EXPERIMENTAL ANALYSIS AND CFD SIMULATION OF MINOR LOSSES IN PIPES

A thesis submitted in partial fulfilment of the requirement for
The award of the degree of

BACHELOR OF ENGINEERING

IN

MECHANICAL ENGINEERING

Submitted by

R. MOUNIKA	**(314126520134)**
REETHU.P	**(314126520117)**
P. MURALI KRISHNA	**(314126520122)**
M.SRAVAN KUMAR	**(314126520101)**
SAHIL ALAM KHAN	**(314126520140)**

Under the guidance of

Dr. RAJESH GHOSH, B.E., M.Tech., Ph.D.

Associate Professor

DEPARTMENT OF MECHANICAL ENGINEERING

ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY & SCIENCES

(Permanently affiliated to Andhra University, Approved by AICTE, and Accredited by NBA & NAAC with 'A' grade)

Sangivalasa - 531162, Bheemunipatnam (Mandal), Visakhapatnam (Dist.),

Andhra Pradesh, India.

2018

ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY & SCIENCES

(Affiliated to Andhra University, Approved by AICTE, Accredited by NBA & NAAC with A grade)

SANGIVALASA, VISAKHAPATNAM (District) – 531162

PRAGNANAM BRAHMA

ANITS

CERTIFICATE

This is to certify that the Project Report entitled "**EXPERIMENTAL ANALYSIS & CFD SIMULATION OF MINOR LOSSES IN PIPES**" being submitted by RAPETI MOUNIKA (314126520134), PARAVADA REETHU (314126520117), PICHIKA MURALI KRISHNA (314126520122), SAHIL ALAM KHAN (314126520140), MULAKA SRAVAN KUMAR (314126520101) in partial fulfillments for the award of degree of **BACHELOR OF TECHNOLOGY** in **MECHANICAL ENGINEERING** of **ANDHRA UNIVERSITY**. It is the work of bona-fide, carried out under the guidance and supervision of **Dr. Rajesh Ghosh**, Associate Professor, Department Of Mechanical Engineering, ANITS during the academic year of 2014-2018.

Approved By

PROJECT GUIDE

HEAD OF THE DEPARTMENT

(Dr. Rajesh Ghosh)
Associate Professor
Mechanical Engineering Department
ANITS, Visakhapatnam.

(Dr. B. Naga Raju)
Head of the Department
Mechanical Engineering Department
ANITS, Visakhapatnam.

PROFESSOR & HEAD
Department of Mechanical Engineering
ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY & SCIENCES
Sangivalasa-531 162 VISAKHAPATNAM Dist A.P.

THIS PROJECT IS APPROVED BY THE BOARD OF EXAMINERS

INTERNAL EXAMINER:

Dr. B. Naga Raju
M.Tech,M.E.,Ph.d
Professor & HOD
Dept of Mechanical Engineering
ANITS, Sangivalasa,
Visakhapatnam-531 162.

EXTERNAL EXAMINER:

ACKNOWLEDGEMENT

On the submission of our project report entitled **"Experimental Analysis and CFD Simulation of Minor Losses in Pipes"** we would like to give our heartiest thanks and gratitude to **Dr.Rajesh Ghosh**, Associate Professor, Department of Mechanical Engineering, Anil Neerukonda Institute of Technology & Sciences, for his continuous motivation, constant support and guidance throughout the past year.

We are very thankful to **Prof.T.V.HanumanthaRao**, Principal and **Prof.B.Nagaraju**, Head of the Department, Mechanical Engineering, Anil Neerukonda Institute of Technology & Sciences for their valuable support and facilities.

We would like to thank the technical staff of fluid machinery lab for their continuous cooperation and their guidance in helping us to understand the technical details of hydraulic machines in the lab.

Finally, we would like to convey our thanks to everyone, who have contributed directly or indirectly for the completion of this project work.

PROJECT ASSOCIATES

R. MOUNIKA	(314126520134)
REETHU.P	(314126520117)
P. MURALI KRISHNA	(314126520122)
M.SRAVAN KUMAR	(314126520101)
SAHIL ALAM KHAN	(314126520140)

ABSTRACT

This project **"Experimental Analysis and CFD Simulation of Minor Losses in Pipes"**, deals with the frictional losses produced in a pipe due to shear stress and viscosity of fluids. This paper contains flow analysis of fluid in different pipe geometry. It focusses on the losses in piping systems, as working fluid through pipes plays an important role in functionality of industries like chemical industries, petroleum industries etc. Whenever there is need of transferring fluids in major piping systems we come across many obstacles such as elbow-junctions, bends, contractions, expansions. All this together affects the overall efficiency by causing major and minor losses in pipes.

The experimental analysis is done by considering the continuity equation. By varying the pipe geometry and flow parameters, the velocity at inlet and outlet of the pipe is calculated. This velocity is used to calculate the coefficient of loss for pipe.

The purpose of this project work is to investigate the steady, incompressible fluid flow and to get familiarize with CFD. The simulations were done using ANSYS FLUENT CFD 14.5 software to observe the effect of changes in velocity of flow, drop in pressure and effect of static pressure, dynamic pressure and stream flow due to change in geometry.

In this project work, analysis of results was done and the results obtained in experiment on different pipe geometry were compared to be closely conforming to the results of ANSYS.

LIST OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

LIST OF GRAPHS

NOMENCLATURE

h_f = head loss due to friction in unit of length

f = friction factor

D = Pipe Diameter

V = Flow velocity

C= Chezy's constant

m=hydraulic mean depth

t = time taken

K = Loss coefficient

h_e = Loss in head due to expansion

h =Manometer Difference

Q = Discharge

V_1=velocity at D_1

V_2=velocity at smaller diameter

k_C=co-efficient of loss due to contraction

k_b=co-efficient of loss due to expansion

k_e=co-efficient of loss due to expansion

CHAPTER 1

INTRODUCTION

1.1 GENERAL:

Pipe network is very common in industries throughout the country, where fluid and gases are transported from one point to another. The pressure loss depends on the type of flow of the fluid in the network, pipe material, and the fluid flowing through the pipe. When any fluid flows through a pipe, the velocity adjacent to the pipe wall is zero and the velocity gradually increases from the wall. Maximum velocity is observed at the centre of the pipe. Due to increase in the velocity gradient, shear stresses are produced in the fluid due to its viscosity. This viscous action attributes to loss of energy which is commonly known as loss due friction or frictional loss.

William Froude stated the following laws of fluid friction under turbulent flow.

For a turbulent flow, frictional resistance is:

1. Directly proportional to V^n, where n varies between 1.5 and 2.
2. Proportional to fluid density.
3. Proportional to surface area in contact.
4. Independent of the pressure
5. Dependent on the nature of the surface in contact.

If losses are minute in a pipe network then the efficiency is higher. Moreover, all networks should be designed to undergo minimum loss.

1.2 OBJECTIVES:

- Calculate the minor losses (due to sudden expansion, sudden contraction and bend) in lab and find the co-efficient of loss for their geometry.
- Modelling of different pipe geometry like elbow, sudden enlarge, sudden contract pipe etc. in ANSYS software.
- Simulation of fluid flow through these pipes.
- Calculation of minor losses with the help of ANSYS.
- Comparison of ANSYS obtained results with experimental obtained results.

1.3 METHODOLOGY

This project can broadly be divided into the following stages.

1) Identifying the problem statement and formulating objectives.
2) Preparation for project:
    a. This includes all preparatory things like literature review, data collection from laboratory etc.
    b. Laboratory practical that are to be undertaken for this project are frictional losses in pipes of different geometry.
    c. Various models of pipes are to be modelled in ANSYS Software for the analysis and comparison of the results from laboratory and ANSYS.
3) Optimization of result:
    a. Flow analysis for fluid flowing through different pipe geometry using data obtained from practical, theoretical and ANSYS methods.
    b. Comparison between ANSYS and experimental results.

CHAPTER 2

LITERATURE REVIEW

Ackeret et al [1] discussed special features of internal flow. He concluded that there is a predominant role played by the equation of continuity, especially if compressibility is involved. If the width of the duct is not growing too fast along its length, separation is followed by reattachment. He observed that in case of internal flow also, three-dimensional boundary layers can appear as in external flow. We have applied equation of continuity to pipes of different geometry when fluid is flowing through it.

Celata et al [2] investigated the possibility of wall roughness effects and geometric deviations for micro tubes ranging from 31 to 326 micro meters. The intent was to model how accurately fluid flow behaved in accordance with the classical Hagen-Poiseuille flow for different diameter micro tubes, and to possibly see around what size deviation from this accepted flow model occurred. An uncertainty analysis was carried out for the Darcy equation, and a slip parameter was incorporated into the laminar velocity profile equation to extrapolate a modified Darcy equation. By this we have simulated the pipes having different geometry.

Hager and Dupraz et al (1985) [3] derived a theoretical equation for obtaining the coefficient of contraction in terms of the contraction ratio, the inlet angle of the contraction and the length ratio of the contracted reach. The flow conditions were those of transitional flow from subcritical to supercritical passing through critical at the minimum depth point through the contraction length. They verified their expression experimentally. Based on this, we have calculated loss coefficient by conducting experiment on different pipes and compared the results with ANSYS results.

Laursen et al (1970) [4] studied the contraction coefficient at sudden expansion at bridge locations. Four distinct flow zones (accretion, contraction, expansion and abstraction) were identified and discussed. It was found that the contraction coefficient varies between 0.7 for about 30% contraction ratio and 1.0 for no contraction. The use of different constrictions for peak discharge measurement by indirect methods was discussed by Matthi (1976) and was outlined in French (1986). We have calculated the loss coefficient and had observed the variations by considering different pipe geometry.

Kindsvater, Carter and Lacy et al (1953) and Kindsvater and Carter et al (1955) [5] carried out an experimental investigation to address the effects of different types of contractions on discharge characteristics. Formica (1955) tested experimentally the various design for channel transition (contraction and expansion). The main results of Formica work are reported in Chow (1959). Basing on this we have conducted the experiment for different discharge for a certain volume.

Rathakrishnan et al and Sreekanth et al [6] studied flows in pipe with sudden enlargement. They concluded that the non-dimensional base pressure is a strong function of the expansion area ratios, the overall pressure ratios and the duct length-to-diameter ratios. They showed that for a given overall pressure ratio and a given area ratio, it is possible to identify an optimal length-to-diameter ratio of the enlargement that will result in maximum exit plane total pressure at the nozzle exit on the symmetry axis (i.e. minimum pressure loss in the nozzle) and in a minimum base pressure at the sudden enlargement plane. By this study, we have come to know that as the diameter of the pipe changes for a given length, pressure variation is observed.

Wick et al [7] has studied the effect of boundary layer on sonic flow through an abrupt cross-sectional area. He observed experimentally that the pressure in the corner of expansion was related to the boundary layer type and thickness upstream of the expansion. He considered boundary layer as a source of fluid for the corner flow. Based on this concept of boundary layer, we have observed the variations in velocity from the centre of the pipe to the extreme walls. At the centre, the velocity is found to be maximum. Due to the relative motion between the fluid molecules, a decrease in the velocity is observed from the centre to pipe walls. At the pipe wall, the fluid molecules come to rest due to the direct contact between fluid molecules and pipe wall. The fluid layer next to this has a velocity nearer to zero and it thereby varies from layer to layer.

Mandal et al. (2008) [8] examined the shape and stability of Taylor bubbles and Taylor drops in liquid-liquid systems. They noted the effect of tube diameter and inclination on the shape and velocity of the Taylor bubble and drop. They have reported that the velocity of both Taylor bubble and drop increases with increase in tube diameter. From this journal we have concluded to note the variation of velocity at different diameters.

