



Ss. Cyril and Methodius University in Skopje



Ansys Fluent Case Study

Air flow in a rectangular duct with three elbows

Developed and curated by the Ansys Academic Development Team

Marija Lazarevikj, Valentino Stojkovski, Zoran Markov

education@ansys.com

Summary

This case study deals with air flow in channels with rectangular cross-section which are mostly used in air distribution systems. The duct has three elbows with the role of fittings which change the direction of air stream. The elbows and the straight sections connecting the elbows are hydraulic resistance causing friction and local losses. These losses can be represented by a total loss coefficient. The aim of the case study is to determine the duct hydraulic resistance coefficient dependency on the Reynolds number by using Ansys Workbench. The CFD results are compared to available theoretical data. Moreover, physical phenomena such as flow separation and secondary flow occurrence are captured and discussed.

Table of Contents

1. Introduction.....	3
1.1 Rectangular ducts	3
1.2 Description of the problem	3
2. Setting up the problem in Ansys Workbench	4
2.1 Preprocessing	4
2.2 Processing.....	4
2.3 Post-processing.....	5
2.3.1 Air velocity distribution	5
2.3.2 Calculation of local loss coefficient.....	7
3. Conclusions	10
4. References	10

1. Introduction

1.1 Rectangular ducts

Ducts are mostly used in air distribution systems to supply or return air. Their shape can be round, square, rectangular or oval. Ducts with rectangular cross sections fit better to building construction – above ceiling and into walls, and they are much easier to install between joists and studs.

Friction losses in ducts depend on air velocity, duct length and diameter (size), and material roughness, while minor losses are caused by duct entrance/exit, sudden or gradual expansion/contraction, open or partially closed valves, and bends.

Bends are fittings used to change flow stream direction. Minor losses from bends are usually measured experimentally and correlated with the flow parameters. The bend-loss coefficient depends on the bend angle, the curvature ratio, and the Reynolds number.

A typical problem to practically solve is obtaining the pressure drop needed to drive the flow for a given duct geometry with added components such as bends at a desired flow rate of fluid with certain physical properties.

1.2 Description of the problem

The subject of this task is such a rectangular duct with three elbows in which air flows. The aim of the task is to determine the hydraulic resistance coefficient at different flow modes.

The rectangular channel analyzed is given in Figure 1, while its geometry parameters are defined in Table 1: L_0 , L_1 , L_2 and L_3 are lengths of the straight sections of the duct, R_1 and R_2 are inner and outer radii of the elbows, respectively, a and b are the width and height of the rectangular section, respectively.

