

Kadkraft Tech Centre

Courses Information

NX/Solid Edge /AutoCAD/KISSsoft/Simufact

Product	Package	Course Details	Minimum Training Duration (In Hours)
NX CAD	CAD-Basic	TR10051- NX- Essentials for NX Designers TR10028- NX-Sketcher Fundamentals TR10100- NX- Drafting Essentials	80
	CAD-Adv	TR10045 NX-Synchronous Modeling Fundamentals TR10043 NX- Synchronous Modeling & Parametric Design	40
	CAD-Adv	TR13150 -NX- Basic Design for Exp CAD Users	40
	CAD-Adv	TR16020 -NX- Sheet Metal	16
NX CAM	CAM-Basic	TR11021 NX- Manufacturing Fundamentals TR11065 NX- Fixed Axis Techniques	48
Solid Edge	CAD-Basic	TR01413 -Solid Edge Fundamentals TR01415 -Solid Edge - Fundamentals with ST	100
	CAD-Adv	TR01416 -Solid Edge Assembly(Advanced)	30
	CAD-Adv	TR01418 -Solid Edge : Surfacing	20
	CAD-Adv	TR01419 -Solid Edge : Sheet Metal	20
AutoCAD	CAD-Basic	AutoCAD 2D only	40
KISSsoft	KISSsoft-Basic	Spur/Helical/Bevel Gear design Shaft and bearing calculations	24
	KISSsys	System level gear box design	30
Simufact	Simufact	Hot Forging and Cold Forming Simulation	40

Course Curriculum

Course Name	<ul style="list-style-type: none">• NX-Sketcher Fundamentals• NX CAD Essentials for NX Designers• NX- Drafting Essentials
Objectives of the Course	<ul style="list-style-type: none">• Sketching in Modeling and in the Sketch task environment• Creating sketches• Sketch constraints• Additional sketch techniques• Sketch on path• User interface• Holes• Coordinate systems• Create expressions• Introduction to sketching• Shell• Datums• Instance arrays• Modeling theory – based on extrude, revolve• Blending and chamfers• Part structure and edits• Basic assembly modeling• Intermediate sketching topics• Assembly constraints• Trim body• Master model concept• Extrude options, including draft and limits• Creating and editing drawings and annotation• Create and edit drawings• Create and edit orthographic, section, and detail views• Associative view scales to model parameters• Create and edit drafting symbols, geometric tolerancing symbols, dimensions, notes and labels• Create associative user-defined view boundaries• Create associative notes and labels

Course Name	<ul style="list-style-type: none"> • NX-Synchronous Modeling Fundamentals • NX- Synchronous Modeling & Parametric Design
Objectives of the Course	<ul style="list-style-type: none"> • Modeling modes and switching between them • Working in history-free mode • Synchronous modeling constraints (make tangent, make parallel, etc.) • Dimension commands • Reuse commands (copy/paste faces) • Pattern/Replace/Delete/Pull Face • Offset region • Documenting design intent (Layers, Feature sets, Product Interfaces) • Editing parametric models (Replace features, Suppression, model updates) • Associative curve operations (Project, Join, Intersect, Wrap/Unwrap, Text) • General pockets and pads (General, Emboss, Offset) • Blending techniques (Overflow, Edge options, Face blends and options) • Design optimization (Optimization study, options, algorithms) • Synchronous modeling (History and History Free Mode) • Design intent and model construction

Course Name	NX- Basic Design for Exp CAD Users
Objectives of the Course	<ul style="list-style-type: none"> • NX part files • The NX user interface • Coordinate systems • Sketch task environment • Datum features • Swept features • Part structure • Using sketches • Trim body • Swept feature options • Hole features • Expressions • Associative copies • Face and Edge operations • Introduction to assemblies • Adding and constraining components • Reference Sets • Top-down assembly modeling • Interpart geometry and references • Component arrays • Revise and replace components • Introduction to drafting

