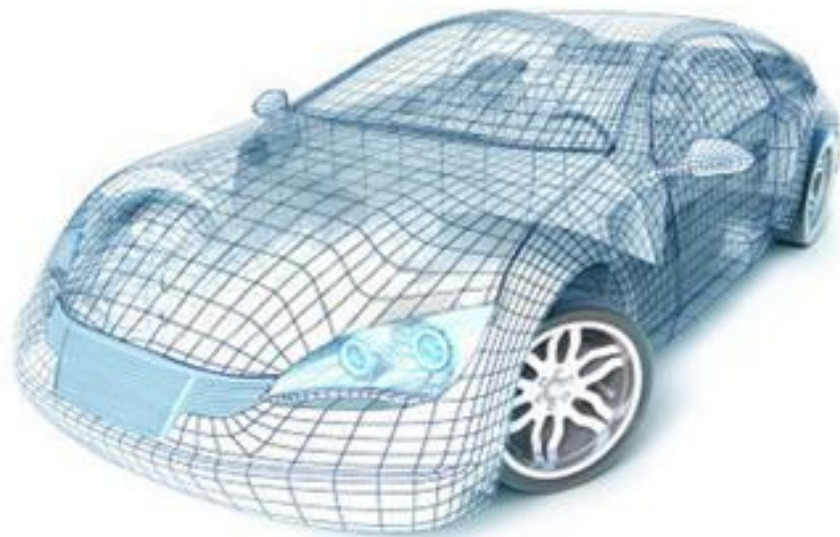


BROCHURE PROFESSIONAL COURSE



CAD | CAM | CAE
CAD | CAM | CAE

Company Profile

g2G Innovation 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 3 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 250+ students in core industries



Professional CAD & CAE Course

Batch Size Limited to 10 Students

Sr. No	Course	Duration	Schedule	Time	Domain Training/Project Work	Placement
1	CAD	6 Months	3 Months: Training on Software 3 Months: Domain Training & Project	6 Hrs per Day / 6 Days a Week	Yes	100% Job Assistance
2	CAE	6 Months	3 Months: Training on Software 3 Months: Domain Training & Project	6 Hrs per Day / 6 Days a Week	Yes	100% Job Assistance

Certification Courses

Sr. No	Course	Duration	Schedule	Time	Domain Training/Project Work	Placement
1	CAD	1.5 Months	1 Months: Training on Software 0.5 Months: Mini Project	2 Hrs per Day / 6 Days a Week or 6 Hrs on Saturday & Sunday	Introduction of Domain	Job Assistance
2	CAE	1.5 Months	1 Months: Training on Software 0.5 Months: Mini Project	2 Hrs per Day / 6 Days a Week or 6 Hrs on Saturday & Sunday	Introduction of Domain	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: CAE

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

- ✓ Hypermesh window
- ✓ 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, Ansys etc.

G) Exporting FE data to various solvers like Nastran, ABAQUS, ANSYS

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

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

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	Syllabus mostly covering only Tool Commands
3 Months Training on one Tool	15-45 Days Training on one Tool
3 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



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Join g2G Innovation and be a Winner

