NX6 Modeling Tutorial

by

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## 1. Getting Started with NX6

- ➤ USB Drive
  - ▪ Always use your own personnel USB Drive, do not use the hard or network drive.
  - ▪ Construct separate directories for each NX homework assignment.
- ➤ Gateway Application
  - ▪ Screen Layout
    - ➤ Top: Top Menu Bar
    - ➤ Middle Top: Status Bar
    - ➤ Left: Left Menu Bar
    - ➤ Center: Graphics Window
  - ▪ Workflows: other applications Modeling, Design Simulation, & Manufacturing
  - ▪ Roles:
    - ➤ Go to Left Tool Bar
    - ➤ Select Roles
      - ▪ Select Essentials
        - • OK to overwrite
  - ▪ Customize: Right Click in Toolbar area to add/delete toolbars
  - ▪ Dialog Boxes: positioned by the Dialog Rail
  - ▪ Command Flow within Dialog Boxes: proceed top to bottom
    - ➤ Red Asterisk: selection/input required
    - ➤ Green Asterisk: selection/input completed
    - ➤ Orange Highlight: current active selection step
    - ➤ Green Highlight: next suggested selection step
    - ➤ OK/ Apply/ Cancel
  - ▪ Navigators:
    - ➤ Part Navigator, located on Resource Bar, pin to keep open
      - ▪ Part Navigator
      - ▪ Assembly Navigator
      - ▪ Operation Navigator
  - ▪ Parts:
    - ➤ Items saved as parts (.prt file)
      - ▪ Models
      - ▪ Assemblies
      - ▪ Drawings
  - ▪ Help:
    - ➤ F1 Key on any function in any application
    - ➤ All Programs/ UGS NX6.0/ NX6 Documentation
- ➤ Set up Defaults
  - ➤ File
    - ▪ Utilities
      - • Customer Defaults
        - ♦ Gateway/ General/ Part: Inches
        - ♦ OK

2. Creating a New 3-D Model (rectangular block with a hole):

Select New Model Template

- ➢ Gateway Application (NX Start/ All Applications/ Gateway):
  - ▪ New
    - • New Dialog
      - ◆ Model Tab: Units: Inches
      - ◆ Drawing Tab: Units: Inches
      - ◆ Simulation Tab: Units: Inches
      - ◆ Manufacturing Tab: Units: Inches
      - ◆ Select Model Tab:
        - ➢ Select Template Name: Model
        - ➢ Input: New File Name: "Block.prt" (use Choose New File Name Icon)
        - ➢ OK
      - ◆ OK (Note: Top Menu Bar reads, "NX6-Modeling-Block.prt(Modified). You are now in the Modeling Application.
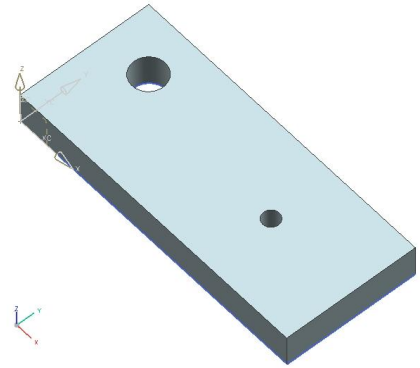
Sketch 2-D Profile on XY Plane

- ▪ Sketch Icon (create 2-D sketch in the XY Plane that you will extrude in the Z Plane)
  - • Create Sketch Dialog
    - ◆ Select Planar Face or Plane: (left click on the XY Plane)
    - ◆ Select Horizontal Reference: (left click on X Axis)
    - ◆ OK (You are now in the Sketcher Application)
  - • Rectangle Icon
    - ◆ Rectangle Dialog
      - ➢ Rectangle Method: Select 3-point (Note Status Bar prompting to "Select the first point of the rectangle)
        - ▪ Cursor Select point (0,0) for first point
        - ▪ Cursor Select second point
        - ▪ Cursor Select third point
  - • Circle Icon
    - ◆ Circle Dialog
      - ➢ Circle Method: Select Circle by Center and Diameter
        - ▪ Cursor Select the center of the hole
        - ▪ Cursor Select the radius of the hole, or input/enter the diameter.
  - • Inferred Dimensions Icon (always dimension in the sketch … aids revisions)
    - ◆ Cursor Select (left click) a single line of your sketch. Move dimension to desired location and left click
    - ◆ Repeat for all critical line dimensions
    - ◆ Cursor Select a single circle radius, and move dimension to desired location.
    - ◆ Cursor Select a single circle center, then Cursor Select a line. Move the distance dimension to desired location
  - • Finish Sketch Icon (automatically returns you to the Modeling Application)

