Proc. CIBSE A: Building Serv. Eng. Res. Technol. **21**(3) 175–178 (2000) Printed in Great Britain B576

Summary The effects on pressure loss of the separation and orientation of closely coupled duct fittings in HVAC systems were investigated using computational fluid dynamics to analyse the pressure distribution in a system containing two 90° bends in two common configurations; an S bend and a U bend. Fittings that are separated by less than 8 to 10 hydraulic diameters of the duct behaved in very different ways depending upon the orientation of the fittings in relation to one another. Further in-depth analysis is required to produce accurate and comprehensive pressure loss coefficient data, and thus improve HVAC systems design.

Effect on pressure loss of separation and orientation of closely coupled HVAC duct fittings

S M Atkin BEng and **Li Shao** BEng PhD CEng MCIBSE
Institute of Building Technology, School of the Built Environment, University of Nottingham University Park, Nottingham, NG7 2RD, UK

Received 27 May 1999, in final form 13 July 2000

List of symbols

ζ Pressure loss factor
p Pressure (Pa)
ρ Density (kg m^{-3})
v Velocity (m s^{-1})
D_h Hydraulic diameter (m)
l Length (m)

1 Introduction

Indoor air quality, thermal comfort level and energy efficiency of buildings are becoming increasingly dependent upon the performance of HVAC systems. The correct sizing of HVAC ductwork and plant relies on the accurate determination of pressure loss throughout the system, which is based on calculations incorporating pressure loss factors (ζ) taken from a variety of sources including the *CIBSE Guide* Volume C[1] and the *ASHRAE Handbook, Fundamentals*[2]. However, previous research has shown that these data are inaccurate and do not cover the wide range of fittings currently available for use in HVAC systems. Such inaccurate sizing will lead to inefficient energy use and poor air quality, with consequent effects on productivity and human health. One possible contribution to these inaccuracies that could be of some significance is the assumption that the pressure loss in a duct fitting is not affected by its proximity to other fittings. However, while this has been investigated in the past[3], no-one has studied the effect on pressure loss in ventilation ducts caused by varying the configuration in which the fittings are connected. The purpose of this study was to examine this phenomenon and determine the degree to which it affects the accuracy of pressure loss factors, so as to provide designers with information on how the separation of fittings affects their designs and to enable them to have more confidence in the viability of those designs. This will be achieved by the use of the computational fluid dynamics (CFD) software FLUENT.

2 Background

The determination of pressure loss factors has been researched for many years because of its importance in HVAC design. However, the data that have been published by both CIBSE and ASHRAE are in considerable disagreement. Such unreliability stems from the failure to account for all the parameters involved when gathering experimental results regarding the pressure loss in duct fittings. The most significant problem is the difficulty associated with defining with sufficient precision the pressure loss due to duct fittings themselves. Shao and Riffat[4,5] have shown that CFD can be used to determine pressure loss factors and hence predict pressure loss, and to a good degree of accuracy provided that a particular set of computational methods are used. Riffat and Gan[3] have supported this using experimental techniques for comparison.

Reynolds number is one of the many factors that affect the pressure loss coefficient of ducts, although it is possible to make this effect negligible by using greater inlet velocities, e.g. 10 m s^{-1} for a duct of 0.5 m diameter[5]. However, this kind of velocity is usually found only in main supply ducts of large ventilation systems because of the requirements for low velocity in branch ducts so that noise levels are kept to a minimum. Low velocity is also of great importance in natural ventilation ducts, which behave in the same way as smaller ventilation ductwork and have the same pressure loss factors as for smaller ducts of similar geometry. Hence, to obtain a set of results that simulate real-world situations, it will be necessary to model the flow at a velocity commonly found in HVAC systems.

The pressure loss factor of a bend can be calculated using the formula

$$\zeta = \frac{\Delta p}{\frac{1}{2}\rho v^2} \qquad (1)$$

where $\frac{1}{2}\rho v^2$ is the dynamic pressure at the duct inlet based on air with a density of 1.2 kg m^{-3} at 20°C and 101 325 Pa (standard atmospheric conditions), and travelling with a velocity of v (m s^{-1}).

However, for two duct fittings in series, the value obtained from equation 1 will include not only the pressure loss due to the bends but also that generated as a result of friction in the spacer (straight duct section separating the fittings). The pressure loss for the bends can therefore be calculated by taking this 'system' pressure loss and subtracting the pressure loss due to friction in the spacer as follows:

$$\Delta p_B = \Delta p_S - \Delta p_F \qquad (2)$$

This value can then be used with equation 1 to determine the pressure loss factor of the bends alone.

