

Getting started with Pro Engineer Wildfire.

Using the Pro Engineer work environment.

1.1. Start Pro Engineer.



Pro Engineer (Wildfire 3.0) can be started with the icon **Pro ENGINEER** or by selecting Start>Programs>PTC>Pro Engineer>Pro Engineer.

Wildfire will open, and the files you make will be stored in the working directory. A different working directory can be selected in File>Set working directory.

The working directory is very important, since all files of a project will be stored in this single directory. It is the project entry point on your hard disk for Pro Engineer.

Note: Trail files (log files) that contain all the executed commands and data you use in a ProE session, are stored in the working directory linked to the window-icon.

1.2. Screen layout and userinterface.

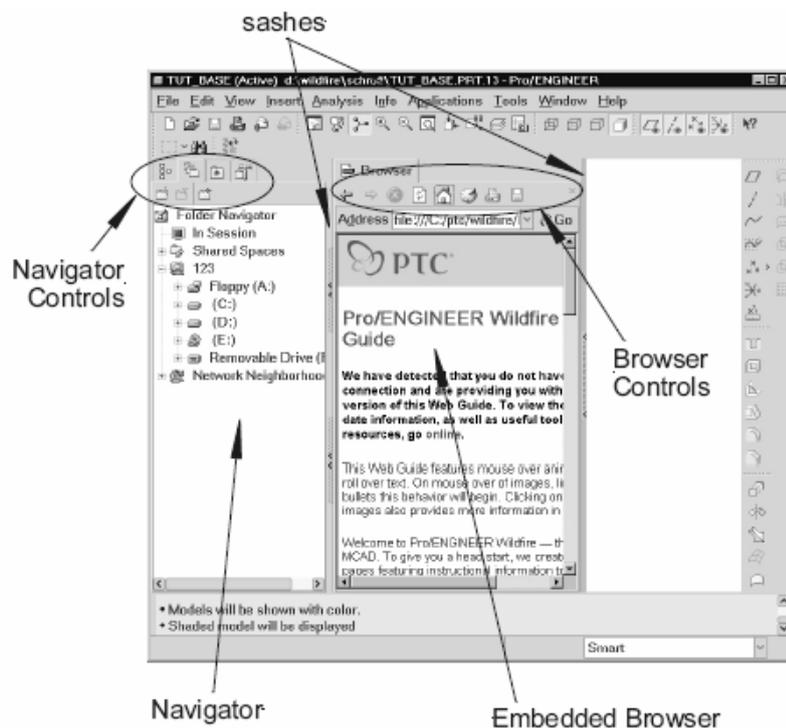


Figure 1 Screen layout

Wildfire is a separate windows-system on top of the operating system of the computer. Every file (object in ProE) you open is a new window.

The Navigator shows the directory tree of the computer when you start ProE. The Navigator will show the part/assembly model tree when in part model mode and assembly mode. It can also reflect the Layer tree of an object once you open a model.

Attached to the Navigator is an embedded (internet)browser. In this browser help screens are available, info screens will appear in Wildfire and if you are online, the internet is available in the embedded browser.

By clicking the sashes you can minimalise the browser and model tree.

When you open an object (File>New or File>Open) the browser space is substituted by the main graphics area.

