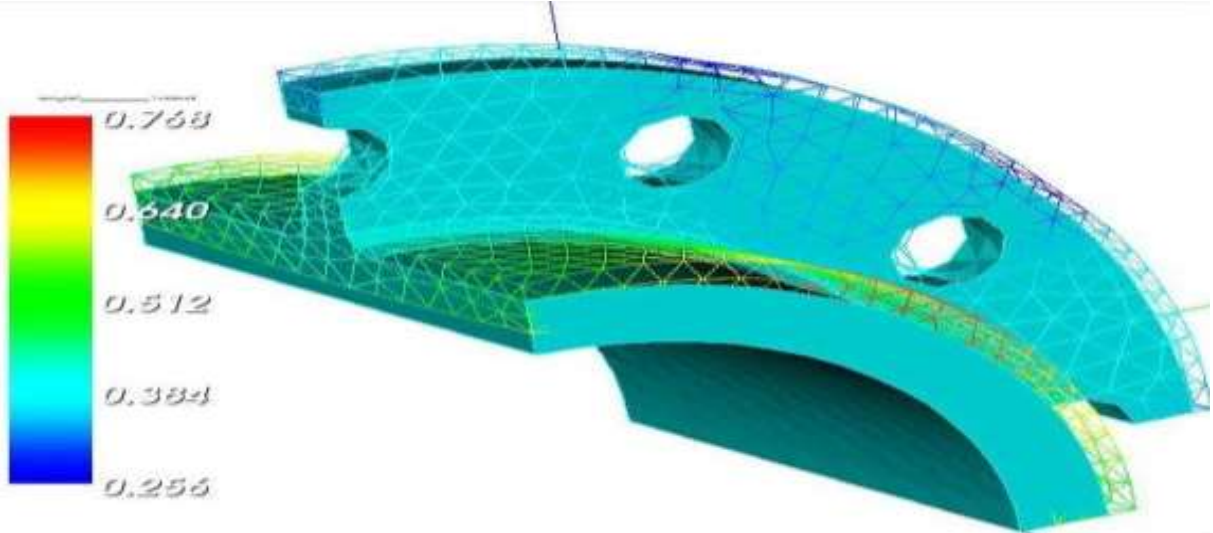




MODEL CURRICULUM



Qualification Name:

Technical Supervisor - Computer Aided Engineering

Qualification Code:

Version: 2.0

NSQF Level: 5

Model Curriculum Version: 2.0

Submitted By:

MSME TECHNOLOGY CENTRE

O/o DC MSME, Ministry of Micro, Small and Medium Enterprises

Govt. of India

A-Wing, 7th Floor, Nirman Bhawan, Maulana Azad Road

New Delhi-110108

Contact No. +91-674-2654700/ 9437491950

Email- msmetcab@gmail.com

COURSES / MODULE TEMPLATE

NOS /Module: INTRODUCTION TO FINITE ELEMENT ANALYSIS (FEA)

NOS /Module Code: MSME/MCCCAE/01

Outcomes:

- Analyze and understand the mechanical property of the component
- understand the concept & principle of computer aided Engineering (CAE).
- Prepare Conceptual design and finalize
- Test against standards and parameters
- Analyze using CAE software.

THEORY HOURS: -60

PRACTICAL HOURS:

THEORY MARKS: -100

PRACTICAL MARKS: 0

Unit No.	Unit Name	Unit level outcomes	Contents (chapters/topics)	TH hours	PR Marks
UNIT-I	Introduction to FEA and CAE	At the end of this Unit the student should be able to: <ul style="list-style-type: none"> • Understand Finite element method. • Work with CAE software • Describe stress strain relationship • solve static Analysis 	Introduction to Finite Element Method, Mechanical, Boundary conditions. Strain- Displacement relations. Stress-strain relations. One Dimensional Problem: Finite element modeling. Local, natural and global coordinates and shape functions. Quadratic shape functions, application of static Analysis with CAE Workbench Mechanical.	10	-
UNIT-II	Application of CAE in Engineering Problem	At the end of this Unit the student should be able to: <ul style="list-style-type: none"> • Understand the axial stress on trusses and frame • To apply proper remote boundary conditions • Knowledge of post processing • Evaluate the results of solution 	Analysis of trusses and frames: Analysis of plane truss with number of unknowns not exceeding two at each node. Analysis of frames with two translations and a rotational degree of freedom at each node. Analysis of Beams: Element stiffness matrix for two noded, two degrees of freedom per node for beam element.	15	-
UNIT-III	Finite Element Modeling	At the end of this Unit the student should be able to: <ul style="list-style-type: none"> • To use of finite element modeling with ANSYS Workbench Mechanical • Apply boundary conditions in ANSYS Workbench Mechanical • Results reading • Optimization of analysis 	Finite element modeling of two dimensional stress analysis problems with constant strain triangles and treatment of boundary conditions. Finite element modeling of Axisymmetric solids subjected of axisymmetric loading with triangular elements. Convergence requirements and geometric isotropy To Subjected To Design Optimization And Parametric Of Post	20	-

			processing Result Of ANSYS Workbench Mechanical		
UNIT-IV	Heat Transfer Analysis	<p>At the end of this Unit the student should be able to:</p> <ul style="list-style-type: none"> • To have Fundamental knowledge of NONLINER • To understand thermal behavior • To describe conduction, convection & radiation. • To use 1D & 2D element of thermal 	<p>Steady state heat transfer analysis: One dimensional analysis of a fin and two dimensional conduction analysis of thin plate. Using the NON-LINER solver to solve the Complex problem in ANSYS Workbench Mechanical Dynamic analysis: Formulation of finite element modeling of Eigen value problem for a stepped bar and beam. Evaluation of Eigen values and Eigen vectors.</p>	15	-

