American Journal of Engineering Research (AJER)
e-ISSN: 2320-0847 p-ISSN : 2320-0936
Volume-12, Issue-10, pp: 01-06
www.ajer.org
Research Paper Open Access

Numerical Method for Large Eddy Simulation of Turbulent Jets

Mohammad A Al-Saleem
The Public Authority for Applied Education and Training , Sabah Alsalem Training Institute, Electrical Department ,Kuwait.

Abstract
In this paper we showed thenumerical method for "Large Eddy Simulation of Turbulent Jets" with software implementation Ansys fluent this paper designed to show the whole process of simulation of large eddy with the geometry designing.There are four main parts of this paper, the first part is describes the brief introduction of the "Large Eddy Simulation of Turbulent Jets", second part is the problem description of the for the large eddy simulation in this part is important to understand the main purpose of the paper, the third part showed the software implementation of the "Large Eddy Simulation of Turbulent Jets" there are different steps shown for the simulation with snaps, the fourth showed the results of the simulation where we can analyses the results of the large eddy simulation for the turbulent jet there is another sub part which concludes the obtained results from the large eddy simulation.

KEYWORDS: *jets; ANSYS; unsteady ;large eddy; turbulence; flow*

Date of Submission: 01-10-2023 Date of acceptance: 12-10-2023

I. Introduction:

We are investigating the "Large Eddy Simulation of Turbulent Jets" it is basically a turbulence's mathematical model which is used in computational fluid dynamics. The simulation of the turbulent flows by numerically solving with Navier-Strokes equations requires the very long length of scale with time and these all directly affect the field of flow. The resolution can be accomplished through the DNC direct numerical simulation it is expensive solution used for simulation of pumps, landing gear, turbulent jets.

For the feasible future turbulent flow can hardly be simulatedreliably via direct numerical simulation (**DNS**) because direct numericalsimulation requires resolution of all persistent eddies. AssumingKolmogorov's description of turbulence valid, the smallest length scale isexpected to be

O (Re^ (-3/4)).

The physical problem:
The physical state of the turbulent jet is given as,

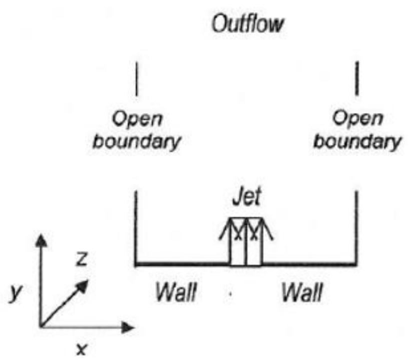


Figure 1: Vertical central region of the bottom side of planar jet.

The size of the domain is represented by the L_x, L_y, L_z which are the (2x4x0.02) m. The planar jet issues vertically from the central region of the bottom side of the domain as shown in the figure 1 there some parameters for the jets which are jet velocity inflow boundary and the temperature is uniform in the computation domain. The z direction must be included in the simulation of the planar jets, 2 grid points to the model of "3D" planar jet with small distance in z direction, the grid resolution must be sufficient at least 80 grid. For the physical problem we would analyses the two cases one for pulsating inlet flow velocity which mathematically can be represent as,

$$V = \bar{V}[1 + A\sin(2\pi f_o t)]$$

Where $\bar{V}$ is the inlet velocity, A is pulsating amplitude, f_o is the pulsating frequency and the t is time for pulsating inlet velocity.

The second we would study the inflow velocity of the jet which mathematically can be represent as,

$$V = \bar{V}\sin(2\pi f_o t)$$

To analyses the above to cases we would perform the simulation in Ansys fluent with user define function UDF.

Simulation Parameters:
To simulate the large eddy simulation for turbulence jet we have the following simulation parameters.

