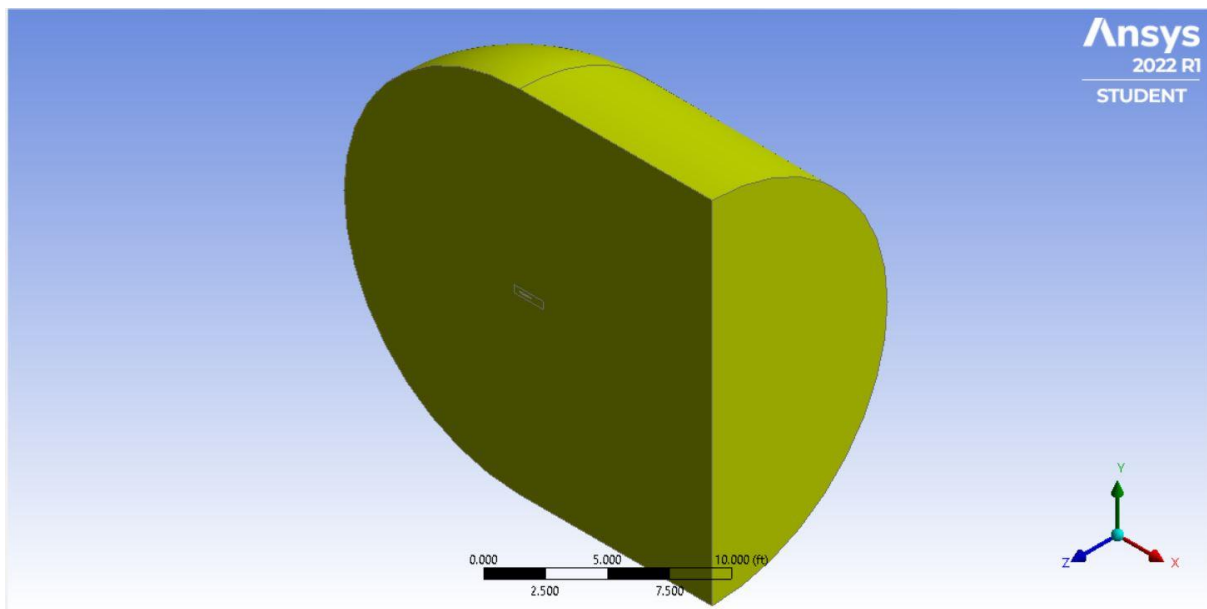
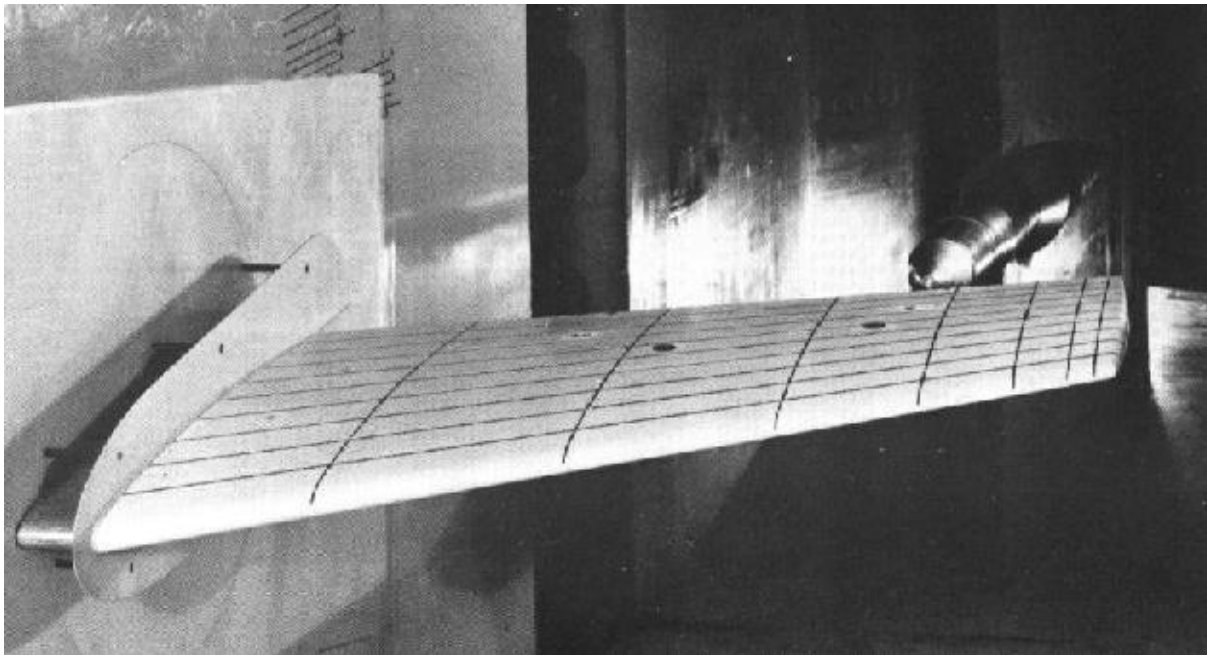


Transonic flow over 3D wing.

To understand flow over a wing is crucial to understand the aerodynamical performance. In current study we have considered an M6 Onera wing. As multiple experimentation is done on this wing the data to validate our simulation is available and will provide as a benchmark for our setup.

