FLOW UNDER HEAD

AbdelhafidBoussouf University Center - Mila
Science and Technology Institutes
Department of Civil Engineering and Hydrolic

The flow of a real fluid generates frictional forces due to viscosity and turbulence. The presence of these forces leads to a pressure drop, which is an irreversible transformation of mechanical energy into thermal energy. Studying fluid flow involves solving the Navier-Stokes equation. However, in practice, this equation can only be solved analytically by making simplifying assumptions. In particular, we must distinguish between two major types of flow: laminar and turbulent.

In fluid mechanics, **load flow** typically refers to the study and analysis of fluid (usually water, air, or other fluids) moving through a system, focusing on factors like pressure, velocity, and flow rate. It often involves evaluating how fluid is "loaded" through various parts of a piping or channel network. Here's a quick breakdown of what load flow entails in different contexts.

In fluid mechanics, **load flow** typically refers to the study and analysis of fluid (usually water, air, or other fluids) moving through a system, focusing on factors like pressure, velocity, and flow rate. It often involves evaluating how fluid is "loaded" through various parts of a piping or channel network. Here's a quick breakdown of what load flow entails in different contexts:

IV.1. Load Flow in Pipe Networks

- This refers to determining the pressure distribution, flow rates, and velocities in a network of pipes.
- **Key Factors**: Pipe diameter, length, friction factors, viscosity, and the impact of pumps or valves.
- **Applications**: Water distribution networks, HVAC systems, and industrial piping systems.
- **Analytical Methods**: Bernoulli's equation, continuity equation, and sometimes more complex iterative methods for network analysis like the Hardy Cross method.

IV.1.1. Load Flow in Power Generation Systems

- Although primarily a term in electrical engineering, **load flow analysis** is often applied to systems like cooling water flows in power plants.
- **Key Considerations**: Flow rate adjustments to match thermal load requirements and ensure efficient cooling.

IV.1.2. Hydraulic and Thermal Load Flow Analysis

- When systems involve both fluid dynamics and heat transfer (such as in thermal management systems), load flow analysis considers both **hydraulic loads** (flow rates, pressures) and **thermal loads** (temperature changes, heat flux).
- **Applications**: Heat exchangers, radiator systems, and HVAC.

IV.1.3. Load Flow Software and Simulation Tools

- To accurately predict load flow in complex systems, computational tools like **CFD (Computational Fluid Dynamics)** software, EPANET, and specialized engineering software are often used.
- **CFD Simulations** provide detailed insights into pressure drops, flow uniformity, and can help visualize the effects of complex geometries, especially when turbulence and transient behaviors are present.

Would you like details on any specific application of load flow or simulation methods?

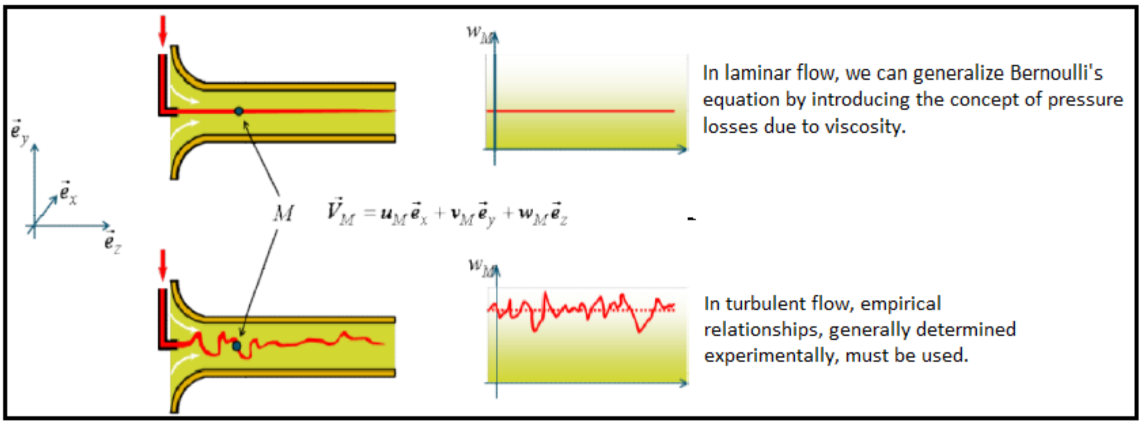


Figure I.V.1. Flow Regimes:

The experiments conducted by Osborne Reynolds (1883) (figure IV.20) involved injecting a colored liquid into a moving mass of liquid within a glass tube. When the drain valve is slightly opened, the colored liquid begins to flow slowly through the glass tube without mixing with the other layers of liquid. The streamlines in the tube remain straight, so the color remains uniform. This flow is called **laminar flow**.

If the valve opening is increased, the flow speed rises, and oscillations can be observed in the tube. The increase in speed causes the colored liquid to mix with the other layers of liquid, with particles being projected in all directions. This flow is called **turbulent flow**.

The **Reynolds experiment** is a classic experiment in fluid mechanics, designed to demonstrate and classify different types of flow regimes—specifically **laminar**, **transitional**, and **turbulent** flows—based on Reynolds number, a dimensionless quantity used to predict flow patterns in different fluid flow situations.

IV.2. Principle

The Reynolds number (Re) is calculated as:

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

where :

- ρis the fluid density,
- v is the fluid velocity,
- D is the diameter of the tube,
- μ is the dynamic viscosity, and
- νis the kinematic viscosity of the fluid.

Reynolds found that:

- For Re<2000, the flow is **laminar** (smooth, orderly flow).
- For Re>4000, the flow is **turbulent** (chaotic, mixing flow).

- For values between 2000 and 4000, the flow is in the **transitional** regime.

IV.2.1. Procedure

1. **Setup the Apparatus**: Attach the transparent tube to the water source and set up the dye injector to introduce dye at the entrance of the tube.
2. **Introduce Flow**: Start the water flow and adjust it to a slow rate. Introduce a thin stream of dye at the tube's entrance.
3. **Observe Laminar Flow**: At low flow rates (low Reynolds number), observe that the dye remains as a straight, undisturbed line, indicating **laminar flow**.
4. **Increase Flow Rate**: Gradually increase the flow rate. At some point, the dye will start to fluctuate, and the line may begin to waver, indicating a **transitional flow** regime.
5. **Observe Turbulent Flow**: As the flow rate is further increased, the dye will mix chaotically with the water, showing **turbulent flow**.
6. **Record Reynolds Number**: For each flow type, calculate the Reynolds number using the flow velocity, tube diameter, and fluid properties.

IV.2.2. Analysis

By observing the dye pattern, you can classify the flow type and verify the approximate Reynolds number range for each flow regime. This experiment shows how flow characteristics change with velocity and allows for a practical understanding of laminar and turbulent flows.

This experiment is foundational in fluid mechanics as it illustrates the effects of viscosity and velocity on flow behavior and the critical role of Reynolds number in predicting flow regimes.

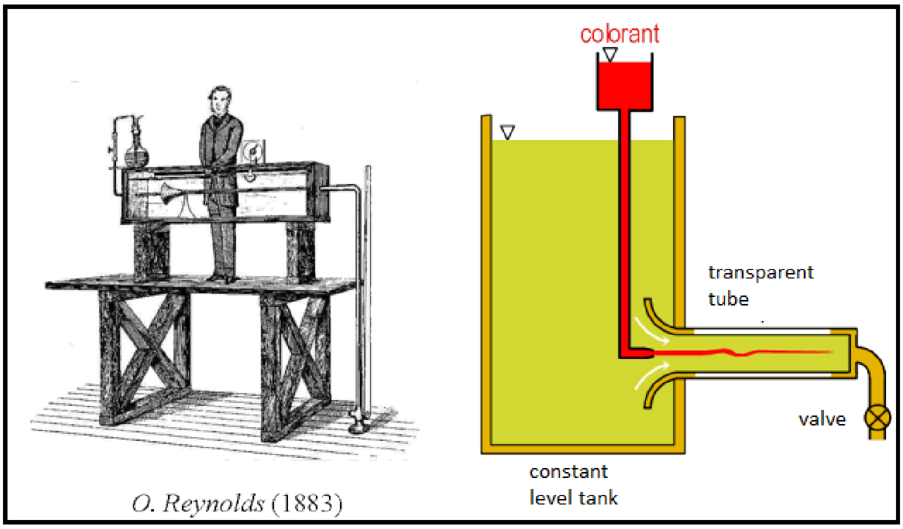


Figure IV.2. Reynolds experiment: experimental device.

