Seventh International Conference on CFD in the Minerals and Process Industries
CSIRO, Melbourne, Australia
9-11 December 2009

NUMERICAL MODELING OF IN-CYLINDER FLUID FLOW IN INTERNAL COMBUSTION ENGINES USING REYNOLDS STRESS TURBULENCE MODEL

H. KHALEGHI[1] AND M.R. NABIFAR[2*]

[1] Associate Professor, Department of Mechanical Engineering, Tarbiat Modares University, Tehran, Iran

[2] Research Assistant, Department of Mechanical Engineering, Tarbiat Modares University, Tehran, Iran
*Corresponding author, E-mail address: mnabifar@yahoo.com

ABSTRACT

In recent years, various turbulence models are widely developed and used to predict turbulent flows. The conventional eddy-viscosity turbulence models (EVTM), such as $k-\varepsilon$ model presents poor prediction in engine fluid flows; therefore, researchers study and investigate other engine turbulence models. In this paper, Reynolds Stress Turbulence Model (RSM) has been employed in conjunction with the non-iterative PISO algorithm to calculate the flow phenomena in an axi-symmetric internal combustion engine chamber during the induction process and compression stroke. The abilities of the model are examined and the results are compared with $k-\varepsilon$ and Algebraic Stress Turbulence Model (ASM) and also compared with experimental data. Qualitative consideration of the results shows that the RSM predicts reasonable agreement and closer results to experimental data and real in-cylinder fluid flow of turbulence phenomena in comparison with ASM model and standard $k-\varepsilon$ model, Although the computing time is more than the others.

NOMENCLATURE

p Pressure (bar)
u Velocity $(\frac{m}{s})$
k Turbulence Kinetic energy (j)
e_s Static energy (j)
t Time (s)
ρ Density $(\frac{Kg}{m^3})$
μ_{eff} Effective dynamic viscosity $(\frac{Kg}{sm})$
ε Rate of destruction of turbulence kinetic energy
r Radius (m)
T Temperature (k)

INTRODUCTION

The performance,efficiency,fuel consumption and exhaust emissions of reciprocating engines are highly dependent on the air-fuel mixing process inside the combustion cylinder. For internal combustion engines, the complexity of the combustion process is further augmented due to the heterogeneous distribution of the liquid fuel inside the combustion chamber. Accordingly, a better understanding of the turbulent fluid flow structure inside the engine cylinder during the induction process and compression stroke is essential to improve the engine design.

In-cylinder fluid flow of internal combustion engine is 3D, unsteady and turbulent and to study details of flow fields, temperature and pressure distributions and turbulence intensity, it is necessary to solve governing equations on flow including conservation (mass, momentum and energy) equations in addition to turbulence equations. In practice, as these equations are non-linear, couple and interrelated and also due to complexity of in-cylinder boundary conditions and boundary changes in connection with out stroke, analytical solution is very difficult and even impossible to achieve, hence for solving equations, numerical methods are widely applied and developed.

Applied calculation methods in multi dimensional internal combustion engine flows are based on two different methods. The first is ICED-ALE solution procedure which for the first time was employed by Hirt [1] for calculation of in-cylinder flow in KIVA computer code [2]. The second method uses SIMPLE procedure of Patankar and Spalding [3] and was employed by Gosman et al. [4] for calculation of in-cylinder flow.

Pressure Implicit by Splitting of Operators (PISO) algorithm is one of the methods based on prediction and correction and for the first time was employed by Issa [5]. This algorithm makes considerable saving in computing time compared with the SIMPLE method and no needs to unsteady condition.

The turbulent flow inside the reciprocating engine has several characteristics which cannot be described by any conventional eddy-viscosity turbulence model (EVTM), such as the two-equation $k-\varepsilon$ model. Due to complex geometry, wall effects, and flow rotation, the turbulence is highly anisotropic. In this research, RSM model is employed for in-cylinder flow in single phase (induction and compression process) and the results are compared with $k-\varepsilon$ and ASM models and also compared with the experimental data.

MODEL DESCRIPTION

To analyze in-cylinder fluid flow and obtain velocity vectors, pressure and temperature distributions, turbulence intensity and flow fields, related equations of mass, momentum and energy conservation together with related equations of turbulence model shall be coupled with each other and then solved. In this research, finite volume method was used for solving governing conservation equations on flow and turbulence equations. In this method, initially fields are determined and/or in-cylinder space divided into separate control volumes with finite dimensions and then governing equations taken form of discrete and in last level, requested parameters are calculated with use of non-iterative PISO algorithm. In turbulence flows, practically averaging behavior shall be used and as a result by averaging, flow equations were changed to time mean equations for turbulence flow. It was supposed that, averaged variables resulted from solving governing equations are able to describe in-cylinder conditions. Also Favre Averaging method was used (density-weighted average).

