

## Computation Fluid Dynamics Simulation of Spark Ignition Engine for Gaseous Fuels by Varying Crank Angles



**G.V. Nagamani, M.Tech, MISTE, AMIE**  
Associate Professor,  
Department of Mechanical Engineering,  
Global College of Engineering & Technology,  
Kadapa, A.P.



**K.Sreelatha**  
Post Graduate Student,  
Department of Mechanical Engineering,  
Global College of Engineering & Technology,  
Kadapa, A.P.

### ABSTRACT:

The internal combustion engine is a heat engine that converts chemical energy of fuel into mechanical energy. Gaseous fuels occur in nature, besides being manufactured from solid and liquid fuels. Gaseous fuels offer several advantages such as clean combustion, high octane number, high availability, and attractive price. They have lower energy content than conventional fuels like petrol or diesel, however they can potentially deliver greenhouse gas emission reductions and air quality benefits compared to conventional fuels. The actual level of benefits in emission reductions varies greatly depending on the gas type, fuel quality, engine technology and driving behaviour. The main aim of this thesis is to simulate the S.I Engine using gaseous fuels. Methane is used as a fuel in this analysis. CFD analysis is done in Ansys IC engine module in the component manifold. The CFD simulation involves running the engine with a constant Compression ratio, Equivalence ratio and at different crank angles. The crank angles are changed from 750deg to 700deg at approximately 2deg variations. The 2D model of manifold and CFD analysis is done in Ansys. The flow of fuel is turbulent K – Epsilon Model. The results determined for observation in this analysis are mass fraction of fuel and temperature distribution inside the manifold at different crank angles.

### 1.LITERATURE REVIEW:

In the paper by Kota Sridhar, etal, simulates the Intake, Compression, Expansion and Exhaust process with hexane fuel combustion. In-Cylinder model for simulating reciprocating.

engine, General strategies for modelling valve opening, closing and Discrete phase Model (DPM) for simulating port injection. In the paper by Jie Wang, etal, a numerical simulation of the influence of different hydrogen fractions, excess air ratios and EGR mass fractions in a spark-ignition engine was conducted. Good agreement between the calculated and measured incylinder pressure traces as well as pollutant formation trends was obtained. In the paper by by İsmet Sezer, the theoretical exergetic evaluation of spark ignition engine operation.

For this purpose, a two-zone quasi-dimensional cycle model was installed, not considering the complex calculation of fluid motions. The cycle simulation consists of compression, combustion and expansion processes. The combustion phase is simulated as a turbulent flame propagation process. Intake and exhaust processes are also computed by a simple approximation method.

### AIM OF THE THESIS:

The main aim of this thesis is to simulate the S.I Engine using gaseous fuels. Methane is used as a fuel in this analysis. CFD analysis is done in Ansys IC engine module in the component manifold. The CFD simulation involves running the engine with a constant Compression ratio, Equivalence ratio and at different crank angles. The crank angles are changed from 750deg to 700deg at approximately 2deg variations.

The 2D model of manifold and CFD analysis is done in Ansys. The results determined for observation in this analysis are mass fraction of fuel and temperature distribution inside the manifold at different crank angles.

## 2.INTRODUCTION:

Internal combustion engines are one of the building blocks of modern civilization. In an internal combustion engine, the combustion takes place inside a confined chamber. All internal combustion engines burn a mixture of air & fuel. The fuel can be gasoline, diesel, methane, propane etc. The piston is the heart of an internal combustion engine. The concept of the piston engine is that a supply of air-and-fuel mixture is fed to the inside of the cylinder where it is compressed and then burnt. This internal combustion releases heat energy which is then converted into useful mechanical work as the high gas pressures generated force the piston to move along its stroke in the cylinder. It can be said, therefore, that a heat-engine is merely an energy transformer. To enable the piston movement to be harnessed, the driving thrust on the piston is transmitted by means of a connecting-rod to a crankshaft whose function is to convert the linear piston motion in the cylinder to a rotary crankshaft movement. The piston can thus be made to repeat its movement to and fro, due to the constraints of the crankshaft crankpin's circular path and the guiding cylinder.