For tubes with variable diameters and fluids of different viscosity and density, there exists a relationship that predicts the transition from laminar to turbulent flow. Between the two values of the Reynolds number (Re), the flow regime is classified as intermediate (if Re = 2000 = Rec, the regime is transitional, critical, or also called incipient turbulent flow). In most common hydraulic problems (except for underground hydraulics), turbulent flow is encountered. When the flow regime is turbulent, friction increases, thereby increasing the pressure drop in a pipeline.

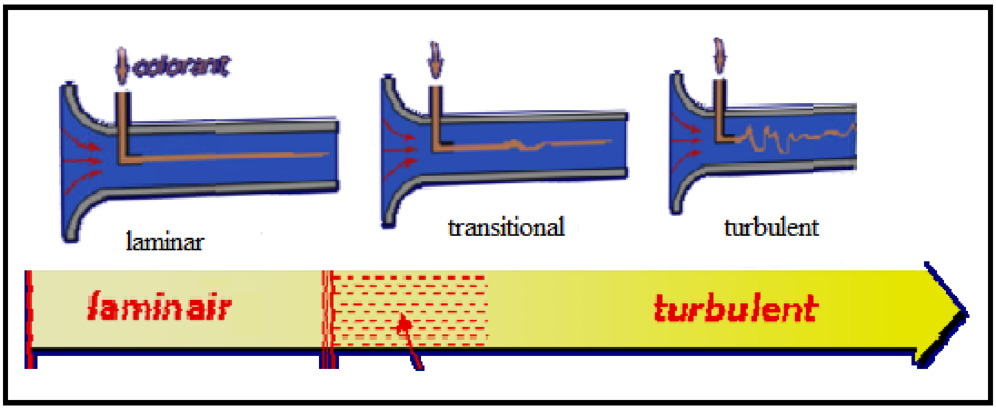


Figure IV.3. Intermediate regime (transient) Rec.

IV.3. Laminar Flow and Regular Head Losses

In laminar flow, fluid particles move in smooth, parallel layers with minimal mixing between them. This type of flow generally occurs at low velocities and with fluids of high viscosity or in pipes of small diameter, where the Reynolds number (Re) is below 2000.

IV.3.1. Key Points

1. **Characteristics of Laminar Flow**: In laminar flow, the fluid velocity profile across the pipe is parabolic, with the maximum velocity at the center and gradually decreasing towards the walls. This smooth and orderly flow minimizes internal fluid mixing and shear stresses.
2. **Regular (Frictional) Head Losses**: In laminar flow, head losses are caused mainly by friction between the fluid layers and the pipe walls. Theselossescanbecalculâtesusing the Hagen-Poiseuille.

Let us start from the Navier-Stocks equation obtained for an incompressible Newtonian fluid:$\rho = \frac{d\vec{V}}{dt} - \vec{\nabla}\rho + \mu\Delta\vec{V} + \rho\vec{g}$

For a steady flow, we have:

$$\frac{d\vec{V}}{dt} = \cancel{\frac{\partial\vec{V}}{dt}}_{0} + (\vec{V}\vec{\nabla})\vec{V} = (\vec{V}\vec{\nabla})\vec{V} = \frac{1}{2}\vec{\nabla}V^2 - \vec{V}\wedge(\vec{V}\wedge\vec{\nabla})$$

where :$\frac{1}{2}\vec{\nabla}V^2 - \vec{V}\wedge(\vec{V}\wedge\vec{\nabla}) = -\vec{\nabla}\rho + \mu\Delta\vec{V} + \rho\vec{g}$

$$\vec{\nabla}\left(\frac{1}{2}\rho V^2\right) - \rho\vec{V}\wedge(\vec{\nabla}\wedge\vec{V}) = -\vec{\nabla}\rho + \mu\Delta\vec{V} - \vec{\nabla}(\rho g z)$$

$$\vec{\nabla}\left(P + \rho g z + \frac{1}{2}\rho V^2\right) = \mu\Delta\vec{V} + \rho\vec{V}\wedge(\vec{\nabla}\wedge\vec{V}) = \mu\Delta\vec{V} + 2\rho\vec{V}\wedge\Omega$$

$$\vec{\nabla}\left(P+\rho g z+\frac{1}{2}\rho V^{2}\right)=\mu\Delta\vec{V}+2\rho\vec{V}\wedge\Omega$$

Let us project this vector equality along a streamline and then onto each of the three axes of a Cartesian frame:

$$\begin{cases}\dfrac{\partial}{\partial x}\left(P+\rho g z+\dfrac{1}{2}\rho V^{2}\right)=\mu\Delta V_{x}\\ \dfrac{\partial}{\partial y}\left(P+\rho g z+\dfrac{1}{2}\rho V^{2}\right)=\mu\Delta V_{y}\\ \dfrac{\partial}{\partial z}\left(P+\rho g z+\dfrac{1}{2}\rho V^{2}\right)=\mu\Delta V_{z}\end{cases}$$

Let us suppose that the laminar flow occurs along the x axis. Under these conditions, we have:

$$\begin{cases}V_{w}=V\\ V_{y}=0\\ V_{z}=0\end{cases} if\ we\ set\ \left(P+\rho g z+\frac{1}{2}\rho V^{2}\right)=Pt\quad\begin{cases}\dfrac{\partial Pt}{\partial x}=\mu\Delta V\\ \dfrac{\partial Pt}{\partial y}=0\\ \dfrac{\partial Pt}{\partial z}=0\end{cases}$$

$$Pt(x,y,z)=Pt(x)$$

$$\frac{Pt}{dx}=\mu\Delta V=\mu\left(\frac{\partial^{2}}{\partial x^{2}}+\frac{\partial^{2}}{\partial y^{2}}+\frac{\partial^{2}}{\partial z^{2}}\right)$$

According to the continuity equation we have; $\vec{\nabla}\vec{V}=0\implies\frac{\partial V_{x}}{\partial x}\cancel{+}\frac{\partial V_{y}}{0\partial y}\cancel{+}\frac{\partial V_{z}}{0\partial z}=0$

$$\frac{\partial V_{x}}{\partial x}=0\iff\frac{\partial V}{\partial x}=0$$

$$\frac{dPt}{dx}=\mu\left(\frac{\partial^{2}V}{\partial y^{2}}+\frac{\partial^{2}V}{\partial z^{2}}\right)=C^{te}\forall(x,y,z)$$

We can deduce that the load varies linearly with the distance traveled by the fluid.

$$Pt_{1}=Pt_{2}+\Delta P_{t}$$

$$\frac{dPt}{dx}=\frac{P_{t2}-P_{t1}}{X_{2}-X_{1}}=\frac{\Delta P_{t}}{X_{2}-X_{1}}$$

$$\Delta P_{t}=-\frac{dP_{t}}{dx}(X_{2}-X_{1})$$

$$P_{t1}=P_{t2}-\frac{dPt}{dx}(X_{2}-X_{1})$$

$$\implies P_{1}+\rho g z_{1}+\frac{1}{2}\rho V_{1}^{2}=P_{2}+\rho g z_{2}+\frac{1}{2}\rho V_{2}^{2}-\frac{dPt}{dx}(X_{2}-X_{1})$$

total pressure (1) = total pressure (2) - regular pressure loss$\Delta P_t > 0$

It then remains to characterize the total pressure gradient$\frac{dP_t}{dx}$

IV.3.2. Poiseuille Flow

Poiseuille flow describes the behavior of a viscous, incompressible fluid flowing steadily through a long, straight pipe with a constant circular cross-section under laminar conditions. Named after Jean Léonard Marie Poiseuille, this flow type is governed by the principles of laminar flow, specifically when the Reynolds number (Re) is below approximately 2000.

- **Parabolic Velocity Profile**: In Poiseuille flow, fluid particles move in parallel layers, with the highest velocity at the center and zero velocity at the pipe walls (due to the no-slip boundary condition).

