ABSTRACT:
A Dual Fuel IC Engine is designed on the concept of the injection of a liquid fuel to start the combustion of the working mixture of a gaseous fuel and air in the engine cylinder. Due to stricter air pollution controls and the rise in the cost of fuels in general and relatively reduced availability of liquid fuels of right quality, the demand for gas-fuelled engine applications began increasing, mainly in the stationary electric power generation and commercial vehicle sectors. However the design and development of such engine needs a lot of time and costs for the conventional prototyping and experimental setup of any IC Engine. These time and cost constraints can be overcome by doing CFD simulation using Ansys Fluent and Ansys IC Engine packages. These simulation packages helps to design and development of efficient IC Engines by predicting the heat transfer rate along with temperature, pressure and pollutants.

Also by understanding the in-cylinder combustion process of dual fuel combustion engine helps us to develop low emission engines. A multidimensional 2D model of Dual Fuel IC Engine was developed using solid works based on which the flow, heat and pollutant analysis was done using Ansys IC Engine package. The meshing was done by tetrahedral element using copper tool. The results were analyzed to get the values of heat transfer rate, temperature, dynamic & average pressures, the torque generated and emissions of combustion process. To validate the Ansys ICE simulation the results were compared with the experimental values and also with Ansys Fluent simulation and found that the predicted values are in conformance with the experimental results.

Keywords:
CFD simulation, Ansys IC Engine, Natural-gas based, Dual Fuel IC Engine, Rate of Heat transfer, Combustion modeling, Diesel combustion.
Fuel Type: Mixture of natural gas + air + other minor constituents

Next, the Valve Lift profile was uploaded.

Next, the Geometry menu of the ICE instance was double clicked to open the Design Modeler (DM) window. Here the previously created Geometry STEP file was imported into the Modeler and the Generate button was clicked to get the Geometry appeared in the DM window. Then a spark point is created with appropriate co ordinates like x, y, z. Then the Input Manager was added to the Design view. In the InputManager the configuration was done to select the two of cylinder faces, Inlet valve and Exhaust Valve. In the Inlet Valve button the Valve body and two faces are selected and necessary parameters such as spray height, spray radius, pilot ignition pipe angle etc were applied. Similarly, the Exhaust valve is also added and corresponding valve body and two faces were selected. The Geometry was updated and then it is decomposed by clicking on Decompose button. Next in the ‘Mesh’ section of the ICE the meshing parameters such as mesh type = Medium, Element length=10 mm etc with Tetrahedral meshing option were set and chosen Animate option to ‘yes’. The Generate option will create the generation of Mesh on the selected surfaces. The Final step, is setting up the solver and running Calculation

III SOLVER SETUP
The solver setup menu is available next to the generate mesh menu. Below are various settings and parameters in each setting that were used to run the solution to solve the equations. Basic settings: Autosave type: Crank Angle and ICE Swirl Number: 1.3
Physics settings: Species
Model: Laminar Flow rate,

Material Input: Ansys ICE Fluent
Mixture Material: Natural gas-air
Boundary Conditions: ice-outlet-exvalve, ice-outlet-invalve

Initialization parameters such as: x,y,z co ordinates,
Temperature=300 K, Turbulent K.E, Turbulent Dissipation Rate, c7h16 and below other parameters are setup
Location of spray: Height: 22 mm from the top of the cylinder,
Spray radius: 15 mm,
Inlet valve angle: 14 degrees,
Pilot injection pipe angle: 90 degrees,
Pilot injection time: 1.5 sec

Engine specifications used:
Bore: 84 mm
Stroke length 82 mm
Connecting rod length: 122 mm
Crank radius: 20.5 mm
Displacement: 1817cc
Engine speed=1800 RPM
Compression Ratio: 16:1. For this study the main fuel used is the mixture of natural gas with air. The Diesel was used for pilot ignition purpose with an ignition time of 1.5 seconds.

IV FLUID FLOW SIMULATION & RATE OF HEAT RELEASE
The distribution of temperature and other thermal quantities under a transient thermal analysis is conducted to obtain the values of Heat transfer rate at various mass flow rates of the natural gas + air fuel mixture.
The rate of heat transfer, pressures, temperature contours and torque are obtained for various mass flow rates. These observations were in turn used to calculate the Efficiency and Brake power.

Figure 3: Total Heat Release rate

Figure 4.1: Total Heat Release rate Vs Crank Angle

Figure 4.2: Total Heat Release rate Vs Crank Angle

V. COMBUSTION SIMULATION

In the simulation of Dual Fuel Engines, the combustion modeling mainly deals with three processes, the first two being of the pilot ignition of Diesel Fuel [3] and third process is of the main combustion of the mixture of natural gas and air. The intermediate species produced by auto ignition in the first process trigger high temperature reactions subsequently in the second process which releases some heat energy. In case of Dual Fuel Engines, the heat released from the above second process ignites the main fuel consisting of natural gas + air mixture, which triggers high temperature reactions and causes the main heat release as well as further release of complete and incomplete combustion products. By choosing time scale combustion model, we can ensure the significance of effect of turbulence on the combustion process.

This combustion of dual fuel i.e. Diesel + Mixture of natural gas and air was simulated using Ansys IC Engine package. A single cylinder, single zone, multidimensional Combustion model was applied. The natural gas composition, Diesel and air properties were copied from various web resources, for combustion analysis. This combustion model includes both laminar and turbulent time scales to calculate the equilibrium concentration of each species. The major species like Methane, Ethane, Propane, Butane and other minor species like Pentane, Hexanes, Nitrogen, Carbon Dioxide, Oxygen etc were used for combustion species [2] and corresponding contours of pollutants such as HCN, NH3, NOx, CO2 etc were obtained.

