

Development of Intake and Exhaust Stroke Flow Simulation in an Engine Cylinder Using CFD Model

Rosli A. Bakar, Semin & Abdul R. Ismail
Automotive Focus Group, Faculty of Mechanical Engineering
Universiti Malaysia Pahang
Locked Bag 12, 25000 Kuantan, Pahang, Malaysia

Abstract - The intake and exhaust stroke flow simulation in an engine cylinder illustrates some of the transient, dynamic-mesh features, event handling, moving grids and cell attachment/detachment by setting up and solving a simplified port-valve and piston problem using STAR-CD computational fluid dynamics. The problem is two-dimensional and represents a simplified four-stroke engine. The simulation commences at bottom dead centre (BDC) and continues for a time interval corresponding to 720° of crank revolution. Inert scalars with the physical properties of air are used to track the intake and exhaust stream through the exhaust induction strokes in engine cylinder. The flow in engine cylinder is turbulent and driven entirely by the motion of the piston and valves. The port boundary pressures are held constant at ambient condition throughout the simulation. Mesh design in problems with a moving mesh and changing cell connectivity is dominated by the need to keep the dynamic parts of the grid simple so that they can be easily changed during the transient run. The formulation of mesh motion in such problems is divided into two conceptual steps. The first deals with connectivity changed which are defined by PROSTAR events. The second step is to specify the grid vertex positions as function of time by supplying a set of PROSTAR grid-manipulation commands to be executed at each time step. In this model the exhaust valve opens within the first 147° and closes linearly by 282°. The intake valve begins to open at 395° and closes by 530°. The piston motion is prescribed by setting the standard kinematics parameter for the connecting rod length, the crank radius and the top dead centre position (TDC).

Keywords: Engine cylinder, intake and exhaust flow simulation, piston motion.

INTRODUCTION

This research was focused to develop the intake and exhaust stroke flow simulation in an engine cylinder using STAR-CD computational fluid dynamics (CFD) model. In this simulation models the engine operation during exhaust and intake strokes. It is made up of three parts. The first is prostar pre-processing, including the problem setup, grid-changing commands and event verification. The second is star analysis, including the PMOV pre-

processing and flow computation. The third is prostar post-processing, including visualization of the flow field at pre-defined times during the analysis and animated display of the flow field transients. The simulation start from the intake port, intake valve, engine cylinder, piston motion, exhaust valve and exhaust port.

STAR-CD is 3D computational fluid dynamics (CFD) model for visualized of Internal Combustion Engines flow in induction, compression, fuel injection, combustion, power and exhaust. The code in STAR-CD includes improvements to the spray, ignition, combustion and turbulent model. The CFD solver provides one of the most effective numerical methodologies available in an industrial CFD code with the high level of accuracy needed for complex unstructured meshes. This is delivered with the speed, efficiency and robustness demanded by engineering design and development cycles. STAR-CD uses state-of-the-art, proprietary numerical schemes to achieve the highest levels of accuracy in both steady and transient simulations, making this solver one of the least sensitive to mesh type and quality, including distorted tetrahedral meshes. Remarkably, this has been achieved without sacrificing efficiency or robustness. So, whatever the choice of mesh or engineering application, the STAR solver will provide the best solution in the shortest time. A particular feature of STAR-CD is its fast CPU performance for transient flows.

STAR-CD was used by any researchers to develop the IC engines computational modeling was fueled by gasoline, diesel or CNG. Any engine researchers used the computational fluid dynamics (CFD) modeling and experiment to visualised and validated their result. Sera [17] has advanced intake system based on a new mixer design was developed to increase intake flow pressure and at the same time generate intake turbulent flow for Compressed Natural Gas (CNG) fuelled operation use experiment and CFD Modeling by STAR-CD, the results show that the new mixer has proven to improve the intake flow pressure and produced higher turbulent ratio in gasoline engine. Chiodi [14] investigation on mixture formation and combustion process in a CNG engine by using a fast response 3D CFD simulation by STAR-CD, the results of their work are mainly intended to set a focus both on the injection system modeling and on the problem related to the influence of the initial conditions on the results. Bianchi [4, 10] was used STAR-CD to studied the

effects of numerical methodology in defining the initial conditions and simulating the compression stroke in the realistic engine and port geometry of D.I. diesel engine and 3D simulations of IC engines intake flows. Riegler [15] was direct coupled 1D/3D-CFD-computation (GT-Power/Star-CD) of the flow in the switch-over intake system of an 8- cylinder SI engine with external exhaust gas recirculation. STARCD was used for engine 3D modeling of the experiments [2-16].

