Introduction

This text guides you through parametric design using Creo Parametric™. While using this text, you will create individual parts, assemblies, and drawings.

Parametric can be defined as *any set of physical properties whose values determine the characteristics or behavior of an object*. **Parametric design** enables you to generate a variety of information about your design: its mass properties, a drawing, or a base model. To get this information, you must first model your part design.

Parametric modeling philosophies used in Creo Parametric include the following:

Feature-Based Modeling Parametric design represents solid models as combinations of engineering features (Fig. 1).

Creation of Assemblies Just as features are combined into parts, parts may be combined into assemblies (Fig. 1).

Capturing Design Intent The ability to incorporate engineering knowledge successfully into the solid model is an essential aspect of parametric modeling.

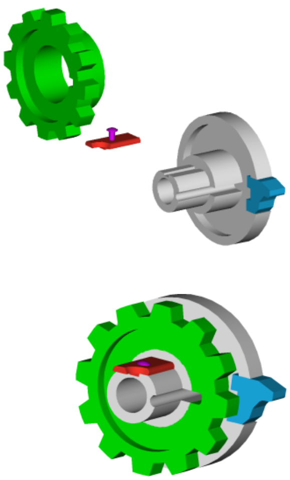


Figure 1 Parts and Assembly Design

Parametric Design

Parametric design models are not drawn so much as they are *sculpted* from solid volumes of materials. To begin the design process, analyze your design. Before any work is started, take the time to ***tap*** into your own knowledge bank and others that are available. Think, Analyze, and Plan. These three steps are essential to any well-formulated engineering design process.

Break down your overall design into its basic components, building blocks, or primary features. Identify the most fundamental feature of the object to sketch as the first, or base, feature. Varieties of **base features** can be modeled using extrude, revolve, sweep, and blend tools.

Sketched features (*extrusions, sweeps, etc.)* and pick-and-place features called **referenced features** (*holes, rounds, chamfers, etc.*) are normally required to complete the design. With the SKETCHER, you use familiar 2D entities (points, lines, rectangles, circles, arcs, splines, and conics) (Fig. 2). There is no need to be concerned with the accuracy of the sketch. Lines can be at differing angles, arcs and circles can have unequal radii, and features can be sketched with no regard for the actual objects' dimensions. In fact, exaggerating the difference between entities that are similar but not exactly the same is actually a far better practice when using the SKETCHER.

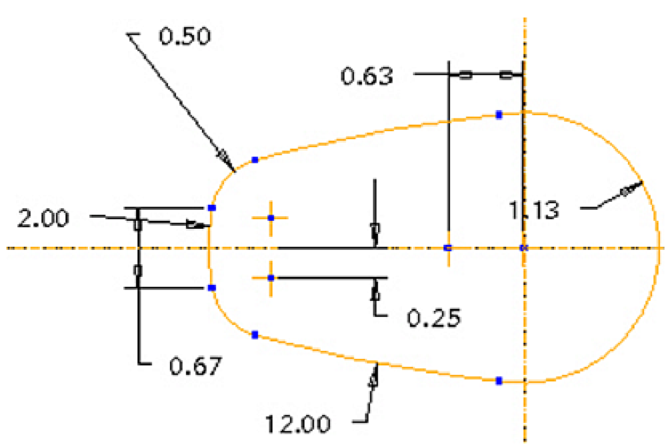


Figure 2 Sketching

Geometry assumptions and constraints will close ends of connected lines, align parallel lines, and snap sketched lines to horizontal and vertical (orthogonal) orientations. Additional constraints are added by means of **parametric dimensions** to control the size and shape of the sketch.

Features are the basic building blocks you use to create an object (Fig. 3). Features "understand" their fit and function as though "smarts" were built into the features themselves. For example, a hole or cut feature "knows" its shape and location and the fact that it has a negative volume. As you modify a feature, the entire object automatically updates after regeneration. The idea behind feature-based modeling is that the designer constructs an object so that it is composed of individual features that describe the way the geometry is supposed to behave if its dimensions change. This happens quite often in industry, as in the case of a design change. Feature-based modeling is diagramed in Figure 4.

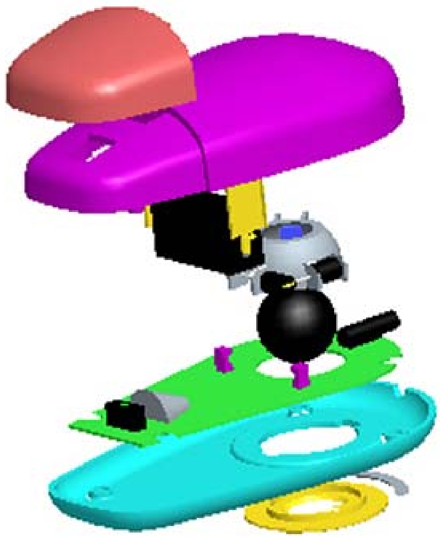


Figure 3 Feature Design (Courtesy CADTRAIN)

