

Basics of Mechanical Engineering-3

Prof. J. Ramkumar

Prof. Amandeep Singh Oberoi

Department of Mechanical Engineering

Indian Institute of Technology, Kanpur

Week 11

Lecture 47: Virtual Lab. Demonstration (Fluid Mechanics, Part 1 of 3)

Welcome to the Laboratory Demonstration of Fluid Mechanics. We are in the course Basics of Mechanical Engineering 3. We discussed the classification of flow in week 8. We discussed Basics 3 of Fluid Mechanics in week 9. Like in the classification of flow, we discussed the Reynolds number, how we determine whether the flow is laminar, turbulent, or in transition between them.

For that, there is a virtual laboratory demonstration designed through the virtual laboratory by Professor Ramkumar and me, which we will try to demonstrate here. I will start with the classification of flow experiment, which is the Reynolds number determination and finding the flow conditions. From that, I will take you to certain theories of fluid mechanics, like Bernoulli's theorem setup, how we develop it, and how we conduct an experiment based on that. After that, what we discussed this week itself, we covered the applications of fluid mechanics, including turbines, pumps, compressors, etc. Some experiments based on those will also be covered in this and the next lecture, where we will have a virtual demonstration of fluid mechanics.

Virtual Labs

fm-ittk.vlabs.ac.in/exp/open-channels/theory.html

Verify it's you

Determination of Reynolds Number at various Flow Conditions

Aim

Theory

Pretest

Procedure

Simulation

Posttest

References

Feedback

Introduction

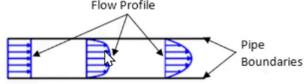


Fig. 1: Fully developed flow in a pipe

Fluid flow in circular and noncircular pipes is commonly encountered in practice. The hot and cold water that we use in our homes is pumped through pipes. Water in a city is distributed by extensive piping networks. Oil and natural gas are transported hundreds of miles by large pipelines. Fig. 1, the inflow, the velocity profile is often uniform but a thin boundary layer develops on the pipe wall because of friction as the fluid moves along the length of the pipe. Fully developed flow occurs when the viscous effects due to the shear stress between the fluid particles and pipe wall create a fully developed velocity profile. In order for this to occur the fluid must travel through a length of a straight pipe. In addition, the velocity of the fluid for a fully developed flow will be at its fastest at the center line of the pipe. On the other hand, the velocity of the fluid at the walls of the pipe will theoretically be zero. Beyond this distance the velocity profile becomes fully-developed (i.e., doesn't change any further with downstream distance).

Components of Experimental setup

28°C Partly sunny

Search

8:44 AM 25-Sep-25

Lecture 47: Virtual Lab. Demonstration (Fluid Mechanics, Part 1 of 3)

Welcome to Virtual Labs

fm-ittk.vlabs.ac.in

Verify it's you

Virtual Labs
An MoE Govt of India Initiative

★★★★★

HOME PARTNERS CONTACT

Mechanical Engineering

Introduction

Objective

List of experiments

Target Audience

Course Alignment

Feedback

Fluid Mechanics Lab

The virtual fluid mechanics lab provides an interesting and interactive way to learn about fluid mechanics and machinery exercises. It uses a practical approach to study how fluids act when they are still and when they are moving. This V-lab shows the steps needed to do experiments, and then includes a simulation to improve learning.

Students can access these studies anytime, which lets them examine results without needing physical equipment. By taking part in these online activities, learners will gain hands-on experience, improve their critical thinking and problem-solving skills, and deepen their understanding of fluid dynamics and how it applies in different areas

Community Links

Contact Us

Follow Us

Sakshat Portal

Phone: General Information: 011-26582050

28°C Partly sunny

Search

8:43 AM 25-Sep-25

2:09 / 41:29

YouTube

Vlab demonstration (Fluid Mechanics)



Volume
Position

$$Q = \frac{V_0}{t} = \frac{0.0003}{17.73} = 0.000017 \text{ m}^3/\text{s}$$
$$V = \frac{Q}{\text{Area}} = \frac{0.000017}{0.000201} = 0.84192 \text{ m/s}$$
$$Re = \frac{\rho V D}{\mu} = \boxed{1340}$$

Type of Flow: Based upon Re The flow is LAMINAR

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



So, how do we start that? I have just opened the Google page here. So, how do we get in there? So, this is the Google page itself. I will write here VLAB mechanical engineering, or you can even write V lab fluid mechanics itself.

