

Indian Institute of Technology (B.H.U)

Sub: ME 544/ME441

CFD lab test

Duration: 2 hours

Marks: 60

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.
- Mention your set number in the answer sheet. If not mentioned, set A will be assumed by default.
- 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 (stccfdem@gmail.com) just after the exam. File not received within 5 minutes after exam will not be considered.
- In the email subject line, mention: **CFD Class test 2019**
- **Slide 1** should contain: your details, roll no., mobile no., email id etc.
- For slide 2 onwards, instructions are given below.
- All the files (.hm, .cas, 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.



QUESTIONS:

External Aerodynamics (50 marks)

1. Download the T38 Talon jet 3D stl model from www.omprakashsingh.com/teaching and develop a CFD model to perform external aerodynamics of the jet. Do not scale the model.
 - A. Remesh the jet with an element size of 1.5 or less. You may use shrink wrap or any other technique. Show the image of remeshed design (**Slide 2, Mark 10**)
 - B. Develop a suitable wind tunnel around the jet and mesh the wind tunnel. Organize various faces of the wind tunnel into inlet, outlet etc. Take front side of jet as inlet. Show the image of the meshed tunnel (**Slide 3, Mark 15**).
 - C. Develop 3D Volume of the system. Show a section of the wind tunnel with jet (**Slide 4, Mark 10**).
 - D. Import the model into fluent and perform CFD analysis. Take inlet velocity as 3m/s, use any turbulent model. Run the model. Show the pressure contour on jet (**Slide 5, Mark 10**).
 - E. Report the magnitude of drag and lift force acting on the jet (**Slide 6, Mark 5**).

Internal Aerodynamics (50 marks)

2. Use the same model of the T38 Talon jet 3D stl model. Develop a CFD model to perform internal aerodynamics of the jet. Take few elements on the nose as inlet and few elements on the two exits as outlet.
 - A. Remesh the jet model using element of size 1.5 or less. Develop 3D Volume mesh of the jet. Show a section of 3D mesh (**Slide 7, Mark: 15**)
 - B. Take few elements on the nose as inlet and few elements on the two exits as outlet. Take velocity of 3m/s, use any suitable turbulent flow. Run the CFD model. Show the pressure contours on the surface of the jet (**Slide 8**) (**Mark: 15**).
 - C. Create a section passing through the centre of the jet touching the nose. Show velocity magnitude, velocity vector and pressure contour on the section (**Slide 9**) (**Mark: 20**).