Roll No: ----------------------------

UNIVERSITY OF PETROLEUM & ENERGY STUDIES

DEHRADUN

End Semester Examination – December, 2017

Program/course: M.Tech CFD **Semester –**
Subject: Introduction to CFD **Max. Marks : 100**
Code : MCFD 701 **Duration : 3 Hrs**
No. of page/s: 4 (including Appendix A)

SECTION A – Objective questions (45 points)

1) **Please either fill in the blanks or answer as True or False (15 pts)**

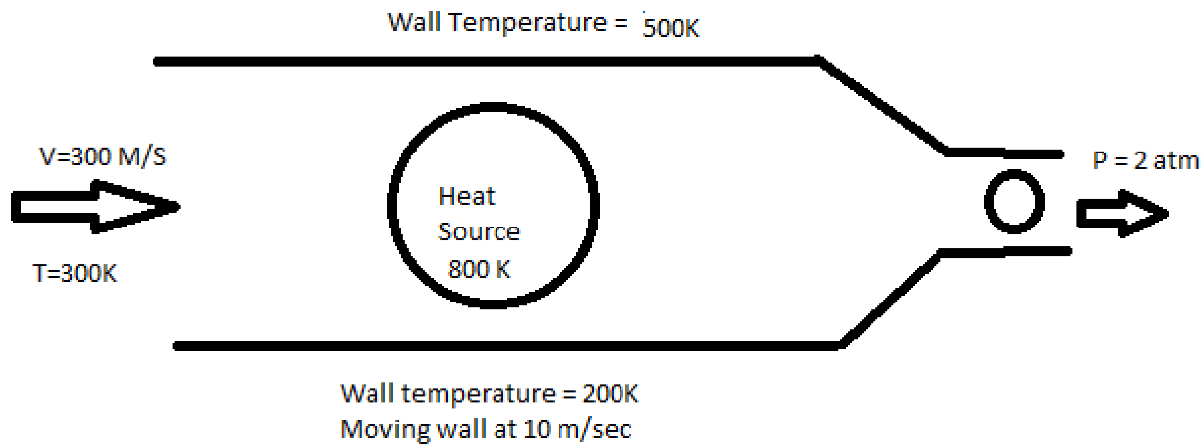
a) If a CFD problem is solved properly, then it is possible to have zero error in the solution of most CFD problems. **(T/F)**
b) While meshing is important, most of the errors in a CFD simulation depend on the post processing and meshing errors can be fixed at the post processing stage**. (T/F)**
c) GAMBIT is a preprocessing meshing software **(T/F)**
d) Reynolds number tells us the interaction of the momentum versus the viscosity in an aerodynamic flow **(T/F)**
e) Any flow with a Reynolds number over 5000 must be a Turbulent flow. **(T/F)**
f) COMSOL is less useful as a physics solver and more useful as a CFD solver due to its finite element interface. **(T/F)**
g) ____ is the process of transforming partial differential equations to algebraic equations.
h) If a partial differential equation and its solution changes over time, then we call it a ____ solution.
i) Relaxation Technique is more appropriate to partial differential equations which do not change over ______________________________________
j) If you have turbulent flow, then it means the particles will be behaving ____________________ and they will be colliding with each other to exchange ______________________ and __________________________.
k) In Aerodynamics, most of the viscous effects are calculated by taking these effects in the ____ rather than the whole flow.
l) In compressible flow when you have a shockwave, the entropy will _____________________ while the velocity will _________________________ across a shockwave.

2) **Please explain in detail the concept of error control in Computational Fluid dynamics problems.** What are the 7 criteria that are used to determine if the solution itself has an error? What is the process of validation? How do you overcome the problems related to error? What is the acceptable rate of error in most engineering problems? What is the cost associated with errors in CFD simulations **(7 pts)**

UNIVERSITY WITH A PURPOSE

3) Take **compressible flow over a flat plate.** Reproduce the conditions required for analysis of this problem. How would you solve this problem in GAMBIT & FLUENT. Write a short tutorial depicting the solution of this problem. Create an overall algorithm **(8 pts)**

4) Please look at the diagram below and answer the following questions.