MODELLING METHODOLOGY

The development of intake and exhaust stroke flow simulation in an engine cylinder was presented in this paper. The specification of the selected engine cylinder model was presented in Table 1.

Table 1. Specification of the engine model [4]

Engine Parameters	Value
Bore (mm)	86.0
Stroke (mm)	70.0
Displacement (cc)	407.0
Number of cylinder	1
Connecting rod length (mm)	118.1
Piston pin offset (mm)	1.00
Intake valve open ($^{\circ}$ CA)	395
Intake valve close ($^{\circ}$ CA)	530
Exhaust valve open ($^{\circ}$ CA)	147
Exhaust valve close ($^{\circ}$ CA)	282
Max. intake valve open (mm)	7.095
Max. exhaust valve open (mm)	7.095
Valve lift periodicity (deg)	360

The mass and momentum conservation equations solved by STAR-CD for general incompressible and compressible fluid flows and a moving coordinate frame the Navier Stokes equations are, in Cartesian tensor notation [1].

$$\frac{1}{\sqrt{g}} \frac{\partial}{\partial t} (\sqrt{g} \rho) + \frac{\partial}{\partial x_j} (\rho \bar{u}_j) = s_m \quad (1)$$

$$\frac{1}{\sqrt{g}} \frac{\partial}{\partial t} (\sqrt{g} \rho u_i) + \frac{\partial}{\partial x_j} (\rho \bar{u}_j u_i - \tau_{ij}) = -\frac{\partial p}{\partial x_i} + s_i \quad (2)$$

where t is time, x_i is cartesian coordinate ($i = 1, 2, 3$), u_i is absolute fluid velocity component in direction, \bar{u}_j is $u_j - u_{cj}$ relative velocity between fluid and local (moving) coordinate frame that moves with velocity u_{cj} , ρ is density, τ_{ij} is stress tensor components, s_m is mass source, s_i is momentum source components, \sqrt{g} is determinant of metric tensor.

$$p = p_s - \rho_0 \cdot g_m \cdot x_m \quad (3)$$

where p is piezometric pressure, p_s where is static pressure, ρ_0 is reference density, the g_m are gravitational field components and the x_m are coordinates from a datum, ρ_0 where is defined.

This specialization of the above equations to a particular class of flow involves for application of ensemble or time averaging if the flow is turbulent,

specification of a constitutive relation connecting the components of the stress tensor to the velocity gradients and specification of the source, which represents the sum of the body and other external forces, if present.

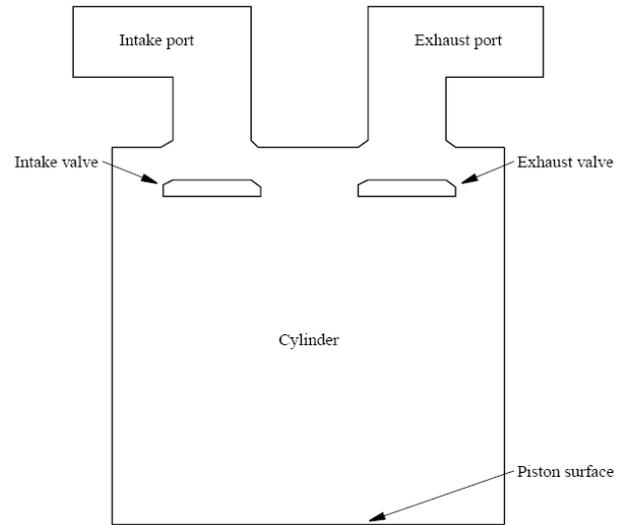


Fig. 1. Engine cylinder with intake and exhaust valves [1]

The fluid in the cylinder is assumed to be air, behaving as an ideal gas. Constant, zero-pressure boundary conditions are assigned at the intake and exhaust port ends. The fluid originally in the intake port is assumed to be a different species called intake with the properties of air. Similarly, the exhaust port is filled with another fluid species called exhaust. The boundary concentrations for the two species are also set appropriately. By tracking the concentration of these species, the fresh charge and exhaust gas distributions can be deduced and global parameters such as volumetric efficiency, etc. can be computed. It should be noted that the model used in this tutorial is too coarse for any realistic computation of these values. The valve lifts follow the curves shown in Figure 2. In this model the exhaust valve opens within the first 147° and closes linearly by 282° . The intake valve begins to open at 395° and closes by 530° . Although this is a very simplistic profile, the technique used here to define valve lift is quite general and can be used for any lift profile. The piston motion is prescribed by setting the standard kinematic parameters for the connecting rod length, the crank radius and the top dead centre position.

