F-16 Aircraft’s aerodynamics simulation

-Rohan Sharma
200104092

- Aim:-
  To analyse the fluid flow of F-16 aircraft using computational fluid analysis using Ansys.

- Theory:-
  The F-16 Falcon is a combat multi role fighter aircraft which has proven its mettle in air to air combats as well as air to surface attacks. With a full load of fuel, the F-16 can handle as much as 9Gs which exceeds the capability of current fighters.
  Computational fluid dynamics aka CFD is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyse problems that involve fluid flow.
  The fundamental basis for all the CFD problems is the Navier Stokes equations which define any single phase fluid flow. These equations can be simplified by removing terms describing viscosity to yield the Euler equations.

- Procedure:-
  1. Modelling of F-16 aircraft
     It is a solid model over which the external flow analysis is done. Some of the sketch commands used in this particular aircraft modelling are line, circle, ellipse, spline etc. While modelling this aircraft intensive care must be taken since any improper way of modelling may lead to over lapping of geometry. Hence it should be seen that no two geometrical entities intersect each other.
  2. Meshing of the Continuum

Here, the continuum from Ansys workbench is imported, then the continuum is divided into different parts like inlet, outlet, symmetry, wall, and the body. The required meshing conditions are applied and the continuum is meshed.

For a CFD analysis, the body should be enclosed in a continuum where the boundary conditions are applied. This enclosure around the aircraft is made in Ansys workbench. The enclosure is only made for the half section of the aircraft since the aircraft is symmetrical about the YZ plane. Hence this saves the computational time. The aircraft is subtracted from the enclosure to get the complete continuum.

The aircraft volume is given a fine mesh size of 30mm x 30mm x60mm.
Once the meshing of the continuum is done. It is when exported to Ansys Fluent where in the flow analysis over the F-16 aircraft is done.

3. <u>Simulation of the Continuum</u>
   The simulation of this continuum is done in Ansys Fluent. In this initially the meshing of the continuum is checked and once the software approves it, the models, materials and boundary conditions are set.
   - <u>Model</u>
     The model used for this kind of simulation is K-E model. This is a two equation model in which one equation corresponds to the turbulent kinetic energy (K) and the other is specific dissipation rate (E).
   - <u>Material</u>
     The working fluid in this simulation is air and is considered to act on the aircraft. Only density

is considered as the material property of the air and is constant.

- Boundary Conditions

  The important boundary conditions in an external flow analysis are Mach number or velocity at inlet of the continuum and pressure at the outlet of the continuum. The inlet boundary condition for continuum is given as 'Mach No 1.7' and the velocity is 580m/s and the speed of sound at this attitude is 340m/s.

- Solution

  Once the boundary conditions are set, the solution method and the controls are set for this simulation. The solution method set for this is the Coupled Solver.

- Results and Discussion:-

I. Residual plots:

It can be seen that the residual results have converged at 500 iterations.

II. Pressure plots:

One can see that at some places the pressure is really high. This is due to the presence of sharp edges, corners and flat surfaces.

III. Turbulent kinetic energy plot:

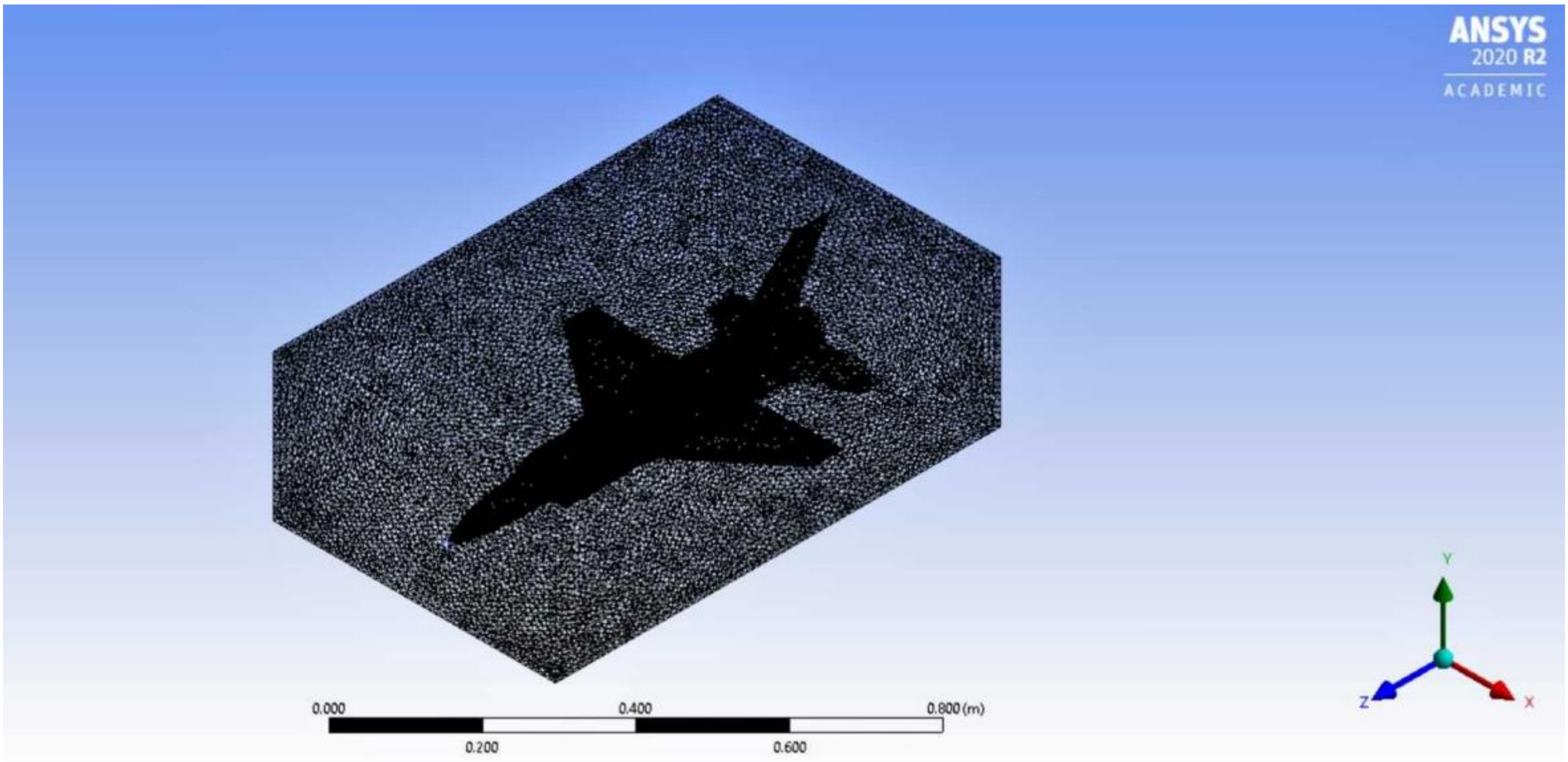
It can be seen that the turbulent kinetic energy increases as the curvature in the shape of the aircraft increase. The turbulent kinetic energy is high at nozzle. This is due to a step provided at the bottom of tail wing.