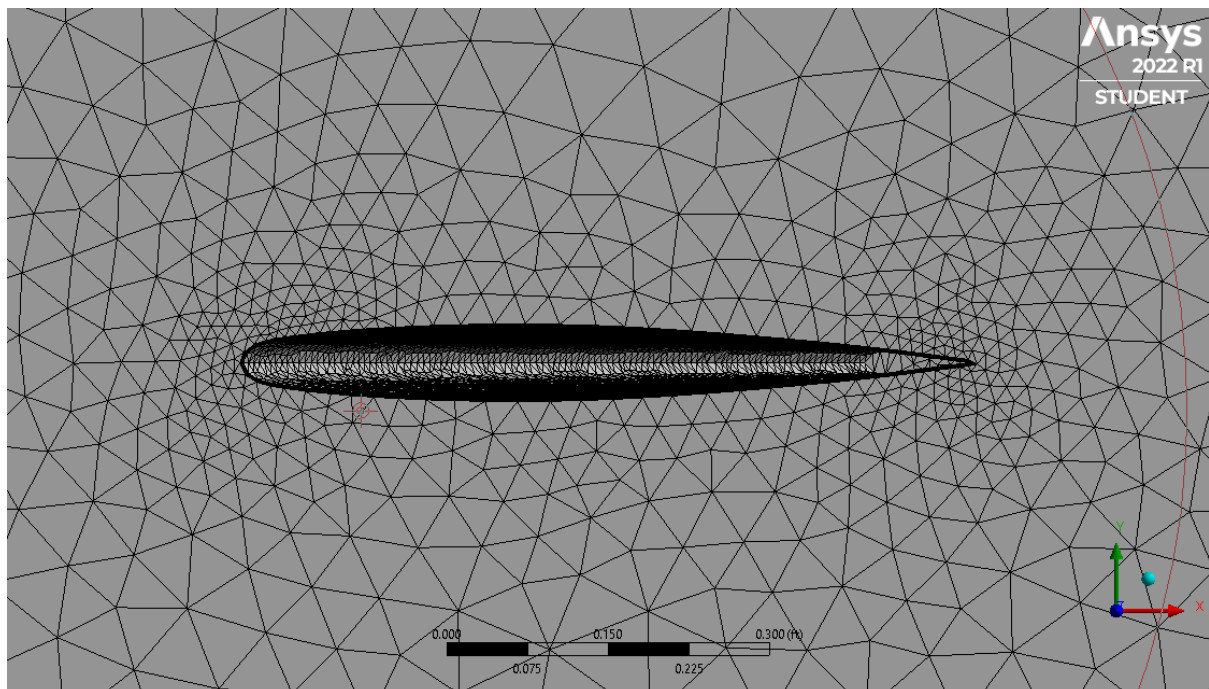
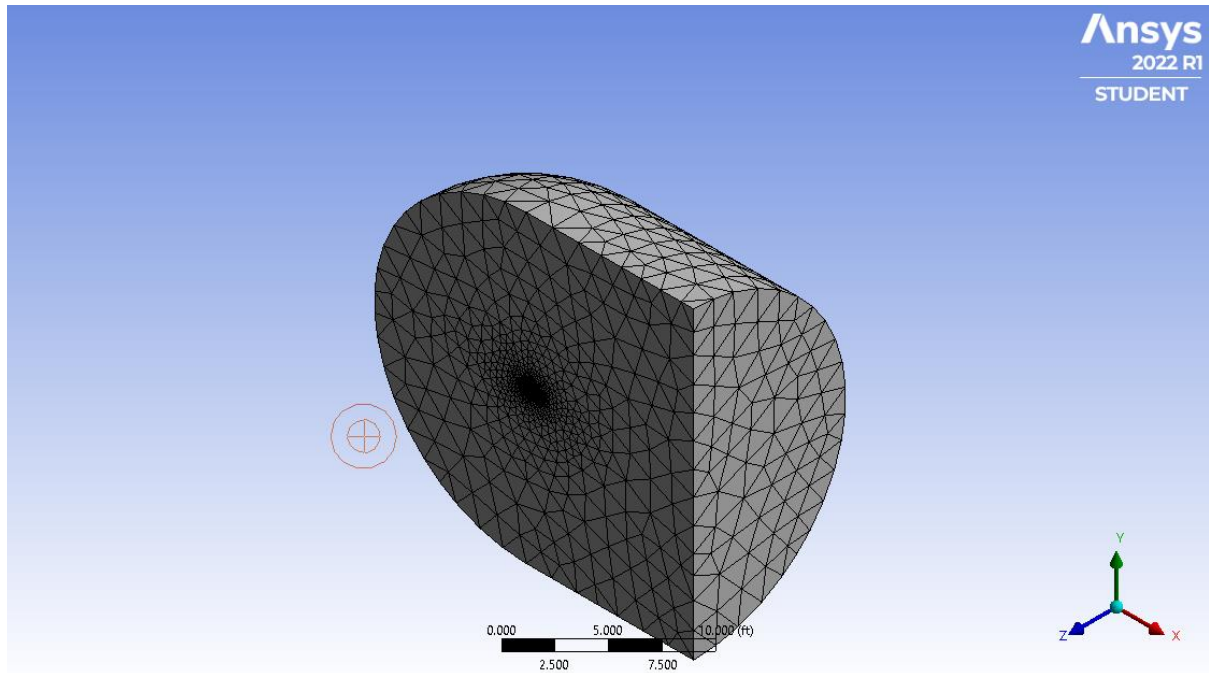
Given below is the image of the wing model which was used for the experiment. And the image below it is the domain created around the model of a wing for our study in ANSYS Design Modeler.



