The effect of geometric parameters on the head loss factor in headers

A. Mansourpour & S. Shayamehr
Mechanical Engineering Department, Azad University of Karaj, Iran

Abstract

Head loss factor *k* of many hydraulic components is available in handbooks but for complex shapes such as headers, there is no standard value. The purpose of this study is to help hydraulic designers to predict a value for the head loss factor in headers based on their geometric parameters. The analysis is carried out solving the Navier-Stokes equations by means of numerical methods. In this study, one parameter is changed and all the others are fixed to predict the effect of that specific parameter on the head loss factor. Results show that:

1- Increasing the chamber diameter increases the head loss factor *k*.

2- Increasing the chamber length increases the head loss factor *k* until it reaches its critical value and then remains fixed.

3- Increasing the outlet diameter decreases *k*.

Keywords: k factor, header, head loss, geometric parameters, Navier-Stokes.

1 Introduction

Nowadays in many pump stations or other utility installations, headers are seen as multiple-water-inlet collectors. Strength and stability design of these kinds of hydraulic devices are mentioned in many handbooks and have a straightforward procedure, but in the field of fluid mechanics and the behavior of fluids inside headers, there are few available studies. Since pressure drop measurement is quite poor in theory, it relies on experimental methods in most cases. Pump sizing and the challenge in selecting the most economical system lies in determining the total head loss. There are a number of important factors in the analysis that need to be studied thoroughly in order to satisfy the requirements for minimum external pressure on fluid-moving equipment. Friction factor, the

WIT Transactions on The Built Environment, Vol 105, © 2009 WIT Press
www.witpress.com, ISSN 1743-3509 (on-line)
doi:10.2495/FSI090331

diameter, the length of components such as valves, fittings and bends are among these factors.

A substantial amount of literature, based on theoretical and experimental methods is available for predicting pressure losses. For example, Darcy-Weisbach equation yields frictional head loss in pipes [1]. There are also some implicit and explicit equations resulting friction factor f required for Darcy-Weisbach equation. Calculations for the other kind of pressure drop – referred to as local head loss – rely more on experimentation [2]. Numerical methods are used to find a value for the loss factor k in the local head loss equation.

The aim of this study is to provide a theoretical overview of previous studies and also to reach adequate correlations through experimentation to predict head loss in headers by means of numerical methods and through different models for turbulent flows.

2 Theoretical overview

2.1 Frictional pipe head loss

There are many equations and correlations which take the geometry of pipes into account for the analysis of head loss. Darcy-Weisbach equation [1] correlates the geometric parameters of the pipe and the velocity of the fluid flow to calculate pressure drop, eqn (1). It is valid for duct flows of any cross section and for both laminar (a smooth and steady flow with Re<2300) and turbulent (a fluctuating and agitated flow, full of sudden changes with Re>2300).

$$h = f\left(\frac{L}{d}\right)\left(\frac{V^2}{2}\right) \tag{1}$$

L is the length of the pipe, d is the diameter of the pipe, f is the friction factor and V is the velocity of the flow. The only problem is to find the form of the function f. In 1939 to cover the transitionally rough range, Colebrook combined the Prandtl's smooth-wall formula, eqn (2), and fully rough formula, eqn (3), into a clever interpolation formula, eqn (4), [1]:

$$\frac{1}{f^{\frac{1}{2}}} = 2.0\log\left(\mathrm{Re}_d f^{\frac{1}{2}}\right) - 0.8 \tag{2}$$

$$\frac{1}{f^{\frac{1}{2}}} = -2.0\log\frac{\frac{\varepsilon}{d}}{3.7} \tag{3}$$

$$\frac{1}{f^{\frac{1}{2}}} = -2.0\log\left(\frac{\frac{\varepsilon}{d}}{3.7} + \frac{2.51}{\mathrm{Re}_d f^{\frac{1}{2}}}\right) \tag{4}$$

where ε is the roughness, d is the length and Re_d is the Reynolds number.

This is the accepted design formula for turbulent friction. It was plotted in 1944 by Moody into what is now called the Moody Chart for pipe friction [1].

