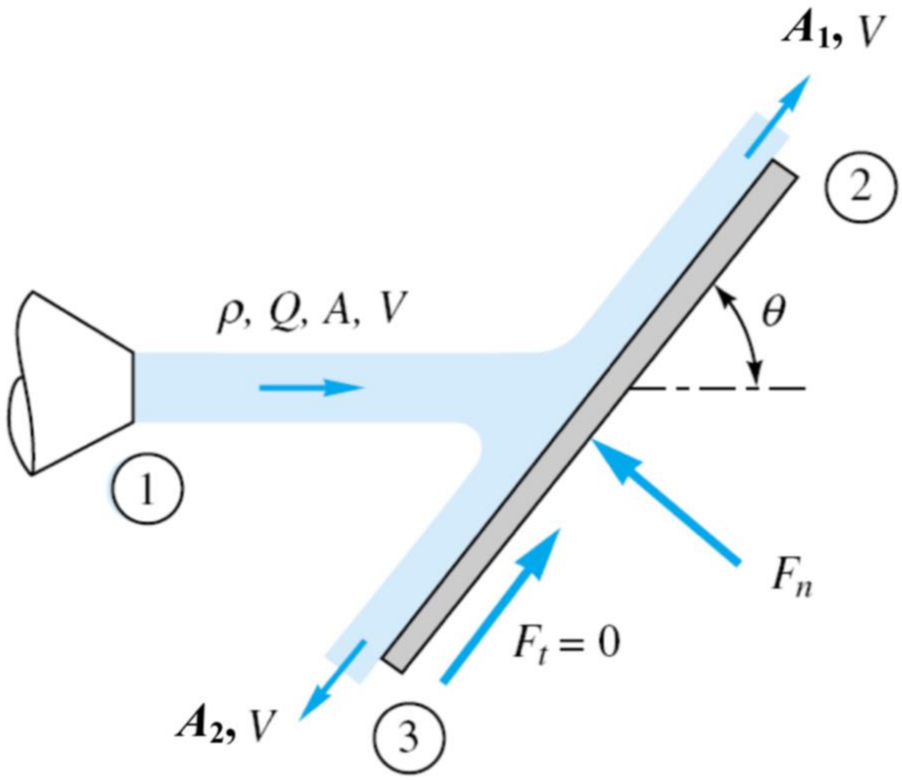
AM5820
Wind Tunnel and Numerical Experiments

Department of Applied Mechanics
Indian Institute of Technology Madras
Chennai – 600036

Numerical Simulation

<u>Impact of jet on Flat Plate</u>



Submitted by:
Nitin Yadav
AM20M004

Objective

To perform CFD simulation of impact of water jet on a flat plate using ANSYS Fluent and get the pressure distribution on normal impact.

Introduction

In solid mechanics, Newton's second law of motion states that a mass that is accelerated requires a force that is equal to the product of the mass and acceleration. The analogy to Newton's second law in fluid mechanics is known as the Impulse momentum theorem. According to Impulse momentum theorem, the impulse applied to an object will be equal to the change in its momentum

When a jet of fluid strikes a solid surface at any angle it will not rebound and rather move over the surface tangentially, which causes a change in linear momentum. This change in linear momentum results in the water jet exerting a force on the surface it is impacting.

By applying the Impulse momentum theorem, the components of this force can be resolved. To apply this theorem, following conditions needs to be satisfied:

- The fluid should be incompressible
- The surface tension forces are negligibly small
- The flow is steady
- The velocity distribution across the cross-section is uniform

Experimental Conditions

We have taken a jet of 6mm diameter, which is impacting normally on a square shaped flat plate of size 60 x 60 mm^2. Simulation is done by taking water as the working fluid with following properties:

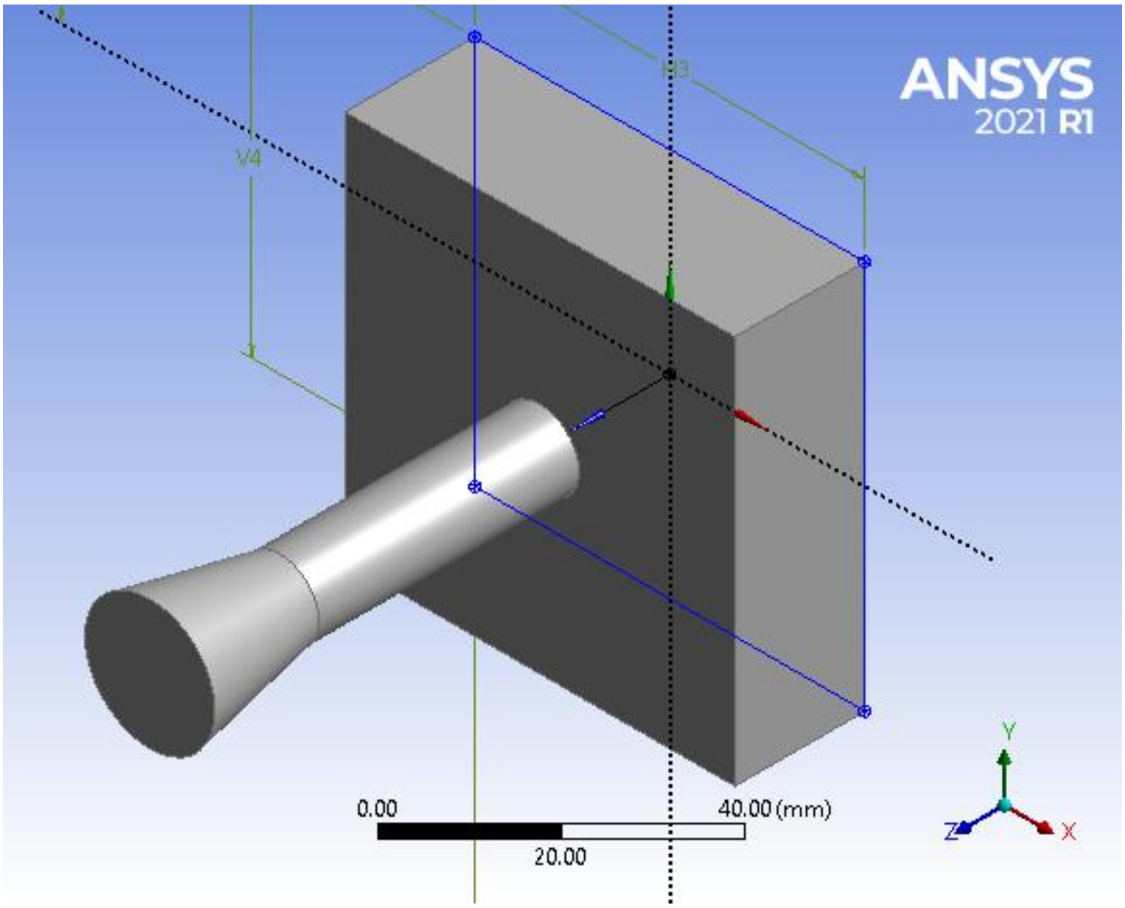
$$\rho = 998.2\, kg/m^3$$

$$\mu = 0.001003\, kg/(m-s)$$

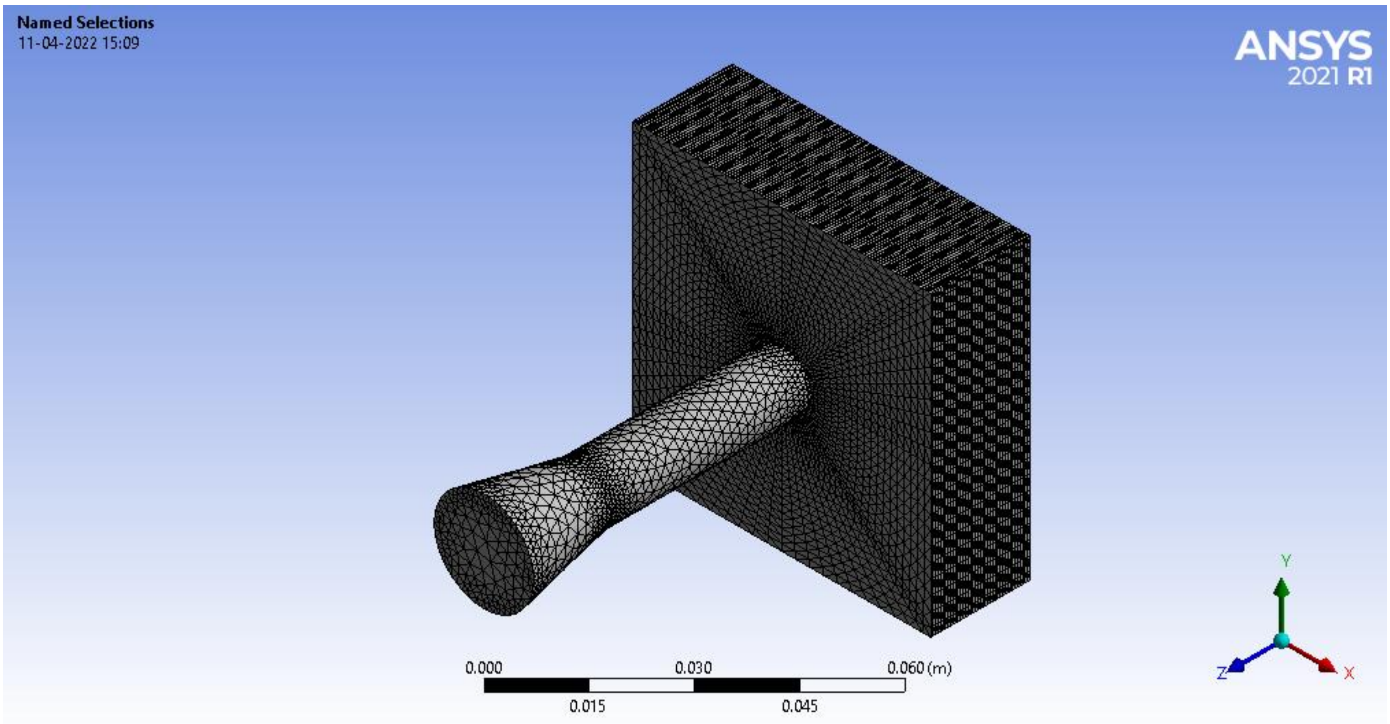
Jet inlet velocity is taken as 7.8595 m/s.

CFD Simulation Process

Geometry



Meshing



To make fine mesh with uniform shape, we have inserted “sizing” & “face meshing” in mesh design as follows:

- Sizing of eight edges of cuboid surfaces. It has 60 divisions with growth rate 1.2
- Sizing of three cross-sections namely jet inlet, jet outlet & jet impacting on plate. It consists of 50 divisions with growth rate 1.2
- Sizing of four edges of cuboid corners. It has 40 divisions with growth rate 1.2
- Face meshing consists of plate & four outlets of cuboid. It has element size of 5.831mm

Total number of elements = 133315

Total number of nodes = 30894

<u>Boundary conditions</u>

To set the boundary conditions as well as making sure that the flow will run along z-axis, we have named every side of geometry as follows:

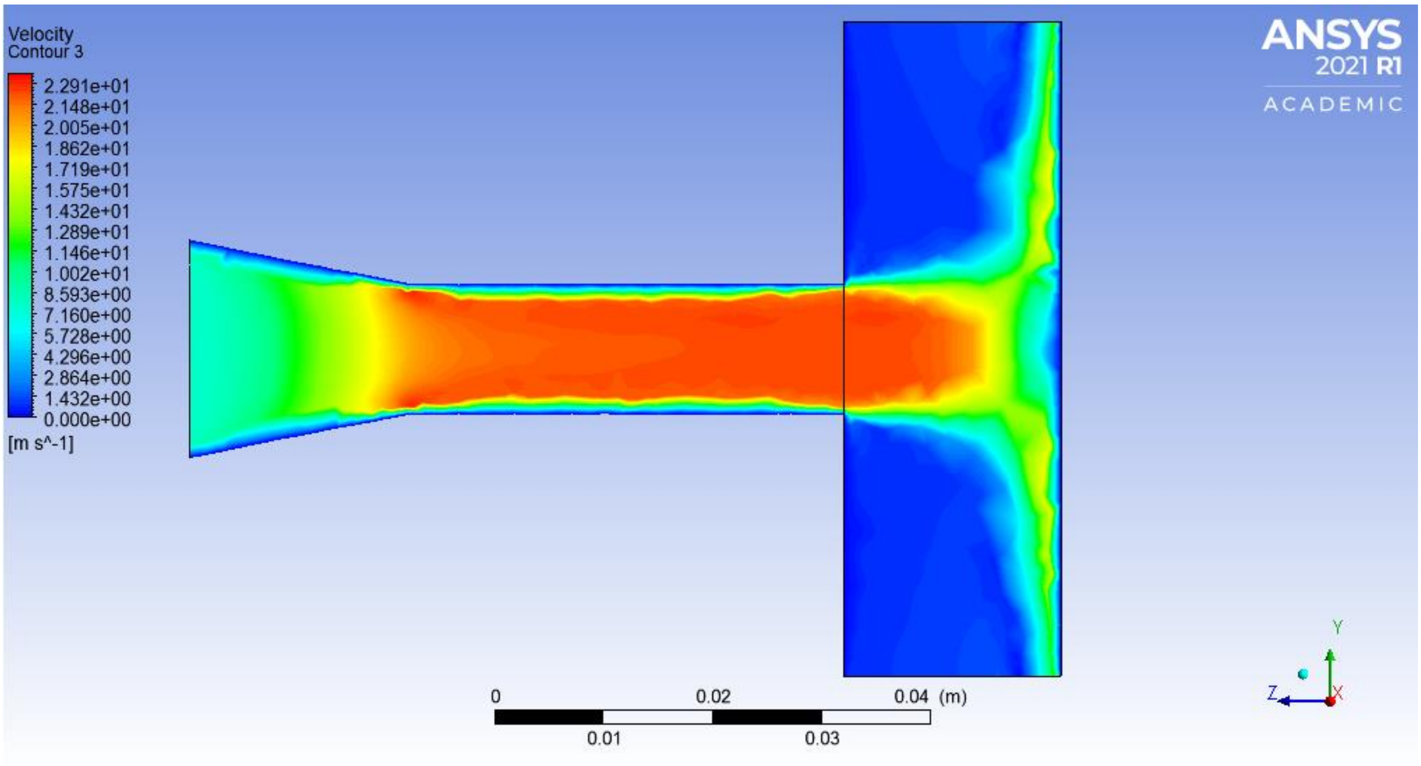
- Inlet for the nozzle inlet
- Outlet perpendicular to plate along four sides
- Wall-plate for flat plate