Create
3-D
Model ⎱
- ▪ Extrude Icon
  - • Extrude Dialog
    - ♦ Select Curve (left click on the sketch in the XY Plane)
    - ♦ Select Vector (Note the Reverse Direction Icon for direction in the Z Plane)
    - ♦ Input Distance (the thickness of your block in the Z Plane)
   - ♦ Apply/ OK
  - • File/ Save

3. Miscellaneous Modeling Aids

- ➢ Reference the Top Menu Visualization Tools:
  - ♦ Fit Icon
  - ♦ Zoom Icon
  - ♦ Zoom In/Out Icon
  - ♦ Rotate Icon
  - ♦ Pan Icon
  - ♦ View Icon (trimetric, top, left, etc)

- ➢ Reference Model Editing Functions
  - ▪ Right click on the model to edit the 3-D Extrusion Function. There are two options:
    - • Select Edit Parameters (edit in current model state)
      - ♦ Input new Distance (thickness in the Z Plane)
      - ♦ OK
    - • Select Edit with Rollback (edit in prior model state)
      - ♦ Input new Distance (thickness in the Z Plane)
      - ♦ OK
  - ▪ Cursor highlight and right click on original sketch in the Graphics Window to edit the 2-D Sketch Function. There are three options:
    - • Select Edit (opens the Sketcher Application)
      - ♦ Modify dimensions that were created using the Inferred Dimensions
      - ♦ Finish Sketch
    - • Select Edit Parameters (edit in current model state)
      - ♦ Left click Inferred Dimension to be modified
      - ♦ Input modified dimension
      - ♦ Apply/ OK
    - • Select Edit with Rollback (edit in prior model state)
      - ♦ Opens the Sketcher Application (the prior model state in this case)

4. Adding an Extrusion to an Existing Model

- ➢ Modeling Application (NX Start/ All Applications/ Modeling):
  - ▪ File/ Open: Open an existing Model
  - ▪ Extrude Icon:
    - • Extrude Dialog
      - ♦ Select Curve (left click on the Model planar face that you wish to place the additional extrusion). You are now in the Sketcher Application
      - ♦ Line Icon
        - ➢ Line Dialog
          - ▪ Select points of the line (Note Status Bar prompting to "Select the first point of the line"). Sketch a triangle.
            - • Cursor Select first & second point of line 1
            - • Cursor Select first & second point of line 2
            - • Cursor Select first & second point of line 3
            - • Inferred Dimensions. Construct critical Dimension Lines.
            - • Finish Sketch. You are now back in the Modeling Application.
      - ♦ Select Vector (Note the Reverse Direction Icon for direction in the Z Plane)
      - ♦ Input Distance (the thickness of your block in the Z Plane)
      - ♦ Apply/ OK
  - ▪ Unite Icon (unite the two extrusions into a body)
    - ♦ Unite Dialog
      - ➢ Select target body (Cursor Select the main body)
      - ➢ Select tool bodies (Cursor Select secondary bodies that you wish to unite with the target (main) body)
   - ➢ Apply/ OK
  - ▪ File/ Save
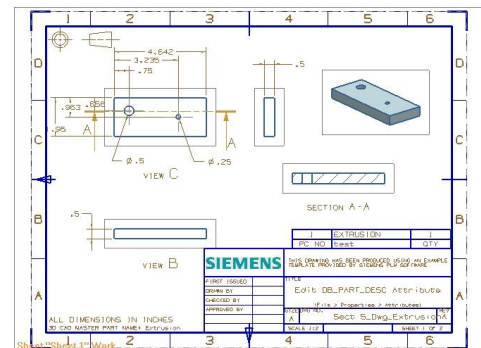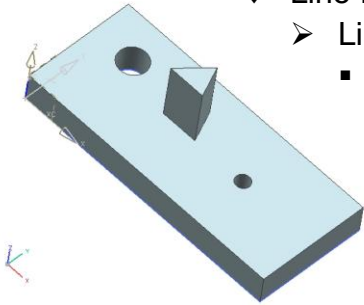
5. Create a 2-D Drawing:

- ➢ Gateway Application:
  - ▪ New
    - • New Dialog
      - ♦ Select Drawing Tab:
        - ➢ Select Template Name: A-Views, Units: Inches, Reference Existing
        - ➢ Input: New File Name: "Dwg_Block.prt" (use Choose New File Name Icon)
        - ➢ Input: Part to create a drawing of:
          - ▪ Choose New File Name Icon
            - • Select Master Part Dialog
              - ♦ Open
              - ♦ Part Name Dialog
              - ♦ Select "Block.prt" (Select the 3-D model that you wish to make a drawing of)

- ◆ OK
  - OK
    - OK (You are now in the Drafting Application, with a 3-view, A-Sheet drawing of your model)
  - Apply Dimensions (Dimension all critical attributes)
    - ◆ Center Mark
    - ◆ Horizontal Dimensions: (Cursor Select two points of reference)
    - ◆ Vertical Dimensions: (Cursor Select two points of reference)
    - ◆ Hole Dimensions
  - Section View
    - ◆ Select Parent View
    - ◆ Select Cut Position
    - ◆ Indicate Center of Section View (orientation of view)
  - Insert
    - ◆ Parts List
  - File/ Save
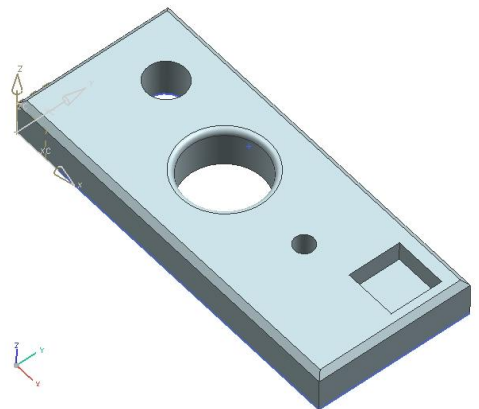
6. Adding Features to an Existing Model
- ➢ Modeling Application:
  - ▪ File/ Open: Open an existing Model
  - ▪ Hole Icon:
    - Hole Dialog
      - ◆ General Hole
      - ◆ Specify Point: (Cursor Select the planar face that you wish to put the hole). You are now in the Sketcher Application.
        - ▪ Coordinates: Absolute: (Input X and Y Coordinates of hole)
        - ▪ OK
        - ▪ Finish Sketch
      - ◆ Form: Simple (Note, you could select Counterbored, Countersunk, etc)
      - ◆ Diameter: (Input value)
      - ◆ Depth: (input value)
      - ◆ Apply/ OK
  - ▪ Chamfer Icon:
    - Chamfer Dialog
      - ◆ Select Edges to Chamfer: (Cursor Select all necessary edges)
      - ◆ Cross Section: Symmetric
      - ◆ Distance: (Input value)
      - ◆ Apply/ OK
  - ▪ Edge Blend Icon:
    - Edge Blend Dialog:
      - ◆ Select Edges to Blend: (Cursor Select all necessary edges)
      - ◆ Radius: (Input value)

- ♦ Apply/ OK
- ▪ Construct a cutout or pocket.
  - • Extrude Icon:
    - ♦ Extrude Dialog
      - ➢ Select Curve (left click on the Model planar face that you wish to place the additional extrusion). You are now in the Sketcher Application
      - ➢ Rectangle Icon
        - ▪ Rectangle Dialog
          - • Select Rectangle Method by 2-points
            - ♦ Cursor Select first & second point
            - ♦ Inferred Dimensions. Construct critical Dimension Lines.
            - ♦ Finish Sketch. You are now back in the Modeling Application.
      - ➢ Select Vector (Note the Reverse Direction Icon for direction in the Z Plane)
      - ➢ Input Distance (the depth of your pocket in the Z Plane)
      - ➢ Apply/ OK
  - • Subtract Icon (subtract the tool body from the target or main body to form a cutout or pocket)
    - ➢ Subtract Dialog
      - ▪ Select target body (Cursor Select the main body)
      - ▪ Select tool bodies (Cursor Select secondary bodies that you wish to subtract from the target (main) body)
      - ▪ Apply/ OK
- ▪ File/ Save

