

Flow and Pressure Drop Characteristics for Various Non-Circular Curved Ducts Used in Automobile HVAC System with Different Bend Angles

Ashok Kumar M and Kannan Kanniah
General Motors Technical Center India

ABSTRACT

Non-circular ducts with strong curvature is commonly used in automobile HVAC system. The geometric cross section of the duct along with the bend angle plays a significant role in pressure drop. The present work aims at computationally analyzing the effect of eight different geometric cross-section ducts with different bend angles ranging from 0 degree to 135 degree on flow and pressure drop characteristics. A comparative study with different duct geometry and the bend angle is performed using fluent code. The effect of duct geometric cross section on secondary velocity and the total pressure loss coefficient for various duct geometries are discussed in detail. It is observed that the curvature of outer wall, slope of top wall and bottom wall plays a significant role in total pressure drop. It is found that the Rectangle-B and Triangle ducts are more sensitive to bend angle whereas Rectangle-A and Semicircle-A are least sensitive to bend angle in terms of total pressure drop.

KEYWORDS: Non-circular duct, curved ducts, pressure drop, CFD

CITATION: M, A. and Kanniah, K., "Flow and Pressure Drop Characteristics for Various Non-Circular Curved Ducts Used in Automobile HVAC System with Different Bend Angles," *SAE Int. J. Passeng. Cars - Mech. Syst.* 6(2):2013, doi: 10.4271/2013-01-1645.

INTRODUCTION

The automotive HVAC unit supplies air to the vehicle cabin through the panel duct, heater duct and the console duct systems. The primary objective of these ducts is to handle the required air flow rate with less pressure drop. However, due to the stringent packaging constraints, these ducts take various non-circular cross-sections with several bends. Flow through these curved ducts are more complex and highly three dimensional. It is well known that the reduction of the pressure drop will be beneficial to reduce the energy consumption and later the cost. Hence, understanding the flow and pressure drop characteristics for flow through non-circular ducts with various bend angles is of paramount practical significance. The study on curved duct began during the 1920s [1]. According to Berger [2], Dean was the first person to study the secondary flow using perturbation method on a Poiseuille flow for fully developed flow in curved pipe. Now such flow depends on a dimensionless number called the dean number, De . Dean number provides a measurement of the importance of inertial and centrifugal force relative to viscous force. Later, huge numbers of literatures are

published for flow through curved pipes. More recently Naphon and Wongwises [3] reviewed almost all the literatures available for flow and heat transfer characteristics of curved, helically coiled and spiral tubes. They re-presented most of the flow and heat transfer correlations published in open literatures for curved tubes. Several literatures are available for flow through curved square ducts as well till 1990s. Taylor and Whitelaw [4] conducted experimental investigation to determine the velocity and pressure distribution in a ninety degree square duct with laminar and turbulent flow. They reported that the secondary flow is more pronounced in laminar flow partly because of thicker boundary layers. They found that the secondary velocities are up to 0.6 and 0.4 of the bulk velocity in the laminar and turbulent regimes respectively. The pressure loss across the bend is about one fifth of the velocity head in the turbulent case. Mess et.al. [5] studied the flow instability in curved square duct experimentally and computationally. They explained the formation of two large Ekman vortices and two pairs of small Dean vortices near the outer wall that result from the primary instability that is of centrifugal nature. They reported that the simulation results obtained by solving

