IIT-Spoken Tutorial OpenFOAM
Student Development Program
Mechanical Department

Nutan Maharashtra Institute of Engineering & Technology, Talegoan, Pune, Maharashtra

01/09/2020 to 05/09/2020 (32 Hrs)

Day I- Schedule

Tutorial No.	Task/Subject	Duration
01	Overview of Open FOAM	120 Min
02	Installing OpenFOAM in Ubuntu Linux	180 Min
03	Setting-up a Test Case in OpenFOAM	180 Min

Day II- Schedule

Tutorial No.	Task/Subject	Duration
04	Creating 2D Channel Geometry and Mesh in OpenFOAM	180 Min
05	Multi-block Meshing of a 2D Geometry in OpenFOAM	180 Min

Day III- Schedule

Tutorial No.	Task/Subject	Duration
06	Creating 3D Pipe Geometry and Mesh in OpenFOAM	180 Min
07	Simulating Hagen Poiseuille flow through a pipe in OpenFOAM	180 Min

Day IV- Schedule

Tutorial No.	Task/Subject	Duration
08	Basic Post Processing using ParaView	180 Min
09	Simulation a 2D Turbulent Flow in a Channel using OpenFOAM	180 Min

Day V- Schedule

Tutorial No.	Task/Subject	Duration
10	Turbulence Modelling in OpenFOAM	180 Min
11	Grid Resolution and Convergence in OpenFOAM	180 Min

Principal
Nutan Maharashtra Institute
of Engg. & Technology
"Samarth Vidya Sankul" Vishnupuri
gaon Dabhade, 410507