<u>Model</u>

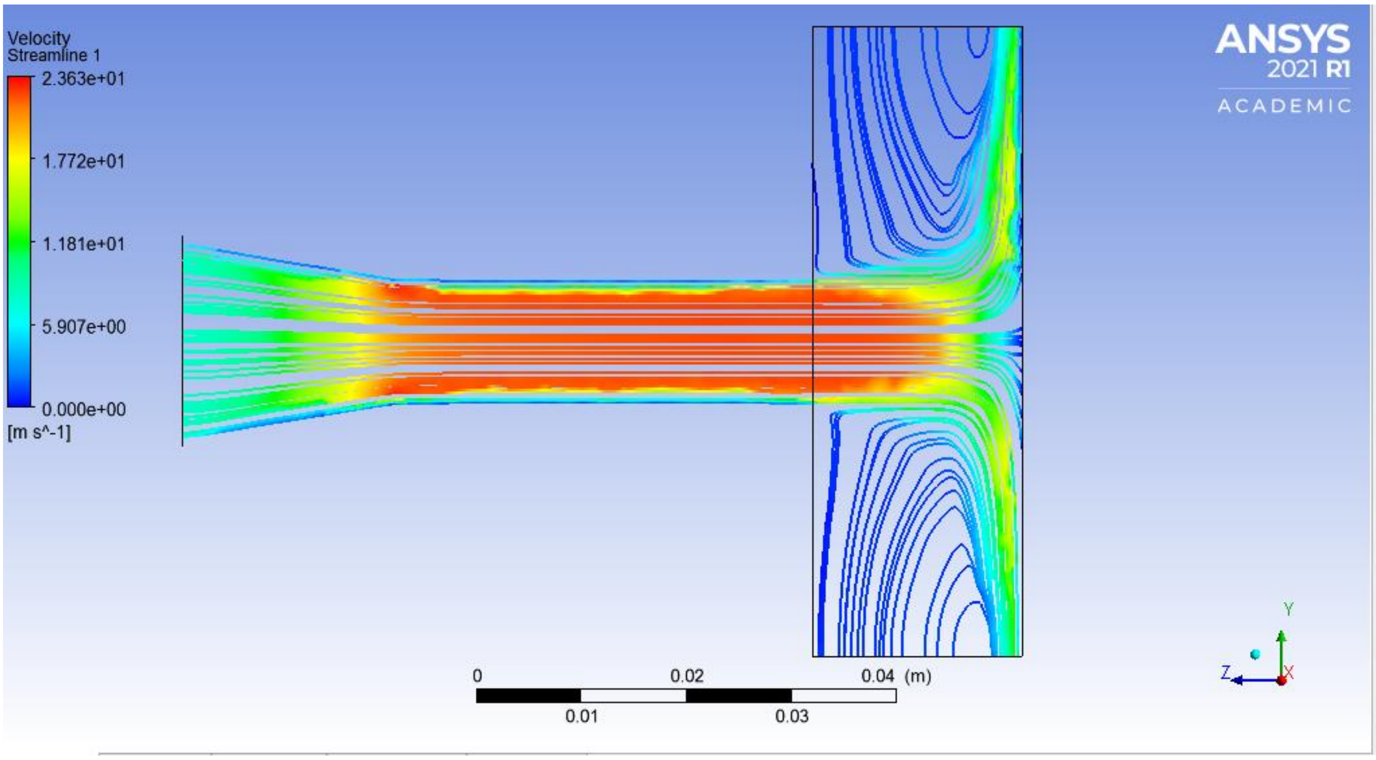
- Energy equation was used
- K-epsilon (2 equations) was used

