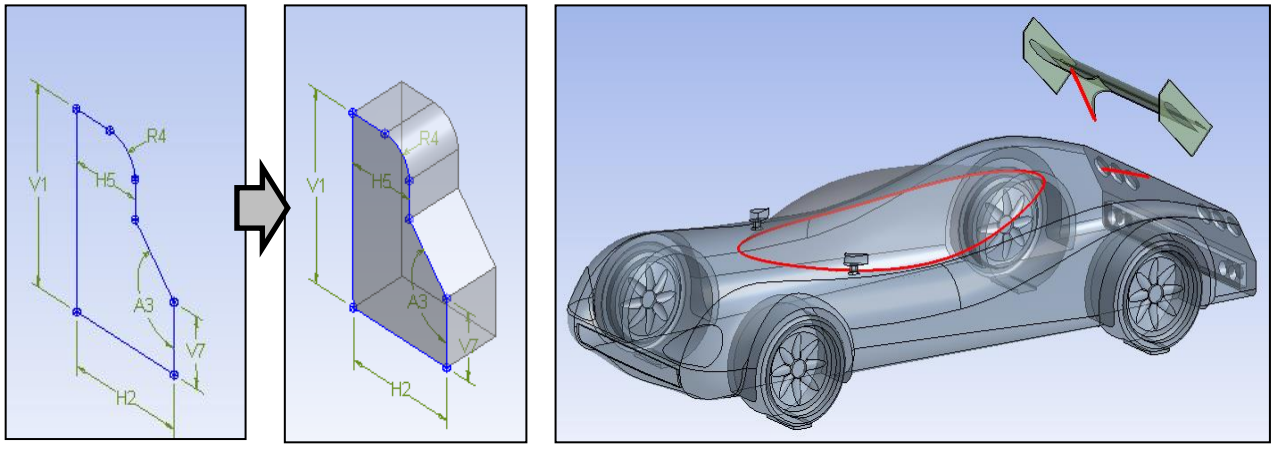
ANSYS CFD Training Agenda

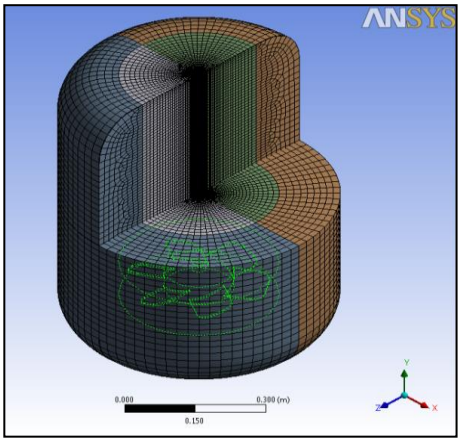
DAY 1:

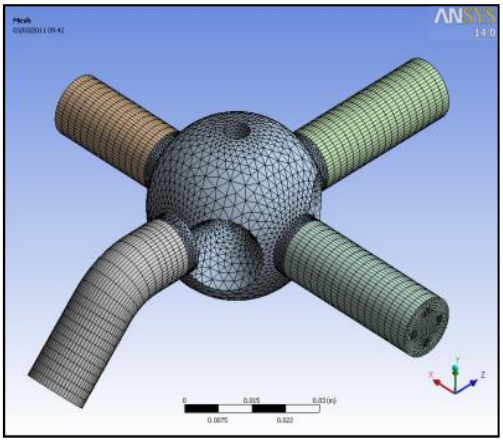
TOPIC	DURATION	LEARNING OUTCOME
Introduction to Numerical Modelling	Pre-Lunch	Introduction to Numerical modelling and its role in in the field of fluid flow and heat transfer Numerical vs Analytical Vs Experimental Analysis Introduction to CFD. What, when to use, and Why? Introduction to Ansys capabilities and CFD applications
Introduction to Ansys Workbench and Design Modeler		Overview of ANSYS Workbench and Design Modeler Interface How to launch Workbench and GUI navigation Capabilities and file structure
Modelling Sketch and 3-D geometry		Concept of Plane & Sketch, Sketching interface & toolbox, How to create Planes and Sketches How to Draw, Modify, Dimension and Constrain sketches How to create and modify 2D and 3D geometry
Lunch Break		
Workshops on sketching and Modeling	Post-Lunch	Hand on Exercise to Creating a sketch and 3d geometry, Use of Various 3D operation, how to modify geometry, GUI navigation and Saving the project
Geometry clean up and Advanced 3-D functions		Body Types & States (Frozen & Active) in DM, Boolean operation, Use of Multi-body parts and Share topology, How to Import generic CAD formats (native & Neutral) Clean up process for corrupted and disconnected geometry, how to simplify geometry for CAE analysis How to Decompose geometry into mesh able sections
Workshops on cleanup		This workshop will demonstrate the import, cleanup, preparation and decomposition of an automotive CAD model.