2.2 Local and minor losses

A local loss is any energy loss, in addition to that of pipe friction alone, caused by some localized disruption of the flow by some flow appurtenances, such as valves, bends, and other fittings. The actual dissipation of this energy occurs over a finite but not necessarily short longitudinal section of the pipe line, but it is an accepted convention in hydraulics to lump the entire amount of this loss at the location of the device that causes the flow disruption and loss. If a loss is sufficiently small in comparison with other energy losses and with pipe friction, it may be regarded as a minor loss. Often minor losses are neglected in preliminary studies or when they are known to be quite small, as will often happen when the pipes are very long. However, some local losses can be so large or significant that they will never be termed a minor loss, and they must be retained; one example is a valve that is only partly open [2].

Normally, theory alone is unable to quantify the magnitudes of the energy losses caused by these devices, so the representation of these losses depends heavily upon experimental data. Local losses are usually computed from the eqn (5).

$$h_l = k_l\left(\frac{V^2}{2g}\right) \tag{5}$$

in which $V = \dfrac{Q}{A}$ is normally the downstream mean velocity, k_l is the loss factor, g is the gravity acceleration [2].

2.3 Numerical analysis

One of the vitally needed skills to assist engineers dealing with complicated problems is the understanding of numerical methods and another thing is the ability to implement them on a computer. Numerical analysis is utilized to ascertain the consequence of various ideas quantitatively and to provide computations that are not practical by hand. Collecting data through experimentation depends heavily on using a corresponding model representing the specifications of fluid flow and physical conditions.

A number of models that are used to simulate turbulent flows in a pipe are presented in the following sections.

2.3.1 K-epsilon model

The K-epsilon is one of the most common two-equation models which was introduced by Harlow and Nakayama [3, 4] in 1968 for turbulent flows. These equations are shown below:

$$k_t = \alpha\left(\frac{k^2.k_x}{\varepsilon}\right) - \varepsilon \tag{6}$$

$$\varepsilon_t = \beta\left(\frac{k^2.\varepsilon_x}{\varepsilon}\right) - \left(\frac{\gamma.\varepsilon^2}{k}\right) \tag{7}$$

where $k = k(x, t)$ is the turbulent kinetic energy, $\varepsilon = \varepsilon(x, t)$ is the rate of dissipation of the turbulent energy, and α, β, and γ are positive constants.

2.3.2 K-omega model

Another two-equation model used widely for turbulence is k-ω. The two transported equations represent the turbulent properties of the flow. k and ω indicate turbulent kinetic energy and the specific dissipation respectively. There are two commonly used k-ω models; Wilcox's and SST.

The SST k-ω turbulence model [5] is a two-equation eddy-viscosity model which has become very popular. The use of a k-ω formulation in the inner parts of the boundary layer makes the model directly usable all the way down to the wall through the viscous sub-layer; hence the SST k-ω model can be used as a low-turbulence model without any extra damping functions. The SST formulation also switches to k-ε behaviour in the free-stream and thereby avoids the common k-ω problem that the model is too sensitive to the inlet free-stream turbulence properties. Authors who use the SST k-ω model often merit it for its good behaviour in adverse pressure gradients and separating flow. The SST k-ω model does produce a bit too large turbulence levels in regions with large normal strain, like stagnation regions and regions with strong acceleration. This tendency is much less pronounced than with a normal k-ε model though [10]. The equations required for this model are available in [5, 10]. Another commonly used K-omega model is Wilcox's. The equations associated with this model are discussed in [11].

2.3.3 Spalart-Allamaras model

Spalart-Allmaras model is a one-equation model which solves a transport equation for a viscosity-like variable $\widetilde{\nu}$ [7, 8]. This may be referred to as the Spalart-Allmaras variable. A modification of the model was proposed in 1995 [6] which also accounts for the effect of mean strain rate on turbulence production.

One of the main advantages of S-A model compared to k-ε is the simplicity in imposing the free-stream and wall boundary conditions [9]. In a near wall region, the model depends on the distance to the wall in order to reproduce the viscous effects in the laminar sub-layer. Far from the wall, the viscosity production term becomes negligible.

3 Modeling

An initial model was developed using a geometry design application with the capability of exporting data to a solver application. The figure is a cylindrical header containing two inlets and one outlet fig.1. The inlet and outlet diameters were taken to be the same by default, and the chamber diameter twice as large as outlet and inlet diameters as well. After dimensioning the geometry, it was exported to the solver application in order to be numerically solved using an iterative method. The experiments were carried out on water with the density of $998.2\left(\frac{kg}{m^3}\right)$ and the temperature of $300\left(°K\right)$.