Rodriguez et al. (2009) [9] studied the frictional pressure drop encountered during horizontal and vertical core flow. They have used both viscous (ρ = 925 kg/m3 and μ = 0.5 Pa-s) as well as ultra-viscous crude oil (ρ = 972.1 kg/m3 and μ = 36.95 Pa-s) in two pipes of diameter 0.0284 m and 0.077m for their study. They have noticed a reduction in frictional pressure with addition of water in all the cases. Further they have modified the model proposed by Parda and Bannwart (2001) to predict frictional pressure gradient. Based on this we have determined pressure drop at different pipe sections. And using this we have calculated the loss coefficient.

Vallentine et al [10] Sixth International Water Technology Conference, IWTC 2001, Alexandria, Egypt (1958) investigated the effect of pipe contraction placed normal to channel axis. His observations covered data that include different diameters of flow. In practical applications we come across different pipes. Basing on this journal we have taken different pipe geometry and found out the geometry of pipe which gives minimum frictional loss.

Delhaye et al (1981), Wadle et al (1989) Schmidt and Friedel et al (1997), Guglielmini et al (1997), Fossa and Guglielmini et al (2002) [11] Researchers have proposed different models for predicting pressure drop in expansion. Some studies are also devoted on the effect of area change on flow regimes (Fossa et al. (2006), Ahmed et al. (2007), Ahmed et al. (2008), Chen et al. (2009). Extracting from this article we have determined pressure drop at different sections when fluid is flowing through pipe.

Timothy J. Rennie et al [12] studied the heat transfer characteristics of a double pipe helical heat exchanger for both counter and parallel flow. Both the boundary conditions of constant heat flux and constant wall temperature were taken. The study showed that the results from the simulations were within the range of the pre-obtained results. For dean numbers ranging from 38 to 350 the overall heat transfer coefficients were determined. The results showed that the overall heat transfer coefficients varied directly with the inner dean number but the fluid flow conditions in the outer pipe had a major contribution on the overall heat transfer coefficient. The study showed that during the design of a double pipe helical heat exchanger the design of the outré pipe should get the highest priority in order to get a higher overall heat transfer coefficient. We have conducted experiment on different pipe geometry to find the coefficient of loss and compared the results with ANSYS results.

J.S. Jayakumar et al [13] observed that the use of constant values for the transfer and thermal properties of the fluid resulted in inaccurate heat transfer coefficients. Based on the CFD analysis results a correlation was developed in order to evaluate the heat transfer coefficient of the coil. In this study, analysis was done for both the constant wall temperature and constant wall heat flux boundary conditions. The Nusselt numbers that were obtained were found to be highest on the outer coil and lowest in the inner side. Various numerical analyses were done so as to relate the coil parameters to heat transfer. The coil parameters like the diameters of the pipes, the Pitch Circle Diameters have significant effect on the heat transfer and the effect of the pitch is negligible.

G.Satish, K.Ashok Kuma, V. Vara Prasad, S k.M.Pasha [14]studied in their paper about the flow through sudden and gradual change of pipe diameter (enlargement and contraction) was numerically simulated with water by unsteady flow in k-epsilon scheme. The major observations made related to the pressure and velocity contours in the process of flow through these pipes. Sudden enlargement creates more severe formation of flow eddies than sudden contraction. Also, the losses are more at the point where the enlargement in the pipe begins. In the sudden contraction, vena-contracta’s are formed at the point of contraction and effect of viscosity is negligible on the pressure drop through sudden contraction.

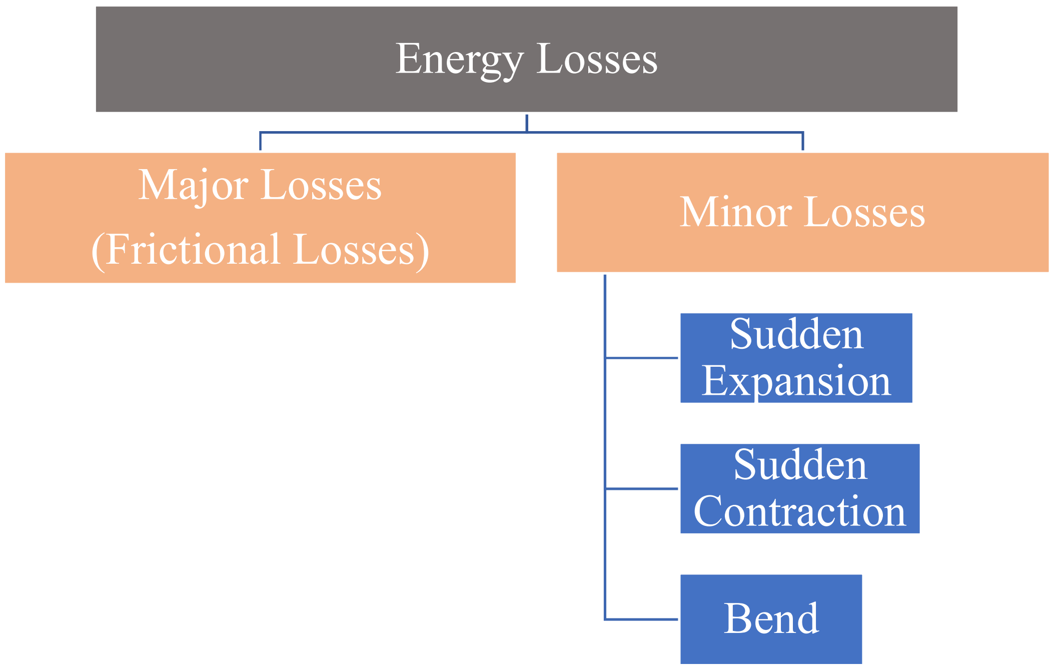
Wan Kai, Wang Ping [15] studied in their paper about Using standard k-ε model with FLUENT software on large diameter CFD numerical simulation of air flow in a 90 ° bent tube. The standard k-ε model belongs to the eddy viscosity model, which adopts closed RANS equations to solve the model. Assuming the air flow rate of 15 m/s the continuous and stable manner flows through the elbow. As the flow rate is small, it can be considered incompressible fluid. By homogenization of the continuity equation and instantaneous Navier-Stokes equations, the Cartesian coordinate system under adiabatic, steady, incompressible fluid flow is governed by the control equation. Basing on this paper we have simulated to 90 degrees bent pipe.

CHAPTER 3

THEORETICAL ANALYSIS

3.1 LOSSES IN PIPES:

When a fluid flows through a pipe, the fluid experiences some resistance, due to which there are some losses in the energy of fluid.



This loss of energy is classified as:

1. Major Energy losses (Friction loss)

2. Minor losses
    - Sudden expansion
    - Sudden contraction
    - Bend in Pipe

3.2 MAJOR LOSS IN PIPE (ENERGY LOSS DUE TO FRICTION):

Friction loss is the loss of energy or “head” that occurs in pipe flow due to viscous effects generated by the surface of the pipe. Friction Loss is considered as a "major loss" and it is not to be confused with “minor loss” which includes energy lost due to obstructions. In mechanical systems such as internal combustion engines, it refers to the power lost in overcoming the friction between two moving surfaces.

This energy drop is dependent on the wall shear stress (τ) between the fluid and pipe surface. The shear stress of a flow is also dependent on whether the flow is turbulent or laminar. For turbulent flow, the pressure drop is dependent on the roughness of the surface, while in laminar flow, the roughness effects of the wall are negligible. This is due to the fact that in turbulent flow, a thin viscous layer is formed near the pipe surface which causes a loss in energy, while in laminar flow, this viscous layer is non-existent.

Friction loss has a few reasons, including:

- Frictional losses rely on upon the states of flow and the physical properties of the system.
- Movement of fluid atoms against one another.
- Movement of fluid atoms against within surface of a channel.
- Bends wrinkles, and other sharp turns in hose or channelling.

In channel flows the losses because of contact are of two types: skin-rubbing and structure grinding. The former is because of the roughness of the inward a piece of the channel where the fluid interacts with the pipe material, while the latter is because of obstructions present in the line of flow maybe a curve, control valve, or anything that changes the course of movement of the flowing fluid.

1. **Darcy-Weisbach Formula:**

In many practical engineering applications, the fluid flow is more rapid, therefore turbulent rather than laminar. Under turbulent flow, the friction loss is found to be roughly proportional to the square of the flow velocity and inversely proportional to the pipe diameter, that is, the friction loss follows the phenomenological Darcy–Weisbach equation. It is a method to calculate friction loss resulting from fluid motion in pipes is by using the Darcy-Weisbach Equation. For a circular pipe:

$$h_i = \frac{flv^2}{2gD}$$

Where,

h_i=Head loss due to friction in unit of length

f= friction factor

D = Pipe Diameter

V = Flow velocity

2. **Chezy's formula:**

In fluid dynamics, the Chezy's formula describes the mean flow velocity of steady, turbulent open channel flow:

$$v = C\sqrt{Ri}$$

Where,

v is average velocity [m/s],

C is Chezy's coefficient [$m^{1/2}$/s],

R is the hydraulic radius (~ water depth) [m], and

i is the bottom slope

3.3 MINOR LOSSES IN PIPE:

Minor losses in pipes come from changes and components in a pipe system. This is different from major losses because those come from friction in pipes over long spans. If the pipe is long enough the minor losses can usually be neglected as they are much smaller than the major losses. Even though they are termed “minor”, the losses can be greater than the major losses, for example, when a valve is almost closed the loss can be almost infinite or when there is a short pipe with many bends in it. There are three types of forces that contribute to the total head in a pipe, which are elevation head, pressure head, and velocity head. Minor losses are directly related to the velocity head of a pipe, meaning that the higher the velocity head there is, the greater the losses will be. Units for minor losses are in length, such as feet or meters, the same as any of the three types of head. A separate head loss coefficient, k, can be determined for every element leading to minor losses. K is a dimensionless parameter to help determine head loss. The coefficient is then multiplied by the velocity head to get the head loss as shown below,

Head loss = head loss coefficient × velocity head

$$h = k \times \frac{v^2}{2g}$$

Where,

h is the head loss

k is the loss coefficient.

v is the velocity

g is the acceleration due to gravity

Each, geometry of pipe entrance has an associated loss coefficient.

The minor loss of energy (or head) happens in the following cases:

1. Loss of head due to bend in the pipe.
2. Loss of head due to sudden expansion.
3. Loss of head due to contraction.
4. Loss of head due to different pipe fitting.
5. Loss of head due to entrance of a pipe.

3.3.1 Loss of head due to sudden expansion:

Expansions are defined when the flow in a pipe goes from a small area to a larger area and the velocity slows down. It is the exact opposite for contractions, the flow goes from a larger pipe to a smaller one and the velocity increases. The loss or energy is due to turbulence, or eddies, formed at the point where the pipe sizes change.

Because of sudden change in diameter across the pipe from D_1 to D_2, the fluid flowing through the pipe is not fit to the unexpected change of the boundary. Thus, the flow separates from the boundary and turbulent eddies are formed as indicated in fig 3.1. The loss of head happens because of the creation of these eddies.

