Layouts

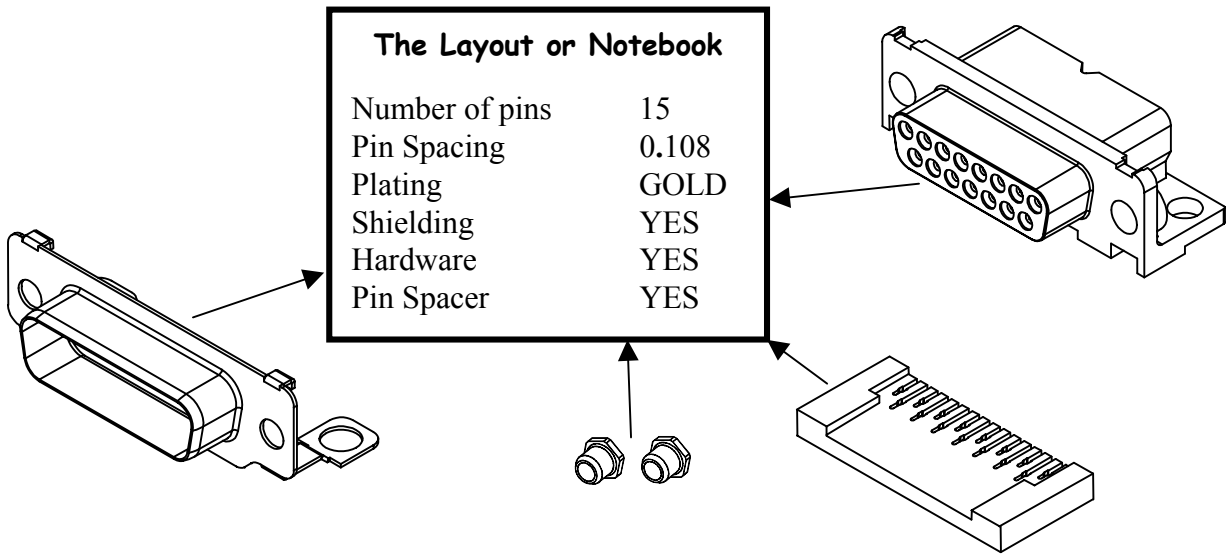
Introduction

Pro/NOTEBOOK is used to create 'layouts'. Layouts are like engineering notebooks, containing key design information and rules. Layouts allow you to create parameters that control the model, just like any other parameter. What's different about the layout is that it is not a model, and it's not a 'layout' at all. This is *not* similar to the old drafting term 'layout'.

Part and assembly models can both 'reference' layouts. When a part references a layout, it 'knows about' all the parameters in the layout. If the parameters in the layout are modified, the parts referencing the layout will update when they are regenerated.

Think of the layout as a repository for all the parameters you need for a project. As the project develops, parameters are added to the layout. As models are developed, they reference the layout. That way, if key design parameters are used by more than one part in the project, the parameter only needs to be modified in the layout, and the entire project will update.

For example, the connector assembly shown below has many components. The customer specifies the number of pins and the desired plating. Other parameters control optional shields, hardware and other components of the assembly. The layout contains all these parameters, and each component of the assembly references this layout. To create a bigger size connector, simply modify the number of pins in the layout and the system builds you a connector that size.



Advantages of Layouts

Layouts are extremely versatile and powerful Pro/ENGINEER objects. With some imagination and perseverance, the possibilities for using layouts are endless. When combined with parameters, relations, and Pro/PROGRAM, layouts can be used to make design decisions based on known rules. Layouts can also be used to create and modify models based on global 'yes/no' and 'numeric' parameters. Some common advantages of layouts are listed below.

- Consolidates engineering rules and critical requirements of the design into one central location
- Used to develop and maintain the basic part geometry for component of assemblies
- Ensures proper fits and sizes of components in assemblies
- Allows the user to drive any number of assemblies, parts and drawings, from one location
- Stored with the design objects in the database
- Provides information about the objects where global parameters are used
- Allows 'automatic' assembly of components, useful in design automation