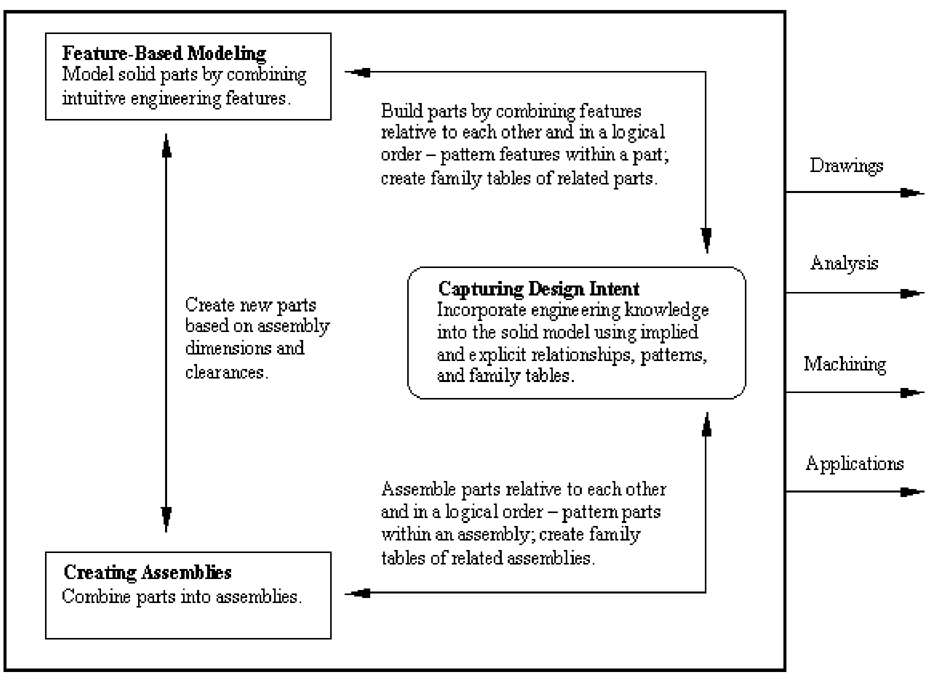
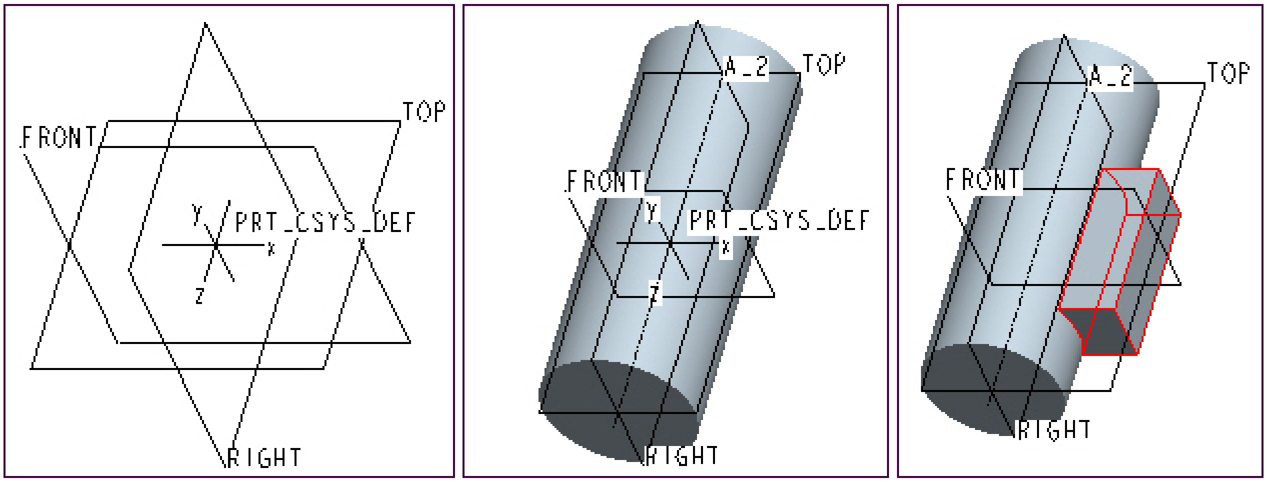


Figure 4 Feature-based Modeling

Parametric modeling is the term used to describe the capturing of design operations as they take place, as well as future modifications and editing of the design. The order of the design operations is significant. Suppose a designer specifies that two surfaces be parallel, such that surface two is parallel to surface one. Therefore, if surface one moves, surface two moves along with surface one to maintain the specified design relationship. The surface two is a **child** of surface one in this example. Parametric modeling software allows the designer to **reorder** the steps in the object's creation.

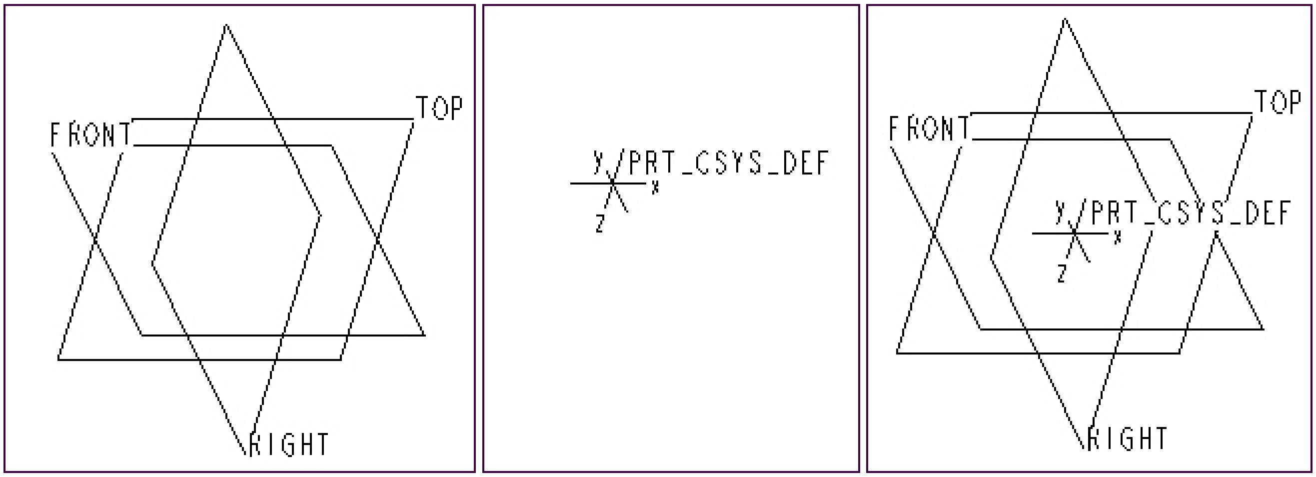
Various types of features are used as building blocks in the progressive creation of solid objects. Figures 5(a-c) illustrates base features, datum features, sketched features, and referenced features. The "chunks" of solid material from which parametric design models are constructed are called **features**.



