Simulation of secondary flow in lubrication pipe with end bends of an Aero engine

Manoj Kumar Mishra[1], V.Reddy[2], Girish K Degaonkar[3]

[1]*Lubrication System Group, AERDC, HAL, Bangalore, Karnataka, India*

---***---

Abstract - *Understanding the oil flow behaviour inside the bend pipes is an essential part of effective lubrication system design in an aero-engine. This paper presents a simulation of the oil flow behaviour in the pressure pipe line of an aero engine. The turbulent hot oil flow behaviour inside the pipe with bend at two ends is simulated using commercially available computational fluid dynamics (CFD) codes (ANSYS CFX 14). The Turbulence model used for this simulation is shear stress transport, SST k-ε turbulent model. The flow characteristics are investigated and presented here at five different sections between inlet and outlet. It was observed that the variation of velocity profiles and formation of Dean Vortices at these five sections are due to curvature in the pipe. The other parameters like velocity, pressure, total pressure, pressure gradient and eddy viscosity vortex contours visualized along the length of pipe is presented.*

Key Words: Aero-engine, pipe with bend, CFD

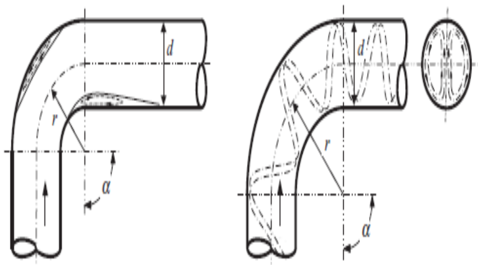
1. INTRODUCTION

Pipe flows under pressure with bends are required in aero engine lubrication system. There are varieties of bend used in aero engine pipes due to complicated engine construction. Lubrication system in aero engine has one pressure, scavenge and vent line. A pressure line pipe in the lubrication system of an aero engine is responsible for transporting the lubrication oil from the oil tank to the engine bearings through the gear pump. The design of this bend pipe is depend on the understanding of flow behaviour. Energy input by the oil pump to the oil flow is needed to make it flow through the pipe .In pipe flow substantial energy is lost due to frictional resistances and bend radius. In this analysis attention was to a very commonly used bend pipe in an aero engine.

1.1 Flow through the bends

The fluid typically enters the pipe with a nearly uniform velocity profile at entrance region. As the fluid moves through the pipe, viscous effects cause it to stick to the pipe wall (the no-slip boundary condition). This is true whether the fluid is relatively inviscid air or very viscous oil. Calculation of the velocity profile and pressure distribution within the entrance region is quite complex. However, once the fluid reaches the end of the entrance region, the flow is simpler to describe because the velocity is a function of only the distance from the pipe centerline, r, and independent of x. This is true until the character of the pipe changes in some way, such as a change in diameter, or the fluid flows through a bend, valve, or some other component. The flow is not fully developed flow at bend sections due to bend.

The pressure loss in pipe bends has three components. One component is the pressure loss due to ordinary surface friction that corresponds to fully developed flow in a straight pipe having the same length as the centerline of the bend. A second component is due to a twin - eddy secondary flow superimposed on the main or primary flow due to the combined action of centrifugal force and frictional resistance of the pipe walls. A third component is due to separation of the main flow from the inner and outer radius of the bend and subsequent expansion of the contracted stream. For bends of small radius of curvature, flow separation and secondary flow dominate. For bends of large radius of curvature, ordinary surface friction and secondary flow prevail. Flow separation and secondary flow are illustrated in Figure 1.



Flow separation Secondary flow

Fig -1: Flow behavior in bend pipe[8]

As noted above, the bend radius ratio r / d is defined as the ratio of the centerline radius of the bend to the inside diameter of the pipe. In this context, a bend of radius ratio 0.5 represents a bend with a sharp (zero radius) inner corner and an outer bend radius of 1.0. The rounding of the corner at the inner wall, or simply beveling the corner, greatly attenuates the separation and reduces the pressure loss. At the opposite extreme, bend losses, excluding friction losses, are at a minimum when the bend radius ratio is at a maximum. Figure 2 explains about the loss coefficients with bend of radius ratio. It shows the development of secondary flow and separation corresponding to bend of radius ratio.