Considerations for Using Layouts

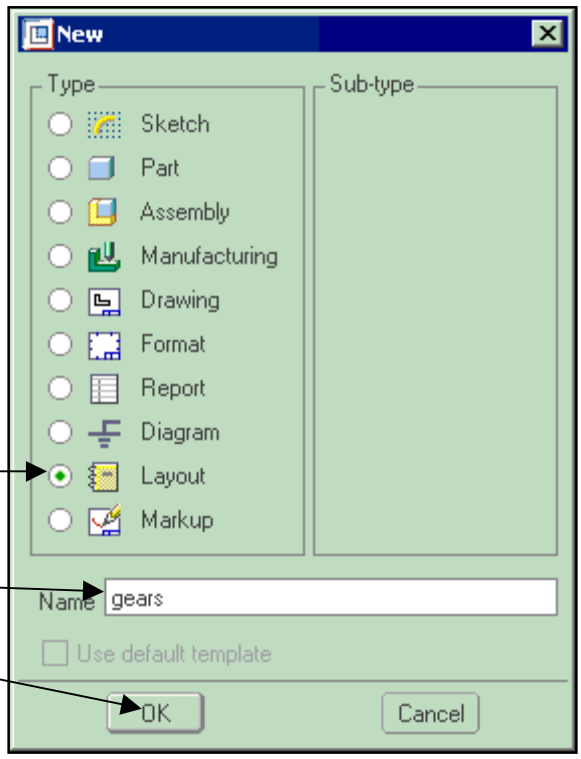
At some point in the design process, it can be useful to capture all of the critical design requirements and place them in a layout. Typically this includes parameters and relations, but can also include simple sketches, balloons, and notes, to make the layout easier to use and maintain. This type of information is cosmetic and is used for annotation of the layout only.

The project leader or engineer can be responsible for the layout, controlling key design parameters for the entire design team. As each team member creates their models, they reference the layout. This approach allows true electronic concurrent or simultaneous engineering to be performed using Pro/ENGINEER.

Although not always possible in today's fast paced work environment, lots of preparation and sound design rules can help to ensure the design intent is carried through, and the design requirements of the project are fulfilled. Using a layout can make it happen.

Creating Layouts

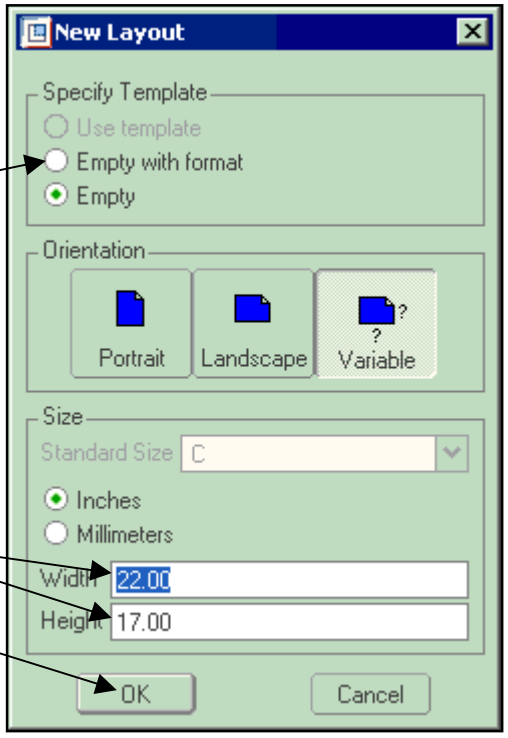
To create a layout, pick **File, New**, pick the **Layout** button, and enter a name in the **New** dialog box. The **New Layout** dialog is displayed and you can choose a drawing format or specify a size for the layout as shown below.



