International Journal of Engineering and Advanced Research Technology (IJEART)
ISSN: 2454-9290, Volume-2, Issue-1, January 2016

Flow Analysis of Intake Manifold using Computational Fluid Dynamics

Ch. Indira Priyadarsini

Abstract—**An inlet manifold or intake manifold is the part of an engine that supplies the fuel/air mixture to the cylinders**. The **intake manifold is essential for the optimal performance of an internal combustion engine. The objective of present paper is to predict and analyze the flow through intake manifold of four cylinder spark ignition engine. One of the important factors is air flow inside the intake manifold; the ideal intake manifold distributes flow evenly to the piston valves. Even distribution is important to optimize the efficiency of the engine. Hence the flow phenomenon inside the intake manifold should be fully optimized to produce more engine power with better combustion and further reduces the emission. Firstly structural analysis has been conducted to set the thickness and material suitability of intake manifold against bursting pressure. Three-dimensional inlet manifold was modeled and numerically analyzed by using the commercially available FLUENT software to study the pressure, velocity and flow characteristics inside the runner. The steady state analysis has been carried out for three different cases: All runners open, 1st & 3rd runners' open and 2 nd & 4th runners open. The predicted results of total pressure loss and total outlet mass flow were discussed. Inlet pipe and plenum connection creates a back step geometry which causes more total pressure loss due to flow recirculation in conventional model. Tapering the geometry is causing more inlet mass flow due to reduction in total pressure loss in the plenum chamber.**

Key words **—Intake manifold, FLUENT, SI engine, Flow analysis**

I. Introduction

An intake manifold is one of the primary components regarding the performance of an internal combustion engine. An intake manifold is usually made up of a plenum inlet duct, connected to the plenum are runners depending on the number of cylinders which leads to the engine cylinder. Intake manifolds have to be designed to improve engine performance by avoiding the phenomena like inter-cylinder robbery of charge, inertia of the flow in the individual branch pipes, resonance of the air masses in the pipes and the Helmholtz effect. Tuning the intake manifold means the intake runners are of proper size and length to produce the highest possible pressure in the cylinder when the intake valve closes. S.Karthikeyan[1] shows, pressure waves for the intake manifold is simulated using 1D AVL-Boost software, to study the internal air flow characteristic for the 3-cylinder diesel engine during transient conditions. The effect of helical, spiral, and helical-spiral combination manifold configuration on air motion and turbulence inside the cylinder of a Direct Injection (DI) diesel engine motored at 3000 rpm. using the CFD tool has been presented by Benny Paul[2]. An investigation of mixture preparation in the intake manifold of a Diesel converted engine into LPG spark-ignition engine operation was explained by M. A. Jemni[3] and he proposed two manifold shapes in order to test the adequate design in view of flow and air-gas homogenization.

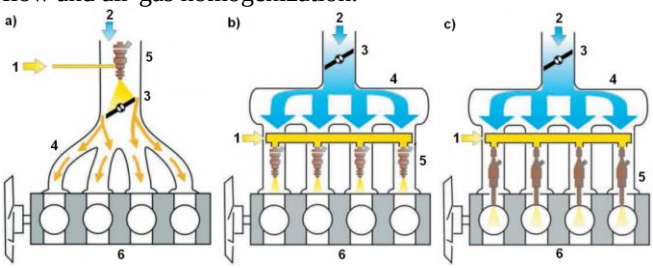

Fig.1 Systems of fuel injection [1]:a) Single Point Injection, b) Multipoint Injection, c) Direct Injection; 1 – Fuel supply, 2 – Air intake, 3 – Throttle, 4 – Intake manifold, 5 – Fuel injector (or injectors), 6 – Engine

Jae-soon Lee[4] performs a study for the optimal design of the intake system by varying the factors which can influence the volumetric efficiency, such as the volume of the plenum chamber, the length of the intake manifold and the pipe length between the surge tank and the plenum chamber. the in-cylinder fluid flow field characteristics of a single-cylinder engine is analyzed by B. Murali Krishna[5] to see the effect of intake manifold inclination at equivalent rated engine speed using Particle Image Velocimetry (PIV) under various static intake valve lift conditions. A.Martínez-Sanz[6] develops a new design of a high performance intake manifold through a combination of CAD and FEM. D. V. Boikov[7] designs the intake passage and shows it has a considerable influence on organization of the processes of mixture formation and combustion in an engine.

