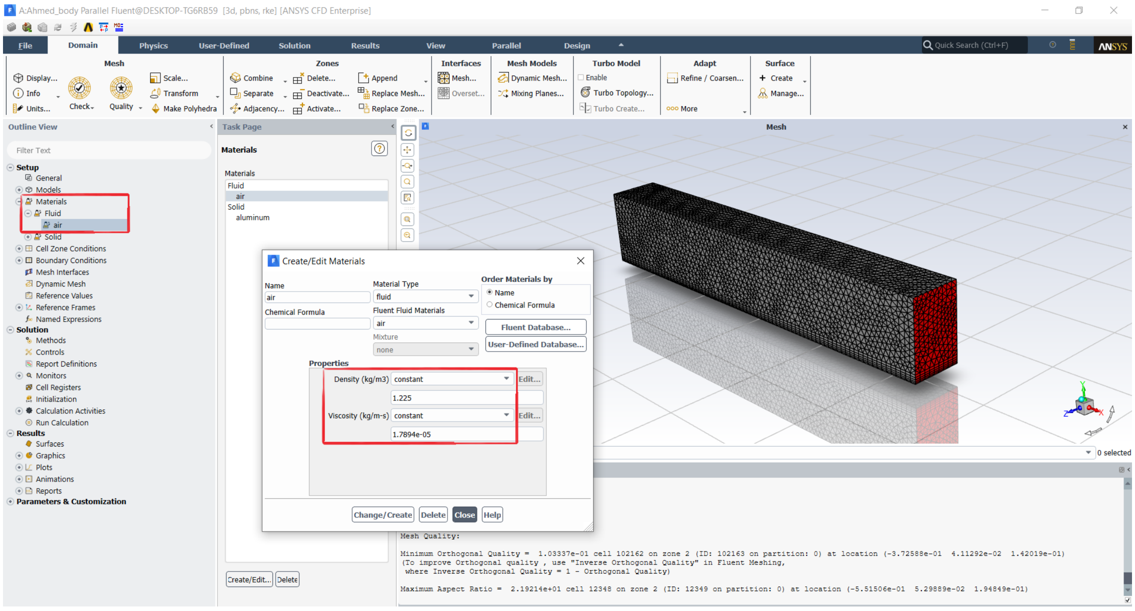


Figure 36: Air properties.

– *Setup* → *Cell Zone Conditions*, double click on *enclosure_enclosure* to check our enclosure is filled with *Fluid* air as in Figure 37:

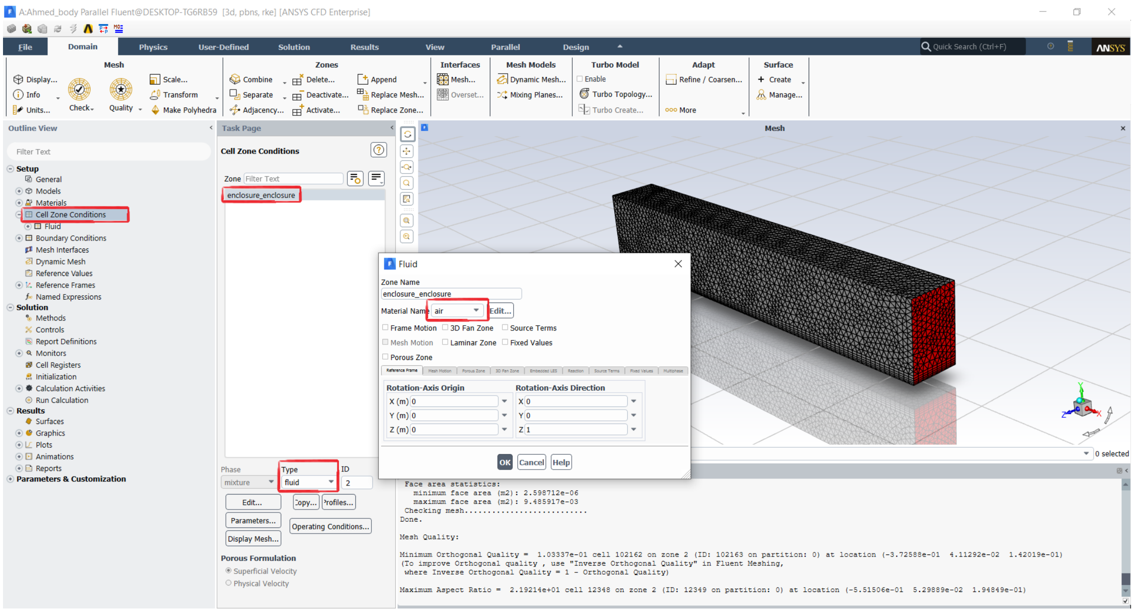


Figure 37: Cell zone conditions.

– Under *Setup* → *Boundary Conditions*, right click on the chosen Named Selection → *Type* to set the correct boundary condition (Figure 38): *side, symmetry* and *top* boundary conditions must be set to symmetry type (no shear stress), *ahmed_surface* and *ground* boundary conditions must be set to wall;

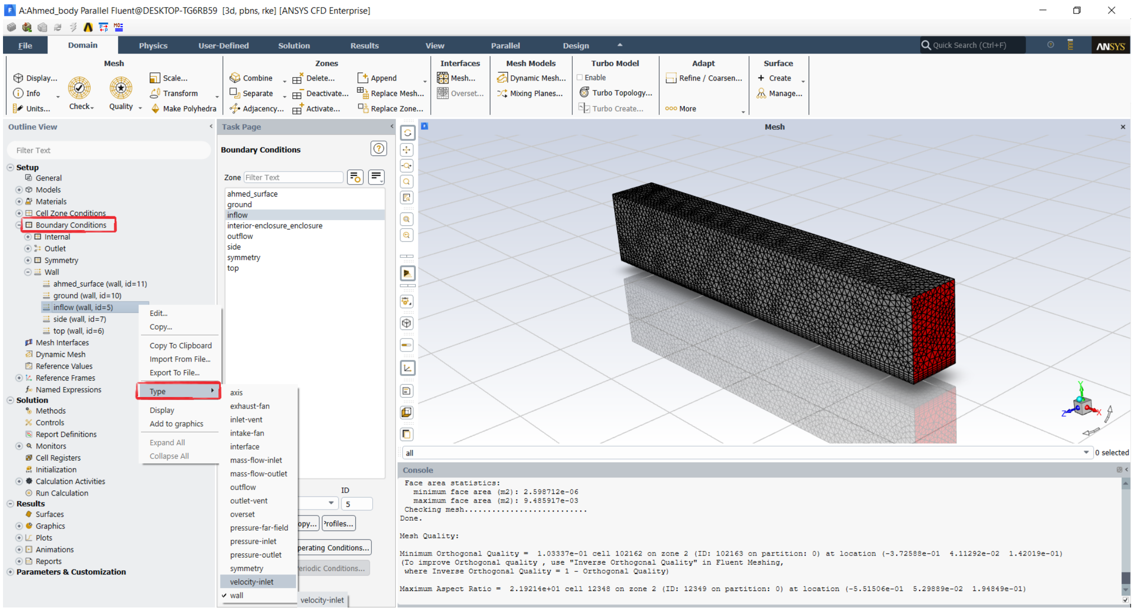


Figure 38: Boundary condition types.

