Available online at www.elixirpublishers.com (Elixir International Journal)

Mechanical Engineering

Elixir Mech. Engg. 163 (2022) 56011-56027

ISSN: 2229-712X

Prediction of Pressure Losses in Various Tunnel Configurations

Satyendra kr. Rajak and Ravi Ranjan Manjul
Department of Mechanical Engineering Ramchandra Chandravansi Institute of Technology, Palamu, Jharkhand.

ARTICLE INFO

Article history:
Received: 6 December 2021;
Received in revised form:
26 January 2022;
Accepted: 3 February 2022;

Keywords
Metro Tunnel;
Scale Model;
Flow Resistance;
Head Loss;
Experiments&CFD Simulation.

ABSTRACT

Tunnels represent a key part of the world transportation system playing a fundamental role both in people and freight transportation, especially in developed countries. Around the world, most major cities and metropolitan areas have metro systems accounting for hundreds of kilometers of underground tunnels and network systems. This study presents the pressure losses in ventilation systems of a long circular tunnel using for a metro purpose length of 700m, and diameter 6.6m, K.M.R.C., Kolkata has been vividly studied and presented in this dissertation. The tunnel which is being driven in the soft soil by using a tunnel boring machine has already developed up to a length of 700m and till the excavation is going on to continue. Auxiliary ventilation is adopted to ventilate the tunnel with the help of an axial-flow fan connected to a duct of approximately 680m in length. The various ventilation related parameters, viz. air quantity, pressure and temperature of the air in various locations inside the tunnel have been studied. In addition, we emphasize the pressure losses in tunnels in various locations i.e., to consider as bend taking place has been calculated. In the Kolkata metro, there is an earth pressure balance TBM is used. EPB excavation provides continuous support to the tunnel face by balancing the earth pressure against the forward pressure of the machine, and the advancement rate is 15 meters every day. In addition, the pressure losses in various tunnel configurations have been computed from the empirical relations as well as computational fluid dynamics (CFD) simulations. The aim of this project work is to correlate the values of pressure losses of analytical results and the computational simulation results with the help of using CFD software and conclude the difference of graph plotted to assess the adequacy of the tunnel and suggest a few measures to improve the ventilation inside the tunnel for building a cordial environment and enhancing the tunnelling operation. This study suggests that pressure losses in field study and CFD simulation are approximately the same.

1. Introduction

Underground ventilation serves many functions, in addition to the modest requirements for sustenance of human life. Because of the extent of the demand and the distance that air must travel from the surface to the working face, ventilation may become complicated and costly. As air travels from the surface to the workings and back, energy transitions and losses are involved in such a process of prime importance, hence, is the accurate determination of these energy losses. Airflow resistance in ventilation system is mainly determined by air speed in this system. Resistance grows with speed enhancing. This phenomenon is called pressure loss. Static pressure, produce by a fan, causes air motion in the ventilation system, which has certain resistance. The higher is the resistance of such a system, the less is the air consumption, moved by the fan. The computational fluid dynamics (CFD) technique, which has been well established and applied in the design of tunnel ventilation system, enables the prediction of fundamental field variables of pressures, velocities, temperature and smoke concentration at any location within computational domain of system being simulated. Application of computational fluid dynamics (CFD) was originally introduced for industrial applications, but today it has also become a common tool for assessing building ventilation, underground space ventilation and environmental performance. CFD is a very powerful technique used for predicting air movement and characteristics. CFD model is based on the concept of dividing the solution domain into sub-zones. Then, for each zone, the mass, momentum, and energy conservation equations are solved, utilizing the processing power of computers. This helps to perform calculations more easily and, in comparison with natural ventilation mathematical models, gives more detailed results. For example, CFD codes are used to predict airflow rate, air velocity and temperature, and airflow patterns inside and around buildings. Much software based on CFD codes have been developed like, Fluent, Phonics, and Fluent. In the last few years, an intensive work has been done using CFD. Comparisons of CFD results with wind tunnel tests have shown good agreement. However, some studies are limited to the use of CFD modelling, and have no access to experimental testing facilities. In this case, it is crucial to validate the implemented CFD code prior to the proposed modelling study in order to avoid producing any misleading results. This is because CFD is considered to be a sophisticated modelling technique.

Tele:
E-mail address: ravimanjul3315@gmail.com

1.1 Statement of the Problem

The main problems occurred in the tunnel due to poor ventilation are given below:

a. The increase in roughness of the wall of the tunnel increases the pressure loss and therefore causes more power consumption in fan motor.

b. Poor ventilation affects the physical and psychological health of humans. Poor ventilation allows for the accumulation of hazardous contaminants. The resulting physical effects on workers are harmful. Psychological effects like stress arise when members know they are constantly exposed to ventilation hazards.

1.2 Objectives

The objectives of the present study are outlined as follows:

a. To predict the pressure losses in various tunnel configurations.

b. To study the ventilation system of a metro project under development and compare the field study data with the simulation results.

1.3 Significance of the Study

The main significance of the study is to make the better environment in the tunnel and overcome the losses occurs in the tunnel. It's expected that the outcome of this investigation will contribute towards the design of ventilation system developing guidelines for control measures, which ultimately will lead to control the tunnel environment and increase safety.

2. Literature Review

The literature review on the ventilation system of tunnel has been carried out under the following three broad headings:

1. Ventilation system of underground tunnel.
2. Design of ventilation system of underground tunnel.
3. Modeling of ventilation system of underground tunnel using CFD software.
4. Pressure losses in underground tunnel ventilation.

2.1 Ventilation System of Underground Tunnel

Kumar and Cox (1989) carried out a series of simulation of forced- ventilated tunnel fire experiment with the Zwenberg tunnel. The effect of changing the wall roughness from smooth to rough-wall values and effect of radiation model were examined. They concluded that the inclusion of the radiation model gives a lower temperature gradient and smeared temperature distribution due to the increase in heat transfer hot gas, ambient air and tunnel wall. Mangs and Keski-Rahkonen (1994) performed a full -scale fire experiment on passengers cars. Rate of heat release, rate of mass change, CO and CO2 production rate, smoke production rate, time varying gas temperature above car were recorded. It provided experimental information for fire and smoke modeling. Oka and Atkinson (1995) performed experiments with a scaled model of HSL Buxton tunnel using 'FROUDE' scaling technique and investigate the limitations of existing empirical equations approach for the critical velocity. New formulation and correlation based on scaled model experimental results for the critical velocity were presented. However, full scale test has not been carried out. They also suggested the existence of an upper limit of critical velocity which will be independent of fire load. Carvel et al. (2001) studied the influence of forced longitudinal ventilation on car fires, pool fires and heavy goods vehicle fires in tunnels. Bayesian probabilistic approach is used to refine estimates, made by a panel of experts, with data from experimental fire tests in tunnels. Results are presented and the implications are discussed. The influence of longitudinal ventilation on heavy goods vehicle fires is predicted to be much larger than the expert's estimates, causing a fire to grow ten times larger than if natural ventilation was used. They have observed that the effect of ventilation on a pool fire in a tunnel depends on the size of the pool. The heat release rate of small pool fires may be reduced by forced ventilation, whereas it may be enlarged for large pool fires. The size of a car fire is not expected to be greatly affected by forced ventilation at low ventilation velocities.

2.2 Design of Ventilation System of Underground Tunnel

Chow and Chung (1998) studied the longitudinal ventilation for smoke control in a tilted tunnel by scale modeling technique. A 1/25 tunnel model of length 2 m with adjustable angle to the horizontal was constructed by transparent acrylic plastics. A small pool fire was put in with smoke generated by burning smoke pellets. Longitudinal ventilation was set up by a fan at one end. Different ventilation rates were adjusted by a transformer on controlling input to the fan motor. Experiments were performed with the tunnel angle varying up to 30o to the horizontal. Observed smoke movement patterns indicated that the shape of the buoyant plume inside the tunnel depends on the tilted angle. Smoke would flow along the tunnel floor due to gravity. The bending angle of the plume depends on Oddny and Aralt (2005) studied the air quality and ventilation fan control based on aerosol measurement in the bi-directional undersea Bomlajord tunnel. Aerosol, NO and CO concentration, temperature, air humidity, air flow and number of running ventilation fans were measured by continuous analyzers every minute for a whole week for six different one-week periods over ten months in 2001 and 2002 at measuring stations in 7860m long tunnel. The ventilation control system was mainly based on aerosol measurement taken by optical scatter sensors. The ventilation turned out to be satisfactory according to Norwegian air quality standards for road tunnels. Betta *et al.* (2010) carried out numerical investigation on an alternative jet fan, known in literature as Banana Jet and it compared its fluid dynamic performances with traditional axial ventilation systems. The alternative jet fan is equipped with inlet/outlet sections inclined at a fixed pitch angle (α) toward the tunnel floor. This approach establishes an alternative solution that is able to provide a safety level equivalent to the traditional solution, in different scenarios. Both systems are installed in a one-way tunnel and two different scenarios (without vehicles and with traffic jam) are considered, in event of fire. The fire was simulated setting heat flux on Heavy Good Vehicle (HGV) surface and comprehensive of radioactive heat flux. Kashef (2010) conducted a research study at the National Research Council of Canada to evaluate the effect of different parameters on the performance of emergency ventilation systems in the event of a fire in a section of a road tunnel. The parameters included: tunnel cross-section width and height, tunnel slope, fire size and location, meteorological conditions, and modes of fan operation. The study aimed at assessing the ability of in-place emergency ventilation strategies to control smoke spread and minimize its impact on tunnel users using both numerical and experimental approaches. Four field fire experiments and eight numerical simulations were conducted. Based on the study results, recommendations were made to optimize the ventilation strategies in the tunnel section. the tunnel angles. Tunnel inclined with higher angles to the horizontal would give larger amount of smoke flow. Ingason et al. (2012) Describe the reduction in the longitudinal airflow velocity due to the fire and hot gases resistances in a large tunnel fire, a theoretical model, taking into consideration the pressure losses over the fire source and obstructions, the thermal stack effects, and the hydraulic resistance induced by the tunnel walls, fire protection boards and a HGV trailer mock-up, is developed and validated using the large-scale tests data from the fire tests performed in the Runehamar tunnel with longitudinal ventilation in Norway2003. Two large mobile fan units were used to create a longitudinal flow within the tunnel and prevent smoke back layering upstream of the fire. One fan was located outside the entrance of the tunnel and the other inside the tunnel. The fire load consisted of a mock-up simulating a heavy goods vehicle (HGV) trailer creating a maximum heat release rates in the range of 66–202 MW. Two methods of calculating the mean temperature related to the thermal expansion and stack effect are proposed and compared.