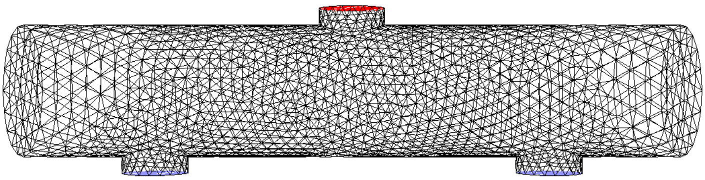

Figure 1: 3D view of the meshed geometry.

Figure 2: Contours of pressure for the initial geometry with equal inlet and outlet diameter.

A number of tests were conducted considering every effective parameter on the fluid behaviour. Individual parameters modifying in each test included the outlet diameter, the chamber length and the chamber diameter. The tests were carried out for each parameter separately and the desired data including the inlet pressure, the outlet pressure, and the outlet velocity were recorded to calculate k factor using the eqn (8)

$$K = \frac{2\Delta P}{\rho V^2} \tag{8}$$

After defining boundary conditions in the solver application, the tests were carried out solving Navier-Stokes equations so that a simulation of the fluid behaviour in the header could be received. Pressure distribution for the preliminary geometric figure is shown in fig.2. The collected diagrams were studied carefully and the required data were gathered. The information was collected for the fluid behaviour in the centerline of the pipe.

Table 1: Head loss factor vs outlet diameter for k-epsilon model.

Outlet diameter(m)	K
2	0.945
2.2	0.761
2.6	0.718
2.9	0.707
3.2	0.701
3.5	0.704

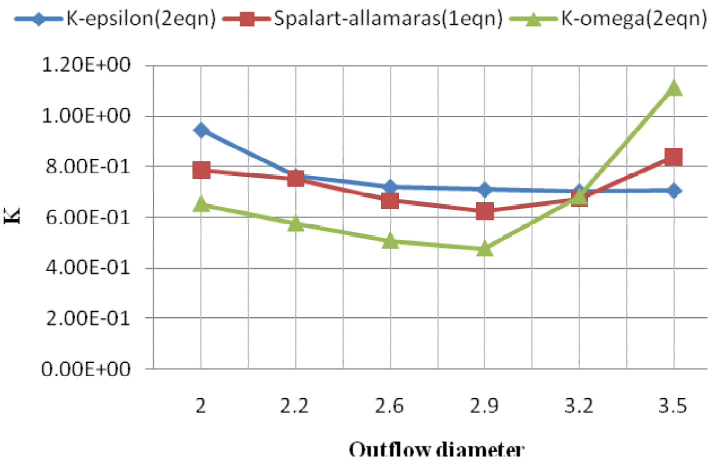

Figure 3: Head loss factor vs outflow diameter.

Table 2: Head loss factor vs chamber length for k-epsilon model.

Chamber length (m.)	K
16	0.453
17	0.535
18	0.658
19	0.789
20	0.945
21	0.971
22	0.964
23	0.962

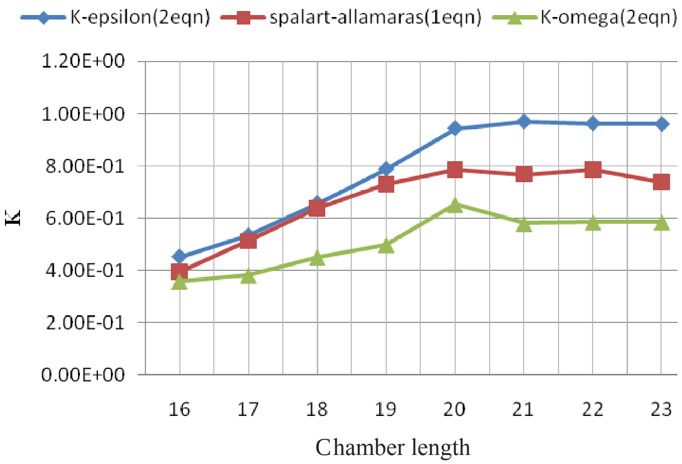

Figure 4: Head loss factor vs chamber length.

Table 3: Head loss factor vs chamber diameter for k-epsilon model.

