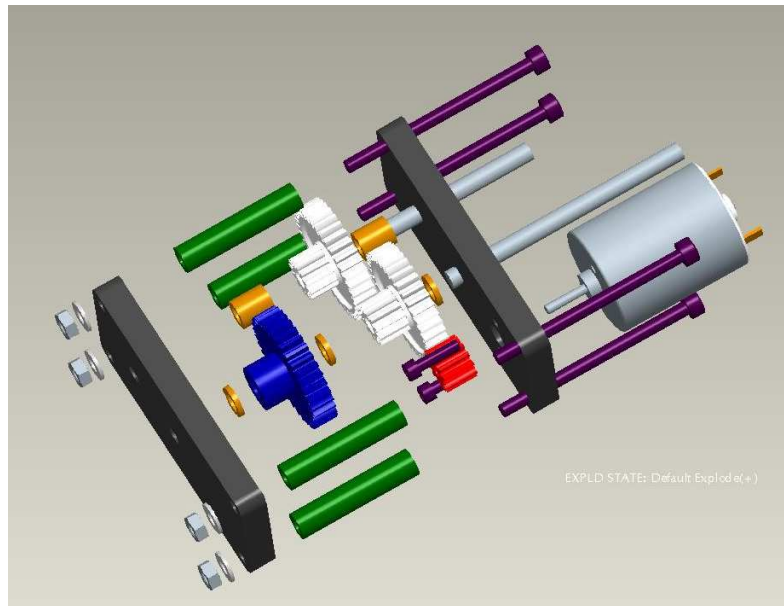


Pro/ENGINEER – Wildfire 2

Introduction Manual



This manual is designed to give the user a very basic understanding of the philosophy of the Pro-Engineer Wildfire (ver. 2.0) package and its user interface. The user is taken through the construction of a number of simple parts, which illustrates a number of the basic commands of Pro-Engineer. The parts are then combined to make an assembly which is then animated analysed using the motion facilities included in the package. having completed this handout the user should be in a position to explore the many other features within the ProE suite of programs. Additional support is available in the package and from web based manuals and tutorials.

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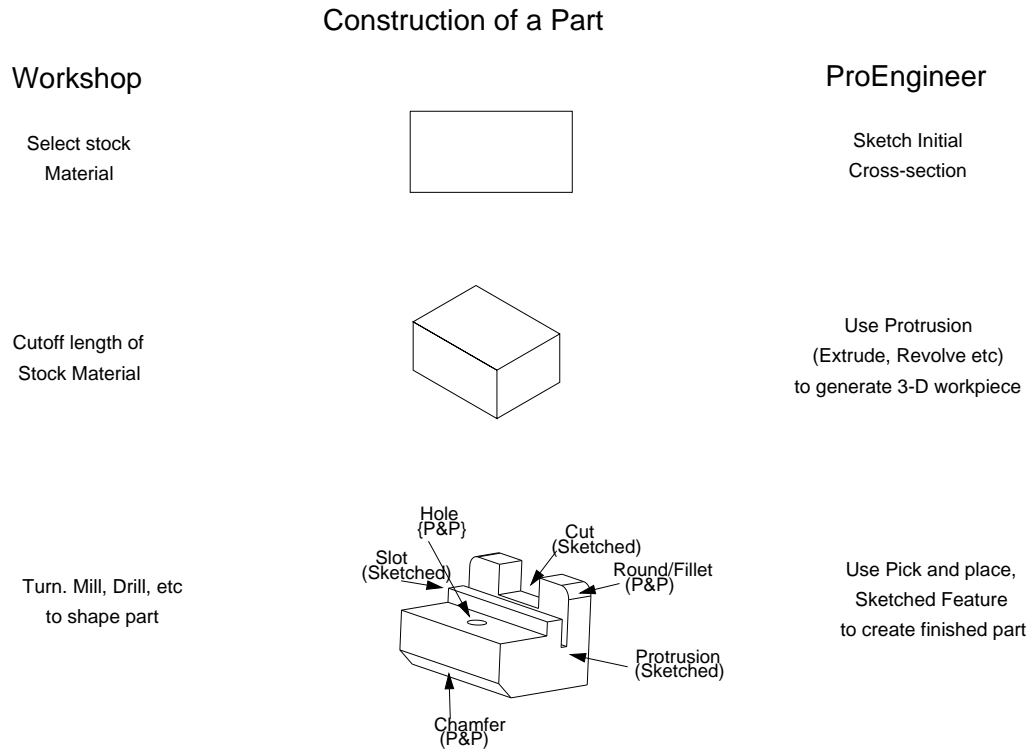


