Artificial Lift

Prof.Abdus Samad

Department of Ocean Engineering

Indian Institute of Technology Madras, Chennai

Lecture-12 Flow Over A Flat Surface or Flow Through Pipe Part 1

Today's lecture is primarily focused on fluid flow through pipes. When discussing fluid flow through pipes, it's essential to understand the surface properties of the pipe material and how they affect fluid flow. You must also grasp how fluid flows from the wellbore to the surface. While some brief descriptions have already been provided, I will begin this lecture with a quick overview of how fluid flows from the reservoir to the separator.

So, let's start by assuming a reservoir below the surface and a wellbore drilled into it. You have a tubing through which you produce oil and gas. The casing is cemented in place, and when it's cemented, production doesn't occur. Therefore, you must perforate it or create a network of holes or pores in the tubing after cementing. This allows fluid to enter the wellbore through these openings, pass through the tubing, and ultimately reach the wellhead.

The fluid goes through a choke at the wellhead before reaching the separator. As the fluid flows from the wellbore to the separator, it encounters various forms of resistance. High-viscosity fluids exhibit more excellent resistance, while low-viscosity fluids, like water, have lower resistance. The opposition also increases when viscosity increases, such as to 10 cP, 20 cP, or 100 cP (centipoise). This resistance is directly related to fluid friction and surface properties.

Why is maintaining pressure necessary? Well, the separator requires a certain level of pressure. The separation efficiency will significantly decrease if you fail to maintain the appropriate pressure or velocity. Additionally, choke performance will also be affected. To sustain the required pressure, you must understand where the losses occur and how much fluid flows. It would help if you considered factors such as tubing size and the amount of gas evolving or coming out of the wellbore as pressure changes.

For instance, if there is a significant pressure drop through the piping, the initial pressure might be 1000 psi, but by the time it reaches the wellhead, it could drop to 10 psi, 20 psi, or 30 psi. Such a rapid change in pressure can lead to bubbles in the oil. As the oil produces more gas due to this pressure change, it alters the fluid properties. In such cases, the wellbore may struggle to make oil and gas. Accumulation of liquid particles at the bottom can occur, leading to a situation where the liquid column grows continuously. Eventually, the hydrostatic pressure of the liquid matches the reservoir pressure, causing no further inflow.

This condition, known as liquid hold-up, can result in the cessation of production. Therefore, it is crucial to understand the pressure drop through the piping. Let's begin by discussing pipe flow, starting with the flat plate. You may recall studying laminar flow and transient flow in pipes, and we'll delve into these concepts further.

This is the velocity 'u,' and the fluid will develop like this. The laminar zone will appear like this, and the fully developed zone will look like this. In this velocity profile, the central line velocity will be higher, while the nearby wall velocity will be lower. This is referred to as laminar or developing flow. As you move from the central line of the pipe toward the wall, the fluid velocity decreases. The lines you see here indicate the magnitude of the velocity. When you approach the wall, the velocity decreases due to the assumption that fluid particles cannot move much near the pipe wall due to friction. This friction immobilizes the particles, resulting in a velocity of 0 near the wall.

When you move away from the wall, fluid layers start sliding over each other. Different fluid layers move one by one, so you encounter higher velocities as you move away from the wall. However, when you reach the center of the pipe again, your velocity decreases once more as you move up. This pattern continues until you reach the tubing's upper wall or the opposite wall. In this case, the center velocity is very high, but the wall velocity is 0. When the wall velocity is 0, it's typically called a 'no-slip' condition. The 'no-slip' condition assumes that particles don't slip and adhere to the wall, resulting in a velocity of 0 on the wall. If particles were slipping, the velocity wouldn't be 0.

Let's talk about the Reynolds number, often denoted as 'Re.' People sometimes use other terms, but 'Re' typically represents the Reynolds number,

$$Re = \frac{\rho V d}{\mu}$$

(or)

$$Re = \frac{V d}{\upsilon}$$

So, ρ represents the density of the fluid, 'v' represents the velocity of the fluid, 'd' represents the diameter, 'ν' represents the kinematic viscosity, and 'μ' is the dynamic viscosity. This is a non-dimensional number. When you use any units, it should be non-dimensional. If it carries any dimensions, you must adjust the units of ρ, v, d, or μ to make it non-dimensional. Calculating the Reynolds number must be made non-dimensional; otherwise, errors or issues may arise, especially when working with field units. Thus, you need to be careful when using the Reynolds number.

The Reynolds number is calculated as ρvd (inertia force) divided by μ (viscous force). The top term represents inertia, and the bottom term describes viscosity. The Reynolds number measures how much fluid tries to move concerning how much it's reduced by viscosity. When the Reynolds number is less than 2000, the flow is typically considered laminar; when it's greater than 4000, it's called turbulent flow.

This applies to laminar flow. It is valid for pipe flow only. If you are dealing with open channel flow or other types of flow, such as open channel flow, the Reynolds number will be different. We are primarily focusing on pipe flow within closed conduits, specifically circular ducts. In laminar flow, the Reynolds number is less than 2000; in turbulent flow, it is more significant than 4000. We refer to it as transition flow between the range of 2000 to 4000.

It would help if you remembered these values because they are quite common. Many times, in interviews or exams, people will ask for these values approximately. Although it's typically written as exactly 2000, sometimes in certain books, you might find values like 2300 for laminar flow or 3800 or 4700 for turbulent flow, but approximately, it's 2000 for laminar and 4000 and above for turbulent flow.

