

COURSE CODE	COURSE TITLE	L	T	P	C
1152ME129	Advanced 3D Modelling using Unigraphics NX	2	0	2	3

Course Category: Program Elective (for Mechanical, Automobile and Aeronautical Branches)

Course Content:

UNIT I INTRODUCTION TO UNIGRAPHICS

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Introduction to Unigraphics NX, About NX Gateway, Getting Started, NX Graphical User Interface - Title bar, Menu bar, Toolbar, Radial toolbar, Selection bar, Cue and status line, Dialog rail, Resource bar, Navigators, HD3D tools, Integrated browser, Palettes, Roles, Full screen, View orientation- trimetric, isometric, View commands, Rotate, Pan, Zoom in/out, Quick pick, Quick pick, categories, Coordinate system- absolute coordinate system, WCS, Absolute coordinate, Work coordinate system. View triad, Multiple graphics window, Information window, Keyboard accelerators, Dialog box File management - Creating new files, Opening files and Saving files.

UNIT II SKETCHER

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Creating Sketches - Profile, Line, Arc, Circle, Fillet, Chamfer, Rectangle, Polygon, Studio pline, Fit spline, Ellipse, Conic Editing sketches - Quick trim, Quick extend, Make corner, Offset curve, Pattern curve, Mirror curve, Intersection point, Derived lines Constraints - Geometric constraints, Auto constraint, Inferred constraint, Dimensional constraints, Auto dimension, Animate dimension, Continuous auto dimension. Basic terminologies - Feature, Body, Solid body, Sheet, Face, Section curves, Guide curves. Creating Primitives - Block, Cylinder, Cone, Sphere, Boss, Pocket, Emboss, Slot, Groove. Feature modeling commands- Creating Extrude features, Creating Revolve features.

UNIT III PART MODELING

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Datums - Creating Datum planes, Axis, Point. Creating Sweep Features- Swept, Sweep along guide, variable sweep, Creating Tube feature Hole - General hole, Drill size holes, Screw clearance holes, Threaded holes Dart, Thread, Shell, Draft, Draft body, Scale Creating Blend and Chamfer. Instance feature - Rectangular array, Circular array, Pattern face, Mirror feature, Mirror body Feature Operations - To Divide face, Trim body, Split body, Boolean commands, User defined feature, Creating Feature group, Layer settings, To measure distance between geometries, To measure angle between geometries, To measure bodies and face geometries, To find geometric properties. Synchronous Modeling.

UNIT IV ASSEMBLY

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Introduction to Assembly Modeling, Assembly approaches Assembly constrains - Angle, Bond, Centre, Concentric, Distance, Fit, Parallel, Perpendicular, Touch align Component array - Linear array, Circular array, Feature instance array Moving a component, Replacing component, Repositioning component, Mirroring assembly. Creating a New Component, Creating a new parent, Assembly clearance, creating

exploded views, Assembly sequencing with motion. Creating deformable parts, finding degrees of freedom. Assembly envelopes.

UNIT V DRAFTING AND DETAILING

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Creating the Sheets and Editing the Sheets, Standard settings. Creating drawing views- Base view, Drawing view, projected view. Section view- Simple section, Stepped section, Half section, Revolved section, Folded section, Unfolded section, Pictorial section, Half pictorial section, Break out section Detail view Creating Broken view Applying dimensions- Inferred Dimension, Horizontal Dimension, Vertical Dimension, Parallel Dimension, Perpendicular dimension, Angular dimension, Cylindrical Dimension, Hole dimension, Diameter Dimension, Chamfer Dimension, Radius or Radius of Curvature Dimension, Radius to Centre, Folded Radius, Thickness Dimension, Arc Length, Horizontal Chain Dimension, Vertical Chain Dimension, Horizontal Baseline Dimension, Vertical Baseline Dimension, Ordinate Dimension. Creating Annotations, Datum feature, symbols, feature control frame, placing datum target symbol. Creating the Centerline, Axis, Hatch and fill options Creating Table and Part list.

TOTAL: 30+30=60 PERIODS