Effect of Junction Geometry on Turbulence and Flow Separation in Pressure Tunnels

Amruta A. Chavan[1], Sushma Vyas[2], Sruthi T. K.[3], Ansh Omane[4], Chaitnya Vishwe[4], Shreyash Arsul[4]

[1]*(Assistant Professor, Department of Civil Engineering, ABMSP;s APCOER, Pune, Maharashtra, India)*
[2]*(Scientist D, CWPRS, Pune, Maharashtra, India)*
[3]*(Scientist B, CWPRS, Pune, Maharashtra, India)*
[4]*(Student, Department of Civil Engineering, ABMSP;s APCOER, Pune, Maharashtra, India)*

Abstract: Pressure tunnel junctions play a crucial role in determining flow behaviour, pressure distribution, turbulence generation and energy losses in hydraulic systems. The geometry of a junction directly influences the movement of water through the tunnel network, making its design an important factor in achieving efficient hydraulic performance. This study examines the effects of different junction geometries on turbulence characteristics and flow separation in pressure tunnels using Computational Fluid Dynamics (CFD) simulations.

Numerical analyses were carried out for both converging and diverging flow conditions with junction angles of 30°, 45°, 60° and 90° and main tunnel diameters of 3 m, 4 m and 5 m. Pressure and velocity distributions were evaluated at selected probe locations around the junction region to assess flow behaviour. For hydraulic interpretation, the pressure values obtained from CFD simulations were converted into pressure heads expressed in metres of water column.

The simulation results indicate that junction geometry significantly affects hydraulic performance. Smaller junction angles promoted smoother flow transitions, reduced turbulence intensity, minimized flow separation, and lowered hydraulic losses. Conversely, larger junction angles generated stronger flow disturbances, larger recirculation zones, and greater energy dissipation. An inverse relationship between pressure head and velocity was consistently observed across all cases. Additionally, increasing tunnel diameter reduced flow velocity and enhanced flow stability. Among the configurations studied, the 30° junction demonstrated the most efficient hydraulic performance, whereas the 90° junction produced the highest turbulence levels and energy losses. These findings emphasize the importance of optimizing junction geometry to improve hydraulic efficiency and reduce operational losses in pressure tunnel systems.

Keywords: Pressure tunnels, junction geometry, turbulence, flow separation, CFD, pressure head, velocity distribution, hydraulic losses, converging flow, diverging flow, hydraulic efficiency.

I. Introduction

1.1 General Background

Pressure tunnels are important elements in the efficient transport of water under high pressure, for example in the case of hydropower, large irrigation networks or major water supply systems[1]. Within these tunnels, the shape and configuration of junctions are of particular importance because they affect the ease with which water can flow from one section to another. Poorly designed junction geometry can induce turbulence, flow separation and unneeded energy losses[2].

Flow separation, in particular, forms low-pressure pockets and swirling zones that interrupt smooth flow. This has the potential to lead to a reduction of the efficiency of the entire system over time and in some cases to influence the structural health of the tunnels due to the variable pressure forces[3]. With increasing demand for robust and energy-efficient hydraulic infrastructure, understanding the effects of junction geometry on flow behaviour has never been more relevant. Because of progress in computational modelling, engineers can now study these interactions in detail and design junctions that reduce flow disturbance and improve the performance of the tunnel as a whole[4].

1.2 Importance of Studying Junction Geometry in Pressure Tunnels

As hydraulic systems grow in size and complexity, the need to minimize energy losses and guarantee reliable operation is becoming increasingly important[1]. Junctions are frequently trouble spots where turbulence builds up and head losses increase, especially when the geometry is too sharp or sudden. This can lead to more wear and tear, more vibration and in some cases, long term structural damage[2].

By studying the impact of different junction geometries on flow properties, engineers can enhance designs and reduce the risk of inefficiency or structural issues. More power can result from even a small increase in flow in a hydropower tunnel. Smoother transitions help keep pressure even and reduce pumping costs for water distribution or irrigation systems[1].

The aim of this work is to show that the implementation of modified or streamlined geometries can be used to control the turbulence, to reduce the separation zones and to significantly improve the hydraulic performance by analysing different junction configurations[4].

1.3 Flow Dynamics and Junction Behaviour: Fundamentals

The flow in pressure tunnels is influenced by a combination of pressure variations, velocity changes and turbulence. Junctions are locations where these effects are more significant as the geometry forces the flow to change direction or to spread out rapidly[5]. This often leads to adverse pressure gradients which are the main cause of flow separation. Once separation occurs vortices form, energy losses increase and pressure recovery becomes difficult[6].