Mass, Momentum and Energy Conservation Equations:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j}\left(\rho u_j\right) = o \qquad (1)$$

Momentum conservation equation is:

$$\frac{\partial(\rho U_i)}{\partial t} + \frac{\partial(\rho U_i U_j)}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \frac{\partial(\mu X_{ij})}{\partial x_j} - \frac{\partial}{\partial x_j}\left(\rho \overline{u_i u_j}\right) \qquad (2)$$

where:

$$X_{ij} = \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} - \frac{2}{3}\frac{\partial U_k}{\partial x_k}\delta_{ij} \qquad (3)$$

Energy conservation equation is coupled the others as follow:

$$\frac{\partial}{\partial t}(\rho e_s) + \frac{\partial}{\partial x_j}(\rho u_j e_s) = \frac{\partial}{\partial x_j}\left[\gamma \frac{\mu_{eff}}{\sigma_{eff}} \cdot \frac{\partial e_s}{\partial x_j}\right] - \frac{\partial}{\partial x_j}(P u_j) \qquad (4)$$

Transport Equations for k and ε :

The transport equation for k is:

$$\frac{\partial k}{\partial t} + U_k \frac{\partial k}{\partial x_k} = -\overline{u_i u_k}\frac{\partial U_i}{\partial x_k} - \frac{\partial}{\partial x_k}\overline{\left(\frac{p}{\rho} + k\right)u_k} + \nu \frac{\partial^2 k}{\partial x_k \partial x_k} - \nu \overline{\frac{\partial u_i \partial u_i}{\partial x_k \partial x_k}} \qquad (5)$$

The transport equation for ε is:

$$\frac{\partial \rho \varepsilon}{\partial t} + \frac{\partial(\rho U_k \varepsilon)}{\partial x_k} = C_\varepsilon \frac{\partial}{\partial x_k}\left(\rho \frac{k}{\varepsilon}\overline{u_l u_k}\frac{\partial \varepsilon}{\partial x_k}\right) + C_{\varepsilon 1}\rho \frac{\varepsilon}{k} P_k - C_{\varepsilon 2}\rho \frac{\varepsilon^2}{k} \qquad (6)$$

The values of constant coefficient for the equations are:[6]

C_k =0.22, C_1=1.8, C_2 =0.6, C_{1w} =0.5, C_{2w} =0.18,

C_ε =0.18, $C_{\varepsilon 1}$ =1.42, $C_{\varepsilon 2}$ =1.92, C_μ =0.09, K =0.41

Reynolds Stress Turbulence Model:

The final Reynolds stress turbulence transport equation is as follow:

$$\frac{\partial \overline{u_i u_j}}{\partial t} + U_k \frac{\partial \overline{u_i u_j}}{\partial x_k} = -\left(\overline{u_j u_k}\frac{\partial U_i}{\partial x_k} + \overline{u_i u_k}\frac{\partial U_i}{\partial x_k}\right) + \frac{\partial}{\partial x_k}\left(\nu \frac{\partial \overline{u_i u_j}}{\partial x_l \partial x_l} + C_k \frac{k}{\varepsilon}\overline{u_k u_l}\frac{\partial \overline{u_i u_j}}{\partial x_l}\right)$$

$$-\frac{2}{3}\varepsilon \delta_{ij} + \Pi_{ij.l} + \Pi_{ij.2} + \Pi_{ij.1w} + \Pi_{ij.2w} \qquad (7)$$

Where Π_{ij} is pressure-strain redistribution tensors. Value of these tensors which indicated by Rodi [7] are as follow:

$$\Pi_{ij,l} = -C_1 \frac{\varepsilon}{k}\left(\overline{u_i u_j} - \frac{2}{3}k\delta_{ij}\right) \qquad (8)$$

$$\Pi_{ij,2} = -C_2\left(P_{ij} - \frac{2}{3}\delta_{ij}P_k\right) \qquad (9)$$

Where P_k is production of Kinetic Energy (K) and is equal to:

$$P_k = -\overline{u_j u_k}\frac{\partial U_1}{\partial x_k} \qquad (10)$$

and D_{ij} is:

$$D_{ij} = -\left(\overline{u_i u_1}\frac{\partial U_1}{\partial x_j} + \overline{u_i u_1}\frac{\partial U_1}{\partial x_i}\right) \qquad (11)$$

and :

$$\Pi_{ij,1w} = C_{1W}\frac{\varepsilon}{k}\left(\overline{u_k u_m}n_k n_m \delta_{ij} - \frac{3}{2}\overline{u_k u_i} - \frac{3}{2}\overline{u_k u_j}n_k n_i\right)f$$

$$\Pi_{ij,2w} = C_{2W}\left(\Pi_{km,2}n_k n_m \delta_{ij} - \frac{3}{2}\Pi_{ik,2}u_k u_j - \frac{3}{2}\Pi_{jk,2}n_k n_i\right)f \qquad (12)$$