Figures 5(a-c) Features

Features generally fall into one of the following categories:

Base Feature The base feature is normally a set of datum planes referencing the default coordinate system. The base feature is important because all future model geometry will reference this feature directly or indirectly; it becomes the root feature. Changes to the base feature will affect the geometry of the entire model [Figs. 6(a-c)].



Figures 6(a-c) Base Features

Datum Features Datum features (lines, axes, curves, and points) are generally used to provide sketching planes and contour references for sketched and referenced features. Datum features do not have volume or mass and may be visually hidden without affecting solid geometry (Fig. 7).

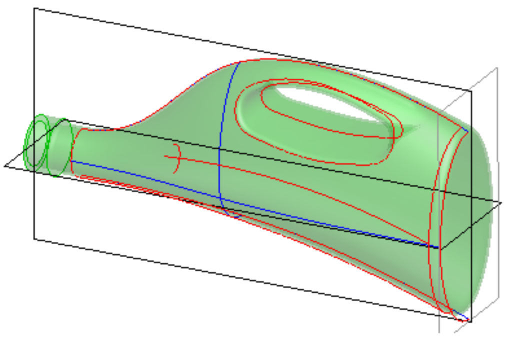


Figure 7 Datum Curves

Sketched Features Sketched features are created by extruding, revolving, blending, or sweeping a sketched cross section. Material may be added or removed by protruding or cutting the feature from the existing model (Fig. 8).

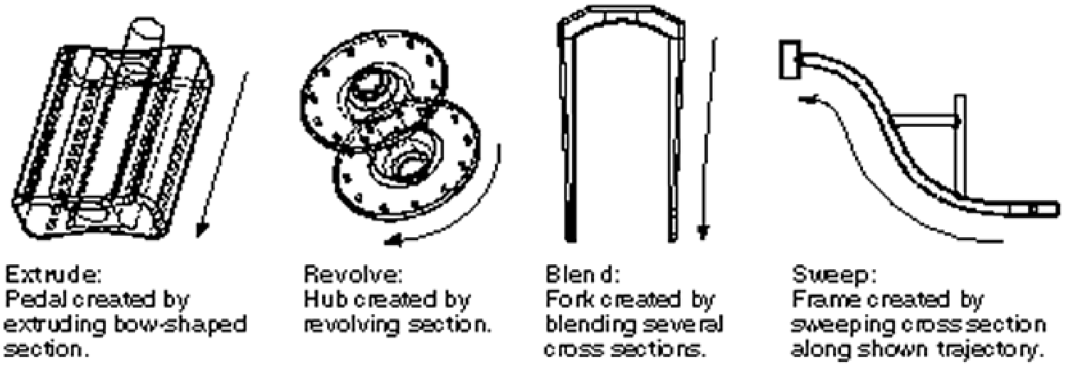
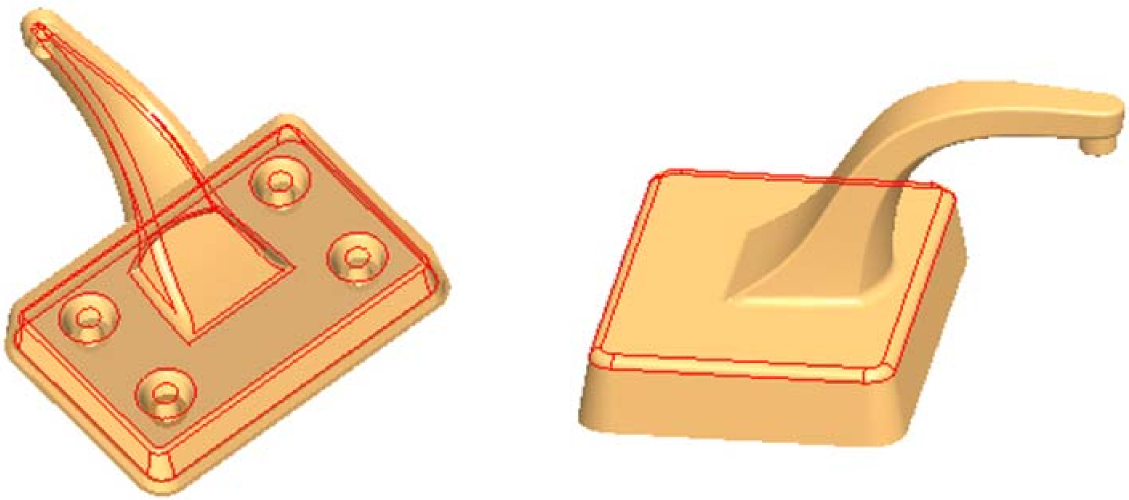


Figure 8 Sketched Features

Referenced Features Referenced features (rounds, holes, shells, and so on) utilize existing geometry for positioning and employ an inherent form; they do not need to be sketched [Figs. 9(a-b)].