	<ul style="list-style-type: none"> • Editing models • Practice projects • Expression operators • System topics • Legacy mating conditions • Typical sheet metal workflow • Preferences and defaults • Base feature tab, contour flange and lofted flange creation • Flange • Convert to sheetmetal and flat solid • Bending sheet metal • Sheet metal corners • Edge rip/Jog/Bead • Sheet metal cutouts • Punch features dimple, drawn cutout and louver creation
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Course Name	<ul style="list-style-type: none"> • NX- Manufacturing Fundamentals • NX- Fixed Axis Techniques
Objectives of the Course	<ul style="list-style-type: none"> • Introduction and Overview • Part analysis for manufacturing • User Interface • Operation Navigator • Machine Coordinate System • Tooling • Visualization/Verification • Post Processing/Shop Documentation • Planar/Cavity Milling • Drilling • Fixed Contour Area Milling • Face Milling • Text Engraving • Wave geometry linker • Advanced cavity milling • Plunge milling • Z level machining • NC assistant • High speed machining concepts • Fixed contour applications • Streamline drive method

Course Name	SE – Fundamentals Solid Edge Fundamentals with Synchronous Technology
Objectives of the Course	<ul style="list-style-type: none"> ▪ Use the Solid Edge interface. ▪ Find and open Solid Edge documents. ▪ Create and save Solid Edge documents. ▪ Access and use Solid Edge Help. ▪ Use Solid Edge learning tools.
Program Modules	<ul style="list-style-type: none"> ▪ <i>Getting started in Solid Edge</i> ▪ <i>Feature modeling overview</i> ▪ <i>Reference planes</i> ▪ <i>Profiles and sketches</i> ▪ <i>Base features</i> ▪ <i>Profile-based features</i> ▪ <i>Treatment features</i> ▪ <i>Reusing features</i> ▪ <i>Constructing base features</i> ▪ <i>Moving and rotating faces</i> ▪ <i>Working with geometric relationships</i> ▪ <i>Constructing treatment features</i> ▪ <i>Constructing functional features</i> ▪ <i>Building assemblies</i> ▪ <i>Manipulating assemblies</i> ▪ <i>Direct editing</i> ▪ <i>Creating drawings of 3-d models</i> ▪ <i>Dimensions and annotations</i>

Course Name	Solid Edge Assembly
Objectives of the Course	<ul style="list-style-type: none"> ▪ This lesson introduces the Solid Edge Assembly interface and discusses the different work flows for creating an assembly with the most commonly used part relationships.
Program Modules	<ul style="list-style-type: none"> ▪ <i>Assembly methods and relationships</i> ▪ <i>Additional relationships</i> ▪ <i>Editing assemblies</i> ▪ <i>Display tools</i> ▪ <i>Explode and section views</i> ▪ <i>Revising assemblies</i> ▪ <i>Designing in the context of the assembly</i> ▪ <i>Assembly inspection</i> ▪ <i>Alternate assemblies</i> ▪ <i>Placing tubing with XpresRoute</i>

Course Name	Solid Edge surfacing
Objectives of the Course	<ul style="list-style-type: none"> ▪ Create Control Drawings ▪ Use 2D geometry to develop 3D curves ▪ Use 3D curves to develop surfaces ▪ Create a solid and add appropriate solid based feature ▪ Tweak
Program Modules	<ul style="list-style-type: none"> ▪ <i>Introduction to surface modeling</i> ▪ <i>Creating and editing curves</i> ▪ <i>Indirect curve creation techniques</i> ▪ <i>Surface creation</i> ▪ <i>Surface manipulation tools</i> ▪ <i>Curve and surface inspection tools</i>

Course Name	Solid Edge Sheet metal design
Objectives of the Course	<ul style="list-style-type: none"> ▪ Create sheet metal parts using both modeling and sheet metal features. ▪ Effectively create and edit sheet metal features. ▪ Create sheet metal deformation features. ▪ Convert a solid model to a sheet metal model. ▪ Create flat patterns. ▪ Create sheet metal drawings.
Program Modules	<ul style="list-style-type: none"> ▪ <i>Introduction to Solid Edge</i> ▪ <i>Getting started in Solid Edge</i> ▪ <i>Reference planes</i> ▪ <i>Profiles and sketches</i> ▪ <i>Sheet metal features</i> ▪ <i>Sheet metal deformation features</i> ▪ <i>Additional sheet metal features</i>