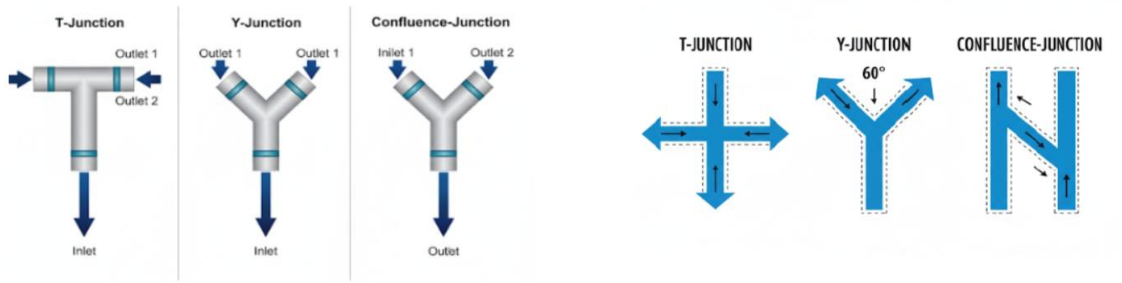
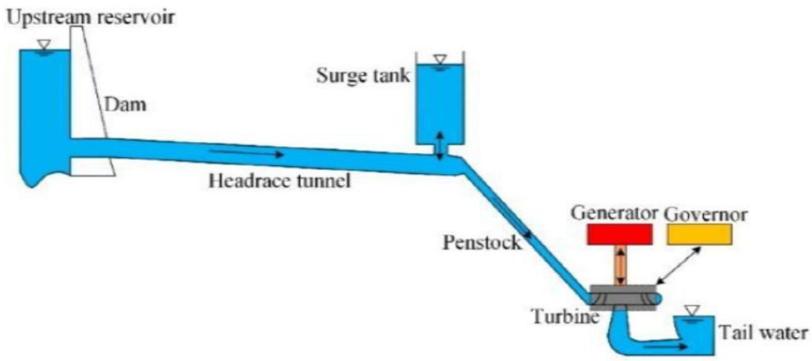


Fig 1.1.Diagram of Different Junction Geometries (Adapted from Miller [3] and Idelchik [2])



Several factors control how severe these effects become, such as:

- Whether the junction is sharp or smoothly curved
- The angle and radius of the junction
- How quickly the cross-section changes
- The strength of the resulting pressure gradients

Modern computational tools make it possible to visualize these effects and evaluate how different geometric designs behave[4]. Research shows that even slight improvements to junction geometry can reduce turbulence and improve flow uniformity, making the entire tunnel system more efficient.

1.4 International Scenario

Researchers worldwide, in particular in countries with advanced hydraulic infrastructures such as Japan, the US and parts of Europe, have extensively studied flow behaviour in tunnel junctions[2,4]. With laboratory experiments and sophisticated simulations, they have sought to understand how turbulence forms at different types of junctions, including T-junctions, Y-junctions and curved connections.

International studies show the importance of:

- Using streamlined or filleted junctions to minimize turbulence.
- Design of complex layouts accounting for pressure recovery.
- Application of advanced numerical models to predict flow behaviour.
- Water-tunnel experiments for validating simulations.

These findings have had an impact on engineering practices around the world, causing designers to implement smoother geometries that help to keep energy efficiency and reduce long-term maintenance issues.

1.5 National Scenario

Pressure tunnels are widely adopted for hydropower projects, water supply schemes and irrigation networks in India. However, research on the hydraulic impact of junction geometry is still increasing. Many legacy systems were designed with traditional geometric layouts, which may not run efficiently under modern operational requirements. Studies done by organizations like CWPRS, CWC and various IITs have highlighted repetitive issues like excessive turbulence, higher pressure losses and unstable flow at junctions which are not well designed[7,8]. These problems not only restrict efficiency, but also create structural stress and generate the need for repairs. As the Indian government is putting more emphasis on enhancing water infrastructure and renewable energy capacity, the significance of optimizing tunnel junctions has grown[4]. But with CFD simulations and advanced modelling techniques, the researchers are now in a better position to assess different junction designs and make recommendations for improvements.

This study contributes to this growing body of work by examining the effect of different junction geometries on turbulence, flow separation and pressure behaviour. The insights gained can help in better design practices, retrofitting solutions and more efficient operation of hydraulic systems in India.

1.6 Need of the Study

Pressure tunnels are a major component of many hydraulic systems, whether hydropower generation, irrigation or large-scale water supply. Junctions in these tunnels are often troublesome areas where the flow is disturbed causing turbulence, separation and undesired energy losses. Many junctions in existing systems, especially older systems, were designed without the benefit of the modern fluid-dynamic understanding or computational tools[2]. This leads to many projects continuing to use sharp or abrupt junction shapes which are not hydraulically efficient[5]. These inefficient junctions are not only energy wasting, but over time they lead to increased pressure fluctuations, which leads to accelerated wear and decreased overall reliability of the system. With the increasing demand for efficient and resilient water infrastructure[1].

This study is needed to fill that gap. The research compares different junction shapes and analyses their effects on turbulence, separation and pressure distribution. It provides insights that can lead to better design choices, smarter retrofitting solutions and more efficient hydraulic tunnels in the future.

1.7 Problem Statement

Results from past research and real hydraulic systems indicate that junctions in pressure tunnels are a significant source of inefficiencies. If the geometry is too sharp or badly designed the flow tends to separate, generating strong vortices, higher turbulence and significant head losses[2,5]. These problems often result in:

- Increased energy consumption and lower system efficiency
- Difficult operation due to unsteady flow patterns
- Higher pressure differentials stressing the walls of the tunnel
- Increased wear and erosion of the material
- Reduced performance of hydropower and water delivery systems

Many of the existing tunnels were built using older design approaches that did not consider detailed fluid-structure behaviour or geometry optimization. This implies that inefficient junction designs are still common and continue to cause problems during operation.

Considering the scale of India’s hydraulic infrastructure and the critical need for reliable performance, knowledge of the effect of junction geometry on flow behaviour is not just useful, but essential[4]. A detailed investigation of these effects can lead to safer, more efficient and sustainable design practices.