Figures 9(a-b) Referenced Features- Shell and Round (Spout is a Swept Blend Feature)

A wide variety of features are available. These tools enable the designer to make far fewer changes by capturing the engineer's design intent early in the development stage (Fig. 10).

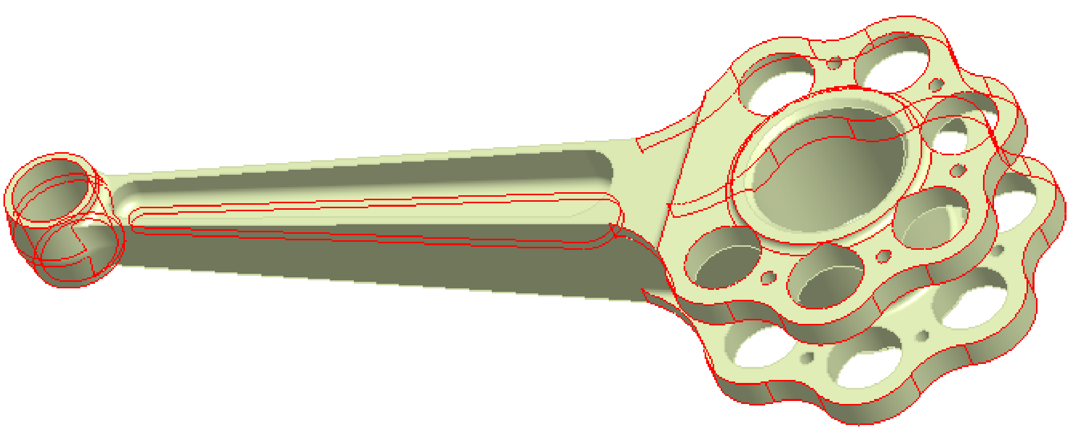


Figure 10 Parametric Designed Part

Fundamentals

The design of parts and assemblies, and the creation of related drawings, forms the foundation of engineering graphics. When designing with Creo Parametric, many of the previous steps in the design process have been eliminated, streamlined, altered, refined, or expanded. The model you create as a part forms the basis for all engineering and design functions.

The part model contains the geometric data describing the part's features, but it also includes non-graphical information embedded in the design itself. The part, its associated assembly, and the graphical documentation (drawings) are parametric. The physical properties described in the part drive (determine) the characteristics and behavior of the assembly and drawing. Any data established in the assembly mode, in turn, determines that aspect of the part and, subsequently, the drawings of the part and the assembly. In other words, all the information contained in the part, the assembly, and the drawing is interrelated, interconnected, and parametric (Fig. 11).

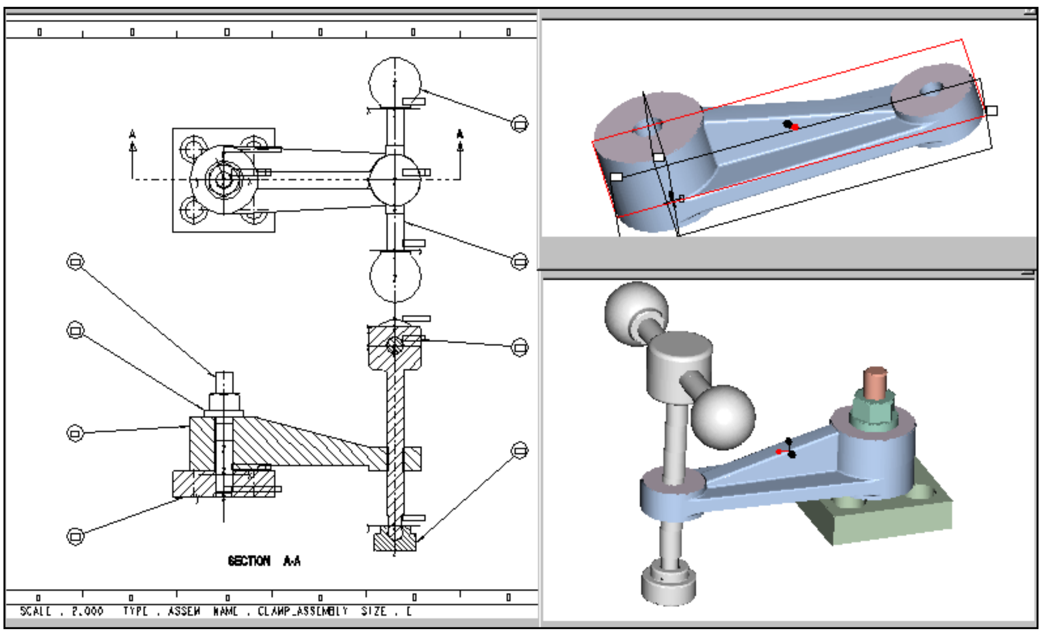


Figure 11 Assembly Drawing, Part, and Assembly

Part Design

In many cases, the part will be the first component of this interconnected process. The *part* function in Creo Parametric is used to design components.