Pick the **Layout** button

Enter the name for the layout here

Pick **OK**



Pick **Empty with format** to use a drawing format for the layout

When using **Variable**, enter the size of the layout's sheet here

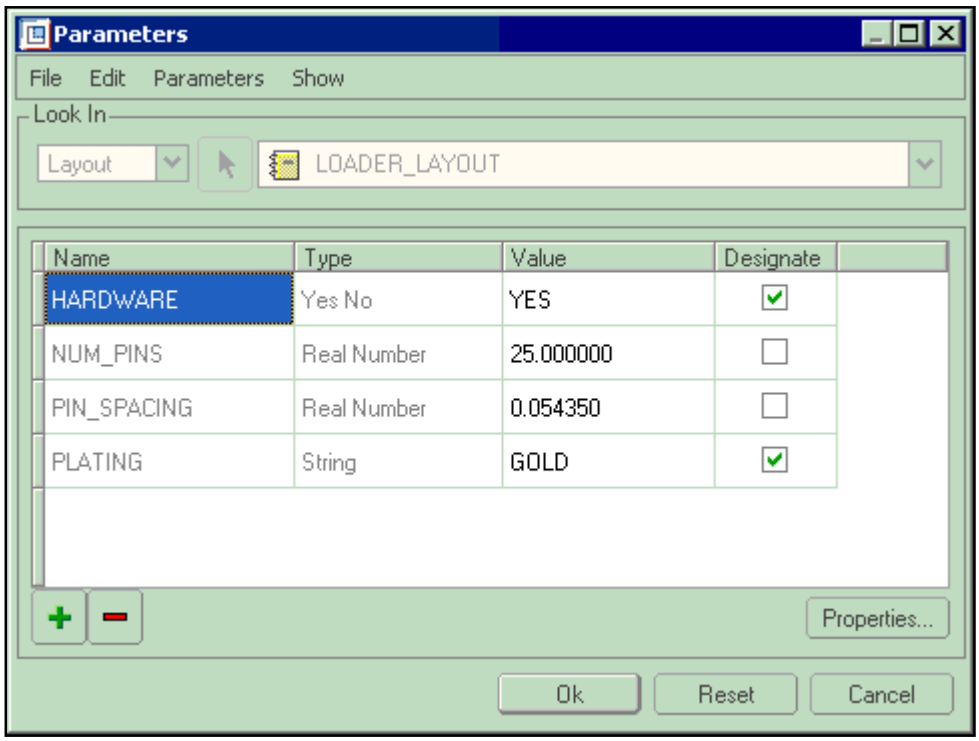
Pick **OK**

Creating Layout Parameters

The main use of *Layouts* are to contain important design parameters. Pick **Tools, Parameters** to create and modify parameters in a layout. The layout **Parameter** dialog box (shown below) is similar to the **Parameter** dialog box used in part and assembly models (see pages xxx-xxx).

Pick the 'plus' sign to add a parameter, pick the 'minus' sign to delete a parameter. The name, type, and values are entered, and the parameters can be 'designated' for use with Pro/Intralink.

When parameters are created in a layout, they are considered 'global' parameters. This is because they are used *outside* the layout by parts and assembly models that reference the layout.



