

UNIGRAPHICS

Curriculum

Solid Modeling (Level 1)

Introduction to CAD/CAM

- Basic approach to Solid Modeling, Operating Systems, Unix Commands

Preference Sketch, Modeling Preferences

Layers Layer Settings, Visible in View, Layer Category, Move to Layer, Copy to Layer

Sub Functions Point, Vector, Plane sub functions,

Sketch Create Sketch, Basic Curves, Point, Associative Point, Rectangle, Fillet, Ellipse, Spline, General Conic, Edit Curve, Constraints, Mirror Sketch, Alternate Solutions, Drag, Show/Remove Constraints, Animates, Add object to Sketch, Add Extracted Curve To/From Reference, Position, Reattach, Activated, Deactivate, Retain Dimension Display, Update Model

Form Feature

Reference Features

- Datum Plane
- Axis CSYS

- Extract
- Sheet from Curves
- Bounded Place
- Thicken Sheet
- Sheets TO Solid Assistant

Swept Features

- Extruded Body
- Revolved Body
- Sweep ANONG Guide
- Tube

Primitives

- Cylinder
- Cone
- Sphere

Special Features

- Hole
- Boss
- Pocket
- Pad
- Slot
- Groove

Feature Operation

- Taper
- Edge
- Face
- Soft Blend
- Chamfer
- Hollow
- Thread
- Instance
- Saw
- Simplify Body
- Wrap Geometry
- Offset
- Scale
- Trim
- Split Body

Library Manger

- User Defined Feature

Associative Features

Boolean Operations

- Create
- Unite
- Subtract
- Intersect
- Edit Feature & Model Navigator
- Parameters
- Positioning
- Move
- Reorder
- Delete
- Replace
- Suppress
- Unsuppressed
- Suppress By Expression
- Remove Parameters
- Solid Density
- Play Back
- Resize Fixed Datum's
- Display Dimensions
- Make Current Feature
- Parent Curves
- Rename
- Object Dependency
- Browser
- Properties
- List Information

Miscellaneous

File

- New
- Open
- Close
- Save
- Save as
- Save All
- Options (Load options, Save Option)
- Import
- Export
- Interoperate
- Plot
- Save Bookmark
- Utilities
- Properties
- Recently Opened Parts

View

- Refresh
- Operation
- Orientation
- Toolbar

Format

- Layout-New, Open, Open Drawing
- Fit All View
- Update Display
- Regenerate
- Replace View
- Delete
- Save
- Save As

Tools

- Expre3ssions
- Macro
- Customize

Analysis

- Distance
- Angle

Advanced Modeling (level II)

Introduction to CAD/CAM

- Basic Approach to Surface Modeling

WCS

- Origin
- Dynamics
- Rotate
- Orient
- Change XC
- YC Directions
- Display
- Save

Curve Operation

- Offset
- Bridge
- Simplify
- Join
- Project
- Combined Projection

- Intersect
- Section
- Extract
- Offset in Face
- Wrap/ Unwrap

Free Form Feature

- Through Points
- From Poles
- From Point Cloud
- Ruled
- Through
- Curves
- Through
- Curve Mesh
- Swept
- Section
- Bridge
- N-Sided
- Surface Extension
- Law Extension
- Enlarge
- Offset
- Rough
- Offset
- Quilt
- Swoop
- Studio Surface
- Styled Blend
- Global Shaping
- Trimmed Sheet
- Fillet
- Foreign

Miscellaneous

Edit

- Curve
- Infer Point
- Free Form Feature

- Face
- Format
- Group
- Attributes
- Pattern
- Group Feature

Tools

- Visual Editor
- Spread Sheet

- Smart Models (Product Definition, Geometric Tolerance)
- Material
- Properties
- Introduction To knowledge Fusin
- Part Families

Advanced Modeling (LEVEL III)

Assembly Structure

- Top Down
- Bottom Up Design

Mating Conditions

- Mate
- Align
- Angle
- Parallel
- Perpendicular
- Center
- Distance
- Tangent

Component

- Add Existing
- Create New
- Create Array
- Substitute
- Reposition
- Mate

- Suppress
- Unsuppressed Component
- Replace Reference Set
- Define Mating
- Verify Mating Alternates
- Part Family Update
- Properties
- Check Clearances

Exploded Views

- Create
- Edit
- Delete
- Hide
- Show explode
- Unexploded
- Hide
- Show Components

Sequencing

- Create Sequence
- Sequence Properties
- Add Assembly Step
- Step Properties
- Make Current Step
- Delete
- Find In sequence
- Show all Sequences
- Rewind to First
- Step Reverse
- Reverse Play
- Play
- Step Forward
- Forward To Last
- Stop

Wave

- Associate Manager
- Geometry Navigator
- Part link Browser
- Part Navigator
- Linked Wave Date

Cloning

- Create Clone Assembly
- Edit Existing Assembly

Reports

- List Components
- Update Report
- Where Used
- Session Where Used
- Assembly Diagram
- Family Report

Context Control

- Find
- Open Locate Components
- Open By proximity
- Show Product Outline Save
- Restore Context
- Define Product Outline
- Set Work/Displayed Part

Envelope

- Wrap Assembly
- Linked Exterior

Miscellaneous

- Delay Interpart Updates
- Update Session
- Variant Configurations
- Edit Component
- Array
- WAVE Geometry Linker
- Reference Sets
- Zones Representations
- Component Filters
- Parts List

Assembly Navigator Assembly Preferences Drawing

- New
- Open
- Delete
- Edit
- Add view (Import, Orthographic, Auxiliary Detail View, Simple, Stepped, Half, Revolved, Fold Section Cut Simple/ Stepped, Half Section Cut From Pictorial View)
- Remove
- Move/Copy
- Align
- Edit View
- Define View
- Boundary Brake
- Section
- Broken View
- Display Drawing
- Update View

Insert

- Dimension (Inferred, Horizontal, Vertical, Parallel, Radius, Radius to Center, Folded Radius Concentric Circle, Arc Length Horizontal)
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- Chain, Vertical Chain, Horizontal
- Base Line, Vertical Base Line).
- Model Parameters[GDT Feature]
- Utility
- ID
- User Defined
- Custom Symbol
- Cross Hatching
- Tabular Note
- Ordinate Dimension
- Weld Symbol

Edit

- Origin
- Section Line
- Section Component
- Leader
- Ordinate Dimension
- Cross Hatch Boundary
- Suppress Drafting Object
- View Dependent edit

Preferences

- Annotation
- Drafting Preferences
- Section Line Display
- View display
- View Label



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