### 2.1.ANALYSIS OF IC ENGINE AT DIFFERENT CRANK ANGLES:

Use FLUENT Launcher to start the 3D version of ANSYS FLUENT, File-Read\_mesh Brows meshed file  
General Settings - Define the solver settings - General -Transient, Dynamic Mesh

- i. Enter 0.1 for Collapse Factor under Layering tab.
- ii. Click OK to close the Mesh Method Settings dialog box.
- iii. Enable In-Cylinder in the Options group box.
- iv. Click Settings... to open In-Cylinder Settings dialog box.

Enter the values as shown in the Table

Parameter	Value
Crank Shaft Speed (rpm)	2000
Starting Crank Angle (deg)	360
Crank Period (deg)	720
Crank Angle Step Size (deg)	0.25
Piston Stroke (mm)	120
Connecting Rod Length (mm)	220
Piston Stroke Cutoff (mm)	0
Minimum Valve Lift (mm)	0

Values for IC Parameters

Click OK to close In-Cylinder Settings dialog box.

### Set up dynamic zones Dynamic mesh—create/edit

#### Define dynamic mesh zone for fluid-outer

- i. Select fluid-outer from the Zone Names drop-down list.
- ii. Ensure that Rigid Body is selected in the Type group box.
- iii. Ensure the selection of **\*\*piston-full\*\*** from the Motion UDF/Pro.  
i.e: dropdown list under Motion Attributes tab.
- iv. Enter the values of X, Y, and Z as 0, 0, and 1 respectively.
- v. Click Create.

#### Define dynamic mesh zone for bowl

- i. Select bowl from the Zone Names drop-down list.
- ii. Retain the default settings for Motion Attributes tab.
- iii. Enter 0.8 mm for Cell Height under Meshing Options tab.
- iv. Click Create.

#### Define dynamic mesh zone for bowl: 019.

- i. Select bowl: 017 from the Zone Names drop-down list.
- ii. Retain the default settings for Motion Attributes tab.
- iii. Retain 0.8 mm for Cell Height under Meshing Options tab.
- iv. Click Create.

#### Define dynamic mesh zone for wall top outer

- i. Select wall top outer from the Zone Names drop-down list.
- ii. Select Stationary in the Type group box.
- iii. Retain 0.8 mm for Cell Height under Meshing Options tab.
- iv. Click Create.

Close the Dynamic Mesh Zones dialog box.

### Creating mesh interfaces:

Mesh interfaces----create/edit

- (a) Enter intf for Mesh Interface.

- (b) Select intf-inner from Interface Zone 1 list.
- (c) Select intf-outer from Interface Zone 2 list.
- (d) Click Create and close the Create/Edit Mesh Interfaces dialog box
- Check mesh:-
- General ----check
- Perform a mesh motion preview
- Display the mesh
- Graphics and animations ---mesh—setup

### Preview the mesh motion

Dynamic mesh---preview mesh motion

- i. Enter 1360 for Number of Time Steps .
- ii. Click Apply and then click Preview.
- Save the case file
- File---write---case
- Compile and load the UDF library
- Define ----user defined---functions----compiled

Click Add... to open Select File dialog box.

- i. Select initialize.c and injection ch4.c from the Source Files list.
- ii. Click OK to close the Select File dialog box.
- Click Build to build the library.
- Click Load to close the Compiled UDFs dialog box.
- Define----user defined----function Hooks
- Click Edit... for Initialization to open Initialization Functions dialog box.
- i. Select my init function::libudf from the Available Initialization Functions list.
- ii. Click Add to add it in the Selected Initialization Functions list.
- iii. Click OK to close Initialization Functions dialog box.
- Close User-Defined Function Hooks dialog box.
- Enable the standard k-epsilon turbulence model

### Models---viscous---Edit

- (a) Select k-epsilon in the Model group box.
- (b) Retain the default settings and click OK to close Viscous Model dialog box.
- Define the species model.:-

### Models—species---edit

- (a) Select Non-Premixed Combustion in the Model group box.
- (b) Click the Chemistry tab.

