Siemens PLM NX 7.5 User Interface

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Starting NX

1) On the desktop, double-click 🌀 or from the start menu select: "**All Programs**" → "**UGS NX 7.5**" → "**NX 7.5**".

2) Wait for program to start.

3) Select "**File**"→ "**New**" or "**File**" → "**Open**".

4) When creating a new file:
    1) Select units (mm or inches) – This cannot be changed later!!!
    2) Select the "Model" tab.
    3) Select a template (e.g., "Model", "Assembly", "Shape studio", "Sheet metal", "Electrical routing")
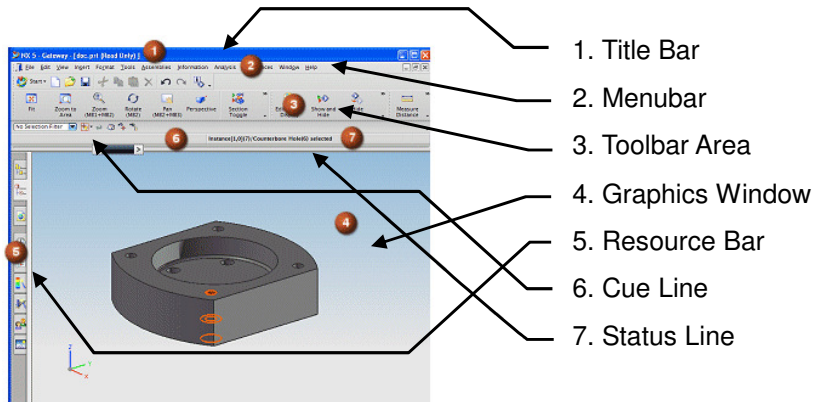   4) Select or type in the directory.
    5) Type in the file name.

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Selecting An *Application*

- Press 🔵 Start▾ to change the default application.
- NX *Applications*:
  - Solid Modeling: "Modeling"
  - Assembly Modeling: "Assemblies"
  - Drafting: "Drafting"
  - FEA "Advanced Simulation"
  - Mechanism Analysis "Motion Simulation"
  - NC Path Generation "Manufacturing"
  - Many other specialized applications
- The application controls what buttons are available.

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Getting Help

1. On-line tutorials Help → Training → NX 7.5 CAST Library → Design

2. Context-sensitive help – press 'F1'

3. On-line documentation Help → Documentation → Design → Modeling

4. See instructor.

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NX User Interface Layout

1. Title Bar
2. Menubar
3. Toolbar Area
4. Graphics Window
5. Resource Bar
6. Cue Line
7. Status Line

From NX on-line documentation.

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Viewing Modes & Control

- Dynamic View Control:
  - Rotate: Use middle mouse button to rotate
        Press right mouse button, then ⟳ to specify rotation point
  - Pan: Use shift key & middle mouse button
  - Zoom: Use mouse wheel to zoom
  - Use ⊞ to fit objects to screen
  - Use space ball: push/pull for zoom, left/right/up/down for pan, twist for rotate

- View Orientation: top, front, right, back, isometric,...
  - Use ⌐ ▾ or right mouse button press in Graphics window

- Render Style: shaded, wireframe, hidden lines removed
  - Use ⬛ ▾ or right mouse button press in Graphics window

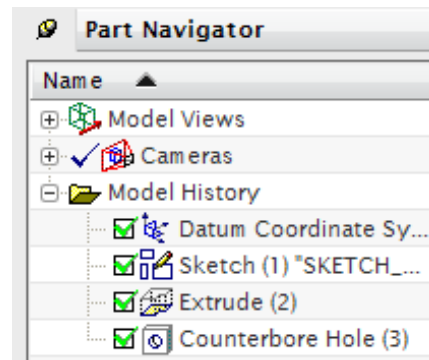MAE 455 Computer-Aided Design and Drafting   6

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Accessing the Part "History"

Press this tab:

Get this window (the
Part Navigator):

Use RMB to perform actions on features.

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Using the Mouse

- **Left button click:** pick a location, object or button
  - Multiple objects can be selected.
  - Hold the shift key down to unselect a selected object.
- **Left button hold & drag:**
  - menu/option-control select
  - dragging selection box corner
- **Middle button:** "OK". Same as <return>.
- **Right button click:**
  - context-sensitive drop-down menu

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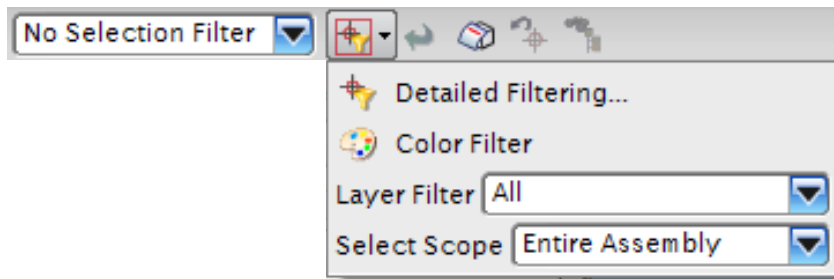West Virginia University.

Actions and Object Selection

- There are two themes for selecting objects:
  1) Pick objects then select action
  2) Select action then pick objects
     – Most Office software uses (1)
     – CAD software uses combination of (1) & (2)
- In NX, the steps of a task may also be identified by:
  – Sections of a dialog box.
  – A row of buttons.

  You can go back to a previous step by pressing the button for that step.

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Selection Filtering

- Use the controls in the selection bar to filter which objects can be selected:

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Controlling Object Appearance

The visual appearance of objects can be controlled by selecting "**Edit**" → "**Object Display**."

Can control:
– Line styles
– Color
– Translucency
– Surface textures (with Visual Studio)

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Using Layers

• Each NX object exists on a layer:

Layers 1-20	Solid geometry
Layers 21-40	Sketch geometry
Layers 61-80	Reference geometry
...	

• When a new object is created, it is put on the "**Working Layer**"
• Use "**Format**" → "**Move to Layer…**" to move objects from one layer to another.
• Use "**Format**" → "**Layer Settings…**" to set each layer as: "**Working layer**", "**Invisible**", "**Selectable**", or "**Visible only**"

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Basic Solid Modeling Process

1) Create datum planes
2) Create a sketch on a plane
3) Sketch curves
4) Constrain curves
5) Perform Sweep
6) Repeat 2-5 a few times to get basic shape.
7) Shell the basic shape if necessary.
8) Add fillets and rounding.

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