II. Methodology

A. Governing equations:

Governing equations:

The conservative form of mass, momentum and energy conservation equations, using Einstein's summation convention over repeated indices, are then given by

Continuity equation:

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = 0$$

Momentum equation:

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial(\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \left[\frac{\partial \sigma_{ij}}{\partial x_j}\right]$$

According to the Stokes's hypothesis which assumes that the bulk viscosity can be neglected, the shear-stress tensor for a Newtonian fluid is given by:

$$\sigma_{ij} = 2\mu(T)S_{ij} - \frac{2}{3}\mu(T)S_{kk}\delta_{ij}$$

Energy equation:

$$c_p\left[\frac{\partial(\rho T)}{\partial t} + \frac{\partial(\rho u_j T)}{\partial x_j}\right] = \frac{\partial p}{\partial t} + u_j\frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j}\left(K\frac{\partial T}{\partial x_j}\right) + \Phi$$

where the viscous dissipation Φ is defined as:

$$\Phi = \sigma_{ij}\frac{\partial u_i}{\partial x_j}$$

B. Steps involved in FLUENT

In CFD simulation following steps is follow:

1. Simplifying the geometry
2. Setting up the model
3. Meshing of the model which includes reduction in geometry complexity
4. Defining boundary conditions
5. CFD-Post for results

III. WORKBENCH MODEL

Geometry is created in ANSYS workbench.

Table1. Dimensions of intake manifold

Section	Diameter (mm)
Intake	38
Outlet 1	30
Outlet 2	30
Outlet 3	30
Outlet 4	30

Table2. Material properties

Aluminum properties	
Youngs' modulus	7e+010N-m2
Poisson's ratio	0.346
Density	2710 kg/m^3
Coefficient of thermal expansion	2.36e^{-5} /^{0}K
Yield strength	9.5e^7 N-m^2

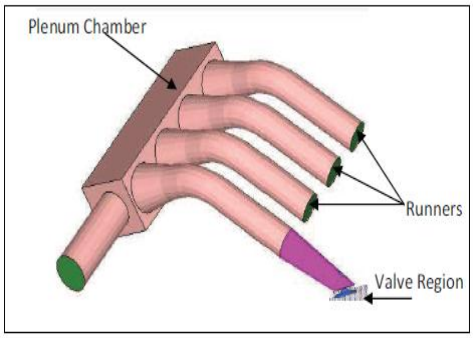

Fig.1 Intake manifold with features

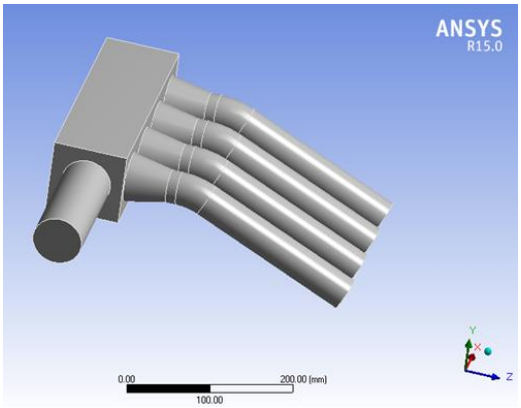

Fig. 2 Manifold Geometry side view

IV. MESHING & BOUNDARY CONDITIONS

Meshing: The accuracy of the results depends highly upon the mesh quality. Thus the choice of meshing scheme (grid pattern) is very important for fluent to provide accurate results. For doing simulation of the intake manifold model we have to do first meshing ,in this technique the flow domain is converted or split into various subdomain primitives like hexahedral and tetrahedral. Care must be taken to ensure proper continuity of solution across the common interfaces between two subdomains, so that the approximate solutions inside various portions can be put together to give a complete picture of fluid flow in the entire domain. We use the tetrahedral mesh for this purpose which imposed on model. Fig. 3 shows the mesh of intake manifold.