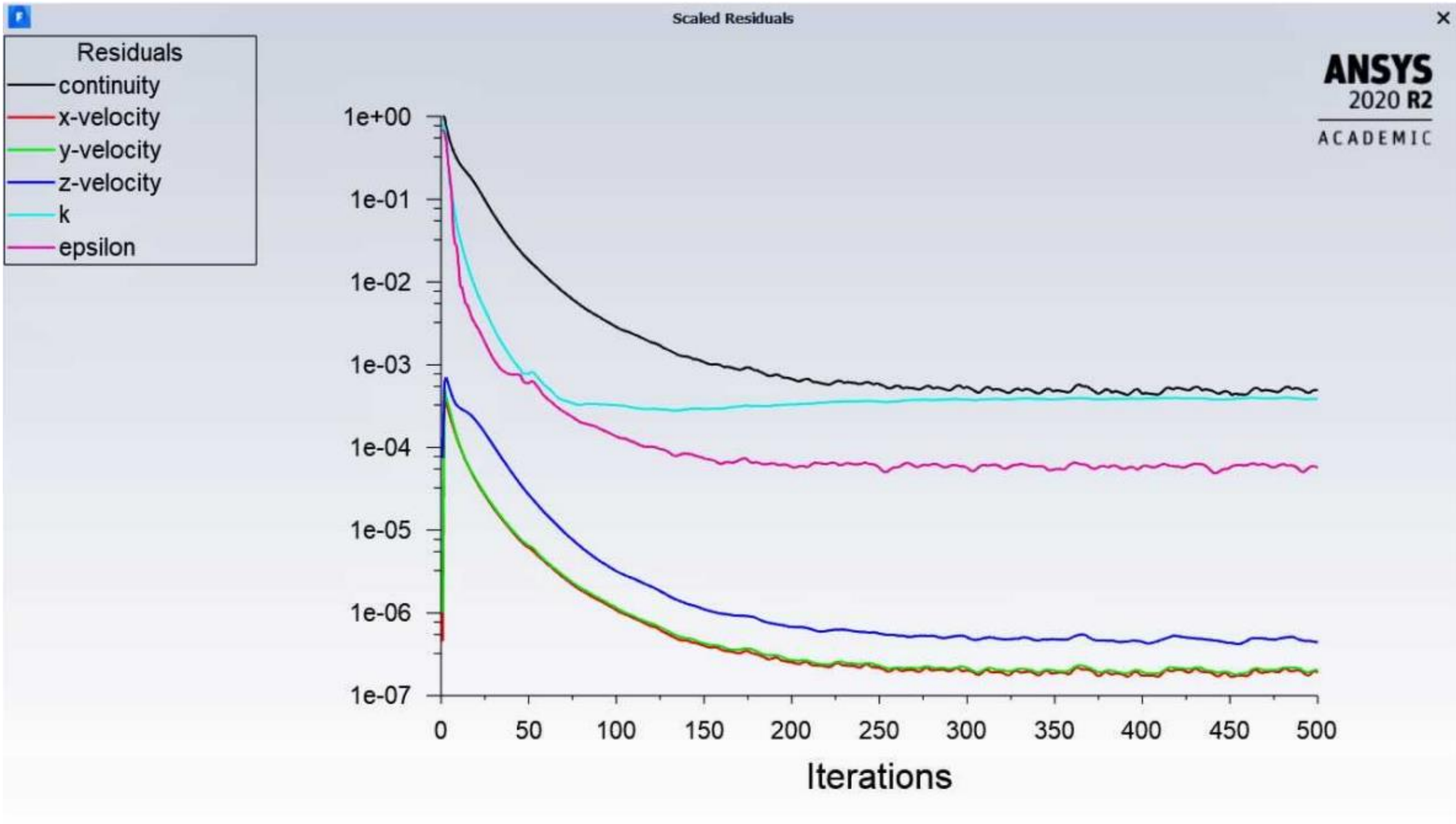
IV. Velocity vector plot for complete continuum:

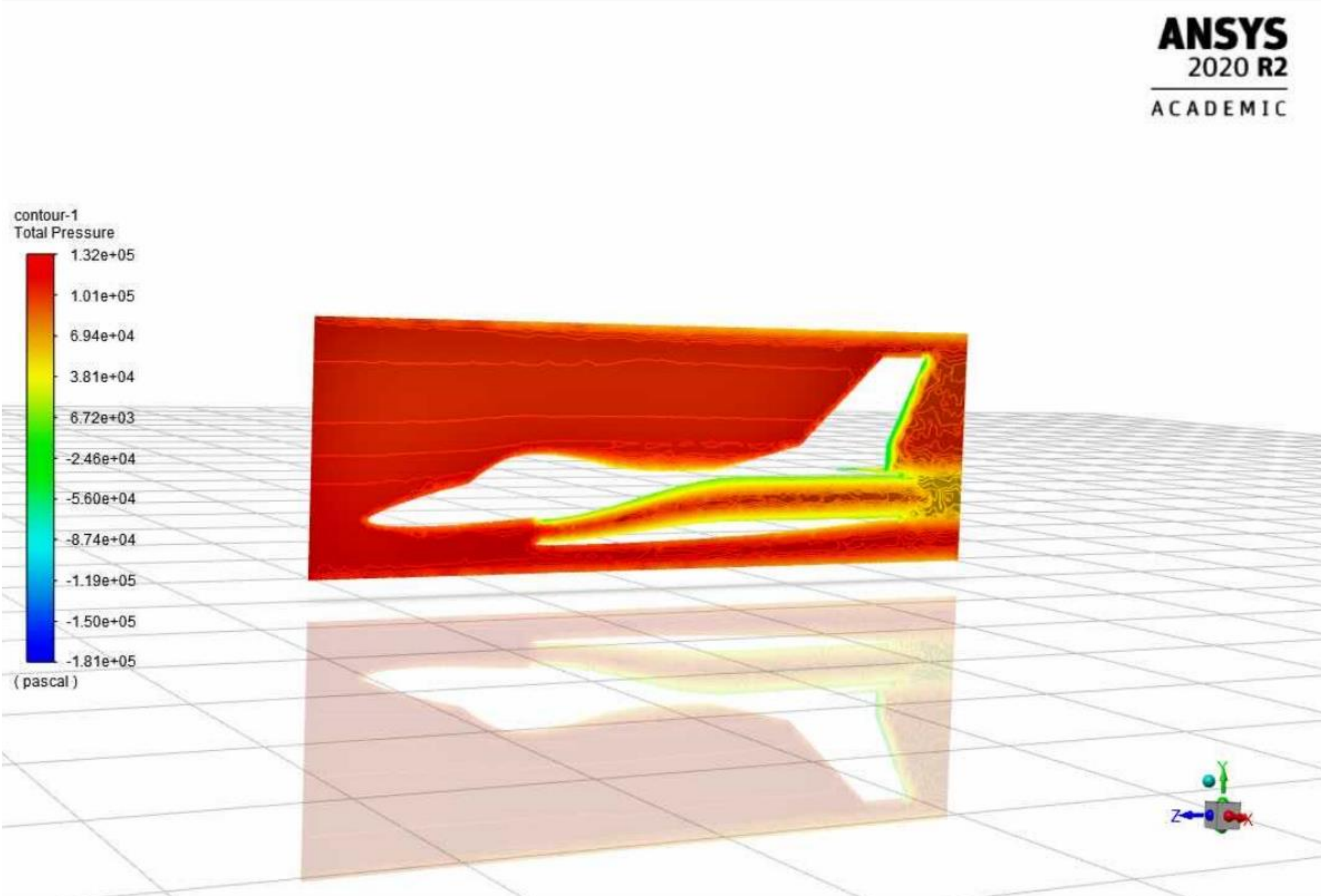
Variation in the velocity is at the aircraft and in the rest of the continuum, it remains constant. This is due to curvature of aircraft and that the rest of continuum doesn't have any shape and is assumed to be filled with air.

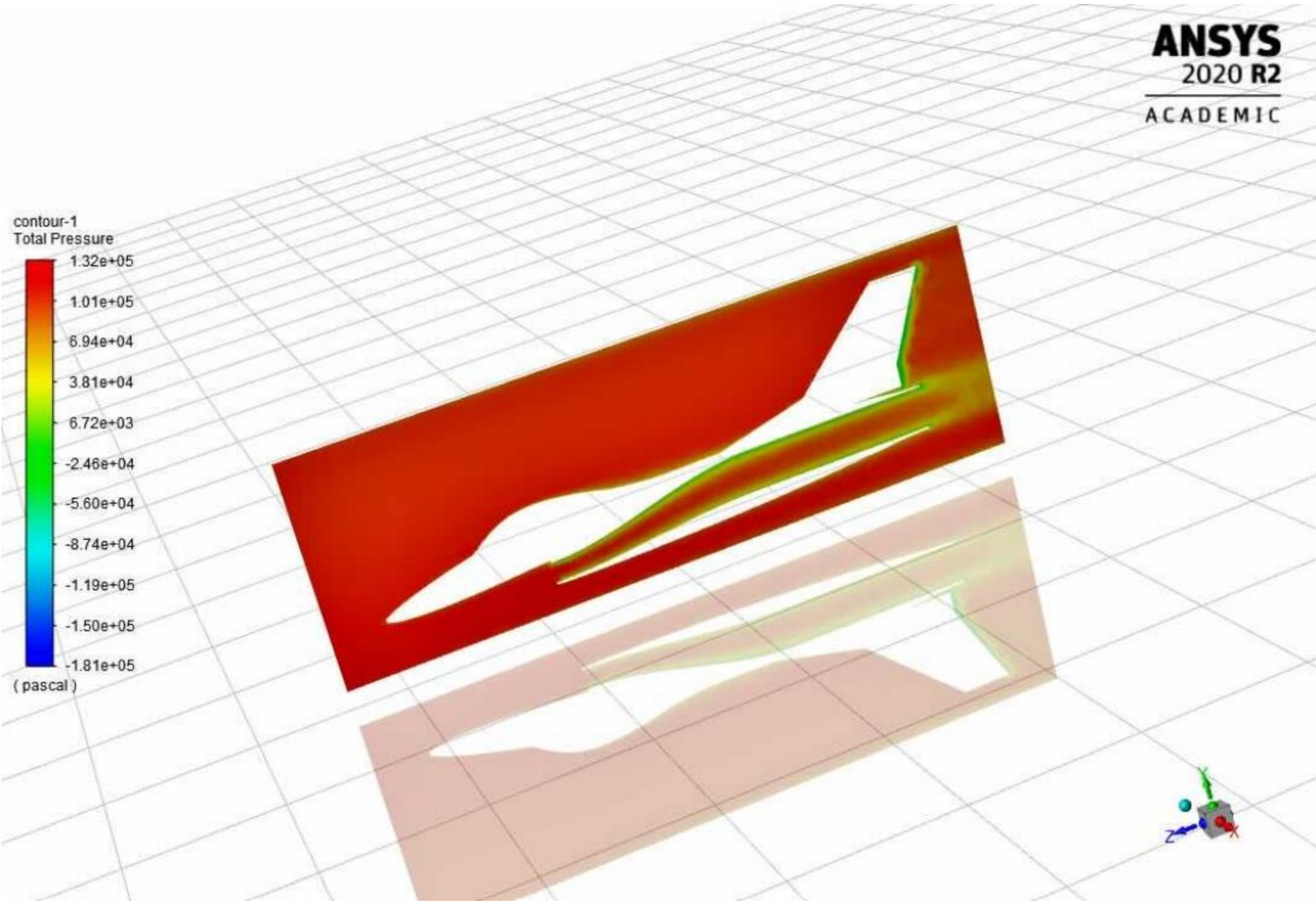
V. Velocity vector for the aircraft:

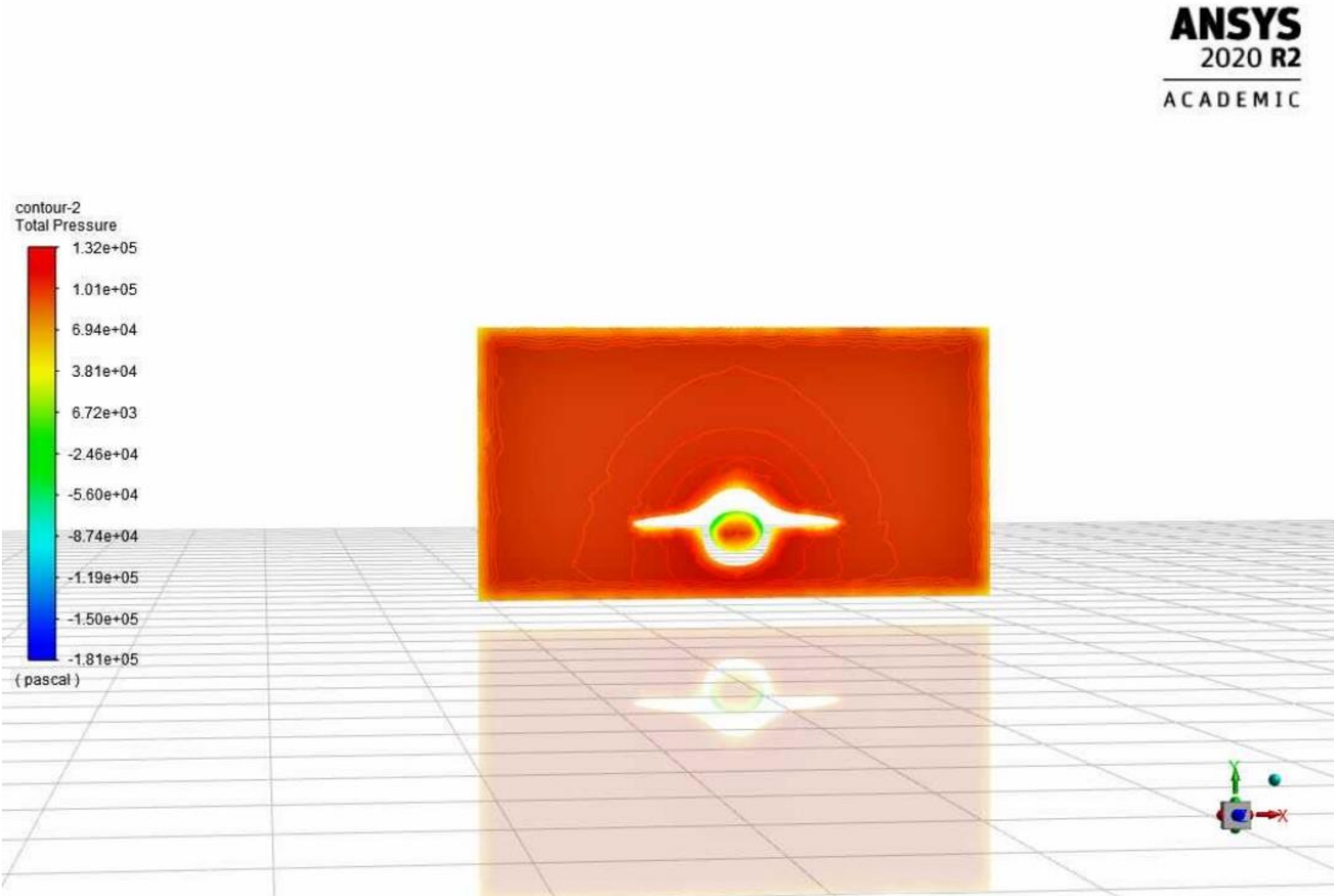
It can be seen that the velocity at the top of aircraft is more than the velocity at the bottom. This is due to the fact that for an aerodynamic shape of a body, the lower portion experiences high pressure with low velocity whereas the upper portion experiences low pressure and high velocity.



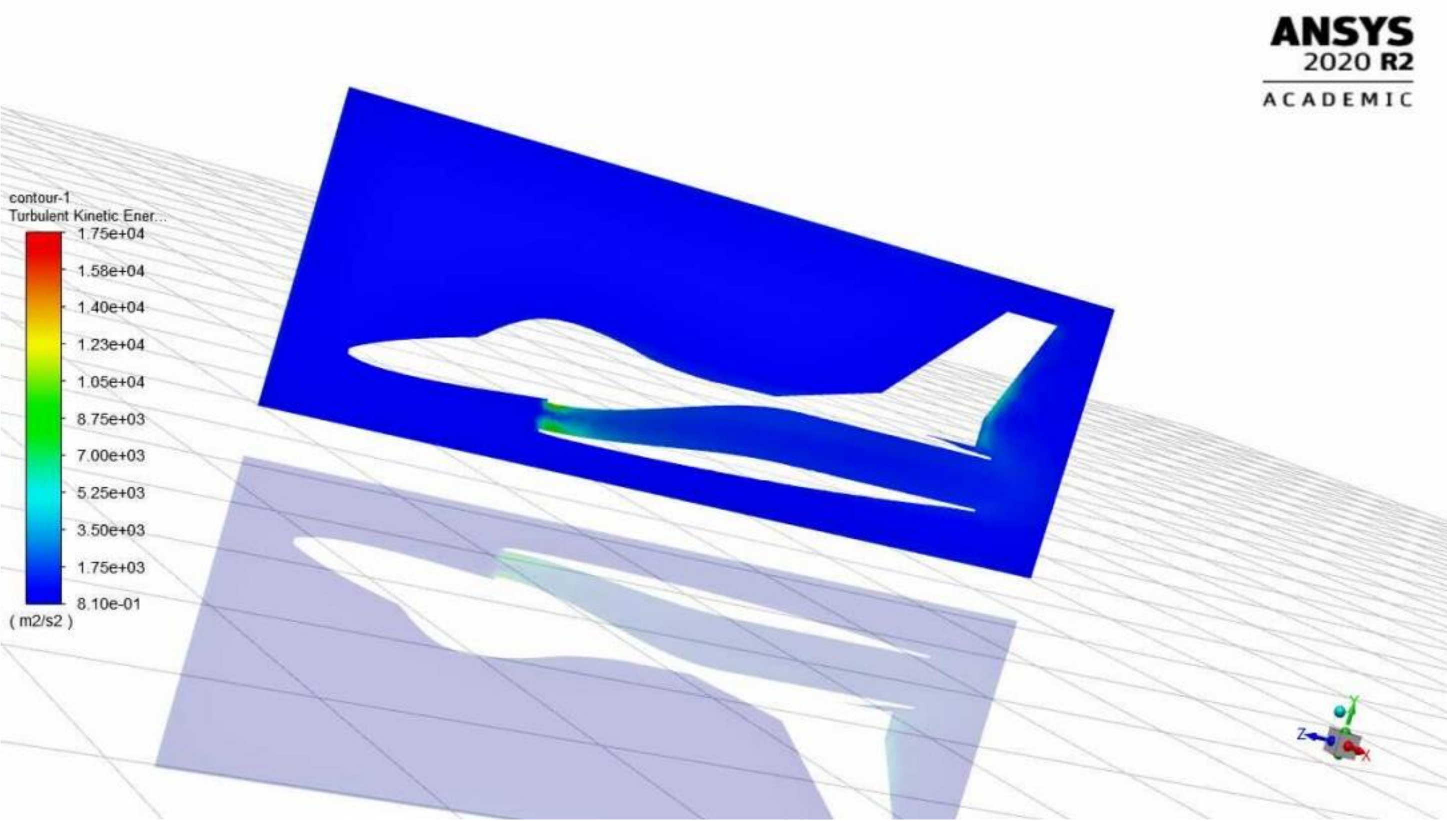

Scaled Residuals
Residuals
continuity
x-velocity
y-velocity
z-velocity
k
epsilon
ANSYS
2020 R2
ACADEMIC
1e+00
1e-01
1e-02
1e-03
1e-04
1e-05
1e-06
1e-07
0
50
100
150
200
250
300
350
400
450
500
Iterations