Figure 1: Comparison of physical and ProE methods of part construction

Part I

Overview of Wildfire

1 Basics

Pro/ENGINEER (Pro/E for short) is a commercial CAD/CAM package that is widely used in industry for CAD/CAM applications. It is one of the new generation of systems that not only offer a full 3-D solid modeller, in contrast to purely 2-D and surface modellers, but also parametric functionality and full associativity. This means that explicit relationships can be established between design variables and changes can be made at any point in the modelling process and the whole model is updated.

The method of constructing a model of an object is very similar to that followed in the production of a physical component. For example the manufacture of the shaped block in Figure 1 would start with the choice of construction environment, the selection of a piece of stock material followed by a series of manufacturing processes, e.g. milling, drilling, welding/sticking. Pro/E has direct analogues for most of these operations as various types of FEATURES which can be combined to generate a complete representation of a PART, Pro/E's terminology for a single component. Features fall into three main categories, Construction, Sketched and Pick/Placed.

1.1 Features

1.1.1 Construction Features

These features are purely used as an aid to the construction of the part, a number of various forms are available the most commonly used are the:

Csys Coordinate systems which aid in the orientation of additional features and the assembly of the part in to subsequent assemblies. CSYS feature is normally the first feature in a part definition and is used as the basis for the placement of all subsequent features.

Datums These are an extension of the idea of construction lines as used on a traditional drawing. The most used type is a DATUM PLANE which allows a 2-D reference plane to be defined in space. Additional forms include DATUM AXES, DATUM POINTS and DATUM CURVES. It is normal to add three DEFAULT datum planes, immediately after the initial coordinate system, to effectively generate default x-y, x-z and y-z planes.

1.1.2 Sketched Features

These features are so named because they all involve the use of the SKETCHER mode within ProE, (see below for more details on its use). The main features that use this functionality are:

Protrusion Using this feature material can be added to/removed from a part by sketching a cross-section and then extruding/revolving/sweeping the section to produce a 3-D solid/cut. A solid protrusion is normally the first non-constructional feature in a part, and is used to produce the base solid entity of the part. In the material removal mode the action is similar to a turning, saw or milling cut.

Rib This allows the user to produce a thin rib or web. This is a limited version of the protrusion function.

1.1.3 Pick & Place Features

Pick and place features tend to refer to simple or standard operations, e.g. the production of HOLES, ROUNDS and CHAMFERS. The action to produce the required effect has been preprogrammed into ProE, thus requiring the user to indicate the position of the operation on the existing model.

1.2 Modification of Features

The parametric nature of ProE means that the modification of features is relatively easy, individual features can be selected and the associated parameters/dimensions

changed. However, it should be noted that ProE produces a HISTORY based model in which features can be dependant on one or more previous features for their definition, e.g. a chamfer on an edge generated by a cut or protrusion. These PARENT-CHILD dependencies mean that when a parent feature is modified its children are automatically revised to reflect the changes. N.B. Care should be taken not to remove references used by child features.

2 Getting Started

The commands given below initiate ProEngineer Wildfire on the CD/DPO version of the CUED interface. If you wish to run ProE on another system please contact your system manager.

Start menu

- Close down unused programs, ProE imposes a large load on the system
- Select **Start** from the main icon bar and then select **Pro/Engineer Wildfire 2** from the **Programs**→**3D Modelling** submenu.

N.B. Some systems default to starting the PTC Application Manager in response to the **wildfire &** command. This is indicated by a small window in the top left hand window, to start ProE select **wildfire2** from the **Start menu**.

