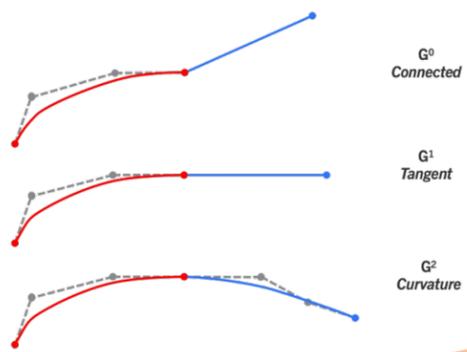




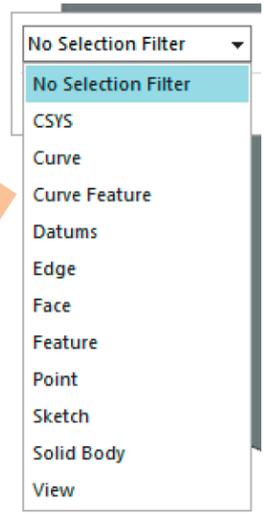
Licensing Error

- You must attach the Clemson licensing bundles every time you log on to a new computer
1. Go to File
 2. Select <Utilities>
 3. Select <Bundles>
 4. Select <ACD10> and click <Add>
 5. Click <OK>



Continuity

Set continuity on splines and edge blends



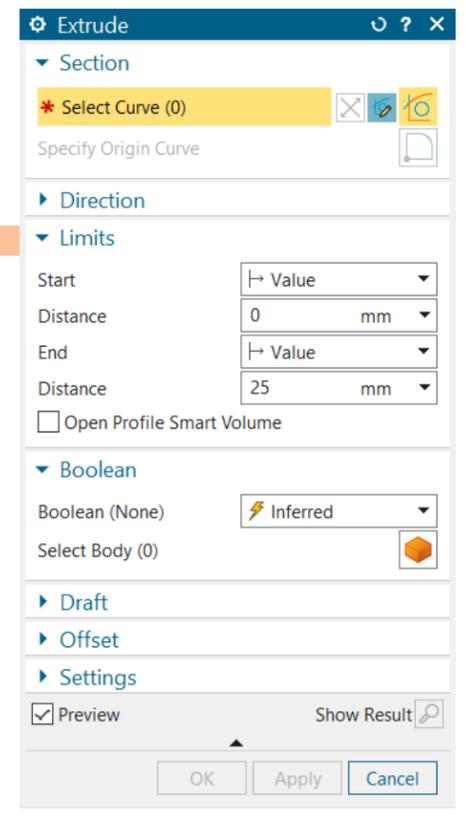
Find a Command

Search Bar

Shows, explains, executes, and attaches commands

Limits

- **Value:** a numeric distance from the starting point (can be positive or negative)
- **Symmetric Value:** Makes both start and end limits equivalent
- **Until Next:** The feature extends until it encounters existent geometry
- **Until Selected:** The feature extends until it encounters the selected geometry
- **Until Extended:** The feature extends until it encounters the planar extension of the selected geometry
- **Through All:** The feature does not terminate



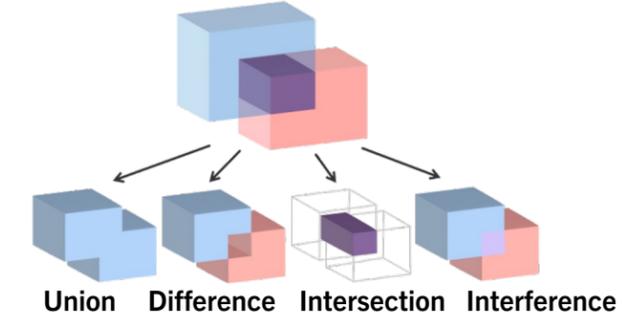
Operations

- Application controls what commands you can see
- Basic CAD application is “Modeling”
- Extrusion and revolution are in the “Home” tab of the modeling application
- Sweeps and lofts are found in the “Surface” tab as “Swept” and “Through Curves” respectively

Commands

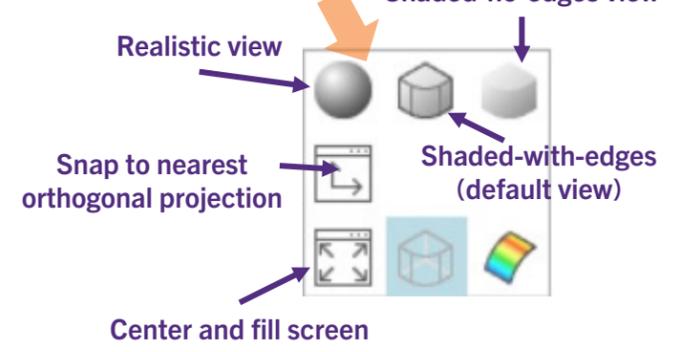
Yellow box shows active parameter
Click Scroll Wheel to advance
Use to reverse direction