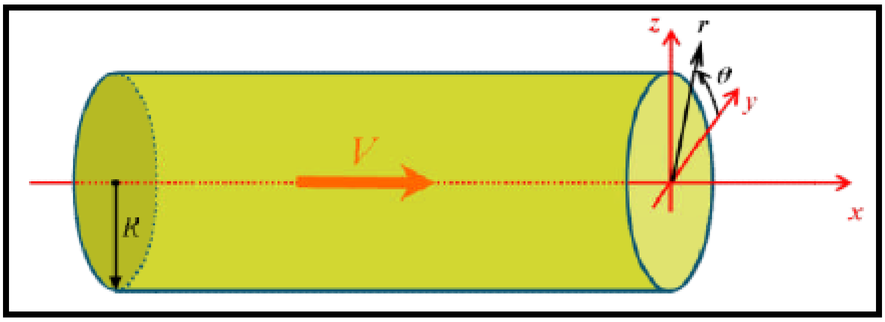


Figure IV.4. Flow of a viscous fluid in a cylindrical tube

$\vec{V} = V\vec{e_x} \Longrightarrow V_\theta = 0$under these conditions we can write$\frac{dPt}{dx} = \mu\Delta V$

The continuity equation imposes$\vec{\nabla}\vec{V} = 0 \Longrightarrow \frac{1}{r}\frac{\partial}{\partial r}(rV_r) + \frac{1}{r}\frac{\partial V_\theta}{\partial\theta} + \frac{\partial V_x}{\partial x} = 0$

$$\frac{\partial V_x}{\partial x} = 0 \Leftrightarrow \frac{\partial V}{\partial x} = 0$$

And the geometry of the system is such that there is symmetry of revolution

$\frac{\partial}{\partial\theta} = 0 \Leftrightarrow \frac{\partial V}{\partial\theta} = 0$ so finally$V(x,r,\theta) = V(r)$

Consequently, the Laplacian is expressed as

$$\Delta V = \frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial V}{\partial r}\right) + \underbrace{\cancel{\frac{1}{r^2}\frac{\partial^2 V}{\partial\theta^2}}}_{0} + \underbrace{\cancel{\frac{\partial^2 V}{\partial x^2}}}_{0} = \frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial V}{\partial r}\right)$$

$$\frac{dPt}{dx} = \frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial V}{\partial r}\right) = C^{te} = A$$

It is then possible to coat the velocity profile: $V(r)$by simple integration

$$\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial V}{\partial r}\right) = \frac{A}{\mu}r \Longrightarrow r\frac{dV}{dr} = \frac{A}{\mu}\frac{r^2}{2} + B$$

$$\Longrightarrow \frac{dV}{dr} = \frac{A\,r}{\mu\,2} + \frac{B}{r} \Longrightarrow V(r) = \frac{Ar^2}{4\mu} + B\ln r + C$$

B and C: constants to be determined using the boundary conditions

In contact with the walls of the pipe, r=R, the fluid is immobile.

$$V(R) = 0 \Longrightarrow \frac{A}{\mu}\frac{R^2}{4} + B\ln R + C = 0$$

On the axis of the pipe, $r = 0$, the velocity is necessarily of finite value:

$$V(0) \neq \infty \Longrightarrow Bln0 + C \neq \infty \Longrightarrow B = 0 \Longrightarrow C = -\frac{A}{\mu}\frac{R^2}{4}$$

$$V(r) = 0 \ \Longrightarrow \frac{A}{4\mu}(R^2 - r^2)$$

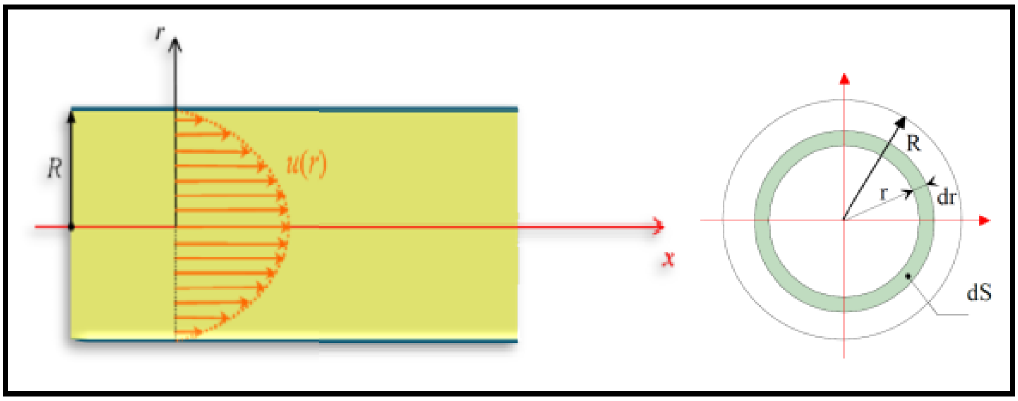


Figure IV.5. parabolic velocity profile.

Calculation of the volume flow rate through a section of the tube (figure.IV.23.)

$$dq_v = V(r)dS \ or \ ds = 2\pi r dr$$

$$q_v = \int_0^R V(r)2\pi r dr = -\frac{A}{4\mu}2\pi\int_0^R (R^2 - r^2) r dr$$

$$q_v = -2\pi\frac{A}{4\mu}\left[R^2\frac{r^2}{2} - \frac{r^4}{4}\right]_0^R = -2\pi\frac{A}{4\mu}\frac{R^4}{4} = -\pi\frac{A}{8\mu}R^4$$

Or $\frac{dPt}{dx} = A \ and \ R = \frac{D}{2}$

$$q_v = -\frac{\pi}{128\mu}\left(\frac{dPt}{dx}\right)D^4$$

We can then define an average velocity of the flow: $V_m = \frac{q_v}{S}$

Moreover, if we consider a pipe of length L, the total pressure loss is expressed.

$$\Delta P_t = (Pt_1 - Pt_2) = \int_2^1 \frac{dPt}{dx}dx \quad with \ \frac{dPt}{dx} = C^{te}$$

$$\Delta Pt = \frac{dPt}{dx}\int_{2}^{1} dx = \frac{dPt}{dx}(X_1 - X_2) \Longrightarrow \Delta P_t = -\left(\frac{dPt}{dx}\right) D^4$$

$$q_v = -\frac{\pi}{128\mu}\left(\frac{\Delta P_t}{L}\right) D^4$$

We can express the total pressure loss as a function of the flow rate or the average flow velocity.

$$q_v = V_m S = \frac{\pi}{128\mu}\left(\frac{\Delta P_t}{L}\right) D^4 \Longrightarrow \Delta P_t = 128\mu L V_m \left(\frac{S}{\pi D^4}\right) = 128\mu L V_m \frac{\pi D^2/4}{\pi D^4}$$

$$\Delta P_t = \frac{32\,\mu\,L\,V_m}{D^2}$$

It is customary to express a pressure loss as a function of the kinetic pressure of flow in the pipe. Kinetic pressure is generated by movement (it corresponds to kinetic energy per unit volume) and is expressed. $\frac{1}{2}\rho V_m^2$

In this case: $\Delta P_t = \frac{32\,\mu\,L V_m}{D^2} = \left(\frac{32\,\mu\,L\,V_m}{D^2}\frac{2}{\rho V_m^2}\right)\frac{1}{2}\rho V_m^2$

$$\frac{64\,\mu\,L}{\rho V_m D^2} = \frac{64\,\mu}{\rho V_m D}\frac{L}{D} = \frac{64\,L}{Re\,D} = \lambda\frac{L}{D}$$

So for a laminar flow in a pipe, we have$\Delta Pt = \lambda\frac{L}{D}\frac{1}{2}\rho V_m^2$ with $\lambda\frac{64}{Re}$dimensionless coefficient is called a pressure loss coefficientregularthis is only valid forRe<2000.

From a practical point of view, this formulation is particularly suitable for evaluating all the pressure losses caused by a hydraulic circuit comprising a succession of different sections of pipe, as illustrated by figure.IV.24.