1) The domain size for jet is given as

L_x = 2m

L_y = 4m

L_z= 0.02m

Volume=0.16m^3

2) The jet width is 0.1m.
3) The mean velocity of the jet at the inflow boundary is 10m/s and 0.
4) Temperature is uniform.
5) The pulsating frequency is 20Hz.
6) For the non-zero mean velocity at the inflow amplitude of pulsating is A=0.3.
7) The grid points are 200.
8) Simulation time is tmax =2 seconds.
9) Simulation step of time of delta tmax=0.01 seconds.

Results and Discussion:
1) We designed the geometry of the turbulence jet in Ansys software with the given parameters Lx=2m, Ly=4m and Lz=0.02m. Figure number 2 is illustrating the geometry of turbulence jet the 200 grid points.

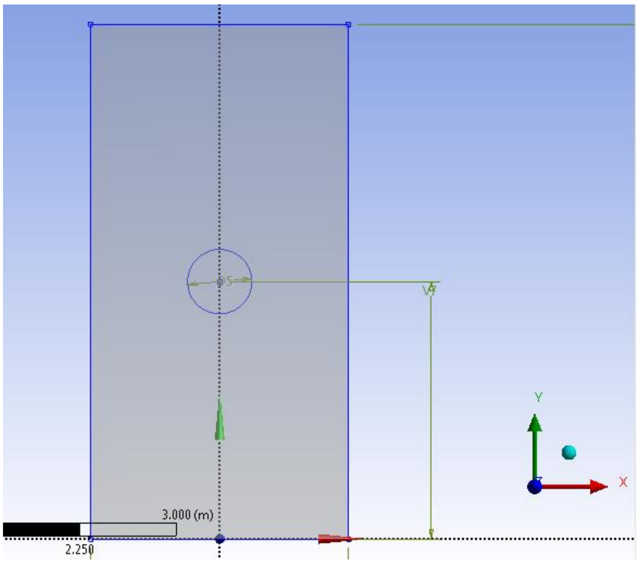


Figure 2a: Geometry of the turbulence jet.

The dimension of the turbulence jet geometry is showing in the figure 2b where D5 is the diameter of the nozzle H1 and H2 are x-axis parameters H=1+1=2m and V4 is the vertical axis 4m.

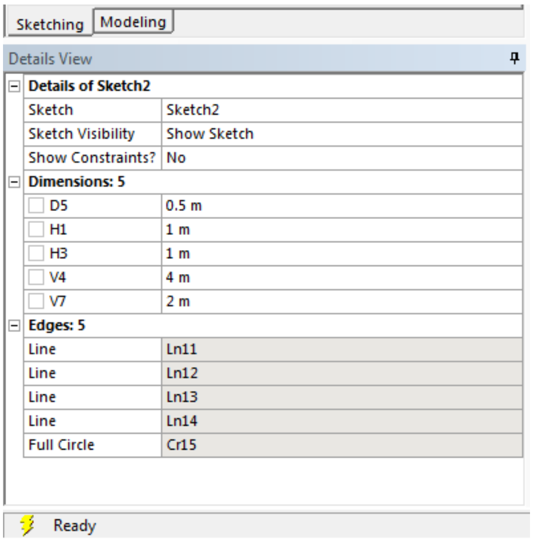


Figure 2b: Dimensions of geometry.

The isometric view of the turbulence is shown below in the figure 2c.