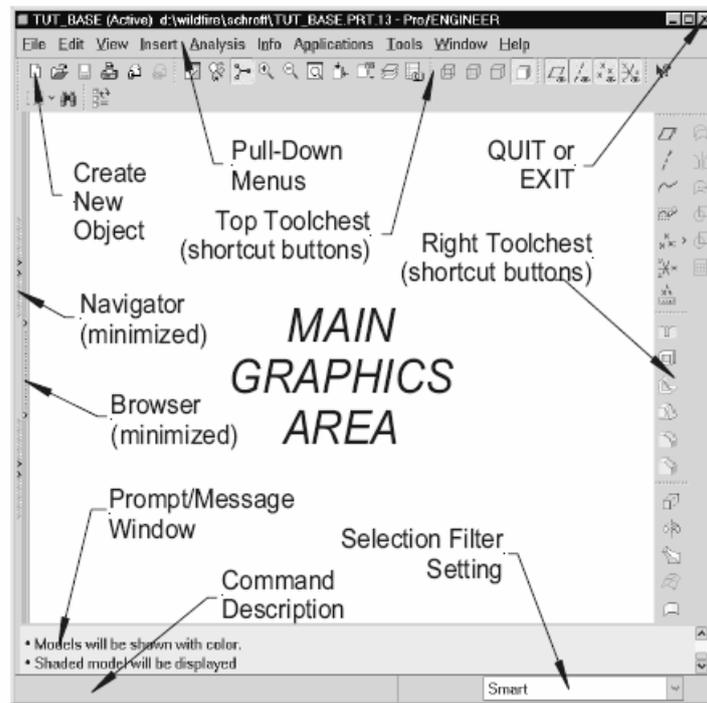


Figure 2 Screen layout in part mode.

As a good practice you better not maximise the ProE window. Sometimes extra input frames appear in the upper right corner of the screen. These frames could disappear behind the main window, and while ProE is just waiting for input, one can have the impression that it stalled.

Next to the pull down-menu also customisable icon tool chests are available. Non-selectable icons are greyed out. This way the design proces is streamlined.

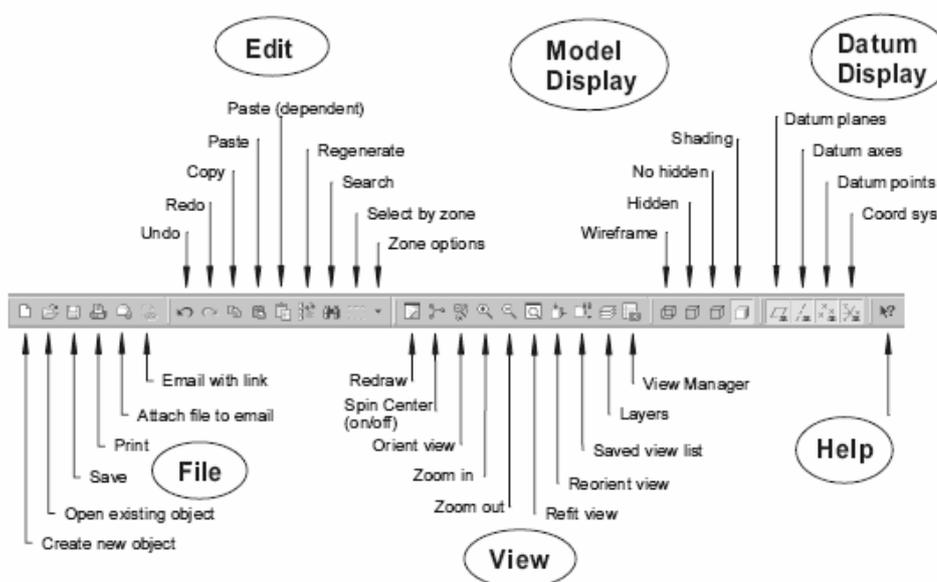


Figure 3 Standard top toolchest

Controlling Wildfire is done in a head-up design manner. This means that you can do anything you'd liked by using a series of mouse clicks, without looking at the keyboard.

Any mouse button has several functions. Most common ones are in the table underneath.

| Function | | Operation | Action |
|---|---|--|--|
| Selection (click left button) |  | LMB | entity or command under cursor selected |
| Direct View Control (drag holding middle button down) |  | MMB | Spin |
| | | Shift + MMB | Pan |
| | | Ctrl + MMB (drag vertical) | Zoom |
| | | Ctrl + MMB (drag horizontal) | Rotate around axis perpendicular to screen |
| |  | Roll MMB scroll wheel (if available) | Zoom |
| Pop-up Menus (click right button) |  | RMB with cursor over blank graphics window | launch context-sensitive pop-up menus |

Figure 4 Mouse button functions

1.3. Files and directories.

All files will be stored in the working directory, which is also the standard directory for Wildfire to look for objects.

In a larger project, you can extend the search over different directories by using option-file variables SEARCH_PATH and/or SEARCH_PATH_FILE. (consult option-file TOOLS>OPTIONS.)

Every object (part, assembly, drawing) you save in Wildfire will be stored in a new version of the file. They are number .1, .2,

These versions will be available until you delete them or purge them (File>Delete>Old Versions).

Purpose of this storing of different version/releases of an object is to use them as an alternative, and to return too when (fatal) errors would occur downstream in the design process.

