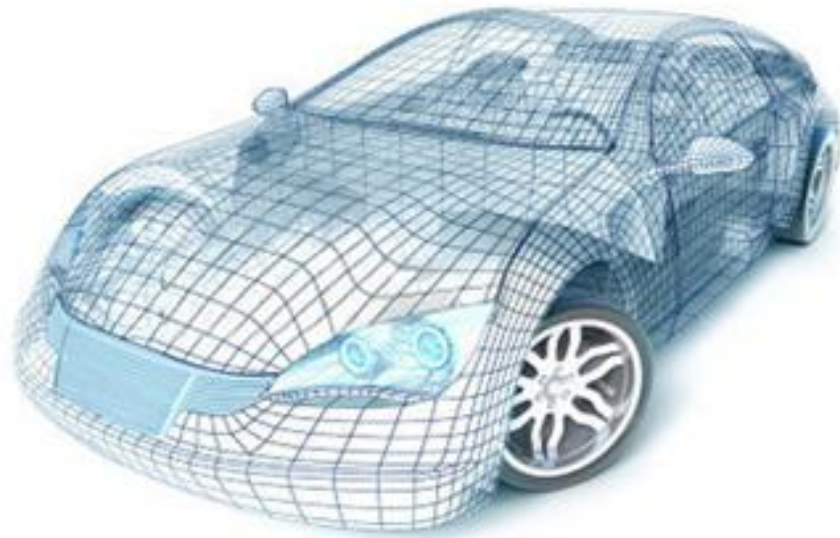


BROCHURE



CAD | CAM | CAE
CAD | CAM | CAE

PG Diploma | Professional Course | Certification Course

Company Profile

G2G Innovation LLP is a corporate training partner of Engineering Services in Pune and offers Industrial Oriented Training in CAD & CAE for Mechanical, Automobile, Aeronautical & Production Engineering Students. Training courses are designed by industrial experts keeping in mind the industrial requirement to enhance the design and analysis skill of engineers and to make them more employable by providing industry oriented domain knowledge in design & analysis fields & to help them start a career in the right direction. Training is embedded with 4-7 months of industrial project from industries across India, Canada, Germany and USA giving the insight of CAD & CAE. Engineers undergoing training at G2G Innovation Training Center are preferred by most of the Automotive Industries, Heavy Industries, Medical Devices Industries & Home Appliances Industries for employment.

We have trained & placed 350+ students in core industries



PG Diploma in Automotive Design

Sr. No	Course	Duration	Schedule	Time	Domain Training/Project Work	Placement
1	PG Diploma in Automotive Design	9 Months (1500 Hr)	2 Months: Training on Software 7 Months: Domain Training- Plastic & Sheetmetal & Real Time Industrial and Live Project	6 Hrs per Day 6 Days a Week	Projects 1. Design of Vehicle Door Trim 2. Design of Vehicle Instrumental Panel 3. Design of Passenger Car Seating 4. Design of Car BIW	100% Job Assistance

Professional Course in Automotive Design

Sr. No	Course	Duration	Schedule	Time	Domain Training/Project Work	Placement
1	Professional Course in Automotive Design	6 Months (1000 Hr)	1 Months: Training on Software 5 Months: Domain Training- Plastic & Sheetmetal & Real Time Industrial and Live Project	6 Hrs per Day 6 Days a Week	Projects 1. Design of Vehicle Door Trim 2. Design of Vehicle Instrumental Panel 3. Design of Passenger Car Seating 4. Design of Car BIW	100% Job Assistance

PG Diploma in Automotive FEA Simulation

Sr. No	Course	Duration	Schedule	Time	Domain Training/Project Work	Placement
1	PG Diploma in Automotive FEA Simulation	9 Months (1500 Hr)	2 Months: Training on Software 7 Months: Domain Training- Plastic & Sheetmetal Real Time Industrial and Live Project	6 Hrs per Day 6 Days a Week	Projects 1. Meshing of Door Trim for Crash Analysis 2. Meshing of Vehicle Instrumental Panel for Crash Analysis 3. Meshing of Passenger Seating for Crash Analysis 4. Meshing of Car BIW 5. Structural, Dynamic, Thermal, Contact Analysis of Vehicle	100% Job Assistance