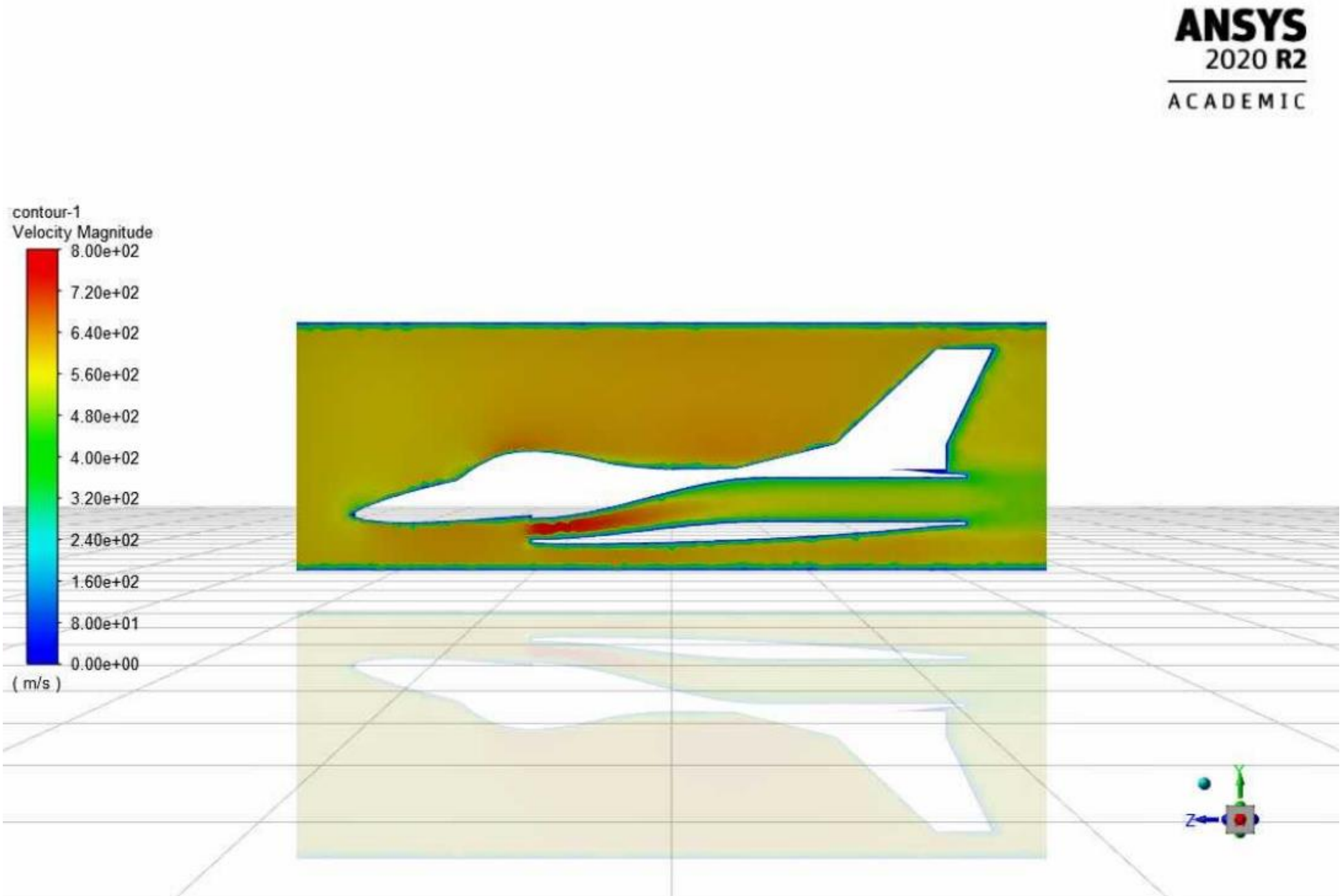

ANSYS
2020 R2
ACADEMIC
contour-1
Total Pressure
1.32e+05
1.01e+05
6.94e+04
3.81e+04
6.72e+03
-2.46e+04
-5.60e+04
-8.74e+04
-1.19e+05
-1.50e+05
-1.81e+05
(pascal)

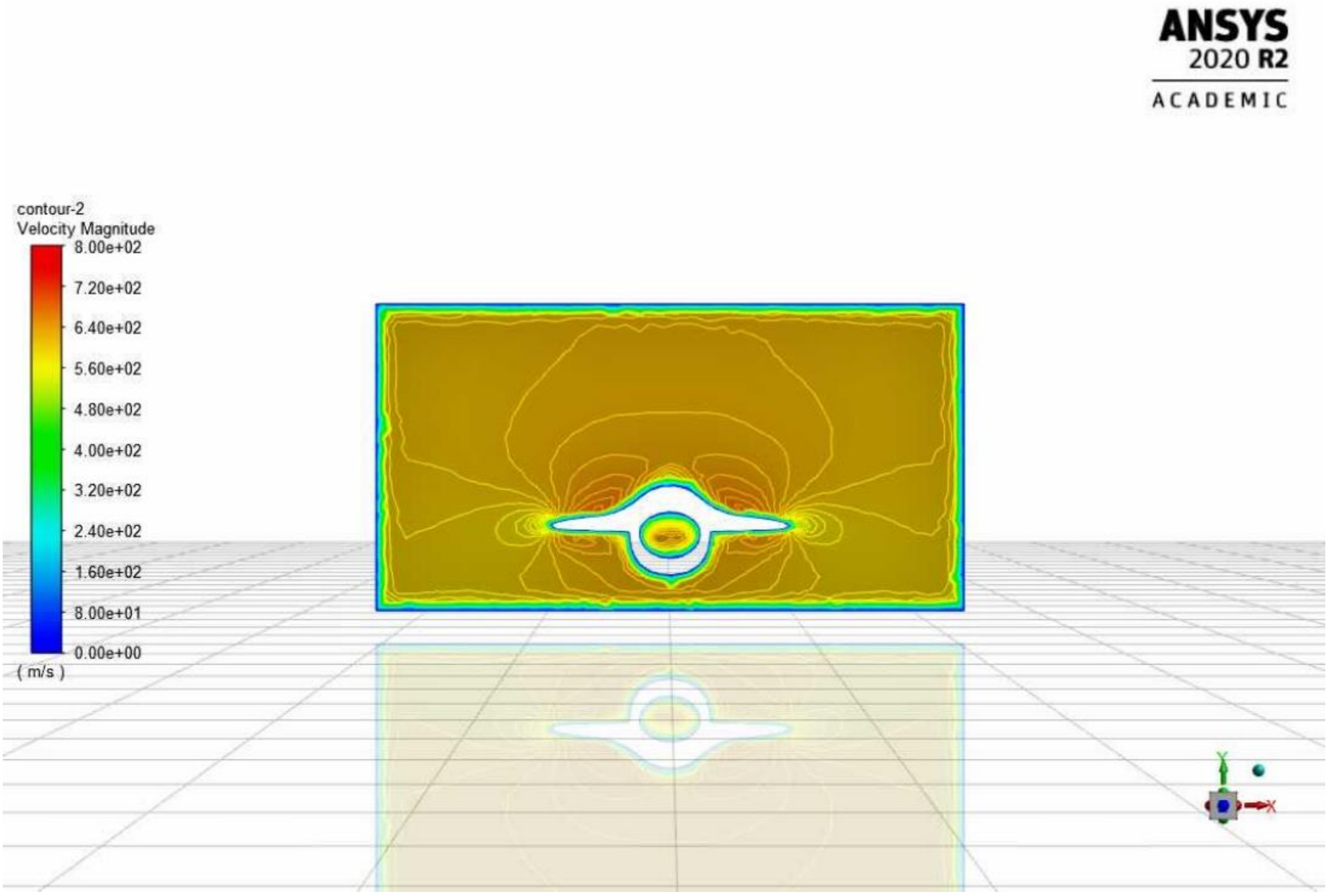

ANSYS
2020 R2
ACADEMIC
contour-1
Total Pressure
1.32e+05
1.01e+05
6.94e+04
3.81e+04
6.72e+03
-2.46e+04
-5.60e+04
-8.74e+04
-1.19e+05
-1.50e+05
-1.81e+05
(pascal)