– For *inflow*, set it as *velocity-inlet* type with the marked settings of Figure 39:

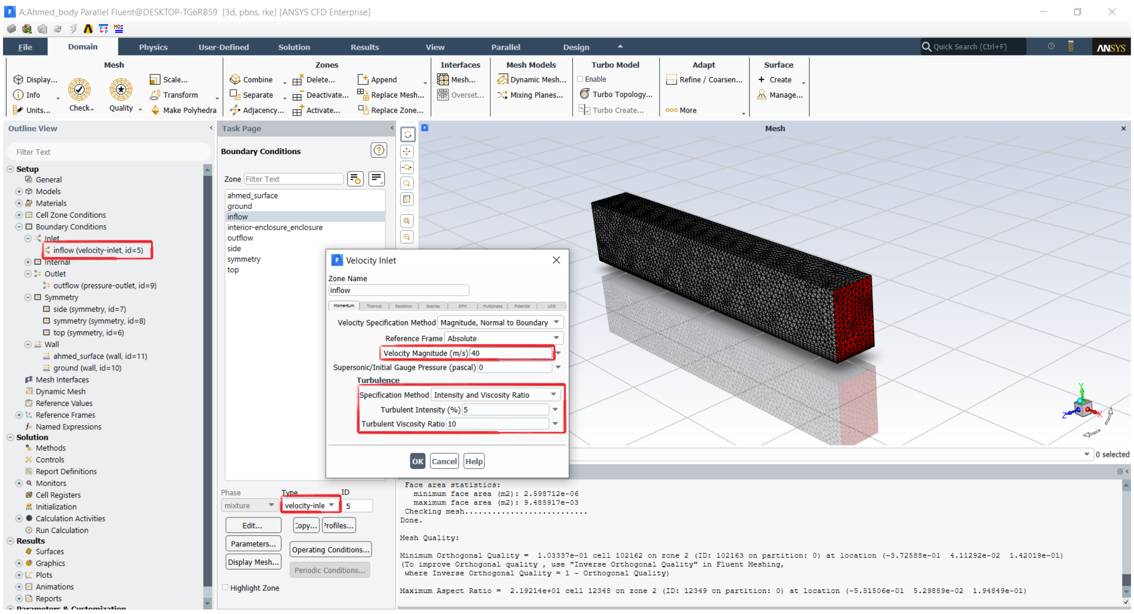


Figure 39: Inflow conditions.

– For *outflow*, set it as *pressure-outlet* type with the marked settings of Figure 40:

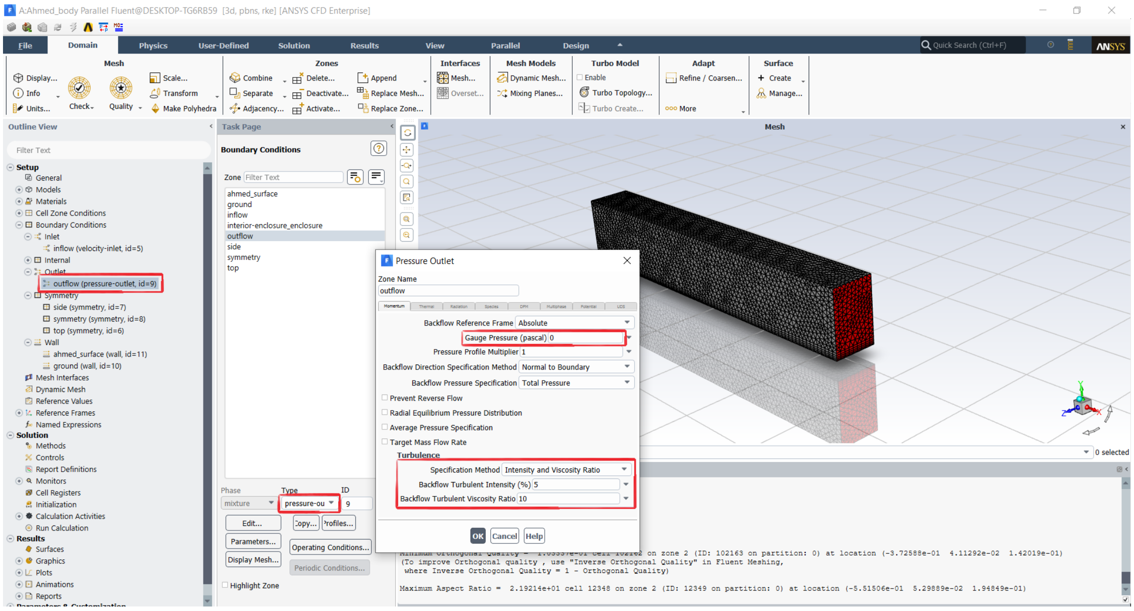


Figure 40: Outflow conditions.

– Under *Setup* → *Reference Values*, specify *inflow* under *Compute from* and $A_x = 0.057516$ m^2 as *Area* (Figure 41):