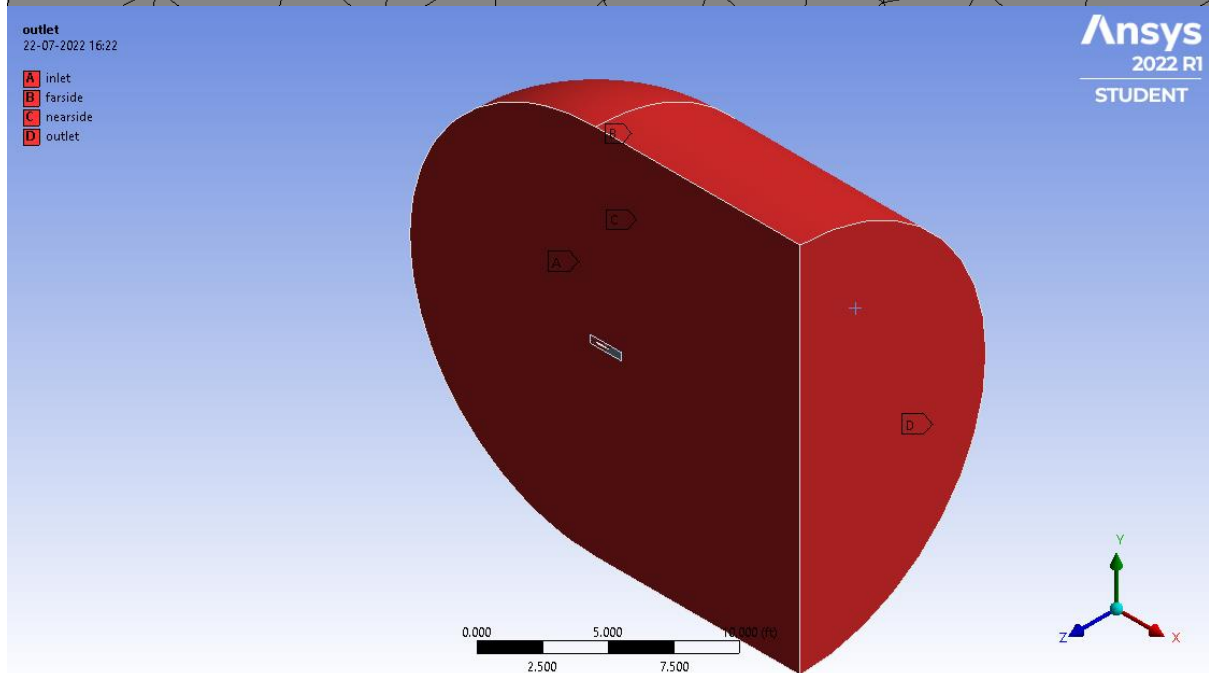
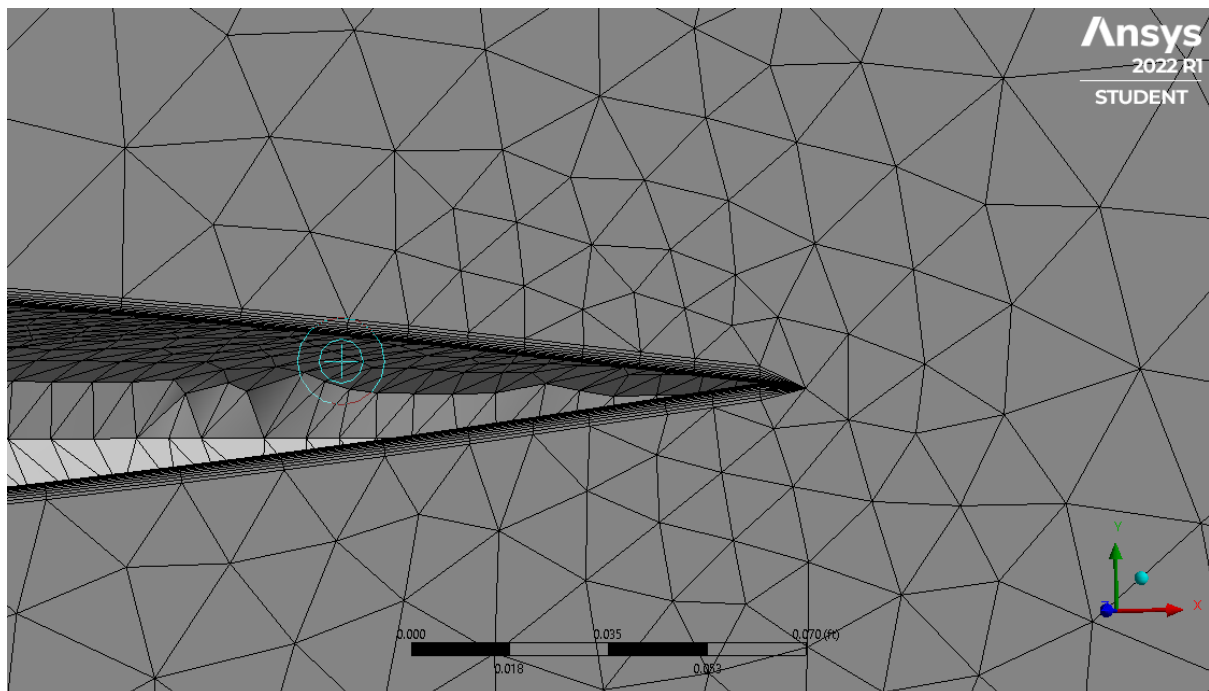
Meshing:

Meshing is crucial task for any simulation as it ensures the accuracy of the numerical result. Here simple meshing is done where a **Frozen (in design modeler)** body is created around the wing which acts as a body of influence. This body of influence will create smaller elements near the wing to have better accuracy while calculating the Drag, Lift and Coefficient of pressure.

