Views

- click in a view to make it active
- 1, 2, 3, 4 – Number of viewports

<table>
<thead>
<tr>
<th>5</th>
<th>top</th>
</tr>
</thead>
<tbody>
<tr>
<td>6</td>
<td>front</td>
</tr>
<tr>
<td>7</td>
<td>right</td>
</tr>
<tr>
<td>9</td>
<td>current sketch view</td>
</tr>
</tbody>
</table>

Zoom/Pan/Rotate

- z – zoom
- z w – zoom window
- z a – zoom all
- dynamic zoom – roll mouse wheel
- dynamic pan – press/hold mouse wheel, move mouse

General

Plan ahead
Read the command line!
Make sure you are at the command prompt before entering a command - use Esc
Finish commands before starting a new one

- ctrl-Z – Undo (works multiple times)
- Esc – Cancel
- Enter – Accept default or Previous Command
- F2 – toggle command window display
- rr – regenerate the display

Selecting objects

- Must have the object within the cursor selection box
- Try to select on object where there is nothing else in the cursor selection box
- Select close to, but not on the very end of, the desired end of a line
- Select in the isometric view when possible
- Circle and arc centers are selected by clicking on the arc

Deleting objects

- 2D geometry – select it and press the delete key
- Features – right click on the name in the Desktop Browser

Changing objects

- Dimensions – double click
- Features – double click on the name in the Desktop Browser
- Color – right click on the part or feature name, select properties, color
- Sketches – left click on the + next to the feature name, select properties, double click on the associated sketch

Setup

- Assist->Options, System Tab, Startup:
  - Show Traditional
- View->Display->Desktop Browser
- Disable OTHO, POLOR, OSNAP, OTRACK on status bar

Initial Sketch Plane

- Display views (4) and make iso view active
- Top (world XY): do nothing (or SS, X, <accept>)
- Front (world ZX): (SS, Z, <accept>)
- Right (world YZ): (SS, Y, <accept>)
- Use Z W to set the initial sketch size

Basic 2D Sketching

- pl or Design->Polyline
- use 0,0 for start point for first sketch
- single click on each vertex
- use C to close the polyline
- when sketching on an existing part, start with the sketch offset from the desired location

Dimensioning and Constraining Sketches

- Must profile first – Part->Sketch Solving->Single Profile
- Must have "Solved underconstrained sketch…"
- Part->2D Constraints
- Part->Dimensioning->New Dimension

Sketched Features

- Part->Sketched Features->Extrude
- Be sure to select the correct operation (join, cut, intersect)

Drawings

- Orient active viewport to desired front view
- Switch to Drawing mode via browser tab
- Right click on layout name, select new view, base view
- Set options, enter switches back to model view
- Place view, enter to finalize
- Right click on base view name, use multiple views option to place other views

New text - Annotate->Text->Paragraph
Dimensions – Annotate->Reference Dimension
Modify text – double click on existing text
Move dimension – click on it, drag with the

Constraint Types

- H - Horizontal
- V - Vertical
- T - Tangent
- P - Parallel
- L - Perpendicular
- C - Collinear
- N - Concentric
- R - Same Radius
- J - Join
- X - Same X value
- Y - Same Y value
- E - Equal length
- M - Mirror
- F - Fix

Printing

- File->Plot
- . Plot device: current room
- . Plot area: Layout
- . Plot scale: Scaled to Fit
- . Full Preview

Using an Existing Face as a Sketch Plane

- Always have at least two views displayed.
- Make selections in the isometric view.
- Part->New Sketch Plane (or SS)
- Select the desired surface by selecting on the surface edge.
- Select the orientation of the sketch plane by:
  - selecting an edge
  - specifying parallel to WCS X, Y, or Z axis
  - Using rotate/flip options
- Accept the orientation of the sketch plane
- WATCH OUT: it is easy to see the new coordinate system in the desired orientation and then pick another menu option. If you don't accept the final orientation, the sketch plane command is terminated without being completed and your sketch plane IS NOT set.