Indian Institute of Technology (BHU)

Computational Fluid Dynamics

Practical exam

Duration: 2 hours Marks: 100

Instructions:

- You are allowed to use internet, any book, mobile, calculator etc.
- You are not allowed to talk to your friends, share any information etc.
- Do not write anything on the question paper except your name.
- Evaluation will be done during the exam hours and based on the report you submit. Report creation instruction is given below.
- To create the report, create a ppt file with name: your_name_roll_no.ppt. After the exam, save as this file in .pdf format. This pdf file has to be sent via email (om.prakashh.singh@gmail.com) just after the exam. File not received within 5 minutes after exam will not be considered.
- **Slide 1** should contain: your details, roll no., mobile no. etc.
- For slide 2 onwards, instructions are given below.
- All the files (.hm, .sim, and .ppt) should also be put in the Dropbox/Google Drive account immediately after the exam. The pdf and ppt file should match when compared.

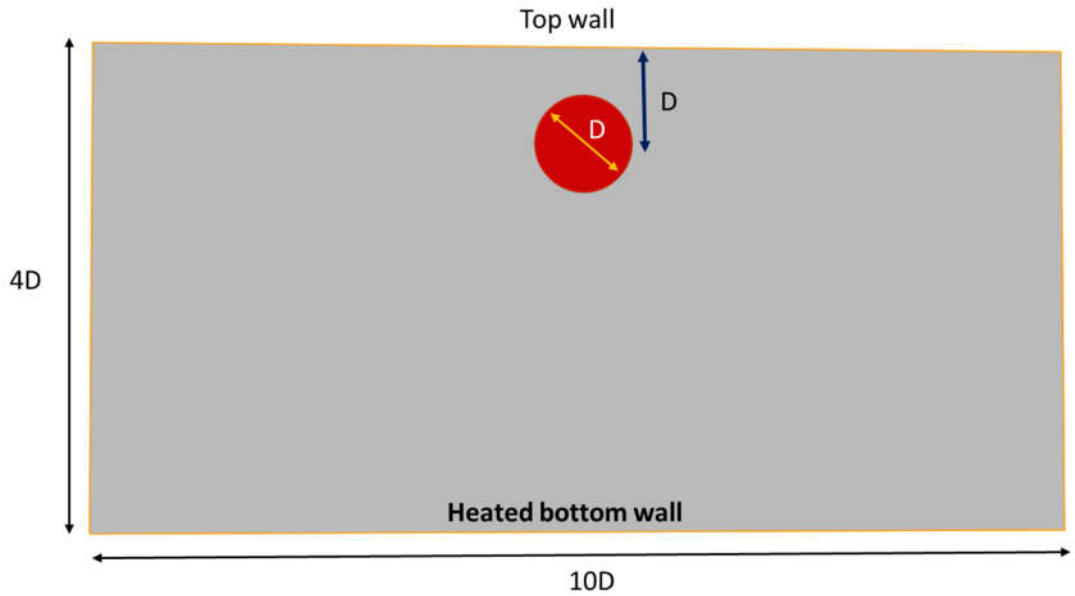

Figure 1

Questions 1. (30 marks)

A cylinder with an explosive gas is place inside an airtight room as shown in the figure 1. The room is initially at 27°C. The cylinder will auto-explode if the average explosive gas temperature reaches 40°C. Due to some unfortunate incident; bottom wall is exposed to a high temperature of 100°C. All other walls

are adiabatic. Find the time when cylinder will explode. Assume that explosive gas has the same physical property as hydrogen. Follow the instructions below.

i. A cylinder of diameter D = 40 mm is placed at the center of the room and is at 40 mm from the top wall. Create the geometry and show the image in **slide no. 2. (marks 2)**

ii. Mesh the cylinder with 1mm and rest of the room with 5 mm grid size. (a) Show the meshed image of whole system (**slide 3**) (b) show the image of zoomed of cylinder and its nearby areas (**slide 4**) (c) save the file in the name as: yourname_roll_no.hm. Put this file in your google drive in appropriate folder. **(marks: 5)**

iii. Develop a 2D transient CFD model with suitable boundary conditions. Use time step = 0.05 seconds or less. Save this model as .cas file and put it in Google drive (**8 marks**).

iv. Run the 2D CFD model.

   a. Generate the temperature contour and show it in **slide 5** (**4 marks**)
   b. Generate velocity magnitude contour and show it in **slide 6** (**3 marks**).
   c. Do you also notice convection in the cylinder? Show the contour (**slide 7, marks 2**)
   d. Show the temperature and velocity contour when the average temperature of the explosive gas reaches 40°C. Also mention the time **(Slide 8, marks 3)**
   e. Describe one strategy to delay the cylinder gas temperature reaching its ignition temperature. **(marks 3)**