After a few seconds, depending on the load on the system, windows will begin to appear on the screen when the start up procedure is complete the user will be faced with a screen display similar to that shown in figure 2.

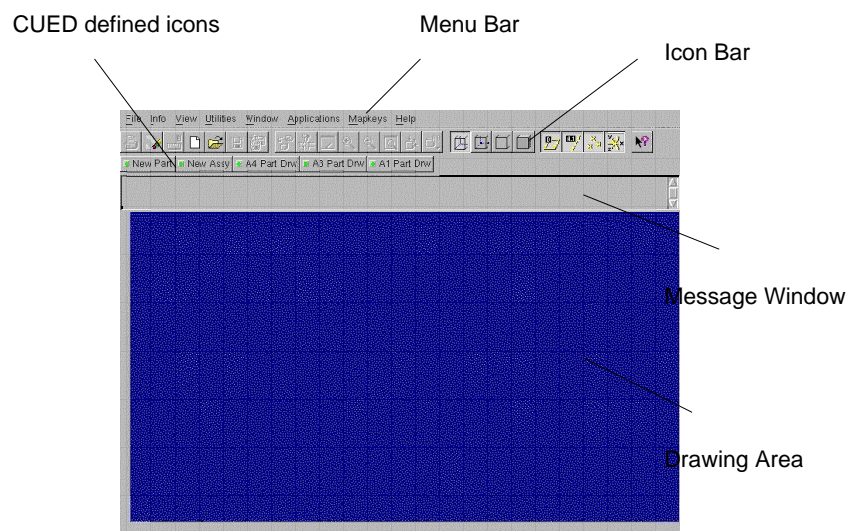


Figure 2: Basic/Start up Screen Layout

3 User Interface

3.1 Start-up Display

At startup a single model window is displayed which contains 5 main areas, each model which is loaded into the system will generate another dedicated window with the same five functional areas.

1. **Menu Bar** Ranged along the top of the screen are a number of pull down menus that access all the functions, many of which are duplicated in secondary context sensitive menus that appear during normal operation. The general operation is similar to that found in any windows compliant software. N.B. When selected the action of the menu item is displayed in the note line at the base of the main display area.

File File management sub menus - Including the ability to import and export foreign file formats.

Edit Allows modification/redefinition etc of items and also a restricted 'undo' function.

View Access to functions to change the display parameters, including orientation, colour, transparency and resolution. (A number of these functions are also available on the icon bar, see below)

Insert Gives direct access to the constructional tools, e.g. create holes, protrusions.

(Sketch) (N.B. Only visible in Sketch mode) Access to functions used while in sketcher mode, e.g. draw line, align, modify.

analysis Functions to investigate dimension and mass properties of a part. (In multi-body assemblies, interference information is available here)

Info Information about the construction, parent-child relationships etc of the current model.

Applications Access to additional PTC applications, see other manuals/help files for further information.

Tools Access to functions to change the user interface, not normally required for general use.

Window Allows user to switch input between windows etc. N.B. To change focus to a particular part window press CTRL-A while the mouse pointer is over that window.

CUED Menu access to a limited number of CUED/user defined Mapkeys

Help Access to ProEngineer manuals. Select **Pro/help** followed by **Products**, in the web browser to gain access to the online ProE manuals.

2. **Icon Bars** Icons that mimic the functions found in the pull down menus can be arranged in three positions below the pull down menus, and to the left and right of the main window. These can be simply adjusted by selecting the RHB or via the Utilities pulldown menu.

- Top Menu Bar By default the main file, view/display configuration function icons are arranged here. Information about a icon's action can be obtained by placing the mouse pointer over an icon. 'Greyed' icons indicate actions that are currently unavailable. N.B. During certain operations additional icons may appear.
- Right Side Icon Bar By default this icon bar displays access to the datum generation icons (identical function to the datum pull down menu see above) together with a number of sets of context sensitive icons.