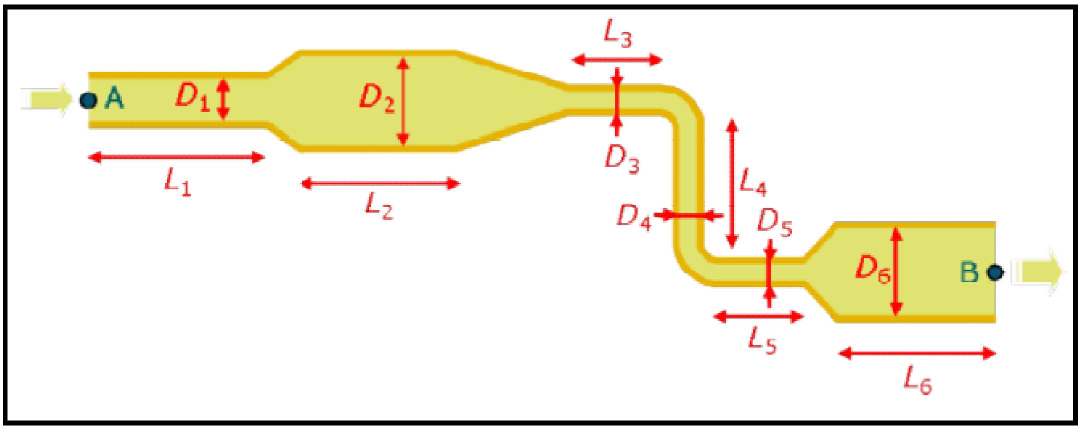


Figure IV.6. A hydraulic circuit comprising a succession of different pipe sections.

We can then generalize the Bernoulli equation

$$P_{tA} = P_A + \rho g Z_A + \frac{1}{2}\rho V_A^2$$

$$P_{tB} = P_B + \rho g Z_B + \frac{1}{2}\rho V_B^2$$

$$\Delta P_t = \lambda_1 \frac{L_1}{D_1}\frac{1}{2}\rho V_1^2 + \lambda_2 \frac{L_2}{D_2}\frac{1}{2}\rho V_2^2 + \cdots + \lambda_6 \frac{L_6}{D_6}\frac{1}{2}\rho V_6^2 + \ singular\ pressure\ losses$$

$$P_{tA} = P_{tB} + \Delta P_t$$

$$P_A + \rho g Z_A + \frac{1}{2}\rho V_A^2 = P_B + \rho g Z_B + \frac{1}{2}\rho V_B^2 + \sum_i \lambda_i \frac{L_i}{D_i}\frac{1}{2}\rho V_i^2 + singular\ pressure\ losses$$

This equation remains valid even if the flow is not laminar ($Re < 2000$) the only difference lies in the expression of the regular pressure loss coefficient whichmust be determined experimentally or taken from charts$\lambda \neq \frac{64}{Re}$

For a turbulent flow, $\lambda \neq \frac{64}{Re}$the regular head loss coefficient is determined by the Colebrook empirical equation.

$$\frac{1}{\sqrt{\lambda}} = -2\log_{10}\left[\frac{\varepsilon/D}{3.7} + \frac{2.51}{Re\sqrt{\lambda}}\right]$$

Allows the calculation of the coefficientλ this equation is an implicit equation that is not easy to handle; we will instead use the Moody diagram, drawn from the previous equationfigure IV.25.

IV.3.3. Singular Pressure Losses – Bélanger's Theorem

Singular pressure losses (or localized pressure losses) are energy losses in a fluid flow caused by specific elements like bends, sudden expansions or contractions, valves, and other obstructions within a conduit. Unlike regular pressure losses, which are related to the length of the conduit and wall friction, singular pressure losses are localized and depend on sudden changes in geometry or elements that disturb the flow.

Bélanger's theorem, named after the hydraulic engineer Jean-Baptiste Bélanger, is a formula used in hydraulics and fluid mechanics to estimate singular pressure losses in a flow. This theorem uses a singular pressure loss coefficient, often denoted as KKK, which depends on the geometry and characteristics of each encountered singularity.

IV.3.4. Formulation of Bélanger's Theorem

In an incompressible flow, Bélanger's theorem gives the singular pressure losses Δhsin the form:

$$\Delta h_s = K\frac{V^2}{2g}$$

where:

- Δhs is the singular pressure loss (m),
- K is the singular loss coefficient specific to the singularity considered,
- V is the average flow velocity of the fluid in the conduit (m/s),
- g is the acceleration due to gravity (9.81 m/s²).

The coefficient K varies according to the nature of the singularity. For example:

- For a bend, K depends on the angle and roughness of the bend.
- For a valve, K depends on the type and opening level of the valve.
- For a sudden expansion or contraction, K is related to the ratio of the inlet and outlet areas.

IV.3.5. Applications of Bélanger's Theorem

Bélanger's theorem is essential for correctly sizing conduit systems and predicting pressure losses in hydraulic and ventilation networks. By accounting for singular losses, pressure and power requirements for maintaining a desired flow rate in a system can be more accurately estimated, which is particularly useful in water distribution networks, cooling circuits, and ventilation and heating systems.

Consider the flow of an incompressible fluid in a horizontal pipe with a sudden widening (Figure IV.25.), which constitutes a singularity:

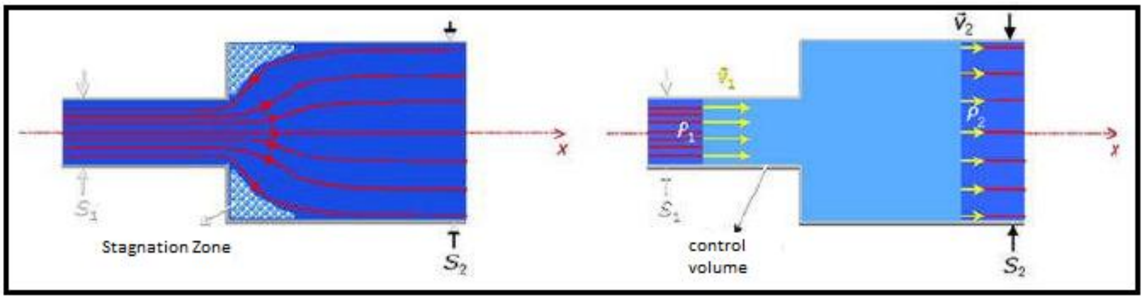
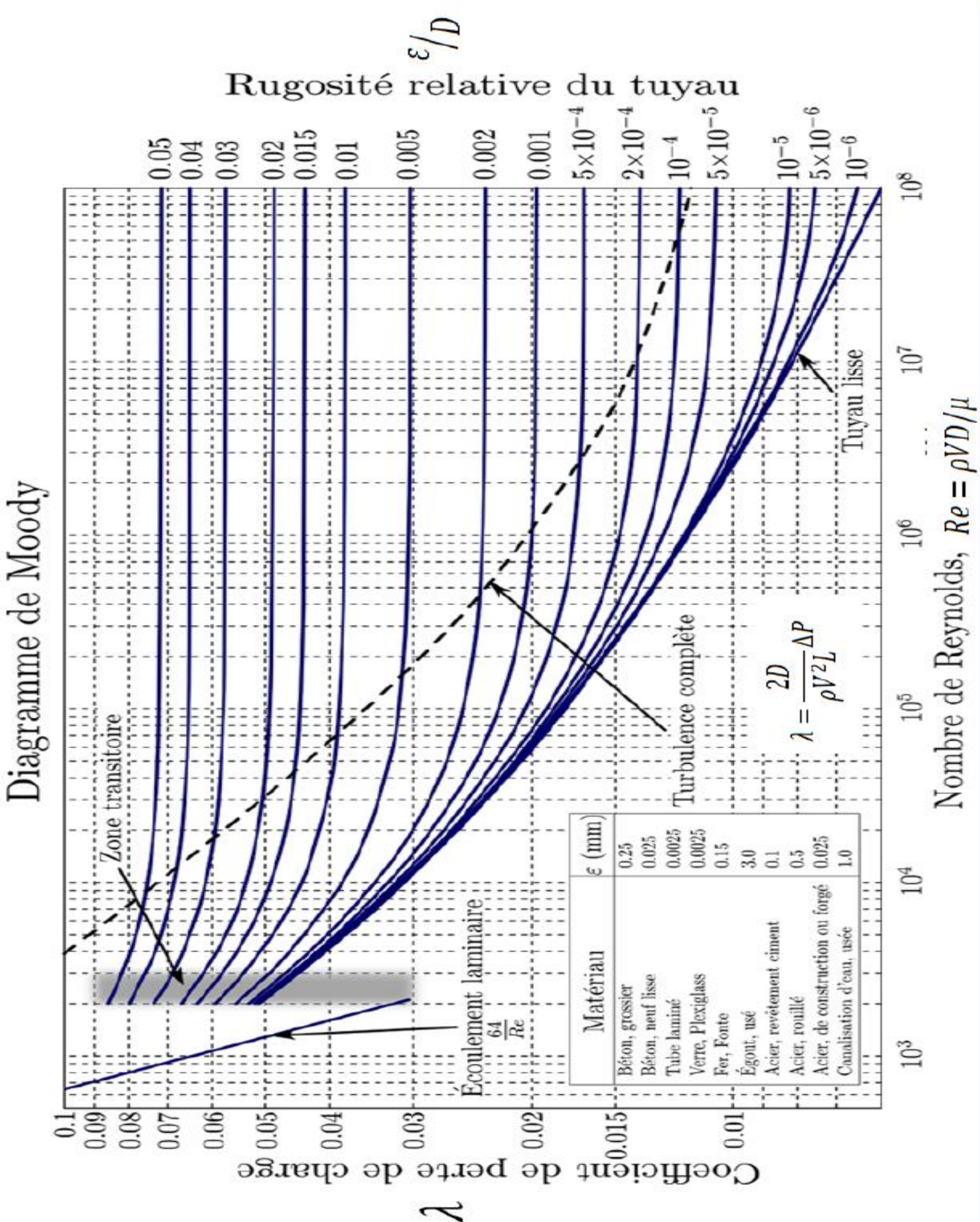