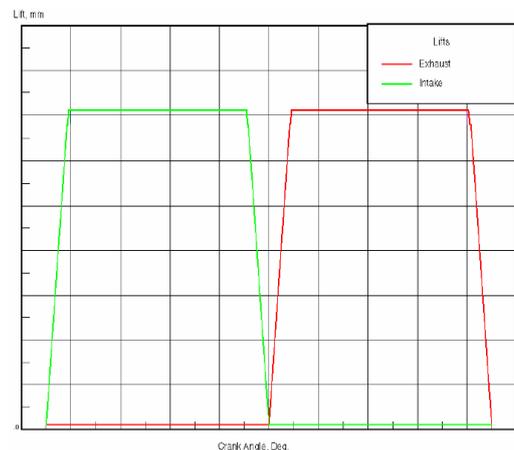


Fig. 2. Valve lift profile

RESULT AND DISCUSSION

Problem Setup Mesh Model

Mesh design in problems with a moving mesh and changing cell connectivity is dominated by the need to keep the dynamic parts of the grid simple so that they can be easily changed during the transient run. The formulation of mesh motion in such problems is divided into two conceptual steps. The first deals with connectivity changes cell removal, addition, reconnection, etc., which are defined by PROSTAR events. The second step is to specify the grid vertex positions as a function of time by supplying a set of PROSTAR grid-manipulation commands to be executed at each time step.

Other considerations include three strategies. The first is the initial mesh must contain all the cells that will be used in the analysis. Thus, it will be seen that the initial grid construction contains overlapping cells in the vicinity of the valves. These cells are rearranged according to the valve position at time = 0.0, i.e. at the start of the calculations. The second is when cells are added or activated, they are still deemed to be connected to the neighbours they had at the time of their removal or deactivation. This imposes sequencing constraints on the activation and deactivation processes. The third is if any part of the solution domain becomes separated from the rest of the flow field during a transient run (e.g. the intake port after the intake valve has closed) the cell material type in that domain must be changed. All material types used must have been defined in the initial setup.

Mesh Definition: The versatility and accuracy of an industrial thermofluids analysis code are very much tied to the flexibility of the computational mesh structure it employs; for this determines both the level of geometrical complexity it can handle and the degree of control it offers over resolution of flow features. The function of the mesh is to fit the boundary surface of the computational domain and subdivide its volume into subdomains or 'cells', used in the numerical solution of the differential conservation equations of the mathematical model. STAR-CD employs a highly flexible mesh system that is probably unique in its combination of features.

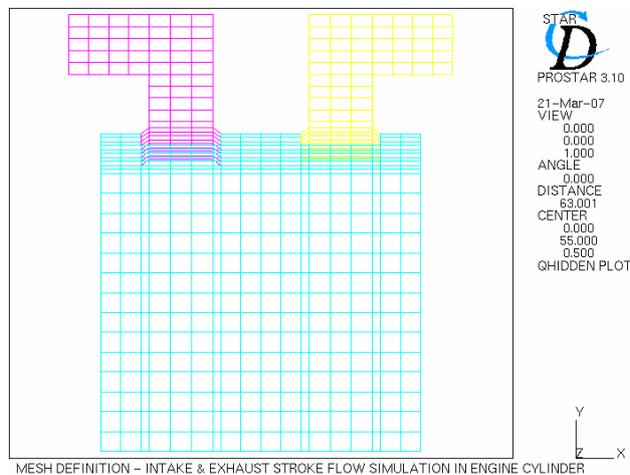


Fig. 3. Mesh definition of model

It should be noted that cell types IVB and IVT (and also EVB and EVT) overlap in space. As the run progresses,

these cells are selectively activated, deactivated and reordered such that no overlap occurs at any time during the solution. It should also be noted that the sides of these blocks of cells are not connected to the cylinder (CYLI) cells. These connections will be made dynamically during the run by using moving mesh commands.