Where n_i is unit vector in direction of x_i and $f = \frac{C_\mu^{3/4}K^{3/2}}{\varepsilon k x_n}$ is the wall correction equation in which x_n is normal distance from wall surface. Also K is the Von-Karman constant.

These equations together with real gas form equation will calculate 11 unknowns.

To compare and evaluate the RSM results, the standard $k-\varepsilon$ model of Launder and Spalding is used in this work as well as ASM model of Lumly et al [8].

STUDY OF CALCULATING SPACE AND MESH GENERATION

The calculation has been run for a reciprocating engine with a valve for gas input and release. Specifications of the engine are indicated in table 1.

Bore (mm)	75
Stroke (mm)	94
Compression Ratio	12.5
Engine Speed (r.p.m)	2000
Length of Connecting Rod (mm)	363.5
Max Opening of Valve (mm)	7.3
Dia. Of Piston Bowl (mm)	50
Dia. Of Cylinder bowl (mm)	65
Piston Bowl Depth (mm)	4
Cylinder Bowl Depth (mm)	10
Inlet Angle to Axis (deg.)	60

Table 1: Specification of modeling engine

General view of mesh generation and A-A and B-B cross section are presented in figure 1 to study different profiles of flow. A-A section and B-B section are respectively located at 12.5 mm from cylinder head and ½ distance between cylinder head and piston. Location of this section is different in various crank angles.

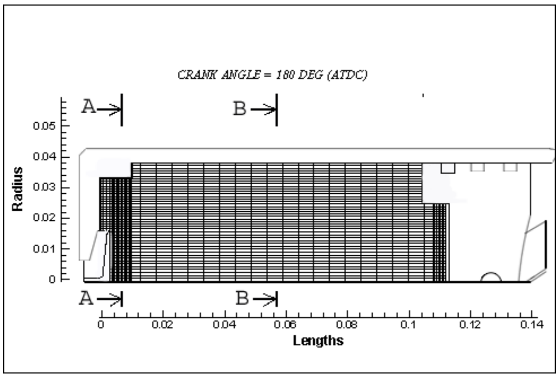


Figure1: Geometry and Grid lines at 180 Deg. crank angle

In aid of independency of results and mesh structure, considering volume of calculation after study, a 45x45 mesh is considered. So the maximum values of grid size happen in 180 Deg. Crank Angle. Also to obtain accurate results, time step shall be formed and determined. For this purpose, time steps of 0.5, 0.25, 0.125, 0.0625 degree of crank angle were studied. As shown in figure 2 for time steps of 0.125 and 0.0625 turbulence intensity profiles are located on each other and as a result, time step of 0.125 degree of crank angle was used for calculations.