Figure IV.7. Horizontal pipe with a sudden widening

Due to its inertia, the fluid does not completely follow sudden changes in direction, resulting in turbulence zones where energy is dissipated. These areas, where the fluid is mostly stagnant, are responsible for singular pressure losses. We select a control volume on which Euler's theorem is applied. It is assumed that on the same cross-section (upstream and downstream of the contraction), velocities and pressures are uniform, so regular pressure losses during the contraction are neglected. Under these conditions, by projecting onto the x-axis, we obtain:



$q_m(V_2 - V_1)$: Projection on x of the resultant of the forces exerted on the control volume.

- upstream thrust $+ P_1 S_1$
- downstream counter-thrust $P_2 S_2$

- thrust of the vertical wall on the stagnant part of the fluid:$+P(S_2 - S_1)$(hydrostatic law)

$$q_m(V_2 - V_1) = P_1S_1 - P_2S_2 + P(S_2 - S_1) = (P_2 - P_1)S_2 \Longrightarrow q_m(V_2 - V_1) = (P_2 - P_1)S_2$$

However, since the fluid is assumed to be incompressible, we must have conservation of the mass flow rate: $q_m = \rho V_2 S_2 = \rho V_1 S_1 \Longrightarrow \quad q_m(V_2 - V_1) = \rho V_2 S_2(V_2 - V_1) = (P_2 - P_1)S_2$

$$P_1 = P_2 + \rho V_2^2 - \rho V_2 V_1 = P_2 + \frac{1}{2}\rho V_2^2 + \frac{1}{2}\rho V_2^2 - \rho V_2 V_1 + \frac{1}{2}\rho V_1^2 - \frac{1}{2}\rho V_1^2$$

$$\frac{1}{2}\rho(V_1 - V_2)^2$$

$$P_1 + \frac{1}{2}\rho V_1^2 = P_2 + \frac{1}{2}\rho V_2^2 + \frac{1}{2}\rho(V_1 - V_2)^2$$

pressure loss due to the singularity

Let us express this pressure loss as a function of the kinetic pressure in the upstream pipe:

$$\frac{1}{2}\rho(V_1 - V_2)^2 = \frac{1}{2}\rho\left(V_1 - \frac{S_1}{S_2}\right)^2 = \frac{1}{2}\rho V_1^2\left(1 - V_1\frac{S_1}{S_2}\right)^2$$

$$P_1 + \frac{1}{2}\rho V_1^2 = P_2 + \frac{1}{2}\rho V_2^2 + \frac{1}{2}\rho V_1^2 K$$

K=$\left(1 - V_1\frac{S_1}{S_2}\right)^2$Coefficient of singular pressure loss due to sudden widening (dimensionless).

We can thus complete the generalized Bernoulli equation.

$$P_A + \rho g z_A + \frac{1}{2}\rho V_A^2 + P_B + \rho g z_B + \frac{1}{2}\rho V_B^2 + \sum_i \lambda_i \frac{L_i}{D_i}\frac{1}{2}V_i^2 + \sum_j K_j \frac{1}{2}\rho V_j^2$$

Coefficient of singular pressure loss associated with each singularity encountered during the flow.

IV.3.6. Application Example

Suppose there is a conduit with a partially closed valve causing a singular pressure loss. If the fluid velocity is 3 m/s and the coefficient K for this valve is 0.8, the singular pressure loss will be:

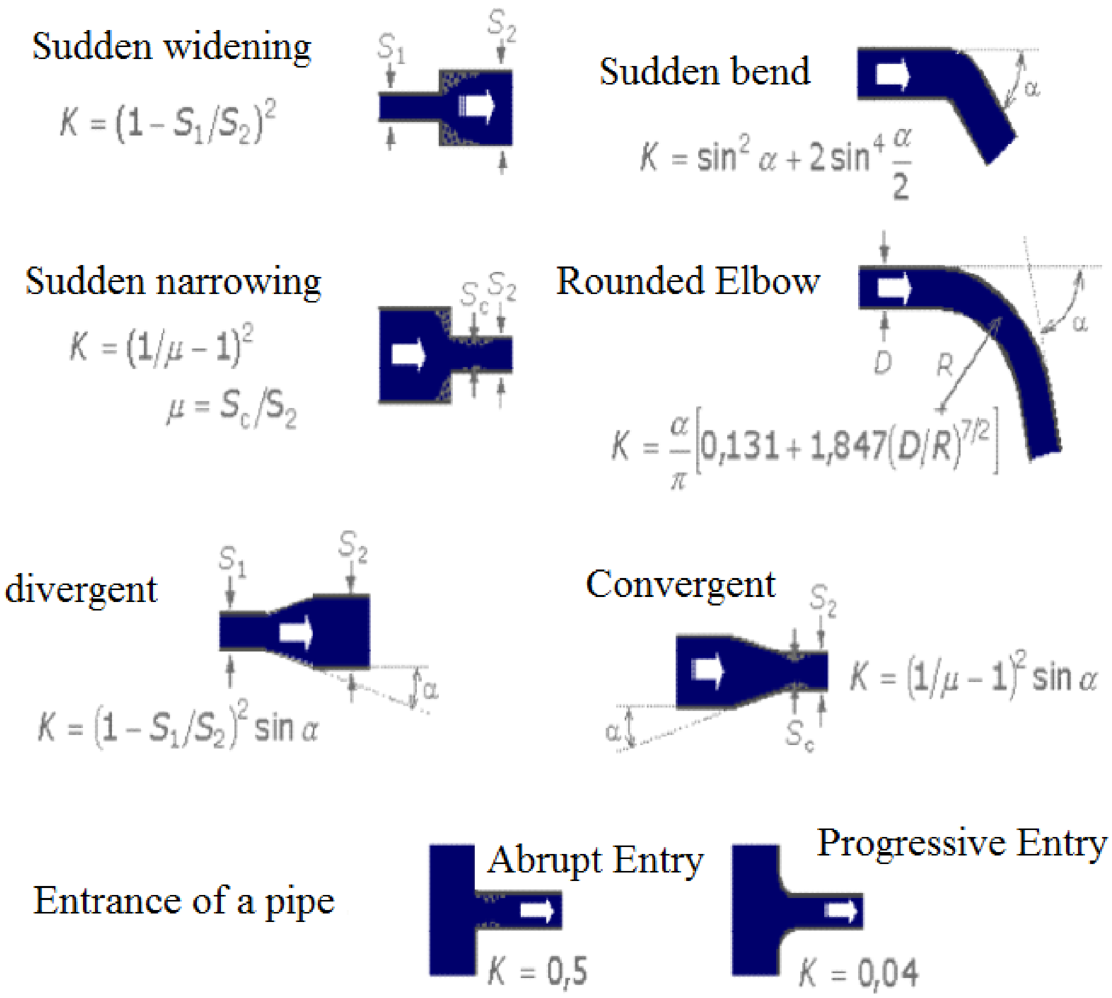
$$\Delta h_s = 0.8\frac{3^2}{2 \times 9.81} \approx 0.37$$

This means an additional water height of 0.37 m would be required to compensate for this energy loss due to the valve.