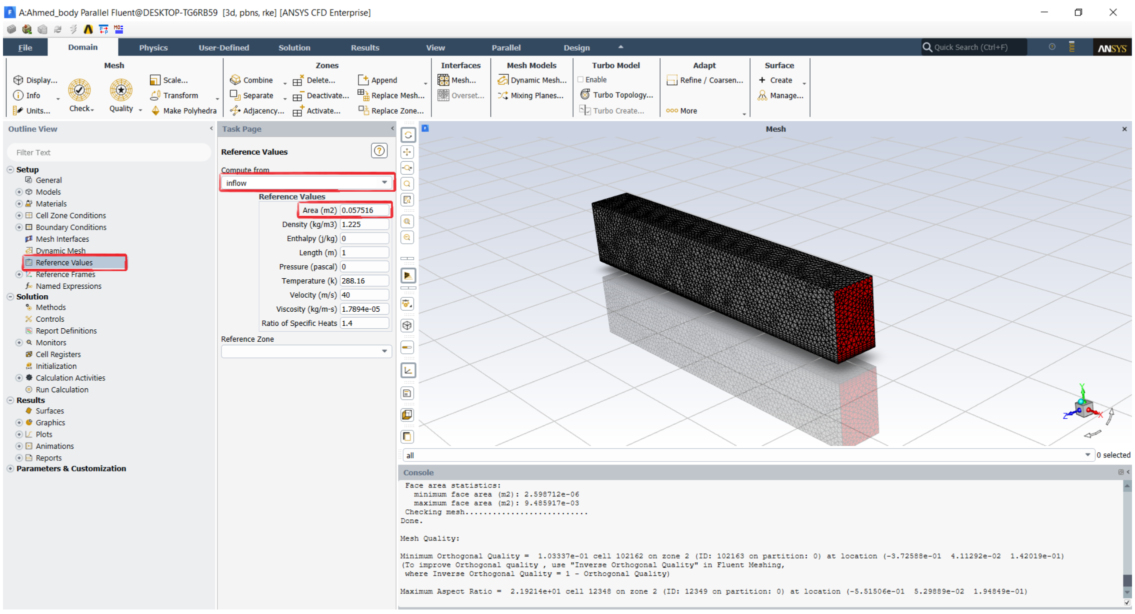


Figure 41: Reference values.

– Under *Solution* → *Methods* select the *Coupled* method for the Pressure-Velocity coupling and *First Order Upwind* (for now) for Momentum, Turbulent Kinetic Energy and Turbulent Dissipation Rate (Figure 42):

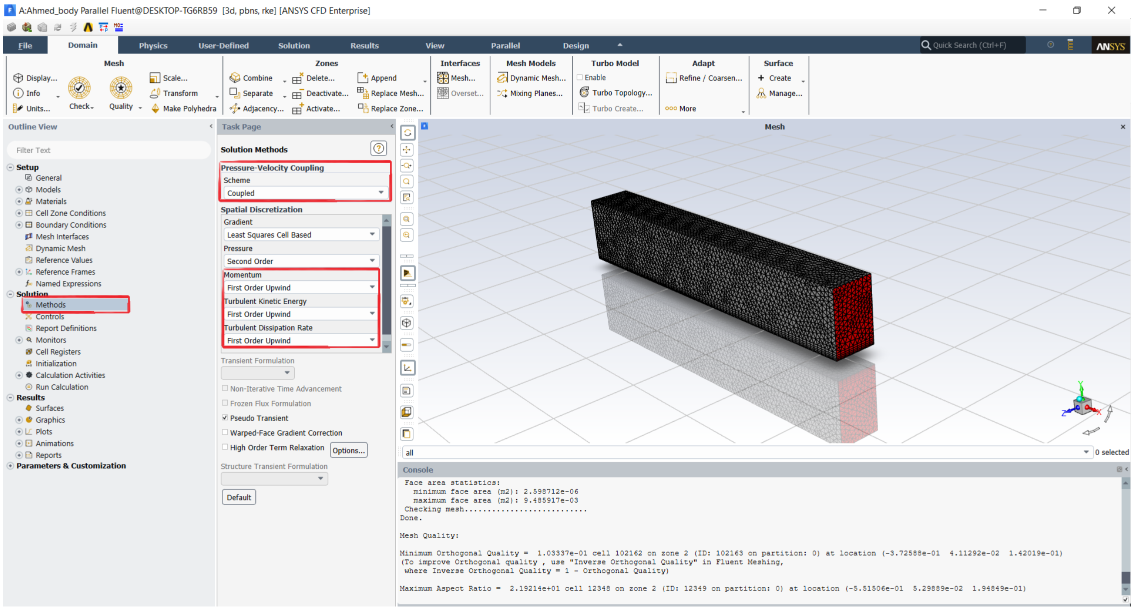


Figure 42: Solution settings.

– Under *Solution* → *Controls* specify 0.3 for both Momentum and Pressure *Pseudo Transient Explicit Relaxation Factors* (Figure 43):

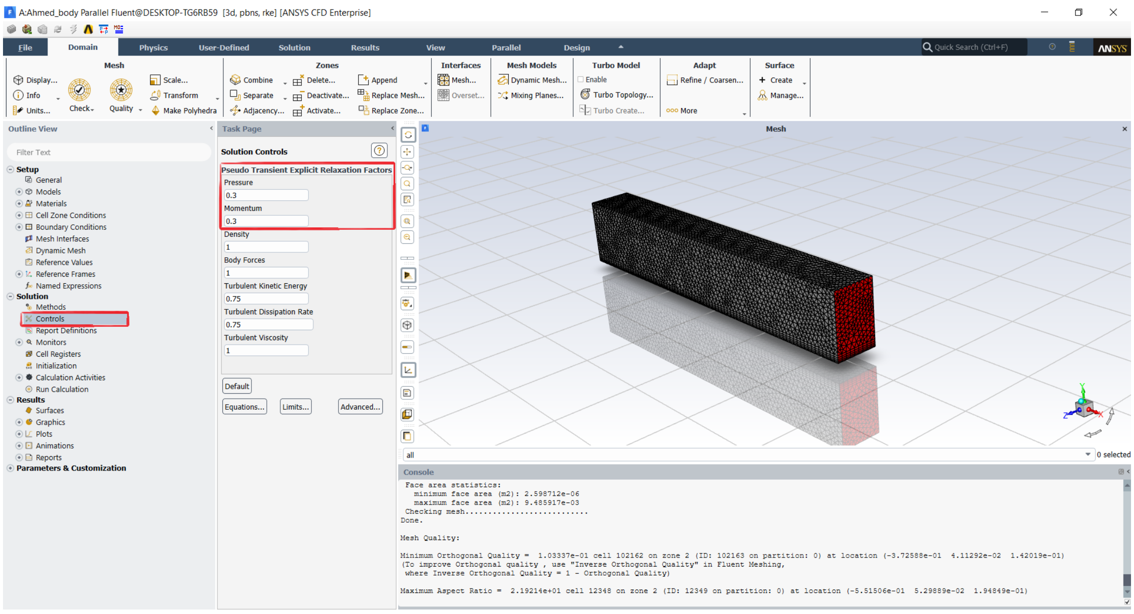


Figure 43: Solution controls.