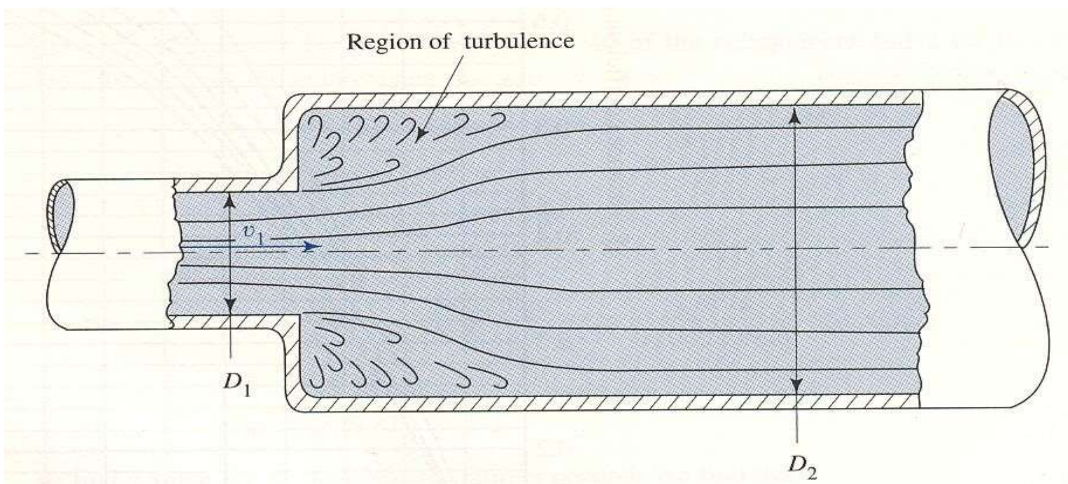


Fig. 3.1 Sudden expansion

Loss of energy is a result of turbulence. Measure of turbulence relies upon the difference in the pipe diameters.

Head loss,

$$h_e = k_e \frac{v_1^2}{2g}$$

Where,

h_e = loss in head due to expansion.

V_1=velocity at D_1

3.3.2 Loss due to sudden contraction:

Sudden contractions are defined when the area of the pipe diameter reduces suddenly along the length of the channel (at the 90-degree plot). The downstream velocity will be higher than the upstream velocity. The streamlines cannot follow the abrupt change of geometry and hence gradually converge from an upstream section of the larger tube. However, immediately downstream of the junction of contraction of area, the cross-sectional area of the stream tube becomes the minimum and less than that of the smaller pipe. This section of the stream tube is known as vena-contracta, after which the stream widens again to fill the pipe. The flow pattern after the vena-contracta is similar to that after an abrupt enlargement, and the loss of head is confined between section 1-1 and section 2-2. Therefore, we can say that the loss due to contraction is not for the contraction itself, but due to the expansion followed by the contraction.

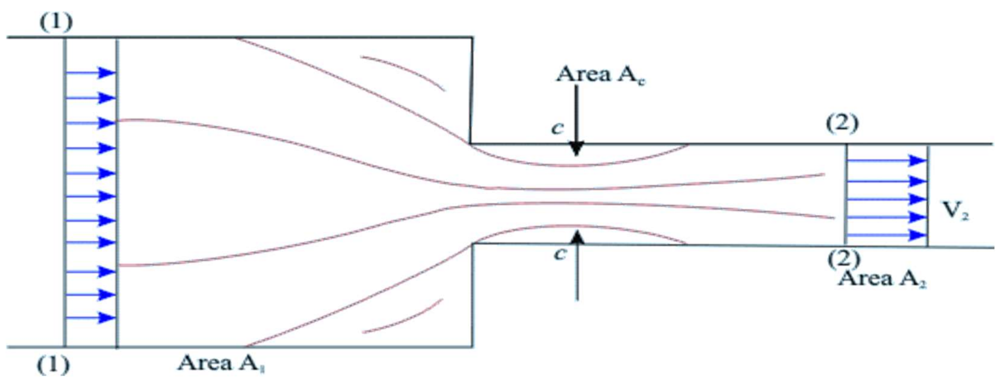


Fig. 3.2 Vena-contracta formed in sudden contraction

Head loss due to contraction is expressed as,

$$h_c = k_c \frac{{v_2}^2}{2g},$$

Where,

v_2=velocity at smaller diameter

k_c=co-efficient of loss due to contraction

3.3.3 Loss of head due to bend in pipe:

Bends are provided in pipes to change the direction of flow through it. An additional loss of head, apart from that due to fluid friction, takes place in the course of flow through pipe bend. The fluid takes a curved path while flowing through the pipe bend as shown in fig.3.3.

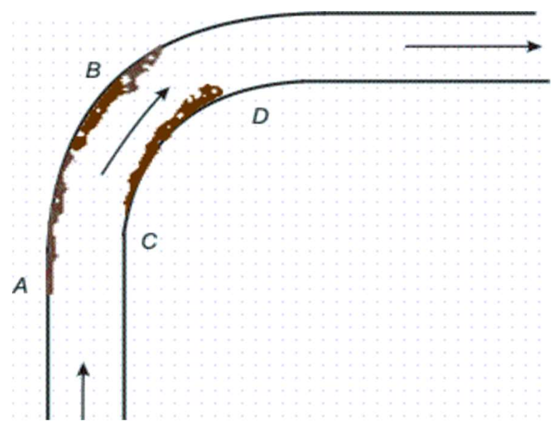


Fig. 3.3 Bends in pipe

Whenever a fluid flows in a curved path, there must be a force acting radially inwards on the fluid to provide the inward acceleration, known as centripetal acceleration. Fluid particles in this region, because of their close proximity to the wall, have low velocities and cannot overcome the adverse pressure gradient and this leads to a separation of flow from the boundary and consequent losses of energy in generating local eddies. Losses also take place due to a secondary flow in the radial plane of the pipe because of a change in pressure in the radial depth of the pipe.

Loss in head due to Bend is expressed as:

$$h_b = k_b \frac{v^2}{2g}$$

Where,

k_b= co-efficient of loss

v = velocity of flow in pipe

3.4 INTRODUCTION TO ANSYS

ANSYS is a general-purpose finite element modelling package for numerically solving a wide variety of mechanical problems. These problems include static/dynamic, structural analysis (both linear and nonlinear), heat transfer, and fluid problems, as well as acoustic and electromagnetic problems.

ANSYS is the standard FEA technique in Mechanical Engineering Department also used in Civil and Electrical Engineering, as well as in the Physics and Chemistry departments. ANSYS provides a cost-effective way to explore the performance of products or processes in a virtual environment. This type of product development is termed virtual prototyping. With virtual prototyping techniques, users can iterate various scenarios to optimize the product long before the manufacturing is started. This enables a reduction in the level of risk, and in the cost of ineffective designs. The multifaceted nature of ANSYS also provides a means to ensure that users are able to see the effect of a design on the whole behaviour of the product, be it electromagnetic, thermal, mechanical etc.

3.4.1 Generic Steps for Solving Any Problem in ANSYS

Like solving any problem analytically, we need to define our solution domain, physical model, boundary conditions and the physical properties in ANSYS. You then solve the problem and present the results, compare to numerical methods, the main difference is an extra step called mesh generation. This is the step that

- Build Geometry
- Define Material Properties
- Generate Mesh
- Apply Loads, and boundary conditions
- Obtain Solution
- Present the Results

Build Geometry: In this stage construct a two or three-dimensional representation of the object to be modelled and tested using the work plane coordinate system within ANSYS.

Define Material Properties: Now that the part exists, define a library of the necessary materials and material properties that compose the object (or project) being modelled.

This includes thermal and mechanical properties of the object.

Generate Mesh: At this point ANSYS understands the makeup of the part. Now define how the modelled system should be broken down into finite pieces.

Apply Loads: Once the system is fully designed, the last task is to apply the system with constraints, such as physical loadings or boundary conditions.

Obtain Solution: In this step we obtain the solution. In this step we need to understand within what state (steady state, transient… etc.) the problem must be solved.

Present the Results: After the solution has been obtained, there are many ways to present ANSYS results, choose from many options such as tables, graphs, and contour plots.

3.4.2 ANSYS Fluent Features

a) Efficient and Flexible Workflow

Fluent is fully integrated into the ANSYS Workbench environment, a platform designed for efficient and flexible workflows, CAD associatively and powerful capabilities in geometry modelling and meshing. The built-in parameter manager makes it easy to rapidly explore multiple design options.

b) Go Faster with High Performance Computing (HPC)

With HPC, ANSYS Fluent delivers CFD simulation solutions faster so that engineers and designers can make better decisions sooner in the design cycle. While ANSYS HPC provides linear scalability on systems with tens of thousands of processors, there is more to HPC than just the number of cores. ANSYS also optimizes processor architecture, algorithms for model partitioning, optimized communications and load balancing between processors to deliver results in breath-taking speed on a wide variety of simulation models.

c) Turbulence Modelling

ANSYS Fluent software places special emphasis on providing a wide range of turbulence models to capture the effects of turbulence accurately and efficiently. Several innovative models such as the Menter–Langtry γ–θ laminar–turbulent transition model™ are available only in Fluent.

d) Fluid-Structure Interaction

Fluent models the effects of solid motion on fluid flow by coupling with ANSYS structural mechanics solutions through the Workbench unified user environment. Fluent users enjoy robust and accurate two-way FSI without the need to purchase, administer or configure third-party coupling and pre- and post-processing software.

e) Heat Transfer & Radiation

Fluent handles all types of radiative heat exchange in and between fluids and solids, from fully and semi-transparent to radiation, or opaque. You can choose from a variety of spectral models to account for wavelength dependencies in a simulation and to account for scattering effects.

3.5 COMPUTATIONAL FLUID DYNAMICS:

3.5.1 DEFINITION AND HISTORY:

Computational Fluid Dynamics (CFD) is the use of computer-based simulation to analyse systems involving fluid flow, heat transfer and associated phenomena such as chemical reaction. A numerical model is first constructed using a set of mathematical equations that describe the flow. These equations are then solved using a computer programme in order to obtain the flow variables throughout the flow domain.

3.5.2 GOVERNING EQUATIONS

The governing equations of fluid flow represent mathematical statements of the conservation laws of physics. Each individual governing equation represents a conservation principle. The fundamental equations of fluid dynamics are based on the following universal laws of conservation. They are,

Unsteady state 3-D equation of continuity: -

$$\frac{\partial\,\partial\rho}{\partial\partial t} + div(\rho u) = 0$$

Momentum equation: -

$$\frac{\partial(\rho u)}{\partial t} + div\ (\rho u \mu) = -\frac{\partial P}{\partial x} + div(\mu grad u)$$

Energy equation: -

$$\frac{\partial(\rho i)}{\partial t} + \mathrm{div}(\rho i u) = -P\mathrm{div}\ u + \mathrm{div}(k\ \mathrm{grad}\ T) + \emptyset + S$$

3.5.3 APPLICATIONS OF CFD:

The earliest adopters of CFD were the aerospace, automotive and nuclear industries. Further growth and development in CFD and its ability to model complex phenomena along with the rapid increase in computer power have constantly widened the range of application of CFD. CFD is applied in a wide range of industries including mechanical, process, petroleum, power, metallurgical, biomedical, and pharmaceutical and food industries.

CFD techniques have been applied on a broad scale in the process industry to gain insight into various flow phenomena, examine different equipment designs or compare performance under different operating conditions.

Examples of CFD applications in the chemical process industry include drying, combustion, separation, heat exchange, mass transfer, pipeline flow, reaction, mixing, multiphase systems and material processing.

3.5.4 ADVANTAGES OF CFD

- No restriction to linearity.
- Complicated physics can be treated.
- Time evaluation of flow can be obtained.
- It has the potential of providing information not available by other means.
- Computational investigation can be performed with remarkable speed. Designer can study the implications of hundreds of different configurations in minimum time and choose the optimum design.
- It gives detailed and complete information. It can provide the values of all the relevant variables (pressure, velocity, temperature, concentration, turbulence) throughout the domain of interest.

3.5.5 DISADVANTAGES

- Truncation errors
- Boundary condition problems
- Computer costs
- Computer storage & speed

3.5.6 CFD ANALYSIS PROCEDURE

Computational fluid dynamics (CFD) study of the system starts with the construction of desired geometry and mesh for modelling the dominion. Generally, geometry is simplified for the CFD studies. Meshing is the discrete process of the domain into small volumes where the equations are solved by the help of iterative methods. Modelling starts with the describing of the boundary and initial conditions for the dominion and leads to modelling of the entire system. Finally, it is followed by the analysis of the results, discussions and conclusions.

