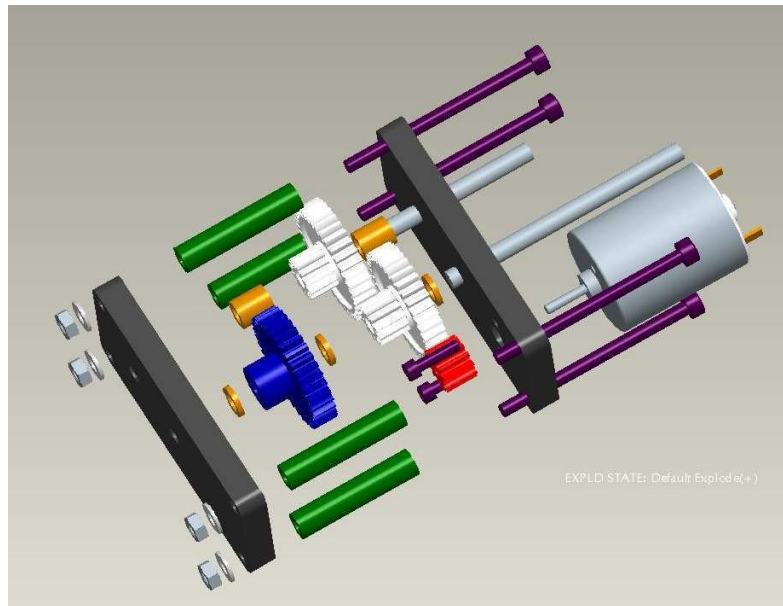# Pro/ENGINEER – Wildfire 3

Introduction Manual

This manual is designed to give the user a very basic understanding of the philosophy of the Pro-Engineer Wildfire (ver. 3.0) 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.

P J G Long   November 2007
S. Tebboth

Contents

Construction of a Part

Workshop   ProEngineer

Select stock   Sketch Initial
Material   Cross-section

Cutoff length of   Use Protrusion
Stock Material   (Extrude, Revolve etc)
                                                                to generate 3-D workpiece

                    Hole
                    {P&P}   Cut
                               (Sketched)
            Slot   Round/Fillet
          (Sketched)   (P&P)
Turn. Mill, Drill, etc   Use Pick and place,
to shape part   Sketched Feature
                                                                to create finished part
                                         Protrusion
                                         (Sketched)
                    Chamfer
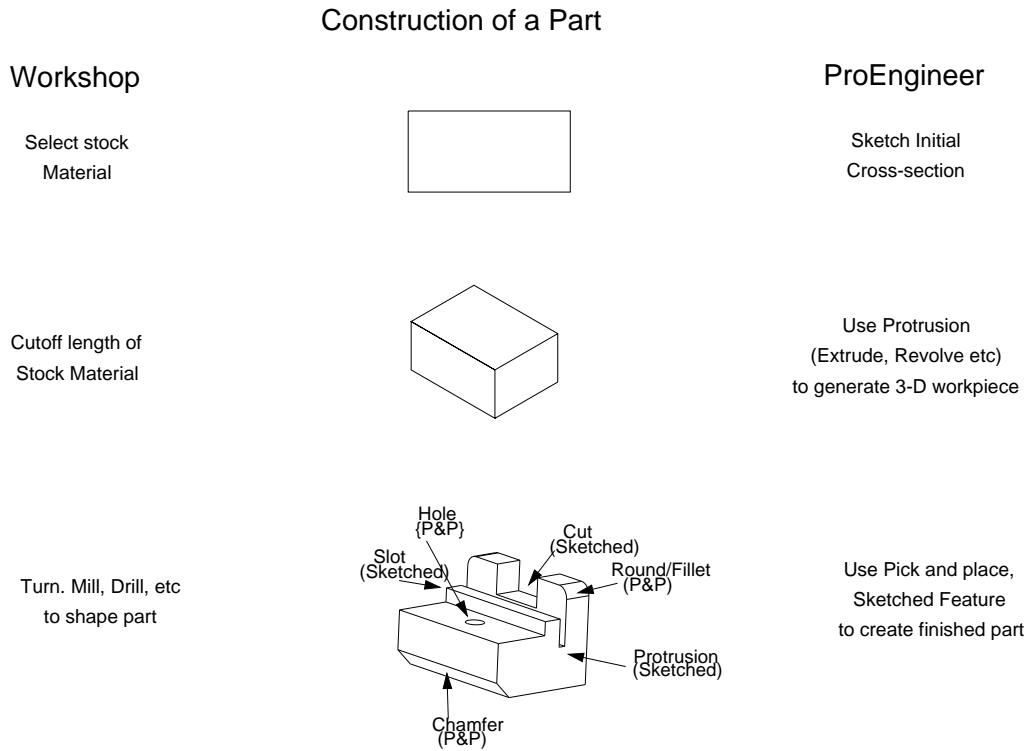   (P&P)

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 EDaL disc, which is similar to many other systems. If you have difficulties 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/E 3** from the **Programs→Drawing/CAD** 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 **wildfire3** 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.
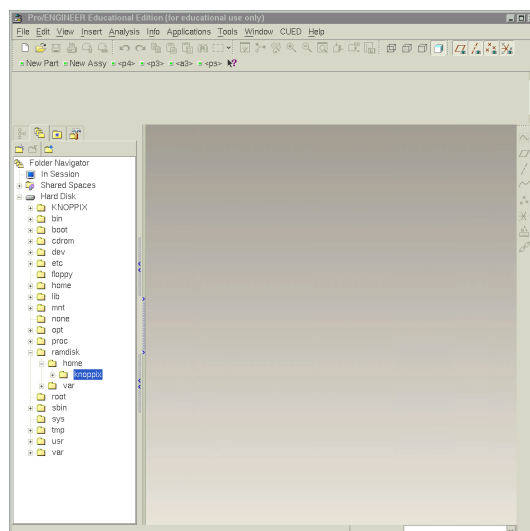

Figure 2: Basic/Start up Screen Layout

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 duplcated 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 simmply adjusted by selecting the RHB or via the Utilities puldown 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 geration 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 accesed 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) in 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.

Key/Mouse		Action
(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 overidable '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 accesed 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 sum-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

pjgl2@eng.cam.ac.uk
Rev.17   15/01/04

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

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