parabolized navier-stoke equations match well with experimental data for velocity profiles and secondary flow pattern. Wiedner and Camci [6] conducted experimental investigations to find the passage flow structure and its influence on end wall heat transfer for 90 degree bend square duct. With the help of pressure and velocity measurements they reported the total pressure loss in various planes normal to the flow direction. Limited literatures are available for flow through non-square curved ducts. Recently, Gauthier et. al. [7] conducted experimental investigations to describe the structure of basic flow for large radius of curvature of small aspect ratio rectangular duct with large radius of curvature with 180 degree bend. They stated that the centrifugal instability through curved duct generates stationary contra-rotating stream-wise vortices, which fill the whole thickness of the channel and develop threshold along both the radial and azimuthal directions. Bahrami et.al. [8] investigated the pressure drop characteristics of fully developed, laminar flow through straight mini and micro channels of arbitrary cross-section. They proposed a compact model that can predict the pressure drop for a wide variety of shapes which is validated well with experimental data collected by several researches. Geyer et.al. [9] studied fully developed flow and heat transfer behavior in periodic trapezoidal channels with a semi-circular cross-section. However they considered the flow to be laminar with Reynolds number ranging from 5 to 400. They studied the effect of channel shape from a zigzag to serpentine for enhanced heat transfer. They explained a new concept of area efficiency to stack the channels effectively. Oosthuizen [10] studied the secondary flow effects in a trapezoidal duct with 180 degree bend. However he used mini-channel trapezoidal duct used in electronics application with parallel top and bottom surface. They reported the variation of f/Re along the channel length and stated that the ratio bottom to top surface width have an important influence on variation of f/Re and the Nusselt number in the outlet channel. In addition to the above mentioned literature, few papers are available on the pressure drop characteristics of complex pipe geometries and ducts used in automobiles. Ghani and Aroussi [11] conducted experimental and computational investigation on air flow through complex HVAC splitter air duct with cross sectional variations and abrupt changes in air flow direction. With the help of PIV and CFD results, they identified the recirculation regions causing higher pressure drop. They added a splitter in the duct to reduce the recirculation region and hence the pressure drop and noise level. Fan [12] systematically studied the effect of mesh, boundary condition and the inlet box on pressure drop characteristics of line pipe. They clearly explained the influence of these parameters on pressure drop calculation using CFD. From the foregoing survey, it is noticed that, most of the literatures for flow through 90 degree square duct are confined to gas turbine application with higher radius curvature and the flow rate. However, due to packaging constraints, the HVAC ducts have less radius of curvature which may leads to separation. Other cross-section ducts such

as semi-circular and Trapezoidal ducts are limited to micro and mini channel flows. Hence, it is clear that no work is reported on a comparative study on effect of various non-circular duct geometric cross sections along with various bend angles for flow and pressure drop characteristics. The present paper addresses this issue which helps the HVAC duct designer for better understanding the effect of geometric cross section and bend angle on flow and pressure drop characteristics.

METHODOLOGY

Computational Model and Input Conditions

Totally about thirty two computational domains were created which consists of eight different standard cross-sections and four bend angles ranging from 0 to 135 degree. The duct with 0 degree (straight duct) is used as a base case while studying the effect of bend angle on total pressure drop. The computational models closely simulate the HVAC ducts used in automobiles. Although the actual HVAC ducts might not be of exact standard geometric shape with sharp corners, usage of standard shapes being generic in nature is considered to be the best model for the purpose of characterizing the geometric parameters. The cross-section area is kept same for all the geometries tested. All the parameters such as the duct geometry, cross-section area, bend angles and the flow rate are deduced from typical HVAC ducts used in automobiles. A computational mesh suitable for finite volume method is generated. Structured mesh is generated for the computational domain which is composed primarily of hexahedral elements but includes coopered wedge elements where appropriate. In order to facilitate the possibility of using low Reynolds number form of SST $k-\omega$ turbulence model, fine clustered mesh is generated near the solid walls. The first node is placed at a distance of 0.05mm from the wall surface which gives a wall $y^+ \sim 1$. The low Reynolds number model integrates the governing equations up to the viscous sub-layer and hence the first computational cell must have its centroid in $y^+ \sim 1$ [13]. An optimum mesh size consisting of two million cells is selected after doing grid independence study. For each case, it took about three hours to get converged result using 4 CPU, 24GB RAM work station. A typical computational mesh and its y^+ is shown in [Fig. 1](#) and [2](#) respectively. For the purpose of computations, a specified mass flow condition is imposed at the inlet. A turbulence intensity of 1% is chosen at the inlet fluid. Constant pressure outlet condition is applied on the duct outlet boundaries.