In the lab of mechanical engineering, when I open it, we have this page of V lab for mechanical engineering. There is a list of laboratories here. These are virtual laboratories that have been developed by different institutions. So, what has been developed by IIT Kanpur? It is the fluid mechanics lab.

So, that I will click here and I will reach on this page. This is fluid mechanics lab developed by IIT Kanpur on virtual environment. That is virtual labs setup which is Ministry of Education initiative. Here, I will click on the list of experiments. In the list of experiments, you can see there are 10 experiments listed, such as determining the loss variables for different types of flow in pipes and open channels. Loss variables include the Reynolds number. Then, there are the characteristic curves of the Pelton wheel, the coefficient of discharge for a given Venturi meter, variable area duct, and verifying Bernoulli's energy equation.

Experiment 3 and 4 are about majorly the basic theory of fluid mechanics what we have discussed here. Experiment 2 is the Pelton wheel, where we discussed pumps, turbines, etc., and the Pelton wheel was discussed, which will be covered later. First, I will start with the very basic experiment, which is the flow in pipes and open channels. You see, when you open this, it shows the determination of the Reynolds number at various flow

conditions. The aim is to measure the friction factor for flow through a pipe over a wide range of Reynolds numbers and compare it with the corresponding theoretical value.

In the theory, the generic theory given here, we have a pipe. The flow comes and changes its behavior when it starts flowing through the pipe. Depending upon the velocity and the pressure, the flow can be laminar or turbulent. What laminar or turbulent means will also be covered. So, what is the test bench here? The test bench consists of a big reservoir and a flow meter so that we can control the flow. There is a control valve, and we have a differential pressure gauge between points A and B. The length between A and B is 0. So, there is a pipe of diameter D. So, there is a flow meter they say test bench consists of flow meter for experiments control valve to control pipe system. The pipe diameter is given, length is given. There is a measuring tank, that is equipped with gauge glass and scale arrangement, differential manometer, reservoir, some theory that flow is laminar like this, flow is turbulent like this. When there are non-number is in a specific range or it is specific beyond range or below the range.

So, Re is $\rho V D / \mu$, which we have already discussed in that unit part. Here one term that is there is Re_{cr} , that is Re critical. The critical Reynolds number is specifically 2300. It is given here for internal flow in a circular pipe. A value of approximately 2300 is often used as the approximate critical Reynolds number for the start of the transition to turbulence. You know this range already.

Re less than equal to 2000 is laminar, between 2000 to 4000 is transitional, more than 4000 is turbulent. Then what is this 2300? So, what is Re critical? Re critical is 2300. It is that when the launch number is 2000, less than 2000, it is laminar.

When it goes more than 2000, the transition does not start immediately. It still stays laminar only after 2300 and on number the transition starts to the turbulence and from 2300 to 4000 it stays transition beyond 4000 it becomes a turbulent flow. Now, when it comes below 23000 it stays the between the transition of the turbulence and laminar unless it reaches 2000. Range always is between 2000 and 4000, it is what transition, below 2000 it is laminar, above 4000 it is turbulence, but there is a critical point 2300 loss number that is for the start of the transition to turbulence. So, this is generic theory.

We have HF here that is Darcy-Weisbach pipe friction factor = $f \times (L/D) \times (V^2/2g)$. There are certain variables given here. All these variables and Moody diagram is there. Let us take a pre-test to understand what these variables are and how we go about it. The first question is: the friction factor is dash when the flow becomes fully turbulent.

When the flow is fully turbulent, that means the friction factor is maximum because the friction causes the flow to become turbulent. If the flow velocity is doubled, then the frictional head losses become. We know that the head losses are proportional to the square of the velocity. So when the velocity is doubled, squaring it becomes 4 times. For laminar flow in a pipe of uniform diameter, relation between Darcy's friction factor and Reynolds number is $f = 64/Re$.

Darcy's friction factor for laminar flow is based upon Poiseuille's law, which is determined by this. Moody's diagram is used to calculate the velocity, pressure, friction factor, and temperature in pipes. It is used to calculate the friction factor. The assumptions of Poiseuille's law are: incompressible fluid?