Wildfire has no absolute undo command, and no time able autosave. The user is responsible for the save keeping of data.

Never change the name of an ProE object in windows explorer: ProE will not find these files anymore when used in an assembly.

1.4. Orientation of the object on the screen

Orientation of the objects is done by:

- Dynamic orientation, pan and zoom with the mouse (fig. 4 mouse button functions)
- Use of saved views (fig. 3 saved view list in the top tool chest): a saved view is an orientation in space and a zoom scale. Saved views are very useful in larger designs to shift from left to right in the design without having to zoom and pan several times. Views can be made and saved in the VIEW>View Manager>Orientation.
- By reorienting the view (fig 3 Reorient view in the top tool chest)
- With the view manager (fig 3 View manager in the top tool chest or View>View manager)

A view is the orientation of the object relative to the plane of the screen. Standard orientation in Wildfire is through the use of plane to look at, and another reference (plane, edge..) you point to a direction (top, bottom, left, right).

View you make in model and assembly mode, are available as views in drawing mode.

1.5. Model tree of a part.

The model tree reflects the making of the part object, and is used to rearrange the design sequence, edit feature dimensions, edit the definition of a feature and lots more.

With tree filters one can influence what is displayed in the model tree, like suppressed features/objects and annotations.

With tree columns one puts in more columns, each containing more information about the features (like feature number, used in error messages.)

When right clicking a feature one can:

- Edit: modify the dimensions of a feature
- Edit definition: edit the definition (options, sketch, dimensions) of a feature
- Delete: delete the feature (take care: all child-features will also be deleted!)
- Suppress: temporarily remove a feature from the model. The feature disappears as if it was never there, and isn't calculated anymore until it is resumed. Suppressed features can be resumed/unsuppressed to once again show them in the model.
- Rename: give the feature a name other the "extrude nr"
- (un)Hide: (un)blank a feature temporarily. A hidden feature will still be calculated in a regeneration.
- Pattern: multi-copy the feature

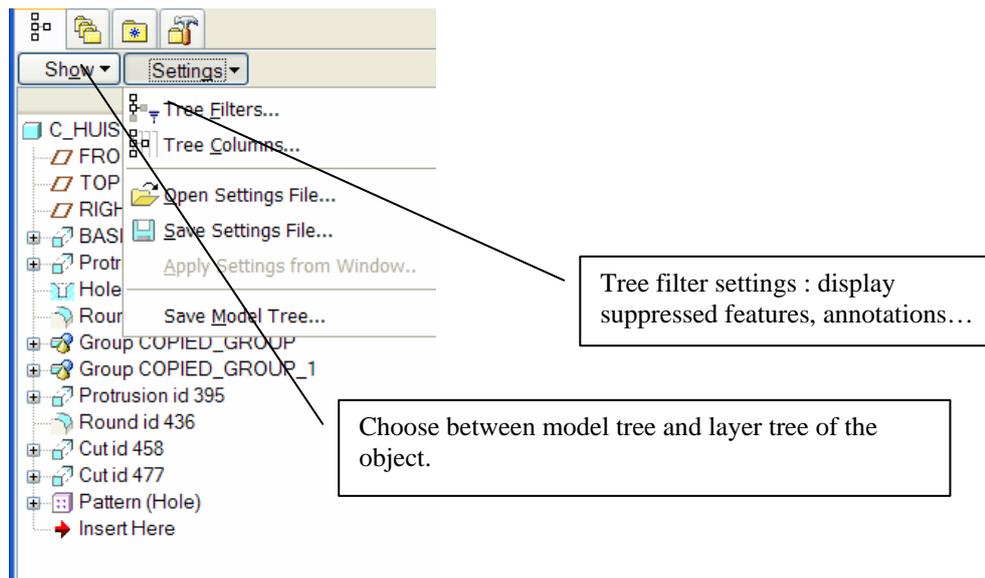


Figure 5 Model tree of a part

1.6. Model tree of an assembly

The model tree in an assembly reflects all the parts and sub-assemblies that make out the assembly.

Next to the component a small sign marks out the state of the component:

- (no sign) Fully constrained: the component is fully placed in space.
- (square) Packaged/partially placed: the component is in the assembly, but not fully placed in space
- ◻ (double square) Constrained to a packaged/partially placed part
- ◻ (square with dot) Constrained with connections: some DOF's are left open for movement.