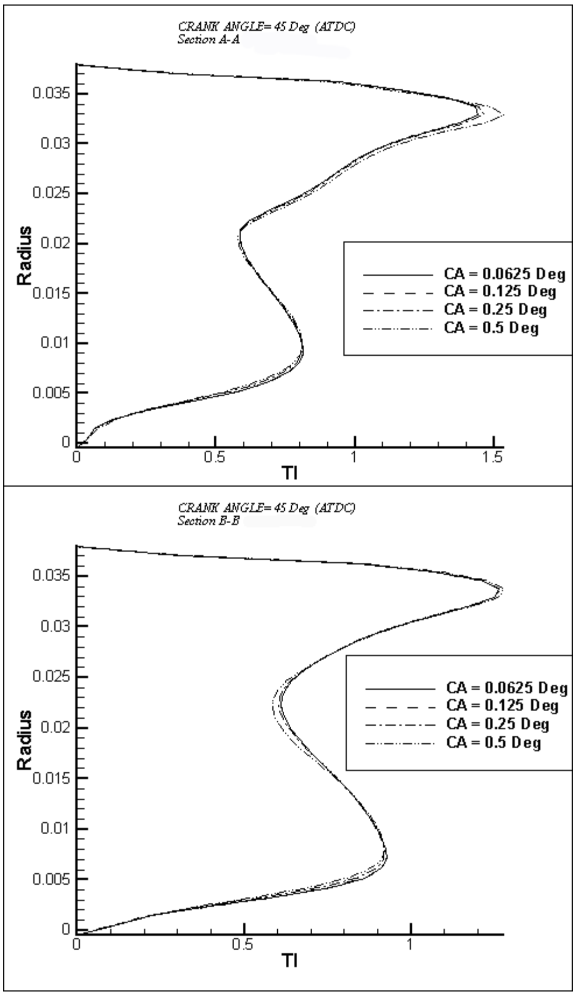


Figure2: Comparison of Turbulence Intensity for Various time step at 45 Deg. crank angle

COMPARISON OF RESULTS OF RSM, ASM AND $k-\varepsilon$ TURBULENCE MODELS

Figure 3 shows turbulence intensity in 100 degree of crank angle for sections A-A and B-B. Turbulence intensity is compared for three standard turbulence models of RSM, ASM and $k-\varepsilon$ and found that results of RSM and ASM models have significant difference with standard $k-\varepsilon$ model.

In figure 4, pressure, temperature, velocity vectors and flow field contours are observed at 180 degree of crank angle i.e. in bottom dead centre.

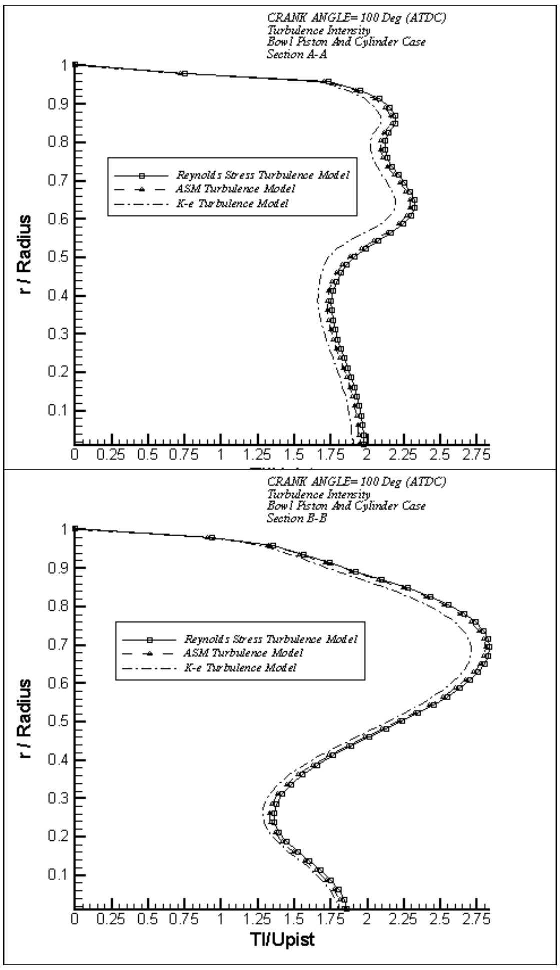
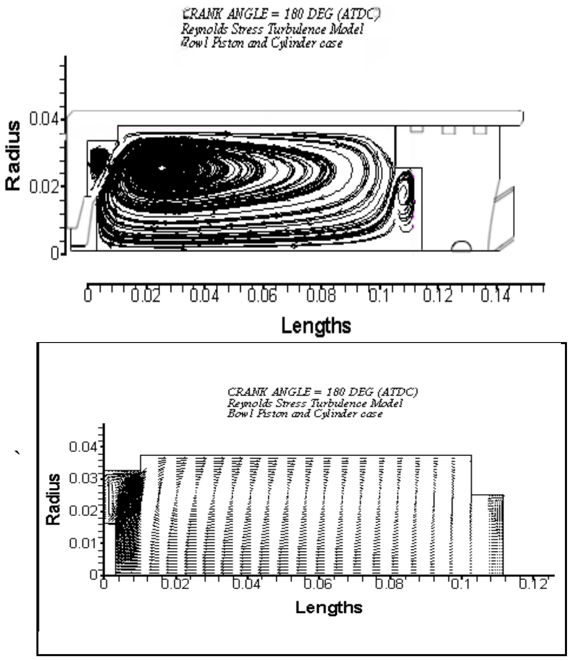


Figure 3: Turbulence Intensity for Various Turbulence Models at 100 Deg. crank angle