During part design (Fig. 12), you can accomplish the following:

- Define the base feature
- Define and redefine construction features to the base feature
- Modify the dimensional values of part features (Fig. 13)
- Embed design intent into the model using tolerance specifications and dimensioning schemes
- Create pictorial and shaded views of the component
- Create part families (family tables)
- Perform mass properties analysis and clearance checks
- List part, feature, layer, and other model information
- Measure and calculate model features
- Create detail drawings of the part

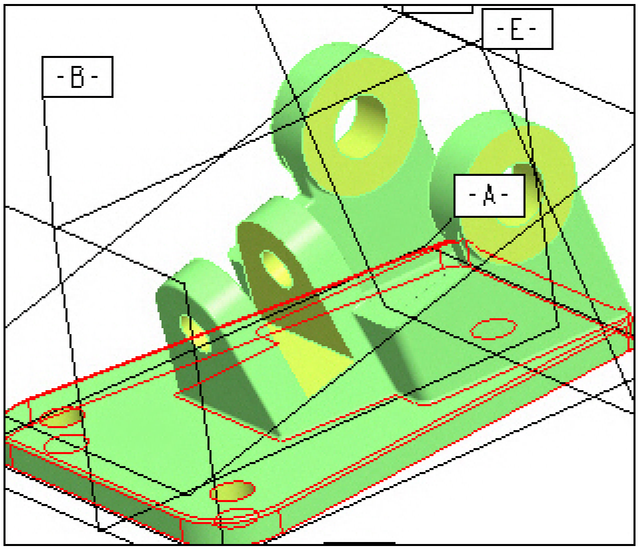


Figure 12 Part Design

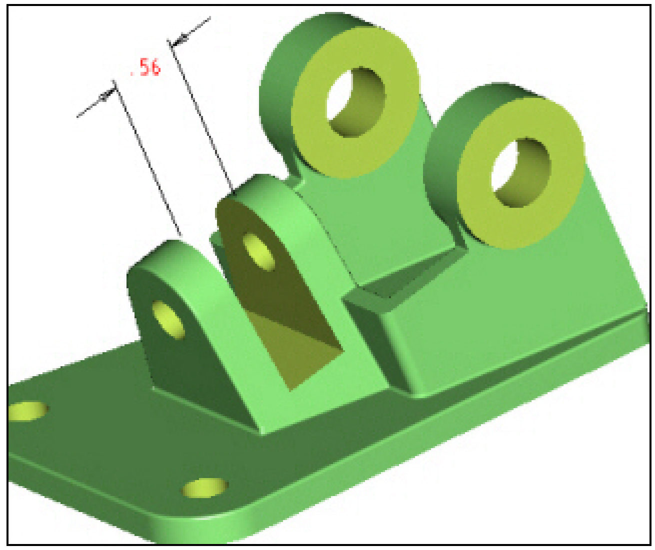


Figure 13 Pick on the **.56** dimension and type in a new value

Establishing Features

The design of any part requires that the part be *confined*, *restricted*, *constrained*, and *referenced*. In parametric design, the easiest method to establish and control the geometry of your part design is to use three datum planes. Creo Parametric automatically creates the three **primary datum planes**. The default datum planes (**RIGHT**, **TOP**, and **FRONT**) constrain your design in all three directions.

Datum planes are infinite planes located in 3D model mode and are associated with the object that was active at the time of their creation. To select a datum plane, you can pick on its name or anywhere on the perimeter edge. Datum planes are *parametric*--geometrically associated with the part. Parametric datum planes are associated with and dependent on the edges, surfaces, vertices, and axes of a part. Datum planes are used to create a reference on a part that does not already exist. For example, you can sketch or place features on a datum plane when there is no appropriate planar surface. You can also dimension to a datum plane as though it were an edge. In Figure 14, three **default datum planes** and a **default coordinate systems** were created when a new part (**PRT0001.PRT**) was started using the default template. Note that in the **Model Tree** window, they are the first four features of the part, which means that they will be the *parents* of the features that follow.

