Fluid Flow Analysis And Flow Path Optimization Of A Centrifugal Fuel Injector For A Turboshaft Aeroengine

Shrutika Dargopatil[1] , Vinith Kumar .C.S[2], PraveenKumar Raikar[3]
Department of Mechanical Engineering
[1] M-Tech, JAIN University, India
[2,3] B-Tech, JAIN University, India

Abstract- *popular in engineering research and advanced design activities. This is primarily because of availability of good scope to achieve high power to weight ratio and increased experimentation along with the past and present experience of designers. Optimization is extensively used in engineering design problems where the emphasis is on maximizing or minimizing certain goals, i e. optimization along with numerical CFD techniques are routinely used. Such optimization techniques actually minimize overall size, of precision aero-engine components like the fuel injectors.*

The theoretical formulations used for obtaining the fuel mass flow rate are derived from the existing fluid dynamics theories taken from the open literature. The present work of optimization is made for specific dimensions of a rotary fuel injector used in two of the high performance defence helicopter's gas turbine engines presently.

The need for doing this study is to meet the thermal design requirements of limiting the combustion chamber temperatures to an optimum designed value which heavily depends on finely divided homogeneous droplet sizes, the SMD (Sauter Mean Dimension) of the fuel that is sprayed as jet of small eddies at the inlet of the primary zone. The experiment is conducted on one of fuel injectors to measure the exit mass flow and hence the discharge for a predefined time, it is basically a pressure-driven cold flow analysis employed to study the performance of the fuel injector under various engine operating conditions. To study the effect of the various inlet and exit pressures at various altitudes and the corresponding effects, a program is written and analysed in MATLAB 1D solver, ICEMCFD, and FLUENT3D solver. The one dimensional optimization algorithm includes. A steady state theoretical formulation using continuity and modified energy equation in the form of one equation flow model. This flow model additionally accounts for the relative wall roughness, the slip factor which is a direct function of the surface finish on the material of the specimen, mechanical losses that include contraction, expansion, exit, bending effects and number of holes required for fuel injection. Further, another simple idealized two-equation model is programmed that gives a crude idea of the mass flow at various rotational speeds alone. The rotational effects that are imposed during the operation of the injector are also studied by modelling the flow in two of the commercial finite element CFD soft ware's and the approximate results of comparison are made with theoretical model and experimental data for clarity of this investigation. An attempt is also made to study some of the numerical schemes expressed in the form of Turbulence Models that are used in the design of experiments of fluid flow, like Direct Numerical simulation as applied to pipe flow studies.

I. INTRODUCTION

Helicopters are highly capable and useful rotating wing aircrafts that have a variety of civilian and military applications. This usefulness lies in their unique ability to take off and land vertically, to hover stationary relative to the ground, and to fly forward, backward or sideways. These unique qualities however come at a price including complex aerodynamics problems, significant vibrations, high levels of noise, and relatively large power requirements compared to a fixed-wing aircraft. Fuel injectors play a dynamic role and provide high performance requirements of these helicopters at various engine operating conditions, at various altitudes from take off to landing respectively.

II.NAVIER-STOKES EQUATIONS OF FLUID DYNAMICS

General: Incompressible and Viscous

- **Continuity**

$$\nabla . V = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \qquad (2.1)$$

- **Momentum**

$$\rho\left(\frac{\partial}{\partial t}+\nabla.V\right)u=-\frac{\partial p}{\partial x}+\mu\nabla^2u+\rho f_x \quad (2.2)$$

$$\rho\left(\frac{\partial}{\partial t}+\nabla.V\right)v=-\frac{\partial p}{\partial y}+\mu\nabla^2v+\rho f_y \quad (2.3)$$

$$\rho\left(\frac{\partial}{\partial t}+\nabla.V\right)w=-\frac{\partial p}{\partial z}+\mu\nabla^2w+\rho f_z \quad (2.4)$$