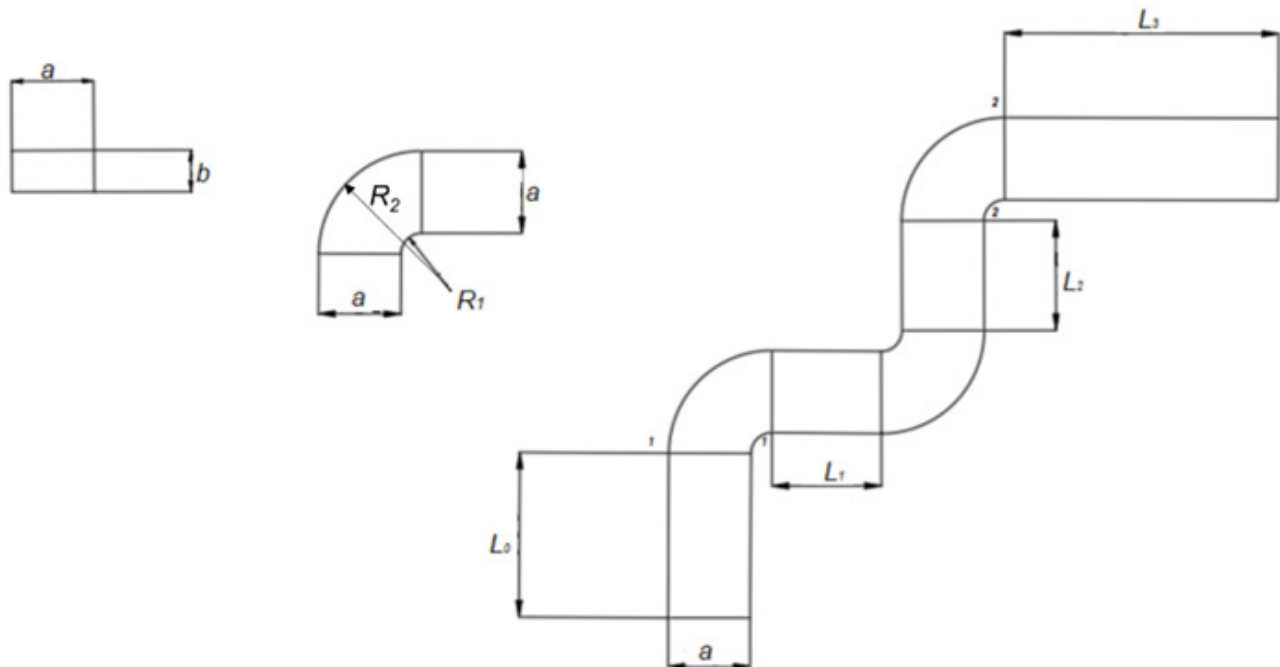


Figure 1 : Rectangular channel with three elbows- notations

Table 1 : Geometry parameters of the channel

a [mm]	b [mm]	L0=L3 [mm]	L1=L2 [mm]	R1 [mm]	R2 [mm]
150	75	1500	450	38	188

2. Setting up the problem in Ansys Workbench

2.1 Preprocessing

Space Claim is used to define the 3D fluid flow domain according to the geometric parameters of the rectangular channel of interest, given in Table 1. Decomposition of the computational domain into 7 parts is done to obtain structural numerical grid. The multizone method is used to generate numerical mesh of only hexagonal elements (Figure 2). The obtained mesh consists of around 433000 nodes.

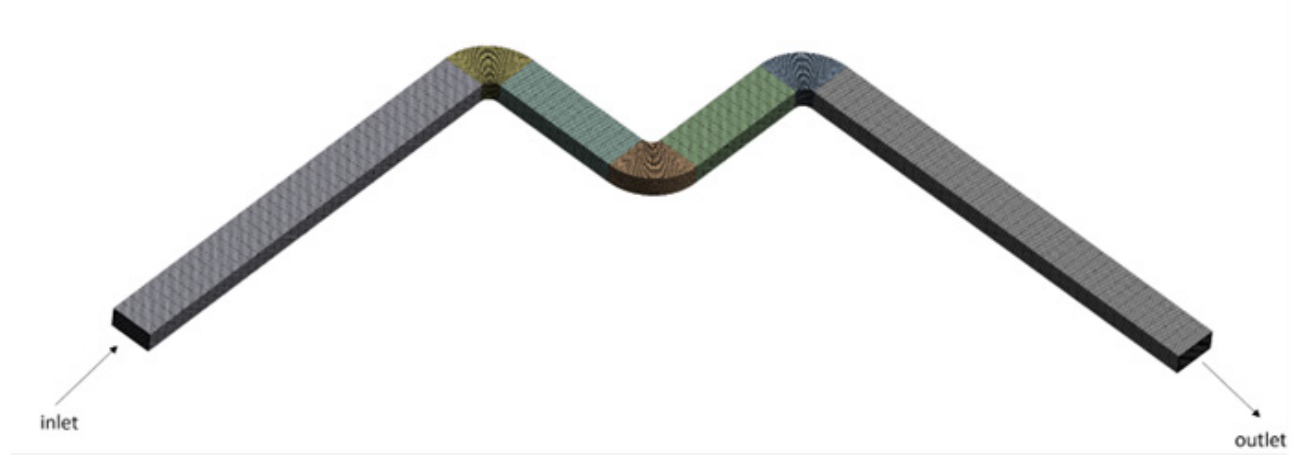


Figure 2 : Generated numerical grid

A boundary layer (Figure 3) is placed at the contact of the fluid with the duct walls to properly capture the viscous effects.