Professional Course in Automotive Design

Sr. No	Course	Duration	Schedule	Time	Domain Training/Project Work	Placement
1	Professional Course in Automotive FEA Simulation	6 Months (1000 Hr)	2 Months: Training on Software 4 Months: Domain Training- Plastic & Sheetmetal Real Time Industrial and Live Project # Instrumental Panel # Door Trim # Seating # BIW	6 Hrs per Day 6 Days a Week	Projects 1. Meshing of Door Trim for Crash Analysis 2. Meshing of Vehicle Instrumental Panel for Crash Analysis 3. Meshing of Passenger Seating for Crash Analysis 4. Meshing of Car BIW	100% Job Assistance

Certification Courses in CAD & CAE

Sr. No	Course	Duration	Schedule	Time	Domain Training/ Project Work	Placement
1	CAD & CAE	6 Months (650 Hr)	1 Month CAD 1 1 Month CAD 2 1 Month Pre-Processor 1 1 Month Pre- Processor 2 1 Month Solver 1 Month Mini Project	4 Hrs per Day 6 Days a Week	Yes	Job Assistance

Salient Features

- Hands-on Experience on Live Project Work from leading Industries
- Instructors with more than 10+ years of Industrial Experience
- Subject Matter Domain Training (Plastic, Sheet Metal, Casting, Foam, Automotive Domain - BIW, Seating, Interior Door Trim Etc.
- Globally Recognized Certification
- State of the Art lab Facilities
- Specialized Training on Software Skill development Like Personality development, presentation skills, assertiveness development, GD&T,
- Opportunity to work on over 100+ Live Project
- 100% Placement Assistance
- Students undergoing Professional Course can compete candidates with 1.5+ Years of experience.

SYLLABUS

Course ID: CAD

Session 1

Introduction to CAD Parametric

Introduction to CAD Parametric, Feature-Based Nature, Bidirectional Associative Property, Parametric Nature, System Requirements, Getting Started with CAD Parametric, Important Terms and Definitions, File Menu Options, Managing Files, Menu Manager, Model Tree, Understanding the Functions of the Mouse Buttons, Ribbon, Toolbars, Navigator, CAD Parametric Browser, Appearance Gallery, Rendering in CAD Parametric, Colour Scheme Used in this Book

Session 2

Sketcher

The Sketch Mode, Working with the Sketch Mode, Invoking the Sketch Mode, The Sketcher Environment, Working with a Sketch in the Sketch Mode, Drawing a Sketch Using tools available in the Sketch Tab, Placing a Point, Drawing a Line, Centreline, Geometry Centreline, Rectangle, Circle, Ellipse, Arc, Sketch, Converting a Weak Dimension into a Strong Dimension, Dimensioning a Sketch Using the Normal Tool, Dimensioning the Basic Sketched Entities, Linear Dimensioning of a Line, Angular Dimensioning of an Arc, Diameter Dimensioning, Radial Dimensioning, Dimensioning Revolved Sections, Working with Constraints, Types of Constraints, Disabling Constraints, Modifying the Dimensions of a Sketch, Using the Modify Button, Modifying a Dimension by Double-Clicking on it, Modifying Dimensions Dynamically, Resolve Sketch Dialog Box, Deleting the Sketched Entities, Trimming the Sketched Entities, Mirroring the Sketched Entities, Inserting Standard/User-Defined Sketches, Drawing Display Options, Dimensioning the Sketch, Dimensioning a Sketch Using the Baseline Tool, Replacing the Dimensions of a Sketch Using the Replace Tool, Creating Fillets, Creating Circular Fillets, Creating Elliptical Fillets, Creating a Reference Coordinate System, Working with Splines, Creating a Spline, Dimensioning of Splines, Modifying a Spline, Writing Text in the Sketcher Environment, Rotating and Resizing Entities, Importing 2D, Drawings in the Sketch Mode.