III. NAVIER–STOKES EQUATIONS (RANS)

The Reynolds-averaged Navier–Stokes (RANS) equations are time-averaged equations of motion for fluid flow. They are primarily used while dealing with turbulent flows. These equations can be used with approximations based on knowledge of the properties of flow turbulence to give approximate averaged solutions to the Navier–Stokes equations. For a stationary, incompressible flow of Newtonian fluid, these equations can be written as:

$$\rho\frac{\partial \bar{u}j\bar{u}i}{\partial xj}=\rho\bar{f}i+\frac{\partial}{\partial xj}\left[-\bar{p}\delta ij+\mu\left(\frac{\partial \bar{u}i}{\partial xj}+\frac{\partial \bar{u}j}{\partial xi}\right)-\overline{\rho u'\imath u'\jmath}\right] \quad (3.1)$$

The left hand side of this equation represents the change in mean momentum of fluid element owing to the unsteadiness in the mean flow and the convection by the mean flow. This change is balanced by the mean body force, the isotropic stress owing to the mean pressure field, the viscous stresses, and apparent stress ($-\overline{\rho u'\imath u'\jmath}$) owing to the fluctuating velocity field, generally referred to as Reynolds stresses.

IV. FUEL INJECTOR DESIGN PRINCIPLES

Fuel injectors are actually orifices which control the flow of fuel, gas or a gas-air mixture into the combustion zone. These orifices may be round or slotted. To fulfill the above requirements, the following factors and their effects on fuel injector performance must therefore be considered.

a. Discharge coefficient and the mass flow rate.
b. Injector pressure drop.
c. Turndown ratio.
d. Velocity of Fuel jets(s).
e. Cone angle or Direction of fuel jet(s).
f. Stabilizing devices.

a) Discharge coefficient and the mass flow rate:

Since the fuel is injected through an orifice, the standard orifice equation may be used to determine the mass flow rate. The orifice is sized to provide the required heat input rate to the combustor. As with liquid injectors, the state of the fuel holes is important and care in manufacture is required to provide holes free of burrs, of equal diameter, and of uniform pitch and angle in order to ensure symmetrical flow distribution into the primary zone of the combustion chamber.

b) Injector pressure drop:

The pressure drop [8] of the fuel over the injector orifice is an important consideration in the design of injector. Sufficient pressure drop is required to produce an even distribution of fuel flow. From literature, it is recommended that for industrial gas turbines the pressure drop over the injector should be about 10% of the supply pressure at full load to provide stable and high efficiency combustion. This produces a uniform turbine inlet temperature distribution and maximizes the operating life of the hot components. Such a pressure drop also improves fuel control, giving stable repeatable operating conditions. The same reference [16] indicates that with too high a pressure drop, results in unstable operation, creating undue thermal s tresses on hot sections.

c) Turndown ratio:

The turndown ratio is a measure of variability of the operating conditions for which stability and combustion efficiency are still acceptable [4]. It is, therefore, directly dependent on orifice geometry and on the fuel pressure drop. For most applications, the ratio should be quite large, requiring significant attention in the design.

d) Velocity of Fuel Jet(s):

For a given fuel, the jet velocity is directly related to the orifice size and pressure drop. The velocity can, however have a significant and direct effect on the combustor performance, since the relative velocities of the air and the fuel jets determine their mixing rates in the combustion zone.

e) Cone angle or Direction of fuel jet(s):

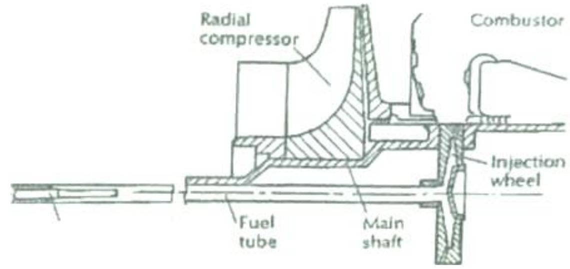
As with the fuel jet velocity, fuel jet direction can also have a significant effect on combustor performance through its effect on mixing rate.