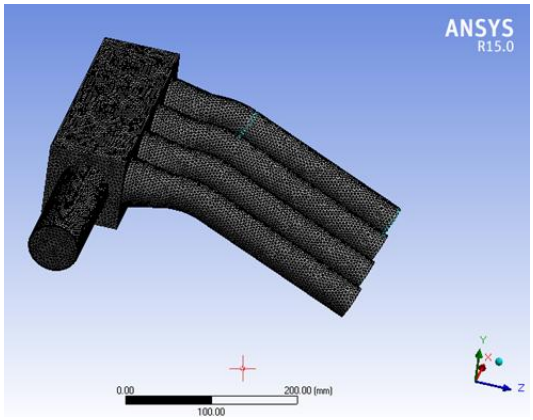

Fig. 3 Mesh of intake manifold

Mesh Inflation Layers:

If we plot a typical velocity profile in the near-wall region, we can see that we have a large change in velocity in the wall normal direction and it is important to CFD simulation that we capture this gradient correctly. To do this, it is required to use inflation layer meshing to accurately capture the boundary layer region for any wall-bounded turbulent flows. Providing a suitable inflation mesh for the geometry is strongly tied to the choice of the turbulence model, and the flow field we are interested in capturing.

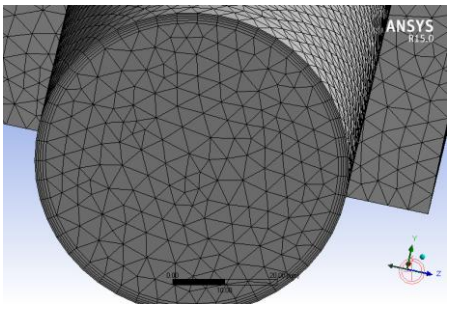

Fig. 4 Inflation inlet

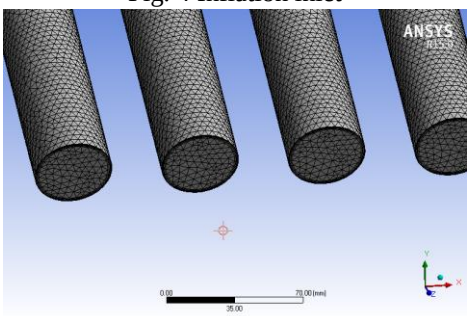

Fig. 5 Inflation outlet

Table 3. Mesh details

Nodes	176396
Elements	575599

Boundary conditions

Boundary conditions are essential to do a simulation. In this problem inlet is open to atmosphere and at outlet suction pressure will act due to pistons down motion. So inlet is chosen as pressure inlet and outlet is chosen as pressure outlet. Pressure inlet boundary condition needs total pressure at the inlet. So from atmospheric condition total pressure at inlet is 0 pa (gauge pressure). The default value of reference pressure in the operation condition is given as 101325. So Gauge pressure (0) = Absolute pressure (101325) – reference pressure (101325). In the Turbulent – specification method needs to choose Intensity and length scale. Turbulent intensity value is assumed as per standard cfd assumption. Turbulent length scale value is assumed as 7% of inlet diameter as per standard cfd assumption. Remaining is needed to keep it as default for this problem.

Table 4. Boundary conditions

Pressure at flow inlet	0 pascal
Intake	38
Pressure at runner outlet	-101325 pascals
Hydraulic diameter	60mm
Turbulence intensity	4.2%
Pressure at runner outlet	-101325 pascals

In the Turbulent – specification method needs to choose Intensity and length scale. Turbulent intensity value is assumed as per standard cfd assumption. Turbulent length scale value is assumed as 7% of inlet diameter as per standard cfd assumption. Remaining is needed to keep it as default for this problem.

V. RESULTS AND DISCUSSION

A. *CASE 1: All runners open*

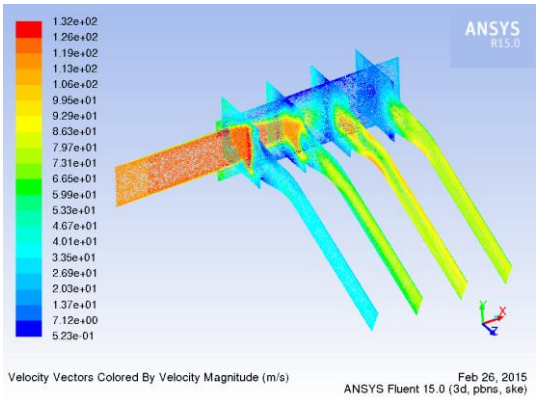

Fig. 6 Velocity vectors (all runners open)