7. Adding Features to a Datum Plane Orientation: (Add a angled tube extrusion onto a block)

- ➢ Modeling Application:
  - ▪ File/ Open: Open an existing Model
  - ▪ Datum Axis Icon:
    - • Datum Axis Dialog
      - ♦ Select objects to define datum axis: (Cursor Select a model edge)
      - ♦ Apply/ OK
  - ▪ Datum Plane Icon: (Add angled Datum Plane)
    - • Datum Plane Dialog
      - ♦ Type: Inferred
      - ♦ Select objects to define plane:
        - ➢ Cursor Select Datum Axis
        - ➢ Cursor Select a Model Face
      - ♦ Angle (Input Angle between Model Face and intended datum plane, pivoted about the Datum Axis)

- ♦ Apply/ OK
  - • To change the Datum Plane angle
    - ♦ Right Click on the Datum Plane
      - ➢ Edit Parameters
        - ▪ Change Angle values
        - ▪ OK
- ▪ Datum Plane Icon: (Add model Center Line Datum Plane)
  - • Datum Plane Dialog
    - ♦ Type: At Distance
    - ♦ Select planar object:
      - ➢ Cursor Select the bottom of model (planar face)
      - ➢ Offset Distance (Input half of the Model Thickness to form a center datum plane)
      - ➢ Cursor Select and hold down on corner of the Datum Plane. Enlarge plane to intersect with the angled Datum Plane.
      - ➢ Apply/ OK
- ▪ Sketch Icon:
  - • Sketch Dialog
    - ♦ Type: On Plane
    - ♦ Select Planar Face: (Cursor Select the Angled Datum Plane)
    - ♦ OK (You are now in the Sketcher Application)
  - • Circle Icon
    - ♦ Select the center point of the circle: (Cursor select the center to be on the Center Line Datum Plane)
    - ♦ Diameter: (Input value)
    - ♦ Finish Sketch
- ▪ Extrude Icon:
  - • Extrude Dialog
    - ♦ Select section geometry: (Cursor Select the circle on the Angled Datum Plane)
    - ♦ Specify Vector
    - ♦ Limits:
      - ➢ Start: Value
      - ➢ End: Through All
- ▪ Unite Icon (unite the two extrusions into a body)
    - ♦ Unite Dialog
      - ➢ Select target body (Cursor Select the main body)
      - ➢ Select tool bodies (Cursor Select secondary bodies that you wish to unite with the target (main) body)
      - ➢ Apply/ OK
- ▪ File/ Save

8. Creating Revolved Extrusions: (Create a shoulder around a cylinder)

- ➢ Gateway Application:
  - ▪ Create a new 3-D Model of a cylinder (circle sketch on XY Plane centered at (0,0), extrude in Z Plane). You are now in the Modeling Application.
  - ▪ Create a Datum Plane on the YZ Plane located at X=0 (make certain to extend the Datum Plane so that it intersects the model at your point of interest)
  - ▪ Sketch Icon:
    - • Sketch Dialog
      - ♦ Type: On Plane
      - ♦ Select object for sketch plane: (Cursor Select the YZ Datum Plane)
      - ♦ OK (You are now in the Sketcher Application)
    - • Line Icon
      - ♦ Line Dialog: (create a rectangular shoulder)
        - ➢ Select points of the line (Note Status Bar prompting to "Select the first point of the line"). Sketch a 3-sided rectangle.
          - ▪ Cursor Select first & second point of line 1
          - ▪ Cursor Select first & second point of line 2
          - ▪ Cursor Select first & second point of line 3
          - ▪ Finish Sketch. You are now back in the Modeling Application.
  - ▪ Revolve Icon
    - • Revolve Dialog
      - ♦ Section: (Cursor Select sketch of the shoulder)
      - ♦ Axis: (Cursor Select the Z Axis)
      - ♦ Angle: 360 Deg.
  - ▪ Unite Icon (unite the revolved shoulder into the main body)
    - ♦ Unite Dialog
      - ➢ Select target body (Cursor Select the main body)
      - ➢ Select tool bodies (Cursor Select secondary bodies that you wish to unite with the target (main) body)
      - ➢ Apply/ OK