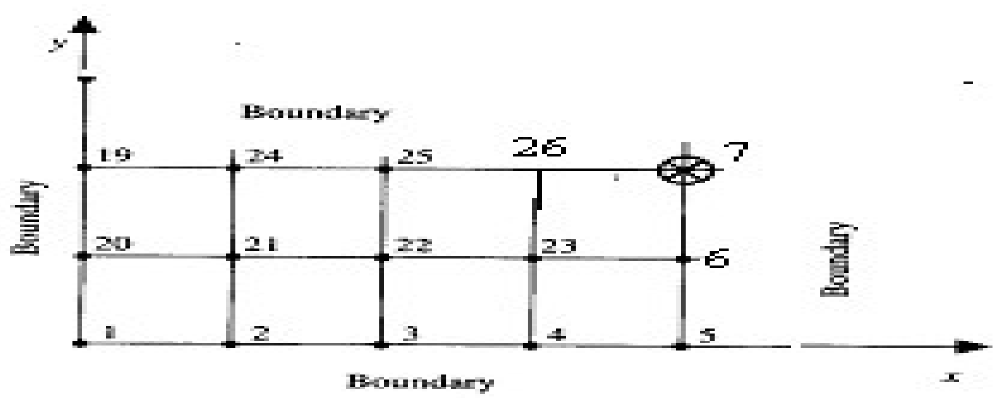
a) Explain the steps required in Gambit to model the flow and the steps required to Mesh it, Use diagrams in your explanation. **(5 points)**

b) Explain the steps for the same problem for FLUENT. Describe all of the steps in detail and use diagrams where appropriate. What kinds of boundary and initialization conditions have been used? **(5 points)**

c) What will be the effect of the heat source on the flow itself? How do you think the flow itself will develop? What kinds of flow conditions are prevalent? What do you think will be the flow situation at the end of the system**? (5 points)**

SECTION B –Numerical questions (30 points)

5) Use the diagram below and apply the Relaxation technique to find the solution. (10 pts)



Use the **Relaxation Technique** to solve for **points 21, 22 and 23 for** ***velocity potential*** by taking 3 iterations. Take boundary conditions points 1, 2,3,4,5 = 200, points 19, 20= 150, points 24, 25,

UNIVERSITY WITH A PURP

26 = 200 and points 6,7 = 400. Find the velocity potential at points 21, 22, and 23, then solve for velocity in uniform flow. Show at least 3 iterations. Write a flowchart for the solution as well.

6) Use the two dimensional flow equations below to solve for **y velocity (v) and energy (e)** using the McCormack Technique. Also write the flowchart as well. **(15 pts)**

$$\text{Continuity}: \quad \frac{\partial \rho}{\partial t} = -\left(\rho\frac{\partial u}{\partial x} + u\frac{\partial \rho}{\partial x} + \rho\frac{\partial v}{\partial y} + v\frac{\partial \rho}{\partial y}\right)$$

$$x\ \text{momentum}: \quad \frac{\partial u}{\partial t} = -\left(u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + \frac{1}{\rho}\frac{\partial p}{\partial x}\right)$$

$$y\ \text{momentum}: \quad \frac{\partial v}{\partial t} = -\left(u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + \frac{1}{\rho}\frac{\partial p}{\partial y}\right)$$

$$\text{Energy}: \quad \frac{\partial e}{\partial t} = -\left(u\frac{\partial e}{\partial x} + v\frac{\partial e}{\partial y} + \frac{p}{\rho}\frac{\partial u}{\partial x} + \frac{p}{\rho}\frac{\partial v}{\partial y}\right)$$

7) Imagine the viscous flow over a curved surface. The flow velocity and the temperature changes only in the y direction. The following data are obtained. calculate the Reynolds number as well as the shear stress and heat transfer at every point from 0 to 0.40 **(5 points)**

Take μ= 1.78 × 10^{-5} kg/(m·s) and k=0.025 W/(m·K), and ρ=1.22

y (m)	u(m/s)	T(C)
O	0	20
0.10	50	28.4
0.20	65.75	38.55
0.30	80.65	48.22
0.40	99.87	59.78

SECTION C –Application questions (25 points)

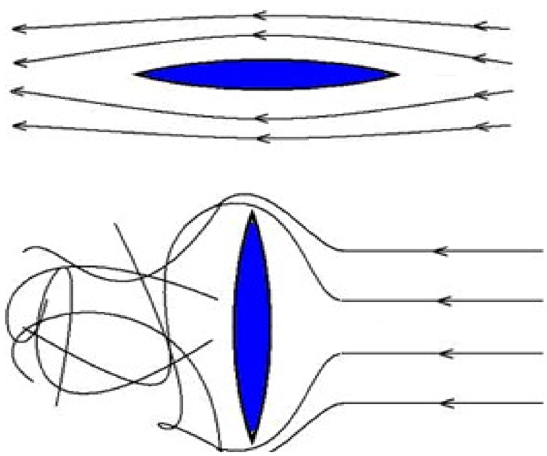
8) Write a FORTRAN code for the problem in Question 5 using the same parameters which has been given in the question. Please make sure that the FORTRAN code is written in a user friendly manner. **(15 pts)**