1.8 Aim

The main aim of this study is to examine how different junction geometries affect the flow inside pressure tunnels, especially in terms of turbulence, flow separation and pressure changes. By using computational simulations, the study focuses on identifying which junction shapes help maintain smoother, more uniform flow and reduce energy losses.

The ultimate goal is to provide practical insights that can support better design and retrofitting strategies for hydraulic tunnels, helping engineers create systems that are more efficient, more stable and longer lasting.

1.9 Objectives

The specific objectives of this study are:

1. To investigate the effect of different junction designs on the flow behaviour in pressure tunnels.
2. The velocity distribution, pressure variation and turbulence characteristics at the tunnel junctions are analysed using CFD simulations.
3. To assess the impact of junction geometry on pressure losses and flow efficiency generally.
4. Compare the performance of different junction geometries and identify configurations that minimize turbulence and flow separation.
5. To formulate recommendations for the design of pressure tunnel junctions to enhance hydraulic performance and operational efficiency.

II. Methodology

2.1 Introduction

The methodology adopted to study the effect of junction geometry on turbulence characteristics, flow separation and pressure distribution in pressure tunnels by using Computational Fluid Dynamics (CFD) is described. The methodology is mainly centred on the development of different tunnel junction geometries, performing numerical simulations and analysis of the resulting flow parameters to understand the impact of the geometric changes on the hydraulic performance.

The objective of this study is to identify junction configurations that lead to a smoother flow transition, reduced turbulence intensity and minimal energy losses. The work as a whole includes the generation of geometry, meshing, CFD simulation and analysis of the simulation results. The methodology adopted in this study is systematic and suitable for evaluating the hydraulic behaviour of pressure tunnel junctions under different flow conditions.

2.2 Problem Identification

Before starting the simulations, a detailed review of previous research papers, journals and technical studies related to pressure tunnels, pipe junctions and CFD analysis was carried out. It has been demonstrated in previous studies that the geometry of tunnel junctions is of significant importance for the behaviour of flows, in particular for the formation of turbulence, pressure fluctuations and flow separation[4].

The following observations were identified from the literature review:

- Junction angle has a strong influence on turbulence intensity and recirculation zones.
- Sudden flow direction changes are liable to increase energy losses.
- Flow separation usually happens around sharp junctions and large branch angles.
- CFD techniques are extensively used to study complex hydraulic flow behaviour in pressure tunnels.

With reference to these observations, the present study aims to analyse the influence of various junction geometries on flow characteristics in converging and diverging pressure tunnel systems.

2.3 SolidWorks Flow Simulation: Theoretical Background

SolidWorks Flow Simulation is a CFD (Computational Fluid Dynamics) tool integrated with the SolidWorks software. It allows engineers and researchers to study the behaviour of fluid flow within a system without the need for physical experiments. In this project, the software was used to analyze how different junction angles and branch diameters affect turbulence, flow separation, pressure distribution, and velocity in pressure tunnels.

The software works by dividing the flow domain into a large number of small cells and solving the fluid flow equations numerically throughout the model. For turbulent flow conditions, it uses RANS-based turbulence models to predict the flow behaviour accurately[4].

SolidWorks Flow Simulation provides several useful features, including three-dimensional flow analysis, visualization of streamlines, prediction of pressure and velocity distributions, and identification of regions where flow separation or recirculation occurs. These capabilities helped in comparing the hydraulic performance of different tunnel junction configurations and understanding the effect of geometry on flow characteristics.

This software was selected for the present study because of its accuracy, ease of use, and seamless integration with the SolidWorks modelling environment.

2.4 Geometry Generation

The geometrical models of the pressure tunnel junctions were developed with the help of SolidWorks CAD software. Different junction configurations were developed by varying the junction angle and branch channel diameter for a constant main tunnel diameter. SolidWorks was used due to its accuracy and flexibility in creating complex three-dimensional tunnel geometries required for CFD analysis.

In total 24 geometrical models were created comprising:

- 12 converging flow junctions
- 12 diverging flow junctions

The geometries were developed for four different junction angles of 30°, 45°, 60° and 90° and for three different branch channel diameters of 3 m, 4 m and 5 m.

The main channel diameter was maintained at 5 m for all the models to keep the uniformity in the comparison between different cases.

2.5 Selection of Tunnel Dimensions

To understand the range of tunnel sizes used in real projects, a review of several hydropower tunnel junctions from around the world was carried out. The review showed that tunnel dimensions vary from project to project based on design and operational requirements.

As the main focus of this study was to investigate the effect of junction geometry on flow characteristics, simplified dimensions were used for the CFD models. Therefore, branch diameters of 3 m, 4 m and 5 m were chosen as representative sizes, while the main tunnel diameter was kept constant at 5 m throughout the analysis.