3 Description of computation

The computational simulations were carried out using the commercial CFD software FLUENT (Fluent Inc.). The simulation was treated as a three-dimensional problem using body-fitted coordinate grid systems generated using the software package PreBFC. This ensures that the curved walls of the bends are modelled accurately, whereas use of the conventional Cartesian coordinate system offered by FLUENT would result in curves being modelled as a series of short straight lines. This could produce unacceptably large errors owing to the pressure loss being dependent upon the surface roughness of the duct.

The system itself was modelled using long sections of duct both upstream and downstream of the bends to ensure that the flow becomes fully developed prior to entering the first bend, and to allow it to re-establish a fully developed profile after leaving the second bend. It is been assumed that the direction of flow is normal to the inlet cross section with a uniform velocity distribution at the duct entrance at a typical HVAC duct velocity of 5 m s^{-1}.

Shao and Riffat[5] demonstrated that the most accurate results are obtained using the k–ε turbulence model, in both its standard form and that based upon the RNG (renormalisation group) theory. Of these two, the standard model gives the slightly better results. However, the RNG model is a fundamentally more plausible turbulence model, as it does not include any of the empirical data required in the standard k–ε turbulence model. This diminishes the grid dependency of the system under the RNG model, and this model was used to carry out this series of simulations.

The duct system simulated was of square cross section and included two 90° bends separated by spacers of varying length from 0 to 3.0 m. The hydraulic diameter of the duct, D_h, was 0.3 m and the radius of the bend, measured with reference to the centre line of the duct, was equal to twice this value. Six simulations were run at spacer length intervals of 0.6 m. This set of tests was carried out for two different configurations of bends, a U bend and an S bend, to determine whether the bend configuration has any effect upon the pressure loss in the system, since previous work has concentrated solely upon the U bend configuration and assumed that the pressure losses are independent of this factor. Figure 1 illustrates the two bend configurations simulated.

Both configurations are symmetrical with respect to a plane defined by the curved axis of the bends, with grid lines being either parallel or perpendicular to the duct axis. Figure 2 shows an example of this grid set-up in the symmetry plane. The grid density in the duct axis direction has been doubled through the bends themselves to ensure accurate simulation of the flow in this region. This is illustrated in Figure 3, which shows the grid set-up for the walls of a typical bend, cut through the symmetry plane.

4 Results and discussion

Figure 4 shows a typical static pressure distribution taken along the duct centre line. It is possible to determine the pressure loss