Session 3

Creating Base Features

Creating Base Features, Invoking the Part Mode, the Default Datum Planes, Creating a Protrusion, Extruding a Sketch, Revolving a Sketch, Understanding the Orientation of Datum Planes, Parent-Child Relationship, Implicit Relationship, Explicit Relationship, Nesting of Sketches

Datums

Datums, Default Datum Planes, Need for Datums in Modeling, Selection Method in CAD Parametric, Datum Options, Datum Planes, Creating Datum Planes, Datum Planes Created On-The-Fly, Datum Axes, Datum Points, Creating Cuts, Removing Material by Using the Extrude Tool, Removing Material by Using the Revolve Tool

Session 4

Options Aiding Construction of Part - I

Options Aiding Construction of Parts, Creating Holes, Hole Dashboard, Important Points to Remember While Creating a Hole, Creating Rounds, Creating Basic Rounds, Creating a Variable Radius Round, Points to Remember While Creating Rounds, Creating Chamfers, Corner Chamfer, Edge Chamfer, Understanding Ribs, Creating Trajectory Ribs, Creating Profile Ribs, Editing Features of a Model, Editing Definition or Redefining Features, Reordering Features, Rerouting Features, Suppressing Features, Deleting Features, Modifying Features

Options Aiding Construction of Part - II

Introduction, Creating Feature Patterns, Uses of patterns, Creating Patterns, Deleting a Pattern, Copying Features, New Refs, Same Refs, Mirror, Move, Select, Mirroring a Geometry, Creating a Section of a Solid Model.

Session 5

Advance Modeling Tools-I

Other Protrusion Options, Sweep Features, Creating Sweep Protrusions, Aligning a Sketched, Trajectory to an Existing Geometry, Creating a Thin Sweep Protrusion, Creating a Sweep Cut, Blend Features, Parallel Blend, Rotational Blend, General Blend, Using Blend Vertex, Shell Feature, Creating a Constant Thickness Shell, Creating a Variable Thickness Shell 8

Datum Curves, Creating a Datum Curve by Using the Curve Button, Creating a Datum Curve by Sketching, Creating a Curve by Using the Intersect Option, Creating a Curve by Using the Project Option, Creating a Curve by Using the Wrap Option, Creating Draft Features

Advance Modeling Tools-II

Advanced Feature Creation Tools, Variable Section Sweep Using the Sweep Option, Swept Blend, Helical Sweep, Blend Section to Surfaces, Blend Between Surfaces

Advance Modeling Tools-III

Advanced Modeling Tools, Toroidal Bend, Spinal Bend, Warp, Transform Tool, Warp Tool, Spine Tool, Stretch Tool, Bend Tool, Twist Tool, Sculpt Tool

Session 6

Assembly Modeling

Assembly Modeling, Important Terms Related to the Assembly Mode, Top-down Approach Bottom-up Approach, Placement Constraints, Package, Creating Top-down Assemblies Creating Components in the Assembly Mode, Creating Bottom-up Assemblies, Inserting Components in an Assembly, Assembling Components, Displaying Components in a Separate Window, Displaying Components in the Same Window, 3D Dragger, Applying Constraints Status Area, Placement Tab, Move Tab, Packaging Components, Creating Simplified Representations, Redefining the Components of an Assembly, Reordering Components

Suppressing/Resuming Components, Replacing Assembling Repeated Copies of a Component, Modifying the Components of an Assembly, Modifying Dimensions of a Feature of a Component

Redefining a Feature of a Component, Creating the Exploded State, References Tab Offset Tab, Explode Line Tab, The Bill of Materials, Global Interference, Pairs Clearance

Session 7

Generating, Editing and Modifying the Drawing Views

The Drawing Mode, Generating Drawing Views, Generating the General View, Generating the Projection View, Generating the Detailed View, Generating the Auxiliary View, Generating the Revolved Section View, Generating the Copy and Align View, Generating the 3D Cross-Section View, Editing the Drawing Views, Moving the Drawing View, Erasing the Drawing View

Deleting the Drawing View, Adding New Parts or Assemblies to the Current Drawing

Modifying the Drawing Views, Changing the View Type, Changing the View Scale

Reorienting the Views, Modifying the Cross-sections, Modifying Boundaries of Views, Adding or Removing the Cross-section Arrows, Modifying the Perspective Views, Modifying Other Parameters, Editing the Cross-section Hatching

Dimensioning the Drawing Views

Dimensioning the Drawing Views, Show Model Annotations Dialog Box, Adding Notes to the Drawing, Adding Tolerances in the Drawing Views, Dimensional Tolerances, Geometric Tolerances, Editing the Geometric Tolerances, Adding Balloons to the Assembly Views, Adding Reference Datums to the Drawing Views, Modifying and Editing Dimensions, Modifying the Dimensions Using the Dimension Properties Dialog Box, Modifying the Drawing Items Using the Shortcut Menu Cleaning Up the Dimensions

