Tutorial: To simulate Hagen-Poiseuille flow in OpenFOAM.

Script and Narration : Saurabh S. Sawant

Keywords: Video tutorial,CFD.

Visual Cue	Narration
Slide 1:	Hello and welcome to the spoken tutorial on simulating **Hagen-Poiseuille flow** in **OpenFOAM**.
Slide 2 : Learning Objectives	In this tutorial we will see: • To create and **mesh 3D cylindrical pipe.** • To **simulate** the **Hagen-Poiseuille flow** having **fixed pressure ratio** across **boundaries.** • To visualize the **velocity contour** in **ParaView**.
Slide 3: System Requirement	To record this tutorial I am using **Linux Operating system Ubuntu** 12.04 **OpenFOAM version** 2.1.1 **ParaView version** 3.12.0
Slide 4: Prerequisites	To practice this tutorial learner should have the knowledge of Basic **Fluid Dynamics** and **Hagen-Poiseuille flow**
Slide 5: Hagen-Poiseuille Flow Diagram Read aloud the given points and show the contents in the diagram with the mouse pointer.	Here is, **Hagen-Poiseuille Flow** Diagram. We can see the **dimensions and boundaries** of the pipe. **Viscosity** of fluid used, that is, water is given. Pressure at the **inlet** is 20 **Pascals** and **outlet** is 0 **Pascals.** As it is an **incompressible flow**, only the pressure difference is of importance.
Slide 6: Formulas and Analytical Solution Read aloud the given points	**Formulas and Analytical Solution:** For **Hagen-Poiseuille flow,** **Pressure drop** along the pipe is: (**P1 minus P2**) equals (**32 mew Uaverage L**) upon (**D square**)

	By substituting the values from the previous diagram, we get, **Uaverage** equals to 0.208 **m/s** **Maximum Velocity** is given as, Two times the **average velocity**, which would be, 0.416 **m/s** **Reynolds Number** for the flow is, **Uaverage** into **D** upon **nu**, that comes out to be, **2080** Hence, the flow is **transient**.
Slide 7: Transient Solver	Type of solver used here is, **IcoFOAM** It is a **Transient Solver** It is used for **incompressible, laminar flow of Newtonian fluids.**
Slide 8: Pressure Boundary Conditions	**Pressure Boundary Conditions** used, At **Inlet: fixedPressure** At **Outlet: fixedPressure** At **Walls: ZeroGradient**
Slide 9: Velocity Boundary Conditions	**Velocity Boundary Conditions** used, At **Inlet: pressureInletVelocity** At **Outlet: zeroGradient** At **Walls: fixedValue**
Show 3dpipe folder. Show the 3dpipe folder	For executing this case, First, Let's create the **case directory** in '**icoFoam'** folder. Give it some name. I have named it as **'3dpipe'**.
Point the mouse pointer from lid driven folder to 3d pipe folder.	To know the location of this folder, go through the tutorial on **lid driven cavity**. Copy the **'0' (zero), 'constant'** and **'system'** folders of **lid driven cavity** problem in the newly created folder.
Go inside the 3dpipe folder.	Let's go inside the **'3dpipe'** folder.

Hover the pointer over the folder inside the 3dpipe folder.	I have already copied the folders into my '**3dpipe**' folder and modified the files in it.
Go into the '0' folder and open P file and show it	Now, let's go into the '**0**' folder. And open the **'P'** file. This is the **pressure boundary condition** file.
Show the pressure boundary condition file and show the dimensions inside it.	Note that the dimensions are in **(meter square) per (second square) (m2/s2)**.
Show the pressure value written	Hence the **pressure** value in **pascals** is divided by **density**, that is, 1000 **Kg/m3 (Kg per meter cube),** and written here.
Close the file	Let's close the file.
Open U file in the same folder and show	File containing **velocity boundary condition** is as shown:
Close the file and come out of the '0' folder	Let's close the file and come out of the **'0'** folder.
Switch back to the slides	To see the **blocking strategy**, let me switch back to the slides.
Slide 10: Blocking Strategy Hover the pointer on the geometry and drag it towards the z direction.	To create a **3D geometry** of a pipe I have made a **2D** circular **geometry** and extruded the length in the z direction.
Point out the numbering pattern.	Numbering Pattern is as shown.
Minimize the slides	To see the **blockMeshDict** file, let's minimize the slides.
Go to folder 'constant' and then 'polyMesh' and open blockMeshDict file and show it.	Let's go into the folder '**constant',** and then '**polyMesh'.** Final **blockMeshDict** file is as shown:
Close the file and come out of the folder 'constant'	Let's close the file and come out of the '**constant'** folder.
Open and show transportProperties file and point at the value viscosity value	We see the '**transportProperties'** file. Note the **dynamic viscosity** value, here, is 1e-06.
Close the file and come out of the 'constant' folder.	Let's close the file and come out to the '**constant'** folder.