The complete CFD analysis procedure can be divided into the following six stages.

a) Initial thinking

It is very important to understand as much as possible about the problem being simulated in order to accurately define it. This stage involves collecting all the necessary data required for the simulation including geometry details, fluid properties, flow specifications, and boundary and initial conditions.

b) Geometry creation

The geometry of the flow domain is created using specialised drawing software.

Usually, 2-D sketches are first drawn and 3-D tools are then used to generate the full geometry.

c) Mesh generation

In this stage the continuous space of the flow domain is divided into sufficiently small discrete cells, the distribution of which determines the positions where the flow variables are to be calculated and stored. Variable gradients are generally more accurately calculated on a fine mesh than on a coarse one. A fine mesh is therefore particularly important in regions where large variations in the flow variables are expected. A fine mesh, however, requires more computational power and time. The mesh size is optimised by conducting a mesh-independence test whereby, starting with a coarse mesh, the mesh size is refined until the simulation results are no longer affected by any further refinement.

d) Flow specification

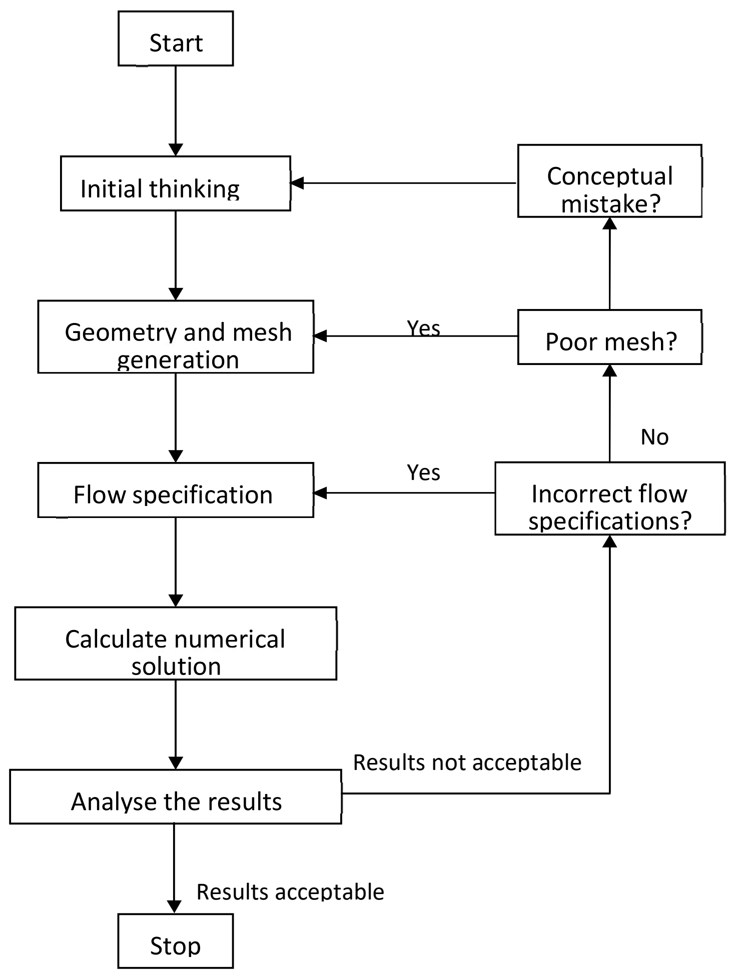
Flow specification involves defining the fluid physical properties, flow models, boundary conditions, and initial flow conditions, as determined in the initial thinking stage.

e) Calculation of the numerical solution

When all the information required for the simulation has been specified, the CFD software performs iterative calculations to arrive at a solution to the numerical equations representing the flow. The user needs also to provide the information that will control the numerical solution process such as the advection scheme and convergence criteria.

f) Results analysis

Having obtained the solution, the user can then analyse the results in order to check that the solution is satisfactory and to determine the required flow data. If the results obtained are unsatisfactory, the possible source of error needs to be identified, which can be an incorrect flow specification, a poor mesh quality, or a conceptual mistake in the formulation of the problem.



CHAPTER 4

PROCEDURE OF EXPERIMENT

4.1 STEPS:

1. First of all, bench valve, gate valve and the flow control valves are opened and after that the pump is started to fill the test rig with water.
2. Air, if present is bleed from the pressure tap points and the manometers by adjusting the bench and flow control valves and air bleed screw.
3. The scales of all the manometer levels are checked when all the valves are fully opened. The level is adjusted with the help of air bleed screw.
4. The reading is recorded for a selected flow from all the manometers after the water levels have steadied.
5. The flow rate is determined by collecting some fixed volume of water (15000 cm^3) in volumetric storage tank with the help of stopper. A digital stopwatch is used to record time. This is used to calculate the discharge of water.
6. Step 4 and 5 are repeated in different pipes for two more flows.
7. The flow rate is adjusted by the control valve and pressure drop across the gate valve is measured from the pressure gauge.
8. Step 7 is repeated for more two flow rates.

Fig.4.1 Apparatus of Minor Loss

CHAPTER 5

OBSERVATIONS
&
CALCULATIONS

5.1 SUDDEN EXPANSION:

5.1.1 EXPERIMENTAL OBSERVATION AND CALCULATION:

OBSERVATION:

Diameter of the smaller pipe before expansion is 19 mm and after expansion is 25.4mm.

Diameter of larger pipe before expansion is 25.4mm and after expansion is 38.1mm.

Table 5.1: Observation table of sudden expansion

Sl. No	Rise in level(cm)	Volume cm^3	Time (sec)	Discharge Q (m^3/sec)	Manometer Difference h(mm)	Head h_e(m)	V_1 (m/sec)	K_e
Pipe 1	10	15000	10	1.5×10^{-3}	25	0.315	5.29	0.22
Pipe 1	10	15000	11	1.3×10^{-3}	19	0.24	4.8	0.2
Pipe 2	10	15000	13	1.1×10^{-3}	7	0.084	2.17	0.35
Pipe 2	10	15000	23	6.43×10^{-4}	2.5	0.03	1.26	0.33

CALCULATION:

For pipe 1,

Q= volume/ time

= 0.015/10

= 1.5×10^{-3} m^3/sec

$A = \pi\frac{d^2}{4} = \pi\times\frac{0.019^2}{4} = 0.28\times10^{-3}\ m^2$

$Q = A\times v_1$

$Q=1.5\times10^{-3} = 0.28\times10^{-3}\times v_1$

$V_1 = 5.29$ m/s

$h_e = h(\frac{S_m}{S} - 1) = 25\times(\frac{13.6}{1} - 1) = 25\times12.6$ mm

$h_e = 0.315$ m

$h_e = k_e\times\frac{{v_1}^2}{2g}$

$\boldsymbol{k_e} = \frac{2gh_e}{{v_1}^2} = \frac{2\times9.81\times0.315}{5.29^2} = 0.22$

5.1.2 FROM ANSYS:

a) Modelling

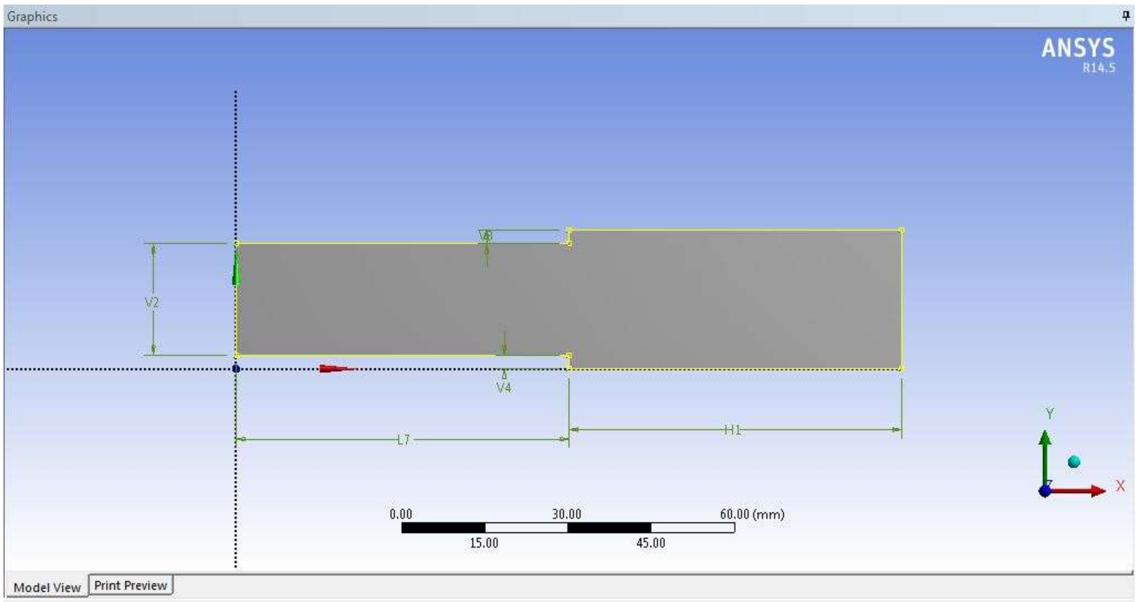


Fig.5.1. Modelling of Sudden expansion of pipe

b) Meshing

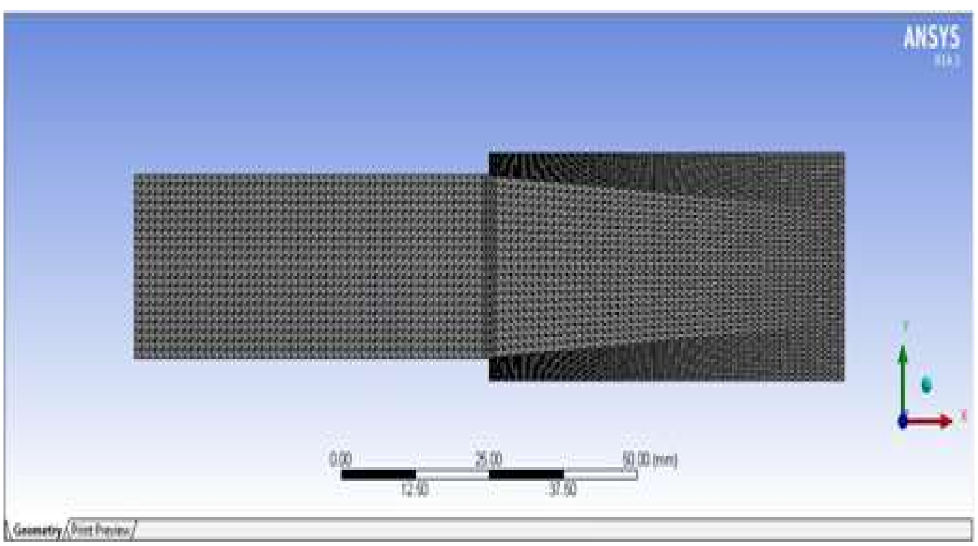


Fig.5.2. Meshing of sudden expansion pipe

c)Stream function

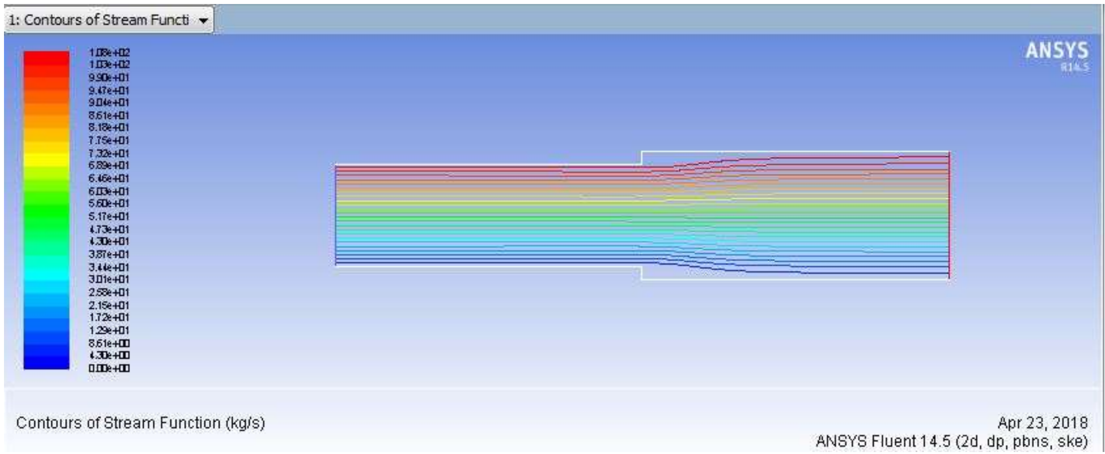


Fig. 5.3: Stream function of flow

Just after the junction, stream line at the surface is disconnected, formation of eddies occurs and further touches the surface.