Other Drawing Options

Sketching in the Drawing Mode, Modifying the Sketched Entities, User-Defined Drawing Formats

Retrieving the User-Defined Formats in the Drawings, Adding and Removing Sheets in the Drawing

Creating Tables in the Drawing Mode, Generating the BOM and Balloons in Drawings

Session 8

Surface Modeling

Surface Modeling, Creating Surfaces in CAD Parametric, Creating an Extruded Surface, Creating a Revolved Surface, Creating a Sweep Surface, Creating a Blended Surface, Creating a Swept Blend Surface, Creating a Helical Sweep Surface, Creating a Surface by Blending the Boundaries

Creating a Variable Section Sweep Surface Using the Sweep Tool, Creating Surfaces the Using the Style Environment of CAD Parametric, Style Dashboard, Surface Editing Tools, Mirroring the Surfaces, Merging the Surfaces, Trimming the Surfaces, Creating the Fill Surfaces, Creating the Intersect Curves, Creating the Offset Surfaces, Adding Thickness to a Surface, Converting a Surface into a Solid, Creating a Round at the Vertex of a Surface, Freestyle modelling environment, Freestyle Dashboard

Working with Sheet Metals Components

Introduction to Sheet metal, Invoking the Sheet metal Mode, Introduction to Sheet metal Walls
Creating the Planar Wall, Creating the Unattached Revolve Wall, Creating the Unattached Blend Wall, Creating the Unattached Offset Wall, Creating Reliefs in Sheet metal Components, Creating a Flat Wall, Creating a Twist Wall, Creating an Extend Wall, Creating a Flange Wall, Creating the Bend Feature, Creating the Unbend Feature, Creating the Bend Back, Conversion to Sheet metal Part, Creating Cuts in the Sheet metal Components

Course ID: CAE – Pre Processor

A) Fundamentals of FEM

- ✓ Introduction to FEM and fundamentals of strength of materials
- ✓ Finite element formulation of 1D element
- ✓ Finite element formulation of 2D & 3D element
- ✓ Introduction to Finite element Method.
- ✓ FEA and Hypermesh
- ✓ Finite element theory & FEM Basics
- ✓ Introduction to CAE Software
- ✓ Types of Analysis

B) Introduction to CAE Pre-Processor

- ✓ Software Window (GUI)
- ✓ Graphics area, Header, Main menu, Macros menu, Permanent menu, Toggles
- ✓ Opening a database file : Displaying element
- ✓ File panel , Collector panel
- ✓ Creating Geometry : Nodes, Lines, Surface, Editing
- ✓ Geometry editing

C) Geometry

- ✓ Nodes, Nodes Edit, Creating and Deleting temporary nodes, calculating distance between two nodes, creating nodes on line, nodes between two nodes, creating points.
- ✓ Creating lines using different methods like, point, linear nodes, creating circle, two point circles, three point circles.
- ✓ Creating surfaces using drag, nodes, spline, skin, from FE model, editing surface, trim with nodes, trim with line, trim with plane, offset.
- ✓ Creating solid with nodes, line and surfaces, performing Boolean operation like union, intersection, cut and remove.
- ✓ Hands on features of quick edit panel like splitting of surface with line and nodes, creating washer , splitting washers, toggling edge, adding and removing points, edge editing, point edit.

- ✓ Defeaturing like suppressing holes, removing surface fillet, edge fillet
- ✓ Mid-Surface extraction of Sheet metal components and plastics using surface pair and auto-mid surface, checking extended surface, replacing edge

D) Meshing

i) 1D

- ✓ Line mesh using lines and nodes, assigning properties and material, dealing with element configuration.
- ✓ Creating 1D elements like bars, beams, rods etc, , creating shell and solid section using hyperbeam and optistruct, editing section.
- ✓ Creating connections using rigids(RBE2 and RBE3) with independent and dependent nodes, weld joints, bolts, .
- ✓ Editing quad, tria, tetra, hexa element using create, combine and split features, replacing nodes, order change of elements, checking element type.

