International Journal of Engineering Research and Applications (IJERA) ISSN: 2248-9622
International Conference On Emerging Trends in Mechanical and Electrical Engineering
(ICETMEE- 13th-14th March 2014)

RESEARCH ARTICLE OPEN ACCESS

Flow Simulation of an I.C. Engine in FLUENT, ANSYS 14.0

DivyanshuPurohit[1], Pragya Mishra[1], Vishwanath Banskar[1]
[1]B. Tech Scholar RJIT, Tekanpur

ABSTRACT:
The design and manufacture of Internal Combustion (IC) Engines is under significant pressure for improvement. The next generation of engines needs to be compact, light, powerful, and flexible, yet produce less pollution and use less fuel. Innovative engine designs will be needed to meet these competing requirements. The ability to accurately and rapidly analyze the performance of multiple engine designs is critical.
The performance of an IC Engine depends upon complex interactions between mechanical, fluid, chemical, and electronic systems. However, the central challenge in design is the complex fluid dynamics of turbulent reacting flows with moving parts through the intake/exhaust manifolds, valves, cylinder, and piston. The time scales of the intake air flow, fuel injection, liquid vaporization, turbulent mixing, species transport, chemistry, and pollutant formation all overlap, and need to be considered simultaneously. Engine analysis using CFD software has always been hampered due to the inherent complexity in

- Specifying the motion of the parts.
- Decomposition of the geometry into a topology that can successfully duplicate that motion.
- Creating a computational mesh in both the moving and non-moving portions of the domain.
- Solving the unsteady equations for flow, turbulence, energy, and chemistry.
- Postprocessing of results and extracting useful information from the very large data sets.

The ANSYS Workbench ICE system provides such an integrated environment with the capabilities integrated to set up most IC Engine designs.
Keywords: Fluid dynamics, intake/exhaust manifolds, CFD, ICE system.

I. Introduction

Fluid Dynamics during the Four Cycles
IC engine design involves several critical decisions which impact and interact with the fluid dynamics. The primary design decisions are

- The specifications for engine type.
- Peak power at a specified speed or RPM.
- The number of cylinders.
- Fuel and emissions characteristics.
- The total volume of the engine.
- Overall "packaging" of the engine including all the sub-systems.

Mechanical and electronics systems may also be specified at this stage, such as using different cam configurations. There may be additional specifications, such as the engine power at idle speed or low RPM. These design decisions impact the computation of the amount of air and fuel needed by the engine and lead to a cascade of design decisions to maximize the overall efficiency of the engine. This efficiency is given by the following equation for engine brake power:

$$P_b = \eta_i * \eta_c * \eta_m * \eta_v * \rho_i * V_d * N / (X * F * Q)$$

Where,
ηi is the indicated (brake) efficiency
ηc is the combustion efficiency
ηm is the mechanical efficiency
ηv is the volumetric efficiency of the engine
ρi is the density of the air at the intake
Vd is the engine displacement volume
N is the rotational speed X is the number of revolutions per power stroke
F is the fuel air ratio
Q is the calorific value of the fuel per unit mass

The primary goal of engine design is to maximize each efficiency factor, in order to extract the most power from the least amount of fuel. In terms of fluid dynamics, the volumetric and combustion efficiency are dependent on the fluid dynamics in the engine manifolds and cylinders.

In increasing order of complexity, the CFD analyses performed can be classified into: -

Port Flow Analysis: Quantification of flow rate, swirl and tumble, with static engine geometry at different locations during the engine cycle.

Cold Flow Analysis: Engine cycle with moving geometry, air flow, and no fuel injection or reactions.

In-Cylinder Combustion Simulation: Power and exhaust strokes with fuel injection, ignition, reactions, and pollutant prediction on moving geometry.

Full Cycle Simulation: Simulation of the entire engine cycle with air flow, fuel injection, combustion, and reactions.