Boundary Condition: Define the attributes of the various cell types, assuming they all have the properties of air. For all materials, the first is specify that the density is to change according to the Ideal Gas Law, the second is switch on turbulence modelling ($k-\Sigma$ model) and the third is switch on the temperature calculation. A scalar variable needs to be set up to track the distribution of the inlet charge in the cylinder. The properties of the inlet scalar are those of air. Another scalar is also needed for the exhaust charge, to determine if any back flow from the exhaust port into the cylinder occurs when the valve opens.

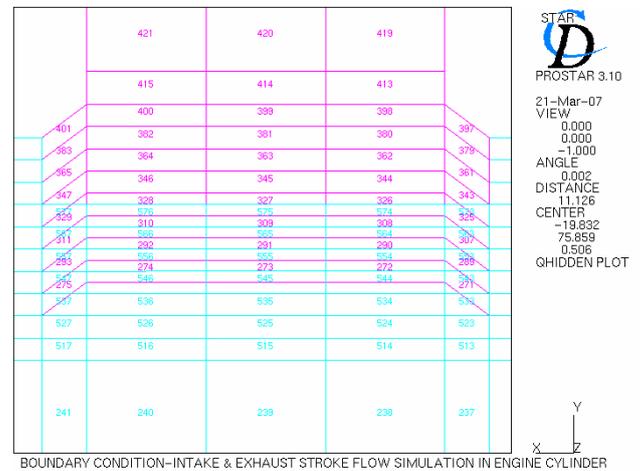


Fig. 4. Boundary condition of model

Boundary conditions for the various boundary regions may now be defined. Regions 1, 2 and 3 are assigned the ATTACH boundary type. This is used for boundary surfaces that will be attached together dynamically during the course of the analysis, i.e. the vertical edges of the IVT, IVB, EVT and EVB domains and the corresponding boundary cell faces in the CYLI domain.

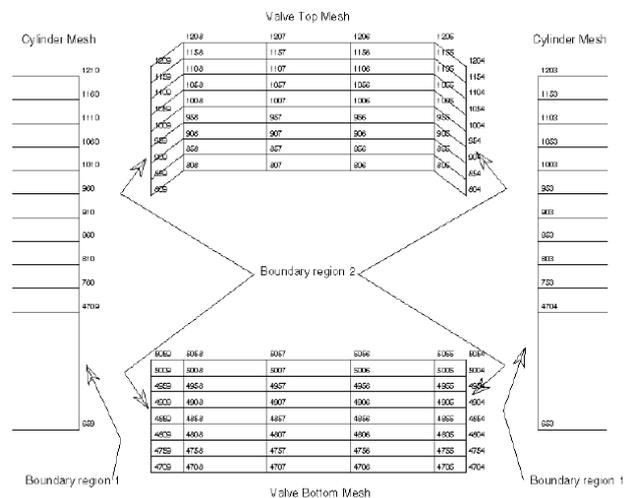


Fig. 5. Boundary region of model

Figure 5. shows an exploded view of the grid near the intake valve, with the boundary regions marked clearly. The parameters needed for the region definition are the coordinate system number and the alternate region number to be used if the boundary faces in question are not attached to anything.

Mesh Visualization: This command sets the r.p.m. value to 1000, the crank radius to 34.95, the connecting rod length to 105, the initial piston position to 0.0 and the top dead centre position to 69.9. Note that all distances are specified in PROSTAR units (mm in this case). The advantage of specifying these parameters is twofold. The first is the piston event times can be specified in terms of its position rather than the equivalent time (e.g. cell layer 1 should be removed when the piston reaches a position of 2.5 mm). The second is these parameters are used by STAR to set the built-in parameter YPST which is used in the EGRID commands to simplify the description of mesh motion.