When right clicking a component one can:

- Edit: modify the dimensions of an assembly constraint.
- Edit definition: edit the placement definition (constraints, connections) of a component
- Delete: delete the component in the assembly (take care: all dependant components will also be deleted!)
- Suppress: temporarily remove a component from the assembly. The component disappears as if it was never there. Suppressed components can be resumed/unsuppressed to once again show them in the assembly.
- (un)Hide: (un)blank a component temporarily. A hidden component will still be recalculated in regeneration.
- Pattern: multi-copy the component

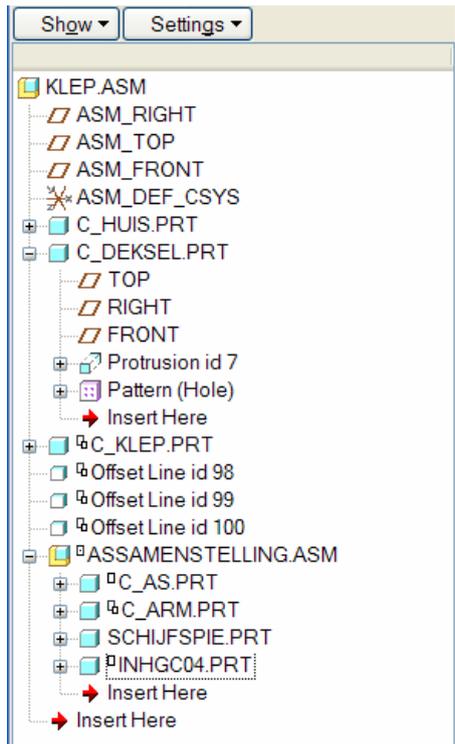


Figure 6 Model tree of an assembly

1.7. Help files

When designing you can always use the help-files.

Help can be used through:

- Help>Help Center
- Click the  button and the command you need help on.
- Right click a command and select Get Help.

Help files are stored locally on the pc, but when online you can also use the “Resource centre” at www.ptc.com. License holders can also use the knowledge database on www.ptc.com.

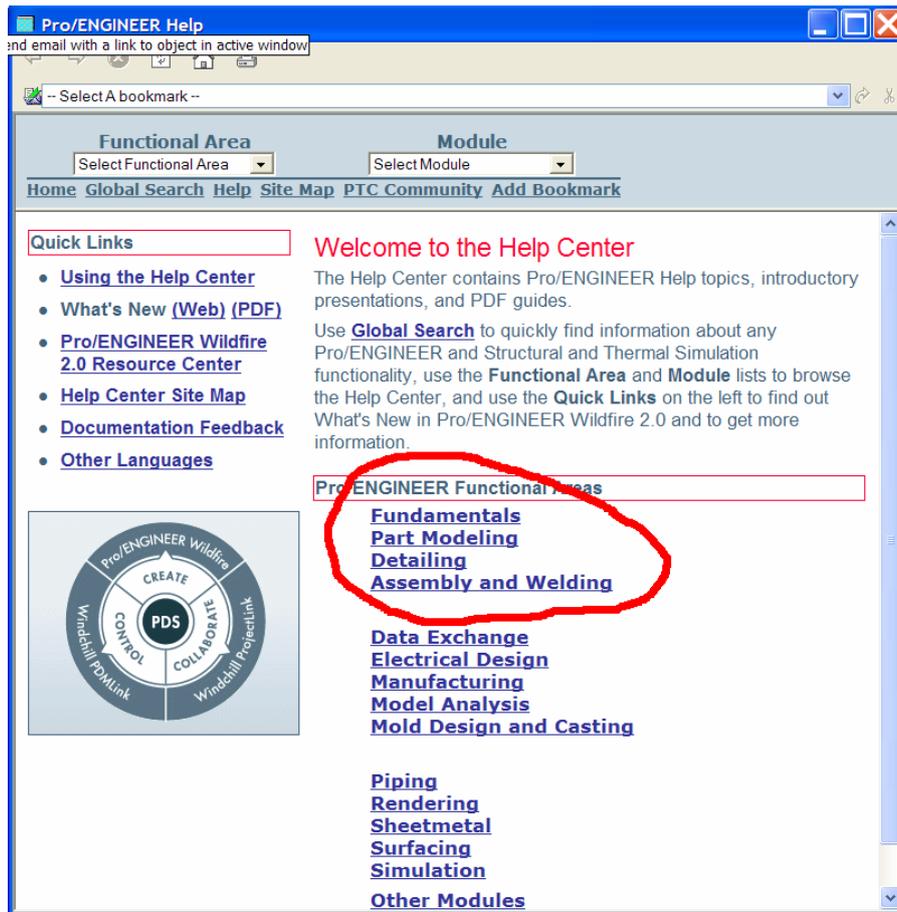


Figure 7 Wildfire help center.