2.3 Modelling of Ventilation System of Underground Tunnel Using CFD Software

CFD modelling has been attempted for tunnel ventilation simulation recently due to the advances in computer hardware and software reliability and efficiency. Much of current tunnel ventilation techniques and assumption are based on simulations and experiences established from other areas. Most of them use standard k-ἑ turbulences model, with convective heat transfer and wall model. Some researchers studied the design of tunnel ventilation system using CFD software. Chow and Leung (1988) used CFD to determine the effect of ventilation velocity on smoke production from a fire in the Zwenberg tunnel. Rectangular tunnel geometry was assumed regardless of the original oval shape of the tunnel. The results were qualitatively agreed with the experiment yet the temperature difference compared with the experiment exceeded by200 K.Simcox *et al.* (1992) carried out a simulation of the king's Cross fire. The CFD results have showed agreement qualitatively with the experiment and the real scenario. The effects of spreading of heat source over different- size areas, varying the heat source power and the effects of different boundary condition were investigated. The results established the importance of trench effect to the rapid spread fire. Kerrison *et al.* (1994) also compared the room fire simulation result from FLOW3D CFD with experimental data. The simulation employed standard k-ἑ turbulences model with buoyancy modification; combustion and radiation were ignored. Results showed the model agreed reasonably well (within +/- 20% for most variables) and the difficulty lay in predicting near wall mass fluxes while the differences associated with the mass flux for the corner fires were within +/-40%. Woodburn (1995) carried out a series of CFD simulations with the data from Buxton tunnel fire experiment. A combination of standard k-ἑ turbulences model and modified k-ἑ turbulences model described by Rodi (1985) were used. It was derived from the second order equations for the individual turbulent shear stress under the specific conditions of near horizontal flow with stable stratification. Better agreement between simulation results and experimental results were obtained with the modified k-ἑ turbulences model. Meeks *et al.* (1997) carried out a CFD simulation with the Jubilee Line Extension Project for the London Underground using computer code STAR-CD. The dynamic flow behavior of hot smoke from tunnel fire was simulated and the results were used to justify the use of a simpler empirical method for quantifying the capacity of emergency tunnel ventilation. Simulation results showed conformity with the critical velocity calculated from empirical equation. Levy *et al.* (1997) customized a general Computational Fluid Dynamics (CFD) code and validated specifically for tunnel application during Phase IV of the Memorial Tunnel Fire Ventilation Test Program. For transverse ventilation, a novel approach was developed for modelling the interaction between ventilation ducts and the tunnel. A network model, comprised of nodes and links, is used to represent the ducts while a field model is used to represent the tunnel. These models interact with each other through boundary conditions. Chow et al.(2003) studied the design and construction of operating rooms in Hong Kong, including the upgrading of the older ones, based on the UK Health Building Notes and Health Technical Memoranda. Observations and three field measurements in a case study have shown that the air flow and some design features were not fully complied with the specified requirements. A CFD analysis supported by field measurements was made to simulate the temperature distribution, airflow pattern and the contaminant dispersion. The study placed an emphasis on the health risk of the airborne bacteria released from the surgical team on the patient, and vice versa. Li and Chow (2003) studied the performance of tunnel safety systems for fire protection and evaluated the ventilation provision using Computational Fluid Dynamics (CFD). Ventilation safety systems commonly used in tunnels are reviewed first. The safety aspects related to tunnel fire and ventilation are then discussed. Airflow induced by an accidental vehicular fire in a tunnel is simulated using CFD. Based on the results, performances of different safety systems are evaluated and compared. The ventilation systems considered are longitudinal, semi transverse, transverse, partial transverse, and combined longitudinal and semi-transverse ventilation systems. Furthermore, sensitivity analyses on the effects of the grid size, and number of iterations on the required computing time and accuracy of the results are carried out. Yuan and You (2007) evaluated the velocity and temperature field of subway station and the optimized ventilation mode of subway side-platform station using computational fluid dynamics (CFD) simulation with the boundary conditions collected for simulation computation through field measurement. It is found that the two-equation turbulence model can be used to predict velocity field and temperature field at the station under some reasonable presumptions in the investigation and study. Hargreaves and Lowndes (2007) conducted the analysis of a series of validated computational studies to assess the effectiveness of alternative auxiliary ventilation system son the mitigation of any adverse environmental conditions experienced within these drivage. A series of steady-state computational fluid dynamics (CFD) models were constructed to replicate the ventilation flow patterns seen at the head end of a drivage during the various stages of a cutting and bolting cycle. The results obtained from these simulations were compared against the data obtained from a series of full-scale ventilation experiments conducted within a rapid development drivage of a representative UK deep coal mine. It is concluded that CFD models may be successfully used to identify the ventilation characteristics associated with the various auxiliary ventilation systems during a typical cutting bolting cycle. Su *et al.* (2008) have reported that Mono draught TM wind catchers are commercial natural ventilation devices, which are primarily driven by wind to produce both extract and supply air flow. The measurement of the net flow rate (extract minus supply) of a Mono draught TM wind catcher ABS 550 for various wind speeds and directions is introduced. The ventilation measurement system uses a cone flow meter and a blower fan. CFD standard k - έ turbulence model is employed to calculate the flow rate. The situation using a blower fan is considered in modelling and the effect of the manometer sensitivity is also discussed. The comparison has indicated a good agreement between measurement and simulation. CFD modelling of the wind catcher is then carried out for the situation of outdoor far field wind. At the same nominal wind speed, the calculated extract flow rate of the wind catcher in a far field wind is roughly twice that for the situation using a blower fan, the wind direction has a small effect on the extract flow rate. The extract and supply flow rates are also calculated for various room pressure due to various wall openings and installation on a flat roof or a pitched roof. The contribution of the buoyancy effect on the flow rates is also discussed in simulation. Migoya *et al.* (2010) reported that during a fire inside a tunnel, the average heat release rate (HRR) is estimated according to the type of vehicle. Frequently, the overall HRR is considered, however it is also necessary to know its time evolution to design real time systems, particularly ventilation, which respond to fire events or signals as fast as possible. Nowadays, there is no well-established and generally accepted procedure to know the power liberated at each instant of time inside an operational tunnel. That procedure could help in taking the correct actions to adapt the tunnel ventilation in order to diminish the effects of the fire and the smoke. They have proposed a method to calculate the heat release rate using sensors that can be installed inside an operational road tunnel. Besides, the location of the fire could also be calculated accurately and quickly. To achieve the previous purposes, a stationary database that depends on HRR, its location, and the ventilation speed is calculated with CFD programs; the data are compared with temperatures measured by the sensors located inside the tunnel. The program used to generate the database is the simplified model UPMTUNNEL. The predictions of the model are compared with the results of calculations carried out using the general-purpose code FLUENT, and with measurements done in a tunnel with a real fire, produced with a fuel tray.

2.4 Pressure losses in underground tunnel ventilation.

Montecinos and Wallace (2010) the calculation of the pressure drop along tunnels is by using Atkinson’s equation. The friction factor in Atkinson’s equation is determined from measured or computed values of airway dimensions and tunnel interior finishing according to mining method and ground support type. This paper re-visits the friction factor according to Colebrook´s relationship and the Darcy-Weisbach equation, which are widely used in mechanical engineering. This method of calculating the friction has the advantage of a more reliable determination since the size of the airway is implicit in the method and only a representative absolute roughness has to be selected to determine the friction factor. Measurements of pressure drop performed at Codelco’s El Teniente mine, Chile, permitted the determination of the absolute roughness for different tunnel sizes and wall finishing. Diego *et al.* (2010) calculated the losses in 138 situations of circular tunnels (varying tunnel diameter, air velocity and surface characteristics), by both traditional and CFD (computational fluid dynamics) means. The results of both methods are compared and adequate correlation has been observed, with CFD values constantly 17% below the values calculated by traditional means. The paper deals with the main problems commonly encountered in the CFD use, meshing and turbulence, and shows guidance on the practical use of this numerical method. It also shows the capabilities of the method in simulation domains including machinery of underground works: road headers, dumpers and excavators. Jade and Sastry (2008) Studies in combining and bifurcating flows in two-way junctions and splits are reported, deviations persist in the estimated shock loss coefficients (SLCs). Due to their complex nature, more work has to be done to be able to correctly predict flow losses at these locations. In two-way splits, the flow at bifurcation is known to be characterized mainly by two recirculating zones in the downstream branches, whereas, in a junction one recirculating zone is observed immediately downstream of the main branch. The relative flow rate in the branches (quantity ratios) is treated as the only parameter defining the losses at these locations. However, the effect of wall roughness, if any, on SLCs becomes important for my ventilation. This paper presents the critical determinants of shock losses at splits and junctions at varying quantity ratios. The flow at 90º split and junction was examined both experimentally and numerically for Reynolds number in the range of 1.0×104 to 1.6×105. With a well-designed laboratory setup, using precision instruments, velocity and pressure measurements were carried out, by using two scales of wall roughness. 3D numerical simulations for these experiments were conducted by using ANSYS CFX modelling tool. CFD simulations yielded results that are reasonably close to experiments, providing high degree of satisfaction in numerically predicting losses across the splits and junctions. However, comparisons of these numerical simulation results with existing literature standards showed significant under-estimation of shock losses by the standard literature by as much as 50%. A clear rise in the magnitude of shock losses was noted in both the approaches, with increased wall roughness. Based on the current studies, predictor equations incorporating quantity ratios are developed for accurate estimation of shock losses. Further investigations on roughness are needed, before one is able to incorporate wall roughness into the prediction of SLC. Tien, (1978) presented his research on shock losses at air crossing, which is a major source of shock-pressure loss in underground ventilation systems, shock losses caused by overcasts come from a combination of losses due to area changes as well as airflow direction change. Based on the results of his analysis, several recommendations were made to reduce shock losses at overcasts. Hartman, (1960) proposed the empirical relationships for calculate the loss coefficients in bends, area changes entries and exits. The equations are applicable for turbulent flows in two-way split or junctions with rectangular opening and in complex geometries, they are not very accurate.

3. Methodology

The methodology of the project work is divided into the following categories:

1. Ventilation survey of the underground tunnel.
2. Simulation of pressure losses in various configurations inside underground tunnel using CFD software.
3. Comparison of the field study data with the simulation results.

3.1 Ventilation Survey of the Underground Tunnel

The quantity and pressure surveys have been carried out in an underground tunnel at Kolkata. During the ventilation survey, the barometric pressure, dry-bulb and wet-bulb temperatures, average air velocities and the quantity of air flowing at different locations inside the tunnel are measured. The details of the underground tunnel are as follows:

3.1.1 Brief background of the underground Tunnel

In Kolkata, roads account for only 4.2 per cent of the total surface area, as against 25per cent in Delhi and 30 per cent in Mumbai. Hence a transport system that did not add to the existing traffic problems of the city has to be developed. The Metropolitan Transport Project (Railways) was therefore set up in 1969. Studies concluded that a Mass Rapid Transit System alone was the solution to the ever-increasing traffic problem. In 1971, a master plan was prepared for constructing five rapid transit lines. Priority was given to the busy north-south corridor between Dum Dum and Tollygunge, covering a length of 16.45 kilometers. The project was sanctioned in June 1972, and the foundation stone was laid by Prime Minister Indira Gandhi on December 29 the same year. After numerous impediments - including court injunctions, irregular supply of raw materials and even non-availability of funds until 1977-78, the dream project was completed on October 24, 1984. The Calcutta Metro, India's first and Asia's fifth such rail system, started partial commercial operations, servicing five stations - spanning a distance of 3.4km - from Esplanade in Central Kolkata to Bhowanipur in the south. A month later, a2.15-km stretch was added to the Metro service - this time in north Kolkata, between Dum Dum and Belgachia. By April 1986, the Metro service was extended up to Tollygunge; it covered 11 stations and an overall distance of 9.79 km. By 1995, the service covered 16.45 km, touching 17 stations, each separated from the other by around one kilometer. Under the pipeline of a new metro project along Eastern part of Kolkata and Howrah to meet the future transport demands of Kolkata, using an optimal mix of over ground rail and underground metro system.