Here are some typical examples of **singularities** that cause pressure losses in fluid flow systems:

- **Bends and Curves**: Changes in direction (often at 90° or 45°) force the fluid to abruptly change its path, creating turbulence and recirculation zones. The sharper the angle, the greater the pressure loss.
- **Sudden Contractions and Expansions**: When a pipeline suddenly narrows or widens, turbulence zones appear due to abrupt changes in cross-section. The pressure loss depends on the ratio between the inlet and outlet sections.
- **Valves and Check Valves**: These flow control devices (such as butterfly valves, ball valves, and check valves) cause pressure losses, especially when they are partially closed.
- **T-Joints and Branches**: In T-joints, the fluid is redirected to a new branch, causing flow disturbances. The pressure loss depends on the geometry and angle of the connection.
- **Grids, Filters, and Screens**: These elements add resistance to fluid passage, resulting in pressure losses due to friction and reduced flow area.
- **Pipe Inlets and Outlets**: Flowing from a reservoir into a pipeline (or vice versa) often creates recirculation zones. For example, inlets into a pipe without profiled fittings increase singular pressure losses.
- **Pipe Turns**: Gradual or sharp changes in direction through turns cause pressure loss due to accumulated turbulence and recirculation zones.

These singularities should be considered in system design to ensure efficient flow, limit pressure losses, and optimize the energy efficiency of the flow system.



IV.3.7. Conclusion

Bélanger's theorem is a fundamental tool for evaluating singular pressure losses in fluid flow systems. It helps anticipate the effect of each singularity in a circuit, thereby improving the accuracy of fluid system sizing and energy efficiency.

IV.4. Water Distribution Network Calculations

Designing and calculating water distribution networks involves determining the flow rates, pressure losses, and pipe sizing required to efficiently deliver water from a source to various endpoints, such as residential, commercial, or industrial buildings. Here's a breakdown of the key steps and considerations in these calculations:

IV.4.1. Flow Rate Requirements

- **Determine Demand**: Calculate the required water flow rates for each endpoint. This depends on the population size, type of buildings, usage patterns, and peak demand times.
- **Demand Estimation**: Use standards and guidelines for domestic, commercial, and industrial water demand to estimate peak and average flow rates. The total demand at each junction of the network is the sum of the demands from connected endpoints.

IV.4.2. Pipe Sizing

- **Select Pipe Material**: Different materials (e.g., PVC, steel, copper) have varying roughness coefficients, affecting flow resistance. Choose materials based on durability, cost, and suitability for the water quality.
- **Determine Pipe Diameters**: Calculate the minimum pipe diameters that can deliver the required flow rates at adequate pressure, while keeping velocity within safe limits (typically 0.6–2.5 m/s for residential networks).
- **Hydraulic Gradient**: Ensure the hydraulic gradient (slope of the pressure drop along the pipe) meets regulatory requirements for maintaining pressure at each node.

IV.4.3. Pressure LossCalculations

- **Frictional Losses**: Calculate pressure losses due to pipe friction using the **Darcy-Weisbach** or **Hazen-Williams** equations, depending on flow type (laminar or turbulent) and fluid properties.
    - **Darcy-Weisbach Formula**: $\Delta P = f\frac{L}{D}\rho\frac{V^2}{2}$, where:
        - f = friction factor (dependent on flow regime),
        - L = length of the pipe,
        - D = diameter,
        - ρ = fluiddensity,
        - V = fluidvelocity.
- **Singular Losses**: Include pressure losses from bends, valves, contractions, and expansions using **Bélanger's theorem** or singular loss coefficients.

IV.4.4. Network Balancing

- **Looped Network Analysis**: For closed-loop networks, balance the flow using methods such as the Hardy Cross or Newton-Raphson methods to ensure each endpoint receives adequate pressure.
- **Adjusting for Elevation**: Account for changes in elevation by calculating gravitational pressure changes (using $\Delta P=\rho g\Delta h$, where Δh is the height difference.

IV.4.5. Pump and Pressure Requirements

- **Determine Pumping Needs**: Calculate the pump power required to overcome total pressure losses in the network, factoring in the highest demand points. Choose a pump that provides sufficient pressure to all endpoints.
- **Pressure Regulation**: Use pressure regulators or booster pumps at points where pressure must be managed due to height differences or distance from the source.

IV.4.6. Verification and Safety Margins

- **Pressure Testing**: Run simulations to verify that all nodes meet the minimum and maximum pressure requirements, especially during peak demand.
- **Safety Margins**: Apply safety factors to account for potential changes in demand, future expansions, and pipe aging.

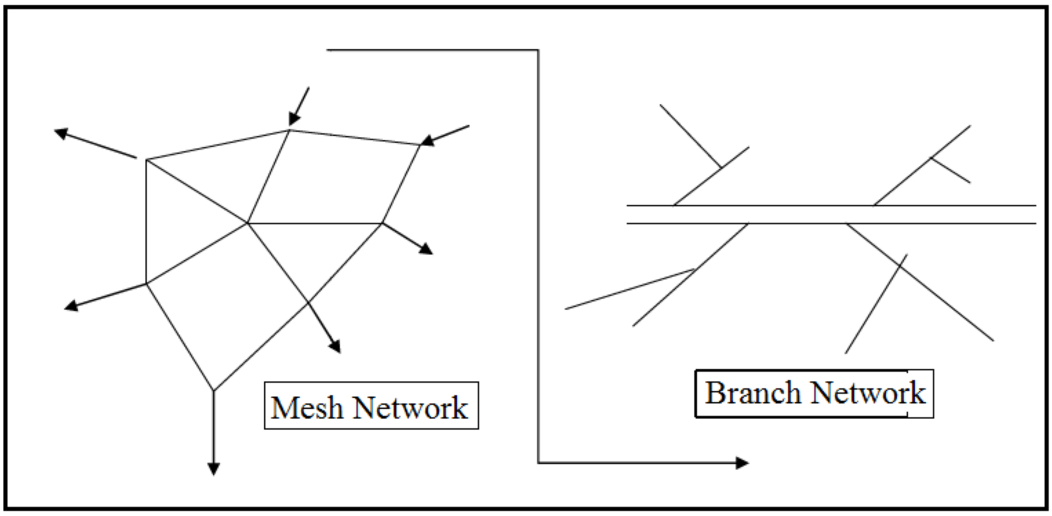
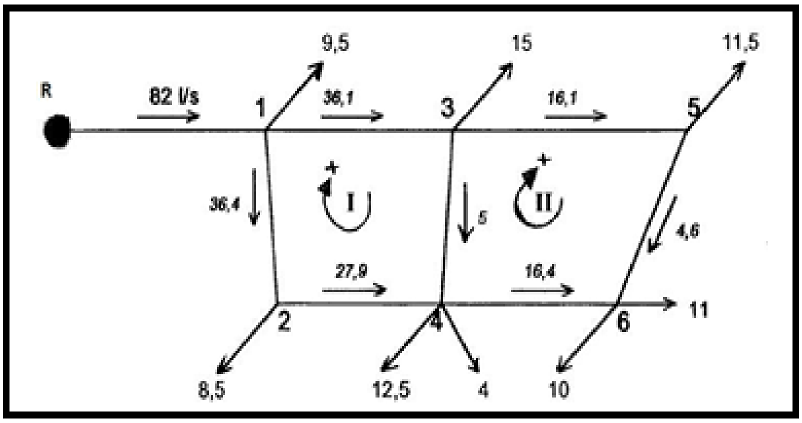


Figure IV.8. Mesh and Branch Networks.

These calculations ensure that water distribution networks are designed to deliver reliable service with minimal energy consumption and optimal pressure throughout the system. Using simulation tools (such as EPANET or WaterGEMS) can help model complex networks and optimize design decisions.

IV.5. Branched Water Distribution Networks

A branched (or dendritic) water distribution network is a layout in which the water flow branches out from a main source or trunk line and spreads through successive branches, resembling a tree structure. This type of network is commonly used in smaller or more localized systems, such as residential areas or rural water supply systems. Branched networks are typically simpler and more cost-effective to build but come with specific design and operational considerations.