Go into the system folder and open the controlDict file. Show it.	Let's go into the '**system'** folder. Now, let's have a look at the '**controlDict'** file.
Show time step value	The **time step** has been set to 1e-03. The solution **converges** after 18 seconds. The final **time step** is kept 19.
Close the file and the Home folder	Let's close the file. Let's close the '**Home'** folder.
Press 'control', 'alt' and 't' keys altogether	Now to **execute** the case, we will, first, go inside the '**3dpipe'** folder through terminal. Let's open the terminal by pressing **'control', 'alt'** and **'t'** keys, altogether.
Type run and press Enter in the terminal.	Type **run** and press **Enter**
Type cd (space) tutorials and press Enter	Type **cd (space) tutorials** and press **Enter**
Type cd (space) incompressible and press Enter	Type **cd (space) incompressible** and press **Enter**
Type cd (space) icoFoam and press Enter	Type **cd (space) icoFoam** and press **Enter**
Type cd (space) 3Dpipe and press Enter	Type **cd (space) 3Dpipe** and press **Enter**
Type blockMesh and press Enter	Now to create the **mesh**, type **blockMesh** and press **Enter**. **Meshing** has been done.
After the meshing is done, type icoFoam to start the iterations	To start the **iterations** type **icoFoam** and press **Enter**. We can see the **iterations** running.
After the iterations are done, type paraFoam for postprocessing the results and press Enter.	Iterations has been done. After the **iterations** end type **paraFoam** for **postprocessing** the results and press **Enter**.
Click on Apply.	Let's click on **Apply** on the left hand side of the **Object inspector menu** to see the **geometry.**
Rotate the geometry by pressing the button of the mouse and move it in the required direction.	Let's rotate the **geometry** for a better view.
Click on the active variable control menu and select U in the drop-down menu	Click on the **active variable control menu** and select **U** in the drop-down menu.
Click on play button	At the top, in **VCR toolbar**, click on **Play** button.
Go to Object Inspector menu, go to Display, click on Rescale data range	Go to **Object Inspector menu**, go to **Display**, click on **Rescale data range.**

go to the toolbar named common, click on Clips and press Apply	To view the half section, go to the toolbar named **common**, click on **Clips** and press **Apply**.
Open the color legent	Let's open the **color legend.**
	We can see the **maximum velocity** is near to the actual **maximum velocity**.
Go to Filters> Data Analysis> Plot Over Lines	To view the graph Go to **Filters**> **Data Analysis**> **Plot Over Lines.**
click on Y axis and press Apply	Let's click on **Y** axis and press **Apply**.
Point towards the parabolic profile	We see the **parabolic profile** for **Hagen-Poiseuille flow**.
Close the graph	Let's close the **graph**.
Close ParaView	Close **ParaView**.
Switch to the slides	Let's switch to the **slides**.
Slide 11: Summary	In this tutorial we have learned: To create **3D pipe geometry.** To simulate **Hagen-Poiseuille flow** for a **fixed pressure ratio.** To visualize the **velocity** results in **ParaView**. This brings us to the end of the tutorial.
Slide 12 : Assignment	As an assignment, Change the **geometry parameters** such as length and diameter. Change the corresponding **pressure ratio.** Use the fluid of different **viscosity.**
Slide 13: About Spoken tutorials	• Watch the video available at the following link • It summarises the Spoken Tutorial project • If you do not have good bandwidth, you can download and watch it
Slide 14: About Spoken tutorials	The Spoken Tutorial Project Team • Conducts workshops using spoken tutorials • Gives certificates for those who pass an online test • For more details, please write to contact at spoken hyphen tutorial dot org

Slide 15: Acknowledgement	Spoken Tutorial Project is a part of the Talk to a Teacher project • It is supported by the National Mission on Education through ICT, MHRD, Government of India • More information on this Mission is available at • spoken hyphen tutorial dot org slash NMEICT hyphen Intro