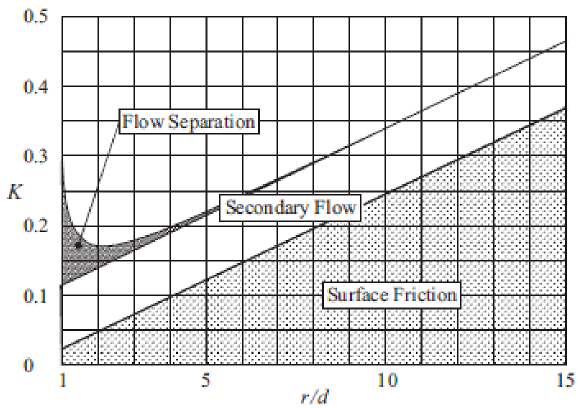


Fig -2: Effect of surface friction, secondary flow and flow separation [8]

1.2 Literature Survey

D., Bhandari., and Dr. S., Singh. [1], CFD analysis carried out by using fully developed turbulent flow. Variation of axial velocity and skin friction coefficient along the length of pipe was analyzed. G,B,Nimadge. And S,V,Chopade.[2], investigated about steady and incompressible flow through T joint by using CFD. The results were compared with testing data and found good agreement with software. Mohammed, A, Al-Y.[3],the study has been carried out by for Y shaped pipe by using CFD. The effect of bend angle, pipe diameter, pipe length, Reynolds number on the resistance coefficient is studied. It was observed that resistance coefficient vary with the change in flow. F,Hellstrom., and L,Fuchs. [4], the steady and pulsative turbulent flows in curved pipes have been computed with two different modeling approaches; the Reynolds Averaged Navier-Stokes (RANS) technique and Large Eddy Simulations (LES).He investigated that different types of small amplitude secondary flow at the inlet affect the flow downstream of the bend. The pulsatile flow in a double bended pipe has also been investigated and the vortex cores are visualized to enable a better insight into the unsteady flow field and the effects of the inflow pulsations. Adib, Z, Jaswar, K., and Yasser. [5], the simulation is performed in time-dependent simulation where oil and water are initially separated by patching the region base on difference in density. Observation on the effect of velocity to the pressure gradient was also simulated at different velocities. Stratis, K., and Michael, F. [6], this paper investigated the flow behavior of oil flow through the scavenge pipe of aero engine. The author provided the methodology for modelling and simulation of the two phase flow of air and oil in the scavenge pipe of an aero engine.

1.3 Governing Equations

The most of fluid flow can be mathematically described by the use of continuity equation and momentum equation. Theoretically, in steady or turbulent flow field the continuity, and momentum equations for incompressible, time dependent and viscous fluid are represented as follows;

a) Continuity Equation

$$\frac{\partial \rho_f}{\partial t} + \frac{\partial(\rho_f v_x)}{\partial x} + \frac{\partial(\rho_f v_y)}{\partial y} + \frac{\partial(\rho_f v_z)}{\partial z} = 0 \ldots\ldots\ldots\ldots (1)$$

Where V_x , V_y, V_z are velocity vectors in the x, y, z directions respectively, ρ_f is the fluid density and t is the time.

b) Momentum Equation

$$\frac{\partial(\rho_f v_i)}{\partial t} + v_i\left(\frac{\partial \rho_f}{\partial t} + \frac{\partial(\rho_f v_x)}{\partial x} + \frac{\partial(\rho_f v_y)}{\partial y} + \frac{\partial(\rho_f v_z)}{\partial z}\right)$$

$$= \rho_f g_i - \frac{\partial P}{\partial i} + R_i + \frac{\partial}{\partial x}\left(\mu_c \frac{\partial v_i}{\partial x}\right) + \frac{\partial}{\partial y}\left(\mu_c \frac{\partial v_i}{\partial y}\right) + \left(\mu_c \frac{\partial v_i}{\partial z}\right) + T_i \ldots\ldots\ldots\ldots (2)$$

c) For this study fully developed flow has been applied at pipe inlet section as shown in figure 11.The equation used for this is;

$$U = W_{max}\left(1 - \frac{r}{R_{max}}\right)^{\frac{1}{7}} \ldots\ldots\ldots\ldots (3)$$

Where W_{max} is the pipe centre line velocity,R_{max} is the pipe radius, and r is the distance from the pipe centre line.