Figure 1. Auxiliary ventilation system via duct during construction of Kolkata metro

3.1.2 Ventilation Surveys

During the ventilation surveys, the average quantity of air and the pressure of air flowing at different locations inside the tunnel are measured. The following ventilation surveys have been carried out at the Kolkata metro tunnel:

1) Pressure survey
2) Quantity survey

3.1.2.1 Pressure Survey

The pressure survey in tunnels is carried out to measure the pressure difference between two points so that the pressure loss over a certain section of an airway or its resistances can be obtained. This is the major requirement for ventilation planning. Tunnel airway can be considered as a smooth conduit with incompressible flow. So, pressure drop due to frictional pressure drop is

$$\Delta P_f = \frac{fLPv^2\rho}{8A}$$

Where;

f = friction factor

L = Length of rubbing surface of mine (m)

P = Perimeter of cross-section (m)

v = Average velocity of air (m s^{-1})

ρ = Density of air (1.2 kg/ m^3)

A = Area of cross-section (m^2)

Equation (8) can be re-written in the form

$$\Delta P_f = \frac{kSQ^2}{A^3}$$

Where;

$k = \frac{f\rho}{8}$ = Coefficient of friction (kg m^{-3})

$S = LP$ = Area of rubbing surface (m^2)

Q = Flow through the airway (m^3 s^{-1})

ΔP_f = Friction Pressure loss (Pa)

3.1.2.2 Quantity Survey

The quantity survey carried out for measuring of the airflow inside the tunnel, which comprises of the following two steps:

1) Measurement of average air velocity at different locations.
2) Measurement of cross-sectional area of the locations.

The average air velocity at different locations inside the tunnel is measured by a vane anemometer. The quantity of air flowing is calculated using the following formula:

$$Q = AV \qquad (18)$$

Where;

A = Area of cross-section (m^2)

V = Average velocity of air (m s^{-1})

3.2 Simulation of pressure losses in underground tunnel using CFD software

The CFD modeling of underground tunnel has been carried out by using the CFD software, Fluent 6.1 and Ansys Fluent v12. The modeling work consists of the following two important steps:

1) Creating basic geometry
2) Solving and analyzing the results.

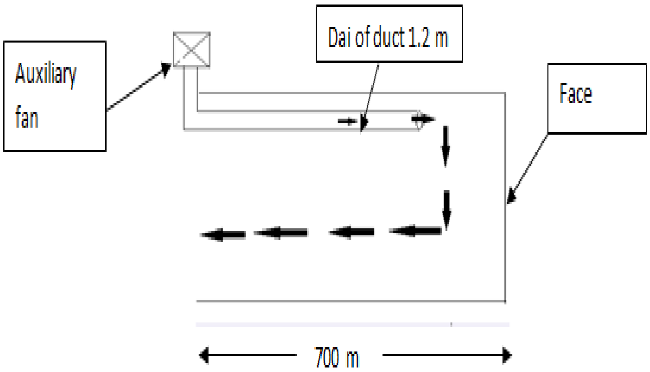


Figure 2 . Schematic line diagram shows the Kolkata metro during construction.

3.2.1 Creating basic geometry

The basic 2-D and 3-D geometry of the panel of underground tunnel was created by GAMBIT 2.1.6, the pre-process of the CFD software. The dimensions of the tunnel are given below:

Diameter of the tunnel = 6.6 m

Length of the tunnel = 700 m

To create the geometry using GAMBIT, Fluent 5/6 solver is used and the following procedure is adopted:

- Create vertex
- Create edge
- Create face
- Meshing the geometry
- Apply Boundary condition
- Saving the case file

3.2.3 Solving and analyzing the results

The Fluent 6.1 CFD code is used for solving and analyzing the results following the steps mentioned below:

- FILE > READ > CASE
- GRID
  > CHECK
  > SCALE
- DISPLAY > GRID
- DEFINE
  > MODEL
  > SOLVER
  > ENERGY
  > VISCOUS
  > MATERIAL
  > BOUNDARY CONDITION
- SOLVE > INITIALIZE
  > INITIALIZE
  > MONITORS
  > RESIDUAL
  > SURFACE
- FILE > WRITE > CASE & DATA
- SOLVE > ITERATE
- DISPLAY > CONTOURS
- REPORT > FLUXES
- DISPLAY > VECTOR

CFD modeling of the underground tunnel is done using a commercial CFD package FLUENT 6.1 and v12. The standard turbulent model is adopted and incompressible airflow is considered. The overall accuracy of prediction by the standard turbulent model has been proved acceptable in simulation of underground ventilation. The structured grid pattern is adopted for the underground tunnel. The intake is represented by a velocity inlet boundary of 6.6 m diameter. The return is represented by a pressure outlet of 6.6 m diameter. The mesh analysis is carried out for all zones in terms of air velocity profiles. The quad grid shape is adopted for meshing the geometry of the tunnel.

3.2.4 Flow specification and boundary condition

For the CFD analysis discussed below, the models were specified as steady state turbulent, single phase, and single species flow. Body force and heat transfer effects are small and hence neglected. The fluid was air with a density of 1.22 kg/m^3 and a viscosity of 1.7894 kg/m-s. At the inlet, the velocity components are specified. At the outlet, pressure was specified as zero. The boundary conditions and other details of the CFD simulations are presented below:

For the Kolkata metro

Velocity of air in the duct =15 m/s
Diameter of the duct =1.2 m
Diameter of the tunnel = 6.6 m
Length of the tunnel = 700 m
No. of nodes =38965
No. of elements= 195651

For various tunnel configurations

Velocity at inlet = 0.60 m/s
Diameter of the tunnel = 6.6 m
Length of the tunnel = 700 m
Pressure at out let = 0 pa
No. of Meshes in elbow = 4774
Meshing type = Quad Map
No. of meshes in sudden contraction = 5098
Meshing type = Quad Sub map
No. of Meshes in split = 4032
Meshing type = Tri Pave
The operating conditions are as follows:
Input temperature = 300K
Outlet temperature = 304K.

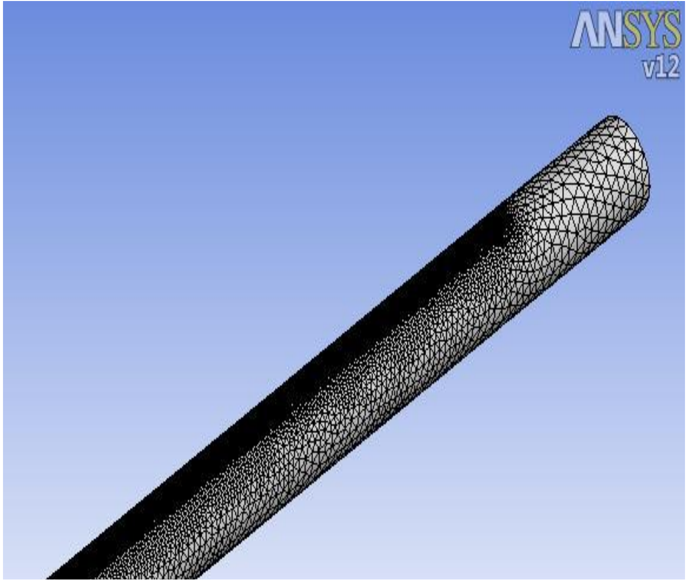


Figure 3. The meshing of the tunnel geometry of Kolkata metro

4.1 Results of ventilation study of underground tunnel of Kolkata metro

The ventilation survey data of the underground tunnel of Kolkata metro are given in Table 1 and the summarized results of CFD simulation of the metro are presented in Table 2.

Table 1. Field study data

Sl.No.	Length of Tunnel (m)	Average velocity (m/s)	Discharge (m^3/s)	Pressure Loss, ΔP_f (Pa)
1	105	0.44	15.04	0.012
2	170	0.45	15.38	0.020
3	277.5	0.48	16.41	0.038
4	289.5	0.49	16.75	0.042
5	295.5	0.50	17.097	0.044
6	307	0.52	17.78	0.050
7	318	0.52	17.78	0.052
8	324	0.55	18.80	0.059
9	337	0.59	20.17	0.071
10	628	0.67	22.91	0.170
11	699	0.70	23.93	0.238

Table 2. Summarized results of CFD simulation for Kolkata metro

Sl. No.	Length of tunnel (m)	Average velocity (m/s)	Discharge (m^3/s)	Pressure loss, ΔP_f (Pa)
1	105	0.43	14.70	0.0118
2	170	0.44	15.04	0.019
3	277.5	0.46	15.72	0.035
4	289.5	0.48	16.41	0.040
5	295.5	0.49	16.75	0.042
6	307	0.50	17.09	0.046
7	318	0.51	17.43	0.050
8	324	0.53	18.12	0.055
9	337	0.58	19.83	0.068
10	628	0.65	22.22	0.160
11	699	0.75	25.64	0.238

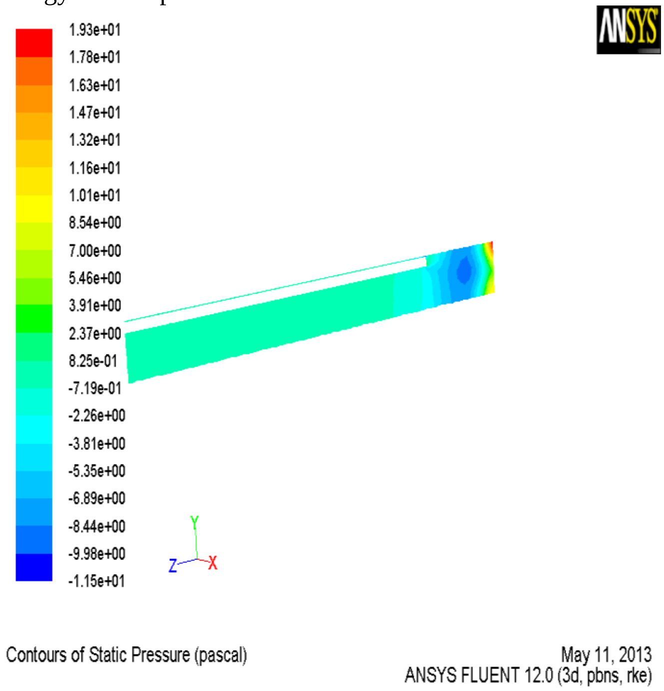
4.1.1 Results of Simulation

The simulation of ventilation survey data with CFD are divided into the following categories:

- Pressure
- Velocity

Pressure

- Fig.12 shows the static pressure contours in slice of tunnel. This simulation results show that the CFD provides a better accuracy.
- The pressure loss in the tunnel is matching with the ventilation study data.
- Since, the coefficient of friction for the rubbing surface of tunnel is less (k = 0.001), the pressure loss determined is also less. Therefore, low pressure loss will help in minimizing the energy consumption in ventilation.



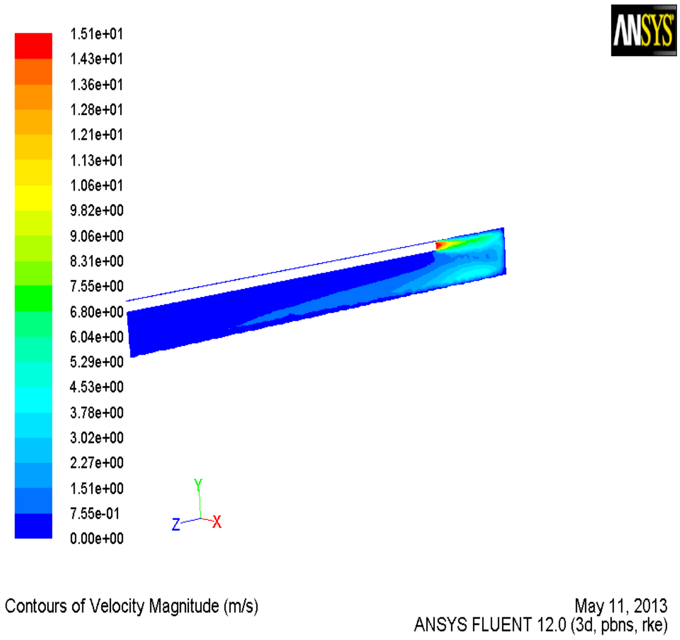
Contours of Static Pressure (pascal)
May 11, 2013
ANSYS FLUENT 12.0 (3d, pbns, rke)

Figure 4. Contour of static pressure for Kolkata metro.

