Concepts of Pro Engineer Wildfire.

1. Introduction.

Pro Engineer first became available in the late 1980's. Pro Engineer was way ahead of its time and the idea of having a powerful solid modelling tool that made it possible to change a design after completion was very exciting! The software is now on its 24th release as and its power and versatility is still unmatched in the CAD market.
The basis behind Pro/ENGINEER is that it is a single database, solid modelling tool. Unlike the earlier drafting tools such as AutoCAD, which created models in 3D with the use of wire frames and 3D faces, Pro/ENGINEER is a feature-based modeller. The solid model is comprised of various features that make up the final geometry. A part may consist of a solid block with many different cuts, holes, rounds and chamfers that complete the final model. Associativity has always been one of the key features of Pro/ENGINEER. What this means is that if a change is made anywhere in the product structure, such as the drawing or assembly, the changes are reflected everywhere. The solid model is the "master" file and all of the deliverables are merely "windows" looking in at the part file. This single database allows the user to make changes very easily without having to go back and update all the drawings and assemblies. However, when using a single database – feature driven modeller, care must be taken and up-front forethought is extremely important. Parent-Child relationships or dependencies among features can cause problems to the new user. If a newly created slot is dimensioned to the centre of a hole, then the hole is removed, what happens to the slot? Since the slot was dimensioned in relation to the hole, the slot is said to be a Child of the hole. This is not necessarily a bad thing, but the user must understand this relationship and try to minimize any of these types of problems before it becomes a major issue. The nice thing about Pro/ENGINEER is that it tries to force the user to build model the correct way. For example, when sketching the profile of a feature, Pro/ENGINEER will not let the user complete the feature if it does not have the appropriate dimension to build that feature. As a result, Pro/ENGINEER geometry is never under-dimensioned or over-dimensioned. The learning curve for Pro/ENGINEER is different for everybody. Some users can pick it up in a week while others may take several months.
In this chapter we describe some properties of ProE, and the basic workflow to illustrate on how to get from a rough sketch to a part.

2. Four important properties of Pro Engineer.

2.1. Feature-Based Modelling.

Earlier drafting tools such as AutoCAD, created models in 2D (and 3D) with the use of wireframes (and 3D faces). Through the use of wire frames, it's the user's responsibility to put in the design exact as many lines, circles and other entities to describe the design. Tools for making geometry just use general mathematical terminology.
Pro/ENGINEER is a feature-based modeller. The solid model is constructed through the use of various features that make up the final geometry. A part may consist of a solid block with many different holes, rounds and chamfers that complete the final model.
A feature is a geometrical entity with a predefined physical shape, but with changing dimensions.

For example, a cylindrical pen is a circular protrusion/extrusion driven by an axis-line. Afterwards this shaft like shape can be rounded or drilled (with holes). All of these features can later be altered in dimension, position and even shape.

There are 3 main categories of features:
- Sketched features:
  o In a sketched feature the base is a 2D sketch which will be extruded, revolved, swept or blend across space to produce a solid.
- Engineering features:
  o Engineering features are features with a prescribed form, such as a hole, a round, a shelling…
- Datum features:
  o Datum features are work features which do not generate geometry, but serve as references items in space to connect or orient geometry on. Planes, axis, points, curves and user coordinate systems are all datum features.
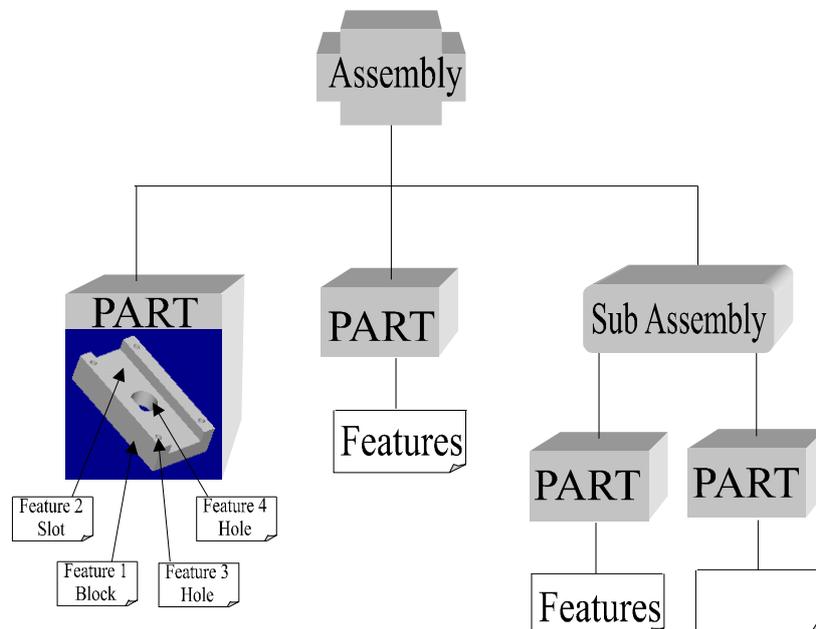 o