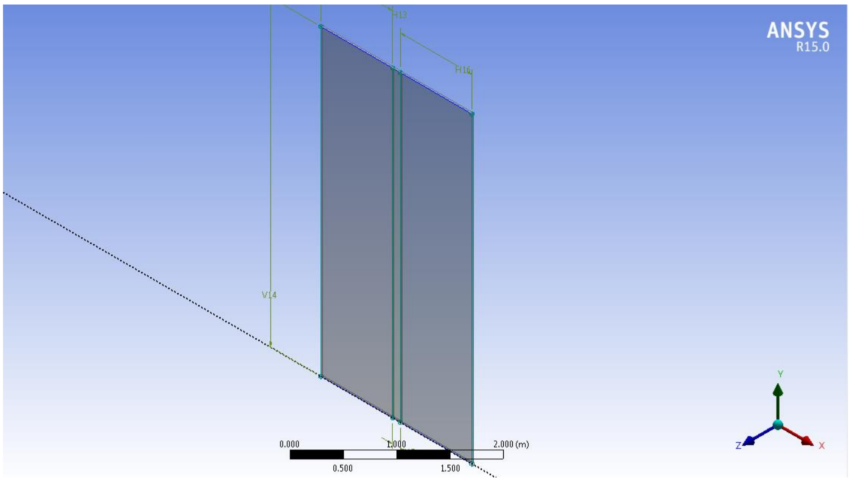


Figure 2b: Isometric view.

We applied mesh to the body design in geometry file the figure 3 is illustrating the mesh view of the turbulence jet.

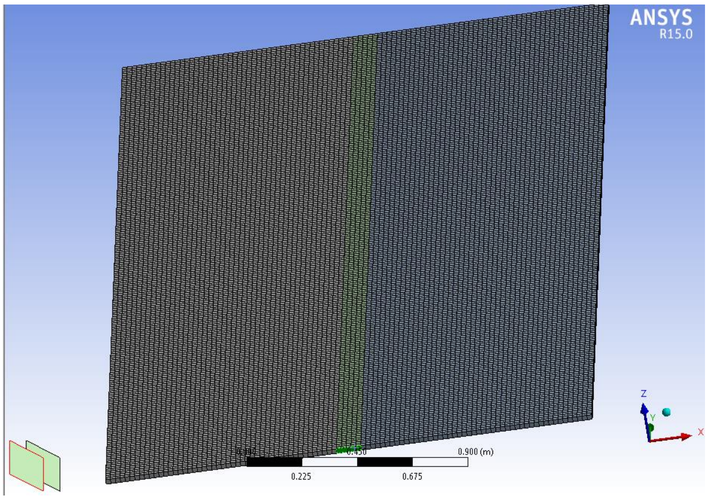


Figure 2b: Mesh view.

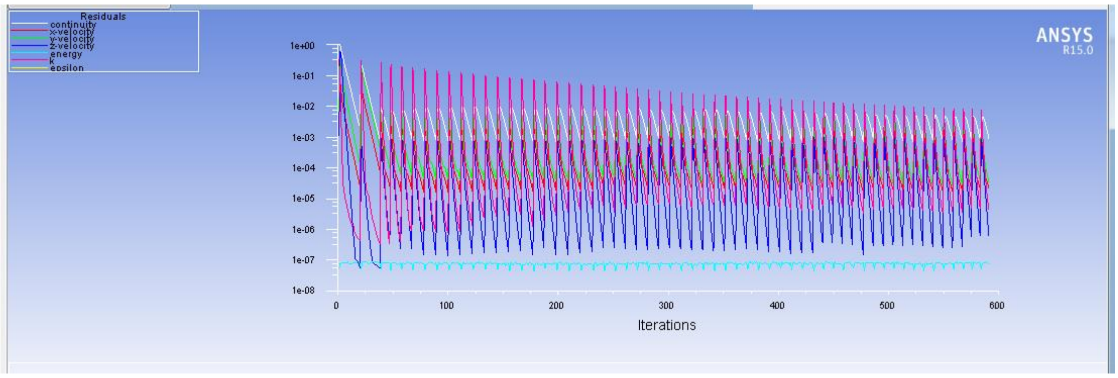
2) We linked geometry file to the fluent and set the solution parameters for the large eddy simulation. The following steps showed the process of simulation.