Yes. Newtonian fluid? Yes. Laminar flow? Yes. All of these are assumptions when we try to apply Poiseuille's law. Submit. We got everything right. 6 out of 6. Now let me come to the procedure. In the procedure, they say to check whether all the joints and sump are not leaking.

Close all the cocks, pressure feed pipes, and manometer to prevent damage and overloading of the manometer. Check the gauge class, meter scale assembly of the measuring tank, and see if it is fixed at the water height and vertically. Check for proper electrical connections to the switch. The actual discharge is measured with the help of the measuring tank. The differential head produced by the flow meter can be found from the manometer.

Start the motor. These steps we will see when we conduct the simulation. Then, we take the next step by regulating the control valve and selecting the corresponding pressure tapings. We try to select the pressure and note the reading. We note down the difference of head from the manometer scale and time required to rise to 10 centimeter.

The actual loss that is determined by manometer readings, the coefficient of friction f is indicated there. Then, we repeat steps 4 to 8. That is, we keep on selecting the height. We keep on closing the valve from 4 to 8. We keep on repeating so that we get multiple readings of discharge valve.

There are certain relationships that are used. For example, $Q_{\text{actual}} = A \times R/t$ where A is area of collecting tank, R is rise of water level taken in meters, t is time taken for the rise of water. V is the velocity, that is the discharge per unit area. That is Q_{actual} per unit area. Then we have friction coefficient $hf = f \times (L/D) \times (V^2/2g)$.

These are all relationships given here, and this is the kind of a table that we will have to develop to find the final theoretical f value. Let me come to the simulation. This is a simulation interface of the Reynolds experiment. You see there is an arrow showing here on power on. This is a setup where you see there is a tank, and this is a sump, that is a reservoir, and from here there is a motor.

When we switch on the power of the motor, the water will start filling and this is the valve. The valve position is also there 0, 1, 2. When we open the valve only, then only water will flow through this pipe. It will start filling this tank. It will fill the tank only when this gate valve is closed.

If the gate valve is open, the water will come here and flow through the gate valve and go to the sump. Let me start the experiment. I will switch on the power. Instructions are here. It says, please wait until the water reaches.

The water is being filled here. You see, water reached here. Then it says, it shows a valve positioning slider to regulate the water flow the valve positioning here is at zero. I will come at position one you see the water is now going through the pipe to the tank, but it is exiting through the gate valve and going to the sump. Now, next instruction is close the gate valve. Also, there is an arrow that shows close the gate valve. When I close this gate, all the water starts filling and to fill the water, the time is being noticed.

The time is 17.72 seconds to fill the water up to 10 centimeters of the tank. Now, the volume of the water filled and the time are the parameters that will determine. These parameters could be put down in the observation table. So, before that, let me see the formulas. We could calculate Re . We can calculate $\nu = \mu/\rho$, we can calculate again Reynolds number by dV/V , where V is velocity, that is Q actual by area of the pipe and Q actual is V_0/t here. Area of the tank, area of the pipe, diameter of the pipe, inside diameter, length of the pipe, kinematic viscosity.

Acceleration due to gravity and the density of water are all given parameters because these will be used in these formulas. So, what is observed here is the time to fill this gate valve at position 1. I will put the time here; it is 17.73. I have entered this value here. Now, V_0 is given, which is the volume of water collected, based on the tank area multiplied by the height, which is up to the 10-centimeter height, and the area is also given here. Now, they say to change the position. I will change the position.

Now, water is again flowing through the pipe toward the tank. Now, I will close the gate valve. The water will fill again. Because the valve is more open here, it is in the almost fully open position. It took very little time to fill.

The time taken is only 3.54 seconds. So, I will record the reading here as 3.54 seconds. Now, this reading is 3.54. There are certain calculations that need to be done here. Discharge is in cubic meters per second.

Velocity is in meters per second. Knots number, flow type. Using these formulas, we can perform the calculations. These calculations could be done in a slide itself or using an Excel sheet. They say export to Excel.

There is a table here. So, it is downloaded here. If I open this using Excel. I am going to put a formula into calculations here. What are the calculations I am going to do?