d)Absolute Pressure

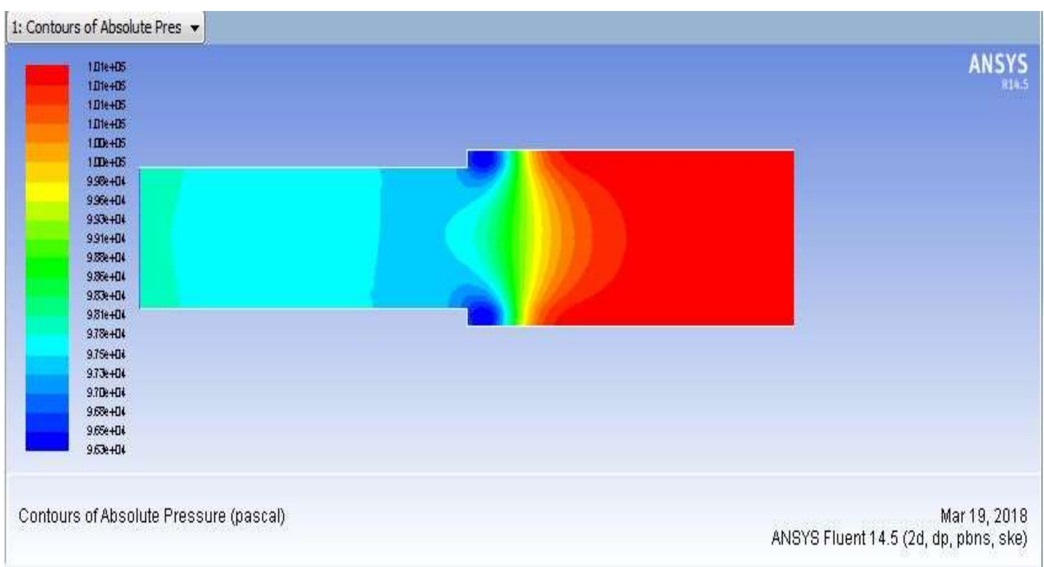


Fig. 5.4: Contours of absolute pressure in pipe

At the inlet, pressure is constant and near the junction, variation in pressure can be shown. And at outlet, pressure is much more than inlet.

e) Velocity

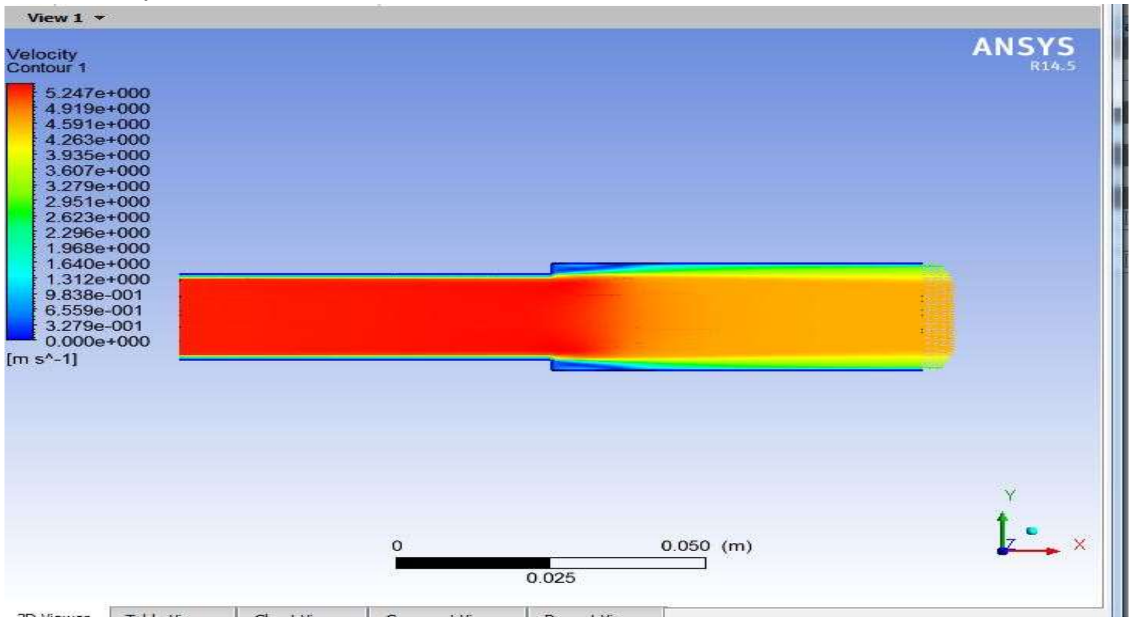


Fig 5.5: Contours of velocity magnitude

There is decrease in velocity of fluid when it reaches the junction. At the right of the junction, velocity decreases in right direction and after some distance, velocity reaches steady state and velocity at outlet is less than inlet.

f) Dynamic Pressure

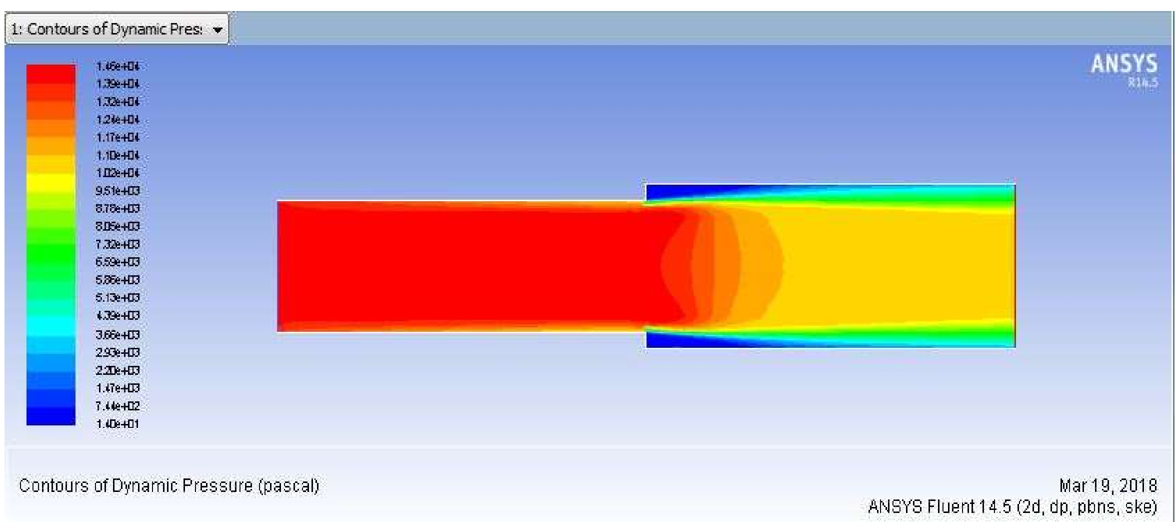
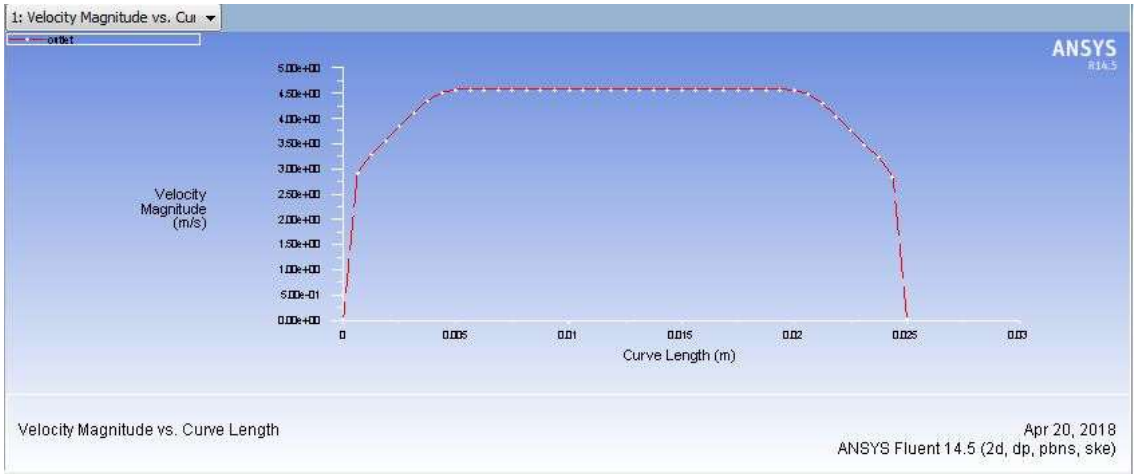


Fig 5.6: Contour of dynamic pressure

It is nearly same as velocity contour because dynamic pressure is directly proportional to square of the velocity.

g) Velocity variation



Graph 5.1: velocity variation at the outlet surface

At the pipe surface, velocity is zero and it is increased as we go away from surface to towards the axis and velocity is maximum at the axis.

h) Result:

Pressure at outlet $=1.01\times10^5$ pa.

Pressure at inlet$=9.73\times10^4$ pa.

Difference in pressure= $(1.01\times10^5) - (9.73\times10^4) = 3.7\times10^3$ pa.

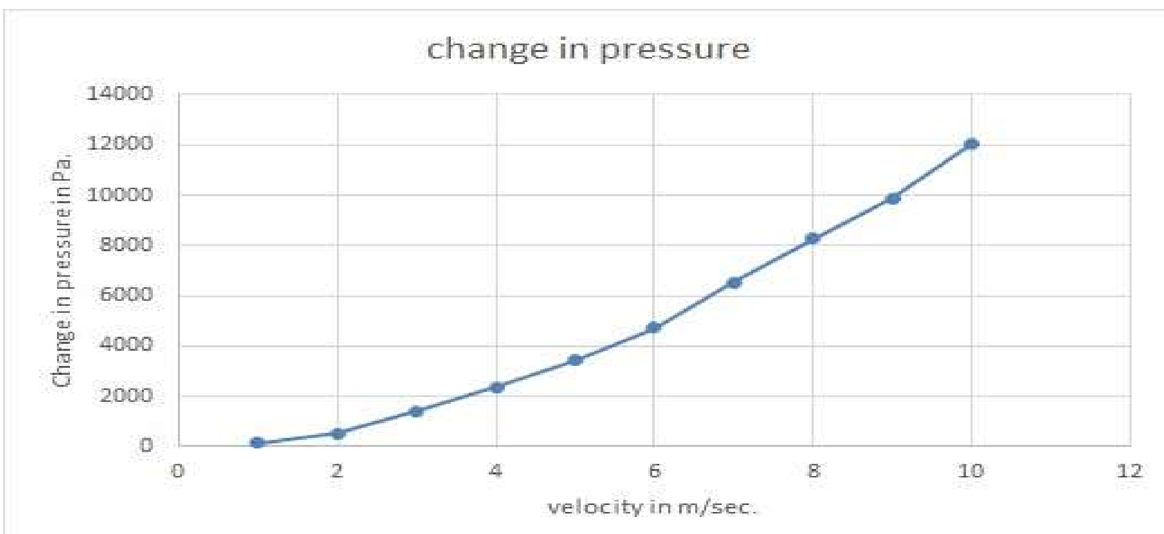
Head difference $= \frac{3.7\times10^3}{\rho g} = \frac{3.7\times10^3}{1000\times9.81} = 0.377$ m