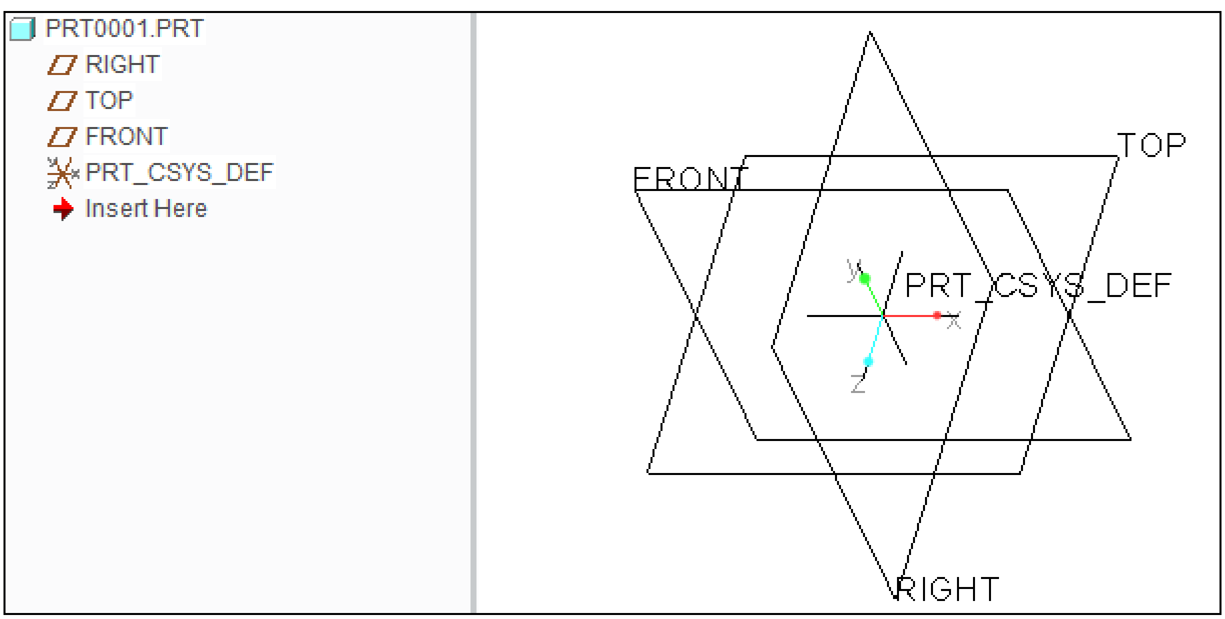


Figure 14 Default Datum Planes and Coordinate System

Datum Features

Datum features are planes, axes, and points you use to place geometric features on the active part. Datums other than defaults can be created at any time during the design process.

As we have discussed, there are three (primary) types of datum features (Fig. 14): **datum planes, datum axes**, and **datum points** (there are also ***datum curves*** and ***datum coordinate systems***). You can display all types of datum features, but they do not define the surfaces or edges of the part or add to its mass properties. In Figure 15, a variety of datum planes are used in the creation of the casting.

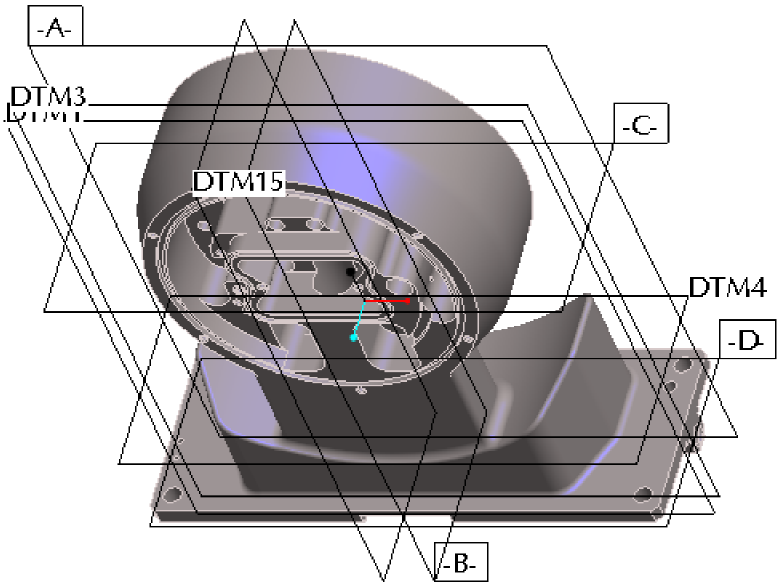


Figure 15 Datums in Part Design

Specifying constraints that locate it with respect to existing geometry creates a datum. For example, a datum plane might be made to pass through the axis of a hole and parallel to a planar surface. Chosen constraints must locate the datum plane relative to the model without ambiguity. You can also use and create datums in assembly mode.

Besides datum planes, datum axes and datum points can be created to assist in the design process. You can also automatically create datum axes through cylindrical features such as holes and solid round features by setting this as a default in your Creo Parametric configuration file. The part in Figure 16 shows the default axes for a variety of holes.