– Under *Solution* → double click on *Report Definitions* to define new quantities to be reported; select *New* → *Force Report* → *Drag* to define a drag report (Figure 44):

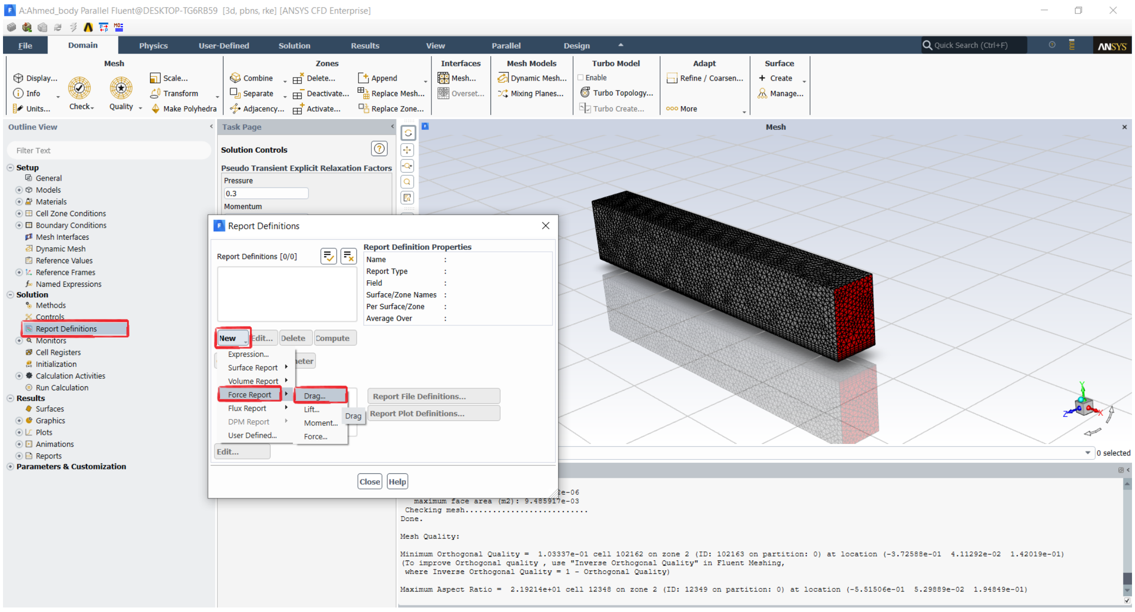


Figure 44: Drag report definition.

– Under *Drag Report Definition* specify the settings of Figure 45 to define a drag coefficient report:

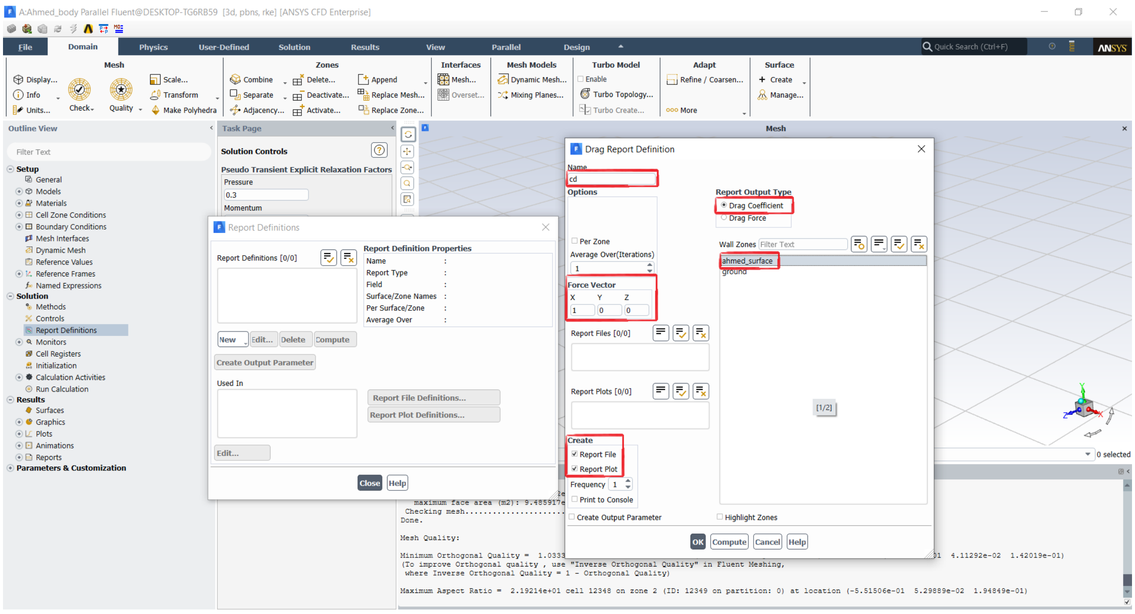


Figure 45: Drag report definition.

– Repeat the previous procedure to define a *Lift Report Definition* if needed.

— Under *Solution* → *Monitor* → *Residual* tick *Show Advanced Options* and set the *Convergence Criterion* to none since we'll define a fixed number of iterations (Figure 46):

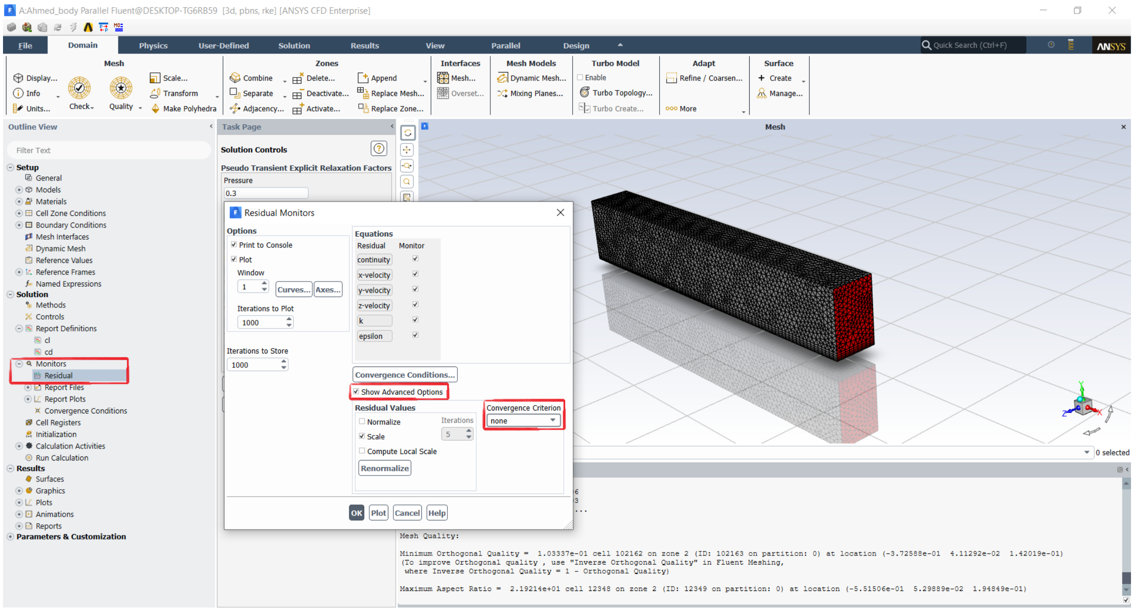


Figure 46: Residual monitors settings.