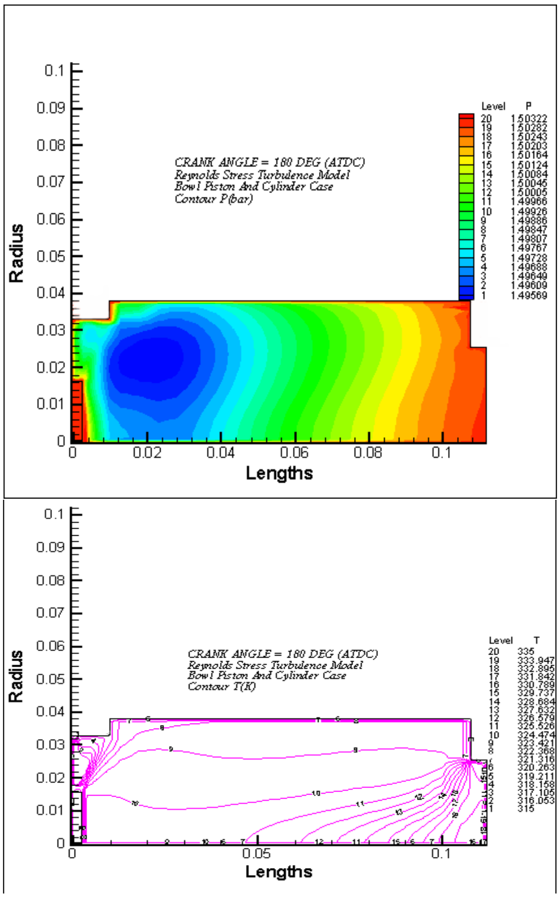
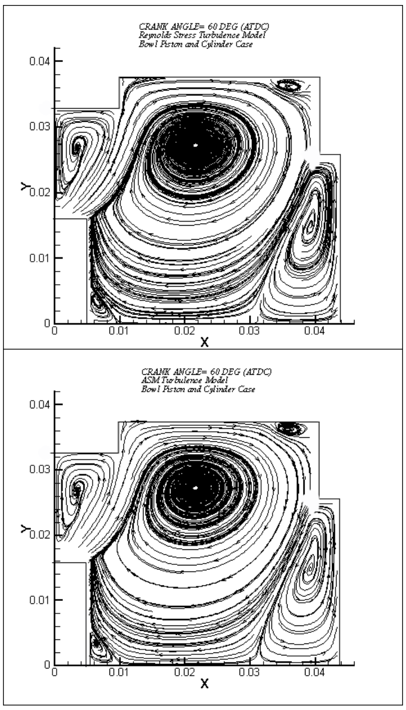


Figure 4: Pressure and Temperature contour, Vector plot, Flow Field for RSM at 180 Deg. crank angle

Flow fields, temperature contours and turbulence intensity contours are presented for different turbulence models at 60 and 340 degree of crank angle on figure 5 to 7 as sample results. As indicated in these figures, flow details in RSM turbulence model are clearer than other models. For example, at 60 degree of crank angle of flow field, small vorticity in upper side of the cylinder isn't described through $k-\varepsilon$ model.