Table 2.1: Summary of Tunnel Junction Types and Dimensions Used in Existing Hydropower Projects Worldwide

Sr. No.	**Type of junctions in use Worldwide**	**Locations**	**Dimensions**
1.	T-Junction	a) Tehri HEP b) Sardar Sarovar Project (Gujarat) c)Lesotho Highlands Water Project (Lesotho) d)Gotthard Base Tunnel (Switzerland)	Tehri HEP T-junction dimensions Main tunnel diameter: 8.5 m Branches diameter: 5.75 m each. plan angle: 45°
2.	Y-Junction	a) Tehri Hydropower Project (Uttarakhand) b) Bhakra–Nangal Project (Himachal–Punjab) c)Nathpa Jhakri HEP (Himachal Pradesh) d)Koyna Hydroelectric Project (Maharashtra) e) Three Gorges Dam – China f) Itaipu Dam – Brazil/Paraguay g) Hoover Dam – USA	Bhakra–Nangal Project Y-junction dimensions Main tunnel diameter:4.88 m Branches diameter:3.35 m
3.	Confluence Junction	a) Koyna HEP (Maharashtra) b) Nathpa Jhakri HEP c)Itaipu Dam (Brazil/Paraguay) d)Robert-Bourassa Project (Canada)	Nathpa Jhakri HEP Confluence Junction dimensions. Upstream Diameter: 10.15 m Downstream Diameter: 6.6 m Junction nature: Transition junction.
4.	Trifurcation Junction	a)Srisailam Hydroelectric Project (Telangana–AP) b) Sharavathi HEP (Karnataka) c)Xiluodu Dam (China) d)Grand Coulee Dam (USA)	Srisailam Hydroelectric Project Trifurcation Junction dimensions. Main tunnel diameter: ≈ 7.0 m Three branch diameters: 3.5–4.0 m each Branch angles:25°
5.	Surge Tank / Surge Shaft Junction	a) Teesta-V HEP (Sikkim) b) Dulhasti HEP (J&K) c)Uri-II HEP (J&K) d)Limmern Pumped Storage Project (Switzerland) e) Snowy Mountains Scheme (Australia)	Teesta-V HEP Surge Tank Junction dimensions Pressure Tunnel Diameter:10.0 m Surge Shaft diameter: 29 m Junction throat diameter: 5.0 m
6.	Manifold-type Junction	a) Nathpa Jhakri HEP b)Three Gorges Dam (China) c)Itaipu Dam (Brazil/Paraguay)	Three Gorges Dam Manifold-type Junction dimensions Upstream of manifold Diameter: 12.8 m Branches from manifold diameter:7.0–7.5 m Junction type: Gradual manifold with multiple outlets Branch spacing:20–30 m
7.	Lateral / angled branch fitting	a)Tehri Hydroelectric Project b)Bhakra–Nangal Project c)Nathpa Jhakri Hydroelectric Project d)Three Gorges Dam (China)	Dimensions Not publicly available.
8.	Restricted-orifice surge tank junction	a)Upper Tamakoshi Hydropower Project (Nepal) b)Bhairabkunda / Punatsangchhu-I & II (Bhutan) c)Deriner Dam Hydropower Project (Turkey)	Upper Tamakoshi Hydropower Project Restricted-orifice surge tank junction dimensions Pressure Tunnel diameter (constructed): 7.40 m Restricted Orifice diameter: 5.00 Main surge chamber diameter (constructed): 24.0 m

2.6 Simulation Procedure

The geometry was then used for CFD simulations with the SolidWorks Flow Simulation software package. The software was used to investigate the flow features in different geometrical configurations of the pressure tunnel junctions.

Simulations were conducted to investigate:

- Velocity profile
- Turbulent flow generation
- Regions of flow separation
- Pressure profile

To keep the simulations consistent, boundary conditions were set as a Q1 = flow rate 100 m3/s, diameter of main pipe is 5m, inlet velocity, no slip wall conditions and flow properties for water at 25oC were assigned to all geometries. The results obtained with SolidWorks flow simulation were later used for comparing the different geometries of the junctions.