Let me put it on a slide. First, I will calculate Q . $Q = V_0/t$. Then the observation table mentioned calculating velocity V , then we talked about Reynolds number, then we talked about the type of flow, right? It is Q , discharge velocity V in meters per second, Reynolds number, and flow type. Let me show you once again: it is Q , discharge velocity V in meters per second, Reynolds number, and flow type.

So, to put all these formulas here and calculate, we can see velocity $V = Q/A$, and $Re = \rho V D/\mu$. The type of flow depends upon Re , based upon the Reynolds number. Now, what is V_0 here? You could see here the V_0 is the volume. That is 0.00300, and the time taken is 17.73. That is 0.0003 divided by 17 seconds.

It was exactly 17.73 seconds. This turns out to be 0.000017 cubic meters per second. This is for valve position one. Everything that I'm doing is for valve position one only. And Q per unit area. The area is given here—you can see the area is given here, which is 0.000201 square meters. I'll put this area here. It is 0.000017/0.000201. This turns out to be 0.84192. It is in meters per second.

Then, I could be calculated. We have given ρ , we have v , and we have D/μ . So, all of this could be calculated. I am not taking much time on this because only this understanding has to be there. When I calculate this, the overall Re value turns out to be around 1340, which means the flow is laminar.

Because Re is less than 2000. This could also be done using an Excel sheet. So, this is an Excel sheet that shows the volume and the time. I can put a formula. So, I can now calculate Q here.

I will put Q , which is volume per unit time. That is, this is equal to cell B3 divided by cell C3. Enter. It shows 1.69×10^{-5} , which is equivalent to what we calculated: 0.00017. A similar formula, I will just copy it here for the full valve open. This is a very high volume.

But this charge is very large because the valve was fully open. It took very little time. So, per unit time, it is around 1/4th of this. So, this is very high. So, this will be 0.0000847.

That means it is 8.47×10^{-5} . This is Q . I will calculate the next thing that was asked, which is velocity V . Velocity is this value. This is equal to Cell D3 divided by the area. The area is 0.000201. This value is 0.084.

This could also be copied here. I can also put the Reynolds number. Okay, to calculate Re , I have to first calculate μ . μ is what? μ is dynamic viscosity.

This is dynamic viscosity, and we have ν as well, which is kinematic viscosity. You can see here that μ is dynamic viscosity, and kinematic viscosity ν is also given. So, using ν and ρ , we have to calculate μ . So, $\mu = \rho \times \nu$, which I can also put here. To calculate Re , we have to calculate or find μ .

$\mu/\rho = \nu$, we know. That means $\mu = \rho \times \nu$, which means dynamic viscosity = density of water multiplied by the kinematic viscosity. This could also be calculated. These values are given there. You see, kinematic viscosity is 1.0060 into 10 to the power minus 6, and ρ is 1000.

So, I can put it here. 1.00 means 5 significant zeros after the decimal place before 1. This formula could be put in Excel. So, this $\mu = \nu \times \rho = 0.000001 \times \rho$.

ρ is 1000. So, this is the value of μ . $Re = \rho Vd/\mu$, or $Re = VD/\nu$. I will use this formula. $Re = \rho Vd/\mu$.

So, putting this formula here, $Re = \rho \times V \times d$, that is velocity, which is calculated in cell E3. This times the diameter. The diameter given here is 0.016 meters. This divided by the calculated value of dynamic viscosity μ , which is in cell F3.

I have got this Reynolds number value as 1346. I will also put here flow type, and this flow type is laminar, right. So, this observation table is taken here, and I can also copy the values for the other time that is taken, that is, 3.54 seconds. So, this Re value is 6745. That is very high.

It is more than 4000. That means this flow is turbulent. This was done using an Excel sheet, or we can do manual calculation. Anyway, I've just explained this for the first experiment. I can enter all the values here, and also the results could be checked.

To check the results, we have to enter the values. Then the results could be checked against what we have calculated. So, these values I can enter here in this sheet. The starch value. Q is calculated as 0.0000169.

Velocity is 0.084181. I am taking these values and putting them here. Then it asks about the Reynolds number. The Reynolds number is calculated as 1346.9, which is 1347. The flow type is laminar. For the second observation, where the time was 3.54 seconds, the discharge value turns out to be Q, which is 8.474×10^{-5} , or 0.0000847. The velocity here is 0.42162.