- i. Ensure Equilibrium is selected in the State Relation group box.
- ii. Select Non-Adiabatic in the Energy Treatment group box.
- iii. Enter 3000000 pascal for Equilibrium Operating Pressure.
- iv. Retain 0.1 for Fuel Stream Rich Flammability Limit.
- v. Enable Inlet Diffusion and Compressibility Effects in the PDF Options group box.

### Click the Boundary tab

- i. Select Mole Fraction in the Specify Species in group box.
- ii. Enter the values as shown in the Table.

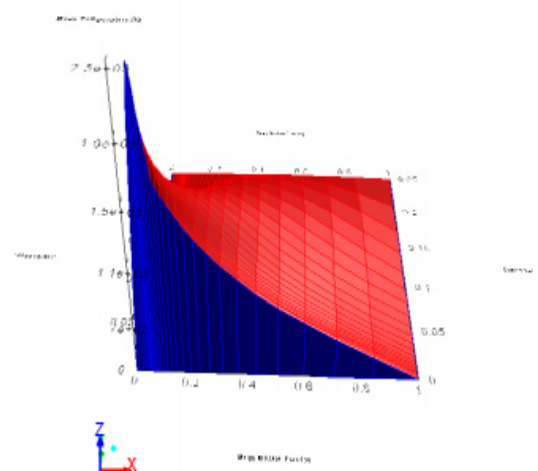
Species	Fuel	Oxid
ch4	1	0
n2	0	0.78992
o2	0	0.21008

Fuel and Oxid Values for Species

Retain 300 K for Fuel and Oxid in the Temperature group box.

Click the Table tab.

- i. Retain the default settings.
- ii. Click Calculate PDF Table.
- iii. Click Display PDF Table... to open PDF Table dialog box.



Close the PDF Table dialog box.

Click OK to close the Species Model dialog box

Save PDF file

File---write---pdf

## Boundary Conditions:

Boundry conditions---Inlet

Select mass flow-inlet from the Type drop-down list

(a) Ensure that Absolute is selected from the Reference Frame drop-down list.

(b) Select Mass Flux from the Mass Flow Specification Method drop-down list.

(c) Select udf fuel -ux::libudf from the Mass Flux drop-down list.

(d) Select Normal to Boundary from the Direction Specification Method drop-down list.

(e) Select Intensity and Length Scale from the Specification Method drop-down list.

(f) Enter 2 mm for Turbulent Length Scale.

(g) Click the Species tab.

i. Enter 1 for Mean Mixture Fraction.

ii. Retain 0 for Mixture Fraction Variance.

(h) Click OK to close Mass-Flow Inlet dialog box.

## Solution:

(a) Select PISO from the Scheme drop-down list.

(b) Set Skewness Correction to 0.

(c) Retain Standard for Pressure drop-down list.

(d) Select Second Order Upwind from the Momentum drop-down list.

(e) Retain First Order Upwind for all other equations.

Set the under-relaxation factors.

## Solution control:

Set the Under-Relaxation Factors for Pressure to 0.5

Enable the plotting of residuals during calculation

Monitors---residuals----Edit

(a) Enter 100 for Iterations to Plot.

(b) Click OK to close the Residual Monitors dialog box.

Set up surface monitor for mass flow rate.

## Monitors----create

(a) Enable Plot and Write in the Options group box.

(b) Enter mass flow rate inlet.out for the File Name.

(c) Select Flow Time from X Axis drop-down list.

(d) Select Time Step from the Every drop-down list.

(e) Select Mass Flow Rate from Report Type drop-down list.

(f) Select inlet from Surfaces list.

(g) Click OK to close the Surface Monitor dialog box.

Set up volume monitor for mass-averaged temperature.

## Monitors---create

(a) Enable Plot and Write in the Options group box.

(b) Enter vol temp. out for the File Name.

(c) Select Flow Time from X Axis drop-down list.

(d) Select Time Step from the Every drop-down list.

(e) Select Mass-Average from the Report Type drop-down list.

(f) Select Temperature... and Static Temperature from the Field Variable drop-down list.

(g) Select fluid-inj, fluid-outer, and fluid-inner from the Cell Zones list.

(h) Click OK to close the Volume Monitor dialog box.

Set up volume monitor for mass-averaged pressure.

## Monitors-----create

(a) Enable Plot and Write in the Options group box.