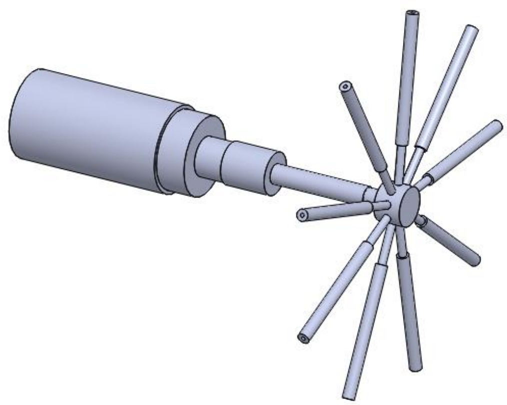
f) Stabilizing devices:

The use of stabilizing devices is essential in burner design. Stabilizing devices usually consist of bluff bodies known as baffle plates or, more commonly, swirling devices placed in the annulus around the fuel injector. The primary use of swirl is to increase the angle of spread and the decay of axial velocity of a jet. It there by increases the entrainment and rate of mixing of the air and the fuel jets, providing improved flame stabilization, giving a wide turn down ratio.

Rotary atomization possesses much appeal as a fuel injection method particularly suited for gas turbine applications, where a rotating shaft is available any way. One of its advantages is the ability to produce fine fuel sprays also at part load or idle conditions, a regime where swirl nozzles exhibit problems with rapidly increasing droplet diameters. Of course, reliable solutions, e.g., dual-orifice nozzles, have been found, but at the expense of increasing design and manufacturing complications. With rotary injection the decrease of the shaft speeds from 100% at full load to 60 % at idle produces an increase in the drop diameters of about 65%. According to Norster’s [16] results, this would yield Sauter mean diameter at idle conditions of the order of 50 micrometer, if the SMD value at full load corresponded to about 30micro-meter, without further complications of the control system. However the use of rotary atomization [3] in gas turbines also causes specific design problems. In order to achieve small droplet diameters, the exit diameter of the injection wheel must be made large. Furthermore, the annular combustor design has to be specifically adapted to the rotating injection, and the fuel has to be supplied through the hollow shaft. Therefore, rotary atomization has only been applied to single-shaft gas turbines. Examples of gas turbines with rotary injection are the family of early single shaft, turbojet and turbo shaft engines developed by Turbomeca in France. Several single shaft engines for propulsion applications are developed by KHD and BMW in Germany.

Fig (1a&b). shows the design of main shaft of the Turbomeca Marboree engine, containing the injection wheel. It is mounted flush in the main shaft drum, which extends aft to the turbine and forward to the centrifugal impeller and the front bearing. Inside this main hollow shaft the fuel supply tube is mounted coaxially. It extends into the front end of the fuel labyrinth seal where it is forced outward along the inner surface of the hollow injection wheel. A dome like structure prevents uncontrolled flows and restricts the fuel to one of the radial inner surfaces. At the crest of the inner spaces two rows of the injection bores originate which are inclined at 35 degree to the radial direction, thus improving the fuel distribution in the combustor.





Fig(1a&1b):Schematic representation of the rotating fuel injection system used in the Turbomeca Marbori turbojet engine.

V. EXPERIMENTAL DATA

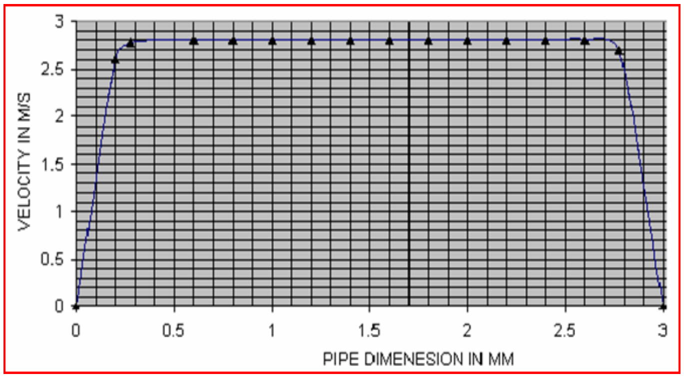


Fig3: Experimental data for exit mass flow.