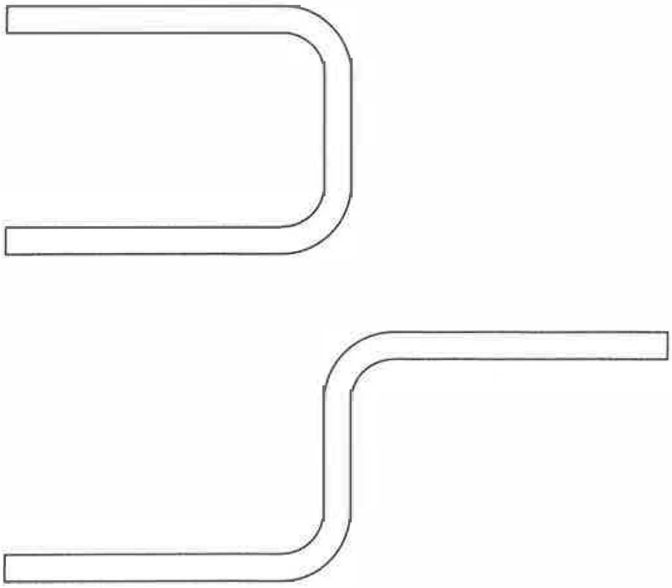

Figure 1 Bend configurations used for simulation

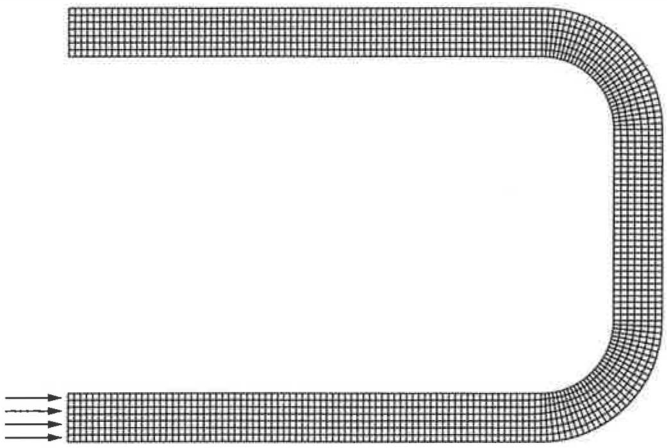

Figure 2 An example of grid set-up

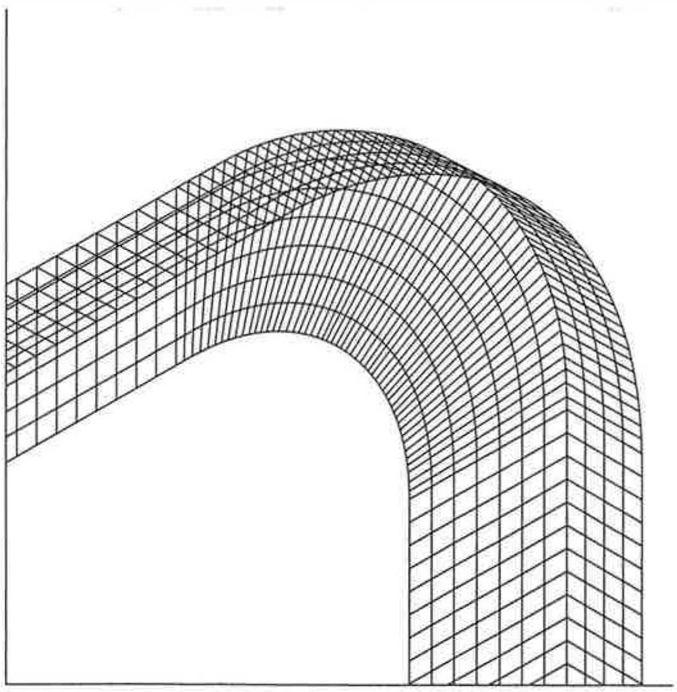

Figure 3 An example of a grid set-up in a typical bend

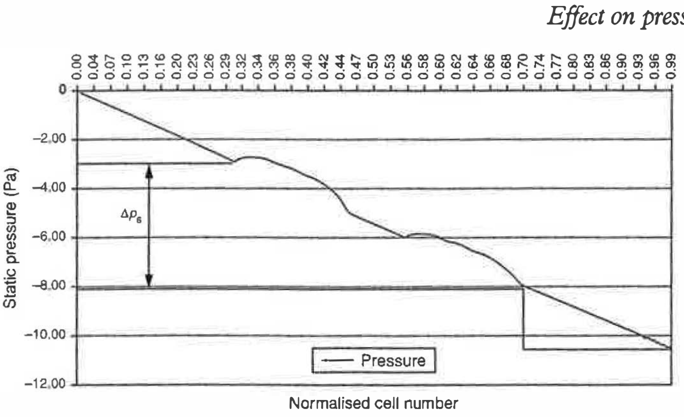

Figure 4 Typical static pressure distribution along a duct centre line; S bend case with 1.2 m separation

across the system due to the bends by reading the pressure difference between the point where the first bend begins to affect the pressure loss and that where the flow has recovered a steady rate of pressure loss after the second bend. Equation 2 can then be used to determine the actual pressure loss due solely to the bends, enabling the system pressure loss factor, ζ_s, to be calculated using equation 1. The values obtained are given in Table 1.

These results reveal an interesting trend, in that the pressure loss in the U bend is highest at a bend separation of zero and decreases as the separation of the bends is increased. However, pressure loss in the S bend increases with separation. This can be seen more clearly in Figures 5 and 6. The values of the pressure loss in the two different bend configurations converge at a separation somewhere between 8 and 10 hydraulic diameters,