The Layout Parameter Dialog Box

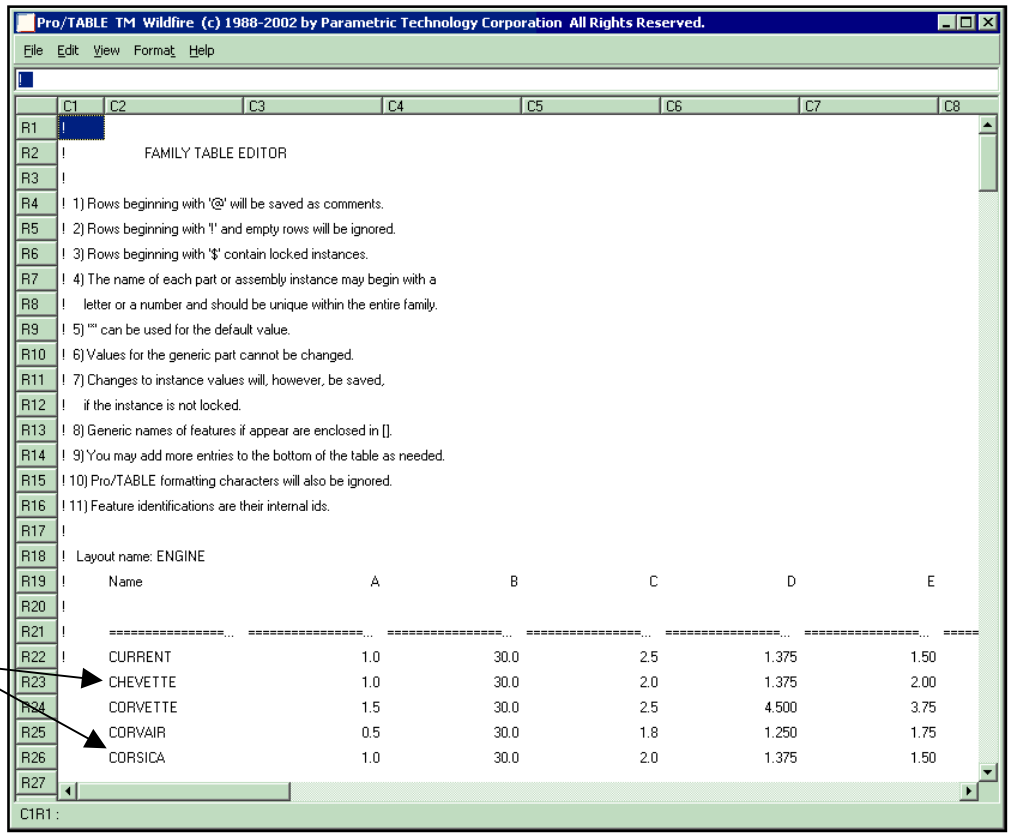
Parameter Sets

After the layout is established and parameters are defined, 'sets' of parameters can be created. This provides flexibility when modifying parameters and allows custom configurations to be pre-defined. Pick **Parameters, Param Table** in the **Parameters** dialog box to access the PARAM TABLE menu shown below.

Add Param	Add parameters to the table.
Del Param	Delete selected parameters from the table.
Apply Set	Apply the selected set of parameters to the layout.
Edit	Edit the parameter sets using Pro/TABLE.
Show	Show the contents of the parameter table.

In the small engine layout example, a set of parameters can be created for each application of the engine as shown below.

Each row of the table is a 'set' of parameters

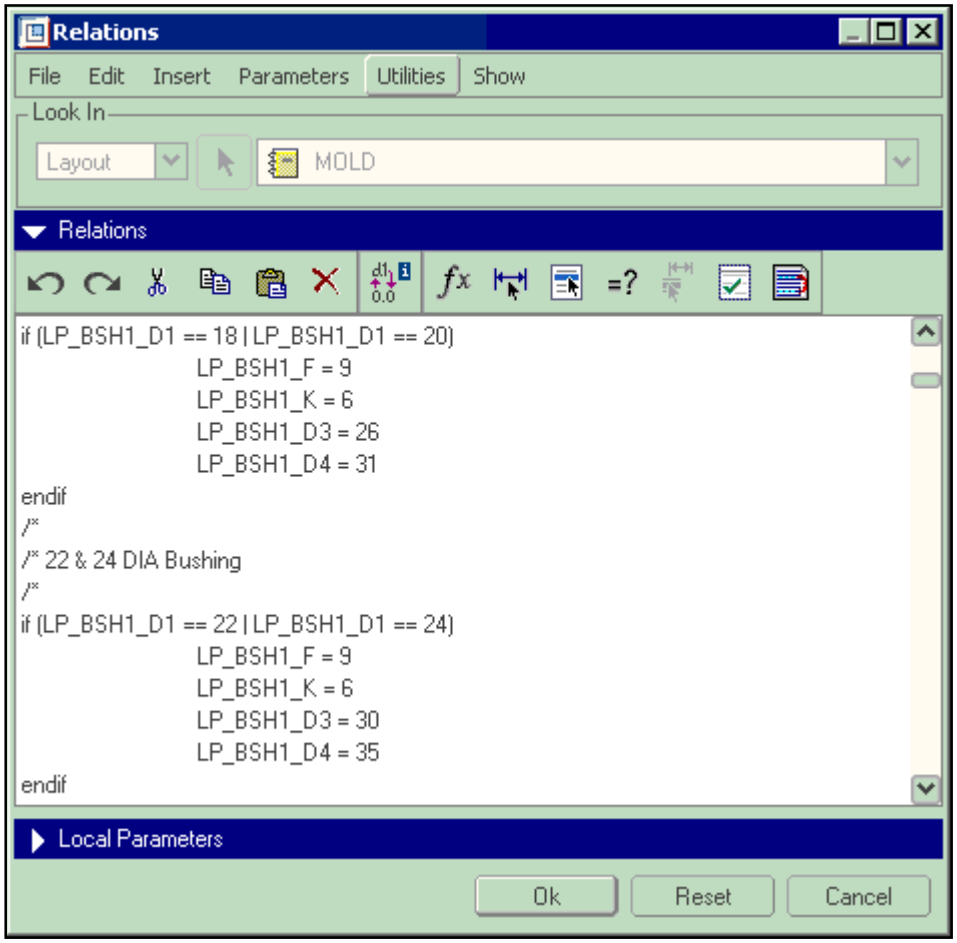


Creating Layout Relations

The layout can also include relations similar to part relations. These relations are used to control parts of the design not covered directly by parameters. For example, in gear design the number of teeth is controlled by the relation: num_teeth = pitch_dia * diametral_pitch

The user enters values for 'pitch_dia' and 'diametral_pitch' in the layout, and the system calculates 'num_teeth' and passes it to the model.