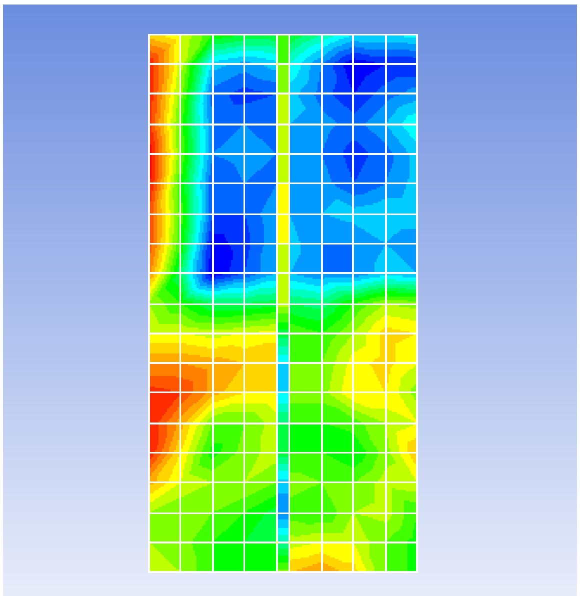
i) In setup for the LES simulation we provide all the solution parameters.
ii) Set boundary Conditions.
iii) User defined function linked with the solution.
iv) Solution Initialization.
v) Performed Calculations.
vi) Plotted the graphs for inlet velocity.
vii) Animation of large eddy simulation.

The following user defined function we linked to the solution of the large eddy simulation.

```
#include "udf.h"

DEFINE_PROFILE(inlet_y_velocity, thread, position)

{

real x[ND_ND], y;
real t=CURRENT_TIME;
#define face_t_f ;
#define f 8314.34;
begin_f_loop(f,thread)

{
F_CENTROID(x, f, thread);
/*y=x[1];*/
F_PROFILE(f,thread, position)=10*(1+0.3*sin(125.6*t));

}
F_CENTROID(x, f, thread);
end_f_loop(f, thread)
}
```

3) After the simulation we obtained the following results for the large eddy simulation of turbulence jet.





4) Table 4 is showing the pulsating inlet velocity at each step of time for the whole duration of 6 seconds.

iter	continuity	x-velocity	y-velocity	z-velocity	time/iter	
1	1.0000e+00	0.0000e+00	0.0000e+00	6.5359e-04	0:00:00	19
2	8.6884e-01	4.7196e-05	4.6805e-05	6.8452e-04	0:00:00	18
3	7.8439e-01	4.4006e-05	4.4204e-05	5.8303e-04	0:00:00	17
4	6.8986e-01	3.6363e-05	3.6475e-05	4.6954e-04	0:00:00	16
5	5.8102e-01	2.9257e-05	2.9208e-05	3.7146e-04	0:00:00	15
6	4.6529e-01	2.4179e-05	2.4155e-05	2.8470e-04	0:00:00	14
7	4.0016e-01	2.1721e-05	2.1745e-05	2.0872e-04	0:00:00	13
8	3.5617e-01	1.8530e-05	1.8533e-05	1.4880e-04	0:00:00	12
9	3.0913e-01	1.4407e-05	1.4375e-05	1.0327e-04	0:00:11	11
10	2.6417e-01	1.0438e-05	1.0381e-05	6.9564e-05	0:00:08	10
11	2.2328e-01	7.2859e-06	7.2307e-06	4.6495e-05	0:00:06	9
12	1.8492e-01	5.1128e-06	5.0679e-06	3.1547e-05	0:00:04	8
13	1.5515e-01	3.7743e-06	3.7514e-06	2.1904e-05	0:00:03	7
14	1.2700e-01	2.7995e-06	2.7781e-06	1.5936e-05	0:00:02	6
15	1.0266e-01	2.0546e-06	2.0445e-06	1.2257e-05	0:00:01	5
16	8.2900e-02	1.5138e-06	1.5125e-06	1.0304e-05	0:00:01	4
17	6.7007e-02	1.1259e-06	1.1367e-06	9.4441e-06	0:00:01	3
18	5.2567e-02	8.4373e-07	8.6134e-07	8.9324e-06	0:00:01	2
19	4.0910e-02	6.3240e-07	6.4446e-07	8.4387e-06	0:00:00	1
20	3.2049e-02	4.8295e-07	4.8813e-07	8.1182e-06	0:00:00	0