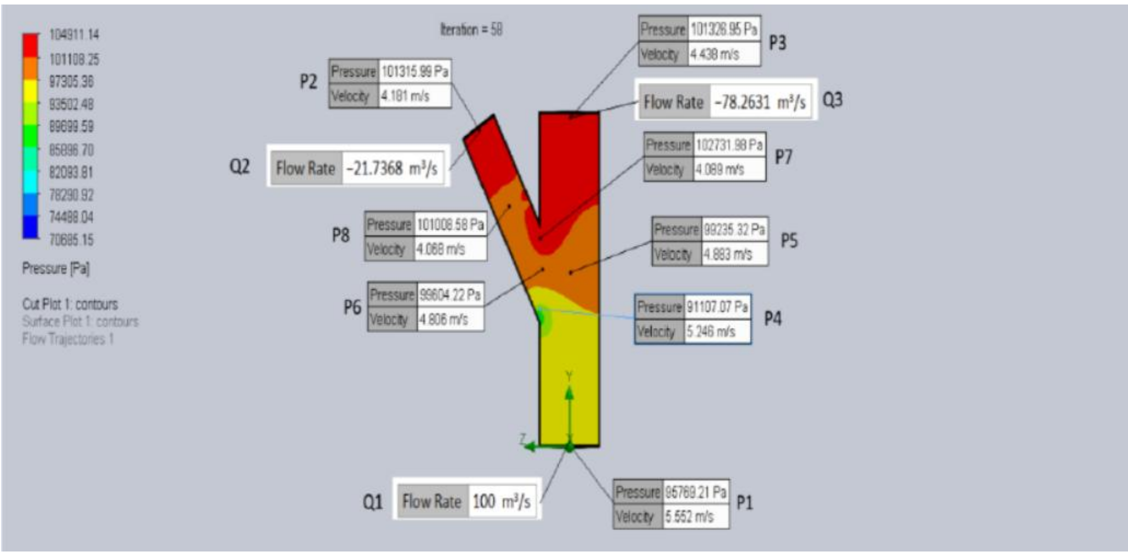


Fig.no.2.1: Probe Arrangement and Simulation Result for Diverging Flow

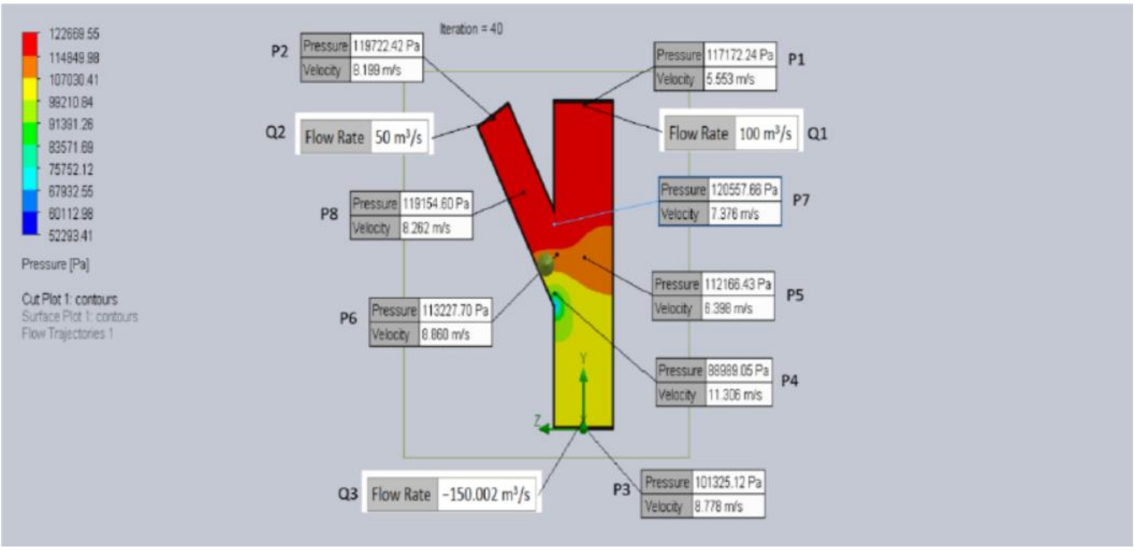


Fig.no.2.2: Probe Arrangement and Simulation Result for Converging Flow

III. Analysis And Discussion

3.1 Introduction

Computational Fluid Dynamics (CFD) simulations were carried out to investigate the effect of junction geometry on turbulence and flow separation in pressure tunnels. Simulations were performed for both diverging and converging flow conditions at junction angles of 30°, 45°, 60° and 90°. Main tunnel diameters of 3 m, 4 m and 5 m were considered.

The CFD software generated pressure values in Pascal (Pa), which were converted into pressure head (m of water) using:

$$H = \frac{P}{1000x9.81}$$

To simplify the analysis, only three probes located near the junction region were selected:
P4 – Upper side of the main tunnel
P6 – Central junction region
P7 – Lower side near the branch connection

These probes adequately represent the hydraulic behaviour in the region where flow interaction, turbulence generation and flow separation are expected to occur.

3.2Pressure Head Analysis

3.2.1 Diverging Flow

The pressure-head results for diverging flow indicate a gradual increase in pressure head with increasing junction angle.

For all diameters, P7 consistently recorded the highest-pressure head, followed by P6 and P4.

Observations:

[1] Pressure head increased from 30° to 90°.
[2] P7 showed the highest-pressure recovery.
[3] P4 consistently exhibited the lowest pressure head.

The difference between probes became more pronounced at larger junction angles.
For the 3 m diameter tunnel:

Table no. 4.1: Pressure Head Distribution at P4, P6 and P7 for Diverging Flow (3 m Diameter)

Angle	**P4**	**P6**	**P7**
30°	9.287	10.153	10.472
90°	10.256	10.350	10.751

A similar trend was observed for 4 m and 5 m diameters.

As the junction angle increases, the flow undergoes a sharper directional change. This causes a reduction in kinetic energy and promotes pressure recovery in the downstream region. Consequently, higher pressure heads are observed at P7.

The increase in pressure head near P7 suggests deceleration of flow and possible formation of recirculation zones close to the junction. These effects become more noticeable at 90°, where flow separation is expected to be strongest.

3.2.2 Converging Flow

The converging flow cases exhibited significantly higher-pressure heads than diverging flow.
For all diameters:
P7 > P6 > P4
remained valid.
Average pressure heads were:

Table no. 4.2: Average Pressure Head (m) at P4, P6 and P7 for Different Tunnel Diameters under Converging Flow Conditions

Diameter	**P4**	**P6**	**P7**
3 m	8.93	12.26	12.95
4 m	10.35	12.98	13.62
5 m	11.18	13.33	13.85

During converging flow, fluid streams merge at the junction, resulting in pressure build-up near the branch connection. The increase in pressure at P7 indicates that the junction region experiences a redistribution of energy from velocity head to pressure head.

The highest-pressure values were generally observed for the 90° junction, indicating increased hydraulic losses and stronger flow interaction.