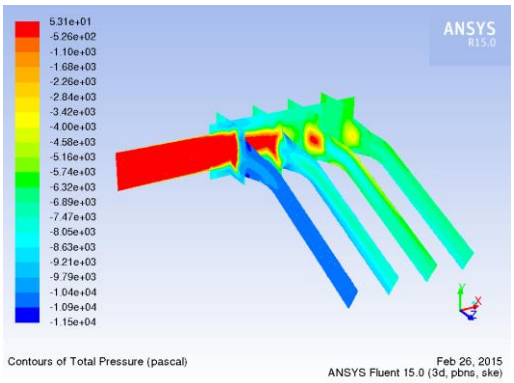

Fig. 7 Pressure contours (all runners open)

Fig. 6 & 7 shows the velocity and pressure contours for all runners open. It is observed that velocity drops as the flow proceeds through the plenum chamber. This is due to sudden increase of the area within the plenum. There is a drop in velocity at the inlet of the runner 1 compared to other runners, due to a sharp bend at the region of runner 1 inlet. Due to the stagnation of fluid occurring at the end corners of the plenum high pressure regions are created.

B. CASE 2: 1st & 3rd runners open:

When 1st and 3rd runners are open, then the other two runners i.e 2nd and 4th are considered as wall in the named selection so that there will be no flow in those runners.

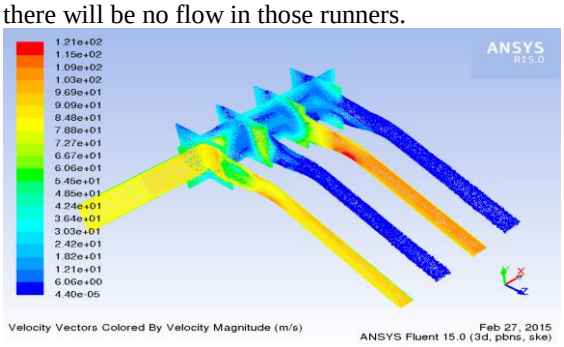

Fig. 8 Velocity vectors (1st & 3rd runner open)

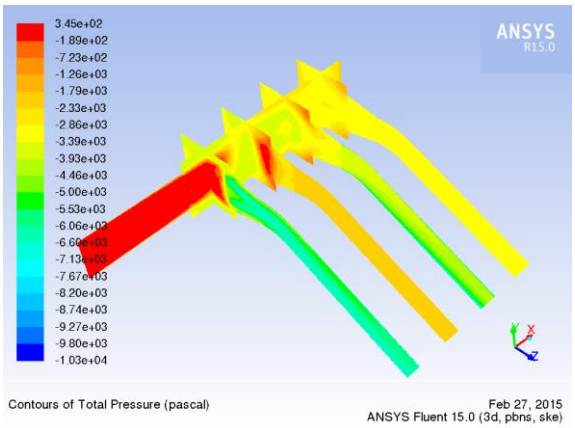

Fig. 9 Pressure contours (1st & 3rd runner open)

From Fig. 8 & 9 it is observed that when runner 1st and 3rd are open the velocity distribution within the plenum changes drastically with higher velocity occurring at inlet region of 1st and 3rd runner. The pressure is higher when 1st and 3rd runners are open compared to 2nd and 4th runner open condition in the plenum. The pressure within is lower comparatively for the condition when all runners are open.

CASE 3: 2nd & 4th runners open

.

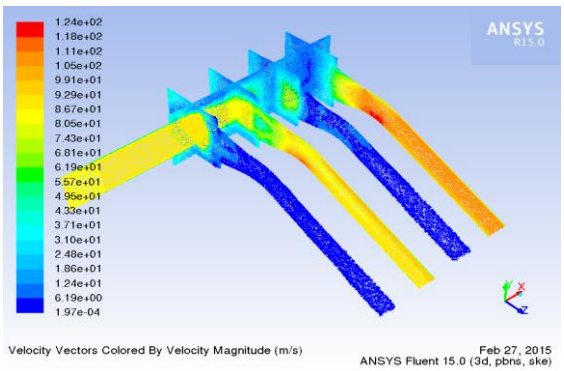

Fig. 10 Velocity Vectors (2nd & 4th runner open)