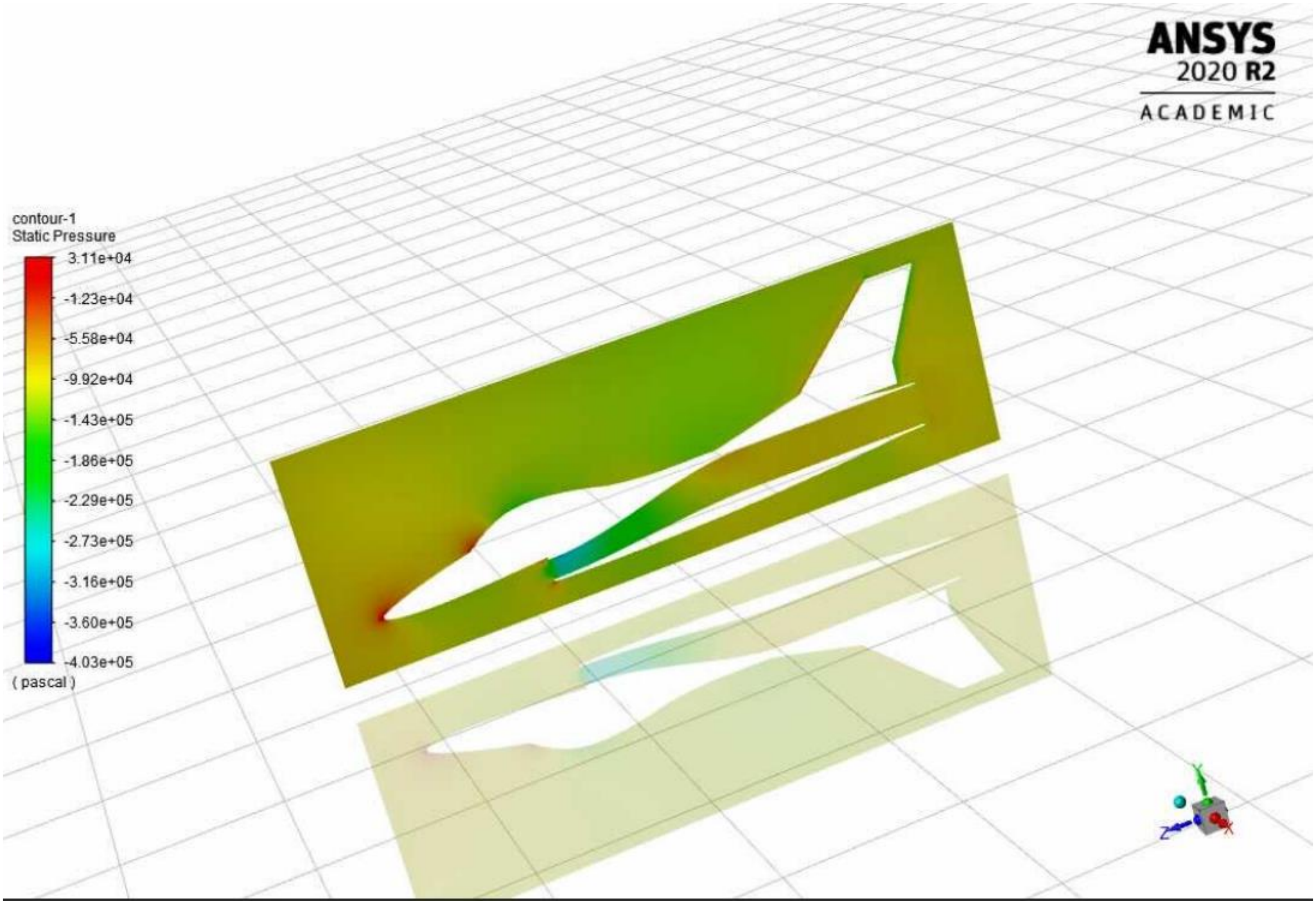

ANSYS
2020 R2
ACADEMIC
contour-2
Total Pressure
1.32e+05
1.01e+05
6.94e+04
3.81e+04
6.72e+03
-2.46e+04
-5.60e+04
-8.74e+04
-1.19e+05
-1.50e+05
-1.81e+05
(pascal)

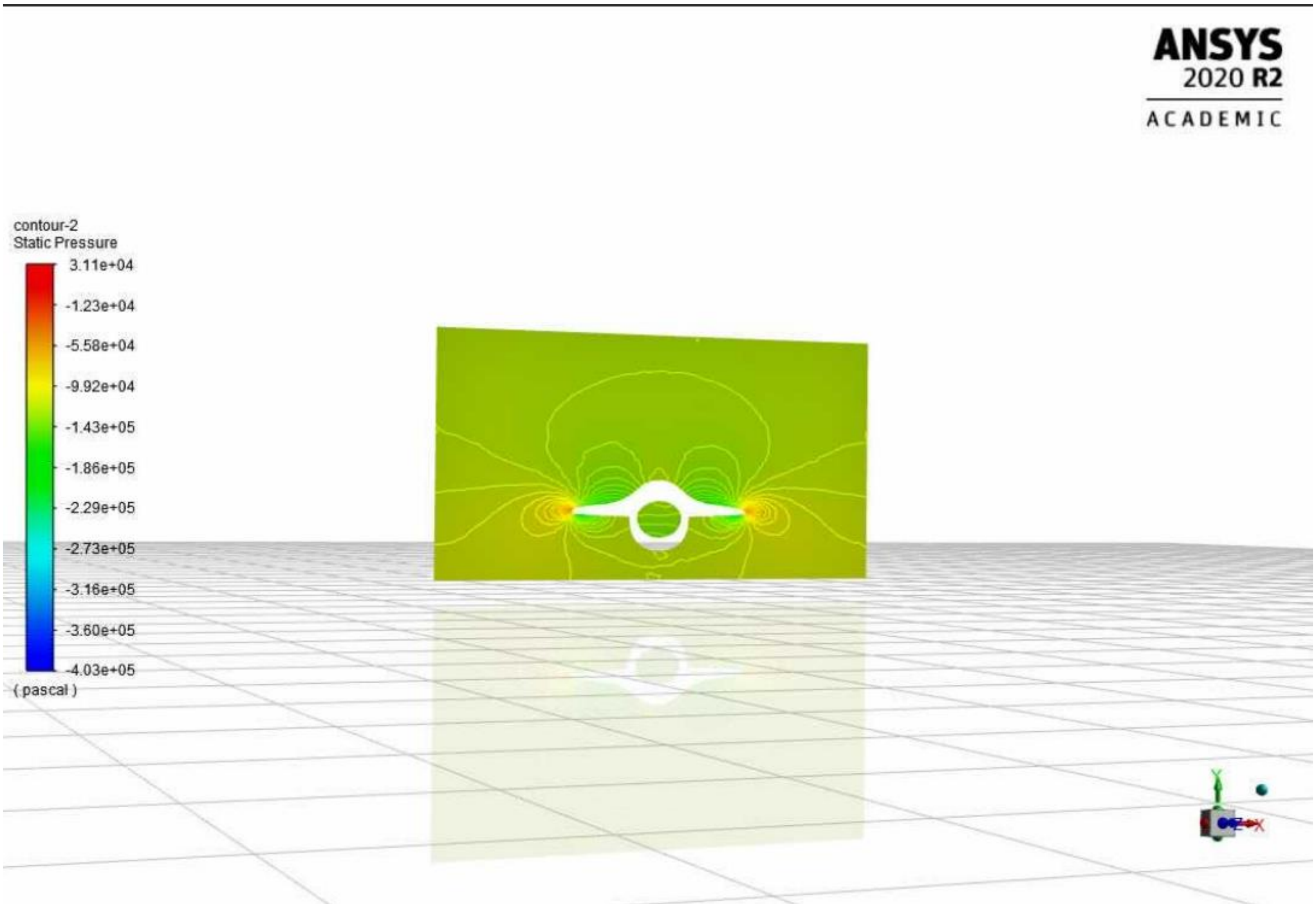

ANSYS
2020 R2
ACADEMIC
contour-1
Total Pressure
1.32e+05
1.01e+05
6.94e+04
3.81e+04
6.72e+03
-2.46e+04
-5.60e+04
-8.74e+04
-1.19e+05
-1.50e+05
-1.81e+05
(pascal)