ii) 2D Meshing

- ✓ 2D meshing using ruled, spline, sking, drag, spin, and element offset,
- ✓ Use of Automesh panel, size and bias meshing, QI optimize meshing, selecting mesh type, like quad, tria, mixed etc, checking the flow alignment order of 2D element , Meshing with interactive and automatic options.
- ✓ Performing element quality criterion like jacobian, aspect ratio, skewness , min and max angle, min length, T-connection, equivalence, checking quality index, setting up-quality parameters, use of clean-up tools,

iii) 3D Meshing

- ✓ Finding type meshing required (Hex or Tet), Meshing with hexa or tet element using, solid map, linear solid, solid mesh, drag , spin, line drag. Performing tet mesh using volume tetra mesh, closed volume mesh, surface mesh, 2d to 3d tetra mesh, tetra remesh. Checking solid element quality like tet collapse, min length, warpage, order checking, split, replace.

E) Introduction to Tool Page

- ✓ Creating components, Organize element, components surfaces, solids etc, to the components, assigning color to individual components, renaming, reordering, finding components, masking for hide and show, deleting, use of translate, rotate, scale reflect project for meshing of components, checking elements, finding duplicates, connectivity, saving failed elements, checking edges, faces, normals, penetrations, counting, nodes elements, properties, components, solid, surface, boundary conditions, calculating mass.

F) Introduction to Analysis and Deck Preparation

- ✓ Checking load types, applying loads like force, pressure, moment, torque, temperature, flux extra on nodes and element option, applying loads using components, surface and lines, with single point constraints, Constraining with all 6 degree of freedom and other free dof.
- ✓ Dealing with control cards, creating load steps with SPC and MPC, creating isotropic, orthotropic, anisotropic material, assigning materials to various components, creating properties, assigning thickness to mid-surface.
- ✓ Deck preparation for linear static analysis, free-free analysis, buckling analysis solver like, optistruct, Nastran, abaqus, Solver etc.

G) Exporting FE data to various solvers like Nastran, ABAQUS, SOLVER, LS Dyna

Course ID: CAE – Solver

1: INTRODUCTION TO FEA AND SOLVER

Introduction to FEA

General Working of FEA, Nodes, Elements, and Element Shapes, General Procedure of Conducting Finite Element Analysis, FEA through SOLVER, 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.

Types of Analysis Structural Analysis ,Thermal Analysis, Fluid Flow Analysis, Electromagnetic Field Analysis, Coupled Field Analysis.

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 SOLVER

System Requirements,

Getting Started with SOLVER: Interactive Mode ,Batch Mode, Starting a New File Using the SOLVER Product Launcher window

SOLVER Output, Window,

SOLVER Metaphysics Utility Menu Window (SOLVER Session)

Utility Menu, Main Menu, Graphics Area, Standard Toolba,r SOLVER Command Prompt ,Command Window Icon, Raise Hidden Icon, Reset Picking, Contact Manage,r SOLVER Toolbar ,Model Control Toolbar, User Prompt Information, Current Settings,

Setting the Analysis Preferences

Units in SOLVER

Other Important Terms Related to SOLVER: 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, SOLVER 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 SOLVER

Self-Evaluation Test

2: BASIC SOLID MODELING

Solid Modeling in SOLVER :Solid Modeling and Direct Generation

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

Considerations before Creating a Model for Analysis - Details Required Symmetry

Creating Geometric Entities- Creating Lines, Creating Arcs, Creating B-Spines, 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 .

Coordinate Systems in SOLVER - 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

3: 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

4: FINITE ELEMENT MODELING (FEM) – I

An Overview of the Finite Element Modeling

Element Attributes- Element Types, Reasons Why SOLVER 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. Element Attributes Table

5: 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, Defining the Meshing Type.

Meshing the Model, Refining the Mesh Locally, Extruding the Mesh, 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, Plotting Pyramid Elements

Meshing the Beam with Orientation Nodes- Creating the Beam Mesh with Orientation Nodes ,Creating the Beam Mesh with Two Orientation Nodes

Improving the Tetrahedral Element Meshes - Improving Tetrahedral Meshed Volumes by Using Volumes, Improving Tetrahedral Meshed Volumes by Using Detached Elements

Some Additional Tips while Meshing the Model

Applying Loads - The Nodal Coordinate System, Loads in Different Disciplines, Types of Loads in SOLVER, 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

6: SOLUTION AND POSTPROCESSOR

Solution - Defining the New Analysis Type, Restarting the Analysis, Setting Solution Controls, Setting Analysis Options, Solving the Analysis Problem.