II. Cold Flow Analysis

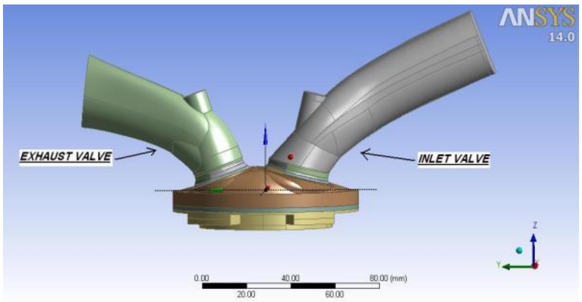
Cold flow analysis involves modeling the airflow and possibly the fuel injection in the transient engine cycle without reactions. The goal is to capture the mixture formation process by accurately accounting for the interaction of moving geometry with the fluid dynamics of the induction process. The changing characteristics of the air flow jet that tumbles into the cylinder with swirl via intake valves and the exhaust jet through the exhaust valves as they open and close can be determined, along with the turbulence production from swirl and tumble due to compression and squish.

Setting up the CFD model for cold flow analysis involves additional work in specifying the necessary information to compute the motion of the valves and piston in addition to the boundary conditions, turbulence models and other parameters. This includes specifying valve and piston geometry, along with the lift curves and engine geometric characteristics in order to calculate their position as a function of crank angle. Since the volume in the cylinder is changing shape throughout the engine cycle, the mesh has to change accordingly. Different approaches to automatically modify the mesh during motion also need to be specified. The CFD calculation is an inherently transient computational problem when involved with moving deforming or dynamic mesh. All the geometric motion is a function of a single parameter, the position of the crankshaft in its rotation, or crank angle

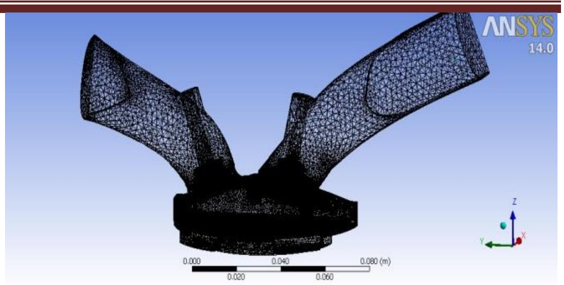
1. Getting Started With ICE

For IC engine analysis in ANSYS there is a separate workbench inbuilt module of ICE which helps in generating complex geometry, mesh, solution of an engine easily.

1. I.C. Engine geometry for simulation



2. Meshing in ANSYS application in **ANSYS ICEM CFD.**



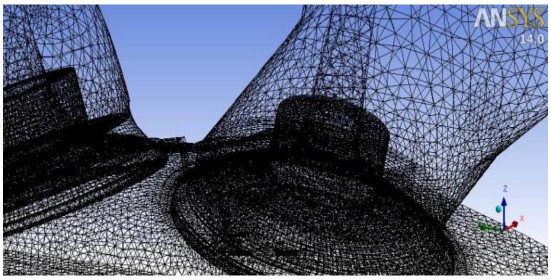
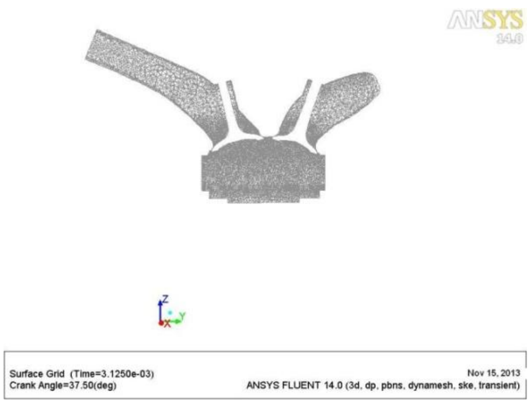
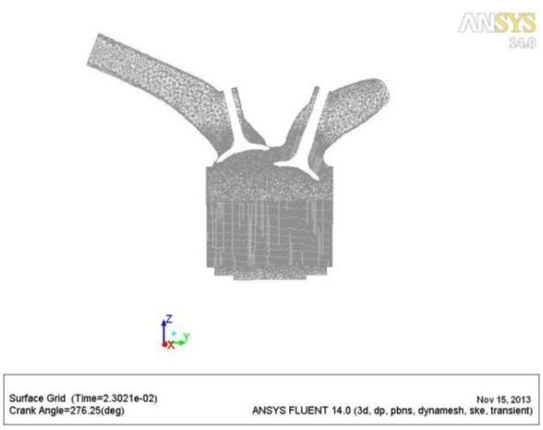


