

Syllabus for 'Industrial training program in CFD solution using Fluent or cfx

Course phasing

By : Karthik.V (M-tech in Aerospace)

Sr.No	Course contents	Date	Student Signature
1	Over view of Fluent GUI		
2	List of commands and applications		
3	Navigation panel		
4	Types of dialog boxes		
5	Reading and writing of files		
6	Grid or case file import		
7	Unit system		
8	Reading and manipulating of imported mesh		
9	Mesh checking and quality report		
10	Scaling and un-scaling mesh		
11	Managing and editing mesh zones		
12	Mesh rotation ,translation		
13	Cell zone and boundary conditions		
14	Cell zone condition (fluid or solid properties, material, source)		
15	Pressure inlet ,Velocity inlet, pressure farfield , farfield, symmetry, pressure outlet etc....Bc's ,		
16	Defining physical properties density, viscosity, thermal conductivity, specific heat capacity etc..		
17	Modeling basic fluid flow defining the turbulence model and solution method		
18	Using solver pressure based or density based		
19	Monitoring the solution convergence, residuals		
20	Printing and plotting of residuals		
21	Editing the solution criteria and number of iteration by interrupting the solver.		
22	Checking and reading of results		
23	Creating contour plots, vector plot of cp ,cl,cm,temperature,pressure etc...		
24	Plotting a graph of various flow properties		
25	Calculation of aerodynamic forces in problem		
26	Animating the result		
27	Creating the report,pictures,data files etc...		
28	Validation of CFD results with experimental data		