IV.5.1. Key Characteristics of Branched Networks

1. **Unidirectional Flow**: Unlike looped networks, branched networks have a single path for water flow from the source to each endpoint. This simplifies flow calculations but makes it more challenging to maintain pressure if demand fluctuates or if there are leaks.
2. **Lower Redundancy**: Since there's only one path to each endpoint, a break or blockage in a branch can disrupt water supply to all endpoints downstream of the issue. This lack

of redundancy means that branched networks require careful design and maintenance to prevent service disruptions.

3. **Reduced Costs**: Branched networks usually require fewer pipes, fittings, and valves compared to looped networks, leading to lower initial construction costs.
4. **Ease of Calculation**: Hydraulic calculations in branched networks are more straightforward because there are no closed loops to balance. This allows for direct calculation of flow rates and pressures at each node.

IV.5.2. Steps in Designing a Branched Network

a. Demand Estimation

- Calculate the water demand at each endpoint based on the usage patterns of connected buildings or areas. This mayincludepeak and averagedemandcalculations.
- Sum the demands along each branch to determine the flow rate required in each section of the network, starting from the farthest endpoint and working back toward the source.

b. Pipe Sizing

- Select pipe diameters to ensure adequate flow rate and pressure at each endpoint. Pipe sizes are typically larger near the source to accommodate higher flows.
- Ensure that the velocity within each pipe section stays within acceptable limits (commonly 0.6–2.5 m/s) to prevent excessive friction losses.

c. Pressure LossCalculations

- **Friction Losses**: Calculate the pressure loss due to friction along each pipe section using the Darcy-Weisbach or Hazen-Williams formula.
- **Singular Losses**: Account for pressure losses from bends, fittings, and valves along each branch. These are particularly important in branched networks, where singular losses can impact pressure more significantly.

d. Pressure at Endpoints

- Ensure each endpoint meets the minimum required pressure, taking into account elevation differences and friction losses.
- If the network covers areas with significant elevation differences, gravitational pressure calculations (ΔP=ρgΔh\Delta P = \rho g \Delta hΔP=ρgΔh) should be included to maintain adequate pressure at higher or lower points in the network.

e. Pump and Pressure Control (If Necessary)

- If the branched network requires additional pressure to reach endpoints, consider adding a pump at the source or pressure boosters at key points along the branches.
- Check that the pump capacity is sufficient to overcome the cumulative pressure losses, especially at peak demand times.

f. Design Verification

- After initial calculations, simulate the network's performance under different demand conditions to ensure reliable operation.
- Include safety margins in pipe sizing and pressure requirements to accommodate potential changes in demand or minor pipe degradation over time.

IV.5.3. Advantages and Disadvantages of Branched Networks

***Advantages*:**

- Lower cost due to fewer pipes and fittings.
- Simple layout, which makes hydraulic calculations more straightforward.
- Suitable for small-scale, localized systems.

Disadvantages:

- Lack of redundancy; a single failure can disrupt service downstream.
- Difficult to maintain consistent pressure across the network, especially in systems with varying elevation.
- Limited adaptability to demand fluctuations without additional pumping or regulation equipment.

IV.5.4. Applications of Branched Networks

Branched networks are typically used in:

- Residential neighborhoods with limited demand variation.
- Rural water distribution systems.
- Temporary or small-scale networks where looped systems are not necessary.

In summary, branched networks are cost-effective and suitable for smaller, localized water distribution systems. However, careful planning is essential to ensure adequate pressure at all points in the network and to minimize the risk of service interruptions.

IV.5.5. Principal Types of Branched Networks

Branched networks, or dendritic networks, are used in various types of fluid distribution systems, each designed with unique characteristics suited for specific applications. The principal types of branched networks include:

1. Radial (Star) Network

- **Description**: In radial or star networks, each branch radiates outward directly from a central source, resembling a star. Each endpoint is connected individually to the central point with its own branch.
- **Advantages**: Simple to design and operate, with clearly defined paths to each endpoint. Any issues or blockages affect only the single branch without disrupting the entire network.
- **Applications**: Typically found in small communities, rural water supply systems, or distribution networks in industrial plants where demand is predictable and manageable.

2. Tree Network

- **Description**: This is the most common type of branched network, resembling a tree structure with main branches dividing into smaller branches, often further branching out to reach multiple endpoints. Eachbranch serves as a conduit for smaller branches.
- **Advantages**: Efficient for distributing fluid to multiple locations with fewer total pipes than a radial system. Suitable for systemswhereendpointsdon'trequire high redundancy.
- **Applications**: Residential neighborhoods, commercial complexes, and irrigation systems where redundancy isn't a priority, and water demand at endpoints is fairly consistent.

3. Composite Network

- **Description**: A hybrid between a branched (tree) and looped network, where the main system follows a tree structure but includes some secondary loops. These loops improve reliability by providing alternate paths to critical areas.
- **Advantages**: Offers greater reliability than a purely branched network due to secondary paths in specific areas, allowing partial redundancy in critical sections.
- **Applications**: Found in suburban areas or smaller urban districts where water reliability is essential, and demand varies more widely. Alsoused in industrial networks requiringmoderateredundancy.

4. Herringbone Network

- **Description**: This layout is characterized by a central main line with short branches extending perpendicular to it, much like a fish skeleton (herringbone pattern). It's designed for efficient parallel delivery from the main trunk.
- **Advantages**: Efficient for linear distribution, reduces material costs by limiting long, extended branches, and minimizes pressure loss by using shorter branches.
- **Applications**: Commonly used in agricultural irrigation, greenhouses, or warehouses where direction flows is straightforward and endpoints are aligned along a central route.

5. Multi-Source Branched Network

- **Description**: This design includes multiple sources, each with its own branching distribution network. These sources may be interconnected to support each other in case of increased demand or failure.
- **Advantages**: Increases network resilience, reduces the impact of source failures, and balances demand by leveraging multiple sources.
- **Applications**: Used in large facilities, such as industrial plants, or complex urban water supply systems where redundancy and load balancing are crucial.

6. Hierarchical Branched Network

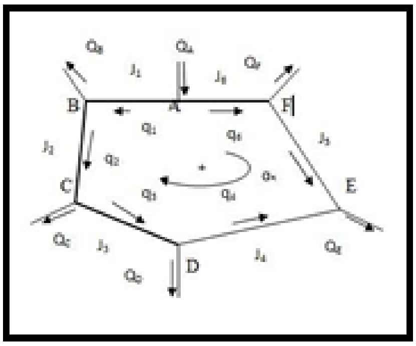
- **Description**: A hierarchical network uses levels of branches organized by demand, where larger pipes serve high-demand areas, and smaller pipes branch out to serve low-demand areas in a systematic hierarchy.

- **Advantages**: Efficiently distributes water across regions with varying demand, minimizes pressure losses in high-demand areas by providing larger conduits.
- **Applications**: Suitable for city water distribution, large facilities, or multi-building complexes where demand can vary widely across locations.

Each type of branched network has distinct benefits and is selected based on the application's size, demand requirements, redundancy needs, and cost considerations. By choosing the appropriate branched network structure, designers can optimize for efficiency, reliability, and cost-effectiveness.

$$h_i = \lambda \frac{l}{d}\frac{V^2}{2g}$$

$$h_i = 8\lambda \frac{l}{\pi^2}\frac{Q^2}{gd^5}$$



IV.7.6. Mesh network

Consisting of a series of sections arranged to form one or more closed loops following its path. Unlike branched networks, a mesh network presents an indeterminacy on the magnitudes and signs (direction) of the flow rates and pressure losses in each section.

$$\ll Q \gg \; and\; H\; where\; A; h_i; d_i; q_i; Q = Q' + \sum q_i{}'$$

Determine "qi" in each section with their direction (sign) and the head losses "hi" along each section, i.e., HiH_iHi.

IV.5.7. Calculation Principle

The method is based on two principles, which are analogous to Kirchhoff's laws in electrical circuits:

a) Flow Rate Equation Principle

At each node, the algebraic sum of the flow rates must be equal to zero.

- **(+) Positive** for incoming flow rates
- **(-) Negative** for outgoing flow rates

Where A: $Q - q_1 - q_4 = 0$

Where B: $q_{1-}q_1' - q_2 - q_6 = 0$

b) Head Loss Equation Principle

Along each loop, the algebraic sum of the head losses must be equal to zero.