Fig: - finer mesh near valve regions

3. **Dynamic meshing of the I.C. engine fluid computational** domain was done and plotting on one plane was done at some crank angles as shown in figures below.





4. Launching **ANSYS FLUENT** for applying materials, boundary conditions etc to calculate the solution.

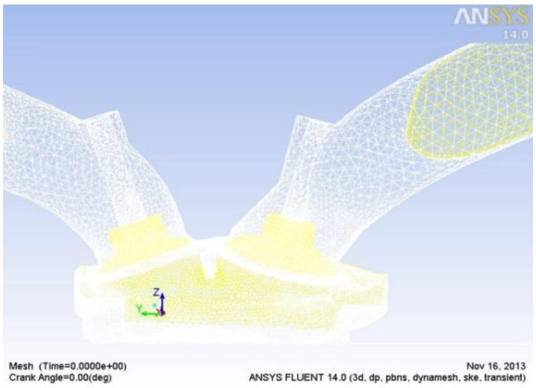
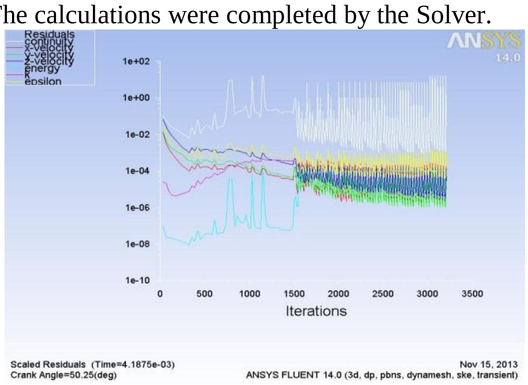


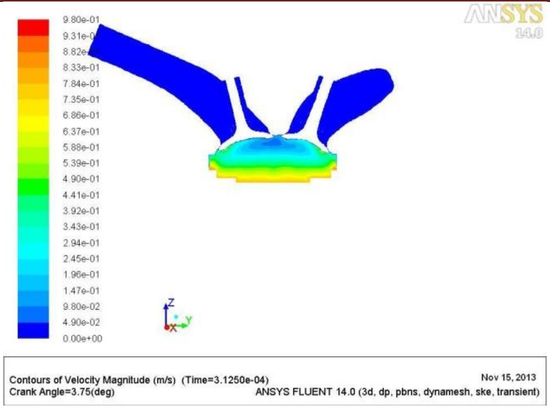
Fig: - Mesh data imported automatically to ANSYS FLUENT

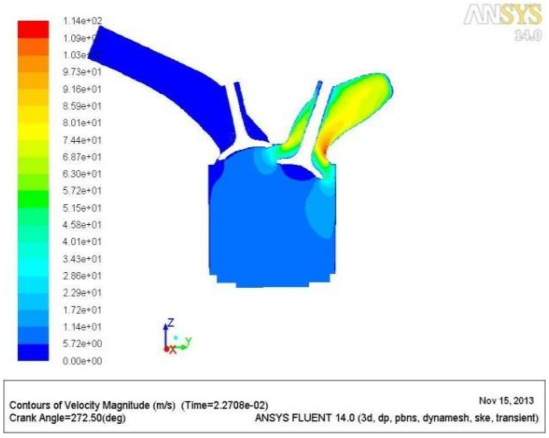
5. The smallest time step consist of 0.25^0 crank angle, therefore 3000 time steps were set for calculation and simulation which will be approximately equal to 720^0 of crank angle (i.e. 1 complete cycle of four stroke engine).