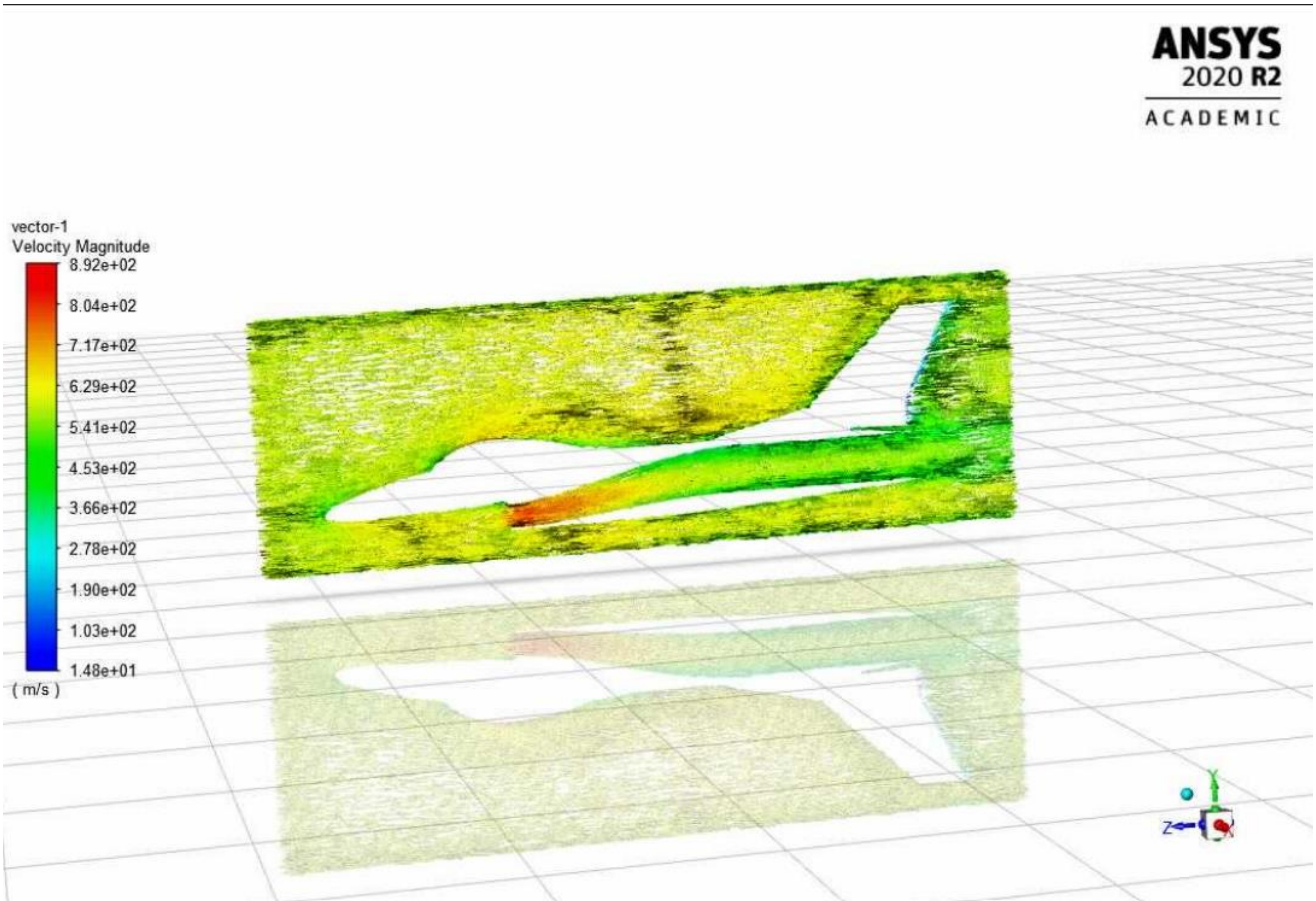

ANSYS
2020 R2
ACADEMIC
contour-1
Turbulent Kinetic Ener...
1.75e+04
1.58e+04
1.40e+04
1.23e+04
1.05e+04
8.75e+03
7.00e+03
5.25e+03
3.50e+03
1.75e+03
8.10e-01
(m2/s2)

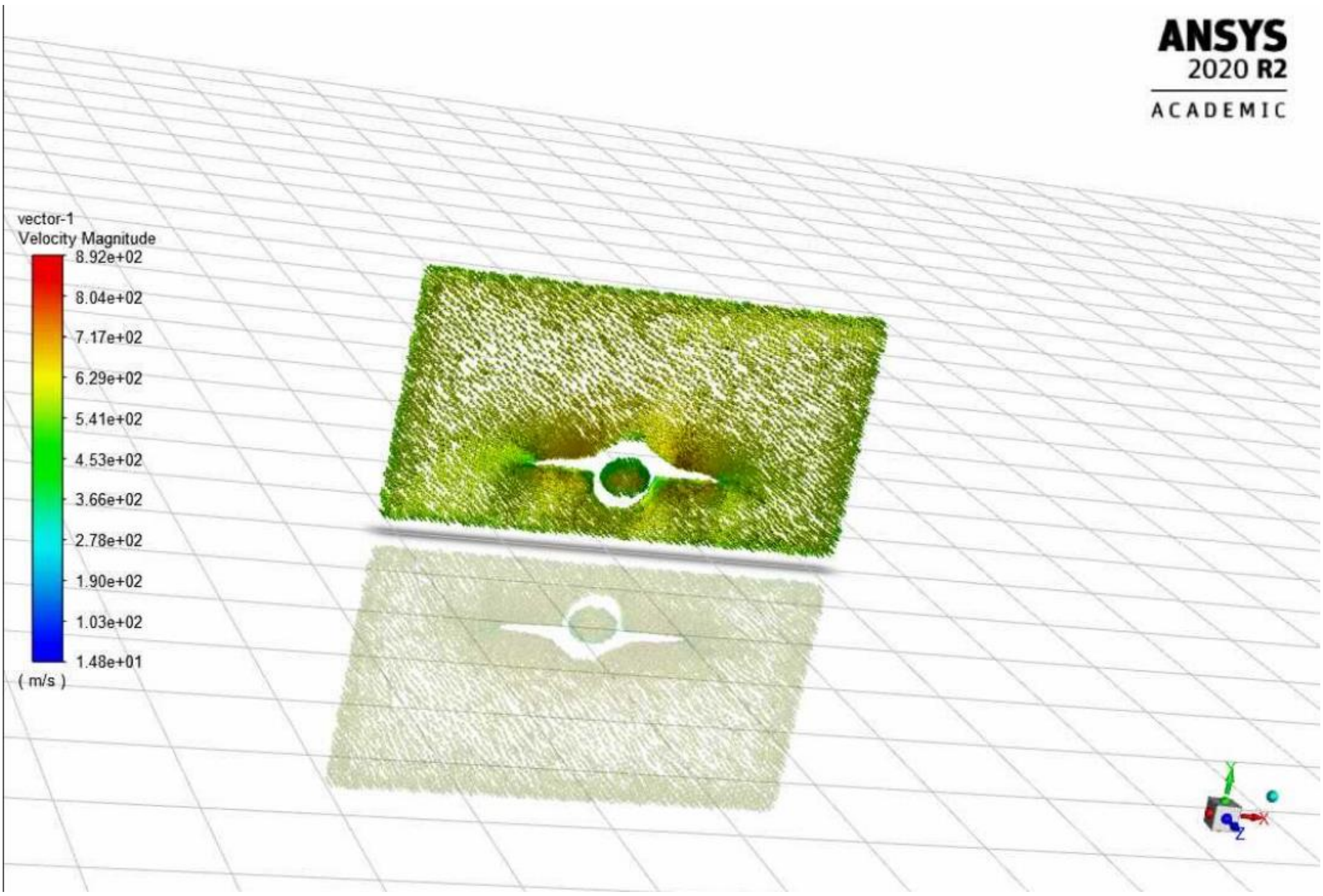

ANSYS
2020 R2
ACADEMIC
contour-1
Velocity Magnitude
8.00e+02
7.20e+02
6.40e+02
5.60e+02
4.80e+02
4.00e+02
3.20e+02
2.40e+02
1.60e+02
8.00e+01
0.00e+00
(m/s)