Post processing the Result: POST1 (General Postprocessor), POST26 (Time-history Postprocessor), Result Coordinate System (RSYS), Displaying the Deformed Shape of the Model, Displaying the Minimum and Maximum Stresses, Listing Reaction Forces, Listing Stress Values at each Node, Query Picking, Path Operations, Load Case Combinations

7: STATIC STRUCTURAL ANALYSIS

Effect of self-weight on a cantilever beam, Analysis of a bicycle handle, Analysis of a stud (pin) Analysis of a master

8: ADVANCED STRUCTURAL ANALYSIS (DYNAMIC AND NONLINEAR)

Advanced Structural Analysis

Dynamic Analysis

Performing the Modal Analysis - Specifying the Analysis Type, Analysis Options, and Applying Loads, Obtaining the Solution, Reviewing Results

Performing the Harmonic Analysis - Specifying the Analysis Type, Analysis Options, and Applying Loads, Obtaining the Solution, Reviewing Results

Performing the Transient Analysis - Specifying the Analysis Type, Analysis Options, and Applying Loads, Obtaining the Solution, Reviewing Results

Nonlinear Analysis - Geometric Nonlinearity, Material Nonlinearity, Boundary Nonlinearity (Changing Status), Performing the Nonlinear Analysis, Specifying the Analysis Type, Setting Solution Controls, and Applying Loads, Obtaining the Solution

9: ADVANCED STRUCTURAL ANALYSIS

Steel tubes and springs structure, Modal analysis of an airplane wing, Nonlinear analysis (material nonlinearity)

10: THERMAL ANALYSIS

Thermal Analysis

Important Terms Used in Thermal Analysis - Heat Transfer Modes, Thermal Gradient Thermal Flux, Bulk Temperature, Film Coefficient, Emissivity, Stefan–Boltzmann Constant, Thermal Conductivity, Specific Heat.

Types of Thermal Analysis Steady-State Thermal Analysis, Transient Thermal Analysis

Performing Steady-State Thermal Analysis- Setting the Analysis Preference, Creating or Importing a Solid Model, Defining Element Attributes, Meshing the Solid Model, Specifying the Analysis Type, Analysis Options, and Applying Loads, Solving the Analysis Problem, Post processing Results, Performing Transient Thermal Analysis, Specifying the Analysis Type and Setting Solution Controls

11: GENERATING THE REPORT OF ANALYSIS

Starting the SOLVER Report Generator, Capturing Images for the Report, Capturing Animations for the Report, Capturing Data Tables for the Report, Capturing Lists for the Report, Compiling the Report, Changing the Default Settings of the SOLVER Report Generator

Error Estimation in Solution- Percentage Error in Energy Norm (SEPC), Element Energy Error (SERR), Element Stress Deviations (SDSG), Maximum and Minimum Stress Bounds (SMXB and SMNB)

What make us Different?

G2G Innovation	Other Training Institute
Syllabus as per Ind. requirement Designed by Industrial Experts covering all the aspects of Design and Analysis and Real Time Industrial Projects	Syllabus mostly covering only Tool Commands
3-6 Months Training on one Tool	15-45 Days Training on one Tool
4-7 Months Domain Training and Project Work from industries	No Domain Training and Project Work
Perfection and Expertise	No Perfection and Expertise
No. of students per batch 10.	No. of students per batch 50-100
Skill Sets meets Industrial Requirement	Skill Sets does not meet Industrial Requirement
Only Two Batches per year	Number of Batches – Mass Production
Job Assistance even after completion of training	Job Assistance till training
We have technical association with Engineering Services in Pune for Domain Training and Project Work	No Domain Training and Project Work
Experts with more than 10 Years of Experience in Domain and Project	No Domain Training and Project Work
Grooming Session for Interview	No



Address

G2G Innovation LP

03, Saket Apartment, Right Bhusari Colony,

Behind Lohia Jain IT Park , Kothrud,

Pune- 411038

Maharashtra, India

Phone: 09975700330 / 8805025992

Emial id: training@g2ginnovation.com

Time: 8.00 Am to 8.00 Pm (Monday to Saturday)

Join G2G Innovation and be a Winner