3.3 Velocity Analysis

3.3.1 Diverging Flow

Velocity distributions showed a trend opposite to pressure head.
For most cases:
P4 > P6 > P7
Average Velocity Values

Table no. 4.3: Average Velocity at Selected Probe Locations (P4, P6 and P7) for Diverging Flow (Dia: 3m)

Probe	Average Velocity (m/s)
P4	4.03
P6	4.06
P7	3.62

Table no. 4.4: Average Velocity at Selected Probe Locations (P4, P6 and P7) for Diverging Flow (Dia: 4m)

Probe	Average Velocity (m/s)
P4	4.02
P6	3.94
P7	3.11

Table no. 4.5: Average Velocity at Selected Probe Locations (P4, P6 and P7) for Diverging Flow (Dia: 5m)

Probe	Average Velocity (m/s)
P4	3.41
P6	3.08
P7	2.90

The highest velocity values were generally recorded at P4. This indicates that the upper portion of the tunnel experiences flow acceleration immediately after interaction at the junction. Velocity decreases progressively towards P7 where pressure recovery occurs.

The reduction in velocity with increasing junction angle suggests increased turbulence generation and energy dissipation. At 90°, the abrupt flow redirection produces stronger disturbances and larger hydraulic losses.

3.3.2 Converging Flow

Converging flow produced substantially higher velocities than diverging flow.
Average velocities obtained were:

Table no. 4.6: Average Velocity Distribution at Selected Probes for Different Tunnel Diameters

Diameter	P4	P6	P7
3 m	11.07	8.43	5.72
4 m	8.09	5.07	3.57
5 m	7.13	3.80	3.05

The highest velocities were consistently recorded at P4. This indicates that flow merging at the junction creates a localized acceleration zone near the upper portion of the main tunnel. P7 exhibited the lowest velocity values due to pressure recovery and reduction in momentum after flow interaction.

3.4 Effect of Junction Angle

The influence of junction angle was clearly observed in both pressure and velocity distributions.

1. For a 30° junction angle, the flow experiences a smooth transition with minimal disturbance. This results in the lowest hydraulic losses and reduces the occurrence of flow separation. Additionally, the velocity distribution remains more uniform throughout the junction, indicating better hydraulic performance and flow efficiency.
2. For a 45° junction angle, the flow experiences moderate disturbance as it passes through the junction. This leads to a slight increase in pressure recovery and marks the beginning of turbulence formation. Although the flow remains relatively stable, minor flow irregularities start to develop compared to the 30° junction angle.
3. For a 60° junction angle, the flow experiences significant interaction between the merging streams, leading to increased velocity fluctuations. This enhances turbulence within the junction and creates stronger recirculation zones, indicating a greater tendency for flow separation and energy loss compared to smaller junction angles.
4. For a 90° junction angle, the flow undergoes maximum disturbance due to the abrupt change in direction. This causes the highest-pressure build-up and significantly reduces flow velocities within the junction. Strong flow separation and intense turbulence generation are observed, leading to greater energy losses and the least efficient hydraulic performance among all the junction angles studied.

The CFD results indicate that increasing junction angle increases the intensity of turbulence and flow separation. Larger angles force the fluid to undergo abrupt directional changes, leading to greater energy losses and non-uniform flow patterns.

3.5Effect of Tunnel Diameter

Comparison among the three diameters revealed that increasing tunnel diameter reduces flow velocity.