$h_e = k_e \times \frac{{v_1}^2}{2g}$

$$k_e = \frac{2gh_e}{{v_1}^2} = \frac{2\times9.81\times0.377}{5.29^2} = \mathbf{0.26}$$

Table 5.2. Change in pressure and corresponding head between inlet and outlet at different velocity

Velocity(m/s)	change in pressure(Pa)	head loss(m)
1	133.7	0.013655
2	516	0.0527
3	1411.3	0.144137
4	2365.9	0.241631
5	3410	0.348266
6	4727.2	0.482793
7	6517.65	0.665653
8	8270.7	0.844694
9	9863.4	1.007357
10	12016.4	1.227245



Graph 5.2: Variation of pressure difference at different velocity

5.2 SUDDEN CONTRACTION:

5.2.1 EXPERIMENTAL OBSERVATION AND CALCULATION:

OBSERVATION:

Table 5.3: Observation table of sudden contraction

Sl No	Rise in level (cm)	Volume cm^3	Time (sec)	Discharge (m^3/sec)	Manometer Difference h(mm)	Head (m)	V_1 (m/sec)	v_2 (m/sec)	K_c
Pipe 1	10	15000	10	1.5×10^{-3}	24	0.315	2.97	5.29	0.22
Pipe 1	10	15000	18	8.18×10^{-4}	7.1	0.106	1.62	2.89	0.4
Pipe 2	10	15000	16	9.0×10^{-4}	4.05	0.051	0.995	1.77	0.31
Pipe 2	10	15000	18	8.18×10^{-4}	3.25	0.054	0.905	1.61	0.41

CALCULATION:

For pipe 1

Q= volume/ time

= 0.015/10

= 1.5×10^{-3} m^3/sec

A= $\pi \frac{d^2}{4} = \pi\times \frac{0.019^2}{4} = 0.28\times10^{-3}$ m^2

Q = $A\times v_2$

$1.5\times10^{-3} = 0.28\times10^{-3} \times v_2$

V_2=5.29 m/s

$h_c = h(\frac{S_m}{S} - 1) = 25\times(\frac{13.6}{1} - 1) = 25\times12.6$ mm

h_c = 0.315 m

$h_c = k_c\times\frac{{v_2}^2}{2g}$

$$\boldsymbol{k_c = \frac{2gh_c}{{v_2}^2} = \frac{2\times9.81\times0.315}{5.29^2} = 0.22}$$

5.2.2 FROM ANSYS

a) Modelling

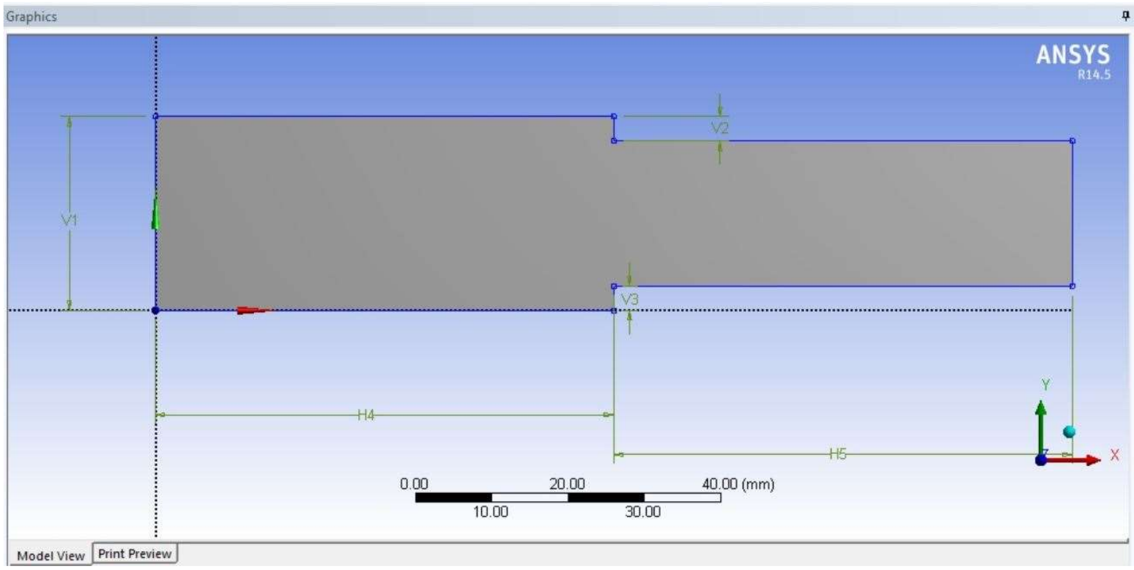


Fig 5.7: Geometry of sudden contraction

b) Meshing

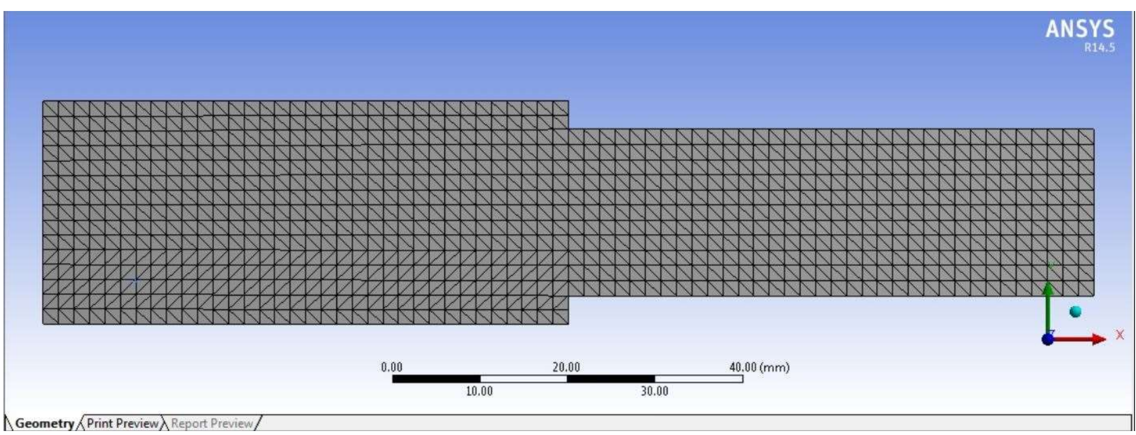


Fig 5.8: Mesh of sudden contraction

c) Stream Function

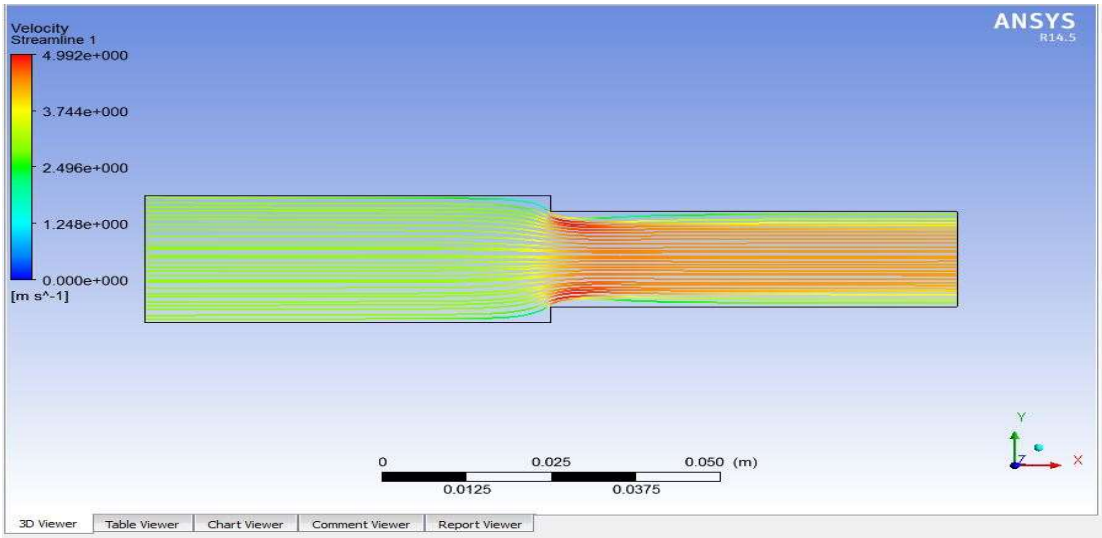


Fig 5.9: Contour of Stream function

In the above fig 5.9, it clearly shows that stream lines change their path at junction and minimum area of flow occurs at just right of the junction which is known as vena-contracta.

d) Absolute Pressure

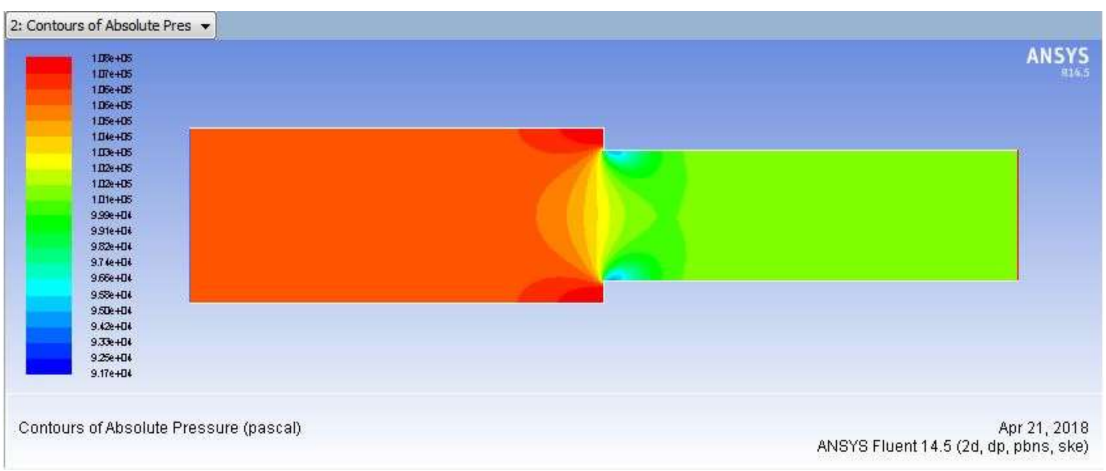


Fig 5.10: Contour of absolute pressure

Pressure decreases when flow is contracted at the junction of the pipe surface having very high pressure, and at outlet (smaller diameter) pressure is lower.

e) Velocity

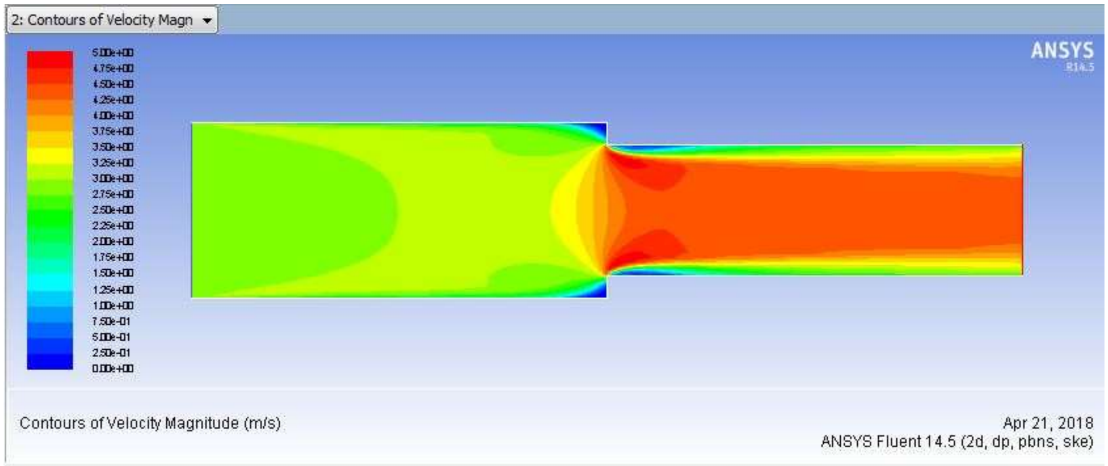


Fig 5.11: Contour of velocity

Velocity increases when stream enters in smaller diameter and velocity is maximum at vena-contracta because area of flow is minimum at that place.