— Under *Solution* → *Initialization* select *Hybrid Initialization* and click *Initialize* (Figure 47):

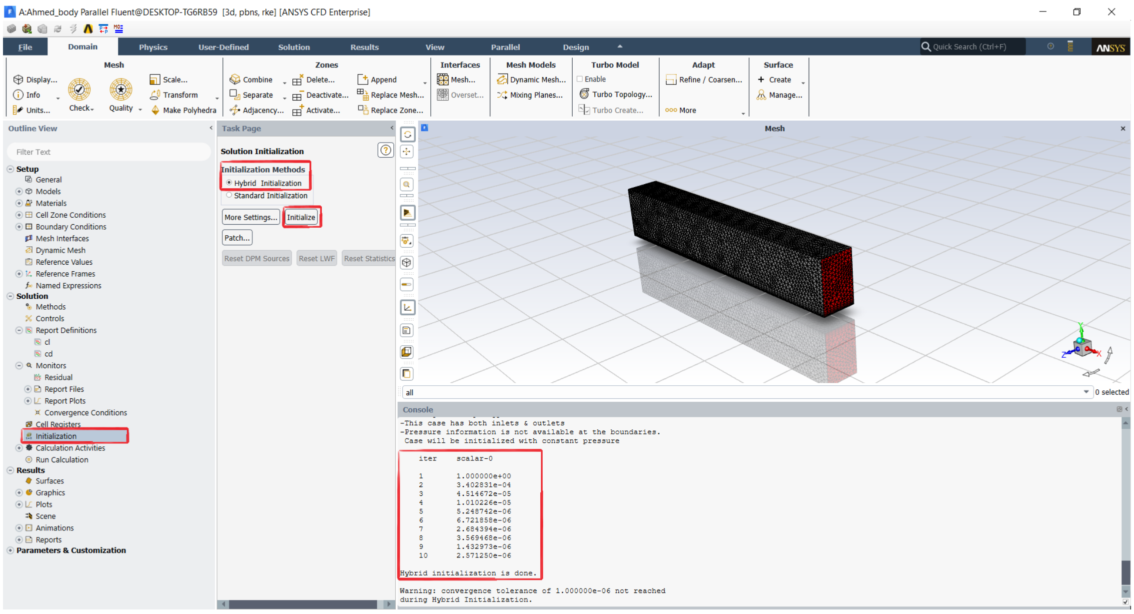


Figure 47: Solution initialization.

– Under *Solution* → *Run Calculation* set 50 as *Number of Iterations* and click *Calculate* (Figure 48); anyway, we can stop the calculation before the 50 iterations if we believe the solution to be acceptable (checking residuals or report plots, for example).

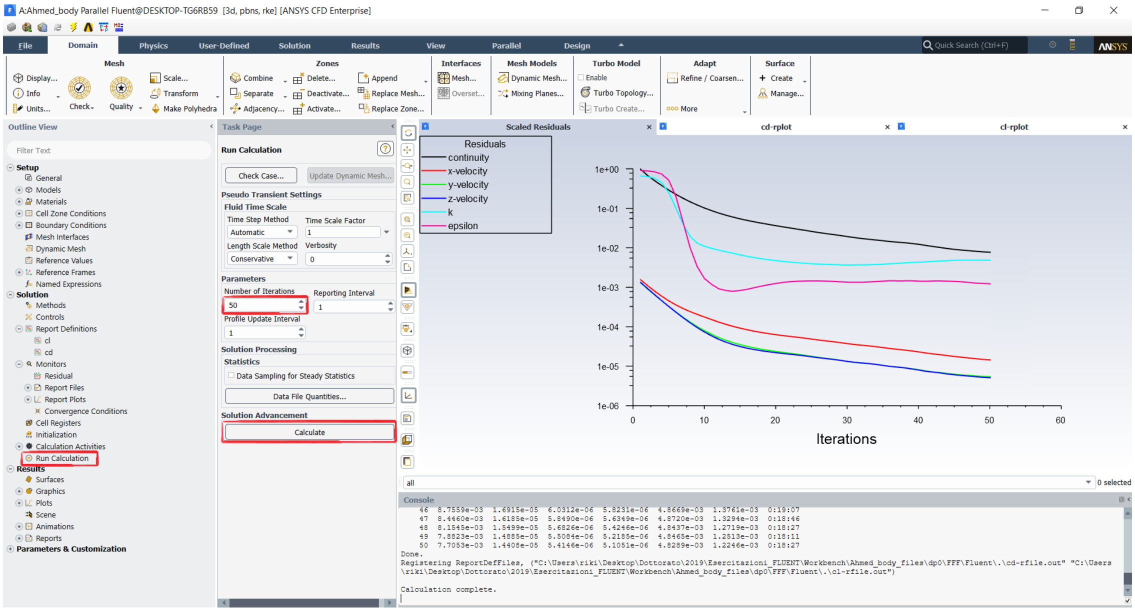


Figure 48: Residual history plot.

– When the solution "converged" (we are employing first order upwind schemes), we can go back to *Solution* → *methods* to select *Second Order Upwind* for Momentum, Turbulent Kinetic Energy and Turbulent Dissipation Rate (Figure 49) to improve the calculations (less numerical viscosity dissipation); run the simulation for 200 additional iterations for example.

