REPORT : ASSIGNMENT 2

Motivation for setup: Knowledge of the case of axisymmetric, steady-state turbulent jet is very crucial in understanding and optimizing processes like mixing and cooling in industries, aircraft propulsion systems (particularly in the exhaust of jet engines), managing the shape of the nozzle of these jets which can help in reducing noise. Understanding the behavior of turbulent jets is essential for improving performance, efficiency, and safety.

My approach for the problem:

1) Creating the geometry file: A simple rectangular 2D geometry is chosen to be the boundary of the mesh (shape of the jet) with length 7 units and breadth 1.2 units. For this I create a .geo file. In this file we define the coordinates of the 4 corners of the rectangle forming the jet, make edges by defining lines from these points and create a loop from these four lines in order to make it a continuous geometry. Then I define a plane surface so that it can be used later to be designated as an inflow/outflow or symmetry plane. I also define surface 1,3 as a transfinite surface as they are walls of the jet so they are a bit complex. Recombine surface parameter helps to provide a structured mesh which helps in convergence of the solution. I use a transfinite curve parameter to divide a section of the geometry into some desired number of segments using some progression with distance. Adding a physical curve helps us to define boundary conditions on these surfaces. A very important step is to add a physical surface for the fluid domain otherwise the domain will not be interpreted as a fluid domain. After defining all the geometry parameters I load this file in gmsh UI and create a mesh then export this file as .su2 file. The .su2 file contains parameters like dimension of the problem (NDIME), number of points (NPOIN), number of elements (NELEM), number of markers (NMARK) and nodes (Coordinates (x, y, z) of mesh nodes).

2) Creating the configuration file: The configuration file is mainly used to define the boundary conditions, solver settings, convergence criteria etc. for a particular problem to be solved with the SU2 suite. For our case of a steady state axisymmetric turbulent incompressible jet I use Euler as a solver coupled with a turbulence model to approximate turbulent effects. The Navier-Stokes solver is noy used because it has a very high computational cost. The parameter MATH_PROBLEM with the value DIRECT typically indicates that the solver is configured to solve the direct

problem, where the primary objective is to directly solve the governing equations of fluid flow for a given set of boundary conditions and initial conditions. In the context of computational fluid dynamics (CFD), this means solving the Navier-Stokes equations or the Euler equations (depending on the flow regime) to obtain the flow field variables such as velocity, pressure, temperature, etc. The INNER_ITER parameter is used to control the number of iterations performed within each nonlinear iteration of the solver to iteratively solve the discretized equations until convergence is achieved. The CFL_NUMBER parameter is used as the CFL number is a scaling factor to limit the time step size. But in steady state simulations it is used to stabilize the solver. It is generally between 0.5 to 1. I use INIT_OPTION to be Reynolds for initializing the solution with Reynolds conditions. One of the most important parts of the configuration file is the definition of the boundary conditions. Here I use markers for each physical curve that is defined in the geometry file. The MARKER_INLET is used to define the boundary conditions of temperature, velocity magnitude and direction of velocity at the inlet surface. MARKER_OUTLET in my case is the pressure outlet so I define its boundary condition to be atmospheric pressure of 101325 Pa. MARKER_SYM is used to define the symmetry boundary condition. MARKER_EULER is used to solve the euler equation on the walls. Some other parameters used are:

OUTPUT_FILES = (PARAVIEW,CSV): this indicates the type of file that will be given as output. Here it gives a .vtu file to be visualized in paraview and a csv file for tracking convergence history

OUTPUT_WRT_FREQ: it indicates with how much frequency the output should be written on screen

CONV_RESIDUAL_MINVAL= 1e-06: This is used to define the convergence criteria. It stands for minimum value of convergence residual.

3) Running the files: I used the command 'SU2_CFD meshconfig.cfg' to run my configuration file after navigating to the location where my meshconfig.cfg file was present. The solution converges in about 100 iterations.

A view of velocity magnitude in my jet is:

Plot of the velocity in y direction over line in the jet shows some turbulent nature in y direction.The turbulence is very frequent near the inlet of the jet but slowly the flow becomes stable towards the outlet of the jet with velocity in y direction = 0. The velocity then remains only in x direction.