The variable r is a CFX system variable defined as;

$$r = \sqrt{x^2 + y^2} \ldots\ldots\ldots\ldots (4)$$

In this equation, x and y are defined as directions for Cartesian coordinate frames respectively, in the selected reference coordinate frame.

d) Shear stress transport model (SST)

The turbulent viscosity of the BSL model is based on assumption that the turbulent shear stress is proportional to the turbulent kinetic energy in the logarithm and wake regions of the turbulent boundary layer.

Turbulent viscosity is defined as;

$$\gamma_t = \frac{a_1 k}{\max(a_1\omega, \ \Omega F_2)} \ldots\ldots\ldots\ldots (5)$$

Where a = 0.31,$\Omega = \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}$,$F_2 = Tanh\,(arg_2^2)$

With $arg_2 = \max\,[\,2\,\frac{\sqrt{k}}{0.09\,\omega\,y}, \frac{500\,\gamma}{\omega\,y^2}\,]$

2. NUMERICAL SIMULATION

2.1 Geometry and mesh

The domain and the meshes were created using ANSYS CFX 14. A sketch of the geometry of the calculation domain is shown in Figure 3. The geometry consists of two bends one is at inlet and other is at outlet. The pipe inlet is connected to gear pump which carries the viscous oil at temperature of 110°C from oil tank and discharged with high pressure of 5 bar. The pipe outlet is connected to engine inlet where the oil is distributed to the bearings for cooling purpose. The neutral length of pipe is 194.05 mm with bending radius of neutral fiber is 40 mm. The inside diameter of the pipe for present work is 14.3 mm. The flow is fully developed turbulent flow considered at inlet with velocity of 1.45 m/sec. Flow is continuously flowing from inlet to outlet due to positive displacement gear pump.

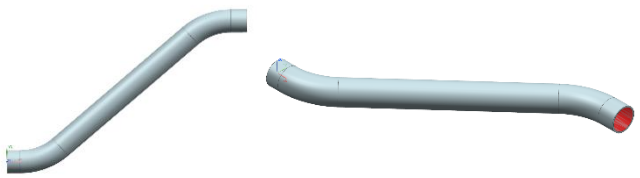


Fig -3: Geometry

A block-structured meshing approach was used to create meshes with only Hex cells. To capture the boundary layer effect accurately, the distance of first node is placed according to the y+ calculation. A y+ value of 1 is taken according to the selected SST turbulence model. The calculated value of first node distance as per the y+ value of 1 is 0.000008 m. The mesh has been generated as per calculated first node distance with a growth rate of 1.2 (Figure 4 shown)

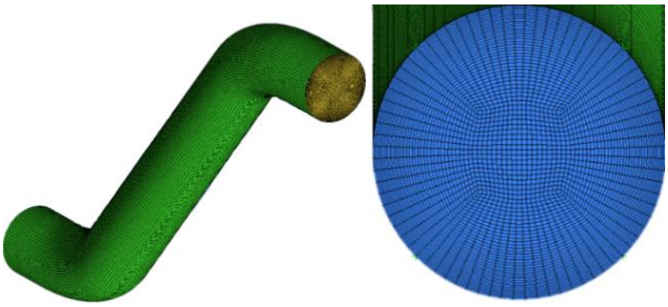


Fig -4: hex meshes

2.2 Boundary conditions

Fully developed velocity profile for oil was introduced at the pipe inlet or oil pump outlet shown in figure 5 and figure 6. The outlet of the pump or inlet of pipe boundary condition was set up as an average static pressure of 5 bar.