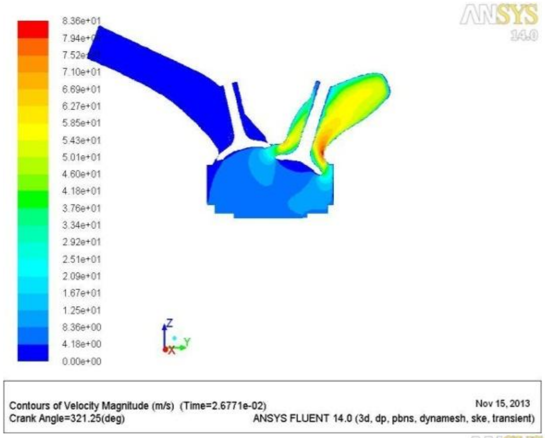
The calculations were completed by the Solver.

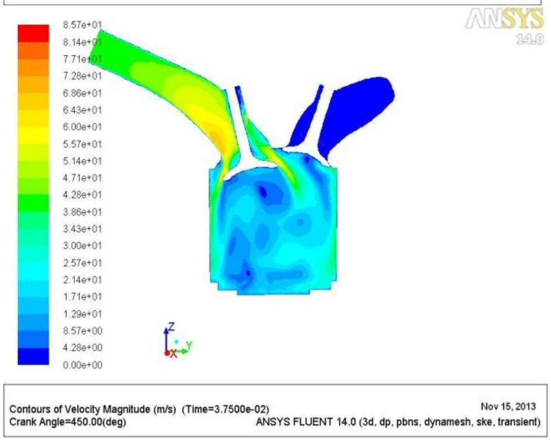


6. **Results of the analysis** were plotted as the contours of velocity magnitude of fluid in the 4-cycles of 4 Stroke IC engine at various time steps and crank angle to check the flow as shown in figures below.

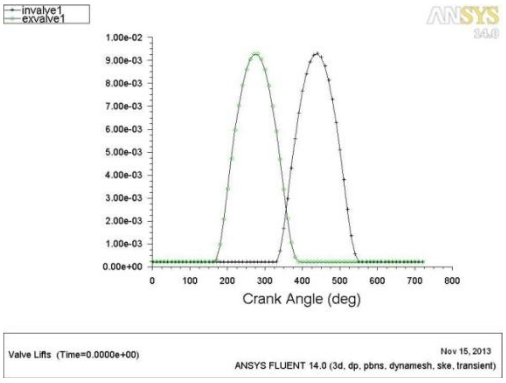


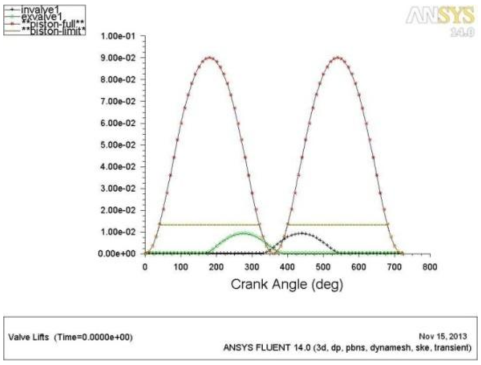






7. **Graph or profiles** were plotted according to the motion and position of piston and valves as shown below.





References

[1] Ganesan .V, Internal combustion engine, Tata McGraw Hill Edu. Pvt. Ltd.
[2] Mathur M.L. and Sharma R.P., DhanpatRai publication.
[3] ANSYS 14.0 Help Library