(b) Enter vol press .out for the File Name.

(c) Select Flow Time from X Axis drop-down list.

(d) Select Time Step from the Every drop-down list.

(e) Select Mass-Average from the Report Type drop-down list.

(f) Select Pressure... and Static Pressure from the Field Variable drop-down list.

(g) Select fluid-inj, fluid-outer, and fluid-inner from the Cell Zones list.

(h) Click OK to close the Volume Monitor dialog box.

Solution initialization - Initialize the solution.

a. Enter 1898675 Pascal for Gauge Pressure.

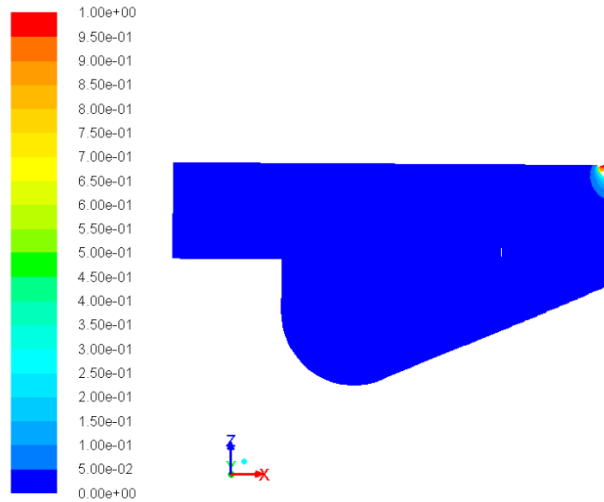
(b) Retain 1 m<sup>2</sup>/s<sup>2</sup> for Turbulent Kinetic Energy and Turbulent Dissipation Rate.

(c) Enter 690 K for Temperature.

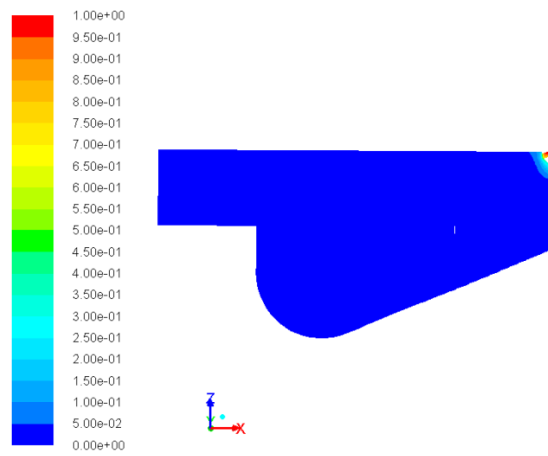
Click Initialize.

Run calculation

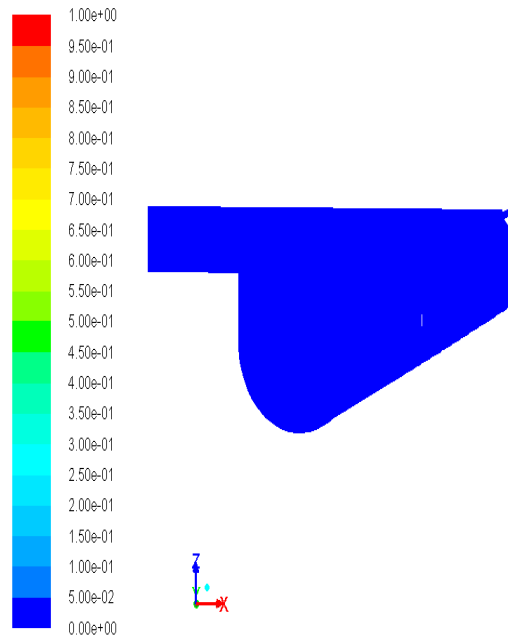
RESULTS OF MASS FRACTIONS AT DIFFERENT CRANK ANGLE



Contours of Mass fraction of ch4 (Time=4.1667e-03)  
Crank Angle=750.00(deg) ANSYS FLUENT 13.0 (3d, dp, pbns, dynamesh, pdf20, ske, transient)