f) Dynamic Pressure

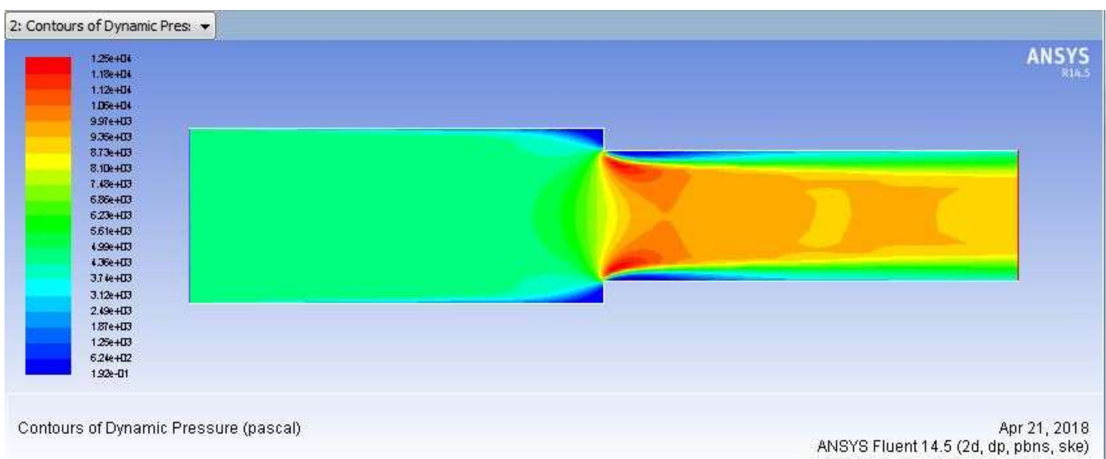


Fig 5.12: Contour of Dynamic pressure

It is nearly same as velocity profile because, it is directly proportional to square of the velocity.

g) Static Pressure

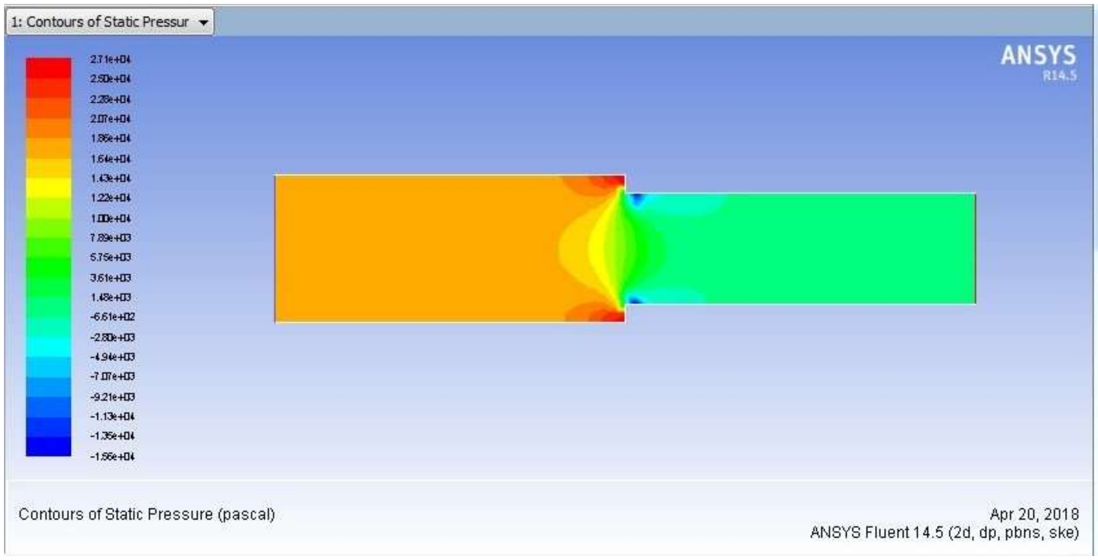


Fig 5.13: Contour of static pressure

From above Fig 5.13, it clearly shows that there is loss in static pressure between inlet and outlet. Pressure at inlet (pipe having large diameter) is more than the outlet (pipe having smaller diameter).

h) Result:

Pressure at inlet=9.35×10^3 Pa.

At outlet=6.0×10^3 Pa.

Difference in pressure=3.35×10^3 Pa.

Head difference $= \frac{3.35\times10^3}{\rho g} = \frac{3.35\times10^3}{1000\times9.81} = 0.342$ m

For pipe 1 and $V_2 = 5.29$ m/s.

$h_c = k_c \times \frac{{v_2}^2}{2g}$

$$\boldsymbol{k_c = \frac{2gh_c}{{v_2}^2} = \frac{2\times9.81\times0.342}{5.29^2} = 0.24}$$

5.3 BEND

5.3.1 EXPERIMENTAL OBSERVATION AND CALCULATION:

OBSERVATION:

Table 5.4 Observation table for bend

Sl No	Rise in level(cm)	Volume (cm^3)	Time (sec)	Discharge (m^3/sec)	Manometer Difference(mm)	Head (m)	v (m/sec)	K_b
Pipe 1	10	15000	10	1.5×10^{-3}	14.2	0.1786	2.96	0.4
Pipe 1	10	15000	15	1×10^{-3}	7	0.09	1.97	0.45
Pipe 2	10	15000	16	9×10^{-4}	16	0.1984	3.16	0.39
Pipe 2	10	15000	24	6.21×10^{-4}	8	0.1	2.18	0.41

CALCULATION:

For pipe 1,

Q= volume/ time

= 0.015/10

= 1.5×10^{-3} m^3/sec

$A = \pi\frac{d^2}{4} = \pi\times\frac{0.0254^2}{4} = 0.645\times10^{-3}\ m^2$

Q = A×v

$Q = 1.5\times10^{-3} = 0.645\times10^{-3}\times v$

V =2.96 m/s

$h_b = k_b\times\frac{v^2}{2g}$

$$\boldsymbol{k_b = \frac{2gh_b}{v^2} = \frac{2\times9.81\times0.1786}{2.96^2} = 0.4}$$

5.3.2 FROM ANSYS

a) Modelling

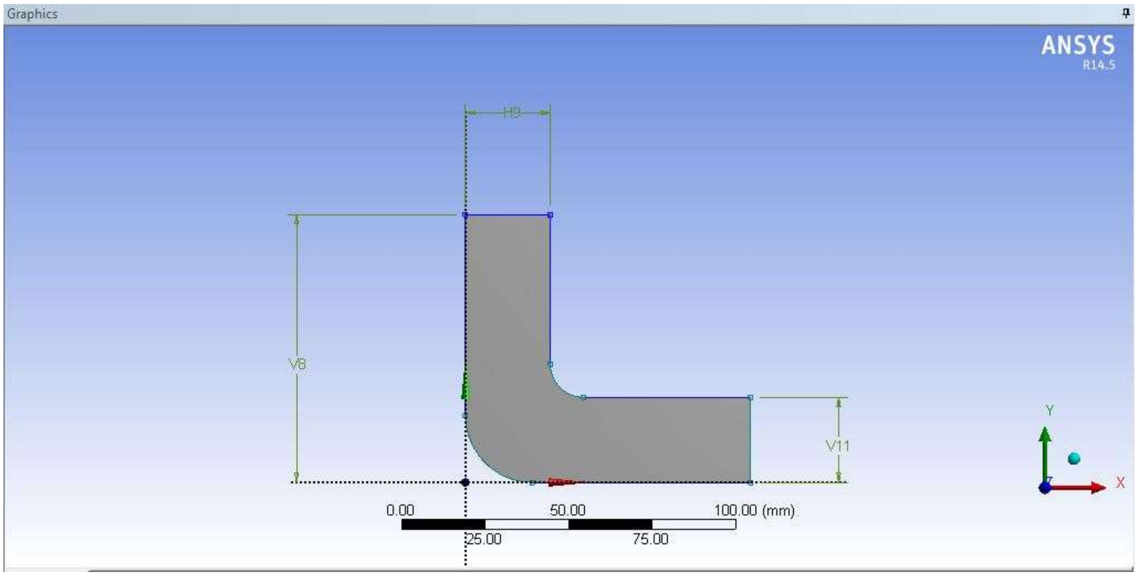


Fig 5.14: Geometry of elbow

b) Meshing

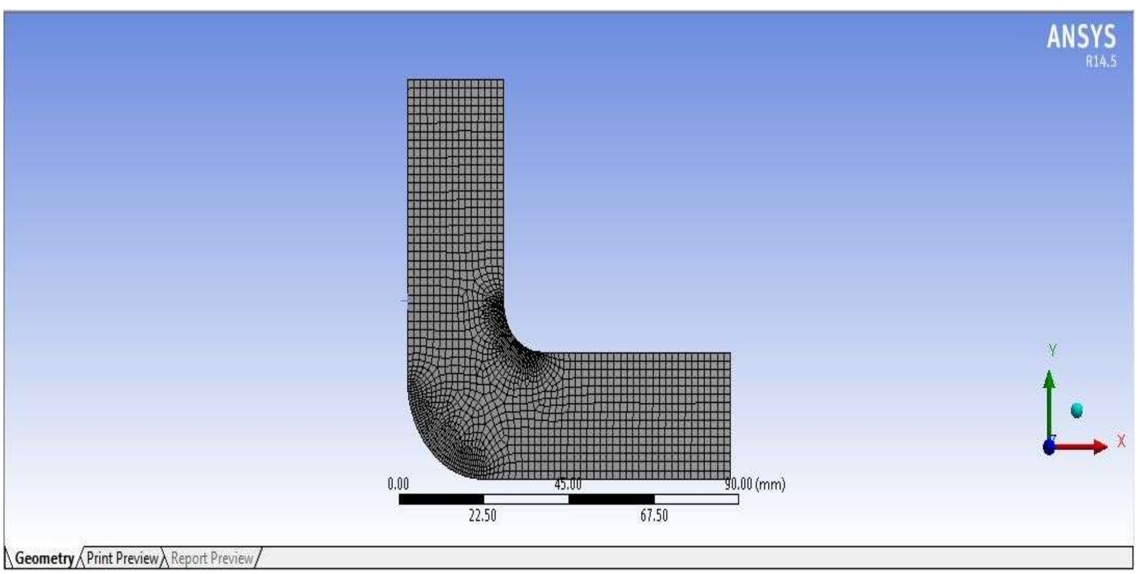


Fig 5.15: Mesh of elbow

c)Velocity

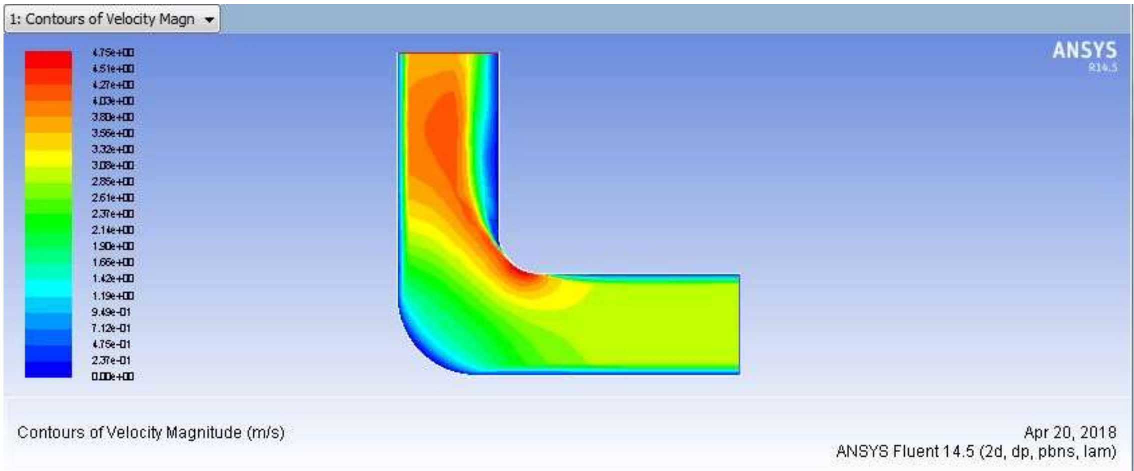


Fig 5.16: contour of velocity

Velocity is maximum at the bend because of the change in direction of flow.