20	3.2049e-02	4.8295e-07	4.8813e-07	8.1182e-06	0.0000e+00	0.0000e+00	0:00:04	20
21	8.5798e+02	1.2494e-04	1.2433e-04	4.3885e-02	2.4682e-01	1.2455e+01	0:00:03	19
22	2.9752e+03	3.3189e-03	3.1842e-03	1.1049e-01	2.2146e-01	1.2130e+00	0:00:06	18
23	1.4560e+03	8.2266e-03	7.6795e-03	7.1368e-02	9.4439e-02	1.5913e-01	0:00:05	17
24	7.8686e+02	4.0821e-03	3.7801e-03	4.0956e-02	6.8835e-02	1.1097e-01	0:00:03	16
25	6.2868e+02	3.8468e-03	3.4927e-03	2.9635e-02	6.7677e-02	1.4151e-01	0:00:03	15
26	5.2308e+02	3.2611e-03	3.1015e-03	2.3169e-02	6.8696e-02	1.3927e-01	0:00:02	14
27	3.8719e+02	2.5015e-03	2.4572e-03	1.8668e-02	6.6757e-02	1.2744e-01	0:00:01	13
28	2.9878e+02	1.8836e-03	1.8661e-03	1.5331e-02	6.2153e-02	1.1297e-01	0:00:01	12
29	2.2927e+02	1.4700e-03	1.3836e-03	1.2347e-02	5.5992e-02	9.9168e-02	0:00:01	11
30	1.7795e+02	1.2574e-03	1.1606e-03	9.7695e-03	4.9479e-02	8.6814e-02	0:00:01	10

II. Conclusion:

The focus of the present project was the large eddy simulation for the turbulence jet. Large eddy simulations of jet (Re ≈ 10^6)is obtained from the performance of the contoured convergent-straight nozzle. Toimprove the nozzle interior unsteady flow modeling and ensure a fully turbulent jet we taken particular care In preliminary work, a systematic parametric study of the effects of different modeling within the nozzle interior was conducted, application of synthetic turbulence, wall modeling and focusing on localized adaptive mesh refinement inside the nozzle.

Overall, the results show significant improvement for both flow field and noise predictions when modeling inside the nozzle was applied, compared to the typical approach based on coarse resolution in nozzle and laminar flow assumption commonly used in most jet simulations. With modeling, the nozzle-exit velocity statistics now exhibit fully turbulent profiles similar to the experimental data, and the far-field noise spectra now more closely match the measurements for all angles and most relevant frequencies.

References:

[1]. http://www.bakker.org/dartmouth06/engs150/12-les.pdf
[2]. https://tcg.mae.cornell.edu/pubs/Pope_NJP_04.pdf
[3]. http://www.afs.enea.it/project/neptunius/docs/fluent/html/th/node94.htm
[4]. http://www.cfd-online.com/Forums/fluent/34012-2d-large-eddy-simulation-fluent.html
[5]. http://www.southampton.ac.uk/~nwb/lectures/GoodPracticeCFD/Articles/Turbulence_Notes_Fluent-v6.3.06.pdf
[6]. http://www.plusnetwork.com/?sp=reim5&q=large%20eddy%20simulation
[7]. https://www.youtube.com/watch?v=l8jDBPN4rXo
[8]. http://www1.eere.energy.gov/wind/pdfs/doewater-10971.pdf
[9]. http://fire.nist.gov/bfrlpubs/fire99/PDF/f99064.pdf
[10]. https://web.stanford.edu/group/ctr/Summer/SP14/07_Jet_noise/02_bres.pdf