Velocity Trend: -

3 m > 4 m > 5 m

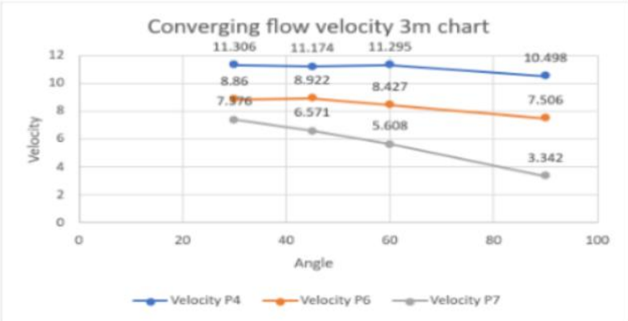


Fig.no.4.1: Converging flow Velocity for branch diameter 3m for each angle

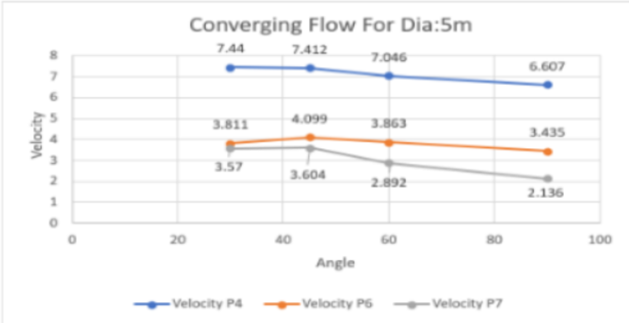


Fig.no.4.2: Converging flow Velocity for branch diameter 5m each angle

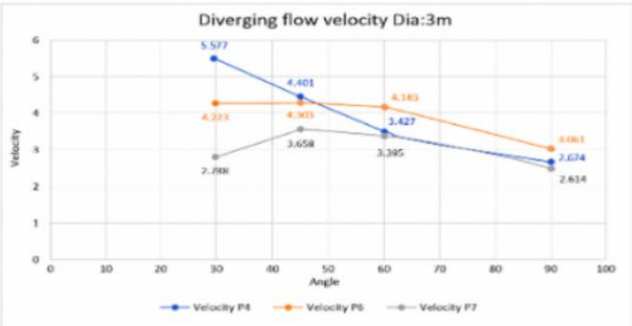


Fig.no.4.3: Diverging flow Velocity for branch diameter 3m each angle

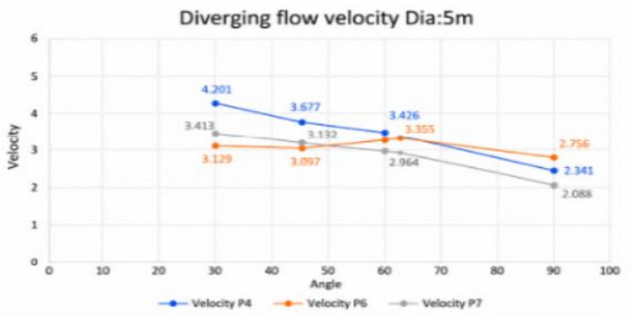


Fig.no.4.4: Diverging flow Velocity for branch diameter 5m each angle

Pressure Head Trend:

Pressure head generally increased slightly with increasing diameter.

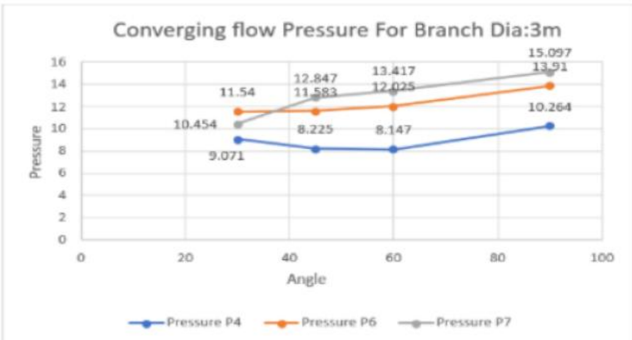


Fig.no.4.5: Converging flow pressure for branch diameter 3m for each angle

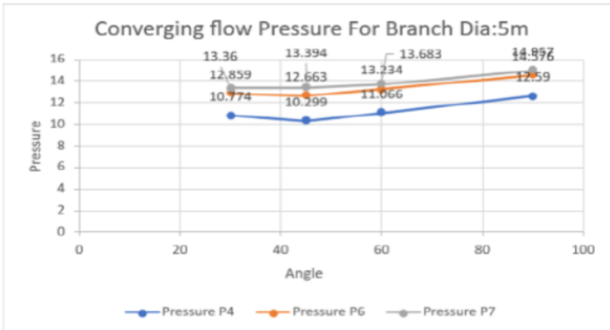


Fig.no.4.6: Converging flow pressure for branch diameter 5m each angle

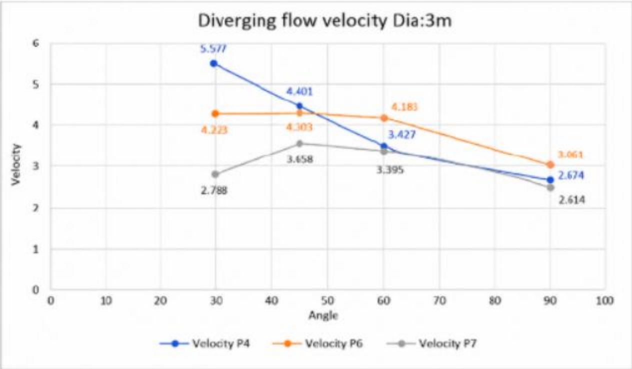


Fig.no.4.7: Diverging flow pressure for branch diameter 3m each angle

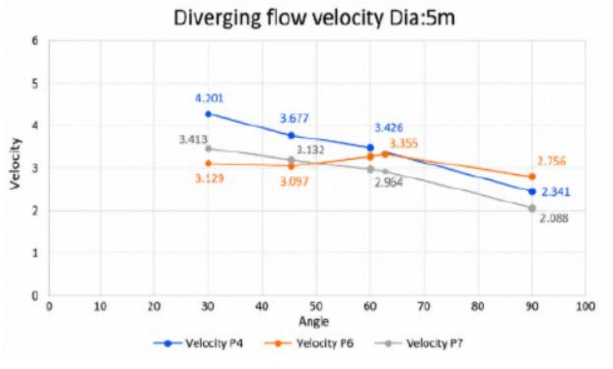


Fig.no.4.8: Diverging flow pressure for branch diameter 5m each angle

Summary:

Larger diameters provide a greater flow area, resulting in lower average velocities. The reduction in velocity decreases kinetic energy and allows pressure recovery within the junction region. Consequently, 5 m diameter tunnels exhibited the highest-pressure heads and lowest velocities.

3.6 Discussion on Turbulence and Flow Separation

The CFD results demonstrate that junction geometry significantly influences hydraulic behaviour within pressure tunnels. The pressure and velocity distributions indicate the presence of: flow acceleration near P4, pressure recovery near P7, velocity redistribution within the junction region, increased hydraulic losses at larger junction angles. The inverse relationship between pressure head and velocity confirms the energy exchange occurring within the junction. Higher pressure heads and lower velocities observed at 90° indicate stronger recirculation zones and greater turbulence intensity. Conversely, the 30° junction produced smoother flow patterns with reduced disturbance. These findings suggest that smaller junction angles are hydraulically more efficient and less susceptible to flow separation.