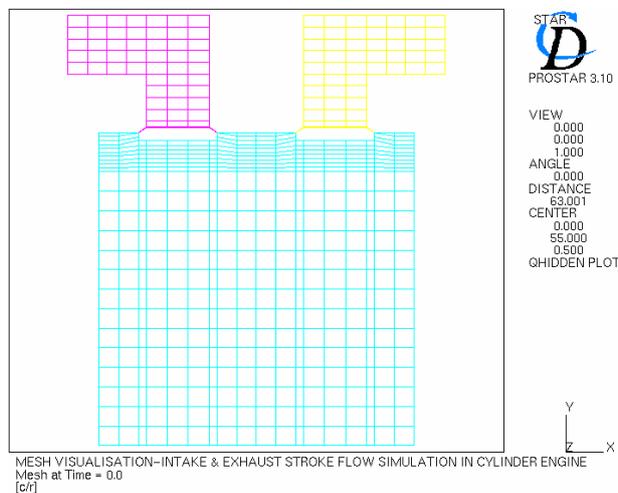


Fig. 6. Mesh visualization at time 0.0

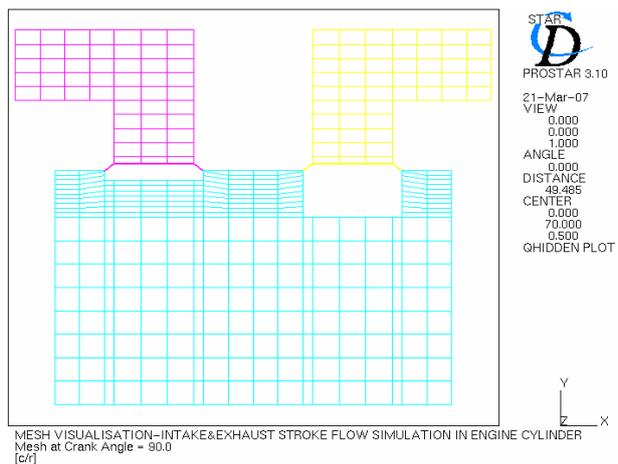


Fig. 7. Mesh visualization at crank angle 90.0

In the piston events, there are 13 cell layers in the cylinder region. The piston events will be defined so that one layer is deactivated at every event between no. 1 and no. 13, corresponding to the piston movement between BDC and TDC. Conversely, the layers are re-activated between event nos. 16 to 28, corresponding to the piston movement between TDC and BDC. Each event occurs when the piston crosses the mid-point of a layer. The

most efficient way of defining the 13 deactivation events is to use the PROSTAR loop facility as follows.

In the intake valve events, were defining of the intake valve events, the natural choice for specifying the occurrence of these events would be the crank angle using command EVPARM. The reason EVPARM is re-specified is because it allows you to use a convenient initial angle which does not have to match the initial position of the piston. EVPARM, DEGREES, 1000, 0.0. The mesh at the intake valve at time $t = 0$ must first be rearranged, i.e. the extra, overlapping cells in that region must be deactivated. Seven cell layers above the valve must be deactivated before the solution starts (the valve is initially closed). This must be done in the correct sequence. For this reason, negative event times are used to establish the sequence of deactivation events that occur before the flow field computation begins. At time $t = 0$, the side faces of the surviving cells below the valve must be connected together. This is done with an attachment event at time $t = 0$. Since event times must be unique, we have to reuse event no. 40 which is already set to occur at this time. The cell faces to be attached are found automatically using the EAMATCH command. For this, we need to group the faces to be matched into two regions. The attachment boundaries belonging to the cylinder have already been assigned to region no. 1; the attachment boundaries belonging to the valve that need to be matched to the cylinder boundaries must be collected and temporarily assigned to region no. 3. The boundary sets to be matched are nominally unequal because of the presence of deactivated cell layers. Nevertheless, the operation is physically valid and also avoids the trouble of picking out the right faces. EAMATCH will generate warning messages which should be ignored. No further intake valve events occur until the crank angle is 180° . At this time, the layer above the valve should be connected to the cylinder. This also means that the cells in the inlet port must be changed to the material type of the cylinder, as they are now physically connected to it. As the valve opens between 180° and 200° , seven cell layers are deactivated below the valve and corresponding cell layers activated above the valve. The side connections between cell layers are also altered to match the new configuration. The final event at 360° will completely close the valve by detaching the last connection (in the layer above the valve) and changing the material type of the cells trapped above the valve.

In the exhaust valve events, a similar sequence of events occurs for the exhaust valve. Note, however, that the opening and closing times are different. Reuse event nos. 30 to 36 for deactivating overlapping cell layers prior to time $t = 0$.