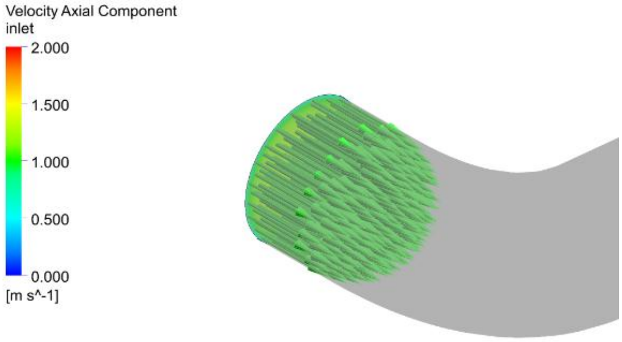


Fig -5: Fully developed velocity profile at inlet

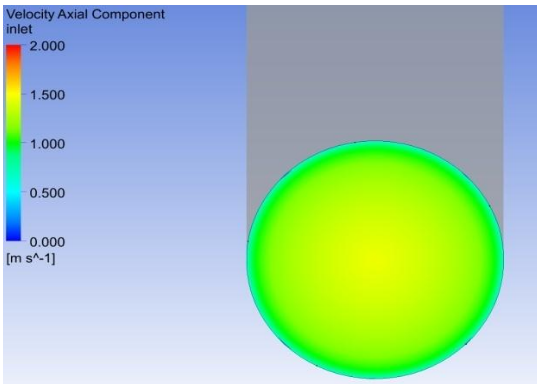


Fig -6: Applied Velocity profile at inlet

The main fluid physical properties and geometry boundary conditions are reported in Table 1 and Table 2 resectively.

Table -1: Fluid properties with geometry details

Physical properties of fluid and pipe			
Fluid properties		Geometry details	
Fluid type	Oil (MIL-PRF-23699)	Pipe material	Stainless steel
Pipe	Pressure line pipe	Pipe ID	14.3 mm
Viscosity	4.17 cSt	Pipe OD	16 mm
Density	943.72 kg/m³	Bend radius	40 mm
Oil mass flow	0.22009 kg/Sec	Length of neutral fiber	194.05 mm

Table -2: Boundary conditions

Model used	SST k-ε turbulent
inlet velocity profile	As per equation no 3
Inlet oil temperature	110 ° C
Pump outlet average static pressure	5 bar

2.2 Solution strategy and convergence

Convergence criteria of 0.00001 is considered

3. RESULTS

This section displaying the property gradients of the model such as velocity, pressure, total pressure, pressure gradient and eddy viscosity vortex in the form of contours along the length of pipe. Dean Vortices visualization has been predicted using this simulation at five different locations as shown in figure 11.A grid independent study has been carried out.

3.1Grid Independent study

A grid independent study was conducted to obtain sufficient mesh density for accurate flow. A grid independent solution exists when the solution does not change when the mesh is refined. The computational grid of230000, 276000, 31000,397000,476000,572000,686000 and 823000 elements were tested for the mesh independent study to find out the optimum size of the mesh to be used for simulation. 230000 showing bad prediction on the oil layer since insufficient amount of elements. Therefore, based on the oil mass flow vs mesh element chart 1, 685691and 823000 cells are the most optimum number of cells required to predict the oil flow behavior in the tested domain and such mesh is going to be used for simulation. Corresponding to this mesh element the flow behavior prediction is better. Quality check has been carried out and all the mesh elements are within the acceptable limit of CFX solver.

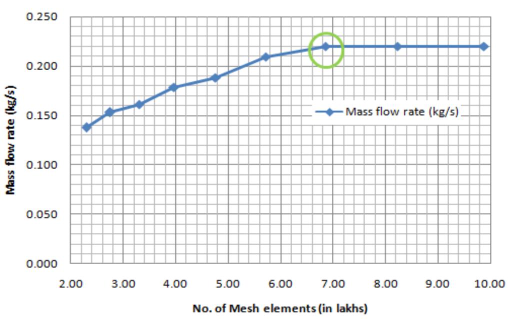


Chart -1: Grid Independent study

3.2 Flow visualization

By using this simulation of hot oil turbulent flow, the velocity and pressure variations have been plotted in figure 7 and figure 8. The inlet velocity is 1.45 m/sec. The figure 7 shows that the velocity is maximum at the entrance region and it varies along the pipe. Maximum velocity observed is at inner curvature and minimum at upper curvature region. The minimum velocity is 0 at the inner curvature of pipe wall at exit region. The color contours shows the pattern of variations in pressure. The pressure at entrance of pipe or pump outlet is 5 bar reduced to 4.9 bar at exit of pipe. The pressure variation is low due to shorter pipe length as shown in figure 8.