Relations are added to a layout by selecting **Tools, Relations**. The Relations dialog box is shown below. See pages xxx-xxx for a full explanation of relations and the available math functions.



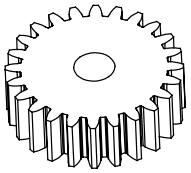
The Layout Relations Dialog Box

Tables

Tables are a convenient method of presenting parameter names and values to the end user of the layout. The layout shown below includes a table containing the important design parameters and their values. Pick **Table, Insert, Table** to create a table in a layout. See the note below.

The end user of the layout simply edits the values of the parameters and then regenerates the layout. The layout shown below also includes imported 2D IGES geometry.

Pitch Diameter	D	3.000
Diametral Pitch	P_D	8.000
Circular Pitch	P_C	0.393
Number of Teeth	N_G	24
Gear Thickness	THICK	2.000
Addendum	A	0.125
Dedendum	B	0.156
Outside Diameter	D_O	3.250
Root Diameter	D_R	2.688
Tooth Thickness	T	0.196
Pressure Angle	Ang_P	20.000
Base Diameter	D_B	2.819
Fillet	R_F	0.020



Sheets, Notes, and Other Tools

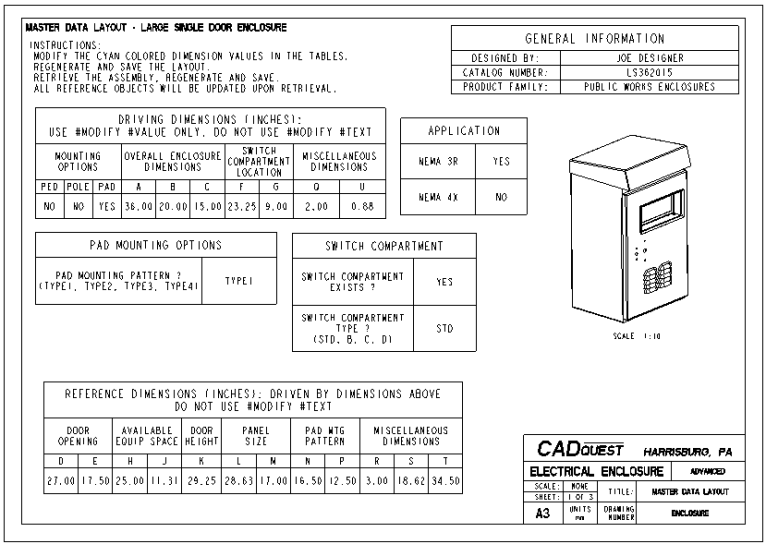
Sheets, notes, draft entities and other drawing related items can be added to the layout. The commands to annotate a layout are similar to those found in detail drawings and are not covered in this textbook.

Note

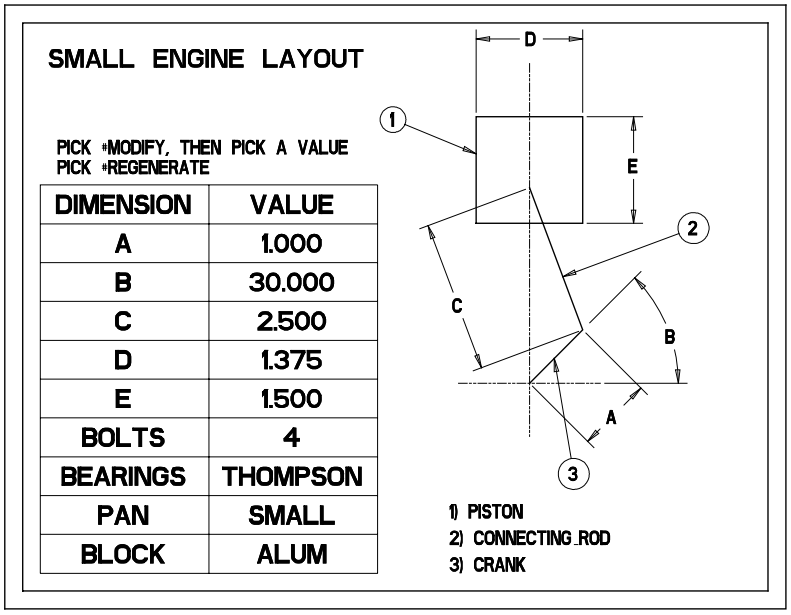
*See the text entitled "**Pro/ENGINEER Wildfire Detail Drawings and Pro/REPORT**" by this author and publisher for complete information about tables, sheets, notes, and other annotation commands.*