Grid and Events Code Generation

In general, such commands would be included in a file created with a suitable editor of your choice. However, since grid movement commands usually represent over half the effort needed to set up a problem, it is convenient to maintain several different EGRID files so as to allow easy investigation of alternative strategies. The easiest way of deploying alternative strategies is to use the technique of file re-direction, in which all PROSTAR pre-processing runs refer to the same file. This file in turn simply redefines the input stream as

coming from the file that actually contains the required variant of the grid movement commands. The intake and exhaust valve lifts at the corresponding time are determined by interpolation of the lift-spline discussed in the previous section. The piston position is directly available via the STAR.

Mesh Development: The mesh structure within the cylinder at any given time is determined by collecting the bottom layer of vertices and placing them at the current piston position. This has the effect of reducing the thickness of only the cell layer adjacent to the piston. The rest of the cylinder mesh remains unaltered. When this layer is eventually removed (at the point where the piston reaches its original mid-thickness location) the bottom vertices of the next layer up are moved to meet the piston surface and the process continues until TDC is reached.

The mesh movement in the intake region is controlled by a similar process. Thus, the vertices at the top and bottom valve surface are set to follow the valve position and the adjacent cell layers above and below the valve are activated or deactivated accordingly. This is done in two stages, for the mesh above and below the valve. A detailed view of the inlet valve region is shown in Figure 5. with the vertex numbering switched on. A similar set of commands are used for the exhaust valve grid.