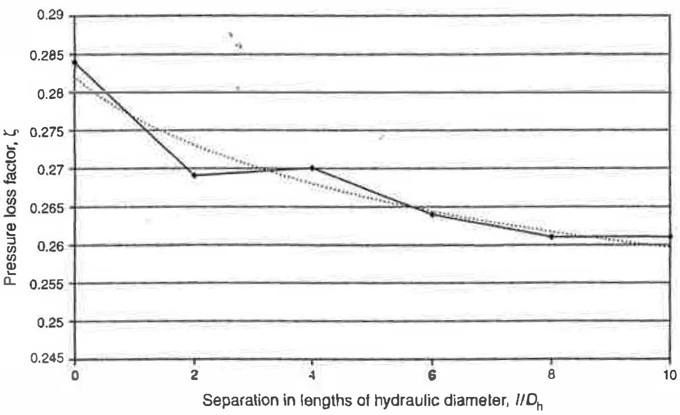

Figure 5 Variation of ζ in a U bend configuration

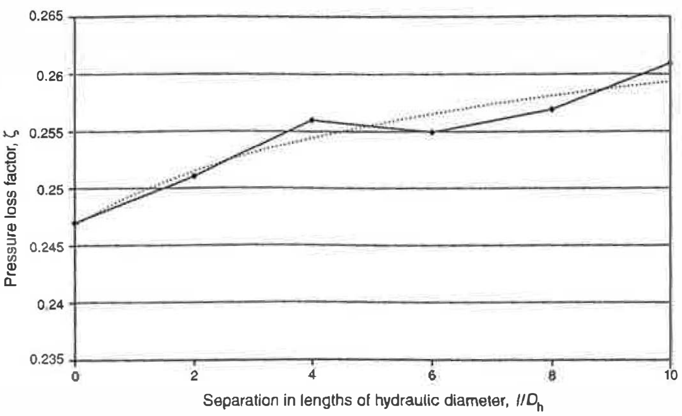

Figure 6 Variation of ζ in an S bend configuration

Table 1 Predicted pressure loss and pressure loss factor for two bend duct systems of different configuration

Separation between fittings (m)	Δp_S (Pa)	Δp_F (Pa)	Δp_B (Pa)	ζ_S	ζ_B
U bend					
0.0	4.26	0.00	4.26	0.284	0.284
0.6	4.63	0.60	4.03	0.309	0.269
1.2	5.25	1.20	4.05	0.350	0.270
1.8	5.76	1.80	3.96	0.384	0.264
2.4	6.32	2.40	3.92	0.421	0.261
3.0	6.92	3.00	3.92	0.461	0.261
S bend					
0.0	3.71	0.00	3.71	0.247	0.247
0.6	4.36	0.60	3.76	0.291	0.251
1.2	5.04	1.20	3.84	0.336	0.256
1.8	5.62	1.80	3.82	0.375	0.255
2.4	6.26	2.40	3.86	0.417	0.257
3.0	6.92	3.00	3.92	0.461	0.261

and it is at this point that the first bend ceases to have any effect on the flow through the second bend.

Examination of flow patterns showed that the differing pressure losses had probably been generated as a result of the flow detaching from the duct wall as it rounded the bend, as shown in Figure 7, and causing a greater or lesser degree of interference depending on which way it was redirected by the second bend.

A fairly simple analysis offers a clearer way to understand why the pressure loss distributions for the different bend configurations are so completely different. The key to this analysis is to regard the double-bend assembly as being 'developed' from a straight duct, as shown in Figures 8 and 9. In each figure the bends are separated farther from each other from (*a*) through to (*d*).

Figure 8 shows the development of an S bend from its simplest case, that of a straight duct with no changes in the flow direction. This results in the minimum possible pressure loss owing to the lack of a fitting loss. The system is then developed through a system with a small 'kink' in the duct to the cases modelled in this simulation with two smooth 90° bends and a spacer of length increasing from 0 to 3 m. At each stage of the development, the pressure loss increases until reaching a stable value when the separation between the bends exceeds 8 to 10 hydraulic diameters of the duct. Figure 8