Figure 1 Part building with features.

2.2. Parametric Design

In a drafting system such as Autocad, a dimension is the result of the measuring of entities or position of entities. This measure will only change when the object or position of it changes. Changes are made with special commands.

In the parametric design, dimensions drive geometry. This means, that when changing a dimension, it really changes the size of the object. This change is reflected throughout the complete design.

A dimension can – by the nature of its definition – also be a mathematical function or can be a function of other dimensions, constants or parameters. This use of functions is made through the use of RELATIONS.
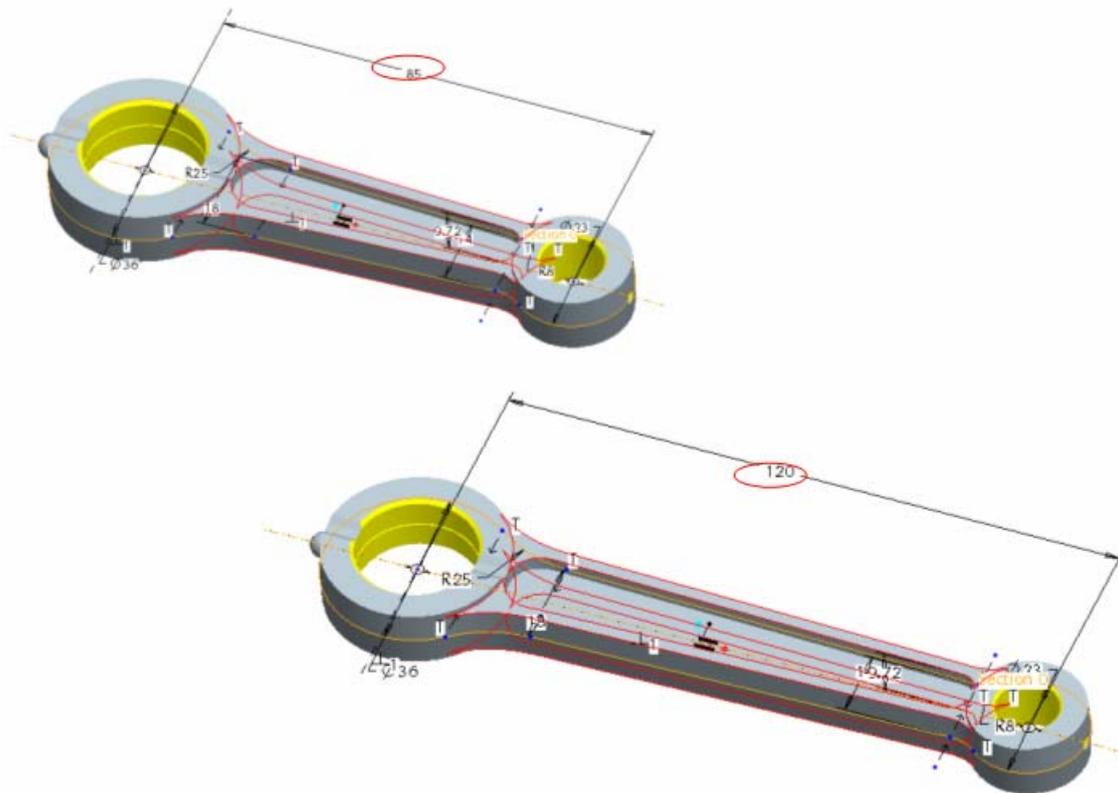
Figure 2 Changing of a dimension changes geometry