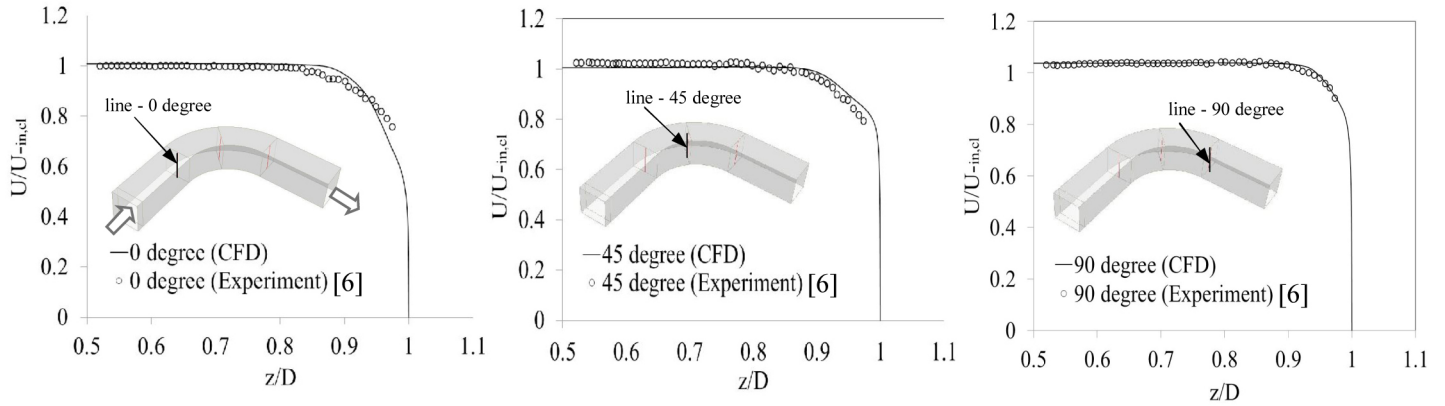


Fig. 3. Mean velocity profiles, U/U_{inlet}

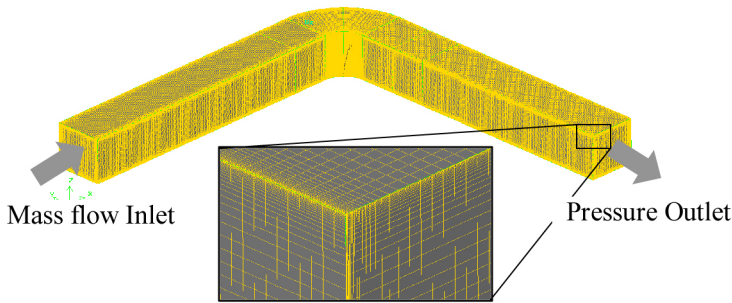


Fig. 1. Computational mesh with input condition for 90 degree curved square duct

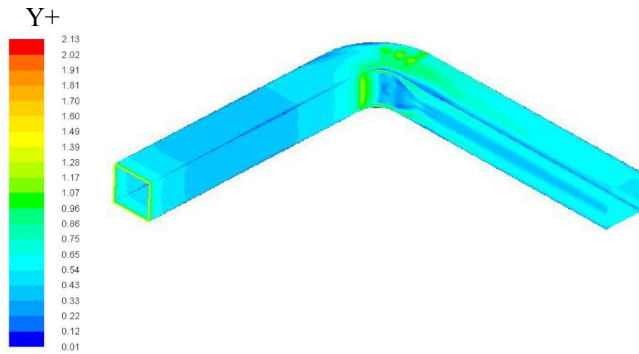


Fig. 2. Contour of wall Y^+ for 90 degree square duct

Numerical Solution

A finite volume based solver Fluent-13 is used for solving the governing continuity, momentum and turbulence model equations (equations 1,2,3,4). SST $k-\omega$ model [14, 15] is used for turbulence closure, since it well predicts the flow with strong pressure gradient and separation [16]. Flow is considered incompressible and constant properties are used. The solution is considered as converged when the residuals fall below 10^{-6} for momentum, continuity and turbulence equations. Change in average total pressure in the measurement planes are continuously monitored so that there

will be no change in the value for consecutive 500 iterations after the solution is converged. In order to find the validity of using the SST $k-\omega$ turbulence model for predicting the flow characteristics in curved ducts, a case with 90 degree duct with square cross-section is selected. The computational model selected for validating the present numerical methodology mimics the experimental set-up of Wiedner and Camci [6] so that the computational results can be compared with the experimental data [6]. The boundary conditions such as the inlet velocity profile and turbulence intensities were applied from the experimental data [6]. The mean velocity profiles and the total pressure loss coefficient with SST $k-\omega$ turbulence model in the plane 0 deg, 45 deg and 90 deg plane shows that there is a good agreement between the computational results and the experimental data[6], refer Fig. 3 and 4. Hence the present computational methodology with SST $k-\omega$ turbulence model can be well taken as validated for the curved ducts.