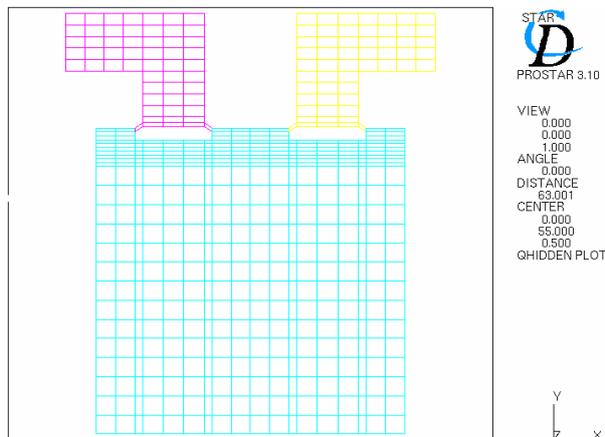


Fig. 8. Mesh animation at crank angle degree 1.0

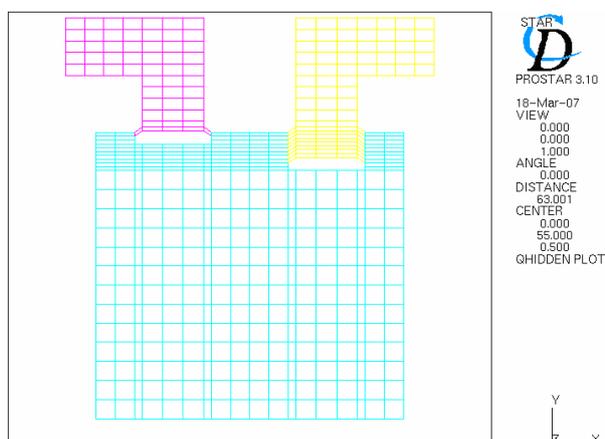


Fig. 9. Mesh animation at crank angle degree 40.0

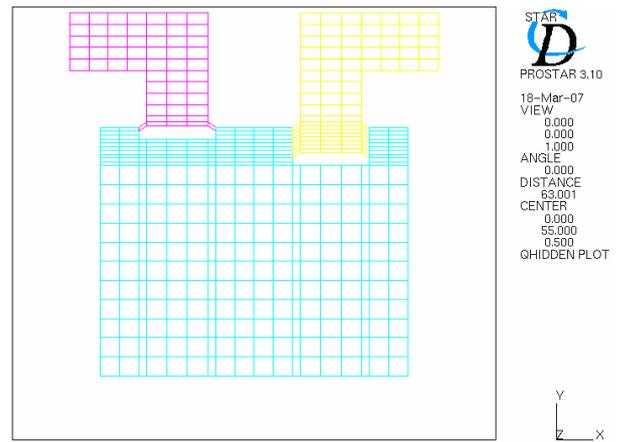


Fig. 10. Mesh animation at crank angle degree 60.0

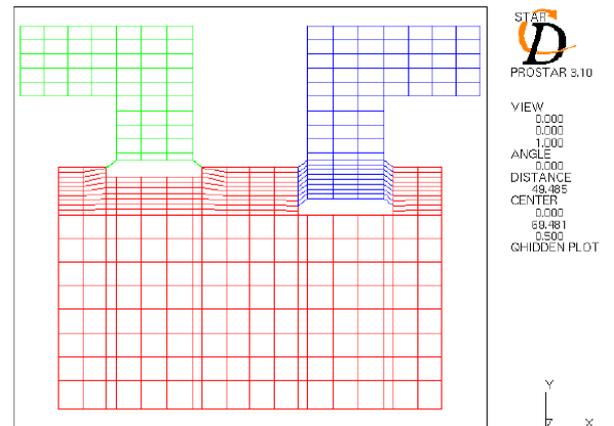


Fig. 11. Mesh animation at crank angle degree 90.0

Post Processing

Post processing of transient moving-mesh solutions presents special challenges. First, there is usually a large amount of data to visualize. Second, the mesh changes with time. The solution data are stored in file at intervals of 5 time steps, as specified in the problem setup section. In this problem, each time step corresponds to 1° of crank revolution, allowing the iteration number to be interpreted as the crank angle. The vertex coordinates per time step are also stored in this file by default. However, the cell status at any stage (activated or deactivated) must be reconstructed by PROSTAR using the events file.

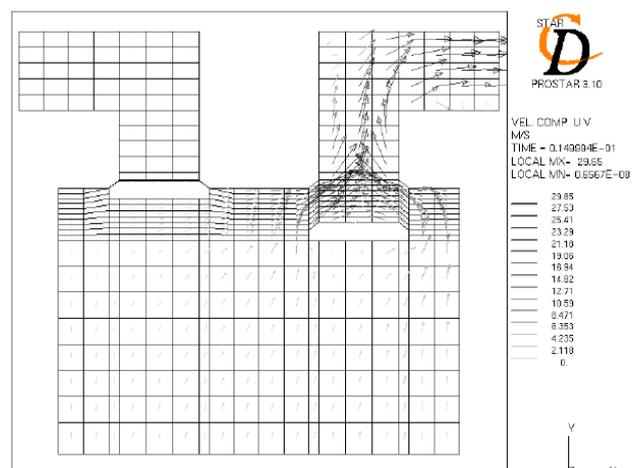


Fig. 12. Flow animation at exhaust valve

Animation Displays: A more powerful means of visualizing transient results is provided by PROSTAR's animation commands. A series of static frames can be saved in a binary neutral plot file and then played back onto the screen to give the appearance of animation. For successful use of this feature, the viewing distances, colour scales and vector lengths must remain fixed between frames.

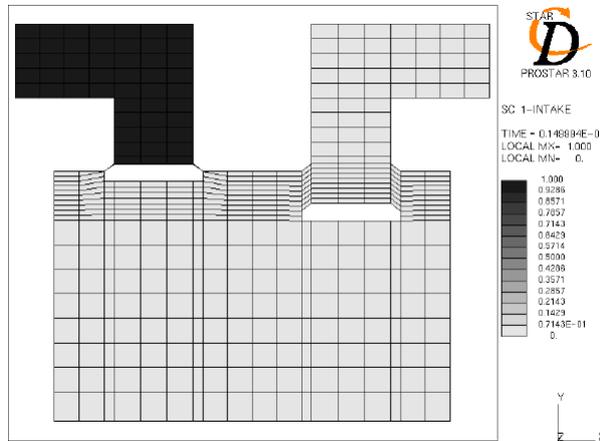


Fig. 13. Animation display at exhaust valve

CONCLUSION

The result of the intake and exhaust stroke flow simulation in an engine cylinder shows that the simulation commences at bottom dead centre (BDC) and continues for a time interval corresponding to 720° of crank revolution. Inert scalars with the physical properties of air are used to track the intake and exhaust stream through the exhaust induction strokes in engine cylinder. The port boundary pressures are held constant at ambient condition throughout the simulation. Mesh design in problems with a moving mesh and changing cell connectivity is dominated by the need to keep the dynamic parts of the grid simple so that they can be easily changed during the transient run. The formulation of mesh motion in such problems is divided into two conceptual steps. The first deals with connectivity changed which are defined by PROSTAR events. The second step is to specify the grid vertex positions as function of time by supplying a set of PROSTAR grid-manipulation commands to be executed at each time step.