VI. RESULTS AND DISCUSSIONS

1. HEAT TRANSFER RATE AT VARIOUS MASS FLOW RATES AND CRANK ANGLES

The Heat Transfer data obtained from the modeled cylinder is shown in fig.3. From the graph the heat transfer co-efficient is in the order of 1.08*102 W/m2K and the maximum heat transfer of 700 watts at a mass flow rate of 1.5 kg/sec.
The total heat transfer rate at various mass flow rates and the total heat release rate at various crank angles were obtained. These heat values [Fig 4.1 & 4.2] were found to be comparable with the extrapolated experimental results of natural-gas based dual fuel engine [1]. The results of natural gas + air combustion are also in agreement with the energy content of bio gas [Fig 4.3 & 4.4]. Also the maximum cycle temperature is observed to be 2160 K.

2. TORQUE VS EFFICIENCY:
After running the calculations in the post processing- >Function Calculator, the torque values are obtained and observed results are plotted in a graphical format as shown below in Fig 5. Here the efficiency of the engine is calculated theoretically. Based on maximum torque of about 29 to 30 N-m.

3. IN CYLINDER PRESSURE AND MASS FRACTION
The maximum pressure observed to be about 10.5 bars approximately as shown in the Fig. 6. The average dynamic pressure is observed to be about 50 bars as shown in the Fig. 7. A graph is drawn between mass flow rate and dynamic pressure which is shown in the Fig. 8. The contours of mass fraction are shown as in Figure 9.

4 TEMPERATURE PROFILES
The temperatures contours are derived from the premixed combustion inside the cylinder as shown in Figure 10. Also the maximum cycle temperature is observed to be 2160 K.
5 COMPARISON OF BRAKE POWER OF CFD SIMULATION WITH EXPERIMENTAL SETUP

6. THE MASS FRACTION OF POLLUTANTS
HCN, NH3, NOx Pollutants are observed to be as per the below figure 11, 12 & 13.

VII. CONCLUSIONS:
1. The model was created using Solid works, and the Heat transfer rate and combustion phenomenon was analyzed using Ansys 16 ICE. The results shows values of Heat Release, Pressure and Temperature are comparable with the Experimental Setup of dual fuel engine working on Natural Gas[1] for the same given equivalence ratio and output power. Hence the developed model is suitable for predicting the Heat transfer analysis and combustion characteristics of a Dual Fuel IC engine.

2. In the Dual Fuel ICEngine with combustion of Natural gas and air mixture, for the given compression ratio of 16:1, the heat transfer rate is observed to be about 700 Joules at about 460 degrees of crank angle. At about 350 degrees of crank angle, the heat release rate is about 580 Joules [1], which is much higher than the heat release rate of 370 Joules as per the experimental setup of Bio-gas air mixture used for comparison[2]. The same can be verified from the fact that the Calorific Value of natural gas is about 48 MJ/Kg, which is much higher when compared to that of Biogas which is about 20.2 MJ/kg [7] as shown in figures 4.3. Similarly the energy content of natural gas is about 11 KWh for 1 Nm3 when compared with 9.67 KWh for 1Nm3 of biogas as shown in Fig 4.4.
The results data from CFD simulation is nearly close with both the experimental and statistical information. Hence CFD simulation is a good alternative for designing and developing Dual Fuel IC engine when compared to prototyping and experimental setup which is expensive and time consuming process.

3. From the results of simulation it can be observed that the average dynamic pressure is about 50 bars at 1800 RPM in this study of natural gas-diesel duel fuel engine. The combustion pressure of LPG-Diesel dual fuel engine is about 57 bars at 1300 RPM itself [3] which support the fact that the LPG engines are poor knock resistance. Also, LNG engines have higher thermal efficiencies compared to LPG Engines [8]. Hence the Natural gas-Diesel based dual fuel engines are much preferable for the commercial vehicles & heavy engine applications.

4. The results of CFD simulation of natural-gas-diesel engine shows that the CO2 emissions are almost negligible when compared to 0.0048 as in case of Diesel only engine. Hence it is proved that the dual fuel engines are less polluting and recommended for better environment. Also, the Natural gas engines produce less emissions particularly NOx and other particulates. Hence the Natural-gas-Diesel can be considered as a choice for dual fuel engine compared to Diesel only engines.

5. The results of CFD simulation using Ansys 16 ICEngine package are comparable with Ansys 15 Fluent package. Hence the Ansys ICE can be used as an alternative to the Ansys Fluent to leverage the benefits already explained at the beginning of this paper.

6. The simulation results of this study with natural-gas diesel engine such as average pressure and mean temperatures are observed to be much higher when compared with a naturally aspirated gasoline direct engine[9]. Hence sufficient care is to be taken in the aspects of engine vibrations, cooling system for the engine, lubrication of engine components etc for natural-gas diesel dual fuel Engine.

7. It is possible to export the results like the heat transfer co-efficient, temperatures etc to Ansys Finite Element Solver to find the thermally induced stresses to come up with better materials to manufacture robust engines to with stand high pressures and temperatures.

REFERENCES:
[2] Shaik Magbul Hussain1*, Dr.B. Sudheer Prem Kumar2, Dr.K. Vijaya Kumar Reddy3 et. al, CFD analysis of combustion and emissions to study the effect of compression ratio and biogas substitution in a diesel engine with experimental verification, IJEST, ISSN: 0975-5462 Vol. 4 No.02 February 2012, pp2, 489, fg 25&26.
[10] John J. Kargul, Director of Technology Transfer, National Center for Advanced Technology Efficient, Use of Natural Gas Based Fuels in Heavy-Duty Engines