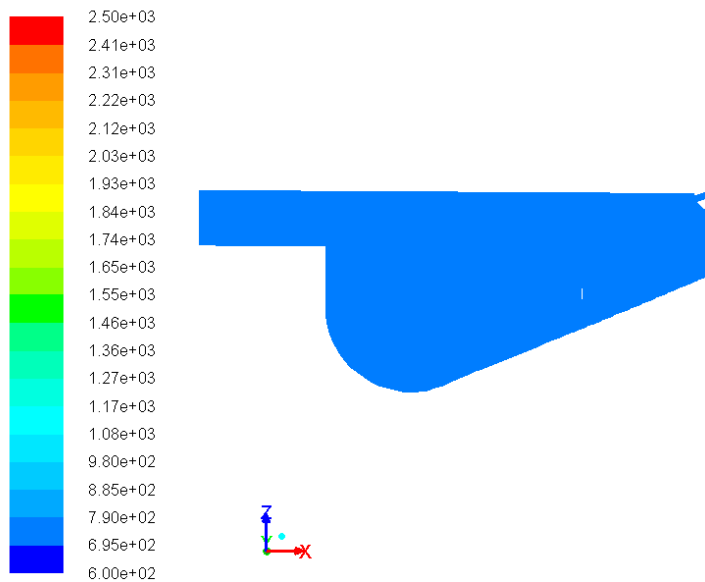


Contours of Mass fraction of ch4 (Time=3.9583e-03)  
Crank Angle=747.50(deg) ANSYS FLUENT 13.0 (3d, dp, pbns, dynamesh, pdf20, ske, transient)

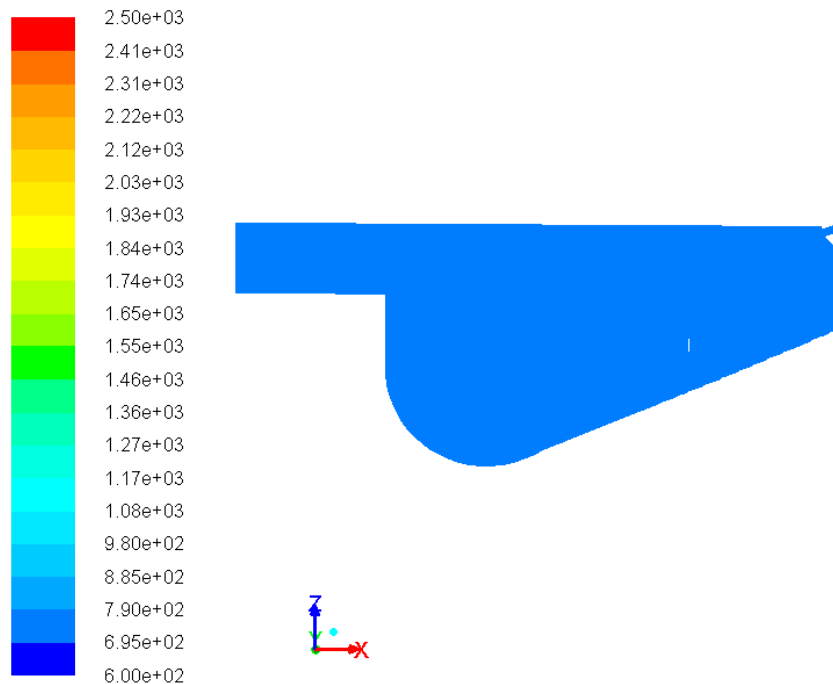


Contours of Mass fraction of ch4 (Time=1.0417e-04)  
Crank Angle=701.25(deg) ANSYS FLUENT 13.0 (3d, dp, pbns, dynamesh, pdf20, ske, transient)

**TEMPERATURE AT DIFFERENT CRANK ANGLE**



Contours of Static Temperature (k) (Time=2.0833e-04)  
Crank Angle=702.50(deg) ANSYS FLUENT 13.0 (3d, dp, pbns, dynamesh, pdf20, ske, transient)



Contours of Static Temperature (k) (Time=1.0417e-04)  
Crank Angle=701.25(deg) ANSYS FLUENT 13.0 (3d, dp, pbns, dynamesh, pdf20, ske, transient)

## CONCLUSION:

The results determined for observation in this analysis are mass fraction of fuel and temperature distribution inside the manifold at different crank angles. By observing the analysis results, the following observations can be made: The mass fraction (i.e) air to fuel ratio is more at the inlet of the manifold and decreasing gradually inside the manifold from crank angle 750deg to 721.25deg with elapsed time of 1.7708e-3secs.

