ABSTRACT

The shape of flow enters the cylinder in internal combustion engine chamber plays very important role. As the change of inlet valve lift depends on crank angle variation on intake process, the shape of flow that enters the combustion chamber changes too. This will influence on the mixing process of flow occurs in cylinder and the combustion process either. With this simulation, wants to be known the shape of flow that enters the cylinder with changing in valve lift depends on crank angle variation.

Using both CFD Fluent 6.0 and visualizing the combustion chamber and valve with transparent material, the visualization of inlet valve lift depends on crank angle variation when intake process occur can be used to predict next improvements that will be proposed.

It is expected with those methods, the visualization of shape flow in cylinder as the change of valve lift depends on crank angle variation can be seen and we can use it in the real condition.

On the modeling mode 1, it is seen that there is a development of region with smaller density on left hand side on the bottom of cylinder starts from crank angle 50° until 90° after top dead center (ATDC).

On the simulation with Fluent 6.0, it is seen the movement of vortex position in long section model. In cross section model, it is seen the movement of two vortexes position whose have opposite rotation direction.

On the modeling mode 2, in the velocity vector in long section model, it is seen the movement of vortex from right side of piston surface into bottom surface of valve. In the velocity vector in cross section model, we can see the movement of two vortexes from first position.

Key words: shape of flow, cylinder, crank angle variation, CFD, visualization.