Reynolds number is defined as the ratio of inertia force to viscous force. It would help if you remember this formula because formulas might not be provided during exams, so knowing this formula is essential. It's a non-dimensional number, and using different units can lead to dimensional Reynolds numbers, which would be incorrect. When you calculate it, ensure that the units are consistent, or verify that you have a non-dimensional Reynolds number after calculation.

When discussing fluid flow in pipe systems, we often talk about friction. To understand this, you must consider how fluid flows over a flat plate. Imagine a flat plate, and initially, the fluid has uniform velocity. As it encounters the flat surface, it forms a thin, uniform layer along the solid surface, the initial laminar zone. After that, there is a transition zone, followed by turbulence, where the flow becomes chaotic and irregular.

So, here I will illustrate it like this: a laminar or entry length, and then this area will be turbulent, which is the transition zone. The velocity is uniform initially in the laminar or inlet zone, but gradually, it may deviate from the smooth flow path. As you reach the turbulent area, the fluid particles move randomly, leading to turbulence. In turbulent flow, the velocity profile looks like this, and there's a very thin layer near the wall known as the viscous sub-layer. Beyond that, you have the fully turbulent layer. Essentially, friction arises primarily from the viscous sub-layer, while the turbulent layer, away from the wall, does not affect it as much. However, turbulence can impact the viscous sub-layer by increasing or decreasing its length.

Later, we will discuss how surface properties affect this. Consider any surface, like this mobile device, when discussing surface roughness. To the naked eye, it may appear smooth, but when viewed under a microscope, you'll see that the surface is not smooth. It's full of irregularities, with valleys and ridges—this is roughness. Roughness increases friction and resistance to fluid flow, resulting in higher pressure drop. How does this happen? Think of it this way: if you're riding a bike on a rough, uneven rural road, your bike will use the same amount of fuel, but your speed will be lower due to the road's unevenness. The bike will move up and down because of the road's irregularities, causing fluctuations in speed. This up-and-down motion is akin to the randomness in fluid flow due to surface roughness, resulting in decreased velocity. However, the energy consumption remains the same, which is why we refer to it as pressure drop in fluid flow terms.

So, when fluid flows over an uneven surface, it encounters a lot of resistance. The reason for this resistance can be explained as follows: When fluid flows over a smooth surface, which appears smooth to the naked eye (although under a microscope, it reveals a rough terrain), the fluid flows smoothly. However, if we consider an inherently rough surface, like cast iron, the fluid encounters difficulties in flowing smoothly.

For instance, a viscous sub-layer is formed when fluid flows over a smooth surface, as shown here. In this case, there is a gap between the surface roughness and the viscous sub-layer. Due to this gap, the surface roughness does not significantly affect the fluid particles.

In the second scenario, where we have a viscous sub-layer and a rough surface, something different occurs. When the surface is rough and the sub-layer is present, the fluid particles interact with the numerous ridges and valleys. When fluid particles attempt to move along a smooth and straight path but encounter obstacles like ridges or valleys, they move up and down, losing energy and velocity. This results in increased fluid resistance and higher pressure drop.

Rough surfaces, therefore, offer more resistance, causing the fluid to flow less smoothly. Conversely, a smooth laminar layer is created on smooth surfaces, allowing the fluid to flow more smoothly. To determine whether a surface is smooth or rough in fluid flow, we compare the roughness height with that of the viscous sub-layer. If the viscous sub-layer is thicker than the roughness, we can assume it's a smooth surface. However, the surface is considered rough when the roughness height exceeds the thickness of the viscous sub-layer by a significant margin. In such cases, the roughness will notably impact fluid pressure drop.

One term we use is 'hydrodynamically smooth surface.' A hydrodynamically smooth surface is one where there is only a small distance between the viscous sub-layer and the surface roughness. If your surface roughness changes within certain limits and doesn't reach the sub-layer, it's considered hydrodynamically smooth. Changing the roughness within this range won't significantly affect fluid pressure drop or friction. However, when you exceed this hydrodynamically smooth range, surface roughness impacts fluid flow and pressure drop.

Another term used is 'sand roughness number.' To estimate roughness, people assume the presence of sand particles on the surface. They use this term to determine if the viscous sub-layer crosses the roughness or remains separate. Sand roughness helps approximate the impact of roughness on fluid flow and pressure drop. Calculating the roughness, including numerous peaks and valleys, can be complex. Therefore, using the sand roughness number simplifies the process.

When discussing pressure drop, you need to consider shear stress. Shear stress (τ) is given by the formula,

$$\tau = \mu \left(\frac{du}{dy}\right)$$

where μ is viscosity. For Newtonian flow, viscosity (μ) is assumed constant. However, in non-Newtonian flow, μ may not be constant. In such cases, shear rate (du/dy) and shear stress vary. There are different flow behaviors, such as Newtonian, dilatant, and pseudoplastic. The fluid's rheology, whether it behaves as a Newtonian or non-Newtonian fluid, can depend on various factors.

Wellbore fluid is often a mixture of many components and fluids, including hydrocarbons of various chain lengths, both shorter and longer. Due to this mixture, it may not strictly exhibit Newtonian, dilatant, or pseudoplastic flow behavior. Its behavior can vary based on conditions. For example, crude oil is sometimes considered non-Newtonian but can exhibit Newtonian behavior under different conditions. It tends to behave as a non-Newtonian fluid when it contains more wax.