I am putting this value here. The Re value here is 6745.9. I will put it as 6746. This flow is turbulent. Right. So, I have entered everything. Now, let me check the results. This is the result.

It is the observation table at the backend of the program that is in this virtual lab. It shows a discharge of 0.00017. So, all my values are very close, and the flow is laminar for the first case. That is the first position of the valve. For the second position of the valve, the flow is turbulent.

There is some change in the value of the Reynolds number. It is okay because the use of the calculator we are using or the decimal places we are using can affect it. The values could change. This is the Reynolds number or the flow condition determination. The next point is post-test questions.

This is a post-test question. A set of five questions is there. The first question is about the relationship between the Reynolds number and flow velocity. We know the Reynolds number is directly proportional to velocity. The velocity equation includes the relations of the velocity equation here.

So, the velocity equation, the first relation is $f = 0.316 \times Re^{-0.25}$. That is $Re^{1/4}$. The velocity equation is valid only for turbulent flow of pressure head, velocity head, and datum head. The flow is laminar when Re is less than 2,000. The flow is turbulent when Re is more than 4,000.

Let me submit. I have gotten 5 out of 5. And for this question, there are explanations also given that this velocity equation is an empirical formula used to estimate the Darcy friction factor for turbulent flow in smooth pipes where a Reynolds number ranges from 4,000 and above. That is 4,000 to 10,000. It shows that the friction factor decreases with an increase in the number.

These you can also calculate if you like. So, there are references through which this was developed. This was the first experiment to determine the flow conditions and to find the Reynolds number. I will take further experiments on certain theories which are discussed in week 9. Now, I will take another experiment on the determination of Bernoulli's equation validation.

I'll come to the fluid mechanics main page. A list of experiments is there. So, I will open experiment number four, which is a variable area duct and verifying Bernoulli's energy equation. You can see here the aim is given as to determine the pressure variation along a converging-diverging pipe section and validate Bernoulli's theorem and its assumptions that the summation of the terms in Bernoulli's equation along the streamline will always be constant. So, to recall Bernoulli's equation, you remember there were the terms given here, that is the pressure head, velocity head, and the potential head.

This summation would remain constant. The setup that is given is something like this. We have a variable diameter pipe. The diameter varies from the outwards towards inwards from both sides; there is a higher diameter here, a lower diameter in the center, and a higher diameter on the right side exit as well. So, there is a sump tank to which the water is taken, and the water always drains towards this as well. There is a measuring tank sometimes that is required to measure the amount of water that is passed through.

There is a control valve to control the speed of the flow, and there are manometers here. So, this is a specific pipe flow, you could see. When the pipe has a higher diameter, the velocity would be something, and there is some pressure. This is a pressure gauge, and when the pipe diameter is reduced, you could see the pressure is also reduced here, meaning the velocity is higher. That is the overall sum of all these three heads: velocity,

pressure, and the potential energy. That stays constant. This is to be determined using the setup that is there.

So, though we have studied this in detail in theory, I am coming to the pre-test questions directly. The first question is regarding Bernoulli's equation: is it derived from Poisson's equations, Euler's equations, Runge-Kutta equation, or Laplace equation? Runge-Kutta and Laplace are only mathematical equations.

They don't have anything to do with the flow as of now. Now, Euler's and Poisson's—it is derived from Euler's equation. For an incompressible inviscid flow, there is Bernoulli's equation, which is derived from Euler's equation. The lift generated in an airplane is based upon this lift in the airplane.

This has also been given as an example. The lift is generated because of Bernoulli's principle. In a horizontal pipe, water flowing through the narrowest part of the pipe will have, since the part we are talking about is the narrowest, the narrowest path means the minimum pressure is there and the maximum speed should be there. So, we have minimum pressure and maximum speed. That is option C. Bernoulli's equation is not applicable here. It is applicable to the point where the flow is present.

In a thermometer, the flow is based on temperature. It is not applicable to a thermometer. Bernoulli's equation is applicable to: it is applicable to an airfoil, it is applicable to a venturi, it is applicable to an orifice. All of these are correct answers. And Bernoulli's equation is based on which law?