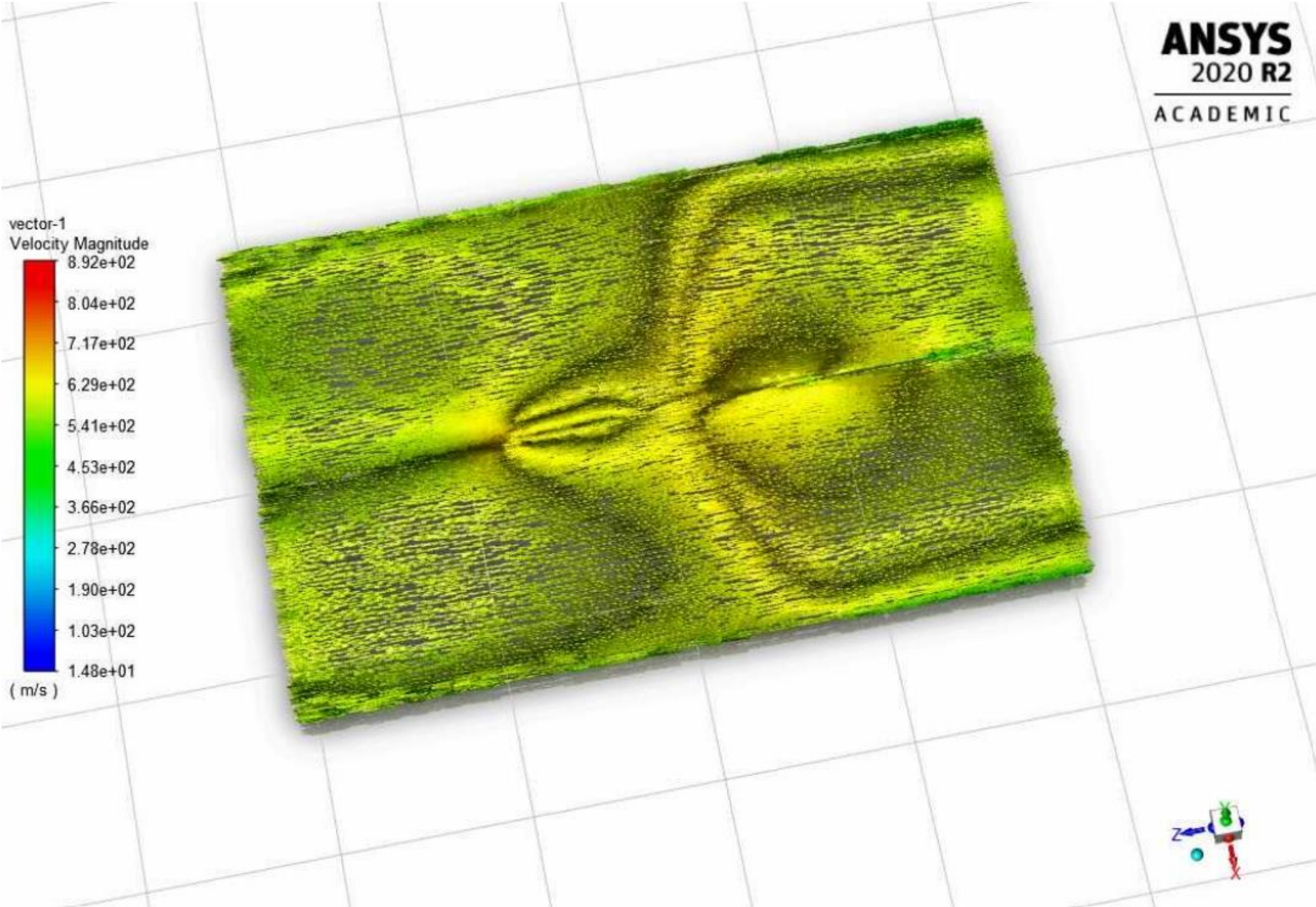

ANSYS
2020 R2
ACADEMIC
contour-2
Velocity Magnitude
8.00e+02
7.20e+02
6.40e+02
5.60e+02
4.80e+02
4.00e+02
3.20e+02
2.40e+02
1.60e+02
8.00e+01
0.00e+00
(m/s)

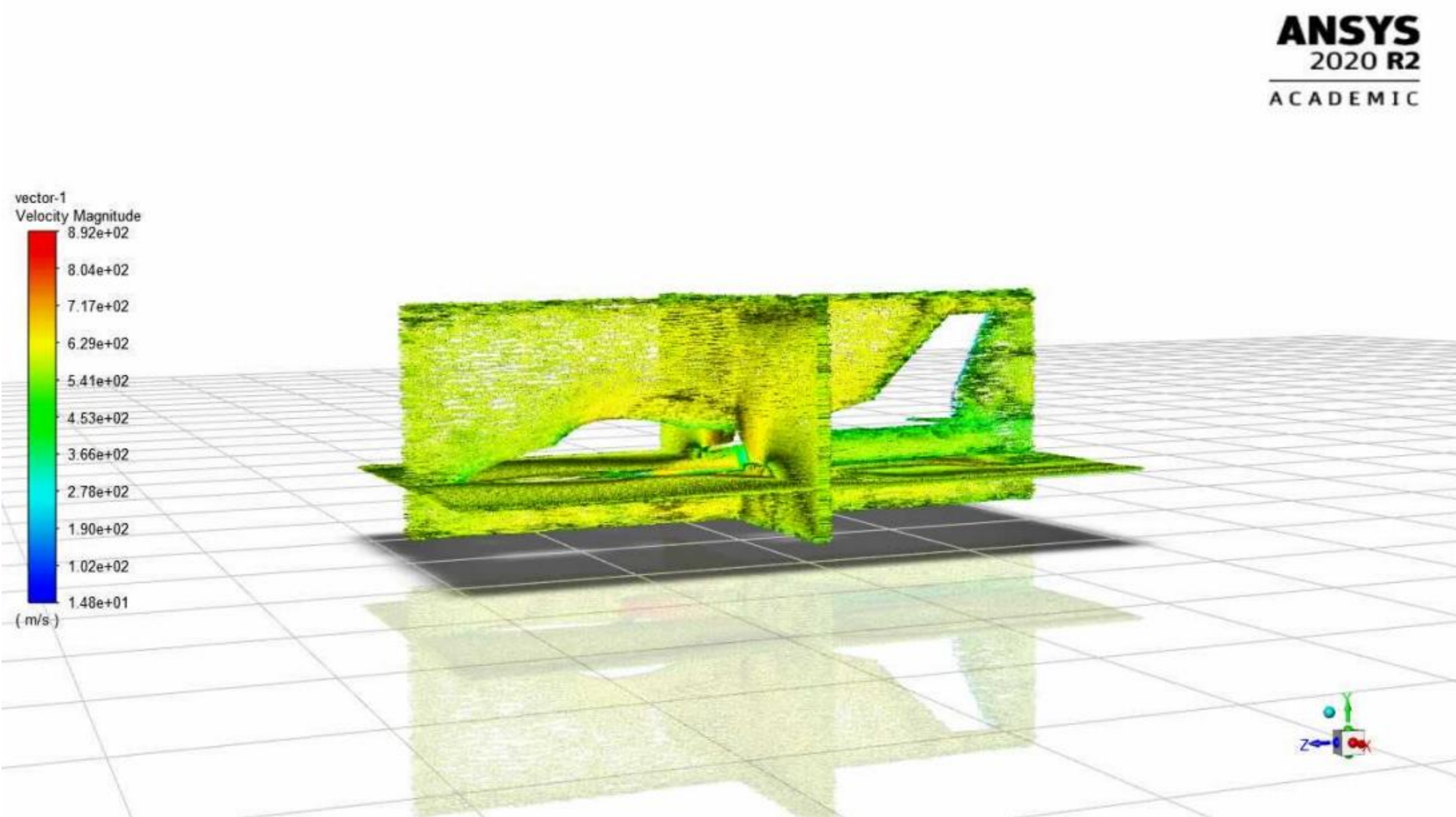

ANSYS
2020 R2
ACADEMIC
contour-1
Static Pressure
3.11e+04
-1.23e+04
-5.58e+04
-9.92e+04
-1.43e+05
-1.86e+05
-2.29e+05
-2.73e+05
-3.16e+05
-3.60e+05
-4.03e+05
(pascal)