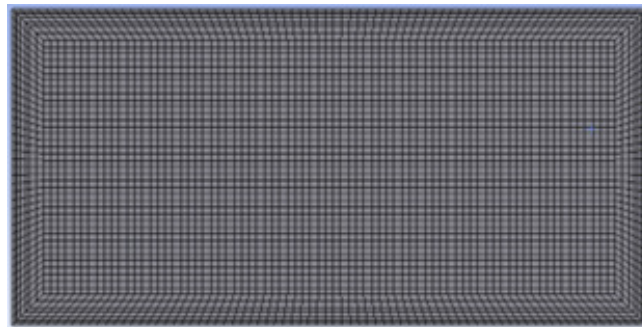


Figure 3 : Defined boundary layer

2.2 Processing

Viscous air flow in the rectangular channel is modeled and simulated in Ansys Fluent. The physical properties of air are constant dynamic viscosity ($\mu=1,7894 \cdot 10^{-5}$ Pa·s) and constant density ($\rho=1,225$ kg/m³). The standard k- ϵ model is used as turbulence model known for its reliable results for industrial applications. Boundary conditions are velocity inlet and pressure outlet. Different velocities at the inlet are assigned for Reynolds number in the range of 104-4·105. The pressure at the duct exit is atmospheric.

2.3 Post-processing

2.3.1 Air velocity distribution

The distribution of velocity in the duct for Reynolds number of 30 000 is shown in Figure 4.

Because of the bending, centrifugal forces appear with direction from the center of the curvature to the outer wall resulting in pressure increase (i.e., lower velocity) at the outer wall where a diffuser effect is present and pressure decrease (higher velocity) at the inner wall when the fluid flows from the straight section to the curved part of the channel. The order is reversed when air flows from the curved part into the straight section of the channel, i.e., velocity is higher at the outer wall and lower at the inner wall. These effects are present in the case of all Reynolds numbers with a difference in the velocity magnitude.



Figure 4 : Velocity contours at Re = 30000

Flow separation appears at the locations where diffuser effect is present and is characterized by vortices. The separation has higher intensity at the inner walls (straight sections of the duct after every bend) due to the inertial forces that act in the region of the bend towards the outer wall. Most of the pressure losses in the duct result from the vortices formation at the inner walls.

In the region of the first elbow, there is no flow separation which means the bend fulfills its role to change fluid stream direction in conditions when the inlet velocity profile is uniform. The flow is separated at the first elbow outlet. The straight section L1 is not long enough to obtain uniform velocity profile due to which the flow remains separated at the inlet of the second elbow; the same applies for the third elbow. Flow separation decreases at the outlet straight section of the duct; the length required to achieve uniform velocity profile, i.e. to decrease the flow separation can be perceived.

Figure 5 shows the velocity contours and velocity profile at the inlet and outlet of each bend. It can be seen that at each elbow inlet (in1, in2, in3) and outlet (out1, out2, out3), the velocity is higher at the inner wall (R1) and lower at the outer wall (R2).