Energy conservation. Momentum conservation, mass conservation, work conservation. The best answer I could find here is energy conservation. That is, the total energy stays constant. And question 7 says Bernoulli's equation is applicable only to, as I have mentioned in Euler's formula, in Euler's law, it is assumed that the flow is incompressible.

That relates the change in pressure between two points in a pipe to changes in kinetic and potential energy. It is Bernoulli's equation only. Bernoulli's equation is applicable to temperature gradient, electronic signals, fluid flow, and mass flow.

Fluid flow is the correct answer. Streamlines over an airfoil follow Bernoulli's law only. Let me submit. I have gotten 10 out of 10. And we have the explanations here as well. Let me see. For example, question 1.

Euler's equation, they say the correct answer is this. Bernoulli's equation is derived from Euler's equation of motion for an inviscid incompressible fluid. Euler's equation describes the balance of forces in a fluid flow and is the starting point for deriving Bernoulli's principle, which relates pressure, velocity, and elevation in a moving fluid.

Let me come to the procedure. It says: check all the joints, and none should be leaking. Fill the tank with clean water. Keep the delivery valve closed. This is a generic procedure. The procedure we will follow in the virtual laboratory demonstration is: switch on the pump and slowly open the delivery valve. Adjust the flow using the control valve.

Allow the system to attain steady state so that it lets water pass through the second overhead tank to the collecting tank. Note the pressure head at different points of the venturimeter and multi-tube piezometer. Close the ball valve of the collection tank and measure the time for the known rise of water.

Change the flow rate and repeat the experiment. Note the pressure head at different points. Certain relations are used for actual flow. That is, actual discharge is $A \times R/t$, where A is the area of the collecting tank. R is the rise of water. t is the time taken.

Pressure head is $h = P/\rho g$. P is pressure. ρ and g are the density of the water and acceleration due to gravity respectively. And we have velocity head = $v^2/2g$. Validation of Bernoulli's equation. So, estimating pressure head and velocity head at cross-sections of variable diameter pipe, the summation of both should be constant. So, this is to be determined here. Let me come to the simulation. This is a simulation setup here. You could see just like we saw in the Reynolds number determination flow condition determination experiment.

This is a similar setup. We have a power on setup here. This is switch on the power. When I switch on the power, the water starts coming. Through this pump and it stops till the valve because the wall is at zero position.

It is not open yet. Then they say, 'Open the valve.' Instructions are also given. Choose a valve position to regulate the water flow. I'll choose position one.

Now the water starts flowing through the pipe. It has started filling here as well, and it goes to the tank. Now the gate valve is closed. Let us fill the tank to the height of some specific centimeter, whatever is given here. So, let me close the gate valve.

So, water is being filled to a height of 14.5 centimeters. Let us note the amount of water being filled here. Based on the positioning of the valve here. So, there are certain readings here, certain observations taken: height is 14.5, time taken is 53.6 seconds, and there are positions of the manometer. For example, h1 is 21.5, h2 is 21.0, h3 is 18.0, h4 is 10.0, h5 is 13.0, h6 is 17.0, and h7 is 18.0.

These are all the values taken. Now, we know the total flow that is energy conservation, total energy is sum of the three heads and $V = Q$ discharge per unit area and $Q = A \times h/t$. These are formulas. The given values are the area of the measuring tank, acceleration due to gravity, and the different pipe diameters here. For the different manometer positions. It is a given table.

For the section of the test point. For where the diameter is 28. The cross section area is this, depending upon the diameter. And distance from the reference point is this. So from the reference point. When we are moving down there, the diameter is variable. It is 28 at the inlet. 28 at the outlet.

In between, positions are there. That is 22.5, 16.5, 14, then again 16.5 and 22.5 of the diameters. Accordingly, the area also varies because this is the area that is going to be used here in this formula. That is why the different area points are given.

Now, they say what is h and t and observation table the manometer readings. And these values h1 is 21.5, h2 is 21, h3 is 18, 10, 21.5, 21, 18, 10. Then we have H5, H6, and H7, as 13.0, 17.0, and 18.0, respectively. That is 13, 17, and 18, for these observations that I did here or entered here.

Now, they say, change the valve positioning. I will change the valve position. Again, it has come here. The water is being filled here. So, depending upon the diameter of the pipe, the velocity changes. That is why the pressure head also changes. Now, I will close the gate valve and see to what level the water fills.