Simulation Results & Analysis

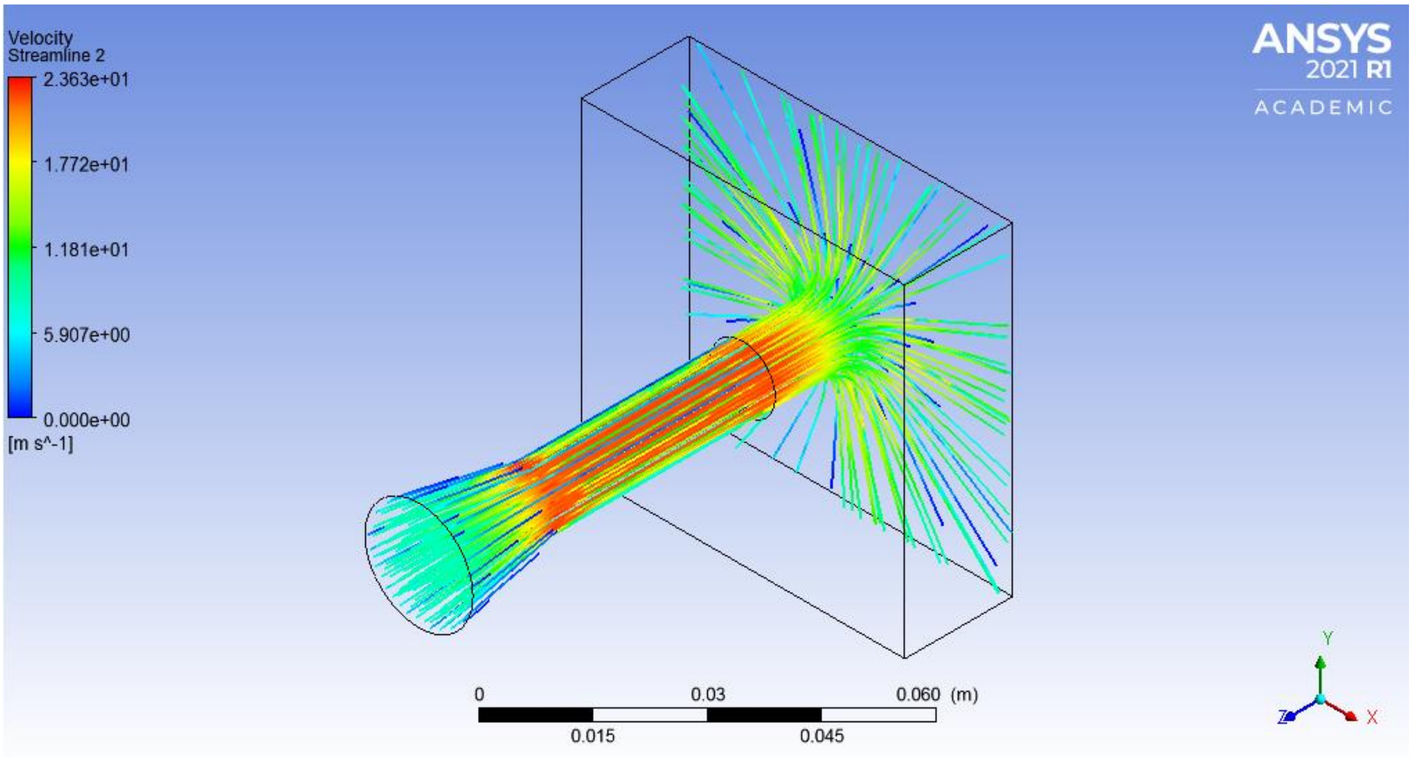
Velocity contours



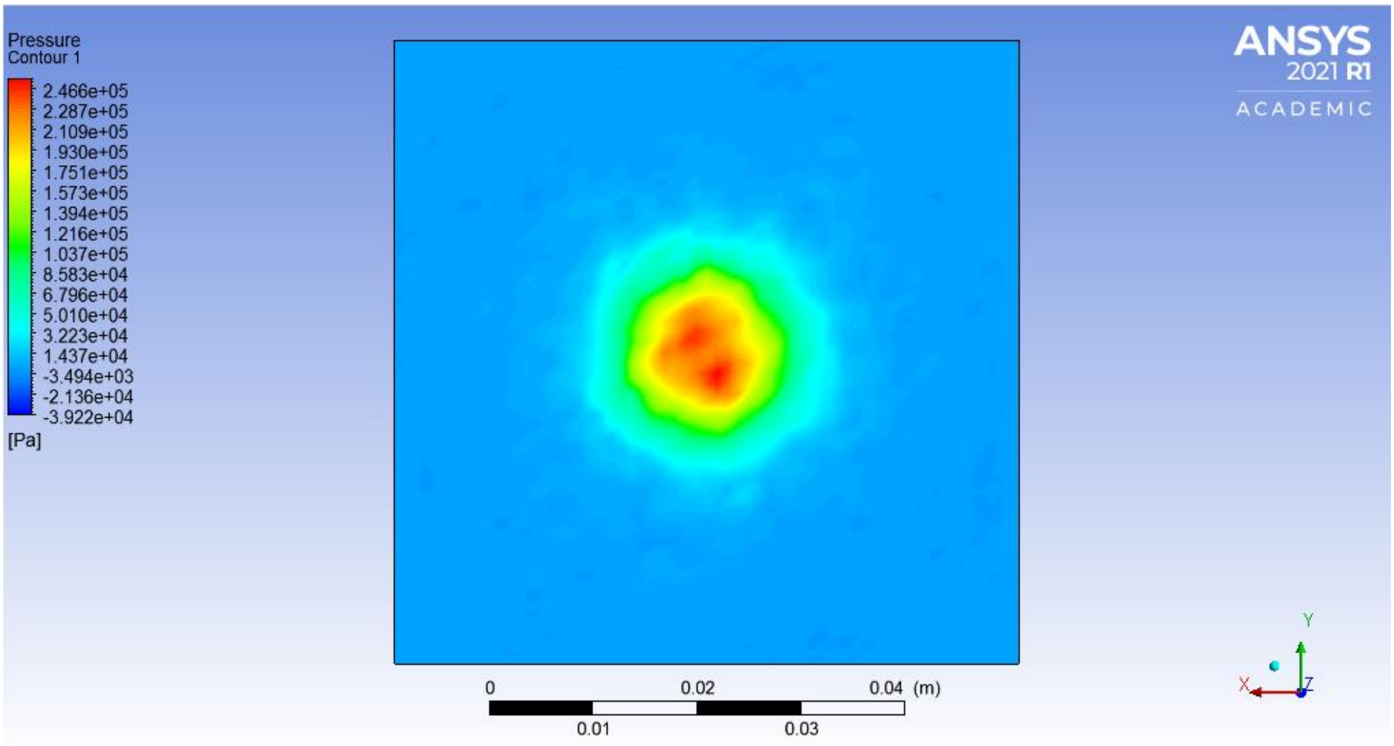
Velocity streamlines (2D)