DAY 2:

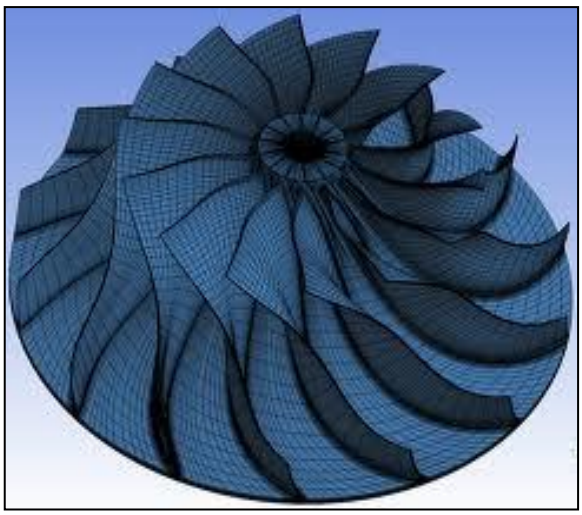
TOPIC	DURATION	LEARNING OUTCOME
Introduction to Meshing Fundamental	Pre-Lunch	Need of mesh and Its process Meshing fundamentals (Efficiency, accuracy & quality) Type of mesh/element (hex, Tet, prism, tri etc.) Introduction to structured and unstructured meshes
Introduction to Ansys Meshing		What is the ANSYS Meshing? How to launch ANSYS Meshing? Overview on Meshing methods & Mesh controls ANSYS Meshing graphics user interface
Workshop on ANSYS Meshing		Basic Hand on Exercise for mesh generation to learn different steps involves Generating a mesh, creating Named Selections, adding Inflation and examining the mesh and its quality
Meshing methods Tetrahedrons		Algorithms for Tetrahedral Meshing Difference between Patch dependent and Patch conformal mesh, Global inflation in tetrahedral meshing
Lunch Break		
Meshing methods Sweep and Multizone	Post-Lunch	Overview and Advantage of Hex Meshing Different methods for Hex Meshing How to create Structured mesh Use of selective meshing in Multiple Bodies part
Workshops on Meshing Methods		Hand on Exercise for mesh generation to learn different steps using various methods. Use of Advanced Size Functions Use of Curvature, Proximity, Global Inflation How to check Mesh statistics and quality
Global Mesh controls		Introduction to Global Mesh Settings and controls Description of Physics Based Settings Smoothing and Transition Inflation Option, Inflation Algorithm Defeaturing Mesh statistics and Quality criteria

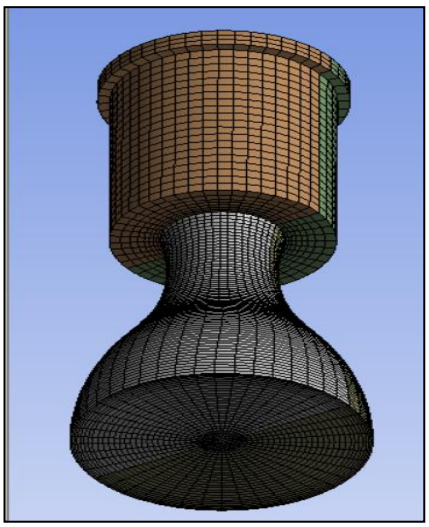




DAY 3:

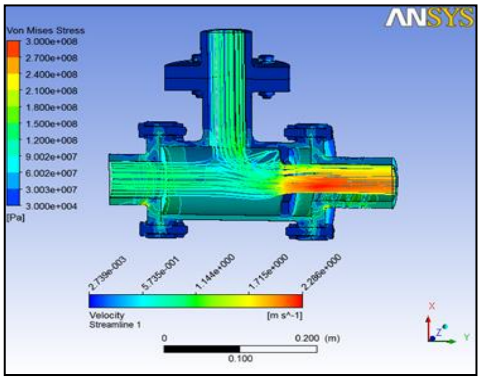
TOPIC	DURATION	LEARNING OUTCOME
Advance size functions and Advance Inflation options	Pre-Lunch	Introduction to various advance option for sizing, Relevance and Relevance Center Use of Curvature & Proximity Smoothing and Transition of mesh Curvature Normal Angle
Workshop on Global Mesh Control		Hand on Exercise to Generate mesh Use of Advanced Size Functions Using Curvature, Proximity, Global Inflation How to check Mesh statistics and quality using various method
Local Mesh Controls		Various local mesh settings (Mesh sizing, Refinement, Match Control, Inflation, etc.) How to apply local controls? Effect of local controls on mesh
Lunch Break		
Workshop on Local Mesh Control	Post-Lunch	Hand on Exercise to Generate mesh using Local control setting Use of Local sizing, Biasing and inflation How to improve quality using Local control
Mesh quality		Impact of the Mesh Quality on the Solution Various methods for checking the mesh quality How to improve quality in Meshing Advance option to improve quality (Pinch, Virtual topology) Tips and tricks
Workshop on - Automotive External flow		This workshop will demonstrate the practical application of ANSYS Meshing to an automotive external aero mode. This includes body of influence, local control, virtual topology etc.