3. **Message Window** This is a scroll-able window in which information on the system status and prompts for inputs are displayed. If the system is not responding look at this window as it normally indicates what is wrong. Typically it is waiting for an user input and the window has been over written.
4. **Display/Drawing Window** The large blue window is the main window in which the model or drawing is displayed. When a model is active the display can be manipulated using mouse/key combinations (see below) and commonly used commands can be accessed via floating menus (Press the RMB over the area). N.B. immediately below this window is a line which continuously displays simple help information, in particular button/menu item actions. General keyboard input is also shown here, notably mapkey shortcuts.
5. **Navigator** Area to the left of the main drawing screen that allows information about the model, directory structure etc to be displayed.
6. **Browser** Embedded web browser, accessible by clicking on the small arrows ('sash controls') at the righthand edge of the Navigator area.

3.2 Part(/Assembly) menus and Model Tree

As soon as a part (or assembly) is loaded, or a new entity started additional information will appear on the screen, notably

1. Additional icons in the top and right hand icon areas associated with image control and feature construction.

In main cases clicking on a top level icon will initiate a series of sub menus/icons. In many cases Pro/E will make educated guesses as to the users selection in these menus, to save time moving the mouse the
2. The Model Tree window gives a graphical representation of the method of the part construction, i.e. the order of Features. The tree can also be used to select individual or groups of features as a alternative to selection in the drawing window. This functionality becomes increasingly useful as the model complexity increases. Features can also be moved in the tree with a simple drag and drop operation provided there are no parent-child conflicts.

N.B. Quick access to feature modifying functions can be gained via the RMB whilst over the Model Tree.

The layout and information displayed is user definable via the icons and pulldown menus at the top of the window.

You should bear in mind that Pro/E is a specialist package, and the designers who use it would normally do so full time, after an extended training period: so don't be surprised if you find it more difficult to use than other CAD software that you may already know.

Central to the use of the package is the ability to obtain the best view of the object you are constructing, e.g. the orientation and the display mode. Access to commands effecting the display are concentrated under the View pull down menu and the top icon bar. In addition the orientation can be manipulated with a combination of the Ctrl/Shift keys and the mouse.

<u>Key/Mouse</u>	<u>Action</u>
(N.B. The Ctrl key can normally be released once the action has been initiated.)	
Ctrl - Middle Mouse Button	Zoom in/out
- Middle Mouse Button	Spin
Shift - Right Mouse Button	Translate

3.3 Sketcher

Central to the use of the 'sketched' features is the use of the sketcher or sketch mode. This is basically a 2-D drawing program which allows profiles to be drawn that are subsequently extruded/revolved/swept etc through 3-D space to modify the item being constructed.

Built into the package is knowledge of how things are typically constructed, e.g. lines are often parallel, similarly sized sketched entities are likely to be identical. The system uses these 'Assumptions' to apply overridable 'Constraints' to the sketch, e.g. vertical/horizontal line, to simplify the drawing the profile.

It should be noted that the assumptions taken by the system are partial dependent on the screen scale.

- **Sketching Tools**

The sketcher has many commands/functions that are accessible either from the SKETCH pull-down menu, via the side icon bar or from context sensitive floating menus accessed via the RMB. In addition a number are mapped to standard CUED hotkey sequences, see below.

The main drawing commands are found in the following sub-menus:

GEOMETRY **Mouse Sketch, Lines, Arc, Circle, Rectangle and Point.**

GEOM TOOLS **Intersect, Trim, Divide, Use edge, Mirror and Move.**

SEC INFO information about the sketched entities, including intersect and tangent points, angles, distances etc.

- **Sketcher Assumptions**

The knowledge model underpinning both operational modes attempts to apply constraints based on the following assumptions about the users intent:

Equal radii and diameters, symmetry, horizontal and vertical lines, parallel and perpendicular lines, tangency, 90 and 180 degrees arcs, collinearity, equal segment lengths, point entities lying on other entities, and centres lying on same horizontal or vertical.

More detailed information on these assumptions can be found in the user manuals.

- **Sketcher Tips**

- Keep sketches simple, you can add other features later.
- Use sketcher grid to aid construction (can also set grid snap from the UTILITIES → ENVIRONMENT menu)
- Do not use **Modify**, i.e. apply physical dimensions, until a section is (nearly) complete.
With the intent manager inactive
- Create sketches in steps, and regenerate each step as you progress.
- Use **Unregenerate** to restore sections to their previous state.