Example Layouts

Layouts are used to control parts and assemblies. The example shown below is an electrical enclosure, containing sheetmetal and solid parts. Using this layout, a new enclosure can be generated at any desired size and configuration in a few minutes. The layout drives the complete assembly, which drives the associated drawings and manufacturing information.



Another example of a layout contains 2D sketched geometry, dimensions, notes, and balloons as shown below. This 2D geometry is similar to draft geometry on Pro/ENGINEER detail drawings. It is not associative to any model, but is created to make the layout easier to use by others. This example layout includes instructional notes and a simple bill of materials.

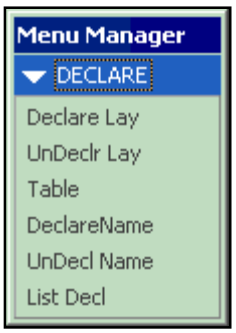


Using Layouts

After the layout is established and parameters are created, it can be used to control a new or existing design. The first step is be sure the layout is *in session*. Then open the part or assembly and pick **File, Declare, #Declare Lay**, then pick the name of the layout. The 'declare' command creates the reference to the selected layout. After the layout is 'declared', all the parameters in the layout are available to the part or assembly model.. You can use the parameters in the layout for relations in the part or assembly to control the design.

The DECLARE menu is shown.

Declare Lay	Declare (reference) the selected layout.
UnDeclr Lay	Un-declare (break the reference) the layout.
Table	Declare global references.
DeclareName	Declare a global datum (see the next page).
UnDecl Name	Un-declare a global datum.
List Decl	List the declared layouts and global datums in the information window.



Part Relations Utilizing Global Parameters

After the layout has been 'declared' in the part, relations can be created in the part to associate dimensions to global parameters. This can be accomplished by adding relations, editing relations, or by modifying the part dimension and entering the global parameter name.

It's a good idea to add *comments* to the relations, especially if your designs are to be used by others. Comments are used to explain the purpose of the relation, and to describe the dimensions, features, and constants involved. Comments must be added ***before*** the relation as shown below.

```
/* The number of teeth is equal to the pitch diameter times the diametral pitch
num_teeth = pitch_dia * diametral_pitch

/* The tooth thickness is equal to the circular pitch divided by 2
tooth_thick = circular_pitch / 2
```

Global Datums and Automatic Assembly

Another advantage of layouts is the use of 'global datums'. Global datums (planes and axes) are created in the layout, and then 'declared' in the parts. Parts having common global datums can be 'automatically' assembled or 'replaced' in assemblies. When creating the global datums in the layout, the direction of the positive/negative sides of the datum is not important, but you must remember which way they are pointing to get the automatic assembly correct.

To create a global datum plane in the layout, pick **Insert, Draft Datum, Plane**, then pick a start point and end point for the datum. Enter a name for the datum: *be careful, you **cannot** rename a global datum*. To create a global datum axis, pick **Insert, Draft Datum, Axis**.

To declare a global datum in a part, pick **File, Declare; #Declare Name** (see the previous page for the menu). Pick an plane and then choose the direction for the positive side of the global datum. The system will change the name of the datum in the part to correspond with the name of the global datum in the layout. Repeat this process in each part you want to replace or automatically assemble in assemblies.

In an assembly, if enough common global datums have been declared in a part, the system will allow automatic assembly. Pick **Automatic** and the system ***Aligns*** the positive side of each global datum plane. Global axes, points, and coordinate system may also be used to automatically assemble components.

Layout Review

To review, the basic steps to using layouts are:

- Gather the key design parameters and rules.
- Create the layout.
- Create the parameters and relations in the layout.
- Annotate the layout with tables, notes, figures, and notes.
- Create the parts and assembly models.
- Declare the layout in each part and assembly.
- Write relations in the parts and assemblies using the parameters in the layout.

When a design change occurs:

- Modify the parameters in the layout then regenerate the layout.
- Regenerate the parts and assemblies, watch them update.