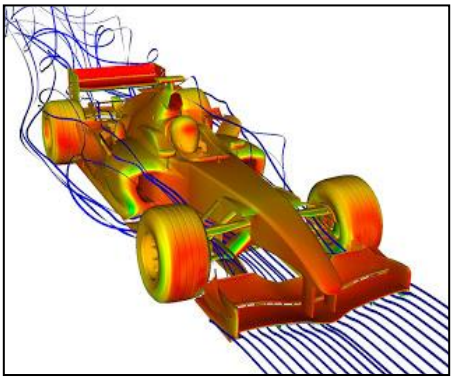




DAY 4:

TOPIC	DURATION	LEARNING OUTCOME
Introduction to ANSYS CFD and FLUENT with demo	Pre-Lunch	Introduction to CFD modelling The basics of what CFD is and how it works Principles of Conservation and scalar transport equation How to go from the original planning stage to analyzing the end results The different steps involved in a successful CFD project
Boundary conditions and cell zone conditions		How to define material properties The different boundary condition types in FLUENT and how to use them How to define cell zone conditions including solid zones, porous media and rotating frame How to define temperature dependent properties How to specify well-posed boundary conditions
Workshop on Internal flow in Mixing tee		Hand on Exercise for Thermal analysis using Fluent, The aim is to learn different steps involved (i.e. reading mesh, defining boundary conditions, selecting material properties, setting up solution monitors, running the simulation and analyzing result) in a successful CFD project
Lunch Break		
Post processing	Post-Lunch	Performing flow field visualization and quantitative data analysis How to do this in FLUENT and in CFD-Post How to create Isosurfaces, Vector plots, Contour plots, Streamlines, XY plotting, Animation creation and more
Solver Basics and settings		How to choose the solver and the discretization schemes How to initialize the solution How to monitor and judge solution convergence and accuracy
Workshop on External aerodynamic (flow over an Airfoil)		The aim of this workshop is to understand various steps requires for external aerodynamics application which includes, Assessing Y+ for correct turbulence model behavior Drag and lift calculation over a NACA airfoil Modifying solver settings to improve accuracy Reading in and plotting experimental data alongside CFD results Producing a side-by-side comparison of different CFD results.





DAY 5:

TOPIC	DURATION	LEARNING OUTCOME
Heat Transfer	Pre-Lunch	How to treat conduction, convection and radiation How to set wall thermal boundary conditions How to solve natural convection problem
Workshop on - Conjugate heat transfer		The aim is to learn different steps involve to simulate various form of heat transfer i.e. Thermal conduction, radiation and Natural convection Case comparison on electronic PCB board
Turbulence modeling		Introduction to Basic turbulent flow and turbulence modeling theory Turbulence models and near-wall options available in FLUENT How to choose an appropriate turbulence model for a given problem How to specify turbulence boundary conditions at inlets
Workshop - Turbulent Flow Past a Backwards Facing Step		The workshop covers many aspects of turbulent flow modeling in Fluent including specifying models and near wall treatments, checking y+, selecting boundary conditions, comparison with experimental results and comparison of results obtained with different turbulence models
Lunch Break		
Transient condition	Post-Lunch	How to set up and run transient calculations How to choose the appropriate time step size for your calculation How to post-process transient data and make animation
Workshop on flow over a cylinder (vortex shedding)		The purpose of this workshop is to introduce good techniques for transient flow modeling. Calculation of drag force and Strouhal number Transient post-processing in CFD-Post Generating images during the simulation
Multiphase modeling		Overview and basic concept of multiphase flow and its modelling Defining boundary condition for multiphase flow
Workshop on tank flush (siphoning Effect)		This workshop teaches the different steps involve in multiphase flow modelling How to do Transient flow modelling How to prepare animation
Tips & tricks and best practices		How to specify well-posed boundary conditions Strategies for minimizing error Best practices for turbulence models and mesh creation