VI. FORMULATION AND THEORETICAL VALIDATION STUDY

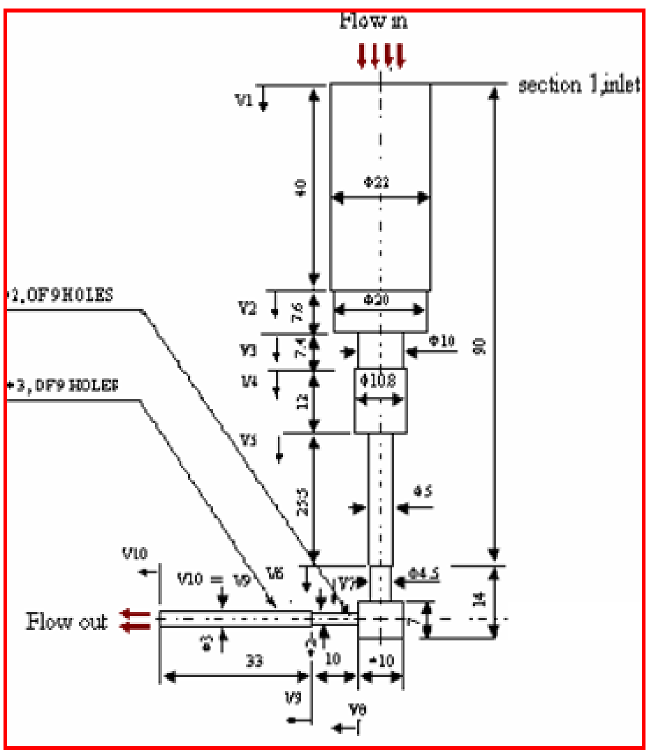
The main objectives of the present work are;

a) To develop a theoretical flow model for fuel mass flow rate to the combustion system and analyse and modify the flow field in the fuel injection system.

b) To conduct the parametric studies to optimize the fuel mass flow rate to meet the fuel requirements at different engine operating conditions.

During this formulation, the pressure is kept constant, and a result of which output has changed from 19.33×10^{-6} m^3/s to 20.83×10^{-6} m^3/s. For every 0.1mm dimension we have got 0.277×10^{-6} m^3/s increase in mass flow. Even though there is a fluctuation of 0.027×10^{-6} m^3/s or 0.055×10^{-6} m^3/s, we have accounted it for uncertainty. The surface finish should be high to reduce the frictional losses. It has to be taken care when manufacturing fuel injector from diameter 4.5 to 5.0mm. In order to get that type of surface finish, honing, buffing, lapping and other high surface finishing operations are done.

This optimization algorithm what we have designed, it accounts for the friction factor, that is going to be occurring between the fluid (aviation kerosene) and the wall of the fuel injector. Friction factor is optimized and it is to be verified with the machining operation inside the fuel injector surface. Contraction factor is a function of surface finish so optimized parameters are going to be the optimized value of contraction factor for a given fuel injector..The scope of the study is limited to establishing the singe and complete mathematical model for a body moving in a rotating frame of reference and to study the resulting dynamic characteristics of injector i.e. good mixing, quality of atomization achieved by rotation, flame attachment characteristics etc.



Fig(2):Schematic layout of injector in section.

VII.FORMULATION 1 (V1)

Diameter	Eq.Dia(mm)	Eq.Area Ratio	Dia values	Continuity Eqn	Simplified (V1) form
d1			22	V1=(d1/d1)^2*V1	1*V1
d2			20	V2=(d1/d2)^2*V1	1.21*V1
d3			10	V3=(d1/d3)^2*V1	4.84*V1
d4			10.8	V4=(d1/d4)^2*V1	4.1495*V1
d5			5	V5=(d1/d5)^2*V1	19.36*V1
d6			4.5	V6=(d1/d6)^2*V1	23.9012*V1
d7			10	V7=(d1/d7)^2*V1	4.84*V1
D1	6	13.44444	2	V8=(d1/D1)^2V1	13.4444*V1
D2	9	5.975309	3	V9=(d1/D2)^2*V1	5.9753*V1

VIII. RESULTS AND DISCUSSION

V1	0.4634	m/s
V2	0.5608	m/s
V3	2.2432	m/s
V4	1.9232	m/s
V5	8.9729	m/s
V6	11.0776	m/s
V7	2.2432	m/s
V8	6.2318	m/s
V9	2.7694	m/s
V10	V9	m/s