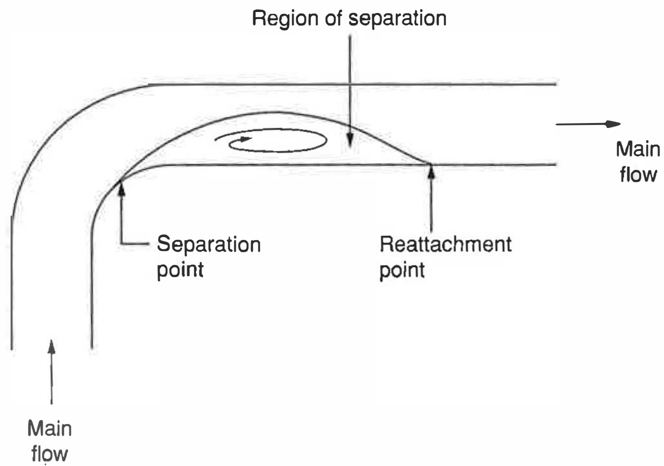

Figure 7 Detachment of flow from duct wall

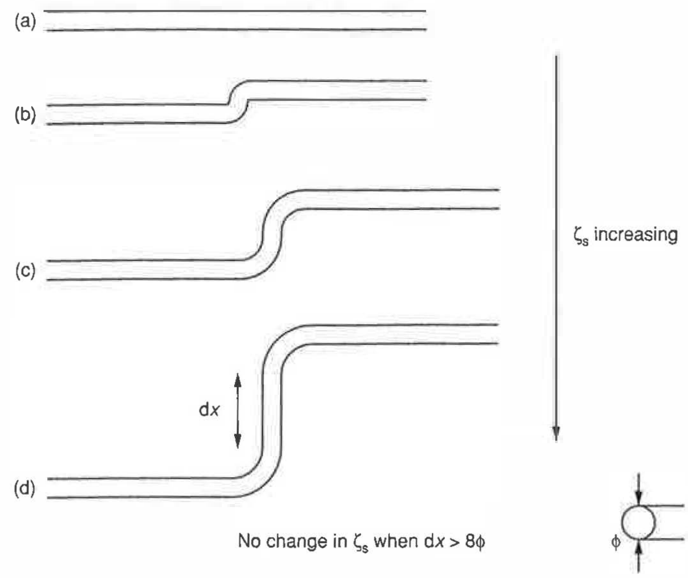

Figure 8 Development of an S bend

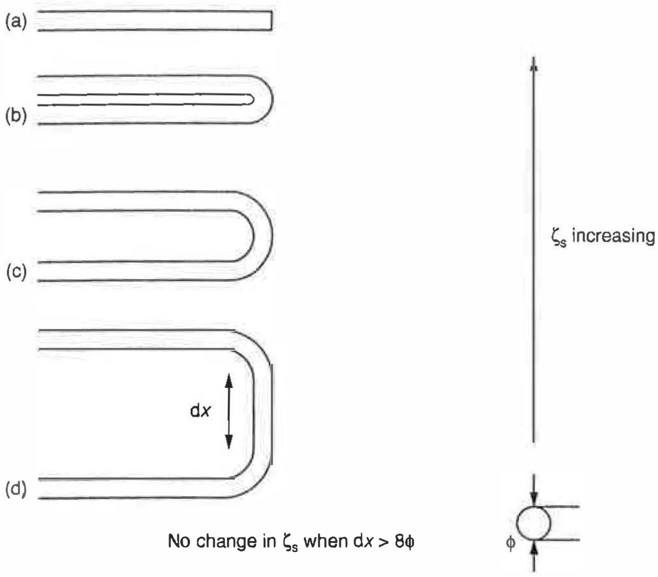

Figure 9 Development of a U bend

follows a similar development for a bend of U configuration. However, the simplest case here is that of a 'capped off' duct (dead end), and hence this suffers from the maximum pressure loss possible as the flow will be completely stopped. As a result, the system pressure loss will actually decrease as the bends are gradually introduced into the system, starting as an abrupt bend in the ductwork and progressing until the bends are fully developed and separated by 8 to 10 hydraulic diameters as for the S bend example.

5 Conclusion

CFD has been used to determine whether the separation between fittings in series in a duct system and the configuration of such fittings has an impact upon the system pressure loss. It has been found that system pressure loss in such a system will behave in very different ways depending upon the orientation of the fittings in relation to one another.

In addition, the separation between fittings does exhibit an effect on pressure loss, although the magnitude of the effect is small for the double-bend system tested. It was also found that the separation effect disappears as the distance between fittings increases to about 8 to 10 hydraulic diameters of the duct.

Since the S bend can be viewed as a U bend with the second elbow rotated about its entrance centre line by 180°, there are a large number of possible system configurations in between that could exhibit any number of differing flow and pressure loss behaviours. An in-depth analysis of such systems coupled with a detailed knowledge of flow field patterns would be required to produce pressure loss coefficient data accurately and to improve design of HVAC systems.

6 References

1 Chartered Institution of Building Services Engineers *CIBSE Guide, Volume C—Reference Data* 5th edition (London: CIBSE) (1986)

2 American Society of Heating, Refrigeration and Air-conditioning Engineers *ASHRAE Handbook—Fundamentals* (Atlanta, GA: ASHRAE) (1989)

3 Gan G and Riffat S B *K*-factors for HVAC ducts: numerical and experimental determination *Building Serv. Eng. Res. Technol.* **16**(3) 133–139 (1995)

4 Shao L and Riffat S B Accuracy of CFD for predicting pressure losses in HVAC duct fittings *Applied Energy 51* (Oxford: Oxford University Press) (1995)

5 Shao L and Riffat S B CFD for prediction of *k*-factors of duct fittings *Int. J. Energy Res.* **19** 89–93 (1994)