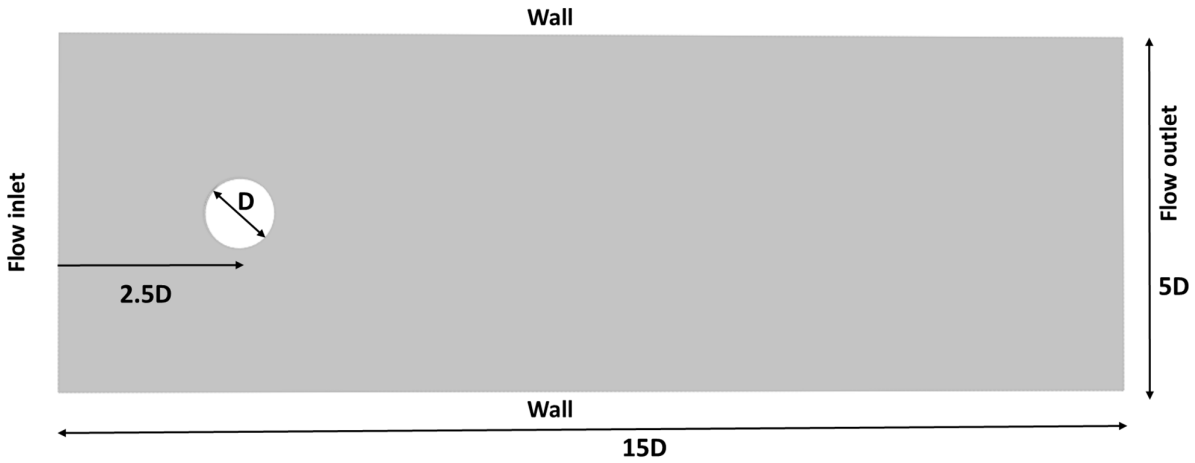

Figure 2

Question 2 (marks 30)

2. Study the effect of combined heat transfer and fluid flow

   a. Figure 2 shows a heated cylinder of diameter D = 40 mm inside a wind tunnel. Air flows from left to right. Develop the design of the system and save the file in Google drive. Show the image **(slide 9, marks 5)**
   b. Mesh the geometry with 3 mm grid size. Generate 10 2D boundary layer mesh around the cylinder. Show the mesh around cylinder **(slide 10)** and whole mesh of the system **(slide 11).** (**Marks 5+5 = 10**).

c. Develop a 2D CFD steady state model with following parameters: laminar flow, v = 0.1 m/s, cold cylinder (same as fluid temperature), pressure outlet boundary condition. Save the model in google drive. Run the model. Show the velocity contour **(slide 12)** and calculate the lift force. **(marks: 5)**

d. Develop a 2D CFD steady model with heated cylinder. Flow conditions remain the same as above but cylinder is now at 50°C. Save the model in Google drive. Show the velocity contour (**slide 13**). Mention any difference in contour plot in (c) and (d). Show the temperature contour plot **(slide 14) (marks 3 + 3 = 6).**

e. Describe two situations in the given problem when (1) flow due to heated cylinder will have significant effect over the flow from inlet side (2) describe a situation when flow from inlet side will dominate over the flow due to heated cylinder. **(Slide 5, marks: 4)**

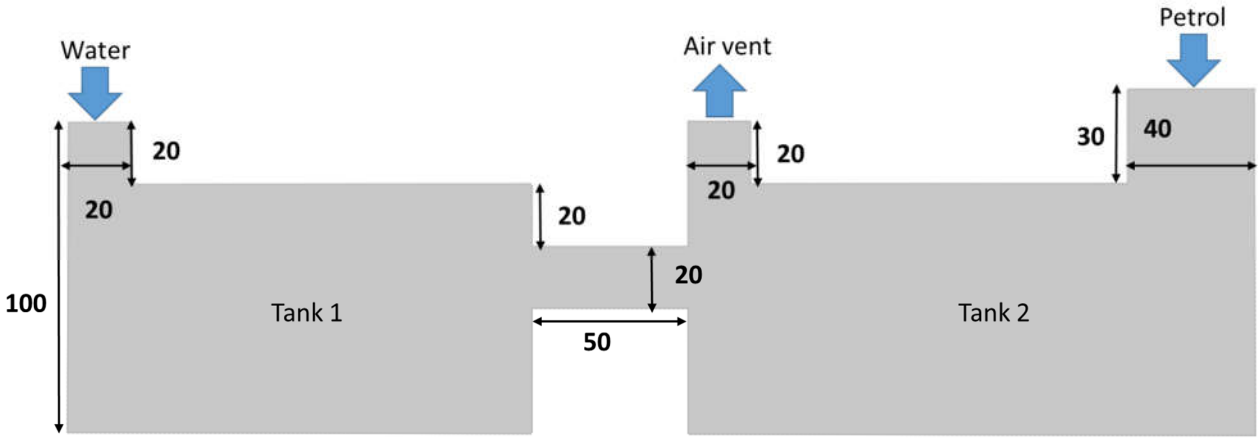

Figure 3

Question 3 (marks 40)

Two water tanks are connected together as shown. By mistake, someone started filling petrol instead of water from tank 2 inlet. Flow rate is same from the inlets i.e. v = 0.2 m/s. Initially both the tanks are empty i.e. filled with air. After what time, water and oil will meet each other and which part of the tank: tank 1, tank 2 or at the connecting pipe?

(a) Design the given model in figure 3 and show it in **slide 15**. Save the model google drive with appropriate name. **(Marks 5)**
(b) Mesh the model with 2 mm mesh size and show the mesh **(slide 17, marks 5)**
(c) Develop 2D CFD multiphase model. Use internet to find appropriate surface tension values between air-water, water-petrol and petrol-air. Save the .cas file in Google drive with appropriate name. **(Marks 20)**
(d) Run the model. Show the contours of petrol volume fraction **(slide 18),** water volume fraction (slide 18) when both these phases meet. Mention the time. **(marks: 10)**