So water is filling up to 14 cm. Time taken is 52.5 seconds. It is filled up to 14 centimeters and these values would again be entered here 21.6, 21.0, 18.2. For h4, it's 10.1, then 12.8, 17.2, and 17.9. 10.1, 12.8, 17.2, and 17.9. 17.2, 17.9. Now, we can calculate here the pressure head, that is p by ρg , that is small h by 100. The h values are there: 21.5. This by 100 would be 0.215. The second manometer would be 0.21. This could be calculated using the Excel sheet as well.

But this calculation is so easy. It is only divided by 100. I am just putting these values here for the first observation only for now. So, this is 21 by 100, which is 0.21. 18 by 100 is 0.18. I am putting these values. So, this is 0.18.

This is 0.10. For 5, it is 13, which means it is 0.13. For 6, it is 17, which is 0.17. And for 7, it is 18, which is 0.18. $v^2/2g$ could also be calculated. g is 9.81 m/s^2 .

This you can calculate using Excel or by using simple calculations in your calculator. I have just calculated it because I will not keep using Excel in each of the experiments. I hope you understand the overall concept. You can use the Excel sheet and put the formulas there. Even more calculations could come in the forthcoming experiments.

You can use an Excel sheet to do that. So, I will just put the values here which I have already calculated. The area is varying here. You could see this area is being varied here. So, I will put the values that I have calculated here. This is 0.005. The next value is 0.013. The third manometer is 0.045. For the fourth one, it is 0.086. The fifth one is again 0.045.

The sixth one is proportional to the area, and the area is the same for the sixth and the second manometer, so it is 0.013. The last would be again 0.005, and the value of E is also calculated. The sum of these two, 0.215 plus these values, is also calculated. I will say calculate. It will give me the sum of 0. This is the area that is calculated. You can see it is almost constant. Let me also check the result. So, when I say check the results, they say we have completed the experiment. It says the sum of the pressure head and velocity head stays constant.

You can see it is almost a constant value. Now, let me come to the post-test questions. Dynamic pressure is $\rho v^2/2$. The pressure term in Bernoulli's equation is static pressure, stagnation pressure, dynamic pressure, or static and dynamic pressure. Here, the pressure term is static pressure only.

Stagnation pressure is defined as $P_{\text{static}} + \rho v^2 / 2$. That is static pressure + dynamic pressure, which is stagnation pressure. The assumptions of flow in Bernoulli's theorem are that the flow is steady, irrotational, and frictionless. All of these are the assumptions that are taken. Bernoulli's equation holds true only for low viscosity and incompressible flow.

Low viscosity with compressibility is applicable for incompressible fluids, and it should be low viscosity in streamlined flow. So, C is the correct option. In a horizontal pipe,

when water is not flowing, the pressure is 3×10^5 pascals. When water starts flowing, it falls to 2.5×10^5 pascals. What is the flow velocity of water?

You can calculate the flow velocity of water using the formula $\rho v^2/2$. This formula, which is given, is $\rho v^2/2$. You can calculate the flow velocity here. So, this flow velocity turns out to be 10 m/s. The Magnus effect is based upon Bernoulli's law. The Magnus effect is related to the spinning of an object that generates a lifting force.

That is based upon Bernoulli's law itself. In which of the following conditions can Bernoulli's equation not be used? A viscous fluid does not allow Bernoulli's equation to be applied. Incompressible flow is when the flow is incompressible, meaning there is no change in density. That is option C, I would say, when the change in density of the fluid is negligible.

Inviscid flow assumes that the viscous effects are negligible. Let me submit 10 out of 10; all of the answers are correct. There are references from which the theory is taken. You can also provide your feedback for any of the experiments here. Share your experience here when you visit this website.

This was Bernoulli's energy equation. I have explained two experiments: the Reynolds number experiment, which is due to the flow condition, explained in detail. When we do the calculations using calculators, pen and paper, or Excel sheets, those Excel sheets or the pen-and-paper method can be used for any observations you take and try to calculate.

I have taken only two experiments in this lecture. Further experiments, which talk about pumps, applications, or turbines, I will cover in the next part of the virtual laboratory demonstration on fluid mechanics.

Thank you.