d)Dynamic Pressure

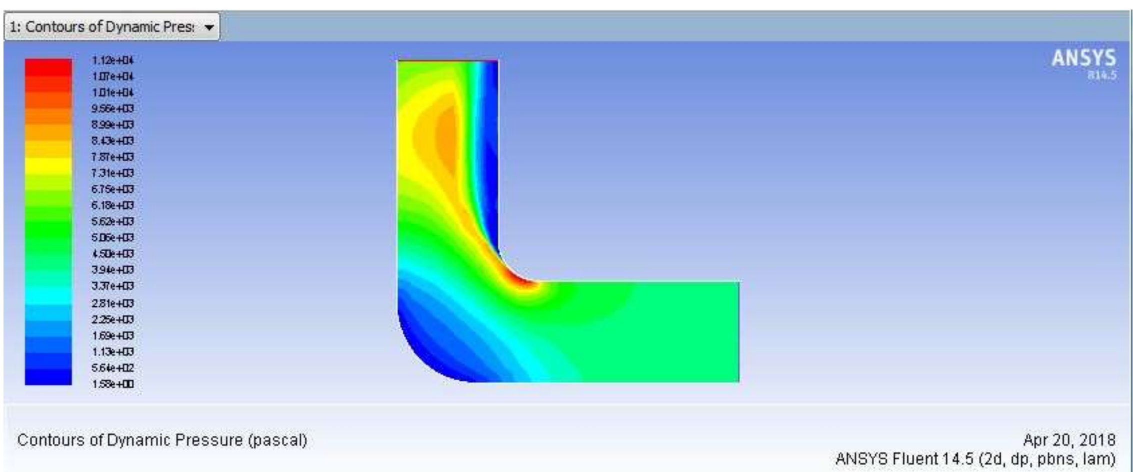


Fig 5.17: Contour of dynamic pressure

e) Static Pressure

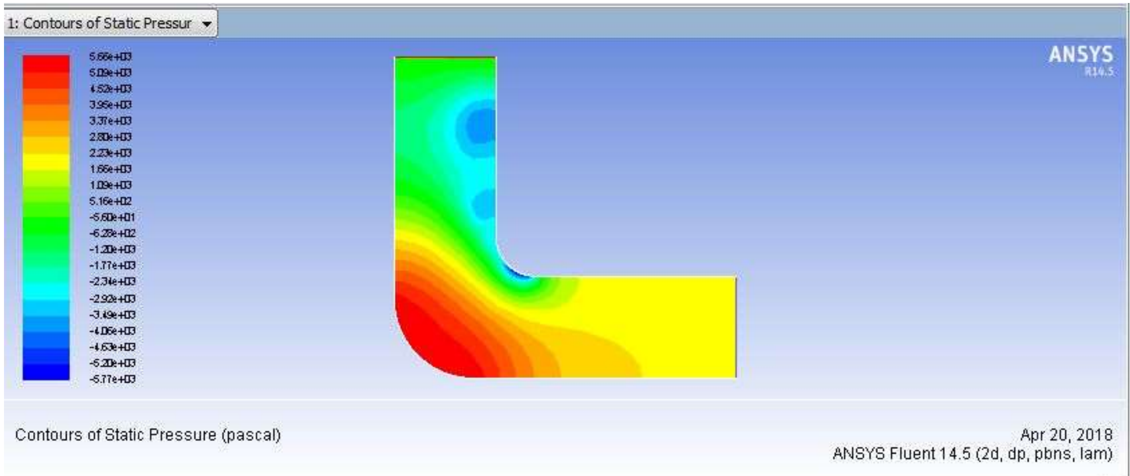


Fig 5.18: contour of static pressure

Static pressure increases at the bend and after the bend static pressure decreases gradually.

g) RESULT:

For bend pipe,

Inlet pressure= 2.23×10^3 Pa

Outlet pressure= -5.6×10^{-1} Pa

Difference in Pressure = $2.286\text{x}10^3$ Pa

$$\text{head loss} = \frac{\text{pressure difference}}{\rho g} = \frac{2.286 \times 1000}{1000 \times 9.81}\text{m}$$

$$=0.233\text{m}$$

Now, k_b = Coefficient of bend

$$\boldsymbol{k_b = \frac{2gh_b}{v^2} = \frac{2\times9.81\times0.1786}{2.96^2} = 0.52}$$

CHAPTER 6

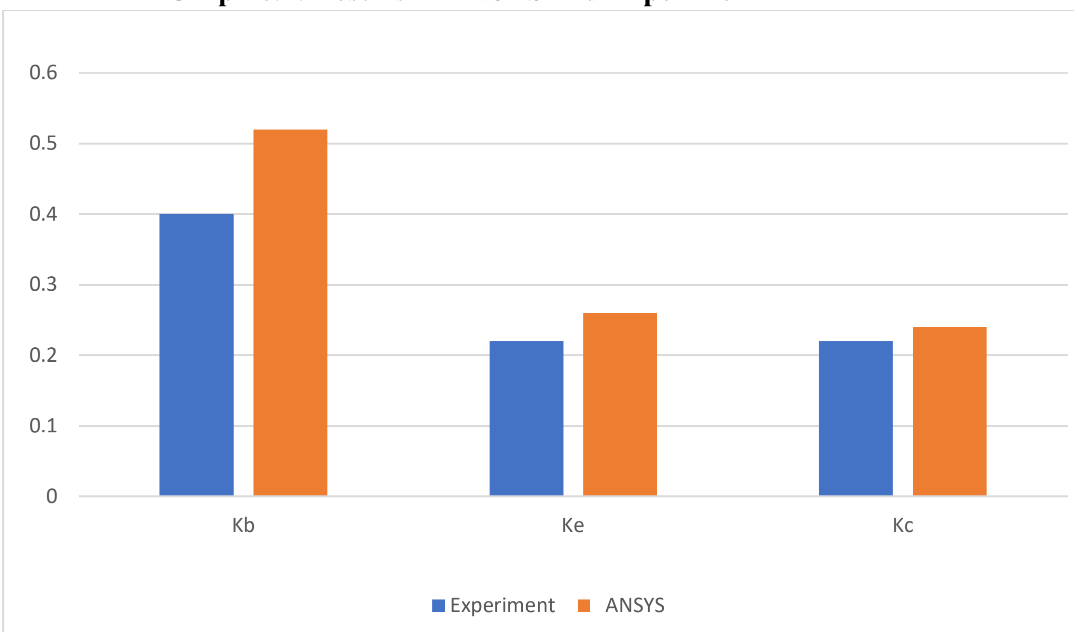
ANALYSIS OF RESULTS

6.1 COMPARISON BETWEEN ANSYS AND EXPERIMENTAL RESULTS

Table 6.1: Results in ANSYS and Experiments

RESULT	K_b	K_e	K_c
From experiment	0.4	0.22	0.22
From ANSYS	0.52	0.26	0.24

Graph 6.1: Results in ANSYS and Experiment



CHAPTER 7

CONCLUSION

The experimental study is conducted and the results obtained from experiment and CFD simulation are compared with each other. The graph shows the variation of head loss with respect to change in the velocity.

From the entire project, it can be concluded that the results obtained from CFD simulation are slightly higher than the results obtained from the experiment.

The variation in the results is because of various parameters acting on the fluid such as resistance between layers, errors in taking the readings, and air acting on the surface of the fluid.

Loss co-efficient of bends is 3% more in ANSYS results than result obtained from experiments. The difference between results is less which means values are approximately same. When the bend curvature is high, the velocity profiles at the bend inlet are shifted towards the inner pipe wall, whereas at low curvature the velocity profiles remain symmetric.

Loss co-efficient of contraction is 1.8% more in ANSYS results than the result obtained from experiments. The values of result are nearly same.

Loss co-efficient of expansion is 0.9% more in ANSYS results than the result obtained from experiments. In this geometry, the results obtained from both ANSYS and experiments are almost same.

REFERENCES

[1] Sovran G., Ackeret, Fluid mechanics of internal flow, Elsevier Publishing Company (1967).

[2] G. P. Celata, M. Cumo, S. McPhail, and G. Zummo, "Characterization of fluid dynamic behaviour and channel wall effects in microtube," International Journal of Heat and Fluid Flow, vol. 27, pp. 135-143, 2006.

[3] Hager, W.H., "Cavity Outflow from a Nearly Horizontal Pipe," Int. J. Multiphase Flow, 25, 349 (1999).

[4] Laursen (1970)**,** "Contraction coefficient at sudden expansion at bridge locations".

[5] Kindsvater, Carter and Lacy et al (1953) and Kindsvater and Carter et al (1955) Effects of different types of contractions on discharge characteristics.

[6] Rathakrishnan, E. and Sreekanth, A.K., Flow in pipes with sudden enlargement, Proceedings of the 14th International Symposium on Space Technology and Science, Tokyo, Japan, p.491 (1984).

[7] Wick, R. S., The effect of boundary layer on sonic flow through an abrupt cross-sectional area change, Journal of the Aeronautical Science, Vol 20, p.675 (1953).

[8] Tapas K. Mandal, "Motion of Taylor Bubbles and Taylor Drops in Liquid−Liquid Systems*",* Ind. Eng. Chem. Res.*,* 2008, 47 (18), pp 7048–7057.

[9] Rodriguez (2009)International Journal of Multiphase Flow Volume 36, Issues 11–12, November–December 2010

[10] Vallentine Sixth International Water Technology Conference, IWTC 2001, Alexandria, Egypt (1958), Effect of pipe contraction placed normal to channel axis.

[11] Delhaye et al (1981), Wadle et al (1989) Schmidt and Friedel et al (1997), Guglielmini et al (1997), Fossa and Guglielmini et al (2002) “Oil–water flows through sudden contraction and expansion in a horizontal pipe–Phase distribution and pressure drop”.

[12] Timothy J Rennie, “Comparison of heat transfer rates between a straight tube heat exchanger and a helically coiled heat exchanger”, International Communications in Heat and Mass Transfer 29 (2), 185-191.

[13] J.S. Jayakumar, “Experimental and CFD investigation of convective heat transfer in helically coiled tube heat exchanger”.

[14] “Comparison of flow analysis of a sudden and gradual change Of pipe diameter using fluent software” G.Satish, K.Ashok Kuma, V.Vara Prasad, Sk.M.Pasha IJSRD - International Journal for Scientific Research & Development| Vol. 2, Issue 05, 2014|

[15] “CFD numerical simulation analysis of small and medium caliber 90 ° circular bend” Wan Kai,Wang Ping Proceedings of the 2nd International Conference on Computer Science and Electronics Engineering (ICCSEE 2013) “Evaluation of head losses in fluid”.