Booleans



Navigation

- LMB: Click to select features
- Shift + LMB: Click to deselect features
- Scroll Wheel: Scroll to Zoom; Click-hold to Rotate
- Scroll Wheel + RMB: Click-hold to pan
- RMB: Click for menu; Click-drag for quick actions



Filters

- RMB click to pull up the selection filter menu
- Filter let you select only a certain type of feature
- Set the selection scope in the righthand menu to limit what you can click on

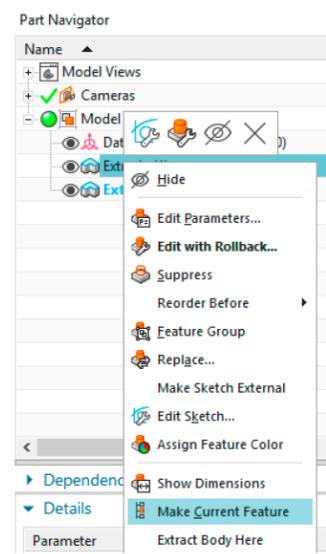
Snap Points

- Click the wheel at the top-center of the screen to set which points you can snap to when selecting



Rollback

- Rollback to a previous feature in the model history
- RMB click on the feature in the “Part Navigator”
- Select “Make Current Feature”

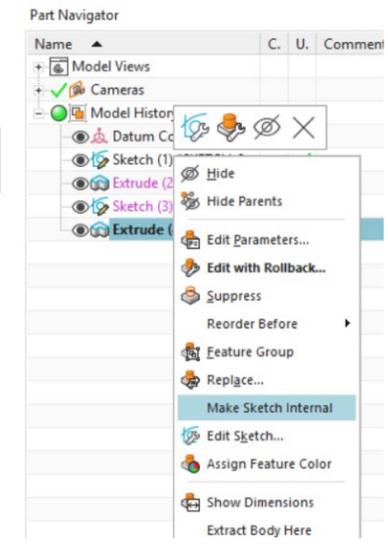


Sketching

- Start at origin and work counter-clockwise
- Define profile, then constraints
- Add as few as dimensions as possible, last
- To use a solid edge in your sketch, click “Project Curve”
- Make sketches used in a single feature internal by selecting <Make Sketch Internal> from the Part Navigator

Selection

- Select the midpoint of a line by clicking the line, then hovering over the midpoint.
- Select the center of a circle by clicking from the inside, or select the curve by approaching from the outside



Siemens NX Cheat Sheet

Section Views

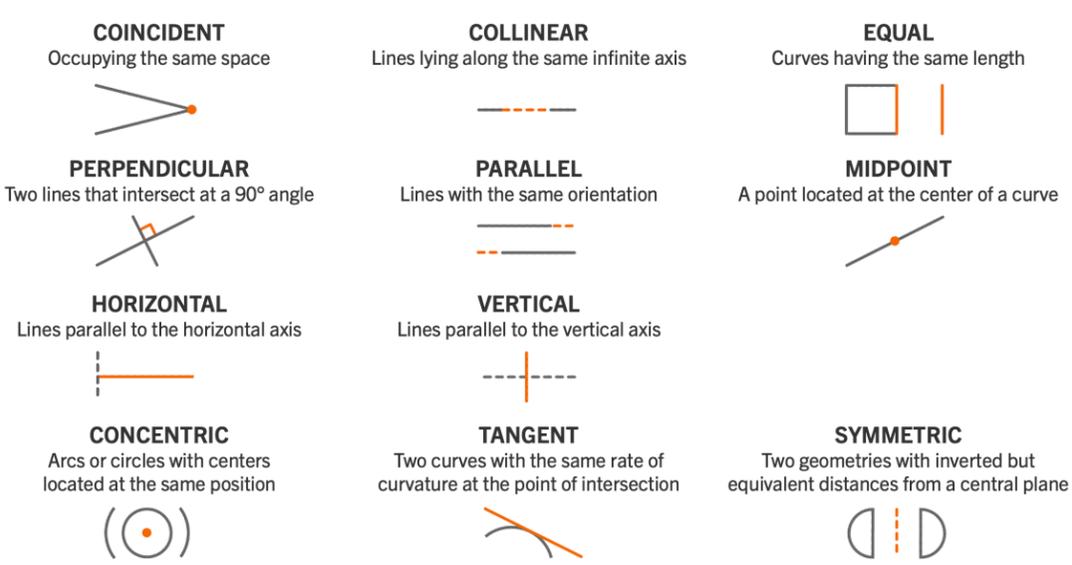
- Go to the Assembly Navigator
- RMB click on “Sections”
- Select “New Section”
- Remove section view: RMB on section view
- Click “Unclip”

Assemblies

- Hover over the constraints to read the tooltip explaining each assembly mate
- Use the “Move Component” command to adjust positions (if needed)

Geometric Constraints

Click objects to constrain
In 1946: Floating tooltip appears with possible constraints (right click selection to make reappear)
In 1980: Constraints anchored at top of window



Parameters/Equations

- Set variables for the model through the Expressions editor, under the “Tools” tab
- You can reference parameters defined in the editor in dimensions
- Use the “=” sign to set dimensions from equations, such as “=width*2”