IV. Conclusion

In the present research, the effect of junction geometry on the turbulence and flow separation in pressure tunnels has been studied through Computational Fluid Dynamics (CFD). Numerical simulations are carried out for converging and diverging flow conditions at junction angles of 30°, 45°, 60° and 90° for tunnel diameters of 3 m, 4 m and 5 m. The hydraulic behaviour of the flow was studied by pressure head and velocity distributions achieved by strategically located monitoring probes around the junction region. The pressure values obtained from CFD simulations were converted to pressure head values expressed in metres of water column for hydraulic interpretation. In order to simplify the study and concentrate on the main interaction region of the flow, the probes P4, P6 and P7 for detailed investigation.

The analysis of pressure-head results revealed that probe P7 always had the highest-pressure head values and probe P4 usually had the lowest pressure head values. Such behaviour is indicative of pressure recovery in the vicinity of the branch connection and the effect of the junction geometry on the energy distribution in the tunnel system. Velocity analysis showed that pressure head and velocity were inversely related. Probe P4 always showed higher values of velocity while probe P7 was slower. This behaviour is consistent with basic fluid flow principles and confirms that energy is redistributed in the junction region.

The results also showed that the increase of the junction angle from 30° to 90° has a significant effect on the hydraulic performance of the tunnel. The increase of the larger junction angles produced higher flow disturbances, more intense recirculation zones and higher turbulence generation. The 90° junction configuration resulted in the highest-pressure build-up and the largest velocity decrease, which is indicative of enhanced flow separation and higher hydraulic losses.

On the other hand, the 30° junction showed a smoother flow transition with lower energy losses and lower turbulence intensity. The junction angle was smaller, allowing the fluid to turn more gently, which reduced separation and increased the hydraulic efficiency. Also noticed was the influence of tunnel diameter. Due to the larger flow area available, the flow velocity was generally reduced by increasing the tunnel diameter. Higher pressure heads and lower velocity gradients were observed in larger diameters resulting in better flow stability.

Comparison of converging and diverging flow conditions showed that converging flow resulted in higher pressure heads and higher velocities due to the interaction and combination of flow streams at the junction. The diverging flow had relatively lower velocities and a smoother pressure distribution.

Based on the results of the CFD obtained in this study, it can be concluded that the junction geometry plays a critical role in controlling the turbulence generation, pressure distribution and flow separation in pressure tunnels. Among the configurations studied, the 30° junction was the most hydraulically efficient, while the 90° junction produced the highest turbulence intensity and pressure loss.

Therefore, in the design of pressure tunnel junctions, smaller junction angles are recommended whenever the site conditions permit, as they contribute to improved flow behaviour, reduced energy losses and enhanced hydraulic performance.

Recommendations

Based on the results obtained from this study, the following recommendations are suggested for the design and analysis of pressure tunnel junctions:

- Smaller junction angles such as 30° should be preferred wherever possible, as they provide smoother flow and reduce energy losses.
- Large junction angles, especially 90°, should be avoided when feasible because they create higher turbulence and flow separation.
- Proper tunnel dimensions should be selected to maintain stable flow conditions and improve hydraulic performance.
- CFD analysis can be used during the design stage to identify potential problem areas and optimize junction geometry before construction.
- Smooth flow transitions should be provided at tunnel junctions to minimize turbulence and improve pressure recovery.
- Both pressure and velocity distributions should be considered during tunnel design to achieve efficient flow conditions.
- Future studies investigating junction dynamics and separation zones are required to fully conclude the hydraulic behaviour of different junction geometries..
- Experimental studies using physical models can be carried out to validate the CFD results obtained in this research.
- More advanced simulation techniques may be used in future work to study flow behaviour in greater detail.
- Further research can also examine the effects of sediment transport, cavitation and unsteady flow conditions in pressure tunnels.

References

[1]. P. Novak, A. I. B. Moffat, C. Nalluri, and R. Narayanan, Hydraulic Structures, 5th ed. Boca Raton, FL, USA: CRC Press, 2017.
[2]. I. E. Idelchik, Handbook of Hydraulic Resistance, 4th ed. New York, NY, USA: Begell House, 2007.
[3]. D. S. Miller, Internal Flow Systems, 2nd ed. Bedford, UK: BHRA, 1990.
[4]. H. K. Versteeg and W. Malalasekera, An Introduction to Computational Fluid Dynamics: The Finite Volume Method, 2nd ed. Harlow, UK: Pearson Education, 2007.
[5]. F. M. White, Fluid Mechanics, 8th ed. New York, NY, USA: McGraw-Hill Education, 2016.
[6]. B. R. Munson, T. H. Okiishi, W. W. Huebsch, and A. P. Rothmayer, Fundamentals of Fluid Mechanics, 7th ed. Hoboken, NJ, USA: Wiley, 2013.
[7]. Bureau of Indian Standards, IS 4880 (Part 3): Design of Tunnels Conveying Water. New Delhi, India: BIS.
[8]. Central Water Commission (CWC), Guidelines for Design of Hydraulic Structures. New Delhi, India.