Continuity equation:

$$\nabla \cdot (\rho u) = 0 \quad (1)$$

Momentum equation:

$$(u \cdot \nabla)u - \frac{1}{\rho} \nabla P + \nu \nabla^2 u = 0 \quad (2)$$

Turbulent kinetic energy equation:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left(\Gamma_k \frac{\partial k}{\partial x_j} \right) + G_k - Y_k + S_k \quad (3)$$

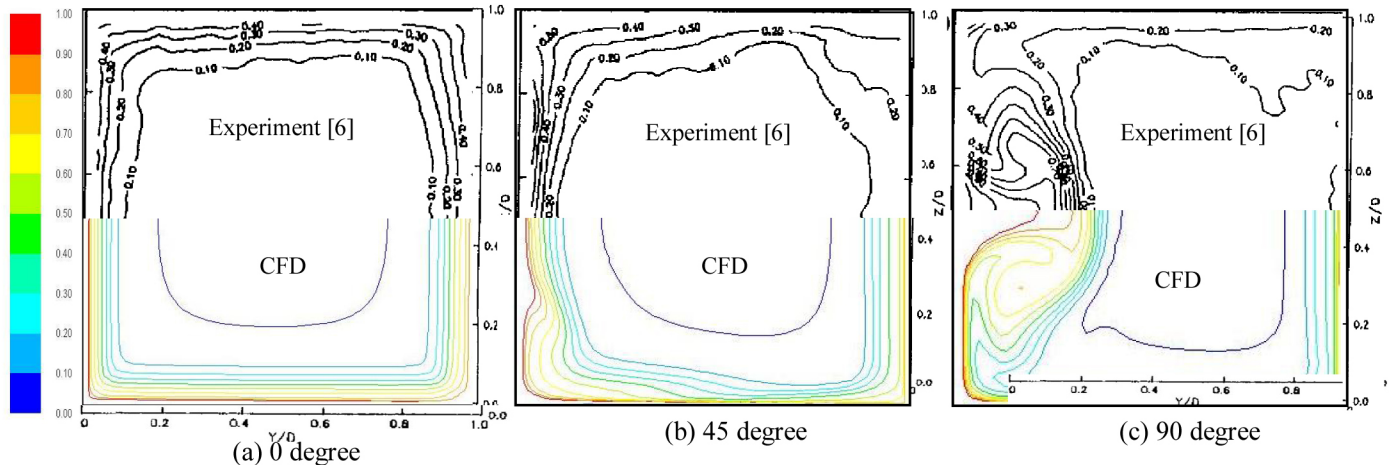
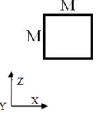


Fig. 4. Total pressure loss coefficient, $C_{po} = (P_{o,in,cl} - P_o) / 0.5\rho U_{in,cl}^2$

Table 1. various duct cross-sections considered for the present study

Shapes	Values (mm)	Shapes	Values (mm)	Shapes	Values (mm)
Square 	M = 70.7	Semicircle-A 	K = 112.8	Trapezoidal-B 	G = 84.8 F = 56.5 J = 70.7
Rectangle-A 	N = 100 Q = 50	Semicircle-B 	K = 112.8	Triangle 	E = 107.4
Rectangle-B 	N = 100 Q = 50	Trapezoidal-A 	G = 84.8 F = 56.5 J = 70.7		

Specific dissipation rate equation:

$$\frac{\partial}{\partial t}(\rho\omega) + \frac{\partial}{\partial x_i}(\rho\omega u_i) = \frac{\partial}{\partial x_j} \left(\Gamma\omega \frac{\partial \omega}{\partial x_j} \right) + G\omega - Y\omega + D\omega + S\omega$$

$$\Gamma\kappa = \mu + \frac{\mu_t}{\sigma_\kappa}, \Gamma\omega = \mu + \frac{\mu_t}{\sigma_\omega}, \sigma_{\kappa,1} = 1.176, \sigma_{\omega,1} = 2.0, \sigma_{\kappa,2} = 1.0, \sigma_{\omega,2} = 1.168$$

(4)

PARAMETERS INVESTIGATED

As mentioned, totally about thirty two computational domains are created which consists of eight different duct geometries with four bend angles ranging from 0 deg to 135 deg. Table 1 shows the various geometric cross-section of the ducts considered for the present study. The cross section area and the mass flow rate (inlet velocity = 5 m/s) is maintained

same value for all the ducts. The bend angles along with the measurement planes considered for the present simulation are tabulated in Table 2. Though the bend angles varies, the radius of curvature is maintained same value i.e. R = 70 mm.

DATA REDUCTION

Numerical results are obtained after solving the governing equations. The results are presented in non-dimensional form. The pertinent dimensionless parameters appearing in the problem are: Static pressure coefficient and Total pressure loss coefficient.

Static pressure coefficient (C_{ps}) = $(P_{s,in} - P_s) / (0.5 \cdot \rho \cdot U_{in}^2)$, where, P_s = Static pressure

Total pressure loss coefficient (C_{po}) = $(P_{o,in} - P_o) / (0.5 \cdot \rho \cdot U_{in}^2)$, where, P_o = Total pressure.