Velocity

- It is useful to investigate the airflow pattern within the tunnel. The computational model indicates the simulated airflow pattern at different positions of the tunnel. It is a slice of the circular tunnel.

Fig. 5 shows the contours of velocity magnitude indicating the variation in the air velocity in the tunnel. The air velocity distribution of the tunnel shows that the distribution of the air velocity is good enough and balanced.



Contours of Velocity Magnitude (m/s)
May 11, 2013
ANSYS FLUENT 12.0 (3d, pbns, rke)

Figure 5. Contour of velocity magnitude of a Kolkata metro straight tunnel

4.1.2 Comparison of simulation results and experimental data of Kolkata metro.

A comparative study between the simulation results and experimental data has been made and the result is shown in graph.

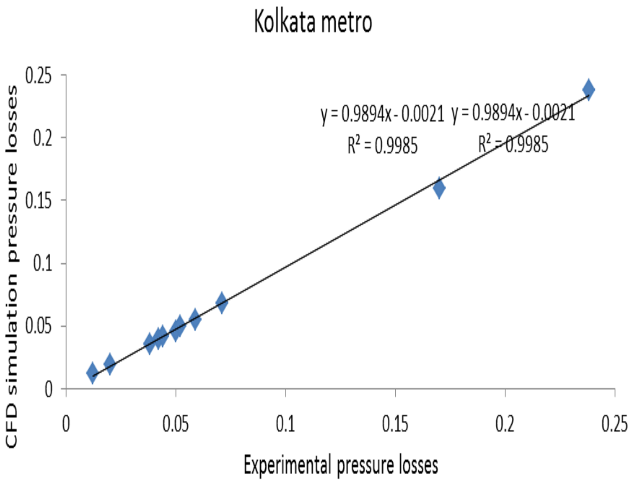


Figure 6. CFD vs. experimental pressure losses

The regression factor is 0.998 so it almost to 1. Means the Experimental values and simulation values is approximately match.

4.2 Results of ventilation study of underground tunnel for elbow bend

Grids of the elbow bend are shown in the Figure 7.

Computational simulation results:

Summarized results of CFD simulation of the metro are presented in Table 3.

4.2.1 Results of Simulation

The simulation of ventilation survey data with CFD are divided into the following categories:

- Pressure
- Velocity

Pressure

- Fig.8 shows the pressure contours. This simulation results show that the CFD provides a better accuracy.
- The regression factor in the case of elbow is 0.715 since the pressure loss in the tunnel is roughly matching with the ventilation study data.
- Since, the coefficient of friction for the rubbing surface of tunnel is less (k = 0.001), the pressure loss determined is also less. Therefore, low pressure loss will help in minimizing the energy consumption in ventilation.

Velocity

- It is useful to investigate the airflow pattern within the elbow bend tunnel. The computational model indicates the simulated airflow pattern at different positions of the tunnel.

Fig. 10 shows the velocity contours indicating the variation in the air velocity in the tunnel. The velocity vectors are shown in Fig. 11. The air velocity distribution of the tunnel shows that the distribution of the air velocity is good enough and balanced.

4.2.2 Comparison of simulation results and experimental data of elbow bend.

A comparative study between the simulation results and experimental data has been made and the result is shown in figure 12.

4.3 Results of ventilation system of sudden contraction in a tunnel

The grids of the sudden contraction in tunnel are shown in the figure 13.

Computational simulation values:

Summarized results of CFD simulation of the sudden contraction are presented in Table 5.

4.3.1 Results of simulation

The simulation of ventilation survey data with CFD are divided into the following categories:

- Pressure
- Velocity

Pressure

- Fig.14 shows the static pressure contours. This simulation results show that the CFD provides a better accuracy.
- The regression factor is 0.954 so pressure losses in the tunnel are roughly matching with the ventilation study data.

Since, the coefficient of friction for the rubbing surface of tunnel is less (k = 0.001), the pressure loss determined is also less. Therefore, low pressure loss will help in minimizing the energy consumption in ventilation.

Velocity

- It is useful to investigate the airflow pattern within the tunnel with sudden contraction. The computational model indicates the simulated airflow pattern at different positions of the tunnel.

Fig.16 shows the velocity contours indicating the variation in the air velocity in the tunnel. The velocity vectors are shown in Fig. 17. The air velocity distribution of the tunnel shows that the distribution of the air velocity is good enough and balanced.

4.3.2Comparison of simulation results and experimental data of sudden contraction

A comparative study between the simulation results and experimental data has been made and the result is shown in figure 19.

4.3.3 Results of ventilation system of split in a tunnel

The grids of a split tunnel are shown in the figure 20.

Computational simulation values

Summarized results of CFD simulation of the split junction are presented in Table 6.

5.1 Results of simulation

The simulation of ventilation survey data with CFD are divided into the following categories:

- Pressure
- Velocity

Pressure

- Fig.21 shows the pressure contours. This simulation results show that the CFD provides a better accuracy.
- The regression factor is 0.172 so pressure losses in the access tunnel are not matching with the ventilation study data.
- Since, the coefficient of friction for the rubbing surface of tunnel is less (k = 0.001), the pressure loss determined is also less. Therefore, low pressure loss will help in minimizing the energy consumption in ventilation.

Velocity

- It is useful to investigate the airflow pattern within the split tunnel. The computational model indicates the simulated airflow pattern at different positions of the tunnel.

Fig. 21 shows the contours of velocity magnitude indicating the variation in the air velocity in the tunnel. The velocity vectors are shown in Fig. 23. The air velocity distribution of the tunnel shows that the distribution of the air velocity is not balanced.

5.1.1 Comparison of simulation results and experimental data

A comparative study between the simulation results and experimental data has been made and the result is shown in figure 26.

Table 3. Summarized results of CFD simulation for Elbow bend.

Sl.No.	Length of Tunnel (m)	Average velocity (m/s)	Discharge (m^3/s)	Pressure Loss, ΔP_f (Pa)
1	105	0.553	18.90	0.01946
2	170	0.554	18.94	0.031
3	277.5	0.559	19.11	0.05
4	289.5	0.558	19.08	0.053
5	295.5	0.554	18.94	0.054
6	307	0.205	6.83	0.0074
7	318	0.555	18.97	0.058
8	324	0.555	18.97	0.059
9	337	0.526	17.98	0.055
10	628	0.513	17.54	0.11
11	699	0.494	16.89	0.103

5.2.1 Shock Loss

We are calculating the shock losses by using this formula

$$\Delta P_s = \frac{X\rho v^2}{2}$$

Where,

X= Shock loss factor

ρ = Density of air (kg/m^3)

V= Avg. velocity of air (m/s)

Value of X= 0.60 (elbow)

= 0.50 (sudden contraction)

= 1 (split junction)

= 0.90 (circular straight tunnel)

(All the value of shock loss factor I have taken from G.B.Mishra book).

Table 4. Shock losses for various tunnel configurations

Types of bend	Shock loss (Pa)
Field(straight tunnel)	0.269
CFD (straight tunnel)	0.308
Sudden contraction (CFD)	1.028
Elbow (CFD)	0.305
Split junction (CFD)	0.227

5.2.2 Suggested measures for improving the ventilation system and environmental condition of the tunnel

Some imperative measures proposed for consideration to improve the ventilation system and environmental conditions of the tunnel are given as follow:

- Minimizing the leakage in the ventilation duct at the joints and holes.
- Supplying higher quantity of air by installing higher capacity auxiliary fan taking due to considerations of the air leakage and pressure loss in the duct.
- Using duct of suitable size to accommodate higher quantity of air supplied by the auxiliary fan and minimizes pressure loss.
- Enhancing the ventilation efficiency of the system either by increasing number of stages of the auxiliary fan or using two auxiliary fans and duct systems in parallel instead of using a single auxiliary fan with such a lengthier duct.
- Avoiding sharp bends in the ducting to minimize pressure losses.
- Improving the smoothness of tunnel wall and contour.

5.3.2 Conclusions

A computational study has been carried out to investigate flow behavior in an underground tunnel. Turbulence model, namely, k-Epsilon is compared with the experimental data from Kolkata metro. It is concluded that validated computational flow modelling can improve the fundamental understanding of the airflow patterns. The distribution characteristics of fluid movement and physical field such as velocity field and pressure field were simulated with CFD approach and computer techniques. The interpretation of the results produced by these CFD models will improve the planning and operation of ventilation systems, in order to improve the dilution and removal of any gas or dust liberated during the cutting operations. It has also been demonstrated that proper flow in different tunnel configuration design can enhance gas control throughout the tunnel whilst maintaining low pressure drop.

The following conclusions may be drawn from this thorough study of losses in tunnel ventilation system of a Kolkata metro and analysis of the field study data and CFD simulation results:

- The average velocity of air flowing in the Kolkata metro tunnel is varying in the range of 0.44 -0.70 m/s.
- The air velocity in tunnel should not fall below 0.15 m/s during construction (Anon, 1979).
- The pressure losses is varies from 0.012 to 0.23 (field study) 0.011 to 0.23(CFD simulation)
- But in other than Kolkata metro CFD losses is roughly match with field data except split junction. It is roughly matching because we are comparing with the straight tunnel field data for all cases.