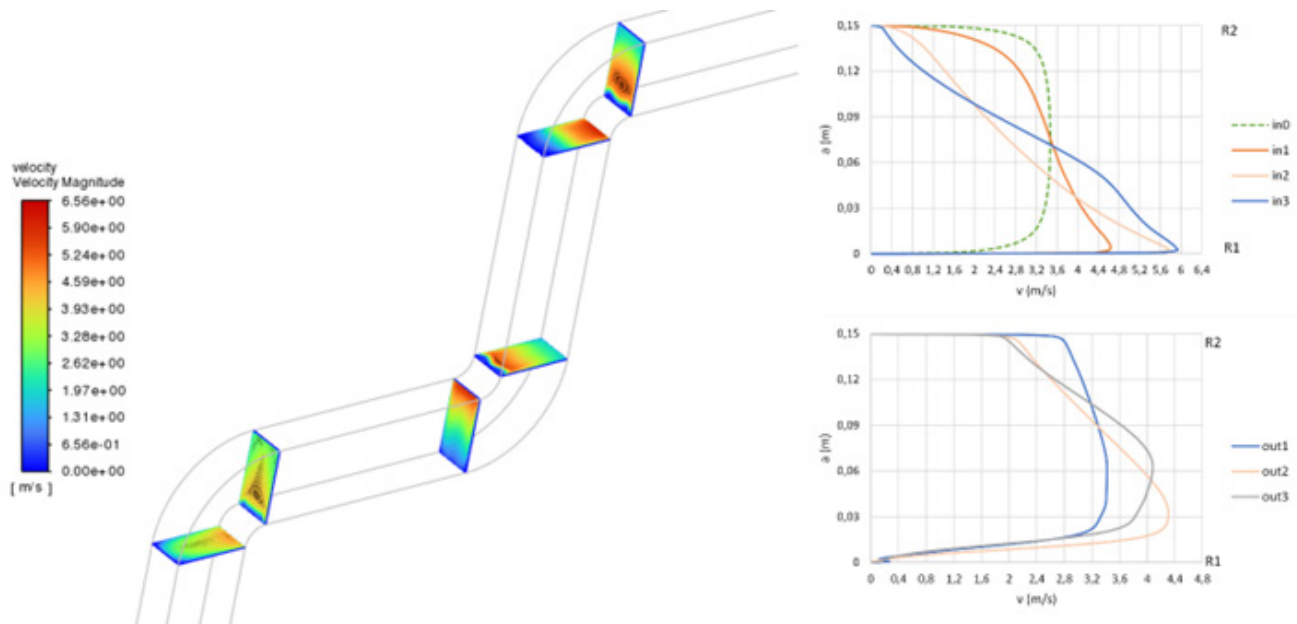


Figure 5 : Velocity contours (left) and longitudinal profile (right) at elbow inlet and outlet.

Figure 6 shows contours of longitudinal velocity in the inlet straight section of the duct cross-section. The contours of the same velocity magnitude are convex and denser (concentrated) around the corners. It can be noted that the mean stream flows from the core to the walls (where the shear is nearly constant) and towards the corner regions (where the shear drops sharply to zero). Thus, the longitudinal velocity contours have shape like the channel cross-section [1]. The velocity profile (Fig.6 right) of the turbulent flow is fully developed and uniform since the relative length of the starting straight section is $l_0/a=10$.

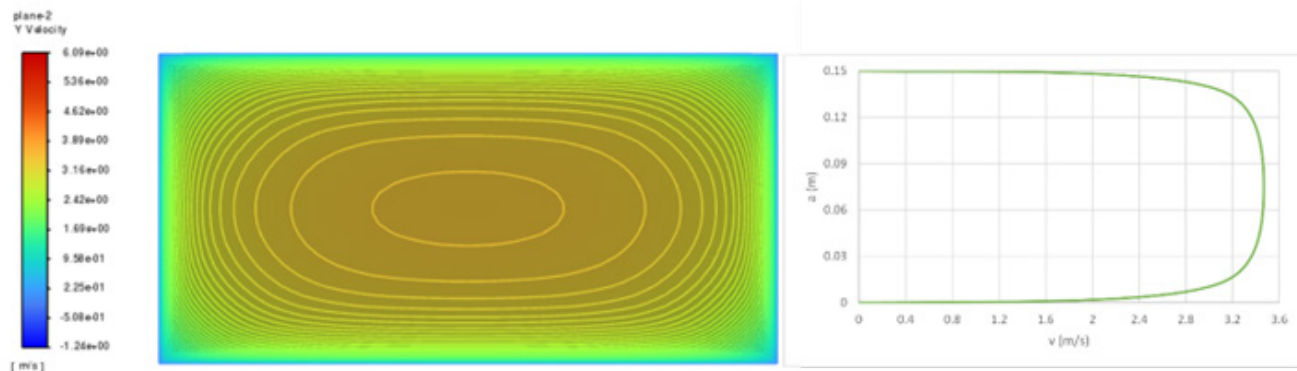


Figure 6 : Longitudinal (axial) velocity contours (left) and uniform velocity profile (right) at inlet section.