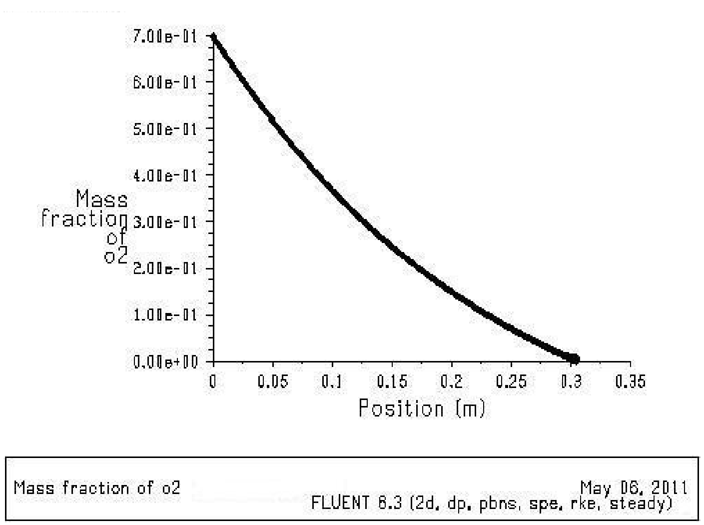
9) Explain the following diagrams and their meanings if they are a result of a CFD simulation. (Please write on the Appendix A and give it along with the answer script. DO NOT WRITE on the answer script, **<u>your answers will only be read from the sheet in APPENDIX A</u>**). **(10 pts)**

UPES
UNIVERSITY WITH A PURPOSE

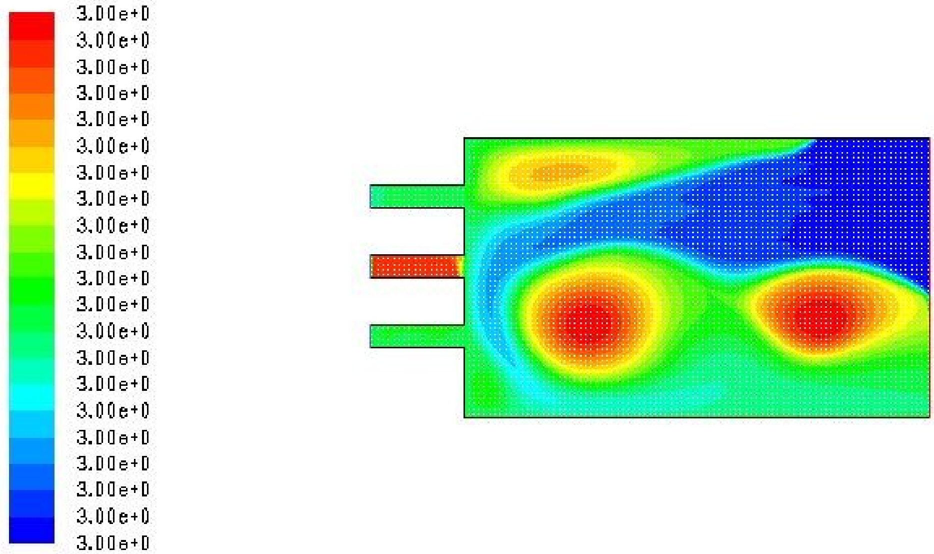
APPENDIX A

(PLEASE WRITE THE ANSWER OF QUESTION 9 ON THIS PAPER AND GIVE IT WITH YOUR ANSWER SCRIPT)

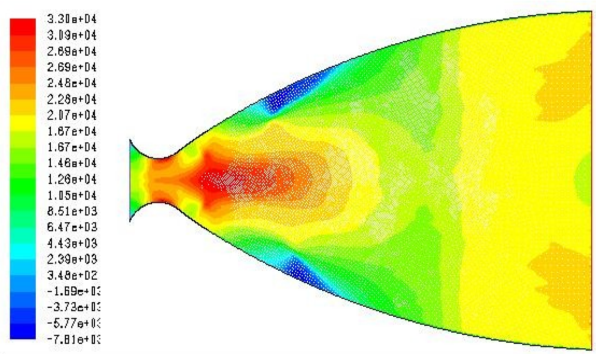




UPES
UNIVERSITY WITH A PURPOSE



Contours of Static Temperature (k) May 07, 2010
FLUENT 6.3 (2d, pbns, lam)



Contours of X Velocity (m/s) (Time=4.5000e+01) Apr 08, 2011
FLUENT 6.3 (2d, dp, dbns imp, rke, unsteady)