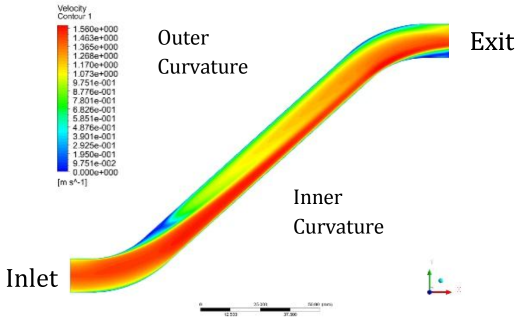


Fig -7: Velocity contour plot

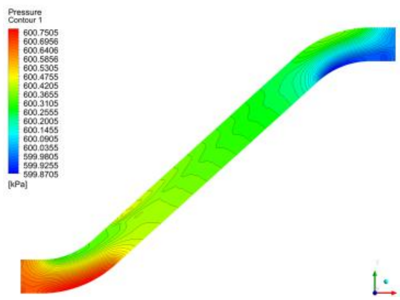


Fig -8: Pressure contour plot

Total pressure contour illustrates that the pressure is high at inlet with fully developed flow. The pressure variation is low towards the outer curvature than inner curvature along the length of pipe (Shown in figure 9).Total pressure slowly decaying towards the outer.

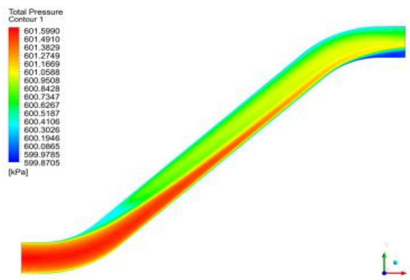


Fig -9: Total pressure contour plot

Variation of pressure gradient is more at bends as shown in figure 10.

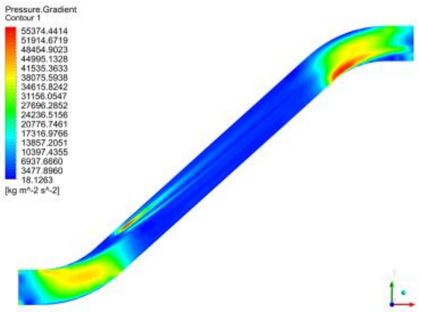


Fig -10: Pressure gradient contour

3.3 Secondary flow visualization

In this analysis the velocity contour has been captured at five locations as shown in figure 11. The flow visualizations are shown in figure 12 at the selected locations. At section 1, the flow is fully developed with uniform velocity contour. As the oil flow attains the bend radius of curvature, the secondary flow will develop at section 2, 3, 4 and 5 (shown in figure 12). Figure illustrates that the secondary flow is dominant due to radius of curvature.