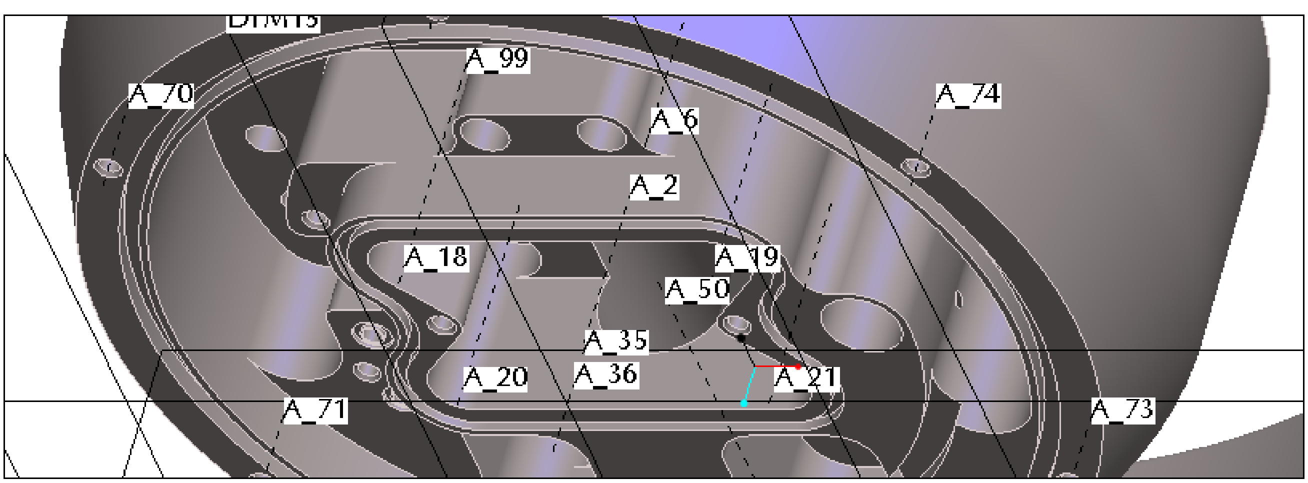
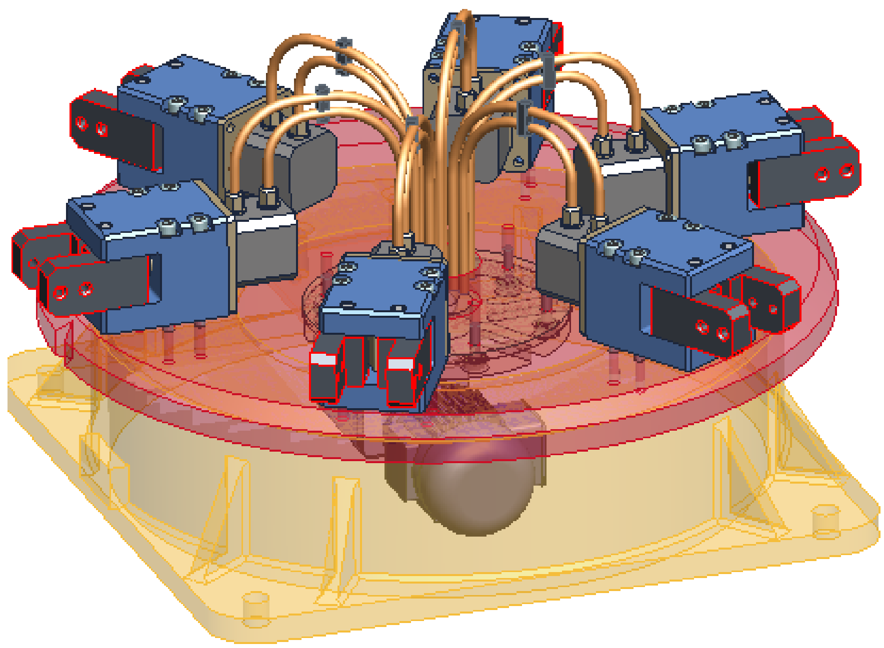


Figure 16 Feature Default Datum Axes

Parent-Child Relationships

Because solid modeling is a cumulative process, certain features must, by necessity precede others. Those that follow must rely on previously defined features for dimensional and geometric references. The relationships between features and those that reference them are termed ***parent-child relationships***. Because children reference parents, parent features can exist without children, but children cannot exist without their parents. This type of CAD modeler is called a history-based system. Using Creo Parametric's information command will list a model's parent-child references and dependencies (Fig. 17).