REFERENCES

- [1]. Adapco, Group. CD., Methodology STAR-CD VERSION 3.15A, Computational Dynamics Limited, 2002.
- [2]. Mastorakos, E., Wright, Y.M., Boulouchos, K., "Liquid Spray Combustion in Diesel Engine, Simulating n-heptane sprays with conditional moment closure", Automotive Application of Star CD, University of Cambridge, Switzerland, 2006.
- [3]. Chen, A., Veshagh, A., Wallace, S., "Intake Flow Predictions Of A Transparent DI Diesel Engine", SAE Paper, 981020, 1998.
- [4]. Bianchi, G. M., Richards, K., Reitz, R. D., "Effects Of Initial Conditions In Multidimensional Combustion Simulations Of HSDI Diesel Engines", SAE Paper, 1999-01-1180, 1999.

- [5]. Sinclair, Rory., Strauss, Tim., Schindler, Peter., "Code Coupling, A New Approach To Enhance CFD Analysis Of Engines", SAE Paper, 2000-01-0660, 2000.
- [6]. Benajes, Jesús., Margot, Xandra., Pastor, JoséV., Gil, Antonio., "Three-Dimensional Calculation Of The Flow In A DI Diesel Engine With Variable Swirl Intake Ports", SAE Paper, 2001-01-3230, 2000.
- [7]. Kaario, Ossi., Larmi, Martti., Tanner, Franz., "Relating Integral Length Scale To Turbulent Time Scale And Comparing K-Eand RNG K-Eturbulence Models In Diesel Combustion Simulation", SAE Paper, 2002-01-1117, 2002.
- [8]. Kaario, Ossi., Martti, Larmi., Tanner, Franz., "Comparing Single-Step And Multi-Step Chemistry Using The Laminar And Turbulent Characteristic Time Combustion Model In Two Diesel Engines", SAE Paper, 2002-01-1749, 2002.
- [9]. Auriemma, M., Caputo, G., Corcione, F. E., Valentino, G., Riganti, G., "Fluid-Dynamic Analysis Of The Intake System For A HDDI Diesel Engine By STAR-CD Code And LDA Technique", SAE Paper, 2003-01-0002, 2003.
- [10]. Bianchi, G. M., Fontanesi, S., "On The Applications Of Low-Reynolds Cubic K-Eturbulence Models In 3D Simulations Of ICE Intake Flows", SAE Paper, 2003-01-0003, 2003.
- [11]. Liu, Yi., Ali, Amr., Reitz, Rolf.D., "Simulation Of Effects Of Valve Pockets And Internal Residual Gas Distribution On HSDI Diesel Combustion And Emissions", SAE Paper, 2004-01-0105, 2004.
- [12]. Fontanesi, S., "Development And Validation Of A Boundary Layer Control System To Increase Intake Port Steady Permeability", SAE Paper, 2004-01-0111, 2004.
- [13]. Koopmans, Lucien., Wallesten, Johan., Ogink, Roy., Denbratt, Ingemar., "Location Of The First Autoignition Sites For Two HCCI Systems In A Direct Injection Engine", SAE Paper, 2004-01-0564, 2004.
- [14]. Chiodi, Marco., Berner, Hans-Jurgen., Bargende, Michael., "Investigation on Mixture Formation and Combustion Process in a CNG-Engine by Using a Fast Response 3D-CFD-Simulation", SAE Paper, 2004-10-3004, 2004.
- [15]. Riegler, Udo. G., Bargende, Michael: Direct coupled 1D/3D-CFD-computation (GT-Power/Star-CD) of the flow in the switch-over intake system of an 8- cylinder SI engine with external exhaust gas recirculation, SAE Paper, 2002-01-0901.
- [16]. Bakar, Rosli.A., Semin., Ismail, Abdul.R: Effect Of Engine Performance For Four-Stroke Diesel Engine Using Simulation, Proceeding The 5th International Conference On Numerical Analysis in Engineering, Padang-West Sumatera, Indonesia, May, 18-19, 2007.
- [17]. Sera, Mardani. Ali., Bakar, Rosli. Abu., Leong, Sin. Kwan., "CNG engine performance improvement strategy through advanced intake system", SAE Paper, 2003-01-1937, 2003.