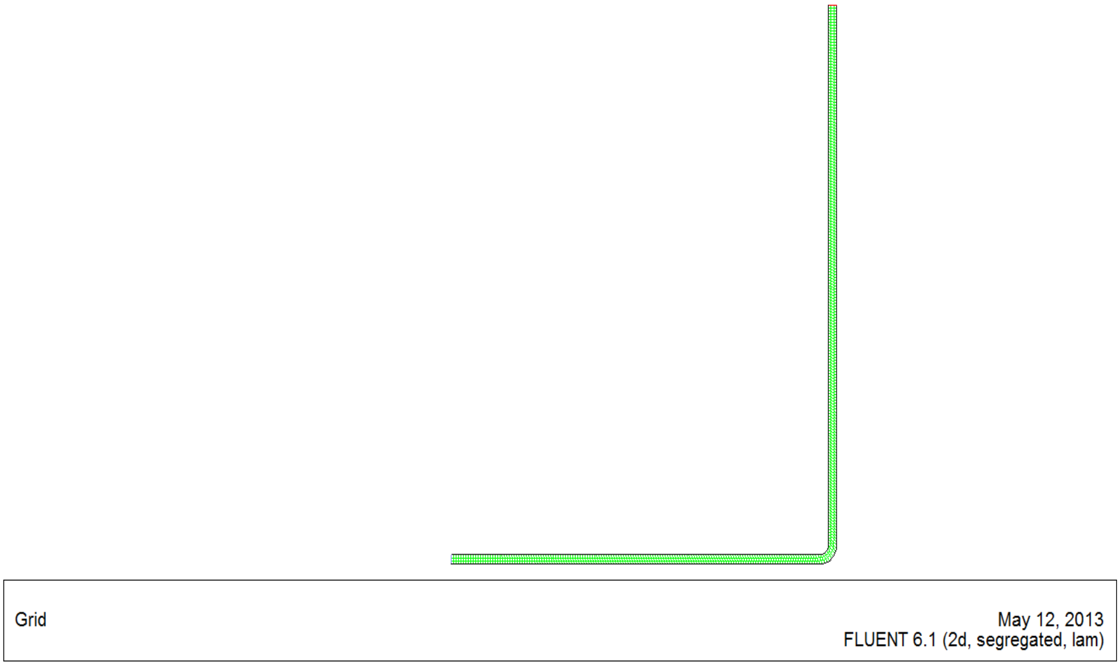


Figure 7. Grid of elbow bend

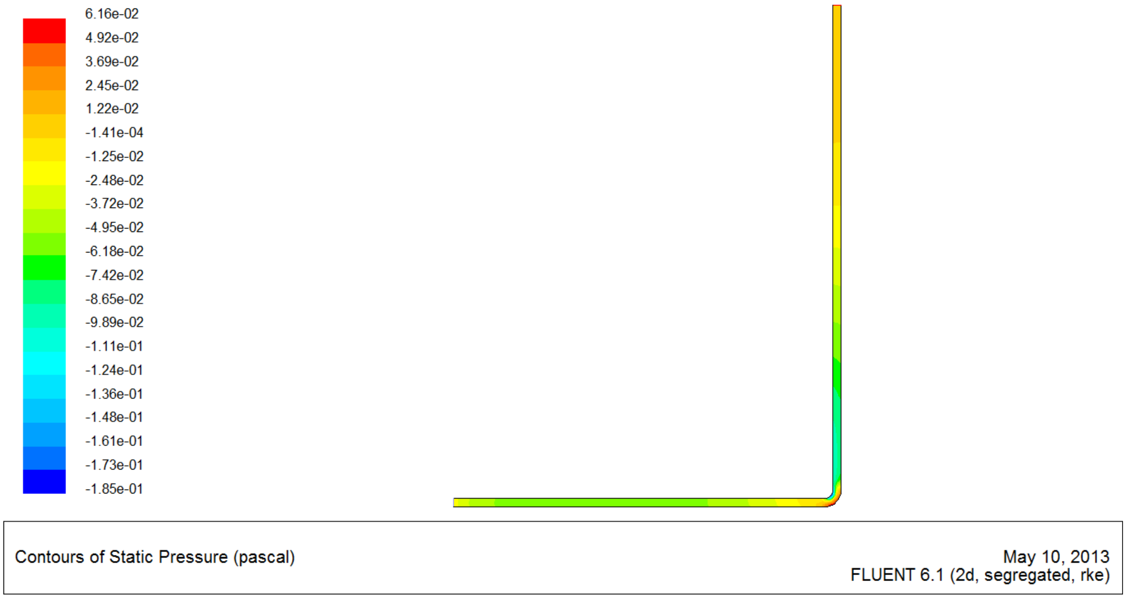


Figure 8. Contour of static pressure for elbow

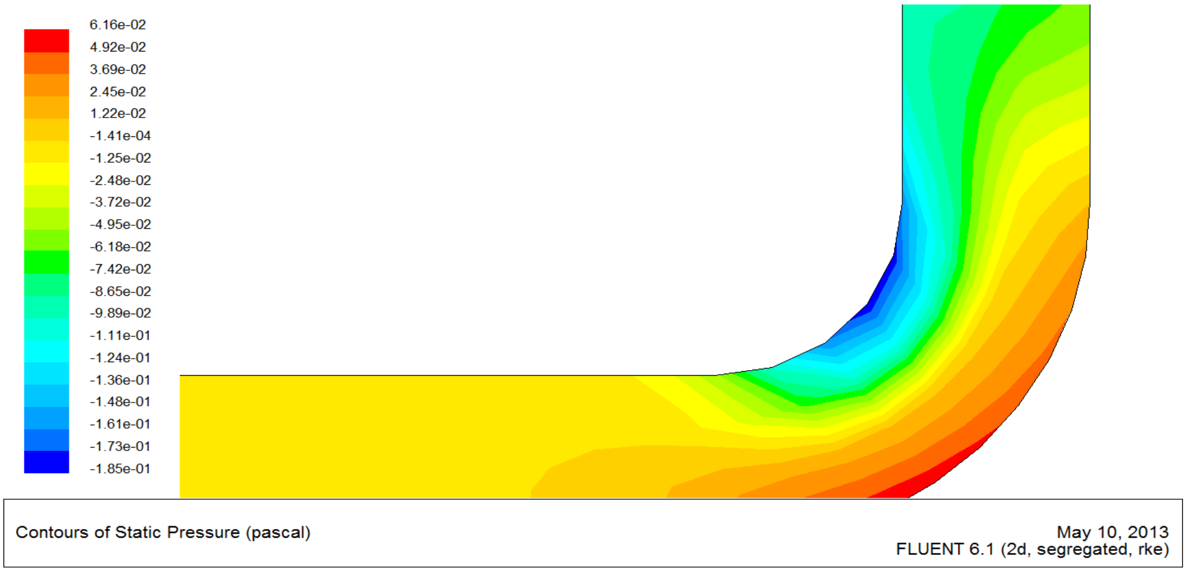


Figure 9. Contours of static pressure for elbow (enlarged view)

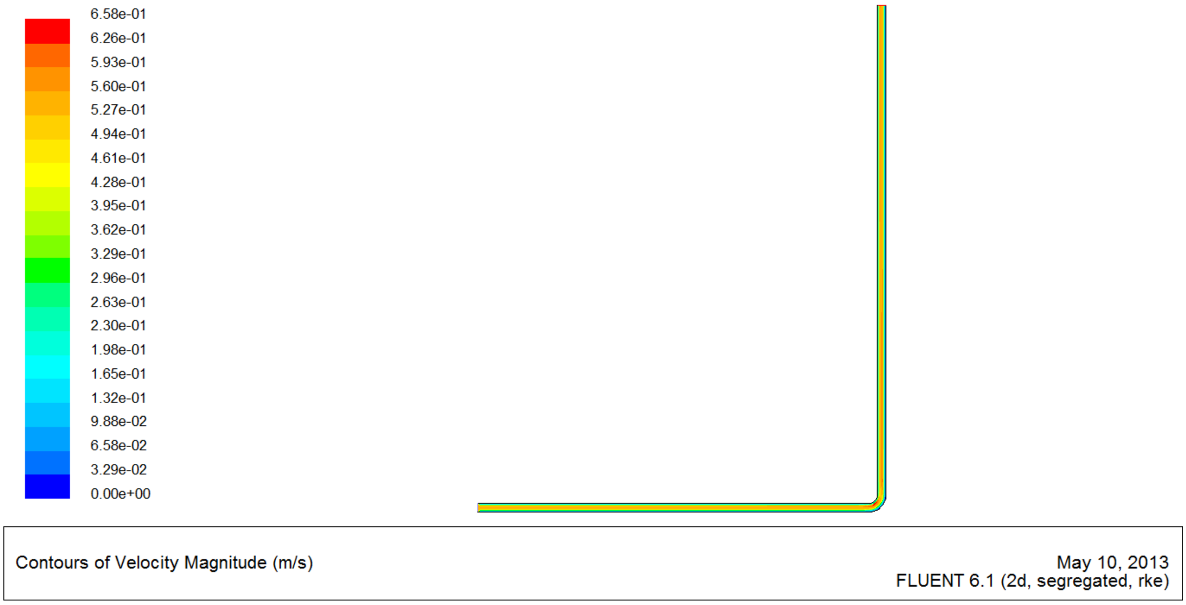


Figure 10. Contour of velocity magnitude for elbow

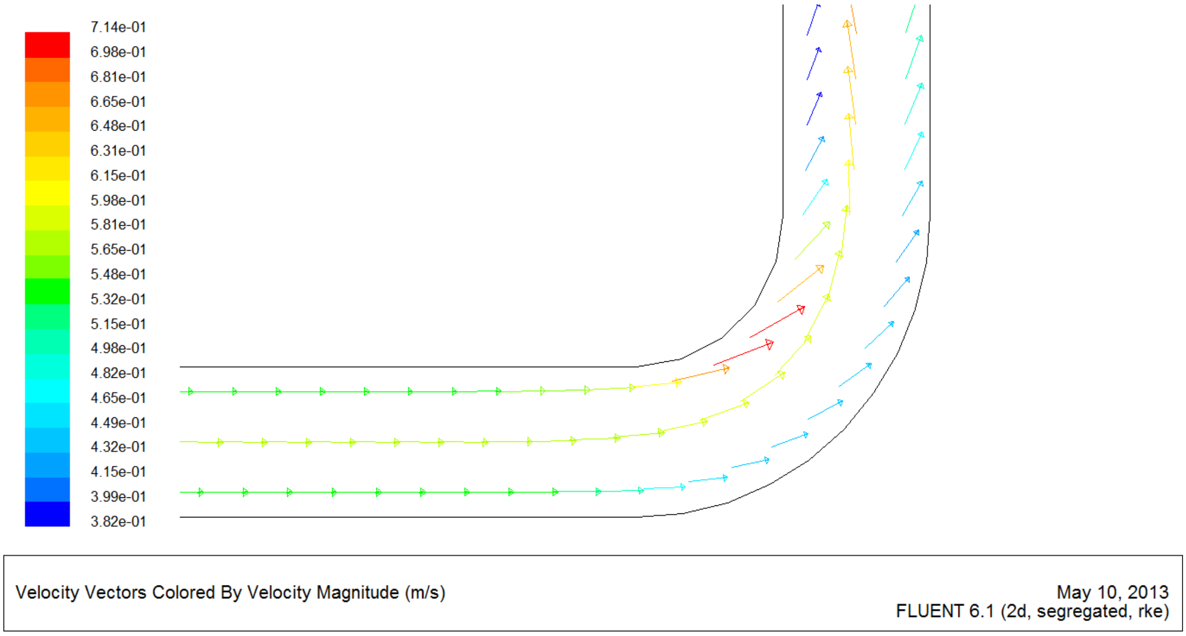


Figure 11. Velocity vectors for elbow (enlarged view)