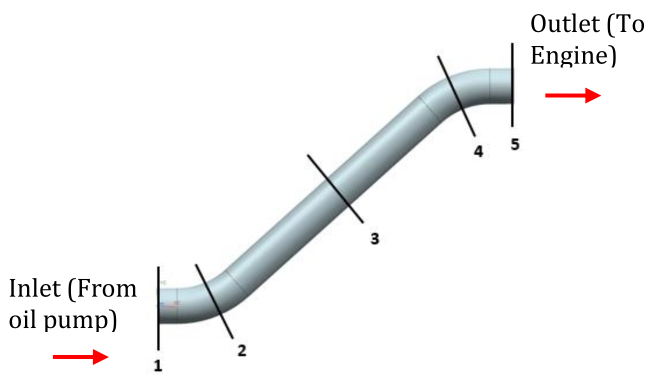


Fig -11: Locations

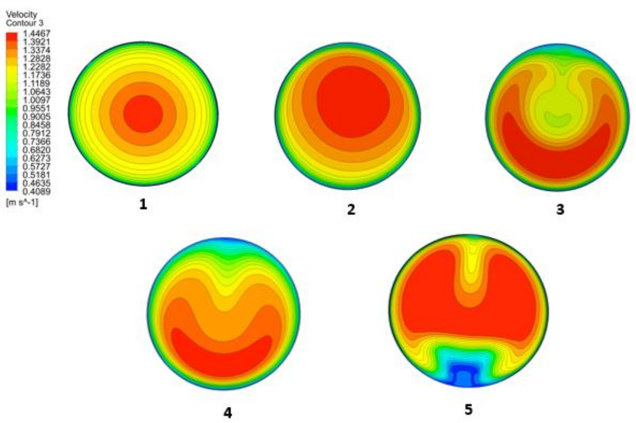


Fig -12: Velocity contour plot at 5 locations

The flow reversals occur in the boundary layer around the whole pipe as shown in figure 13. When the flow field is totally reversed, the secondary flow at the inlet to the first and second bend formed mainly Dean Vortices, see figure 14. The flow field is not symmetric at the exit of first and second bend, see figure 15. The Dean vortices in bend sections are not equal strength at all time and alternately dominate the secondary flow field. This Dean Vortices were simulated with the Isosurface of λ2=0.15 (shown in figure 15).

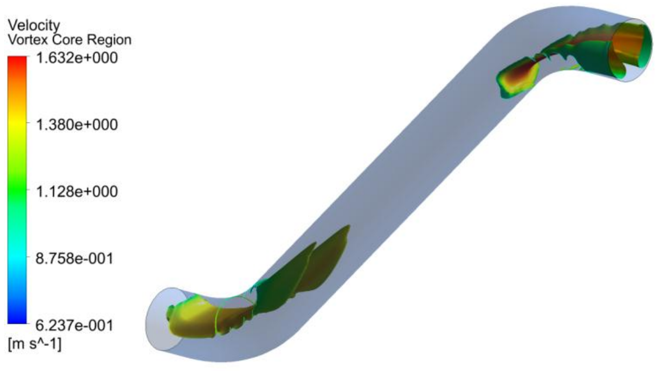


Fig -13: Velocity vortex plot

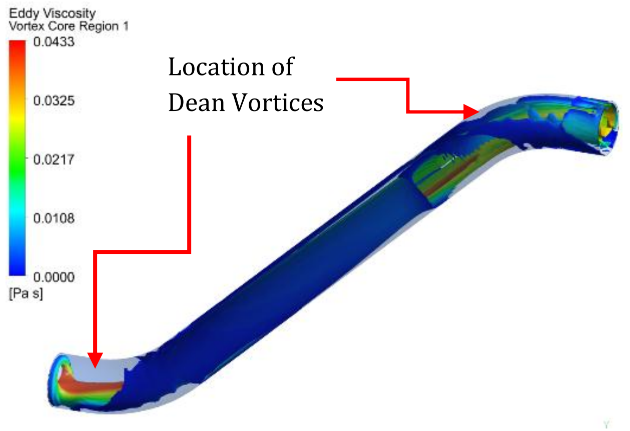


Fig -14: Eddy viscosity vortex core region plot

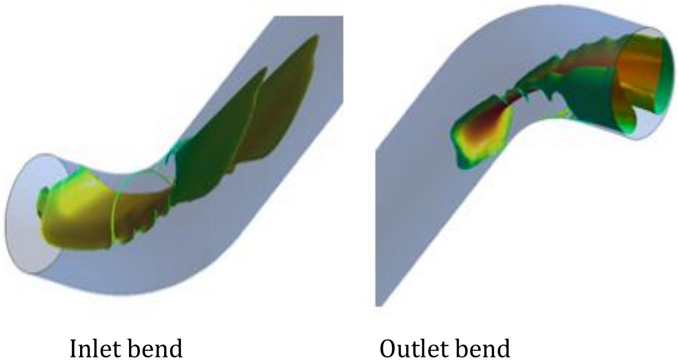


Fig -15: Isosurface of λ_2=0.15 at bend section

International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395 -0056
Volume: 04 Issue: 06 | June -2017 www.irjet.net p-ISSN: 2395-0072

Velocity streamline plot, figure 16 shows the development of secondary flow at curvature.