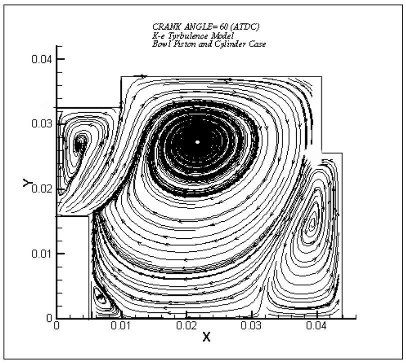


Figure 5: flow fields for Various Turbulence Models at 60 Deg. crank angle

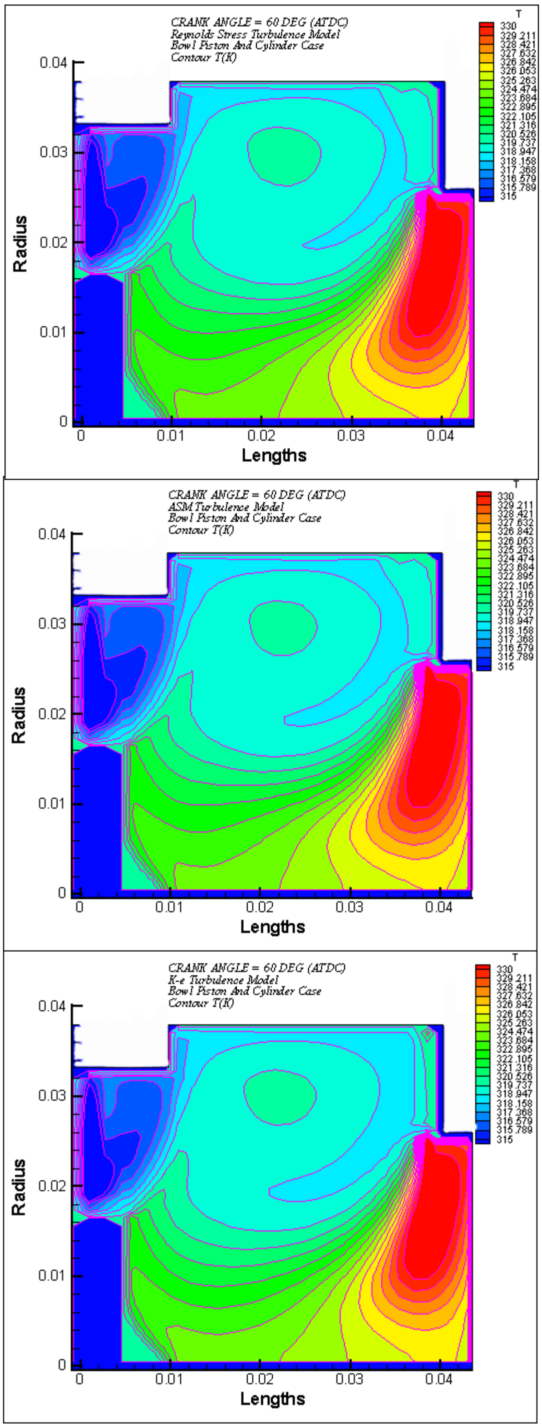


Figure 6: Temperature contours for Various Turbulence Models at 60 Deg. crank angle

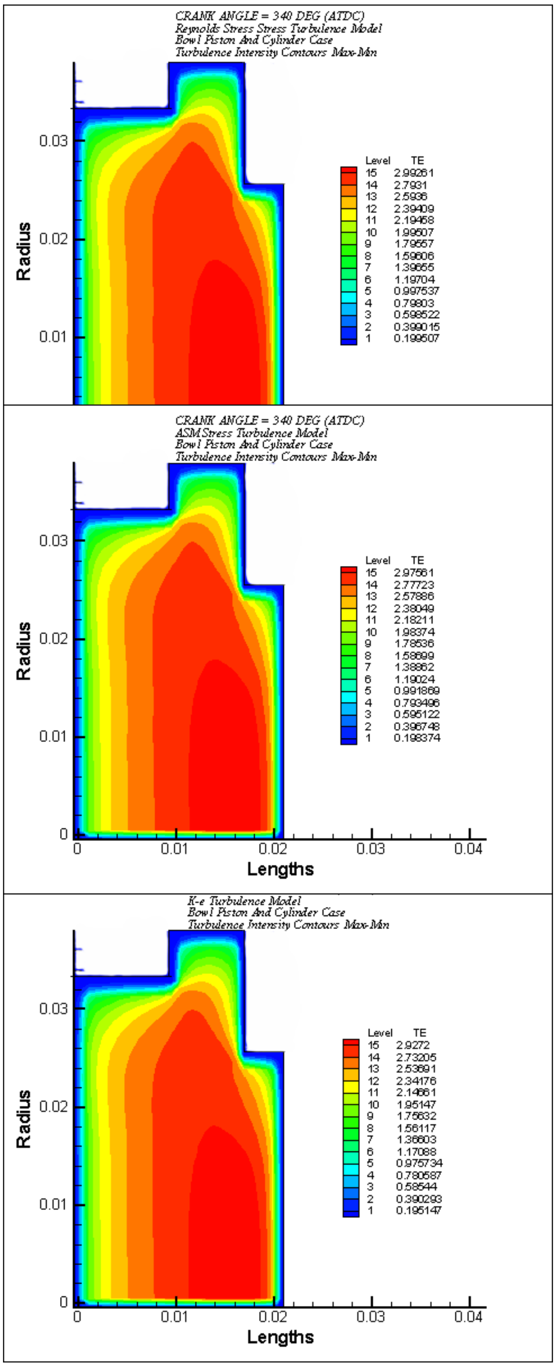


Figure 7: Turbulence Intensity contours for Various Turbulence Models at 340 Deg. crank angle

