

ANJUMAN COLLEGE OF ENGINEERING AND TECHNOLOGY,

SADAR, NAGPUR

DEPARTMENT OF MECHANICAL ENGINEERING

TECHNICAL TRAINING (2015-16)

ANSYS SYLLABUS

PART-I

INTRODUCTION TO FEA AND ANSYS

Introduction to FEA: General Working of FEA Nodes, Elements, and Element Shapes General Procedure of Conducting Finite Element Analysis FEA through ANSYS Effective Utilization of FEA FEA Software Advantages and Limitations of FEA Software

Key Assumptions in FEA: Assumptions Related to Geometry Assumptions Related to Material Properties Assumptions Related to Boundary Conditions Assumptions Related to Fasteners

Important Terms and Definitions: Strength (Resistance to Deformation) Load Stress Strain Elastic Limit Ultimate Strength Factor of Safety Lateral Strain and Poisson's Ratio Bulk Modulus Creep Engineering Materials

Introduction to ANSYS: System Requirements, Getting Started with ANSYS, Interactive Mode, Batch Mode, Starting a New File Using the ANSYS Product Launcher window

ANSYS Metaphysics Utility Menu Window (ANSYS Session): Utility Menu Main Menu Graphics Area Standard Toolbar ANSYS Command Prompt Command Window Icon Raise Hidden Icon Reset Picking Contact Manager ANSYS Toolbar Model Control Toolbar User Prompt Information Current Settings Setting the Analysis Preferences

Units in ANSYS

Other Important Terms Related to ANSYS: Dialog Boxes Graphics Display Panning, Zooming, and Rotating the Model Dividing the Graphics Area The Pan-Zoom-Rotate Dialog Box Graphics Picking Using Mouse Buttons for Picking ANSYS Database and Files Saving the File Resuming the File Clearing the Database Some Basic Steps in General Analysis Procedure Points to Remember while Performing an Analysis, Exiting ANSYS.

BASIC SOLID MODELING

Solid Modeling in ANSYS: Solid Modeling and Direct Generation

Solid Modeling Methods: Bottom-up Construction Top-down Construction

Considerations before Creating a Model for Analysis: Details Required and Symmetry

Creating Geometric Entities: Creating Lines Creating Arcs Creating B-Splines Creating Fillets between Intersecting Lines Creating Areas

Creating and Modifying Work planes: Display Working Plane Show WP Status WP Settings Offset WP by Increments Offset WP to Align WP with.

Dr. Akash M. Langde
Professor and Head
Mech. Engg. Deptt.
Anjuman College of Engg. & Tech.
Sadar, Nagpur

Coordinate Systems in ANSYS: Global Coordinate System, Local Coordinate System, Active Coordinate System, Display Coordinate System, Nodal Coordinate System, Element Coordinate System, Results Coordinate System, Creating New Coordinate Systems, Deleting Existing Coordinate.

ADVANCED SOLID MODELING

Advanced Solid Modeling: Creating Volumes Extruding Entities Extending the Line
Creating Complex Solid Models by Performing Boolean Operations Modifying the Solid Model
Scale, Move, Copy, Reflect.
Deleting Solid Model Entities: Importing Solid Models, Importing the IGES File, Importing Models from Pro/ENGINEER, Importing the Model from Unigraphics.

PART-II

FINITE ELEMENT MODELING (FEM) – I

An Overview of the Finite Element Modeling Element Attributes, Element Types, and Reasons Why ANSYS has a Large Element Library. Real Constants, Material Properties, Multiple Attributes.
Assigning Multiple Attributes before Meshing, Assigning Default Attributes before Meshing, Modifying Attributes after Meshing, Verifying Assigned Attributes and Element Attributes Table.

FINITE ELEMENT MODELING (FEM) – II

Finite Element Modeling (FEM) – II, Mesh Generation, Mesh Density, Meshing the Solid Model, Setting Element Attributes, Defining the Mesh, Defining the Entity to be Meshed and Defining the Meshing Type.

Meshing the Model, Refining the Mesh, Locally Extruding the Mesh and Transitional Pyramid Elements.
Requirements for Creating Pyramid Elements: Creating Transitional Pyramid Elements (Hex-to-Tet Meshing), Converting Degenerate Tetrahedral (20 nodes) Elements into Non-degenerate (10 nodes) Tetrahedral Elements and Plotting Pyramid Elements.

Meshing the Beam with Orientation Nodes: Creating the Beam Mesh with Orientation Nodes and Creating the Beam Mesh with Two Orientation Nodes.

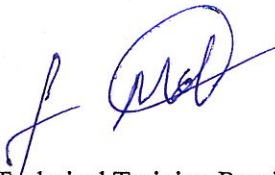
Applying Loads: The Nodal Coordinate System, Loads in Different Disciplines, Types of Loads in ANSYS, Load Steps, Sub steps, and Time Applying Loads.

Deleting Loads: Deleting DOF Constraints, Deleting all Loads and Load Step Options, Deleting all Loads Applied on Solid Model, Deleting all Loads Applied on Finite Element Model.



Technical Training In-charge

Prof. S. Mohiuddin



Technical Training Provider

Astral Infometrics Pvt. Ltd.



H. O. D.

Prof. Dr. Akash Langde
Dr. Akash M. Langde
Professor and Head
Mech. Engg. Deptt.
Anjuman College of Engg. & Tech.
Sadar, Nagpur

CERTIFICATE

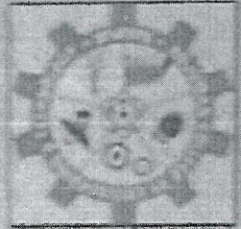
Certificate

Awarded to Mr / Ms.

Aarif Bader

Student of

Mechanical Engineering Department



Anjuman College of Engineering and Technology

for successful completion of Technical Training Program on

Ansys

conducted between

17th Sept 2016 to 25th Oct 2016

Dr. Khalid Al-Labadi
Dr. Khalid Al-Labadi
Dr. Said Al-Harbi
Dr. Said Al-Harbi
Dr. Said Al-Harbi
Dr. Said Al-Harbi