After 721.25deg, the air fuel mixture is in the range of 0.05, it is not changed with change of crank angle. So, the maximum power is reached when crank angles are varying from 721.25 to 701.25deg, since the mass fraction is less (i.e) air fuel mixture is richer. The static temperature is reached up to 25000C inside the manifold from crank angle 750deg to 721.25deg.

After 721.25deg, the temperature has remained constant at around 7900C to 6950C. So it can be concluded that the crank angles between 721.25deg to 701.25deg gives the better performance.

## REFERENCES:

1. Computerised Simulation of Spark Ignition Internal Combustion Engine by Kota Sridhar, R.B.V.Murali, Sk.Mohammad Younus, K.Mohan Lakshmi, IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE), e-ISSN: 2278-1684 Volume 5, Issue 3 (Jan. - Feb. 2013), PP 05-14
2. Simulation of combustion in spark-ignition engine fuelled with natural gas-hydrogen blends combined with EGR by Jie Wang, Zuohua Huang, Bing Liu, Xibin Wang, Frontiers of Energy and Power Engineering in China, June 2009, Volume 3, Issue 2, pp 204-211.
3. Mathematical analysis of spark ignition engine operation via the combination of the first and second laws of thermodynamics by İsmet Sezer, Atilla Bilgin, Published 8 December 2008. DOI: 10.1098/rspa.2008.0190
4. CFD Analysis of in-Cylinder Flow and Air-Fuel Interaction on Different Combustion Chamber Geometry in DISI Engine by B. Harshavardhan & J. M. Mallikarjuna, International Journal on Theoretical and Applied Research in Mechanical Engineering (IJTARME), ISSN : 2319 – 3182, Volume-2, Issue-3, 2013.

5.Engine Performances and Emissions of Second-Generation Biofuels in Spark Ignition Engines: The Case of Methyl and Ethyl Valerates by Francesco Contino, Fabrice Foucher, Fabien Halter, Guillaume Dayma, Philippe Dagaut, Christine Mounaïm-Rousselle, SAE International, 2013-24-0098, DOI:10.4271/2013-24-0098.

6.Effect of port gas injection on in-cylinder stratification in a CNG fuelled SI engine by Manish Garg; R.V. Ravikrishna, Progress in Computational Fluid Dynamics, An Int. J., 2014 Vol.14, No.4, pp.244 – 258

7.3-D CFD Analysis of the Mixture Formation Process in an LPG DI SI Engine for Heavy Duty Vehicles by Gisoo Hyun and Mitsuharu Oguma, NEDO (NewEnergy and Industrial Technology Development Organization

8.Impact of Fuel Properties and Flame Stretch on the Turbulent Flame Speed in Spark-Ignition Engines by Pierre Brequigny, Christine Mounaïm-Rousselle, Fabien Halter, Bruno Moreau, Thomas Dubois, SAE Papers, 2013-24-0054, DOI: 10.4271/2013-24-0054

9.Combustion Modeling Of S.I. Engine For Prediction Of Turbulent Flame Speed by Madhusudan Chunilal Barot, Sudeep Sunil Kolhar, Anurag Mani Tripathi, International Journal of Engineering Research & Technology, Vol.2 - Issue 4 (April - 2013), e-ISSN: 2278-0181

10.CFD Analysis of I.C. Engine Processes by Ashish Rangpariya, Swapnil Solanki, D.C.Solanki, IJSART - Volume 1 Issue 5 –MAY 2015 ISSN: 2395-1052

## **FUTURE WORK:**

1.Air to fuel ratio can be determined for different gaseous fuels.

2.By varying crank angles power reaches maximum and engine performance can be improved.

3.By varying crank angles at manifolds temperature decreases and mass fuel ratio can be improved.

4.By changing the different fuels mass fuel ratio can be determined so that better fuel is selected for better performance of the engine.