COURSES / MODULE TEMPLATE

NOS /Module: Create & Modify Part Model using higher end CAD Software

NOS /Module Code: MSME/MCCCAE/02

Outcomes:

- After completion of course Student should be able to
- Understand advance Computer aided design software (UNIGRAPHICS CAD) as compare to other CAD software.
- Create 2D geometric sketches by using UNIGRAPHICS CAD software.
- Develop 3D solid & surface modeling by using advanced command.
- Understand assembly constraint & develop different types of assembly design.
- Understand design generative & interactive drafting.
- Apply knowledge in create complicated modeling & creative/innovative solution.

THEORY HOURS: - 30 PRACTICAL HOURS: 90 THEORY MARKS: - PRACTICAL MARKS: 100

Unit No.	Unit Name	Unit level outcomes	Contents (chapters/topics)	PR hours	PR Marks
UNIT-I	Introduction To Unigraphics CAD	At the end of this Unit the student should be able to: <ul style="list-style-type: none"> • Understand advance computer aided design. • Understand Different types CAD s • Understand advanced features added & changed since new release software. • Analyze difference between various software as compare to UNIGRAPHICS CAD & UNIGRAPHICS CAM. • Execute the concept of hardware & software. 	Capability of CAD Software and Introduction to UNIGRAPHICS CAD & UNIGRAPHICS CAM. Description of the feature that have been added or changed since new Release software. Criteria for selection of CAD workstations, Single Design Process, Design criteria, Geometric modeling, entities, 2D & 3D Primitives. Different Types of cad software. Also comparison of various CAD Software. CAD software features. Concept of hardware & software.	10	10
UNIT-II	Unigraphics Sketcher & Solid Modeling	At the end of this Unit the student should be able to: <ul style="list-style-type: none"> • Have fundamental knowledge of UG-CAD software. • Create complicated geometry sketch • Understand Adding Geometric & Dimensional Constraint to sketches. • Perform Drawing sketches for solid models. • Understand various sketching tools. 	Introduction of Unigraphics-CAD. History of cad & UG. Technical terms related to UG-CAD. Drawing sketches for solid models. Creating sketches in the Sketch task environment & Modeling Environment. Understand Various Sketching Tool. Editing, Extruding, Revolving sketches. Adding Geometric & Dimensional Constraint to sketches.	20	30

		<ul style="list-style-type: none"> Understand & Create sketches in the Sketch task environment & Modeling Environment. 			
UNIT-III	Advance Solid Modeling	<p>At the end of this Unit the student should be able to:</p> <ul style="list-style-type: none"> Understand Working with Datum plane, Coordinate System, and datum axes. Understand Advance Modelling Tool. Apply advance editing, modifying, creating feature. To execute Boolean, extrusion termination operations. Create 3D model design. 	Working with Datum plane, Coordinate System, and datum axes. Specifying Boolean operation, Specification other Extrusion Termination option. Advance Modelling Tool like creating various types of Hole, Grooves, Slots, Chamfer, and Edge Blend. Pattern Feature Tool, Mirror Feature Tool, and Sweeping Sketches along guide curve. Creating swept, Tubes or cables, Threads, Shell Features.	20	40
UNIT-IV	Assembly Design & Surface Modeling & Drafting	<p>At the end of this Unit the student should be able to:</p> <ul style="list-style-type: none"> Understand surface modeling concept. Create surface model. Understand concept of assembly constraint. Understand concept of Drafting. Understand different types of view. Use different engineering symbols 	Various Assembly constraints. Working with bottom up and top down assembly. Design in context. Generating bill of material. Tool Develop & design in UG-CAD. Surface Modelling Feature, Family table. Generating, Editing, and Dimensioning the Drawing views. Types of Drawing View, Modifying the properties of Generated drawing view, printing tools, print, plot.	40	40

COURSES / MODULE TEMPLATE

NOS /Module: Analyses & Simulation of CAD Model using CAE Tools

NOS /Module Code: MSME/MCCCAE/03

Outcomes:

- After completion of course Student should be able to
- Able to Analyze and understand customers need
- Able to Discuss and finalize product needs

- Prepare Conceptual design and finalize
- Test against standards and parameters
- Analyze using CAE software
- interpret of output
- Apply CAE in Tool Design