STUDY OF FLOW FIELDS DURING THE INDUCTION AND COMPRESSION PROCESS IN THE CYLINDER

During the induction and compression process three main vorticities are formed in cylinder. The first at the back of valve, the second in piston bowl and the third which is the most steady, at the middle of cylinder. Two related vorticities to valve and piston reached to their peak in 45 and 60 degree of crank angle and then with continuation of induction process and after bottom dead centre is decreasing step by step and failed out and only main

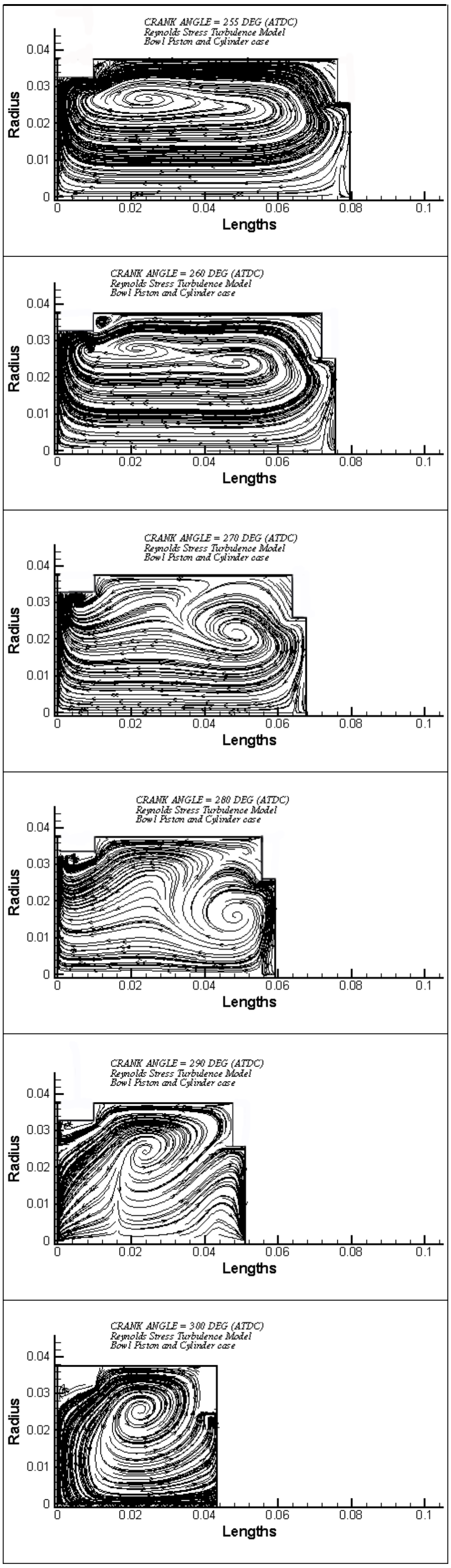


Figure 8: Flow Field for RSM at 255,260,270,280,290, 300Deg. crank angle during the reverse of flow circulation vorticity continue its life. Main vorticity as shown in figure 8 in 260 degree of crank angle also takes form of decrease and become reverse side and after passing from

30 degree i.e. in 290 degree it is completely reverse and remains in this position till end of compression stoke and top dead centre. This phenomenon is presented in figure 8 for 255 to 300 degree of crank angle.

COMPARISON OF RESULTS WITH EXPERIMENTAL DATA

The results of RSM, ASM, $k-\varepsilon$ turbulence models are compared with experimental data with use of results indicated in reference [9] for the engine with specification of table 2. The engine is without piston bowl and cylinder head plate and these two parts are supposed completely flat. Such a comparison carried out for velocity vectors and turbulence intensity at 90 degree crank angle in cross section at 15 mm distance from the cylinder head plate. As indicted in figure 9, results of turbulence model of RSM from ASM model are closer to the experimental results and these two models have long distance in rendering flows phenomena in comparison with $k-\varepsilon$ model.