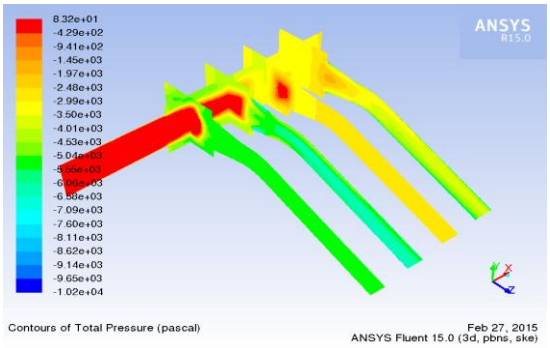

Fig. 11 Pressure contours (2nd & 4th runner open)

Fig. 10 & 11 shows the velocity and pressure flow when 2nd and 4th runners in open condition. It is observed that when 4th runner is open the velocity increases at the inlet region and the pressure is low when 2nd and 4th runners are open compared to 1st and 3rd runner open condition in the plenum. The pressure within is lower comparatively for the condition when all runners are open.

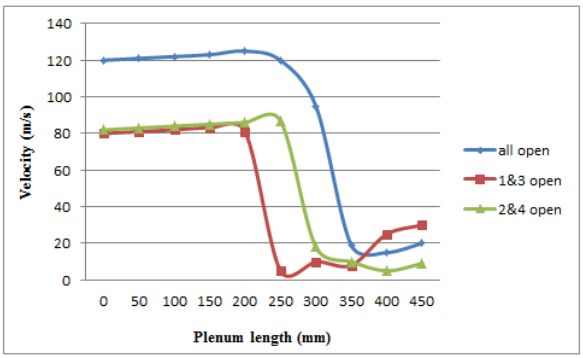

Fig. 12 Velocity variations with plenum length

From Fig. 12 it is observed that a sudden drop in velocity occurs for all the three conditions as the flow occurs from inlet of the plenum to the runner.

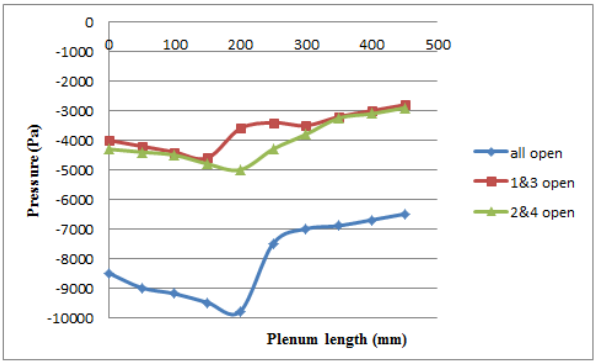

Fig. 13 Pressure variations with plenum length

From Fig.13 it is observed that the pressure is higher when 1st and 3rd runners are open compared to 2nd and 4th runner open condition in the plenum. The pressure within is lower comparatively for the condition when all runners are open.

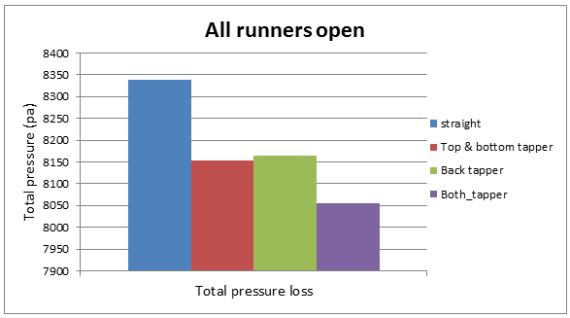

Fig. 14 Total pressure of different geometries

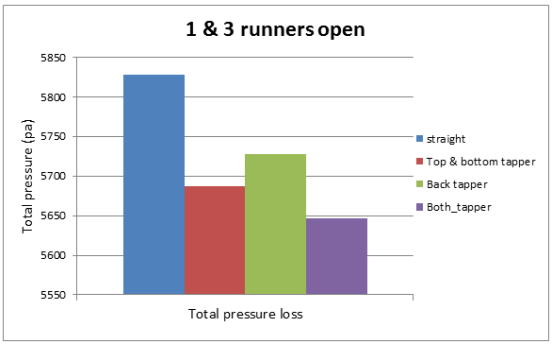

Fig. 15 Total pressure (1&3 open)

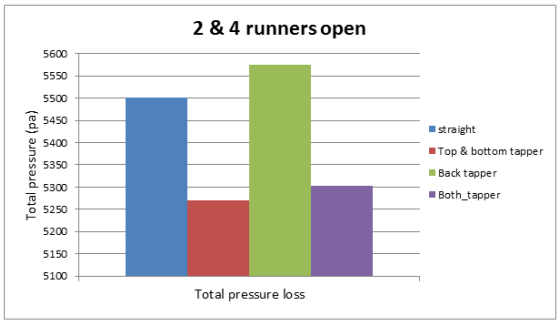

Fig. 16 Total pressure (2&4 open)