3.4 Mapkeys - Hotkeys/Keyboard Shorts

The complexity of a full function CAD package means that it can be difficult/time consuming to access a particular command/set of commands. ProE offers a hot key functionality in addition to the pulldown menus/icons and floating menus to aid efficiency. There are a set of CUED standard hot key sequences that should be available on all systems. Most are two character commands that can be most easily accessed assuming a right hand mouse operation and a left hand on the keyboard. (See the table at the end of this document)

Figure 3: List of the standard MDP mapkeys.

CUED Design Office - ProEngineer Wildfire v.1 - Hotkey Defaults															
<p style="text-align: right;">pjgt2@eng.cam.ac.uk Rev17 15/01/04</p>															
General Commands	aa Okay accept	dd Done	dr Done Return	ds Done Select	ff Flip direction	fx/fq Feature eXit	ra Regenerate All	rg ReGenerate	xf cancel Feature	ee Edit	ef Edit Feature	sc Set Colour			
File Options	fa Find Assembly	fd Find Drawing	fw Find part	na New Assembly	nw New Part	nd New Drawing	cw Change Window	qe/se Quit / save Erase	qp Quit / save Purge	qs Quit & Save	qw Quit Window	sp Save Purge	sw Save Window	cd/wd Change Directory	wn Window New
Environment Options Normally preceded by 'e' (Environment)	ew Wireframe	eh Hidden line	en No hidden line	es/vs Shaded	ed display Datum Planes	ea display Axes	ec display Coord Sys	e\ display Spin Center	ep datum Points	er Reset/toggle datums	et Enable Transparency	te disEnable Transparency	gs Grid Snap	dt Default Tiff colour map	at Alternate Tiff colour map
View Options Normally preceded by 'v' (View)	va All/refit	vr Repaint	vd Default	rv reset	vz Zoom	vp/zv Previous	v\ spin	ve View Explode	ev unExplode View	vf Front	vt Top	vm Bottom	v> right	v< left	vb Back
Blank/Display Layers Normally preceded by 'b'-(Blank) 'x'-(eXpose)	bb/xx all	ba/xa Axes	bc/xc Cuts	bd/xd Datum Planes	bh/xh Holes	bn/xn Notes	bt/xt cosmetic e.g. Threads	by/xy sYmbols	bz/xz Dimensions	F1 / F8 xDatum Fn Plane					
Sketcher Options Normally preceded by 's' (Sketcher)	sa Sketcher Alignment	sd Sketcher Dimension	se use Edge	sf Fillet	scl/cl Centerline	st Trim	sx delete	xs undelete	sq Sketcher Quit	sr Reset view	ss Start Sketcher	sv Sketch View (2D)			
Create Options Normally preceded by 'c' (Create)	ch Create Hole	cr Create Revolve	cc Create Chamfer	cs Create Shell	ce Create Extrude			slh Centerline Horizontal	slv Centerline Vertical	sm Move					
Detail Options Normally preceded by 'd' (Detail)	da show Axis	ad erase Axis	dc Clip line	dd show dimension	de modify/Edit text	df Flip Arrows	dl create note with Leader	dm Move	dn create Note	dp mod attach (Pointer)	da Display Axis	de modify/Edit text		dt move Text	dx Erase dimension
Plotter/Printer Options Normally preceded by 'p' (Printer)	dm Move	df Flip Arrows	dv switch View	dc Clip line	dn create Note	dl create note with Leader	dx Erase dimension	da show Axis	dp mod attach (Pointer)	da Display Axis	de modify/Edit text		dt move Text	dt move Text	dt move Text
Detail Options Normally preceded by 'd' (Detail)	dm Move	df Flip Arrows	dv switch View	dc Clip line	dn create Note	dl create note with Leader	dx Erase dimension	da show Axis	dp mod attach (Pointer)	da Display Axis	de modify/Edit text		dt move Text	dt move Text	dt move Text

N.B. Hotkey text appears at the bottom of the drawing window, use <DELETE> or <CR> to erase errors