Fig (5): Section velocities

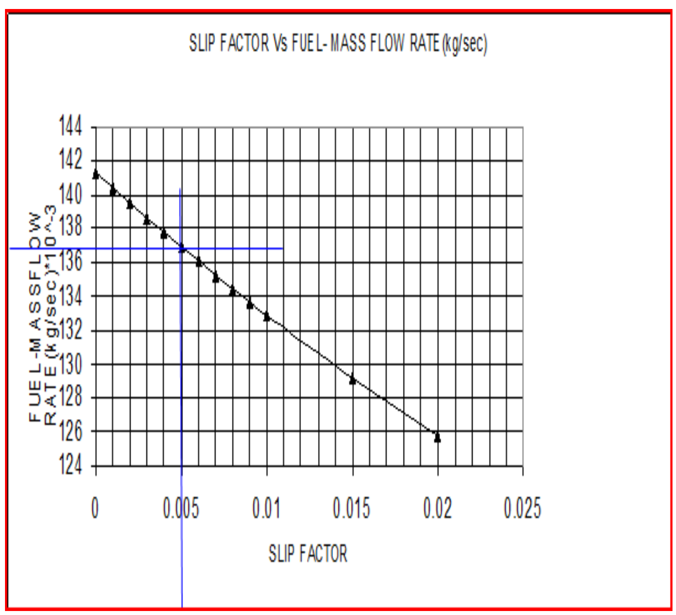


Fig (6): Effect of slip factor

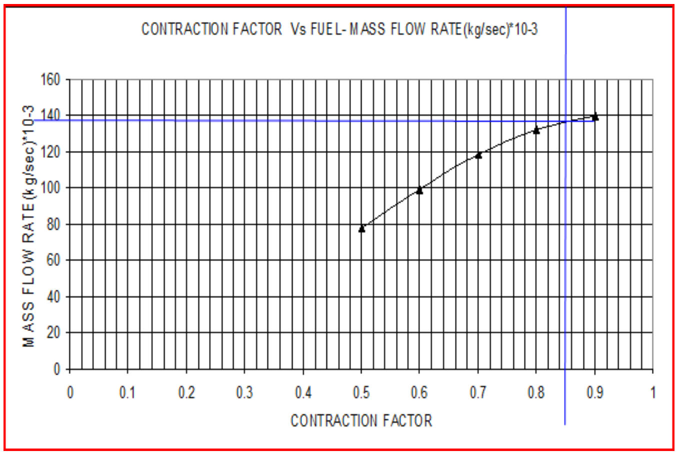


Fig (7): Effect of contraction factor

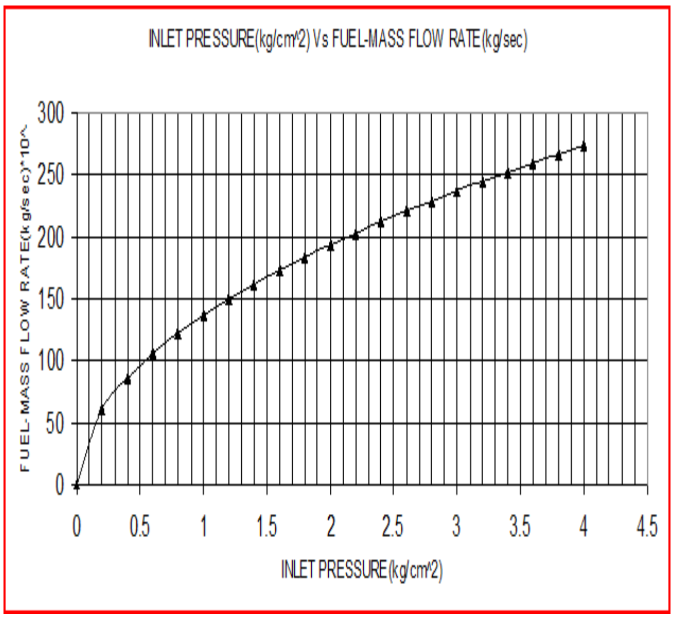


Fig (8):Effect of inlet pressure

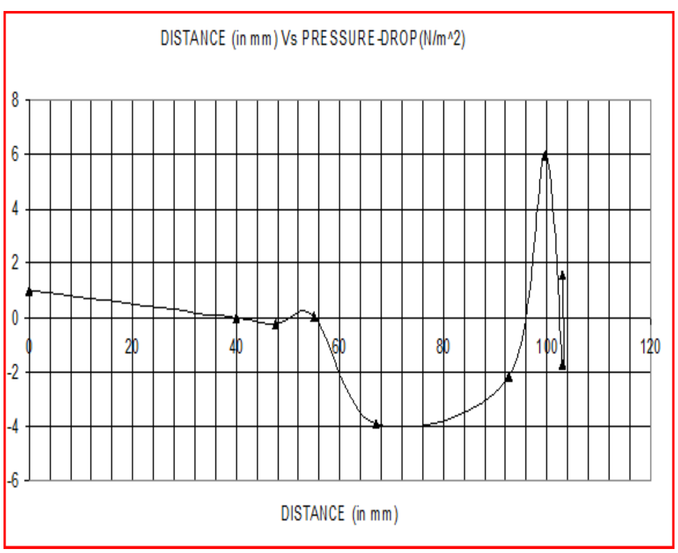


Fig (9): Pressure drop across the injector

XI. NUMERICAL INVESTIGATIONS

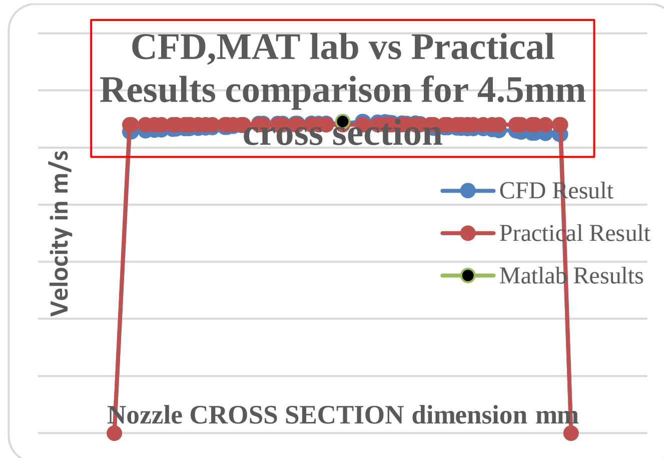
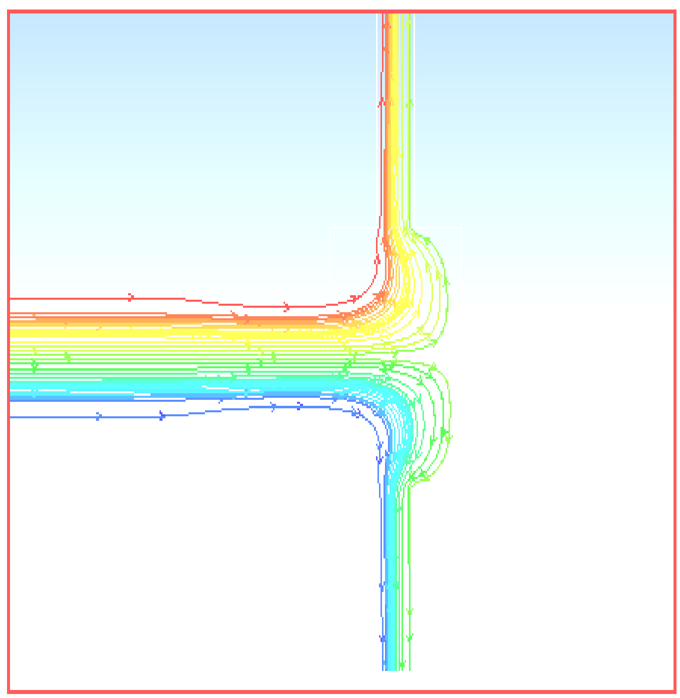


Fig (9)**:** Three dimensional Velocity profile VY (avg) at the exit of Nozzle.

The theoretical validation flow model includes examination of several aspects, which very difficult to implement in numerical viscous flow CFD tools like FLUENT 2D and FLUENT 3D etc.

We use SIMPLE algorithm for pressure velocity coupling as flow model and we have used second order upwind schemes. There are several iterative methods available in FLUENT for solving simultaneous equations that result from discretization of equations of conservation of motion for the finite volume grid chosen.