When compare with base model as well as all modified models

- Due to less total pressure loss, inlet mass flow rate of modified geometry of all taper is higher than others. Tapering is not causing any impact on the recirculation zone reduction. So except Runner 1 remaining Runners are improved in carrying the mass flow

CONCLUSIONS

After analysis, the concluded points are as follows:

- All the cases show that Runner 4 is working efficiently because it's located away from the pressure loss or recirculation region.
- Total pressure loss for case 1(all runners open) is 8338 pa, for case 2(1st and 3rd runners open) is 5827 pa, for case 3(2nd and 4th runners open) is 5500 pa.
- Outlet total pressure for Case 1(all runners open) is runner 1 is -10543 pa, runner 2 is -8415 pa, runner 3 is -6896 pa, runner 4 is -7499 pa. Case 2(1st and 3rd runners open) is runner 1 is -6554 pa, runner 3 is -4807 pa. Case 3(2nd and 4th runners open) is runner 2 is -6193 pa, runner 4 is -4769 pa.

ACKNOWLEDGMENT

I would like to express my deepest appreciation to all those who provided me the possibility to complete this paper. A special gratitude I give to our M.Tech thermal engineering project co-coordinator Dr.M.V.S.Murali Krishna, whose contribution in stimulating suggestions and encouragement, helped me to coordinate my project students especially in writing this paper.

REFERENCES

[1] Karthikeyan S, Hariganesh R, Sathyanadan M, Krishnan S, " Computational Analysis Of Intake Manifold Design And Experimental Investigation On Diesel Engine For LCV", ISSN: 0975-5462, vol. 3 no. 4, (March 2011).

[2] Benny Paul, Ganesan V, "Flow Field Development In a Direct Injection Diesel Engine With Different Manifolds", IJEST: Vol. 2, No. 1, pp. 80-91, (2010).

[3] Jemni M .A, Kantchev G, Abid M. S, "Intake Manifold Design Effect On Air Fuel Mixing And Flow For An LPG Heavy Duty Engine", IJEE:Vol. 3, Issue 1, pp.61-72, , (2011).

[4] Jae-soon Lee, Keon-Sik Yoon, "A Numerical and Experimental Study on the Optimal Design for the Intake System of the MPI Spark Ignition Engines" , KSME Journal, Vol 10, No. 4 pp.471-479, (1996).

[5] *B. Murali Krishna, A. Bijucherian, and J. M. Mallikarjuna, "Effect of Intake Manifold Inclination on Intake Valve Flow Characteristics of a Single Cylinder Engine using Particle Image Velocimetry" , World Academy of Science, Engineering and Technology 68 (2010).*

[6] Martínez-Sanz A., Sánchez-Caballero S, Viu A. and Pla-Ferrando R. , " Design And Optimization Of Intake Manifold In A Volkswagen Car" ANNALS of the ORADEA UNIVERSITY. Fascicle of Management and Technological Engineering, Vol X (XX) ,NR2, (2011).

[7] *Boikov and M. A. Grigorev. "Fouling Of Diesel Engine Intake Passage With Carbonaceous Deposits" , Chemistry and Technology of Fuels and Oils, Vol.33,No.3,(1997).*

[8] Gary D. Bourn, Ford A. Phillips and Ralph E. Harris, "Technologies To enhance The Operation of Existing Natural Gas Compression Infrastructure Manifold Design For Controlling Engine Air Balance, DOE Award No. DE-FC26-02NT41646, (2002).

[9] *Rosli Abu Bakar, "Design and Development of a New CNG (compressed natural gas) Engine", Vol. 72351, (2002).*

[10] *Mardani Ali Sera, Rosli Abu Bakar and Azhar Abdul Aziz, "Effect Of Air Fuel Mixer Design On Engine Performance And Exhaust Emission Of A CNG Fuelled Vehicles", WEC 22-25 (July2002).*

Ch. Indira Priyadarsini

Completed M.Tech in thermal engineering from JNTUH, Hyderabad and registerd Ph.D in Osmania University, Hyderabad.
I am interested in Internal combustion engines: performance, emission, design and analysis using FEM & FLUENT