THEORY HOURS: -30

PRACTICAL HOURS: 270

THEORY MARKS: -

PRACTICAL MARKS: 100

Unit No.	Unit Name	Unit level outcomes	Contents (chapters/topics)	TH hours	PR hours
UNIT-I	Introduction to ANSYS Workbench and APDL, FEA	At the end of this unit student should be able to <ul style="list-style-type: none"> • Understand Finite element method. • Work with ANSYS Graphical user interface • Describe stress strain relationship • solve static Analysis 	Introduction to Finite Element Method of solving field problems using ANSYS workbench. User Interface of ANSYS Workbench Mechanical, Boundary conditions. Strain-Displacement relations. Stress-strain relations. One Dimensional Problem: Finite element modeling. Local, natural and global coordinates and shape functions. Quadratic shape functions, application of static Analysis with ANSYS Workbench Mechanical.	02	20
UNIT-II	Analysis & Procedure	At the end of this unit student should be able to <ul style="list-style-type: none"> • Understand the axial stress on trusses and frame • To apply proper remote boundary conditions • Knowledge of post processing • Evaluate the results of solution 	Analysis of trusses and frames: Analysis of plane truss with number of unknowns not exceeding two at each node. Analysis of frames with two translations and a rotational degree of freedom at each node. Analysis of Beams: Element stiffness matrix for two noded, two degrees of freedom per node for beam element. Solving the using ANSYS Workbench Mechanical & details of result reading in post processing.	04	30
UNIT-III	Meshing, Static structure Analysis using Ansys	At the end of this unit student should be able to <ul style="list-style-type: none"> • To use of finite element modeling with ANSYS Workbench Mechanical • Apply boundary conditions in ANSYS Workbench Mechanical • Results reading • Optimization of analysis 	Finite element modeling of two dimensional stress analysis problems with constant strain triangles and treatment of boundary conditions. Finite element modeling of Axisymmetric solids subjected of axisymmetric loading with triangular elements. Convergence requirements and geometric isotropy To Subjected To Design Optimization And Parametric Of Post processing Result Of ANSYS Workbench Mechanical	04	40

UNIT-IV	Introduction to Thermal Analysis	<p>At the end of this unit student should be able to</p> <ul style="list-style-type: none"> • To have Fundamental knowledge of NON-LINER • To understand thermal behavior • To describe conduction, convection & radiation. • To use 1D & 2D element of thermal 	<p>Steady state heat transfer analysis: One dimensional analysis of a fin and two dimensional conduction analysis of thin plate. Using the NON-LINER solver to solve the Complex problem in ANSYS Workbench Mechanical Dynamic analysis: Formulation of finite element modeling of Eigen value problem for a stepped bar and beam. Evaluation of Eigen values and Eigen vectors.</p>	04	30
UNIT-V	Finite Element & Factor of Safety	<p>At the end of this unit student should be able to</p> <ul style="list-style-type: none"> • To knowledge basic of 3D element • To understand the factor of safety concept • To predict the behavior of analysis • To use ANSYS Workbench Mechanical simulation tools 	<p>Finite element formulation of three dimensional problems in stress analysis. And design Optimization over the safety factor to predict the failure of design and stress value in ANSYS Workbench Mechanical</p>	04	30
UNIT-VI	Transient Structure Analysis	<p>At the end of this unit student should be able to</p> <ul style="list-style-type: none"> • Understand the dynamics of Analysis • Describe the steady state conditions over the analysis • To have validation of analysis • To execute the knowledge in tooling. 	<p>Finite Element formulation of Dynamic transient analysis result validation and supporting data file to simulate the real world problems into ANSYS Workbench Mechanical & Application of computer aided engineering in tool and die engineering.</p>	04	30
UNIT-VII	Introduction to Hypermesh	<p>At the end of this unit student should be able to</p> <ul style="list-style-type: none"> • Understand the Geometry creation • Understand the editing of geometry. • Understand the Manual Auto Meshing. 	<p>Node, Line, Surface, Primitives, Solid, Point edit, Node edit, Lines edit, Surface edit, Edge edit, De feature, Mid surface, Different manual meshing options, Auto meshing, 3-D meshing</p>	04	30
UNIT-VIII	Meshing, Static	<p>At the end of this unit student should be able to</p>	<p>Check elements, Quality criteria, Save and rectify elements, Knowledge of different quality parameters, Different</p>	04	60

	<p>structure Analysis using Ansys</p>	<ul style="list-style-type: none"> ● Understand the Quality Criteria ● Describe the Optristruct & Hyper view. ● Understand the Capabilities of hyper form. ● Understand the one step analysis. 	<p>collectors, Defining load and constraints, Running analysis, Viewing results in Hyper view, Updating collectors, Capabilities of hyper form, Introduction to One Step and Incremental Analysis, Performing One Step Analysis</p>		
--	--	--	---	--	--

COURSES / MODULE TEMPLATE

NOS /Module: Employability Skills

NOS /Module Code: MSME/ES/02

THEORY HOURS: 60

PRACTICAL HOURS: -

THEORY MARKS: 100

PRACTICAL MARKS: -

Refer Standard Curriculum developed by NCVET. (60-hours-MC-Employability-Skills)