2.3. Associativity.

Associativity still is one of the key features of Pro/ENGINEER. Associative means is that if a change is made anywhere in the object structure, such as the drawing or assembly, the changes are reflected throughout the complete structure. The solid model is the "master" file and all of the deliverables are merely "windows" looking in at the part file. This single database concept allows the user to make changes very easily without having to go back and update all the drawings and assemblies. However, when using a feature driven modeller such as Pro Engineer, care must be taken and up-front forethought is extremely important. Parent-Child relationships, this is dependencies among features, drive the complete design.
For instance, if a newly created slot is dimensioned to the centre of a hole, then the hole is removed, what happens to the slot? Since the slot was dimensioned in relation to the hole, the slot is said to be a Child of the hole. It will therefore fail to be regenerated when the hole is gone.
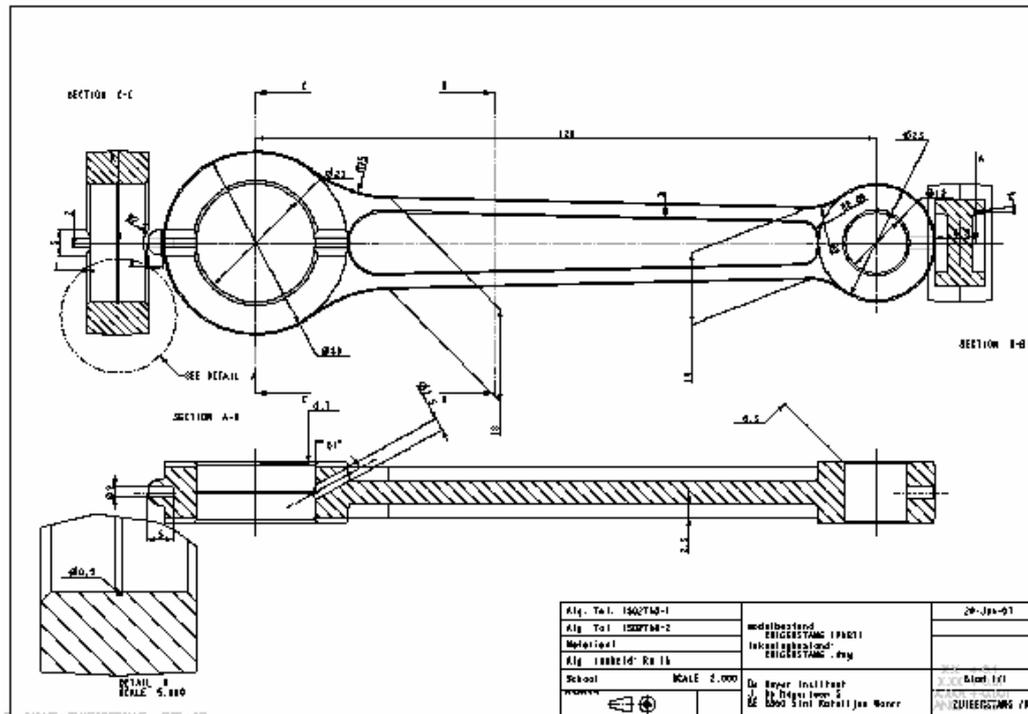
Figure 3 Drawing automatically changed after changes to the model

2.4. Design intent.

Design intent really is the final goal a user has with its design.
Pro/ENGINEER tries to capture the intent of the design and as such tries to force the user to build models in a correct way. When sketching the profile of a feature, Pro/ENGINEER will not let the user complete the feature if it does not have the appropriate dimension to build that feature. As a result, Pro/ENGINEER geometry is never under-dimensioned or over-dimensioned, but is said to be fully constrained.
Pro/ENGINEER will try to help the user and often proposes choices, or has automated context sensitive behaviour, which results in lesser mouse clicks, and lesser explicit choices to be made by the users.
Some examples of this capturing of the design intent:
In sketcher mode ProE will try to find a logical form (vertical, horizontal, parallel shapes) in the crude shape the user makes.
In part mode, ProE will try to predict what the user is up to. If you want to create a plane, and you select a plane, proE will suggest an offset plane. If you select 3 points, ProE suggest a plane through the 3 points. If you select a plane and an axis, ProE suggests a plane through the axis, angled to the original selected plane.
When assembling parts, ProE suggests assembling constrains depending on the entity the user selects.