Figure 7 shows tangential velocity distribution in duct cross-section at the following locations: at the inlet and outlet of the first bend, second bend and third bend, respectively (in1, out1, in2, out2, in3, out3). The main fluid stream is parallel to the channel axis. The presence of secondary flow in the plane perpendicular to the direction of the main (primary) flow can be seen, which is typical for turbulent flows in straight ducts of non-circular cross-section or curved pipes of any cross-section. The average velocity field includes not only the longitudinal component but also transverse components that create a secondary flow.

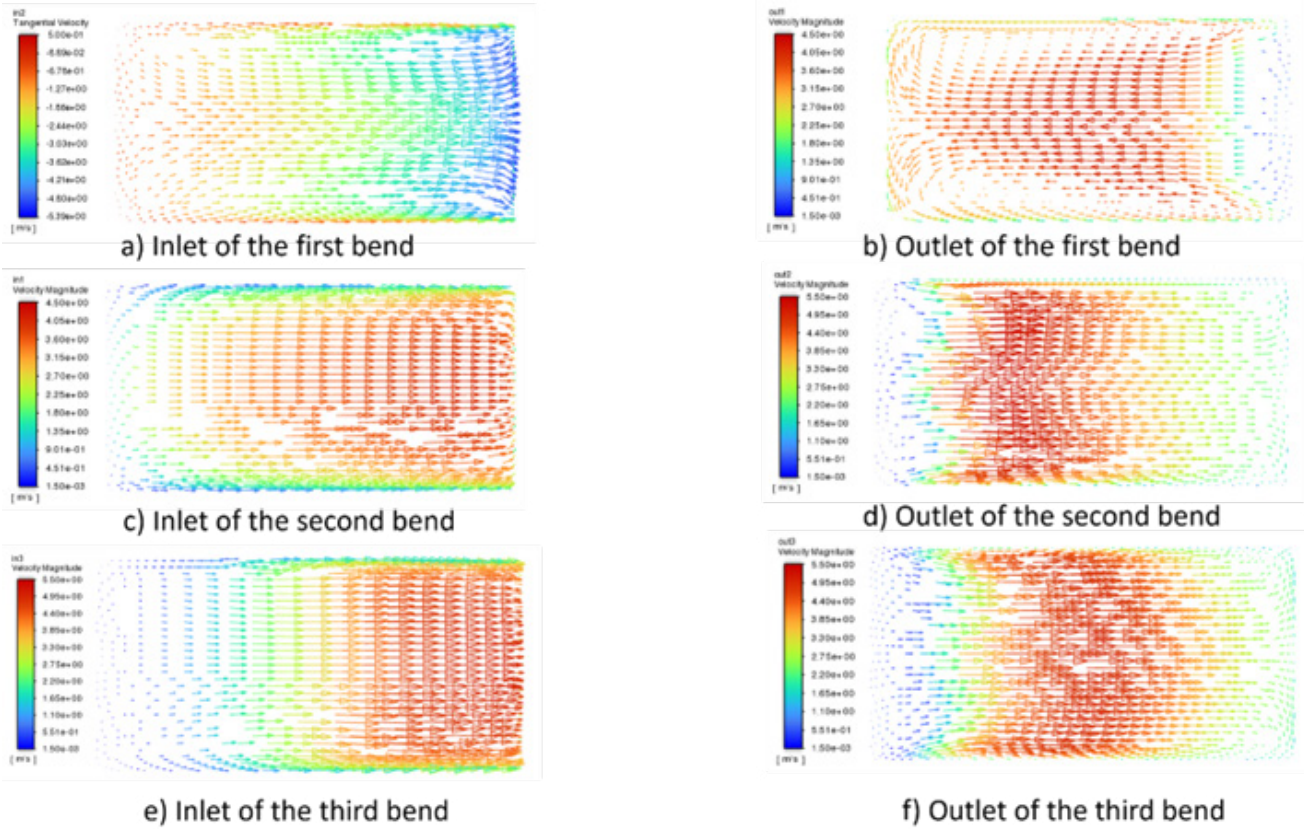


Figure 7 : Tangential velocity vectors at different cross-sections throughout the duct.

Secondary flow occurs due to the appearance of centrifugal forces and the presence of boundary layers at the walls.

2.3.2 Calculation of local loss coefficient

The hydraulic resistance coefficient of bends depends on the Reynolds number, relative length of the starting straight section l/a , the angle of bend δ , relative radius of curvature R/a , aspect ratio b/a .

For a non-circular channel, hydraulic diameter needs to be defined to calculate the Reynolds number. Hydraulic diameter is calculated as:

$$D_h = \frac{4A}{P} = \frac{4ab}{2(a+b)} = 100\text{mm}$$

where A is the surface of duct cross-section and P is the wetted perimeter of the cross-section.

Reynolds number is calculated as:

$$Re = \frac{vD_h}{\nu} = \frac{\rho v D_h}{\mu}$$

where v is the average air velocity in the duct cross-section and ν is the kinematic viscosity of air.

2.3.2.1 Theoretical calculations

According to the available data from Idelchik [2], for the three-bend rectangular channel with aspect ratio $b/a=0.5$, the Reynolds number is expressed as:

$$Re = \frac{va}{\nu} = \frac{\rho va}{\mu}$$

The total loss coefficient is sum of friction loss coefficient ξ_{fr} and local loss coefficient ξ_{loc} :

$$\xi = \xi_{fr} + \xi_{loc}$$

The friction loss coefficient is considering the friction losses occurring in the straight sections between the elbows (with lengths L_1 and L_2) and each elbow itself (with length $[R_0\pi/a180^\circ]*\delta$). It is calculated as:

$$\xi_{fr} = \lambda \frac{L_1 + L_2}{a} + 3\lambda \frac{R_0}{a} \frac{\pi}{180^\circ} \delta = 9.5\lambda$$

where the friction coefficient for smooth walls (zero roughness) is determined by the equation:

$$\lambda = \frac{0.316}{Re^{0.25}}$$

For the analyzed geometry, $R_0/a=0.75333$ and the bend angle $\delta=90^\circ$.

The local loss coefficient according to Idelchik [2] is: $\xi_{loc}=1.5 \cdot K$. The coefficient K depends on total length between the bends l_k and width a ratio, l_k/a . For $l_k/a=(l_{k1}+l_{k2})/a=(3a+3a)/a=6$, $K=1$.

The local loss coefficient is: $\xi_{loc}=1.5$.

The total loss coefficient is determined by:

$$\xi = k_\Delta k_{Re} \xi_{loc} + \xi_{fr}$$

where the roughness coefficient for smooth walls is $k_\Delta=1$ and the Reynolds number coefficient is given by Idelchik [2] in table 2.

Table 2 : Value of Reynolds number coefficient.

$Re \cdot 10^{-4}$	1	1.4	2	3	4	6	8	10	14	20	30	40
k_{Re}	2.2	2.03	1.88	1.63	1.56	1.34	1.14	1.02	0.89	0.8	0.83	0.9

2.3.2.2 CFD calculations

Loss coefficients are expressed by the ratio between the lost energy due to hydraulic resistance (*i.e.* head loss) h_l and the kinetic energy (velocity head) $v^2/2g$.

$$\xi = \frac{h_l/v^2}{2g} = \frac{\Delta p_{tot}/\rho g}{v^2/2g}$$

which depends on the type of local resistance and the flow regime.