- *hi* is **positive** if the flow in the section aligns with the proposed direction.
- *hi* is **negative** if the flow is in the opposite direction.

For example, for loop ABCD:

$$h_{AB} + h_{BC} - h_{CD} - h_{DA} = 0$$

c) Hardy-Cross Method

Consider a loop with multiple sections. Initial flow rates q_i' are proposed for each section, respecting the first principle $h = R.q^n$.

$$\sum h_i = \sum Rq^n = 0$$

This iterative method helps balance flow rates within each loop by adjusting flows and recalculating head losses until the principles are satisfied throughout the network.

$q =$ actual flow rate, q′ differs from the actual flow rate q by an amount Δq.

Δq: correction to be applied to q′ to satisfy the second principle.

$$\Delta q = \frac{-\sum Rq'^n}{\sum nR\, q'^{n-1}} \frac{Algebric}{arithmitic}$$

IV.6. Conclusion

Branched water distribution networks are widely used due to their simplicity, cost-effectiveness, and suitability for areas with predictable water demands. These networks typically feature a main supply line that branches out into smaller sections, effectively delivering water to multiple locations.

The design and analysis of branched networks rely on principles similar to those in electrical circuits, such as conservation of flow at each node and balanced head losses across loops (when present). Flow calculations, often done using the Hardy-Cross method, ensure that each section meets pressure and flow requirements, with corrections applied to proposed flow rates to achieve optimal performance.

While branched networks are efficient for straightforward applications, they have limitations in terms of redundancy and resilience. In the event of a section failure, alternate water paths are limited, potentially impacting service. Thus, they are ideal for systems where reliability is not as critical or where backup systems are in place.

In conclusion, branched water distribution networks offer a practical solution for efficiently meeting water demands in residential, commercial, and some industrial applications. By applying sound hydraulic principles and iterative calculation methods, engineers can design these networks to deliver water reliably and efficiently

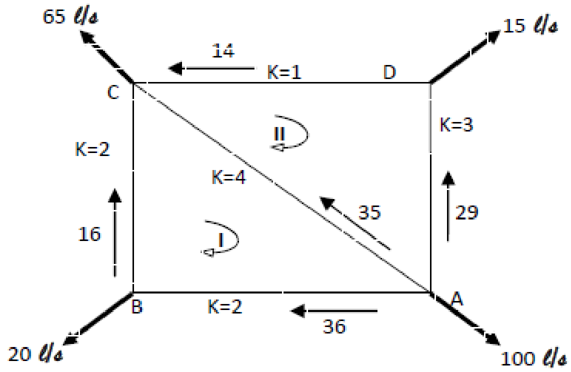
I.V.6. Application

Check the equation of the nodeswith the corrected flow rates.

Solution Correction of flow rates n=2

$$\Delta q = \frac{\sum Rq^2}{\sum 2Rq} \quad with \ \ n = 2$$

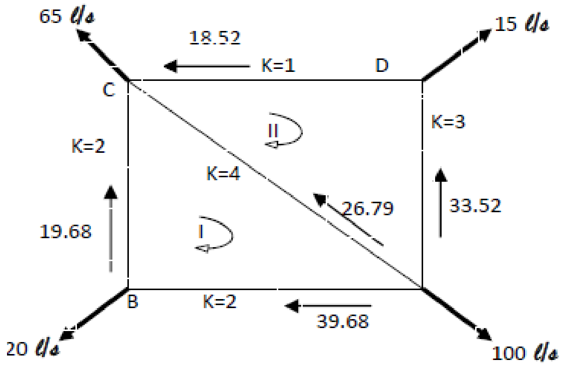
$$K = \lambda \frac{l}{12.1d^5}$$



1st correction:

Maille (I)	K q^2	2kq	Δq
AB	2592	144	3,68
BC	512	64	
AC	-4900	280	
∑.	-1796	488	

Maille (II)	K q^2	2kq	Δq
AD	-2523	174	-4,52
DC	-196	28	
CA	4900	280	
∑.	2181	482	



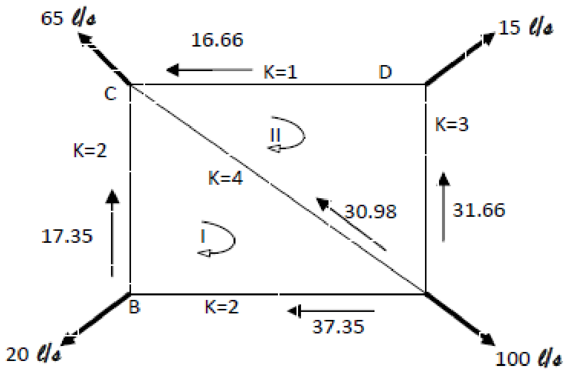
$$Where\ error\ = \frac{\Delta q}{q_{min}} = \frac{3.60}{16.5} = 20{,}16\%$$

2st correction

2^{eme} correction

Maille (I)	K q^2	2kq	Δq
AB	3149,00	158,72	-2,33
BC	774,60	78,72	
AC	-2870,82	-214,32	
∑.	1052,79	451,76	

Maille (II)	K q^2	2kq	Δq
AD	-3370,77	-201,12	1,86
DC	-342,99	-37,04	
CA	2870,82	214,32	
∑.	-842,95	452,48	



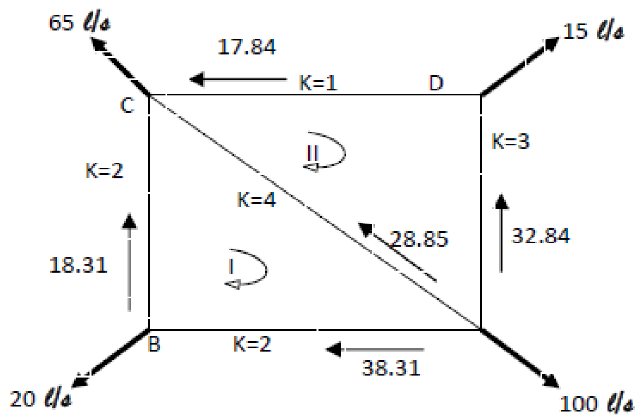
$$Where\ error\ = \frac{\Delta q}{q_{min}} = \frac{1.86}{16.66} = 11.18\%$$

3st correction

3^{eme} correction

Maille (I)	K q^2	2kq	Δq
AB	2790,05	149,4	0,96
BC	602,05	69,4	
AC	-3839,04	-247,84	
Σ.	-446,95	466,64	

Maille (II)	K q^2	2kq	Δq
AD	-3007,07	-189,96	-1,18
DC	-277,56	-33,32	
CA	3839,04	247,84	
Σ.	554,42	471,12	



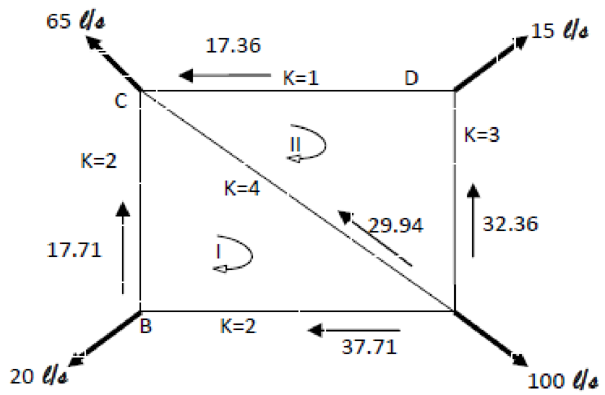
$$Where\ error\ = \frac{\Delta q}{q_{min}} = \frac{0.96}{17.84} = 6.60\%$$

4st correction

4^{eme} correction

Maille (I)	K q^2	2kq	Δq
AB	2935,31	153,24	- 0,6
BC	670,51	73,24	
AC	-3329,29	-230,8	
Σ.	276,53	457,28	

Maille (II)	K q^2	2kq	Δq
AD	-3235,40	-197,04	0,48
DC	-318,27	-35,68	
CA	3329,29	230,8	
Σ.	-224,37	463,52	



$$Where\ error\ = \frac{\Delta q}{q_{min}} = \frac{0.48}{17.36} = 2.80\%$$

According to the result of the 4th correction we note that the error is less than 5%, e <5%, we stop the corrections and we adopt the corrections of the final flow rates.