Next, an inflation layer is created which makes prism elements which further improves the accuracy.



Inflation layer: (10 *layers*)



Setup: The viscous solver used for this study is **Spalart-Allamaras** (1 equation model). This model was designed for external aerodynamic simulation and has been performing well so we use this model. As we are simulating for a **mach number of 0.86** we enable the energy equation. Here, we use a **pressure-based solver** for the simulation as the mach number is lower and density-based solver shouldn't make much difference.

Furthermore, the Boundary conditions are as follows:

Inlet, Outlet, Far-side (Angle of attack 3)	<ul style="list-style-type: none"> • <i>Pressure = 45.826</i> • <i>Mach number = 0.8395</i> • <i>X-cord = 0.9986</i> • <i>Y-cord = 0.0534</i>
Near-side	<i>Symmetry</i>
Wing	<i>Wall (no-slip)</i>

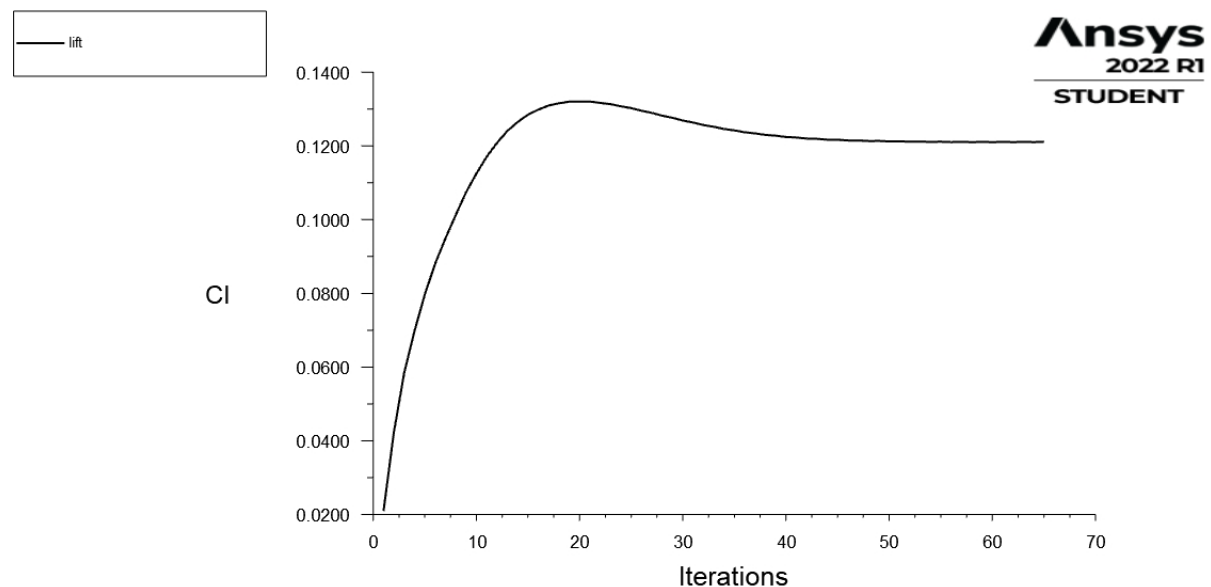
A **coupled solver** is used for the pressure-velocity coupling.

Report definitions for Coefficient of drag and lift are created with Coefficient of pressure and Mach number are exported as solution data.

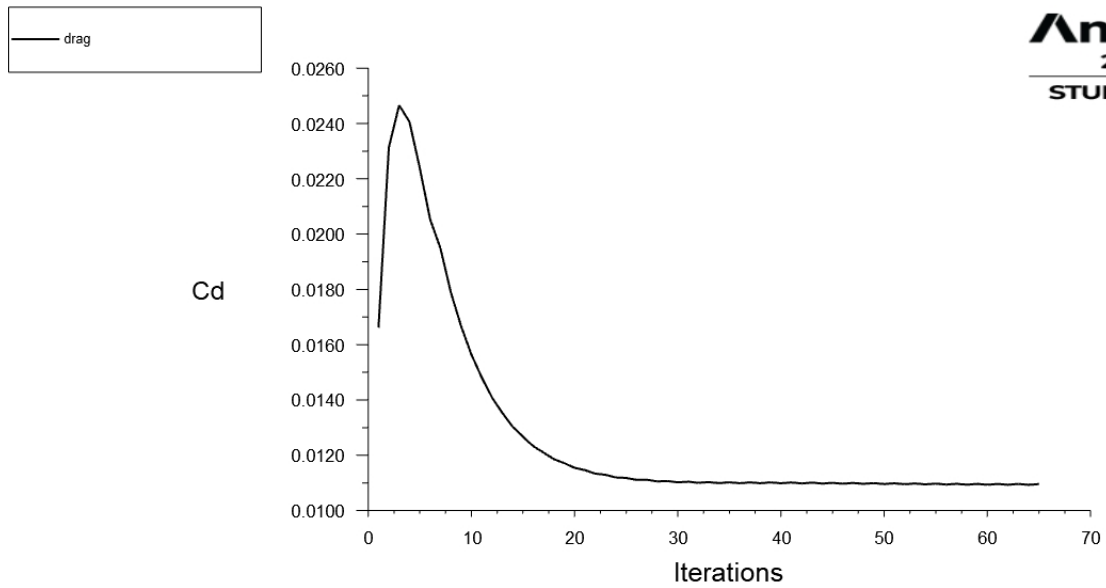
Results:

After standard-initialization and solving for 600 iteration we get a convergence. The results are provided below:

Coefficient of lift:



Coefficient of Drag:



Ansys
2022 R1
STUDENT

Computed values for lift and drag:

drag		

	Cd	

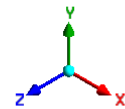
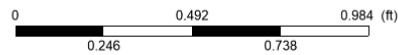
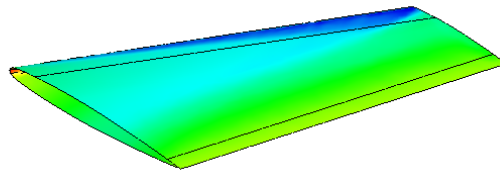
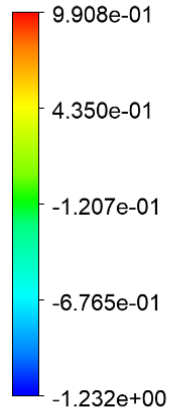
	drag	0.010929451
lift		

	Cl	

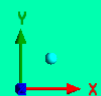
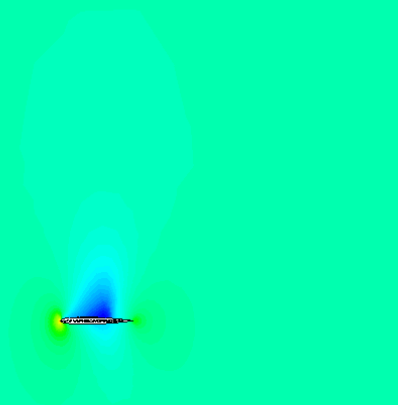
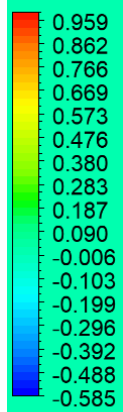
	lift	0.12105288

Coefficient of pressure on the wing: Animation: [Coefficient of pressure](#)

Pressure Coefficient
trailing_edge

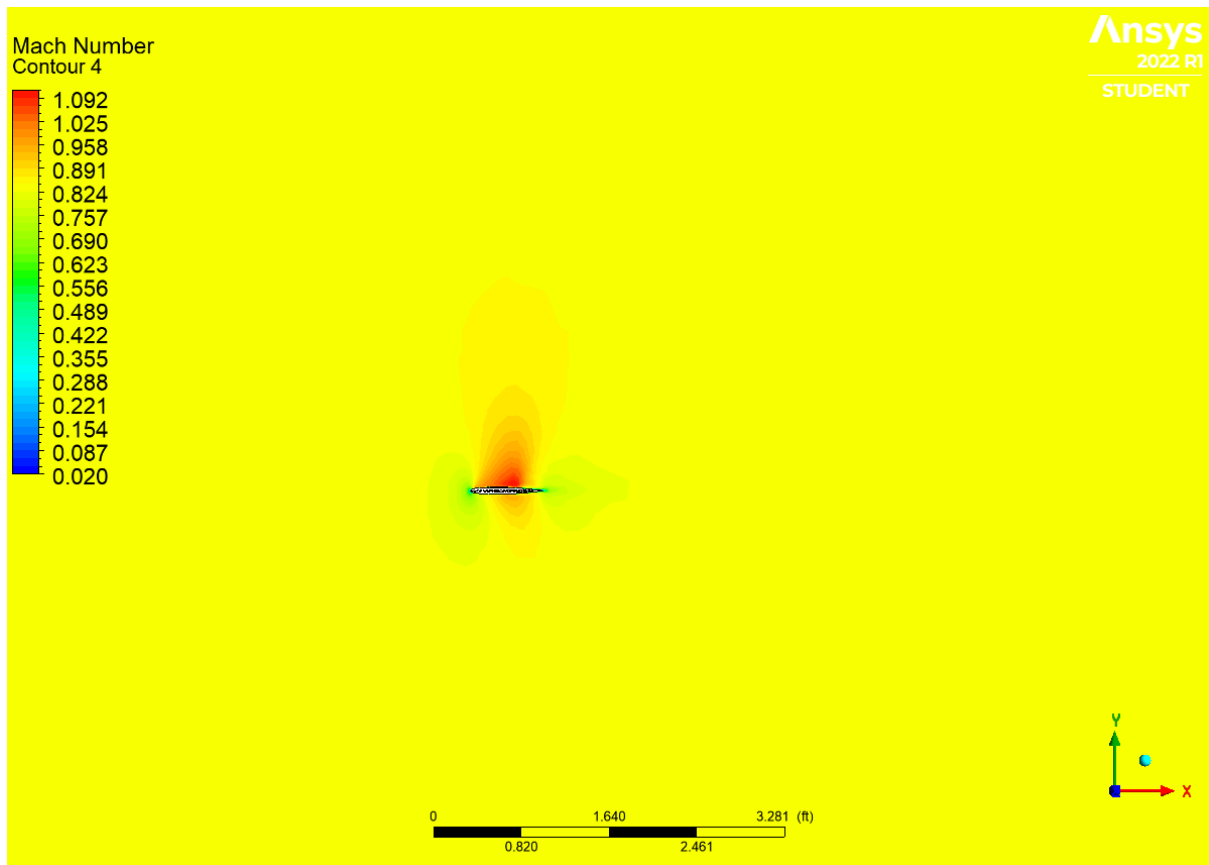
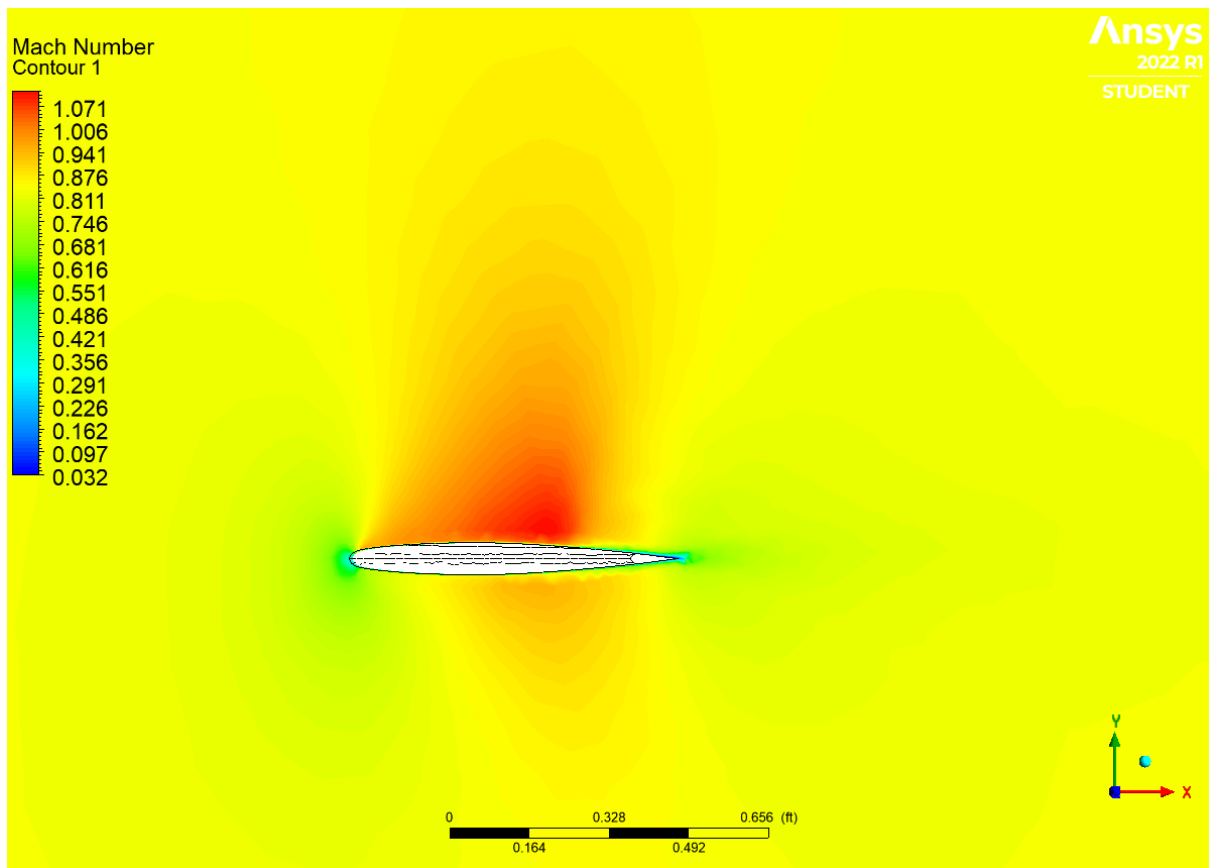