The local loss coefficient ξ depends on the total pressure difference between the inlet in the first elbow and outlet of the last elbow, the air density ρ and air average velocity v :

$$\xi = \frac{2(p_{tot1} - p_{tot2})}{\rho v^2}$$

where p_{tot1} and p_{tot2} are total pressures at inlet of first elbow and outlet of last (third) elbow, respectively. In these calculations, an average value of velocity in several cross-sections in front of the first elbow is taken into consideration.

2.3.2.3 Comparison of CFD and theoretical results

The dependency of the hydraulic loss coefficient on the flow mode is given in Figure 8. Numerically obtained results are compared to the available theoretical data. It can be seen that very good agreement is achieved for Reynolds number in the range of 30,000-300,000 with a maximum error of -6.4%. According to the CFD results, the total loss coefficient decreases with Reynolds number.

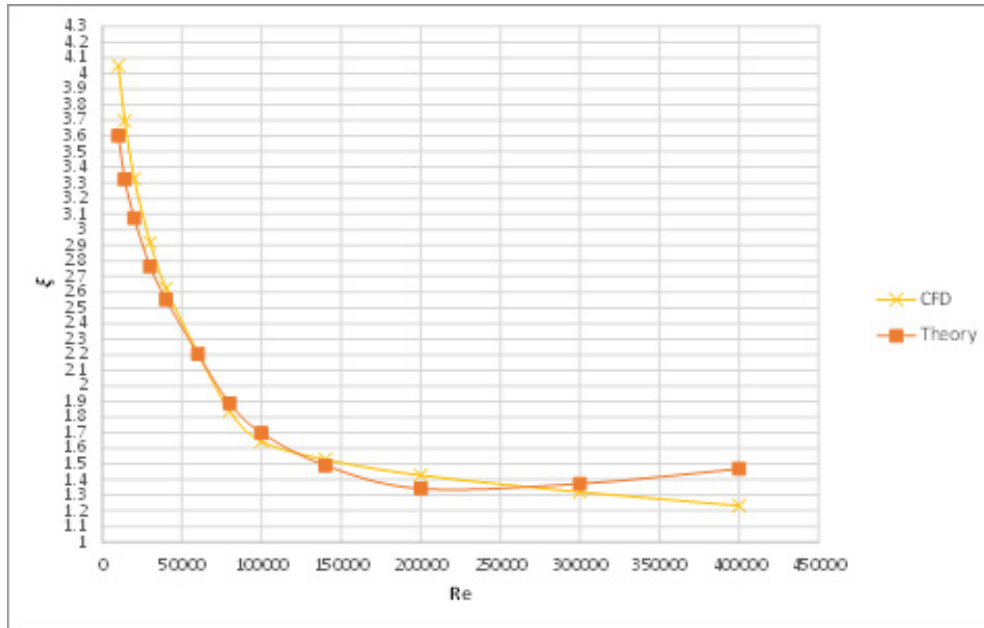


Figure 8 : Hydraulic loss coefficient dependency on Reynolds number - comparison between CFD and theoretical results

Figure 9 shows the loss coefficient of each bend. Lowest resistance is obtained for the first bend and highest resistance is seen in the case of the last bend. Different hydraulic resistance is obtained even though the elbows have the same geometry because of their different positions in the channel system and different velocity profiles at each elbow inlet (Figure 5 right). Moreover, the first bend has a lower pressure drop compared to the other two elbows due to the inlet velocity profile uniformity.