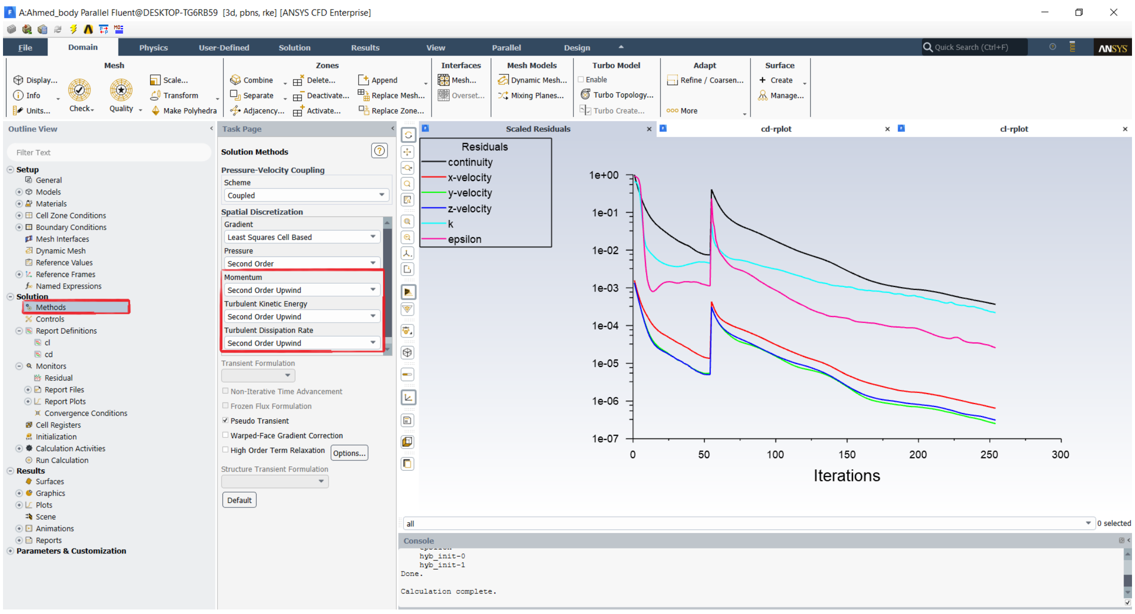


Figure 49: Switching to Second Order Upwind.

The history plot of drag coefficient C_D is reported in Figure 50; we can see that with First Order Upwind scheme (=till 50 iterations) for all the transported quantities (velocities and turbulent quantities), C_D "converged" to 0.45, while the Second Order Upwind scheme (=above 200 iterations) decreased C_D to 0.334. This is in perfect agreement with the upwind schemes: first order scheme is more diffusive than second order and simulates a higher viscosity flow with higher drag coefficients.

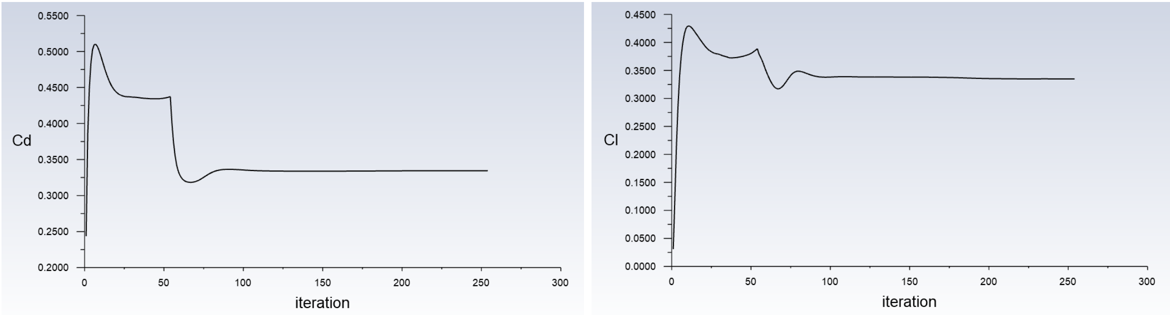


Figure 50: Drag & lift coefficients history plot.

The comparison of the computed drag coefficient to other experimental and numerical results is reported in Figure 51; as we can see our result is in good agreement with both experimental and numerical data from literature, even if it is computed through a RANS simulation with a not so refined mesh.

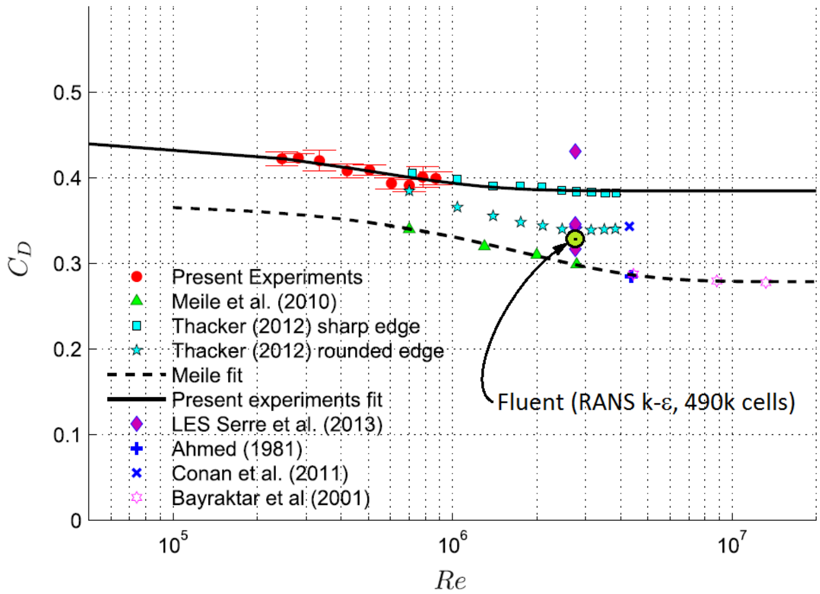


Figure 51: Computed drag coefficient vs other results (from [2]).

– *File* → *Save Project* and close Fluent.

2.4 Visualization of results in CFD-Post

In WB, right click on *Results* → *Edit* to launch CFD-Post (Figure 52):