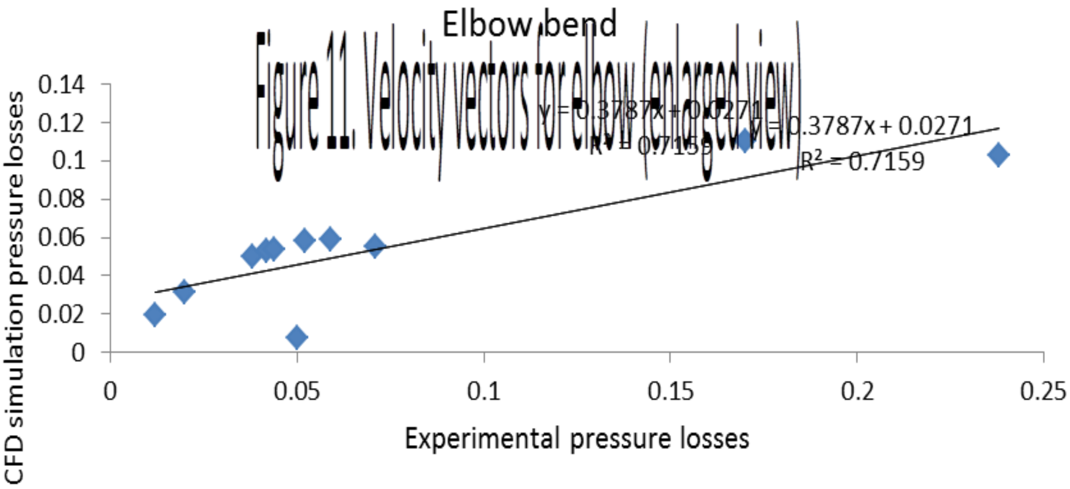


Figure 12. CFD vs. experimental pressure losses in elbow bend

Figure 13.Grid of sudden contraction

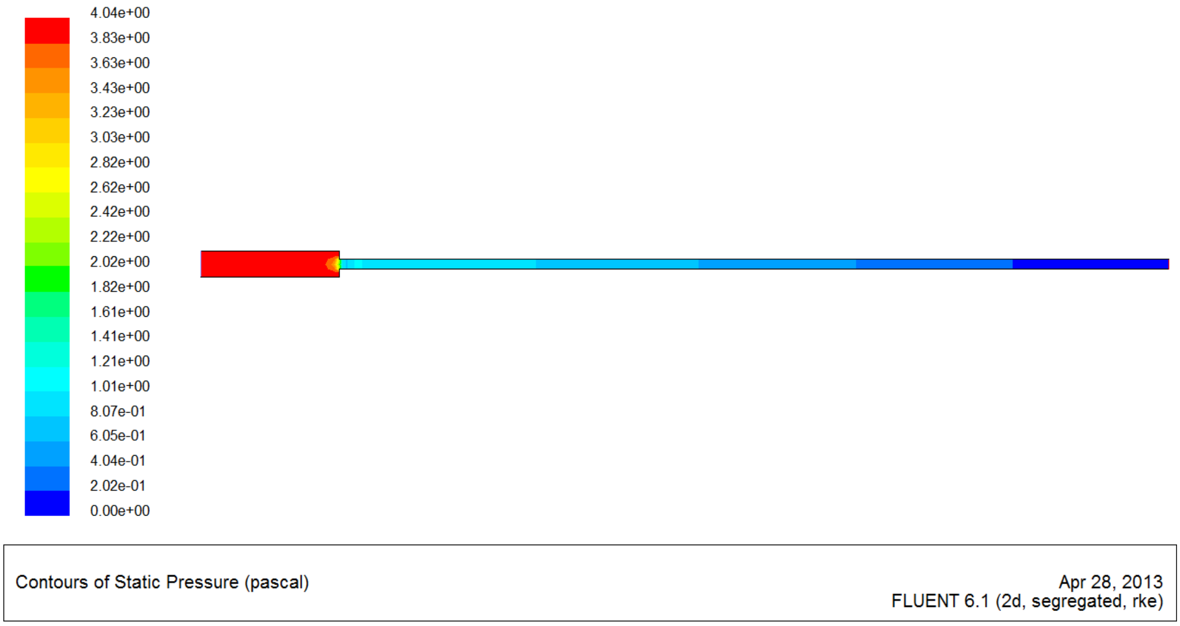


Figure 14.Contour of static pressure for sudden contraction

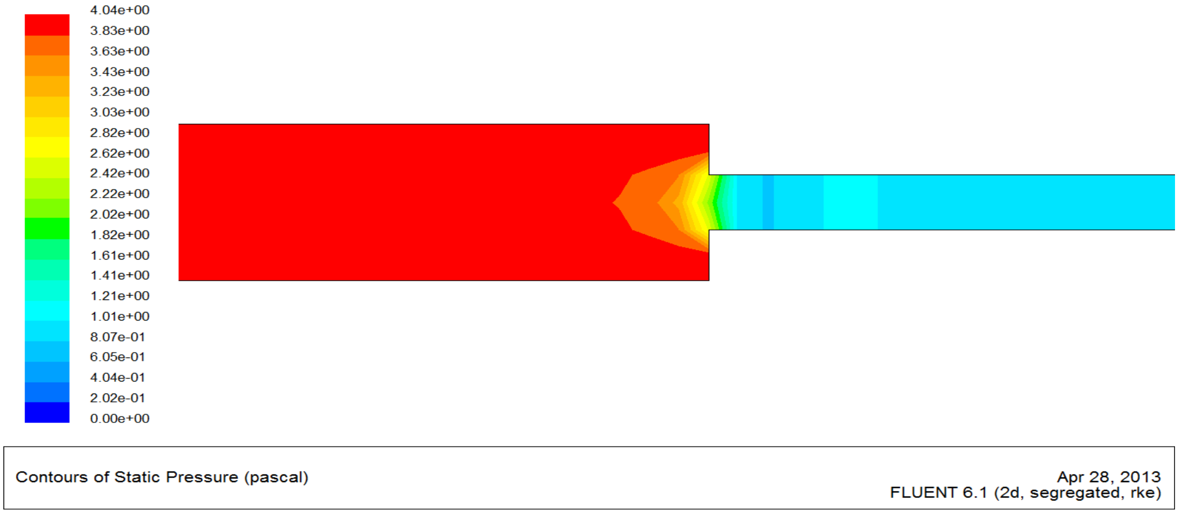


Figure15. Contours of static pressure for sudden contraction (enlarged view)

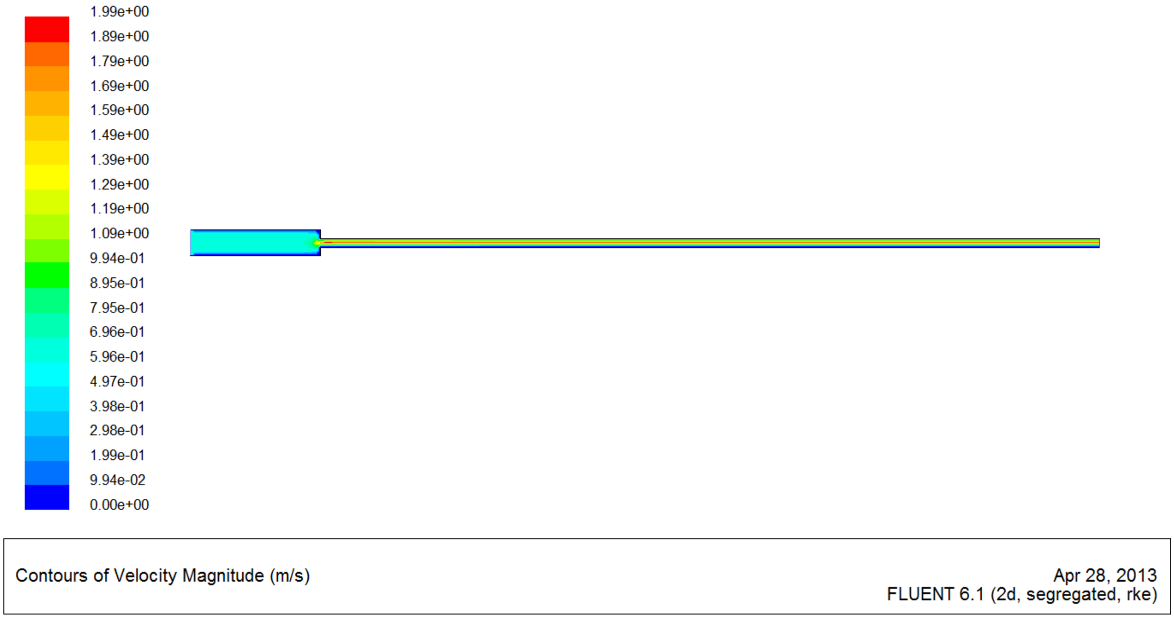


Figure 16.Contours of velocity magnitude for sudden contraction

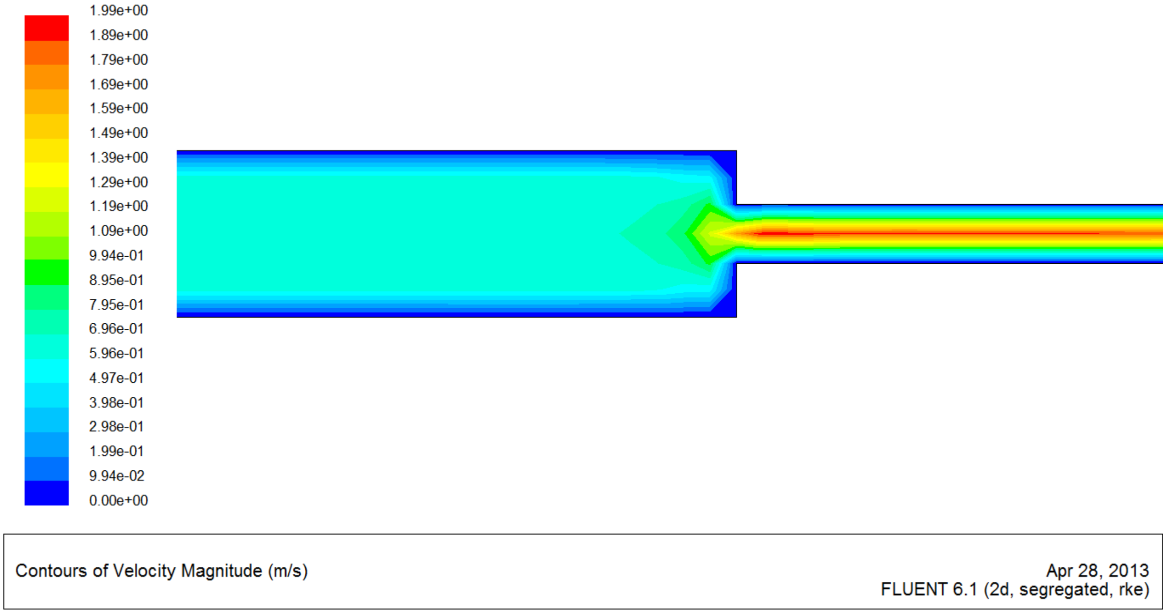


Figure 17.Contours of velocity magnitude for sudden contraction (enlarged view)

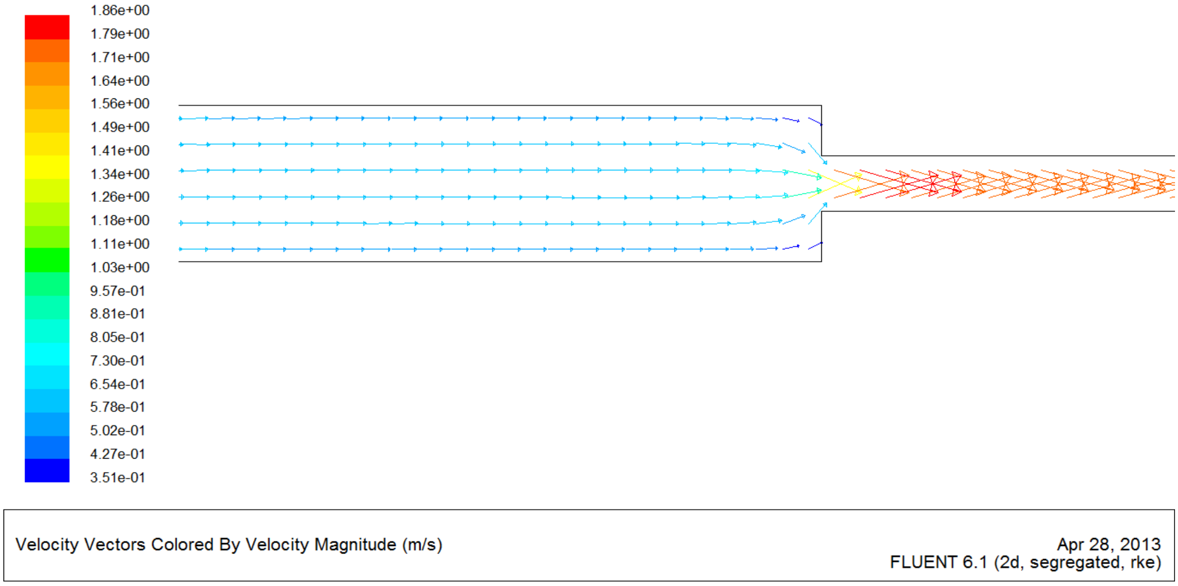


Figure 18.velocity vectors for sudden contraction (enlarged view)