Chamber diameter(m)	K
3.8	0.889
4	0.945
4.2	1.07
4.4	1.25
4.6	1.49

4 Results and discussion

Data collected while increasing the outlet diameter proved that there would be a gradual decrease in *k* factor of the head loss according to k-epsilon model. For the outlet diameter expansion, as it is expected, there would be a linear decrease in the velocity of the flow considering continuity equation for incompressible flows which means the sum of the transformed energy from hydraulic grade line(HGL) to energy grade line (EGL) decreases – pressure loss reduction. Figures resulted from k-epsilon model (fig. 3), table 1 best confirm the argument, partly due to conducting the experiments for fully developed turbulent flows.

Chamber geometric parameters - length and diameter - were noticed to be as important to decide for the total pressure drop of the header. A fairly steady rise for k was recorded when increasing chamber diameter using k-omega model fig.5. The results for k-epsilon table 3 and Spalart-Allmaras models showed a more significant increase especially for bigger chamber diameters. More pressure loss is the consequence of the added distance, the fluid covers. Similarly, increasing chamber length leads to growth of k, as it is shown in fig.4, table 2. But as the chamber gets bigger in length, the curve becomes stable which means, beyond a certain limit, k factor appears to be independent of the chamber length. The reason partly lies in the impact of vortices on pressure loss which is more noticeable for smaller chambers.

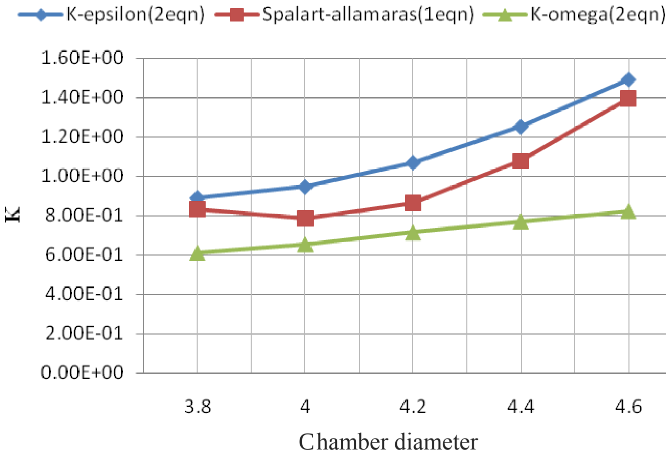

Figure 5: Head loss factor vs chamber diameter.

5 Conclusion

Head loss calculation in complex shapes such as headers depends on accurate modeling and is computed by numerical analysis, solving Navier-Stokes equations using an iterative method because it is insoluble by theory or by reference to standard empirical data. The parameters affecting k factor of head loss in headers were proved to be the outlet diameter, the chamber length and the chamber diameter. K-epsilon model holds better precision and provides more accurate data for turbulent flows in comparison with k-omega and Spalart-Allmaras models.

References

[1] White, F.M., *Fluid mechanics*, McGraw-Hill, Forth edition, pp. 340&348.

[2] Larock, B.E., *Hydraulics of pipeline systems*, CRC press , (2.2.5) , 2000.

[3] Scott-Pomerantz, C.D., *The K-epsilon model in the theory of turbulence*, University of Pittsburgh, pp. 1 , 2004.

[4] Harlow, F.H. & Nakayama, P.I., *Transport of turbulence energy decay rate*, Los Alamos Sci. Lab., LA-3854, 1968.

[5] Menter, F. R., *Zonal Two Equation k-ω Turbulence Models for Aerodynamic Flows*, AIAA Paper 93-2906, 1993.

[6] Dacles-Mariani, J., Zilliac, G. G., Chow, J. S. and Bradshaw, P., *Numerical/Experimental Study of a Wingtip Vortex in the Near Field*, AIAA Journal, 33(9), pp. 1561-1568, 1995.

[7] Spalart, P. R. and Allmaras, S. R., *A One-Equation Turbulence Model for Aerodynamic Flows*, AIAA Paper 92-0439 , 1992.

[8] Spalart, P. R. and Allmaras, S. R., A *One-Equation Turbulence Model for Aerodynamic Flows*, La Recherche Aerospatiale n 1, 5-21, 1994.

[9] Lorin, E, *An accurate positivity preserving scheme for the Spalart-Allmaras turbulence model*, page 3 , 2003.

[10] Menter, F. R., *Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications*, AIAA Journal, vol. 32, pp. 269-289, 1994.

[11] Wilcox, D.C., *Re-assessment of the scale-determining equation for advanced turbulence models*, AIAA Journal, vol. 26, pp. 1414-1421, 1988.