3. Part building.

A part which is modelled in ProE consists of a base feature – which generates some solid material – and extra features to put extra material on, or to take material away.
The part is the kernel part from which is derived technical drawings, which is used to build assemblies, or can be machined in ProNC to make tool paths.
The flow to get from scratch to part is roughly described here.

3.1. Creation of the base feature.

To make a sketched feature, we need a 2D-sketch. This is a (rough) sketch which describes the topology of the form of the feature. Since it is a 2D-sketch, we need to orient the sketch in space, on a datum plane.
Topology of a sketch describes the number of lines and arcs (and other geometric elements) that makes out the sketch. The sketch in figure 2 consists out of 11 lines, and 1 arc.
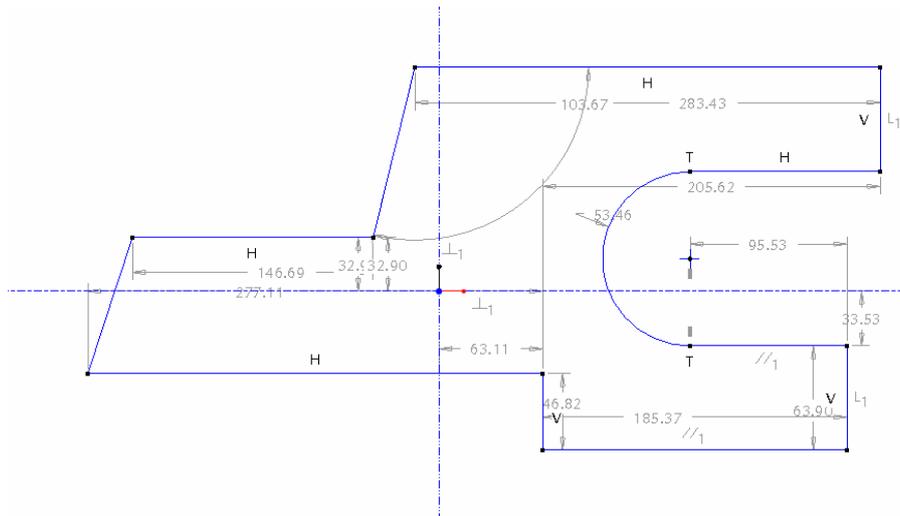

Figure 4 A rough sketch.

A rough sketch can be converted in geometry by using geometrical constraints, and dimensions.
Geometrical constraints available in ProE are listed in table 1 and serve to make relations between geometrical elements. For example, "parallel lines" makes 2 lines parallel to each other.
On the sketch there are marked out by constraint symbol, which are also listed in table 1.

When the sketch is constrained as needed for the intended design intent, the next step is to make the scheme of dimensions as is needed on the technical drawing.
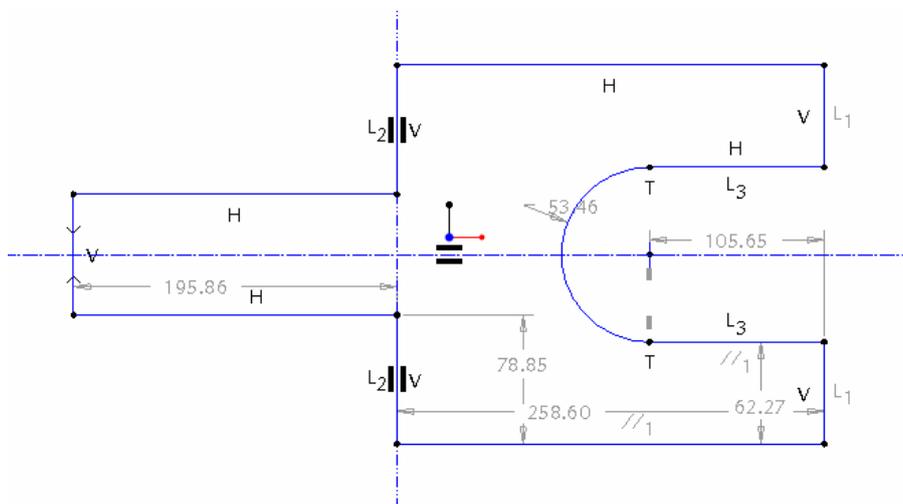

Figure 5 Sketch with constraints (perpendicular, collinear, ...)

