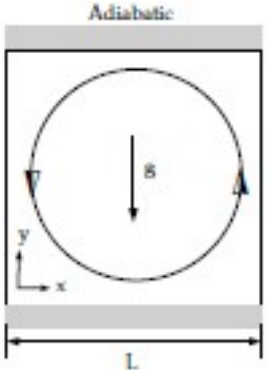
Name: **Enrolment No:**	UPES

UNIVERSITY OF PETROLEUM AND ENERGY STUDIES
End Semester Examination, December 2018

Course: Commercial CFD Software Applications **Semester: III**
Programme: M.Tech CFD
Time: 03 hrs. **Max. Marks: 100**
Instructions: Q9 a) is compulsory, Q9 (b) is having internal choice

SECTION A

S. No.		Marks	CO
Q 1	Explain about: 1. Turbulent Viscosity ratio 2. UDF 3. Frozen Flux formulation 4. Pseudo Transient method	8	CO1 & CO2
Q 2	Explain the various industrial applications where CFD is being used?	4	CO1
Q 3	Consider a flow over a airfoil at 7 m/s having length 1 m, calculate the Y+ value (μ=1.8*10^-5 kg/ms, y= 10^-3 m)	4	CO3
Q 4	Discuss about the Inertial Frame of Reference System?	4	CO2
	SECTION B		
Q 5	Discuss about the two-equation and four-equation turbulence models and explain in which type application area we use them?	10	CO2
Q 6	What do you understand by moving reference frame system? Write about interface treatment of Multiple Reference Model for the moving parts and explain how the interface treatment of absolute velocity formulation and relative velocity formulation occurs in the domain?	10	CO1
Q 7	Explain about the volumetric reactions? Discuss about the modeling of reaction source term in volumetric reaction using laminar finite rate model? **(Or)** Define Mixture Fraction. Explain the Mixture Fraction theory approach for modeling Non-Premixed Combustion?	10	CO2
Q 8	Explain the various parameters, based on which we can able to choose the appropriate radiation model?	10	CO3
	SECTION-C		
Q 9	(a)Write the Radiative Heat transport equation? **(5-Marks)** (b) A square box of side L has a hot right wall at T = 1500 K, a cold left wall at T = 500 K, and adiabatic top and bottom walls. Gravity acts downwards. A buoyant flow	20	CO4

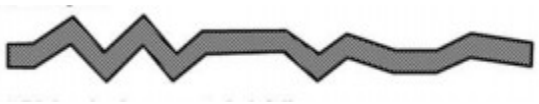
	develops because of thermally-induced density gradients. The medium contained in the box is assumed to be absorbing and emitting. All walls are black. The objective is to compute the flow and temperature patterns in the box, as well as the wall heat flux, using the P1 radiation model. Explain in detail procedure how to carry out the numerical analysis in **FLUENT**, and to compare their performance for different values of the optical thickness aL. The working fluid has a Prandtl number of approximately 0.71, and the Rayleigh number based on L is 5 X 10^5. The Planck number k / ($4\sigma L T_0^3$) is 0.02, and measures the relative importance of conduction to radiation; here $T_0 = (T_h + T_c) = 2$. Three values for the optical thickness are considered: aL = 0, aL = 0.2, and aL = 5. ρ = 1000 kg/m^3 k = 15.309 W/mK, $\mu = 10^{-3}$ kg/ms, $\beta = 10^5$ 1/K, g = -6.96 x 10^{-5} m/s^2, c_p= 1.1030x10^4 J/kgK, a = 0, 0.2, 5 1/m, L = 1 m. **(15-Marks)**  Fig: Schematic diagram for given problem **(Or)** (b)A three-dimensional box 0.2m x 0.2m x 0.2m has a hot wall of aluminum at 473 K. All other walls are made of an insulation material and are subject to radiative and convective heat transfer to the surroundings which are at 293 K. Gravity acts downwards. The medium contained in the box is assumed not to emit, absorb, or scatter radiation. All walls are gray. The objective is to compute the flow and temperature patterns in the box, as well as the wall heat flux, using the surface-to-surface (S2S) model. Explain how do you carry out the numerical simulation for the mentioned problem statement in detail with proper justification **(15- Marks)**		
Q 10	(a)What is the role of orthogonal quality and skewness in grid generation?**(5-Marks)** (b) Suggest the domain length and how will you perform grid generation for an airfoil with protrusion at 0.50chord location, Reynolds number is 100000. **(5 – Marks)** (c) Consider the flow over an airfoil with protrusion at 0.50c location at Reynolds number 100000. Explain how the transient numerical analysis can carried out using the FLUENT for the mentioned problem in detail at different angles of attack (0^0 to 20^0) and discuss about the expected results obtained from simulation? **(10 – Marks)**	**20**	**CO3 & CO4**

Name: **Enrolment No:**	UPES

UNIVERSITY OF PETROLEUM AND ENERGY STUDIES
End Semester Examination, December 2018

Course: Commercial CFD Software Applications **Semester: III**
Programme: M.Tech CFD
Time: 03 hrs. **Max. Marks: 100**
Instructions: Q9 is having internal choice

SECTION A

S. No.		Marks	CO
Q 1	Explain about: 1. Non – Iterative Advancement 2. High Relaxation factor 3. Turbulent Intensity 4. Grid Adaptation	**8**	**CO1 & CO2**
Q 2	Explain the Biomedical and environmental applications where CFD is being used?	**4**	**CO1**
Q 3	Consider a flow over a corrugated dragonfly airfoil at 5m/s having length 101mm, calculate the value of y ($\mu=1.8*10^{-5}$, y+= 1)  Figure: Corrugated Dragonfly Airfoil	**4**	**CO3**
Q 4	Define the Grid independence study? Explain its importance?	**4**	**CO3**

SECTION B

Q 5	Discuss about the one-equation and four-equation turbulence models and explain their applications?	**10**	**CO1**
Q 6	Explain the advantages and disadvantages of Rooseland and Discrete transport radiation models?	**10**	**CO1**
Q 7	Write about interface treatment of Mixing Plane concept, algorithm and explain how the interface treatment of absolute velocity formulation and relative velocity formulation occurs in the domain?	**10**	**CO2**
Q 8	Develop the various chemical configurations for mixture fraction? Explain the restrictions on the mixture fraction approach? **(Or)** Explain in detail about the Premixed and Non-Premixed combustion models?	**10**	**CO4**

SECTION-C			
Q 9	Explain about: 1. Eddy dissipation concept model 2. Zimont Turbulent Flame Closure Theory 3. Moving Reference Frame system 4. Laminar Flamelet Model **(Or)** (a)What is User Defined Function? Explain about Complied and Interpreted UDFs? Suggest which type of UDF is best? **(10-Marks)** (b)Consider that grid generation has to carried out for axial flow compressor, suggest which type of grid you consider and why do you choose that particular grid generation with detailed explanation? What is the role of Aspect ratio, skewness and orthogonal quality in grid generation? **(10 Marks)**	**20**	**CO3 & CO4**
Q 10	Consider the flow over a Formula-1 race car travelling at 215 km/h, suggest the type of solver to use, turbulence model, value of turbulent intensity and viscosity ratio. Explain how to carry out numerical analysis using commercial CFD Software with proper explanation? How can you reduce the drag coefficient, explain with the help of any flow mechanisms?	**20**	**CO4**