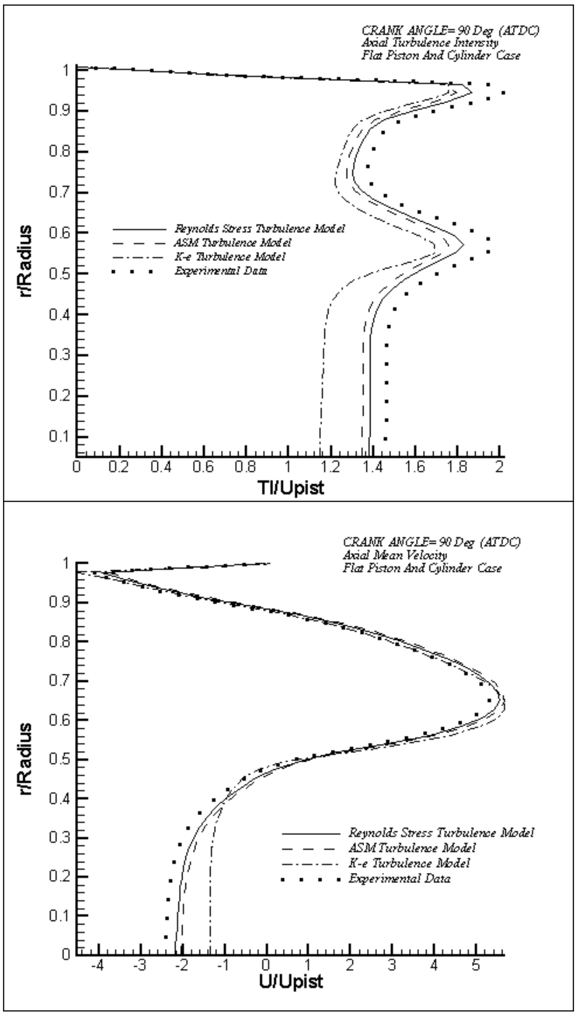


Figure 9: Axial mean velocity and turbulence Intensity profiles at 90Deg. crank angle for modelling and exp. Data

Bore (mm)	75
Stroke (mm)	94
Compression Ratio	10.5
Engine Speed (r.p.m)	1000
Inlet Angle to Axis (deg.)	30

Table 2: Specification of experimental engine

CONCLUSION

In this study, Reynolds Stress Turbulence Model (RSM) has been employed in conjunction with the non-iterative PISO algorithm to calculate the flow phenomena in an axi-symmetric internal combustion engine chamber during the induction process and compression stroke. The abilities of the model are examined and the results are compared with $k-\varepsilon$, ASM turbulence models and also experimental data. Qualitative consideration of the results shows that the RSM predicts reasonable agreement and closer results to experimental data and real in-cylinder fluid flow of turbulence phenomena in comparison with ASM model and standard $k-\varepsilon$ model, Although the computing time is almost double. Flow circulation after closing valve in compression stroke begins to redirect before top dead centre till completely reverse against initial direction. PISO algorithm has considerable ability to simulation of the turbulent in-cylinder flows.

REFERENCES

C. W. HIRT, A. A. AMSDEN and J. L. COOK, "An arbitrary lagrangian-Eulerian computing method for all flow speeds", *J. Comput. Phys., Vol 14, No 3*, 1974.

A.A.AMSDEN, T.D.Butlre, P.J.O'ROURKE and J.D.RAMSHAW, "KIVA-a comprehensive model for 2-D and 3-D Engine Siulation", *SAE 850554* , 1985.

B.E.LAUNDER and D. B. SPALDING, "Mathematical models of Turbulence", *Academic press, London*, 1972.

A. D. GOSMAN, R. J. R. JOHNS and A. P. WATKINS, "Development of prediction methods for in-cylinder processes in reciprocatig engines", *Plenum Prfess,New York* , 1980.

R. I. ISSA, "Solution of the Implicity discretised fluid flow equations by operating spiliting", *J. Comput. Phys, Vol 62, No 1*, 1986.

GIBSON,M.M. and RODI, W., "A Reymolds stress closure model of turbulence applied to the calculation of a highly curved mixing layer",*J.Fluid Mech.,Vol.103*, 1981.

RODI, W.,"Turbulence Models and their Applications in Hydraulics", *book publication of the Int. Association for Hydraulic Research Netherlands*, 1980.

TSAN-HSING SHIH, JIANG ZHU, JOHN L.LUMLY, "Anew Reynolds stress algebraic equation model", *Comput. Methods Appl. Ene., 126*(1995)287-302.

B. AHMADI-BEFRUI, B. A. D. GOSMAN, R. I. ISSA and A.P. WATKINS, "EPISO-An Implicit non-iterative solution procedure for the calculation of flows in reciprocating engine chamber", *Comput. Methods Appl. Engrg. , Vol 73*, 1989.

A.D.GOSMAN, Y.Y.TSUI AND A.P.WATKINS, "Calculation of 3D air motion in model engines", *SAE 540229.*

B. AHMADI-BEFRUI, A. D. GOSMAN, R. I. ISSA and A.P.WATKINS, "The DICE computer codes for prediction of laminar and turbulent flow and heat tranfer in idealised motored diesel engine combustion chambers", Imperial Colledge, Mech. Eng. Dept., Fluid section,Nov. , 1990.