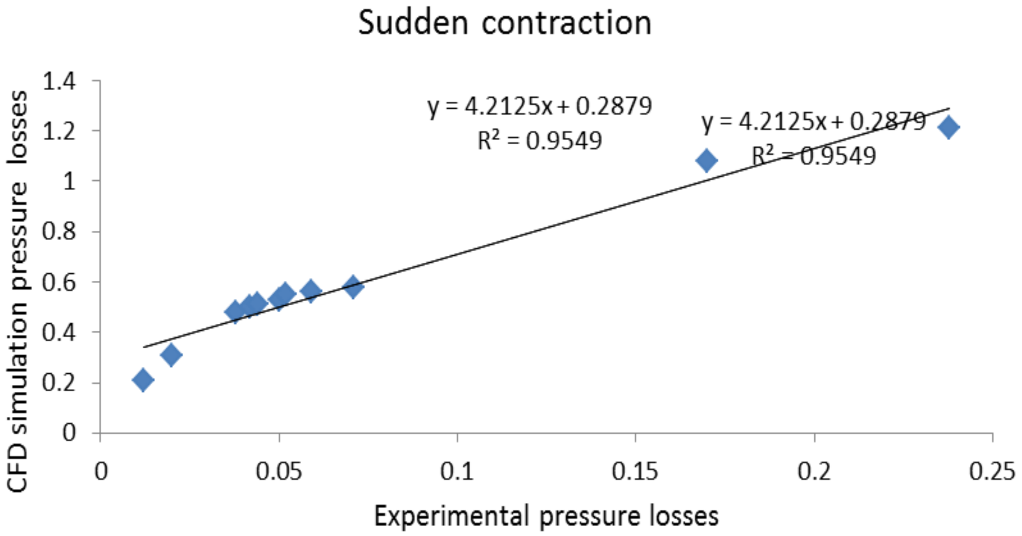


Figure 19. CFD vs. experimental pressure losses in sudden contraction

Figure 20. Grid of split tunnel

Figure 20. Grid of split tunnel

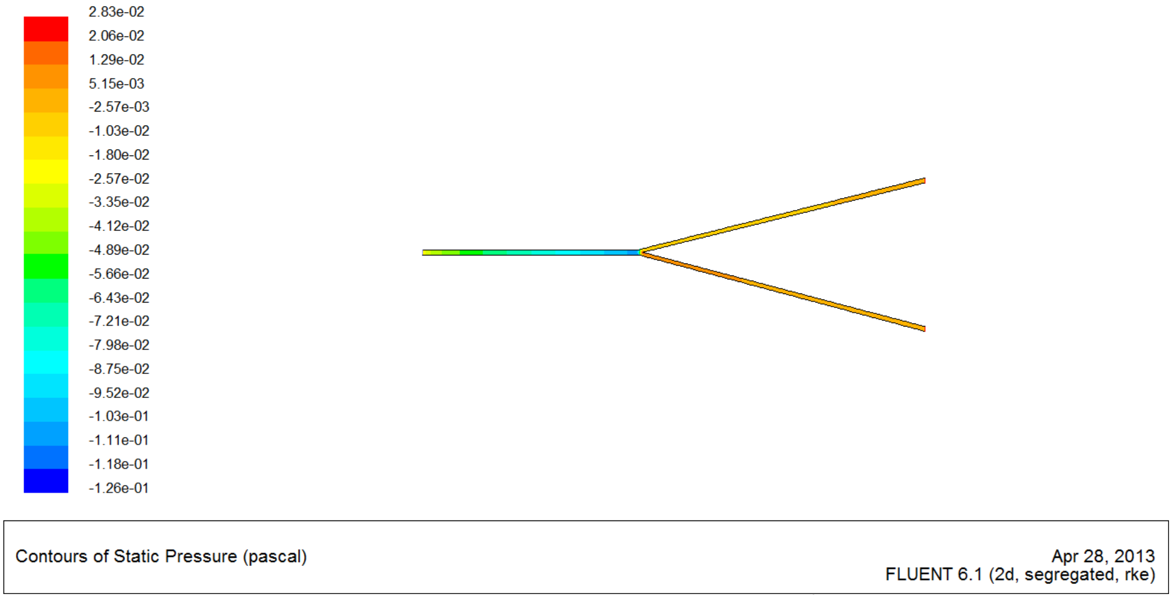


Figure 21. Contours of static pressure for 30^0split

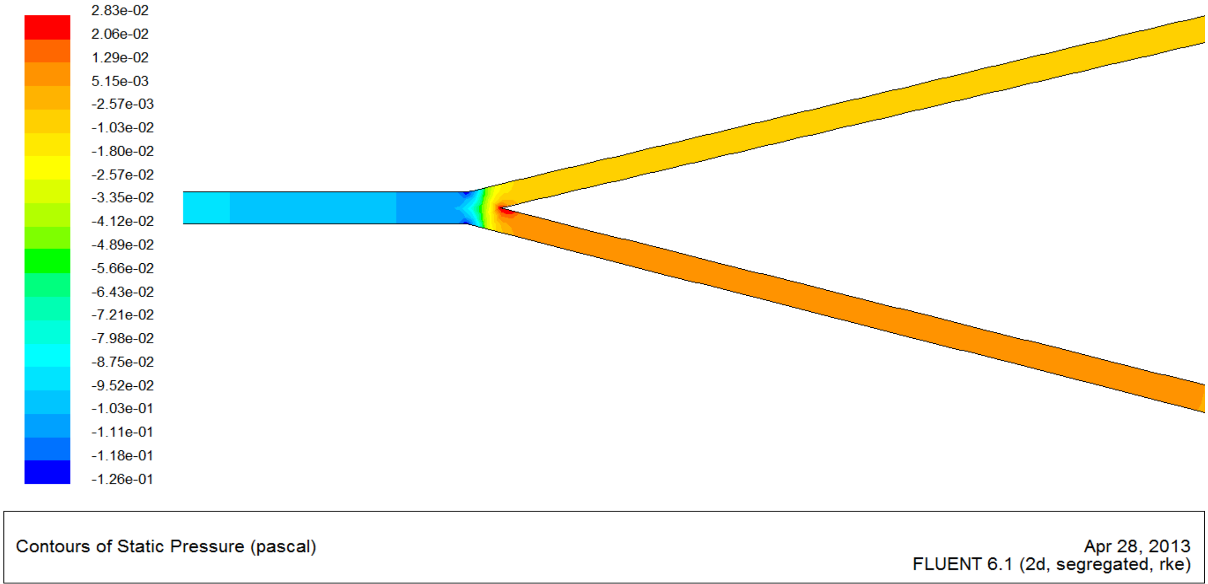


Figure 22. Contours of static pressure for 30^0 split (enlarged view)

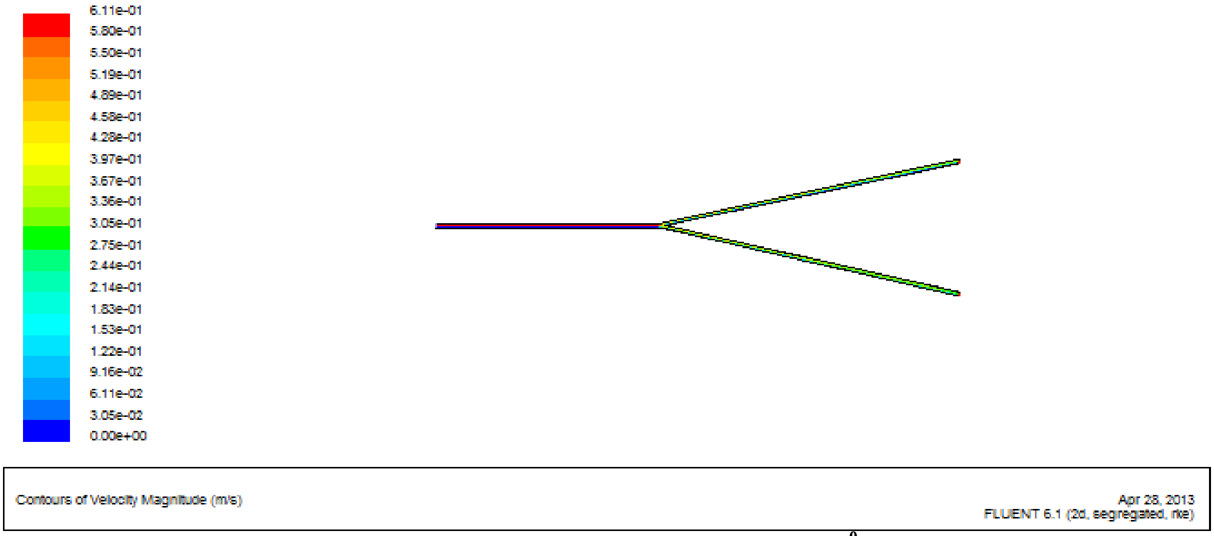


Figure 23.Contours of velocity magnitude for 30^0 split

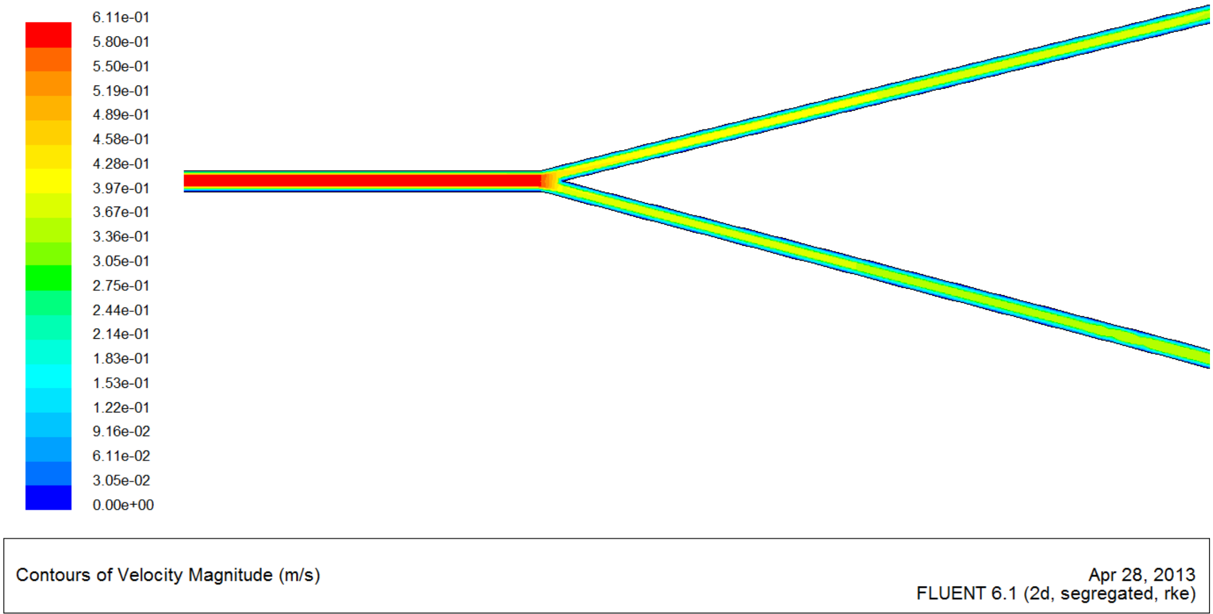


Figure 24.Contours of velocity magnitude for 30^0 split (enlarged view)