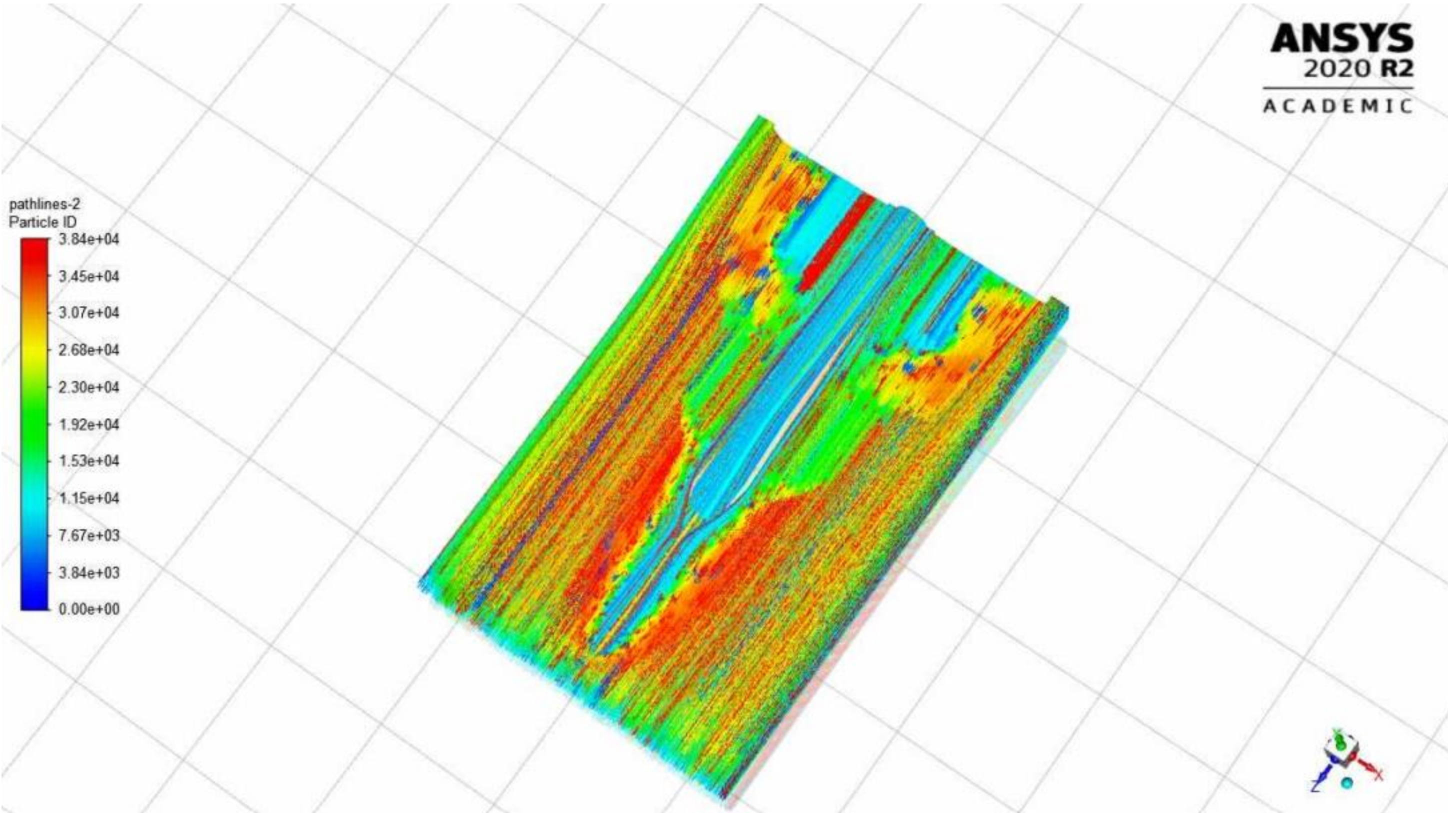

ANSYS
2020 R2
ACADEMIC
contour-2
Static Pressure
3.11e+04
-1.23e+04
-5.58e+04
-9.92e+04
-1.43e+05
-1.86e+05
-2.29e+05
-2.73e+05
-3.16e+05
-3.60e+05
-4.03e+05
(pascal)


ANSYS
2020 R2
ACADEMIC
vector-1
Velocity Magnitude
8.92e+02
8.04e+02
7.17e+02
6.29e+02
5.41e+02
4.53e+02
3.66e+02
2.78e+02
1.90e+02
1.03e+02
1.48e+01
(m/s)

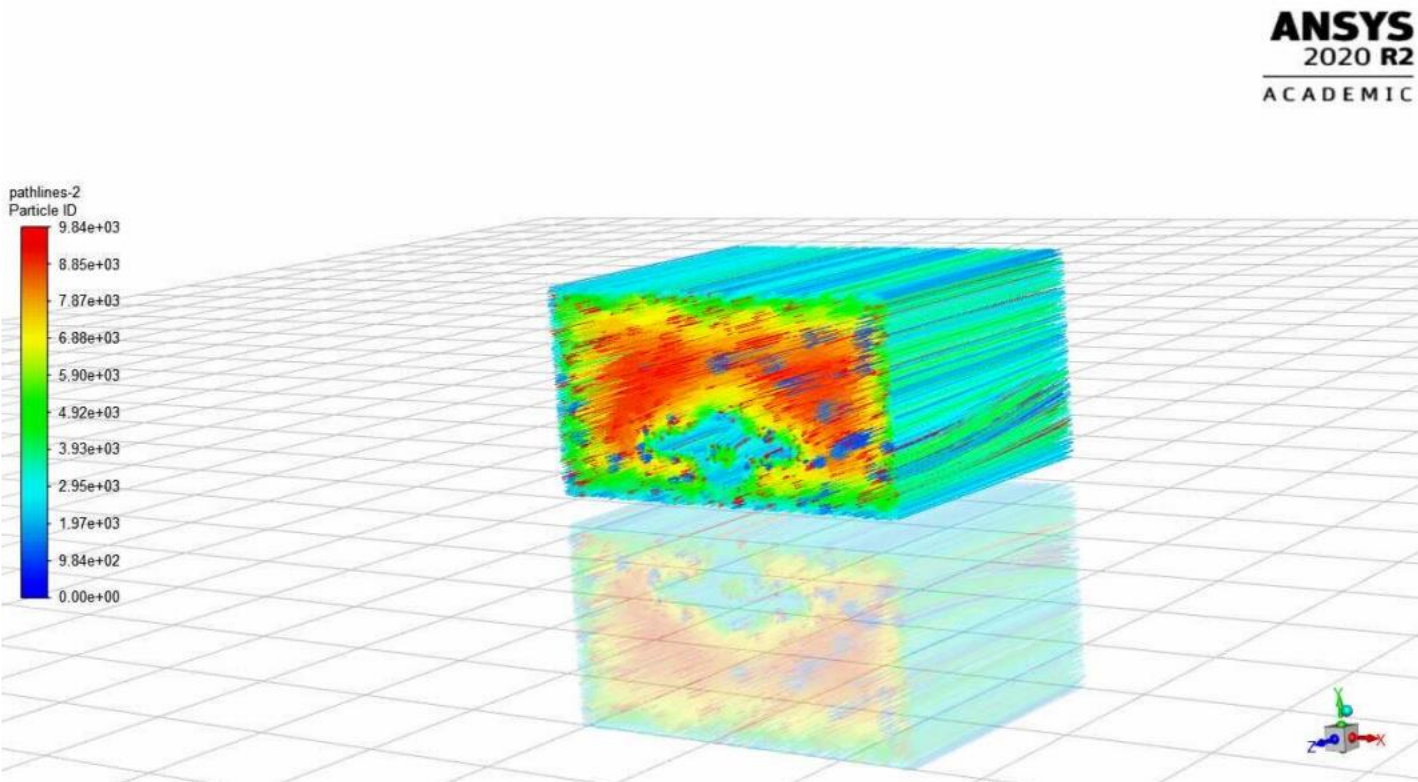

ANSYS
2020 R2
ACADEMIC
vector-1
Velocity Magnitude
8.92e+02
8.04e+02
7.17e+02
6.29e+02
5.41e+02
4.53e+02
3.66e+02
2.78e+02
1.90e+02
1.03e+02
1.48e+01
(m/s)


ANSYS
2020 R2
ACADEMIC
vector-1
Velocity Magnitude
8.92e+02
8.04e+02
7.17e+02
6.29e+02
5.41e+02
4.53e+02
3.66e+02
2.78e+02
1.90e+02
1.03e+02
1.48e+01
(m/s)


ANSYS
2020 R2
ACADEMIC
vector-1
Velocity Magnitude
8.92e+02
8.04e+02
7.17e+02
6.29e+02
5.41e+02
4.53e+02
3.66e+02
2.78e+02
1.90e+02
1.02e+02
1.48e+01
(m/s)



ANSYS
2020 R2
ACADEMIC
pathlines-2
Particle ID
3.84e+04
3.45e+04
3.07e+04
2.68e+04
2.30e+04
1.92e+04
1.53e+04
1.15e+04
7.67e+03
3.84e+03
0.00e+00



ANSYS
2020 R2
ACADEMIC
pathlines-2
Particle ID
9.84e+03
8.85e+03
7.87e+03
6.88e+03
5.90e+03
4.92e+03
3.93e+03
2.95e+03
1.97e+03
9.84e+02
0.00e+00