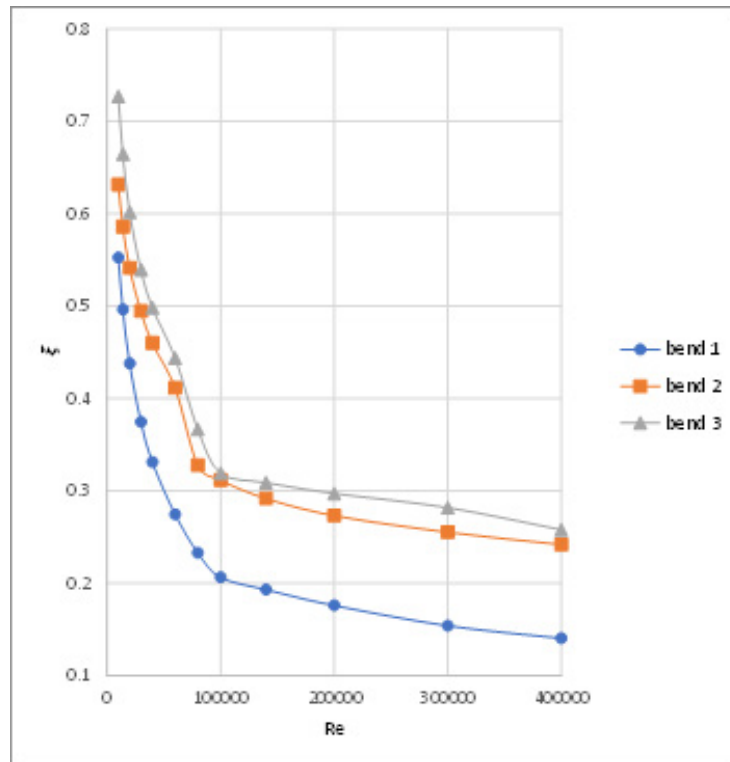


Figure 9 : Loss coefficient of bends.

3. Conclusions

Ansys Workbench is used as a tool to model and simulate turbulent flow of air in a rectangular duct used in HVAC systems. The whole process of geometry creation, meshing, solving and data processing is performed in Ansys Workbench which integrates all steps. The numerically obtained results are compared with available theoretical data. Good agreement is achieved especially in Reynolds number range of 30,000-300,000. Flow separation and secondary flow phenomena characteristic for turbulent flow in non-circular channels with curvatures are analyzed. It was concluded that the hydraulic resistance coefficient of elbows with same geometry depends on the bend position in the fluid flow system. In addition, the inlet velocity profile uniformity contributes to lower resistance.

4. References

- [1] Frank M. White, Fluid Mechanics, 4th Edition, McGraw Hill, Boston, 1999
- [2] I. E. Idelchik, Handbook of hydraulic resistance, Begell House, USA, 2007

© 2024 ANSYS, Inc. All rights reserved.

Use and Reproduction

The content used in this resource may only be used or reproduced for teaching purposes; and any commercial use is strictly prohibited.

Document Information

This case study is part of a set of teaching resources to help introduce students to topics related to fluids.

Ansys Education Resources

To access more undergraduate education resources, including lecture presentations with notes, exercises with worked solutions, microprojects, real life examples and more, visit www.ansys.com/education-resources.

Feedback

Here at Ansys, we rely on your feedback to ensure the educational content we create is up-to-date and fits your teaching needs.

Please click the link here out a short survey (~7 minutes) to help us continue to support academics around the world utilizing Ansys tools in the classroom.

ANSYS, Inc.
Southpointe
2600 Ansys Drive
Canonsburg, PA 15317
U.S.A.
724.746.3304
ansysinfo@ansys.com

If you've ever seen a rocket launch, flown on an airplane, driven a car, used a computer, touched a mobile device, crossed a bridge or put on wearable technology, chances are you've used a product where Ansys software played a critical role in its creation. Ansys is the global leader in engineering simulation. We help the world's most innovative companies deliver radically better products to their customers. By offering the best and broadest portfolio of engineering simulation software, we help them solve the most complex design challenges and engineer products limited only by imagination.

visit www.ansys.com for more information

Any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. All other brand, product, service and feature names or trademarks are the property of their respective owners.

© 2024 ANSYS, Inc. All Rights Reserved.