Introduction to Pro/ENGINEER

On Line Reference Material

In Pro/ENGINEER, selecting Help – ProE Help System will start the system browser (in this case Internet Explorer) and bring up the screen shown below.

Welcome to PTC Help

PTC is continuing its commitment to provide integral intranet enhancements through the latest series of software products, including the innovative changes in this release of PTC Help.

PTC Help is integrated with fully functional, highly flexible CAD/CAM and Data Management software available from PTC. It is easy to locate a desired topic. PTC Help can be installed on a Web server, allowing Web clients to access PTC Help without having direct access to file server machines, thus reducing network traffic and enabling a LAN/WAN configuration.

What's Available in PTC Help?

PTC Help offers:

- A new help system with integrated table of contents, index, and search capability
- Full certification of Internet Explorer 4.0 and later, and Netscape 4.06 and later
- Web server installation capabilities
- Full context-sensitive help, allowing access to PTC Help with a click of the mouse
- Expanded context-sensitive help in dialog boxes
- See Also hotspots for related topics

Selecting the Contents button in the upper right corner will yield a screen that has tabs on the left as shown below.

With the Contents tab selected, expanding the Using Sketcher and double clicking on About Sketcher Mode…. will display information in the right window on that topic.
Picking on the Index tab and scrolling through the index until Protrusion – SOLID is reached and double clicking on SOLID will bring up information on that topic in the right region of the screen.

About Sketcher Mode with Improved Intent Manager

Improved Intent Manager enables you to dynamically dimension and constrain geometry as you sketch. You can turn off Intent Manager by clicking Intent Manager. Before you enable Intent Manager for an existing section, make sure the section is successfully regenerated. Any extra dimensions found by Sketcher will be converted into reference dimensions.

To set Sketcher to use Intent Manager by default, set the configuration option to "sketcher_intent_manager" to "yes".

Overview Demo

When you enter Sketcher mode for the first time, the Read Me alert dialog box appears, instructing you to review the Overview Demo. To prevent this dialog box from showing up for future sessions, set the "sketcher_readme_alert" configuration option to "no."

Terminology in Sketcher
To search for specific information, select the Search tab. Enter what you wish to search for in the box provided (in this case spline) and press the Find button. Double pick on the appropriate topic (About 2D Geometry) in the list provided to display detailed information on the right side of the screen.

To Create a Protrusion (basic)

1. Choose Feature > Create > Solid > Protrusion.
2. Specify a method of adding material by choosing an option from the Solid OPTS menu. The options are:
   - Extrude: Create a feature by extruding its section.
   - Revolve: Create a feature by revolving its section.
   - Sweep: Create a feature by sweeping a section along a trajectory.
   - Blend: Create a feature by blending multiple sections.
   - Use Quit: Create a feature by adding material to a quilt.
   - Advanced: Create features using advanced feature forms.
3. Select an option from the lower portion of the Solid OPTS menu:
   - Solid (Default): Create a feature as solid.
   - Thin: Create a thin feature by specifying the feature's thickness as a parameter.
   
Choose Done from the SOLID OPTS menu.
About 2D Geometry

You create geometry for drawing views by sketching. This chapter describes different drafting options and tools you can use to modify geometry. While you are drafting, the drawing grid, layer, and group functions may be useful.

Sketching Geometry

You can access drafting options in the Draft Create menu by using the Draft option in the Legacy Dwg menu.

Locating Points

When locating points for sketched geometry (using the Adv Geom... option in the Draft Create menu, followed by the Point option), the Get Point menu displays the following options:

- **Pick Point**: Locates any point on the screen.
- **Vertex**: Locates a point at the endpoint as well as midpoint of an open curve (line, arc, spline), the center of an arc or circle, or the intersection between two entities.
- **On Entity**: Locates a point anywhere along an entity.