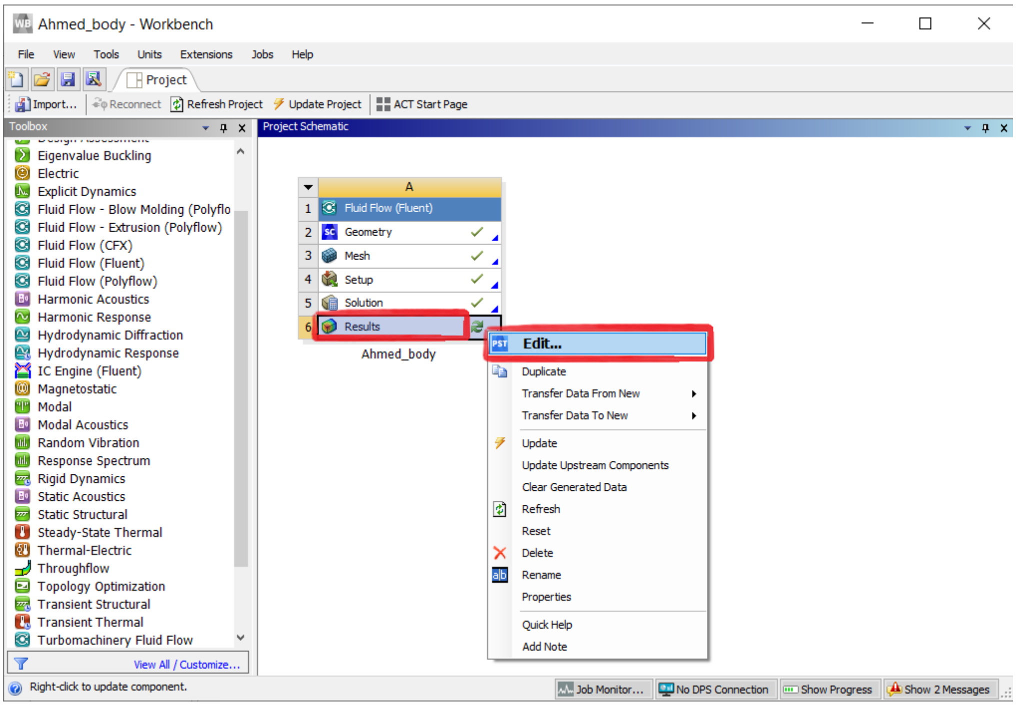


Figure 52: Starting CFD-Post from WB.

– In the *Outline* tab, under *Cases* → *Ahmed_body* → *enclosure-enclosure*, select only the *ahmed_surface* and the *ground* components. Click on the *Location* icon Location and select *Line*, then specify the marked settings as in Figure 53 → *Apply*:

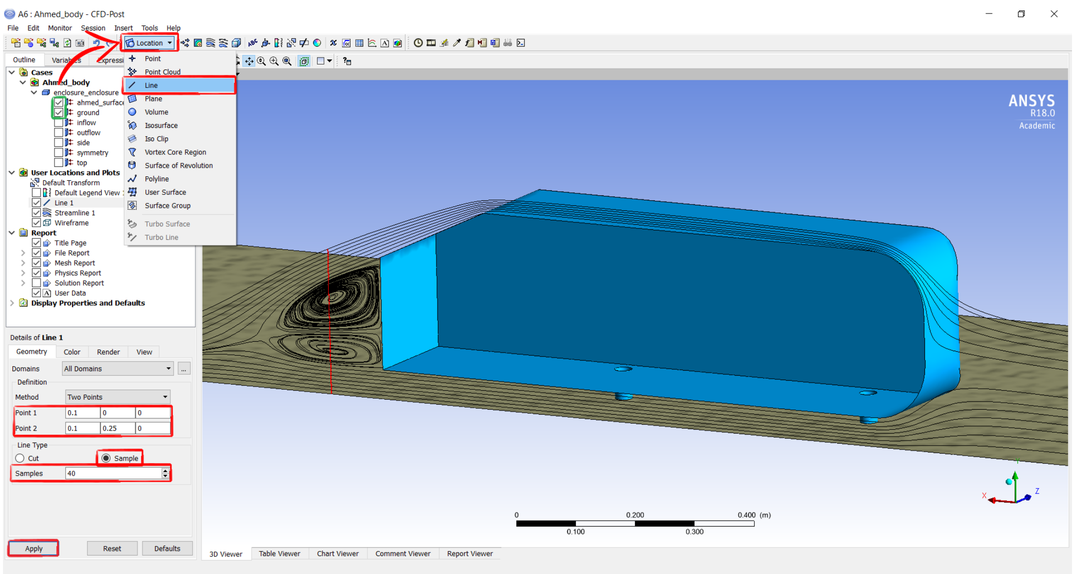


Figure 53: Location definition.

– Click on the *Streamline* icon and specify the marked settings as in Figure 54 → *Apply*, to view the streamlines in the symmetry plane in the rear separation zone:

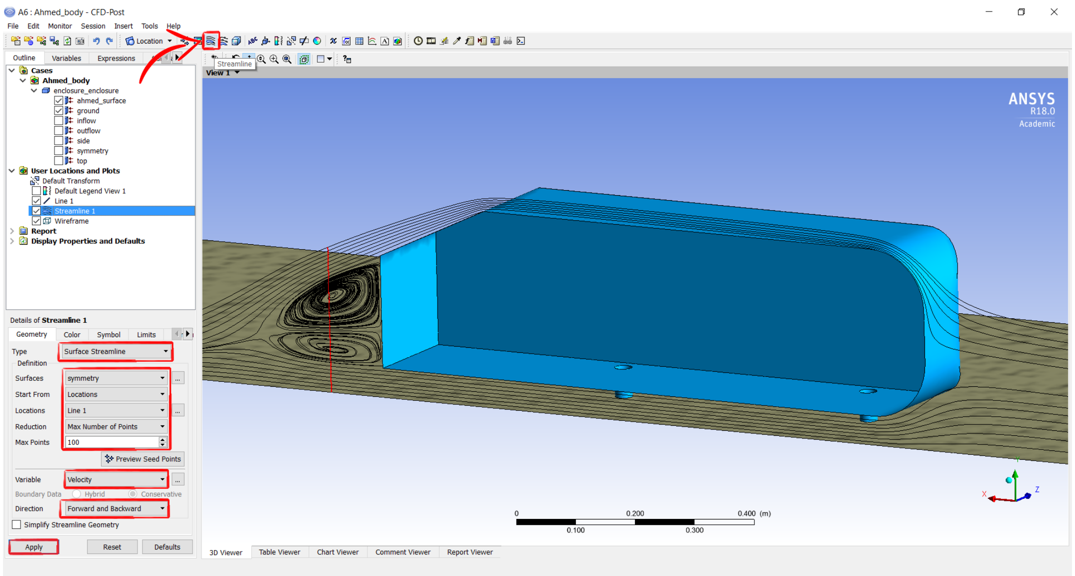


Figure 54: Streamline definition.

A comparison between computed and experimental (mean) streamlines along the symmetry plane in the rear separation zone is reported in Figure 55: although the relative position of recirculation bubbles is not exactly computed, we can see that an approximate width of 200mm is computed with good agreement to the experimental width.