The help centre is organised by module, and can be searched on keywords or by functional area. Most important functional areas are circled in red.

1.8. Printing

Anything you do in Pro E can be printed, from screen shots to shaded images and drawings. Most important is the printing of technical drawings. Printing is easily done using the Windows Print Manager. Wildfire will take care of standard line styles and line width. We only need to look after physical formats and scaling.

Quickest and best way to plot a technical drawing is with these options (in the format tab of the printing dialogue box):

- Full plot: scaling will be as selected in the drawing itself.
- With format: with the selected format (title block, frame) of the drawing.

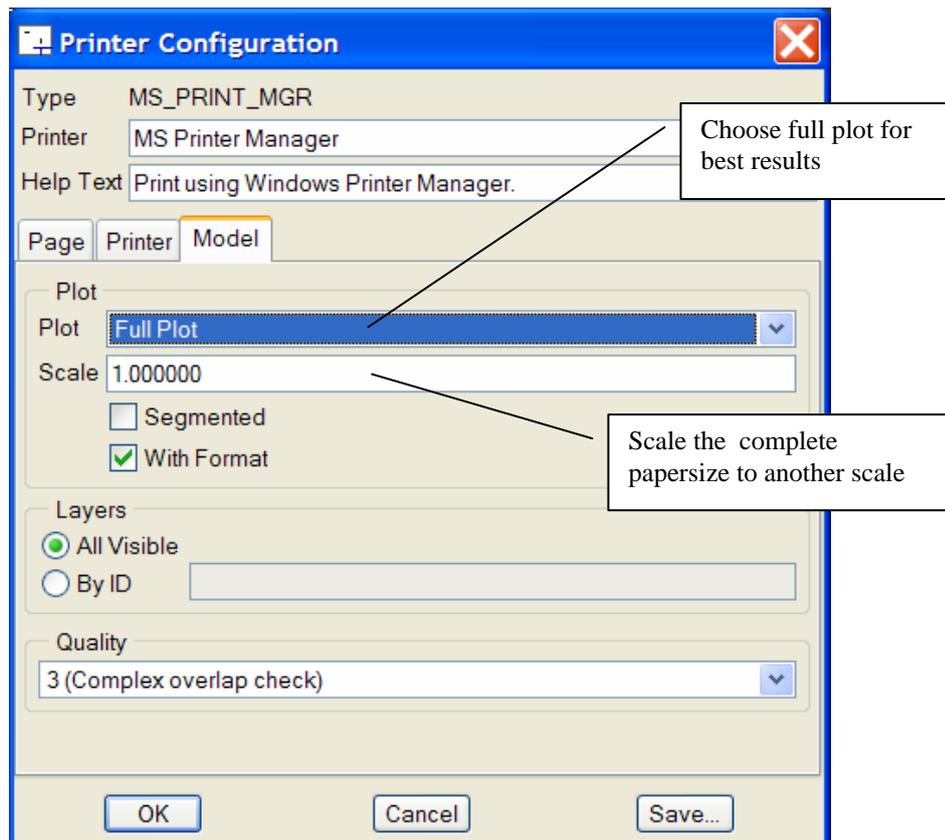


Figure 8 Printer box selections for optimal results

If you want to print on a smaller paper, select “Segmented plot”. Wildfire will cut the drawing in smaller parts, which can be printed on a smaller printer. You get a jigsaw like printout which you can assemble to a big paperformat.

1.9. Exiting Pro Engineer.

Exiting Pro Engineer is done File>Exit, or by closing the windows.

Remember to first save anything you need to save!

Wildfire will not autosave anything in a standard installation. It will not ask to save anything, so what you do not save, will be lost.