Table 1 Available geometrical constraints

| Constraint	Symbol
Midpoint	M
Same points	▫
Horizontal entities	H
Vertical entities	V
Point on entity	–O– – –
Tangent entities	T
Perpendicular entities	⊥
Parallel lines	//₁
Equal radii	R with an index in subscript
Line segments with equal lengths	L with an index in subscript (for example, L1)
Symmetry	→•←
Entities are lined up horizontally or vertically	--┆
Collinear	=
Alignment	Symbol for the appropriate alignment type.
Use Edge/Offset Edge	— ○
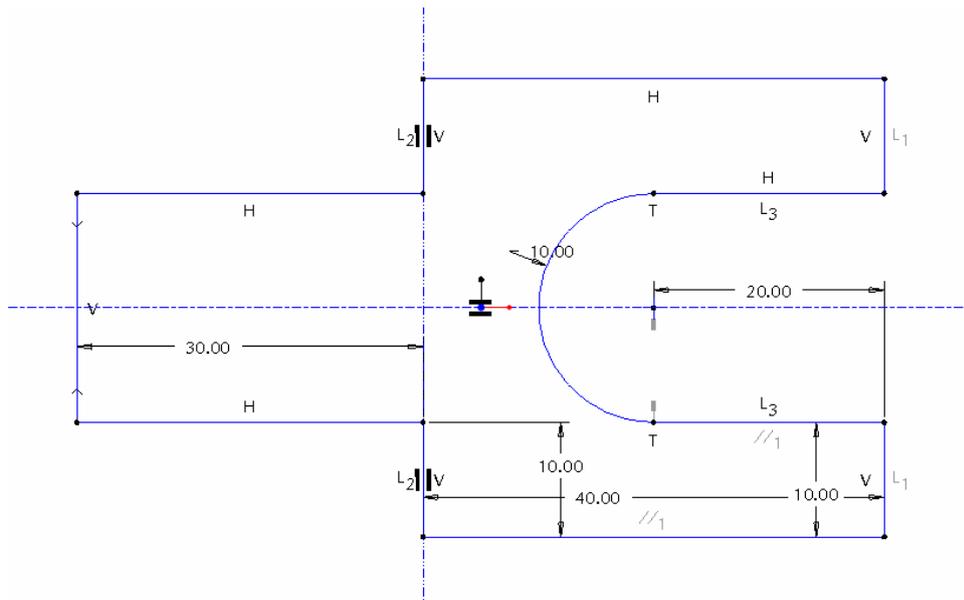
Figure 6 Fully constrained sketch with dimensions.

When the sketch is fully constrained, and dimensioned, we can make a solid by extruding, revolving or sweeping the sketch in space.
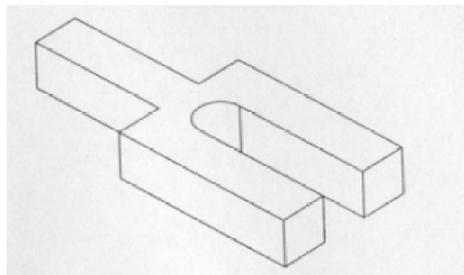

Figure 7 Feature extrude

Important in sketches
- Put symmetry lines in first - Sketch you geometry - Implement constraints - Use the dimension scheme you would like on your technical drawings - Modify dimension values as desired.