Pressure Coefficient
Contour 4

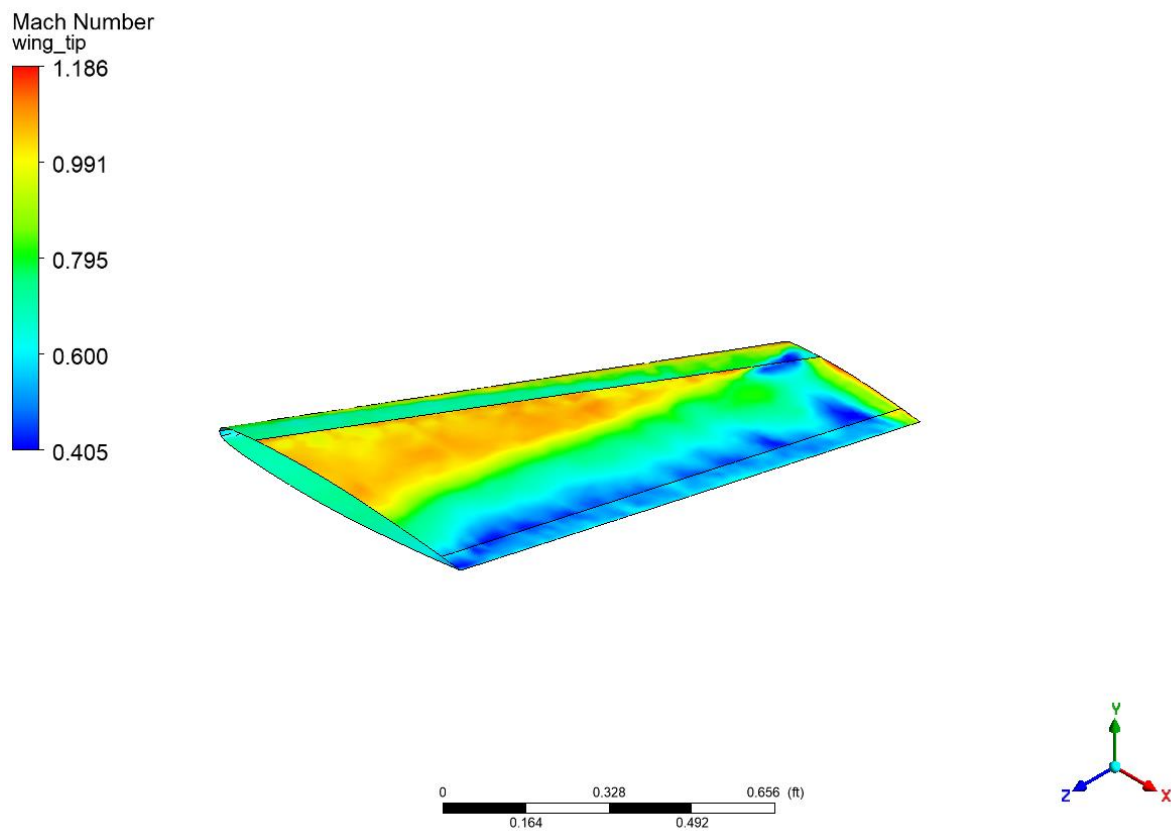


Ansys
2022 R1
STUDENT

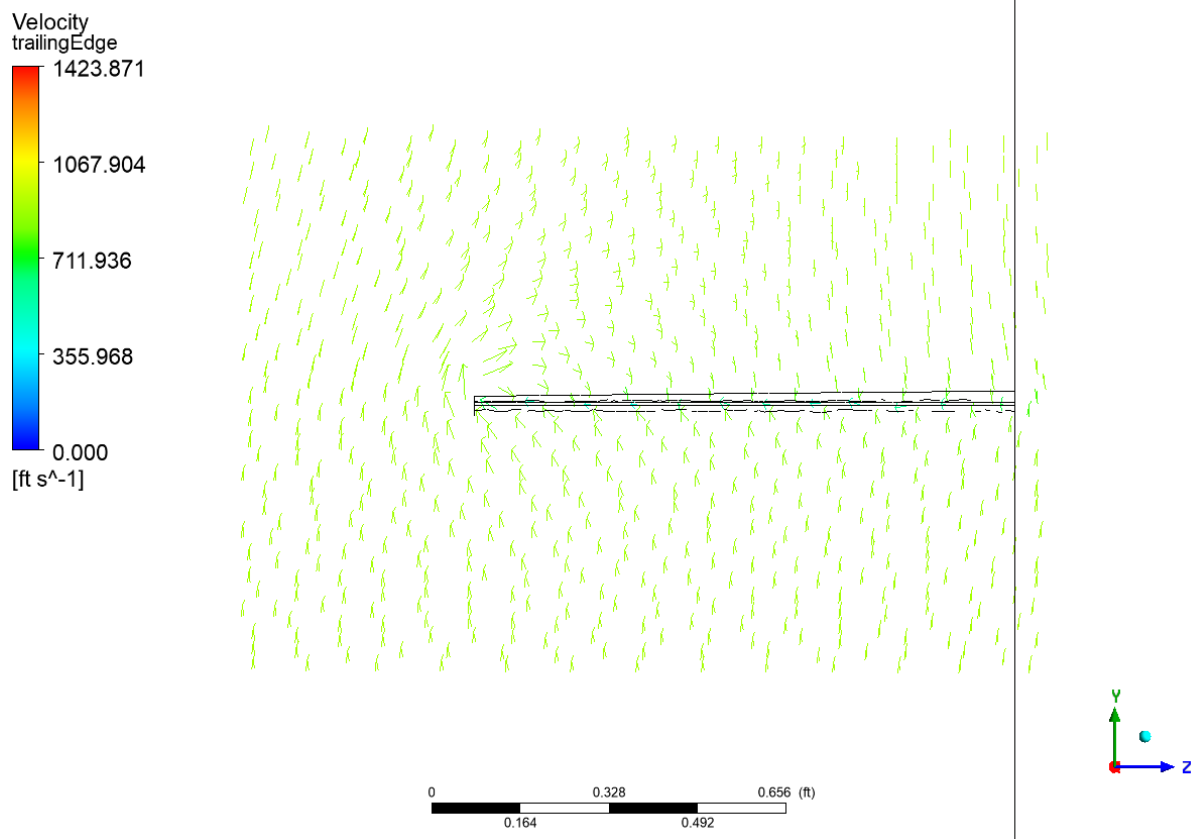
Mach number contour plot:



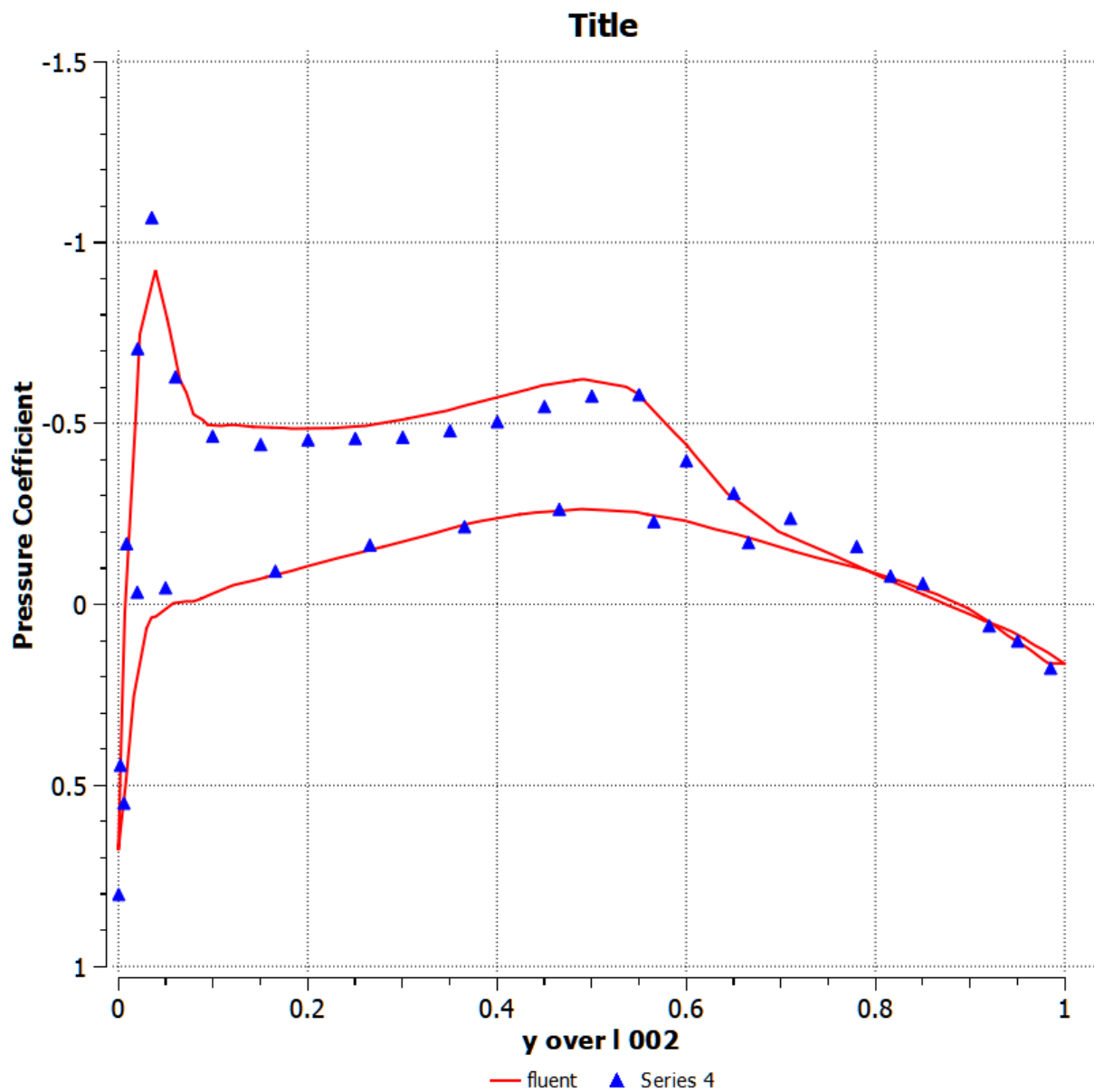
Mach number on the wing:



Vectors of velocity at the trailing edge of the wing:

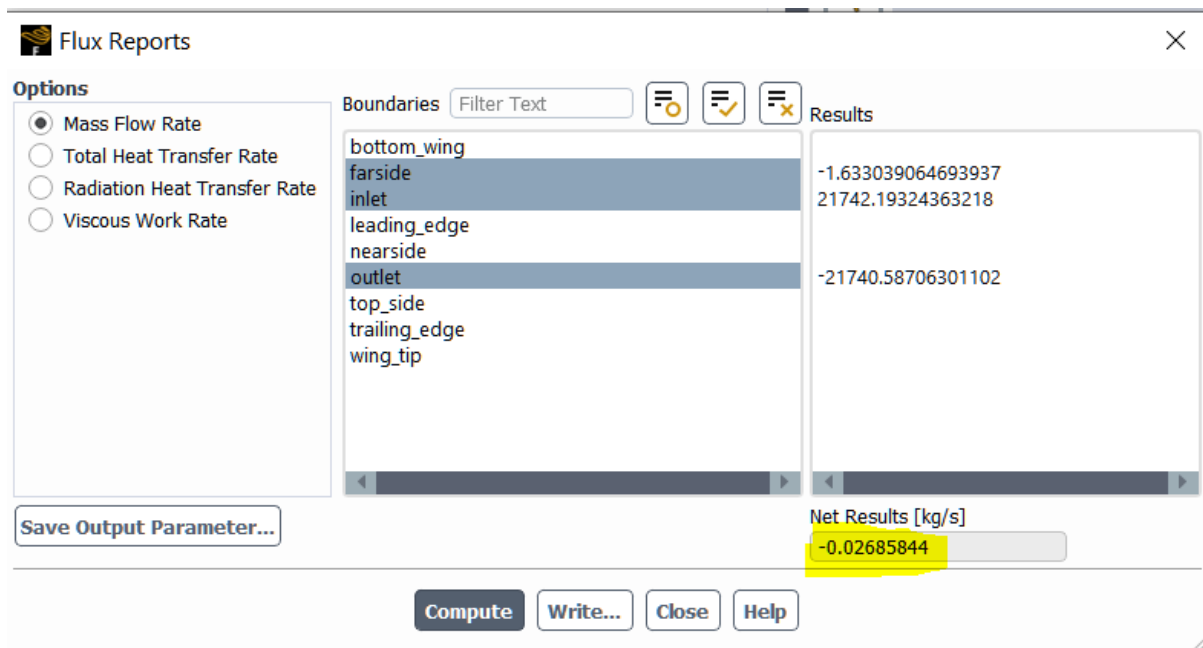


Validation was done with the data provided by the experimental result:



The y/b line at location 0.2ft for Coefficient of Pressure was done. As one can see a close alignment with the experimental values. Here, Series 4 is the experimental values.

A good practice to see the simulation is good is to check the Mass flow rate incoming and outgoing of the domain.



Although the value is lower, it can go lower with a good mesh. But this is appreciable.

Conclusion: The Onera M6 wing simulation takes us through all steps of the CFD workflow from creating a domain, Meshing and through the setup. This gives a good idea about external flow simulation. As the validation and verification is done for this setup we can safely conclude the simulation is done and good.