Fig(10):Initial flow field

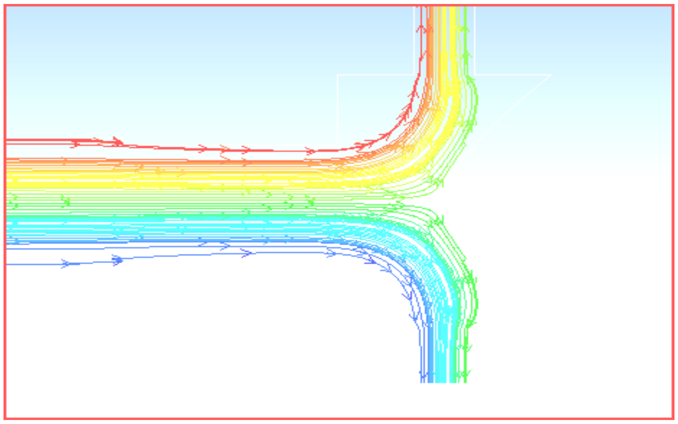


Fig (12): Modified flow field

X. ACKNOWLEDGEMENT

The authors gratefully acknowledge the contributions of Prof. Emeritus. Dr.S.S.Desai, Formal Head of CTFD and Senior scientist, for his valuable comments on the graphs obtained from numerical solutions and the turbulence modeling.

REFERENCES

[1] Fluid Mechanics fifth edition Frank M.White, McGraw Hill,© 2003
[2] Fluid Dynamics Robert W Fox and Mc Donald fifth edition John Wiley and sons©2004.
[3] MELLOR, A. M.Simplified physical model of spray combustion in a gas turbine engine, Combust. Sci. Technol. 8,101 109 (1973]
[4] Computational Methods for Fluid Dynamics third revised edition Joel Ferziger/Milovan Peric. ©2001
[5] Finite Element Computational Fluid Mechanics-A.J.BAKER. A Series In. Computational Methods In Mechanics and Thermal Sciences, © 1987
[6] Applied Fluid Mechanics-Robert.L.Mott, 5th Edn- © 2000,Prentice Hall Inc,pp-161,165.
[7] J. Mathieu/J.Scott-An Introduction to Turbulent Flow-Cambridge University Press-©2000,pp-57,367
[8] Pressure Losses in Ducted Flows -A.J.Ward Smith,Butter Worth London- © 1971.
[9] Tuncer Cebeci Peter Bradshaw-Momentum Transfer in Boundary Layers-A Series In Thermal &Fluid Engg,Hemisphere Publishing Corporation,McGraw Hill-© 1977.
[10] Finite Analytic Methods In Flows & Heat Transfer-Ching Jen Chen,Richard Bernatz,Kent D.Carlson,Wanlai Lin. Published in 2000 by Taylor &Francis-NY.
[11] Introduction to Fluid Dynamics-Stanley Middleman ©2000.pp,218-225
[12] Experimentation for students of Engineering-volume -3, Fluid Mechanics, Thermodynamics,Heat Transfer © 1997-S.M.Fraser,R.S.Hill,J.F.T.MacLaren,S.D.Probert.pp,22-163.
[13] An Informal Introduction to Theoretical Fluid Mechanics-James Light hill-CLARENDON PRESS OXFORD- © 1985.pp, 160-165.
[14] Computational Fluid Dynamics Principles &Applications-J. Blazek- © 2001,ELSEVIER.
[15] Fundamentals of Computational Fluid Dynamics (scientific computation]-Harvard Lomax, Thomas H.Pulliam,David W.Zing-©Springer Verlag Berlin Heidelberg- © 2001.
[16] Combustion Treatise, Design of Modern Turbine Combustors-Edited by A.M.Mellor,Vandert University,Nashville,Tennessee,USA-Academic Press Harcourt Brace Jovanovich, Publishers © 1990
[17] g.n.abrahamovich :the theory of turbulent jets-technical editing by leonh.schindel-THE MIT PRESS ,MIT, CAMBRIDGE , © 1993 by MIT,pp,391-435(jets in finite space of gas turbine combustion chamber].
[18] Finite Elements in Fluids-vol-3, edited by R.H.Gallaher,O.C.Zeikiwicz,J.T.Oden,M.Morandi cebeci &C.Taylor John Wiley & Sons-1978. pp,1-50.