3.2. Other features.

After the base feature is made, we can put in extra features to make more solid material, or to remove material. Extra features include engineering features like holes, rounds, chamfers, shells.
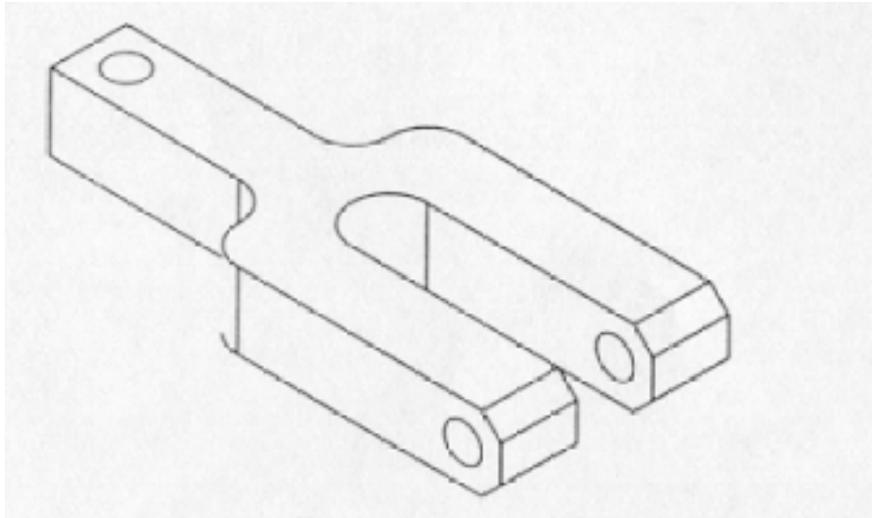

Figure 8 Extra features: rounds, chamfer, holes

3.3. Finished model: applications.

When the part is finished it can be used to derive technical drawings, to make tool paths, to study its strength by means of FEM, to make dynamic analysis, etcetera.
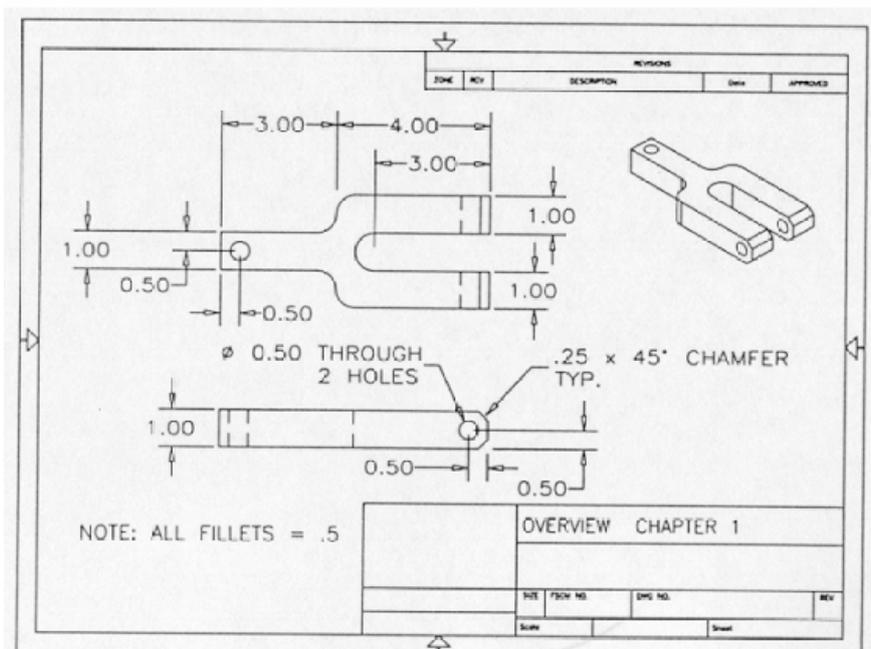

Figure 9 technical drawing of the part

4. Tips and good practices while modelling.

- Always use standard datum-planes to position your object in space.
- Finish as much as possible of the outside of the object – without rounds or chamfers, before starting an the inner forms.
- For drafts, rounds and chamfers, use the appropriate features.
- After finishing the outside, then start working on the inside of your design.
- Use simple sketches. Better use one more sketch instead of a very complex one.
- In sketches:
    - Select reference lines with care: they will be they dimensional references (and therefore directly connected to the production of the final part).
    - Put symmetry lines in first
    - Sketch you geometry
    - Implement constraints
    - Use the dimension scheme you would like on your technical drawings
    - Modify dimension values as desired.
- Keep features which are independent in your design, apart, without parent-child relations.
- If there is symmetry in your design, implement it.
- For holes and cylindrical cuts, use the feature hole.
- Put logic in your design: give names, use parameters and relations.