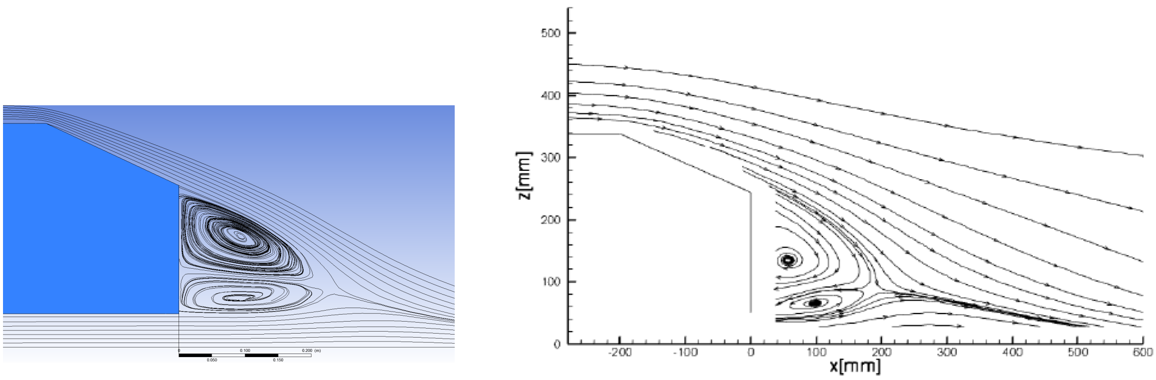


Figure 55: Comparison of streamlines in the rear separation zone (from [4]).

A comparison between computed and experimental (mean) streamwise velocity profiles along the symmetry plane in the rear separation zone is reported in Figure 56: the computed profiles show a discrete agreement with experimental ones.

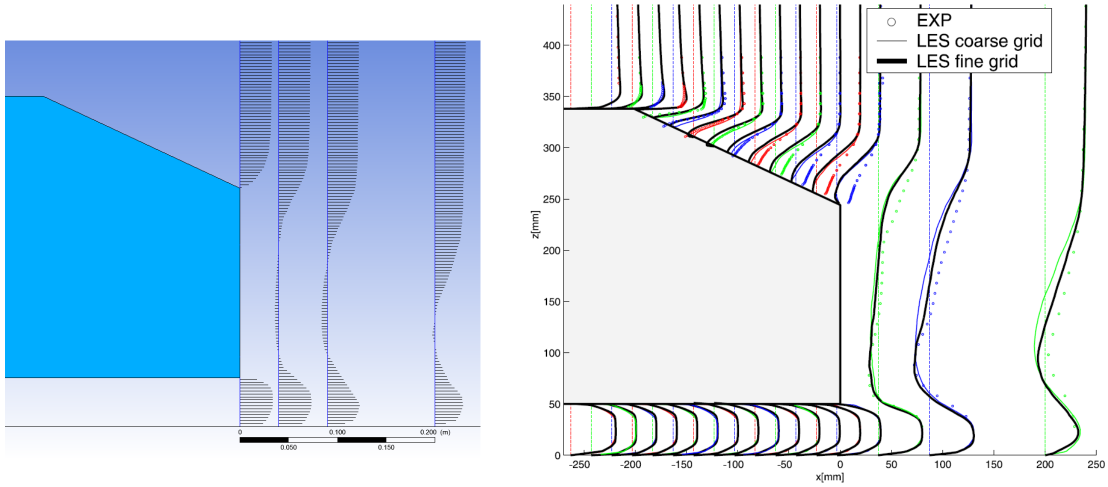


Figure 56: Comparison of streamwise velocity profiles in the rear separation zone (from [3]).

CFD-Post is a powerful tool to analyze and display the results of a CFD simulation in a graphical way; be careful to choose a meaningful way to present your results (see Figure 57).

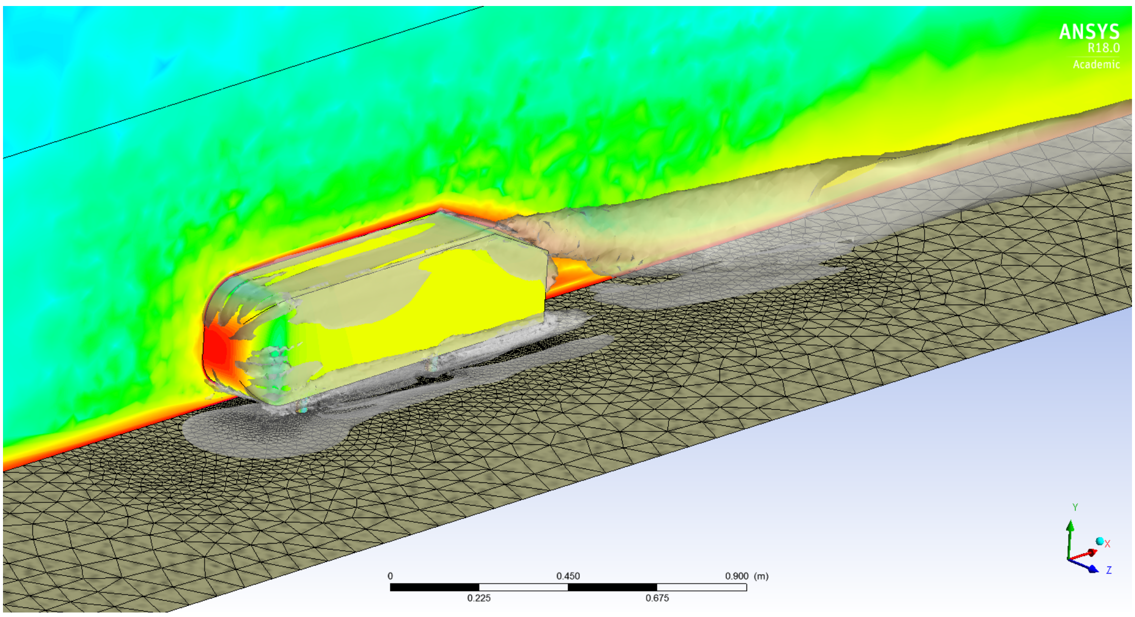


Figure 57: Suggestive CFD image with CFD-Post.

References

[1] S.R. Ahmed, G. Ramm, and G. Faltin. Some salient features of the time-averaged ground vehicle wake. In *SAE Technical Paper*. SAE International, February 1984.

[2] F.J. Bello, T. Makela, L. Parras, C. del Pino, and C. Ferrera. Experimental study on ahmed's body drag coefficient for different yaw angles. *Journal of Wind Engineering and Industrial Aerodynamics*, 157:140 – 144, 2016.

[3] C. Hinterberger, M. Garcia-Villalba, and W. Rodi. Large eddy simulation of flow around the Ahmed body. In R. McCallen, F. Browand, and J. Ross, editors, *Lecture Notes in Applied and Computational Mechanics/The Aerodynamics of Heavy Vehicles: Trucks, Buses, and Trains*. Springer, New York, 2004.

[4] H. Lienhart, C. Stoots, and S. Becker. *Flow and Turbulence Structures in the Wake of a Simplified Car Model (Ahmed Model)*, pages 323–330. Springer Berlin Heidelberg, Berlin, Heidelberg, 2002.