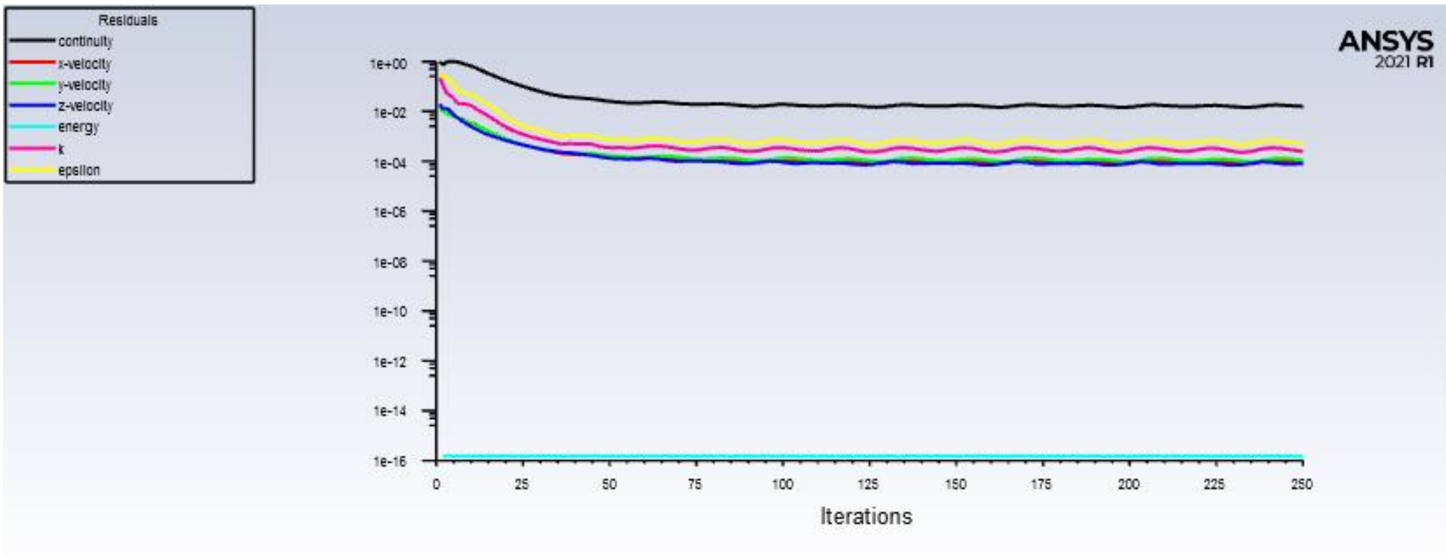


Velocity streamlines (3D)



Pressure contours on plate





Conclusion

It can be seen from pressure plot, that the pressure is maximum at the center in (red region) of plate with magnitude close to 246.6 kPa.