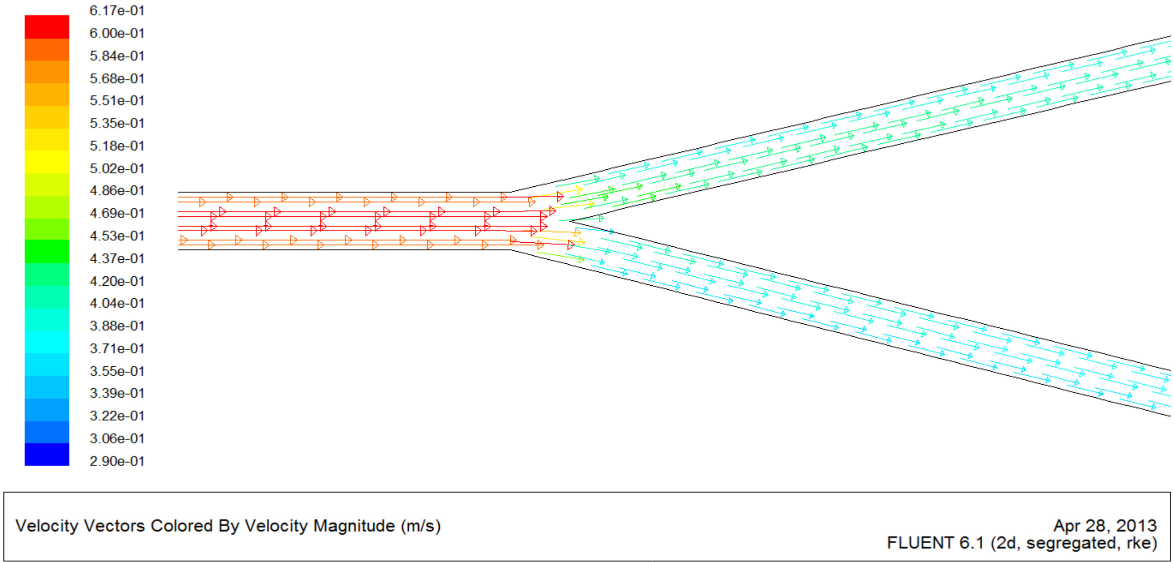


Figure 25.Velocity vectors for 30^0 split (enlarged view)

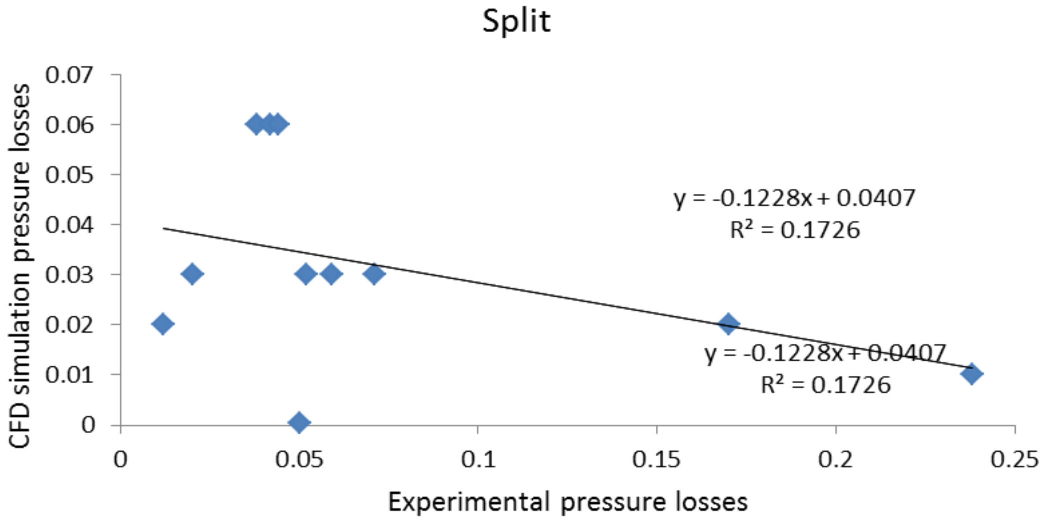


Figure 26. CFD vs. experimental pressure loss in split junction.

References

1.Abata D. L.Bunting, B.G., Johnsons, J.H., and Robb, J., 1978, “Monitoring of Gaseous Pollutants from Six Explosives Tested in an Underground Mine,” Paper Mo. 79-334 presented SME fall meeting, Tucson, AZ, October.

2.Anon., 1979, Code of Federal Regulations, Title 29, Occupational Safety and Health Administration, Department of Labor (see Federal Register, Vol. 44, No.29 U.S. Government Printing Office, Washington, February 9, pp. 8655-8670).

3.Constant, J., 1972, “The Ventilation of Road Tunnel-A General Approach to the Problem,” Tunnels and Tunneling (London), Vol. 4 No. 3 May-June, pp.213-223.

4. Cook, M.A. 1974, the Science of Explosives, IRECO Chemicals, Salt Lake City, UT.

5.Danziger, N.H., 1979, Private communication.

6.Metsch, W.W., 1976, “Meeting Design Needs to Control Subway Environment,” Specifying engr., Vol. 35 No. 3, March, pp. 86-91.

7.Pursall, B.R., and Powell, E.M.T., 1976, “Evaluation of the Design and Performance of Various Types of Ventilation System for a Number of Proposed Tunnels,” Proc. Second International symposium on Aerodynamics and Ventilation of Vehicle Tunnels, British hydromechanics Research Association, Crane field, Bedford, England, March 23-25, pp. G3-31 to G3-62.

8.Rodgers, S.J., 1976, “Analysis of Non coal mine Atmospheres,” USBM, Find Report, Contract S0231057 (21 May) and Contract S0231057, modification No. 5 (5 may)

9.Rodgers, S.J., 1977, “Analysis of Toxic Fumes Generated During Shot Firing,” USBM, Final Report, Contract J0166092, PB 280116, March.

10.Roehlich, F., Rodgers, S. J. Mausteller, J. W., 1973, “Air Pollution Control in Vehicular Tunnel,” Paper presented at First FCP Research Progress Review, Federal Highway Administration, U. S. Department of Transportation, San Francisco, September.

11.Rossi, B.D.,1971, “Control of Noxious Gases in Blasting Work and New Methods of Testing Industrial Explosives,” (in Russian), Translated by M. Tamari and R. Konder, NTIS, Tech. Translation, No. TT70-50163.

12.Szechy, K., 1996, The Art of Tunneling, Akademiaikiado, Budapest, Hungary.

13.http://www.howden.com/en/industries/tunnelventilation/default.htm.

14.Hartman, H.L., (1960) ‘pressure Changes in Splits and junctions in mine Ventilation Circuits,’ Tr. Am. Inst. Min. Met. Pet. Engrs. 217; 163.

15.Hinsley, F. B., (1950-51) ‘Natural and Mechanical Ventilation,’ Tr.I.M.E. 110;651

16.Mishra, G.B., (1964), Mine Ventilation, (Text Book)

17.Simulation of a longitudinal ventilation system: memorial tunnel case.

18.Tunneling and Underground Space Technology 23, 539–551.

19.Gribble, Don. 2009. Natural ventilation, harnessed by New Kingdom Egyptian tomb builders, may explain the changed floor levels in the Valley of the Kings tombKV5. Tunneling and Underground Space Technology 24, 62–65.

20.Potter, Merle C., Wiggert, David C., 2002. Mechanical de fluids. Thomson Mexico, p.277.

21.McPherson, M.J., 2007. Subsurface Ventilation and Environmental Engineering.

22.Heselden, A.J.M., 1976. Studies of fire and smoke behavior relevant to tunnels. In: Proceedings of Second International Symposium on the Aerodynamics and Ventilation of Vehicle Tunnels, BHRA Fluid Engineering, Cambridge, UK, pp. 23–25.

23.Hinkley, P.L., 1970. The flow of hot gases along an enclosed shopping mall: a tentative theory, Fire Research Note No. 807, Fire Research Station, Borehamwood, Herts, UK.

24.Laage, L., Pomroy, W., Weber, T., 1987. Computer modeling of the effect of mine-fire induced ventilation disturbances on stench fire warning system performance. US Bureau of Mines, BUMINESIC-9154, Twin Cities Research Centre, Minneapolis, MN, USA.

25.Lea, C.J., 1994. Computational modeling of mine fires. Mining Engineer (London) 154(394), 17–21.

26. Simcox, S., Wilkes, N.S., Jones, I.P., 1992. Computer simulation of the flows of hot gases from fire at King’s Cross Underground Station. Fire Safety Journal 18 (1), 49–82.

27.Pomroy, W.H., Carigiet, A.M., 1995. An analysis of underground coal mine-fire incidents in the United States from 1978 through 1992. In: Proceedings of the Seventh US Mine Ventilation Symposium, Lexington, KY, USA, June 5–7.

28.Leitner, A., 2001. The fire catastrophe in the Tauern tunnel: experience and conclusions for the Austrian guidelines. Tunneling and Underground Space Technology 16 (3), 217–223.

29.Burnett, J., 1984. Fire safety concerns for rail rapid transit systems. Fire Safety Journal 8(1), 3–7.

30.Chow, W.K., Li, J.S.M., 2001. Case study: vehicle fire in a cross-harbour tunnel in Hong Kong. Tunneling and Underground Space Technology 16 (1), 23–30.

31.Vauquelin, O., Megret, O., 2002. Smoke extraction experiments in case of fire in a tunnel. Fire Safety Journal 37 (5), 525–533.

32.Gao, P.Z., Liu, S.L., Chow, W.K., Fong, N.K., 2004. Large eddy simulations for studying tunnel smoke ventilation. Tunneling and Underground Space Technology 19 (6), 577–586.

33.Hu, L.H., Huo, R., Li, Y.Z., Wang, H.B., Chow, W.K., 2005. Full-scale burning tests on studying smoke temperature and velocity along a corridor. Tunneling and Underground Space Technology 20 (3), 223–229.

34.Miclea, P.C., 1991. Application of SES and MFIRE computer simulation programs for tunnel fire modeling—a comparative study. ICF, Kaiser Engineers Inc., Oakland, CA, USA.

35.Guelzim, A., Souil, J.M., Vantelon, J.P., Sou, D.K., Gabay, D., Dallest, D., 1994.Modelling of a reverse layer of fire-induced smoke in a tunnel. Fire safety science In: Proceedings of the Fourth International Symposium, pp. 277–288.

36.Laage, L., Greuer, R., Pomroy, W., 1995. MFIRE Users Manual ver. 2.20, Twin Cities Research Center, US Bureau of Mines, Minneapolis, MN, USA, US Bureau of Mines Training Workshop on the ‘‘MFIRE’’ mine fire and ventilation simulator.

37.Kennedy, W.D., Gonzales, J.A., Sanchez, J.G., 1996. Derivation and application of the SES critical velocity equations. ASHRAE Transactions: Research 102 (2), 4044.

38.Oka, Atkinson, Y., G.T., 1996. Control of smoke flow in tunnel fires. Fire Safety Journal25 (4), 305–322.

39.Dziurzynski, W., Nawrat, S., Roszkowski, J., Trutwin, W., 1997. Computer simulation of mine ventilation disturbed by fires and the use of fire extinguishers. In: Proceedings of the Sixth International Mine Ventilation Congress, Pittsburgh, PA, USA, May 17–22.

40.Chow, W.K., 1998. On smoke control for tunnels by longitudinal ventilation. Tunneling and Underground Space Technology 13 (3), 271–275.

41.Edwards, J.C., Hwang, C.C., 1999. CFD analysis of mine-fire smoke spread and reverse flow conditions. In: Proceedings of the Eighth US Mine Ventilation Symposium, Rolla, MO, USA, and June 14–17.

42.Wu, Y., Bakar, M.Z.A., 2000. Control of smoke flow in tunnel fires using longitudinal ventilation systems—a study of the critical velocity. Fire Safety Journal 35 (4), 363–390.

43.Cheng, L.H., Ueng, T.H., Liu, C.W., 2001. Simulation of ventilation and fire in the underground facilities. Fire Safety Journal 36 (6), 597–619.

44.Modic, J. 2003. Fire simulation in road tunnels. Tunneling and Underground Space Technology 18 (5), 525–530.

45.Kymar, S. and Cox, G., Radiant heat and surface roughness effects in the numerical modeling of tunnel fires, 6th int. Symposium Aerodynamics and Ventilation of Vehicle Tunnel, Dueham UK, BHRA, pp.515-526 (1988).

46.McKinney D., Brunner D., M. Deng, Miclea P. C. ICF Kaiser Engineer, Inc. Critical velocity of air versus detailed modeling using computational fluid dynamics technique in tunnel fires, APTA presentation (1994).