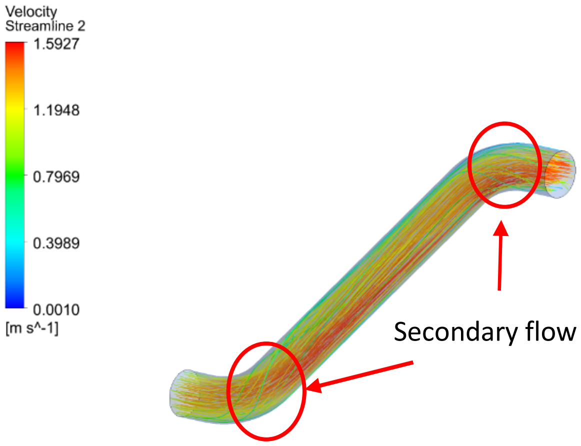


Fig -16: Velocity streamline plot

4. CONCLUSIONS

The review has considered for hot oil flow in aero engine pressure pipe.

- ANSYS CFX 14 has been utilized for the prediction of flow behavior in end bend pipe of inside diameter 14.3 mm.
- Mesh independent study has been achieved to decide on the optimum mesh size to be used in the simulation process.
- Velocity and pressure prediction has been observed along the length of pipe with inlet flow velocity of 1.45 m/sec at oil temperature of 110° C.
- A fully developed flow applied at pipe entrance region.
- The study shows the development secondary flow due to bend.
- Development of Dean Vortices shown. The solution is stable up to a critical Dean number ~ 956. Simulation shows the Dean Vortices are more at curvature region than straight portion. This shows that the Dean number is more than 956 at curvature.

REFERENCES

[1] D., Bhandari. and Dr. S., Singh., " Analysis of fully developed turbulent flow in a pipe using computational fluid dynamics", International Journal of Engineering Research & Technology (IJERT) Vol. 1 Issue 5, July - 2012 ISSN: 2278-0181.

[2] G,B,Nimadge. and S,V,Chopade.," CFD Analysis of flow through T-junction of pipe", International Research Journal of Engineering and Technology (IRJET), Volume:4 Issue:2,Feb-2017, e-ISSN: 2395 -0056, p-ISSN: 2395-0072.

[3] Mohammed, A, Al-Y.,*and Basel, A, S., "CFD Prediction of Stratified Oil-Water Flow in a Horizontal Pipe", Asian Transactions on Engineering (ATE ISSN: 2221-4267) Volume 01 Issue 05, Nov 2011.

[4] F,Hellstrom., and L,Fuchs., "Numerical computations of steady and unsteady flow in bended pipes", 37th AIAA Fluid Dynamics Conference and Exhibit 25 - 28 June 2007, Miami, FL,(AIAA 2007-4350).

[5] Adib, Z, Jaswar, K., and Yasser.,"M, A., CFD Simulation for Stratified Oil-Water Two-Phase Flow in a Horizontal Pipe", The 1st Conference on Ocean, Mechanical and Aerospace., 19, Nov 2014.

[6] Stratis, K., and Michael, F., "Simulation of the Air-Oil Mixture Flow in the Scavenge Pipe of an Aero Engine", Advances in Remote Sensing, Finite Differences and Information Security, ISBN: 978-1-61804-127-2.

[7] P. Bhramara, V. D. Rao, K. V. Sharma , and T. K. K. Reddy, "CFD Analysis of Two Phase Flow in a Horizontal Pipe – Prediction of Pressure Drop", World Academy of Science, Engineering and Technology International Journal of Mechanical, Aerospace, Industrial, Mechatronic and Manufacturing Engineering Vol:2, No:4, 2008

[8] Donald C. Rennels and Hobart M. Hudson "Pipe Flow: A Practical and Comprehensive Guide, First Edition", John Wiley & Sons, Inc. Published 2012 by John Wiley & Sons, Inc. © 2012

[9] B,R,Munson.,T,H,Okiishi.,W.,W.,Hue.,and A.,P.,Rothmayer. "Fundamental of fluid mechanics", John Wiley & Sons, Inc.7th edition

[10] K,A,Hoffmann., and S,T,Chiang., "Computational Fluid dynamics", Vol I,II,III.,A publication of Engineering Education system,Wichita ,Kansas,67208-1078,USA

BIOGRAPHIES

Manoj Kumar Mishra, has completed his post-graduation in Thermal power (Specialization- Aerospace propulsion) from Cranfield University, England, UK. He is working at Aero Engine research and Design Centre (AERDC), HAL. He has more than 10 yrs. of Experience in research field. His area of research is design, development, testing and validation of aero engine lubrication system and its components.