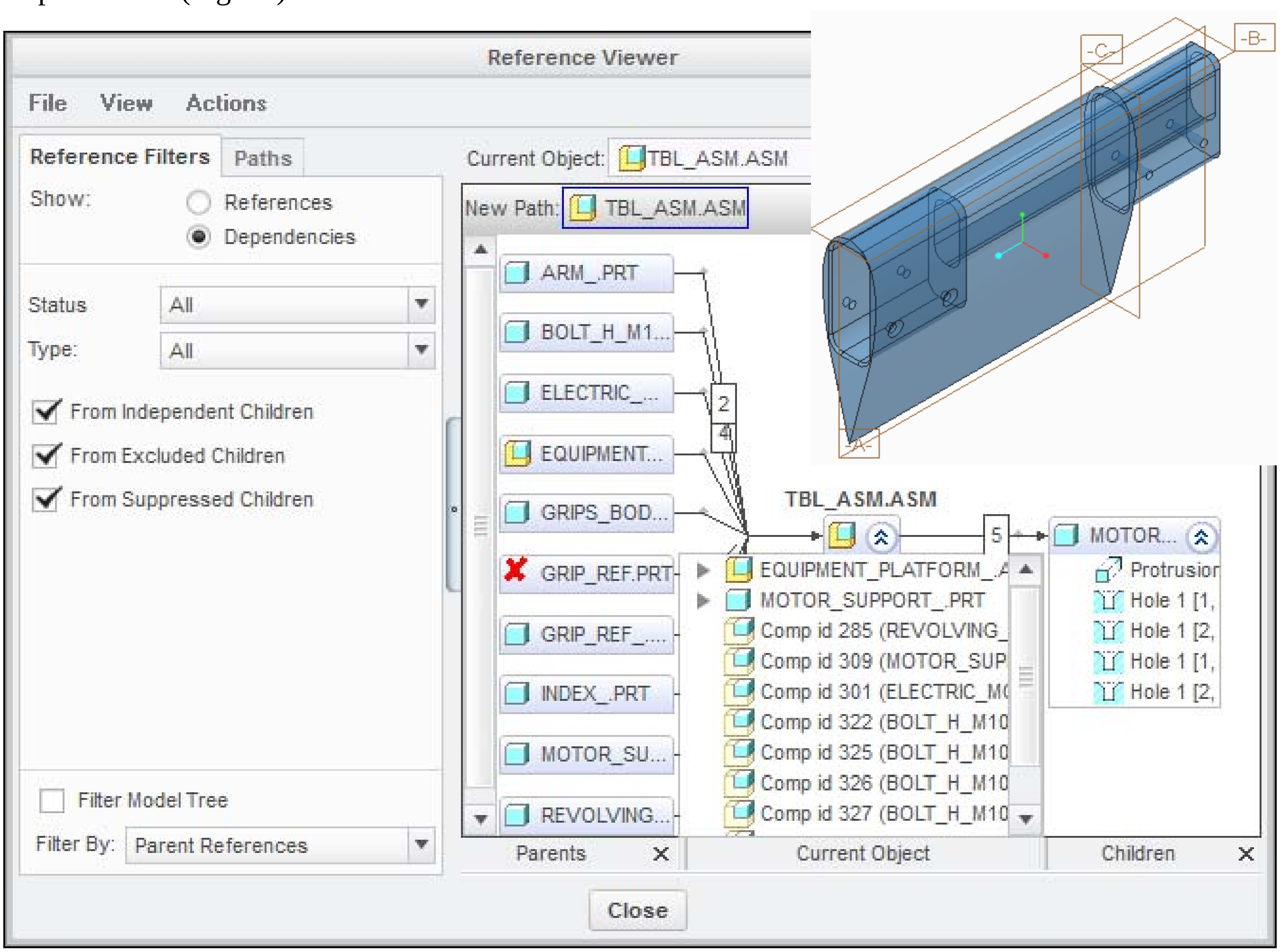


Figure 17 Model Information

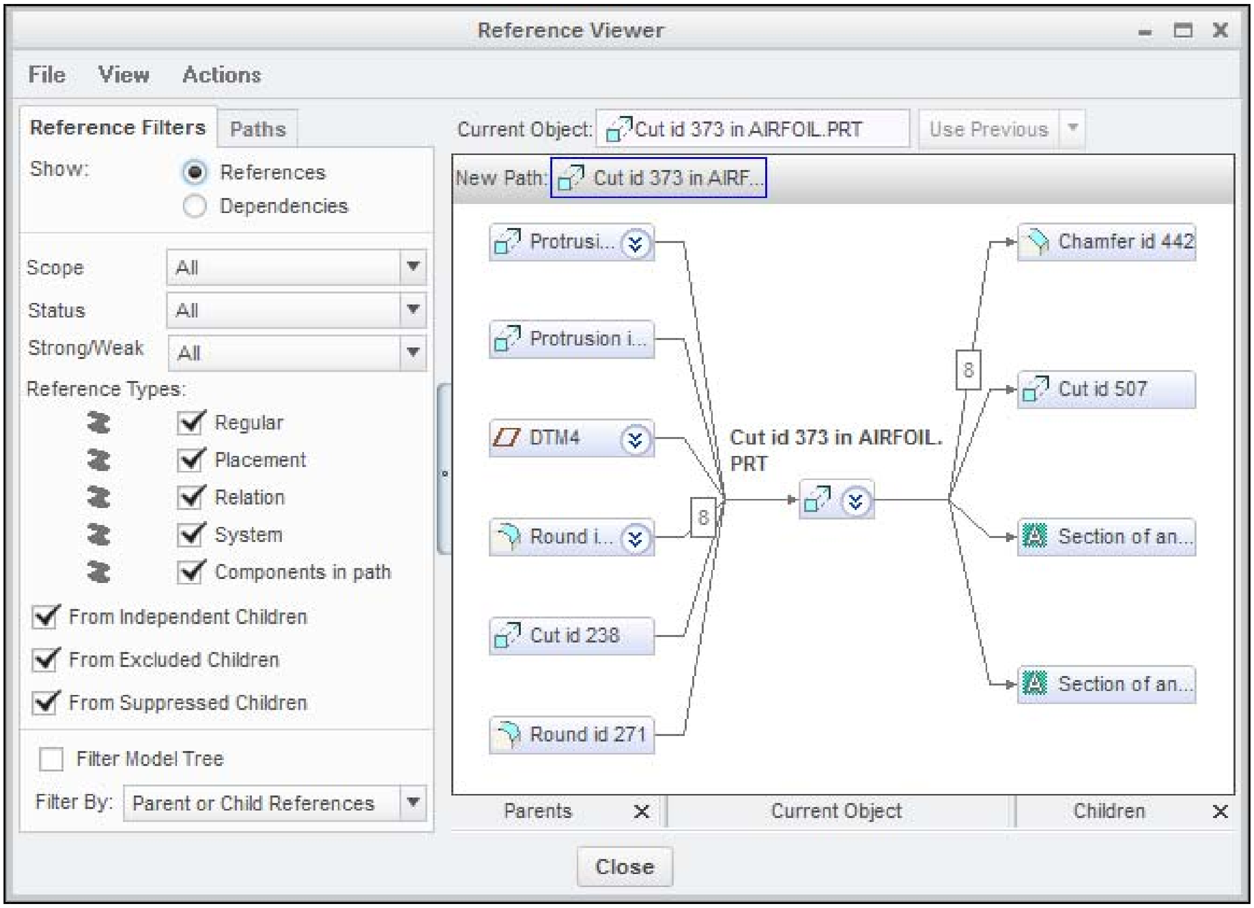


Figure 18 Parent-Child Information

The parent-child relationship (Fig. 18) is one of the most powerful aspects of parametric design. When a parent feature is modified, its children are automatically recreated to reflect the changes in the parent feature's geometry. It is essential to reference feature dimensions so that design modifications are correctly propagated through the model/part. Any modification to the part is automatically propagated throughout the model (Fig. 19) and will affect all children of the modified feature.

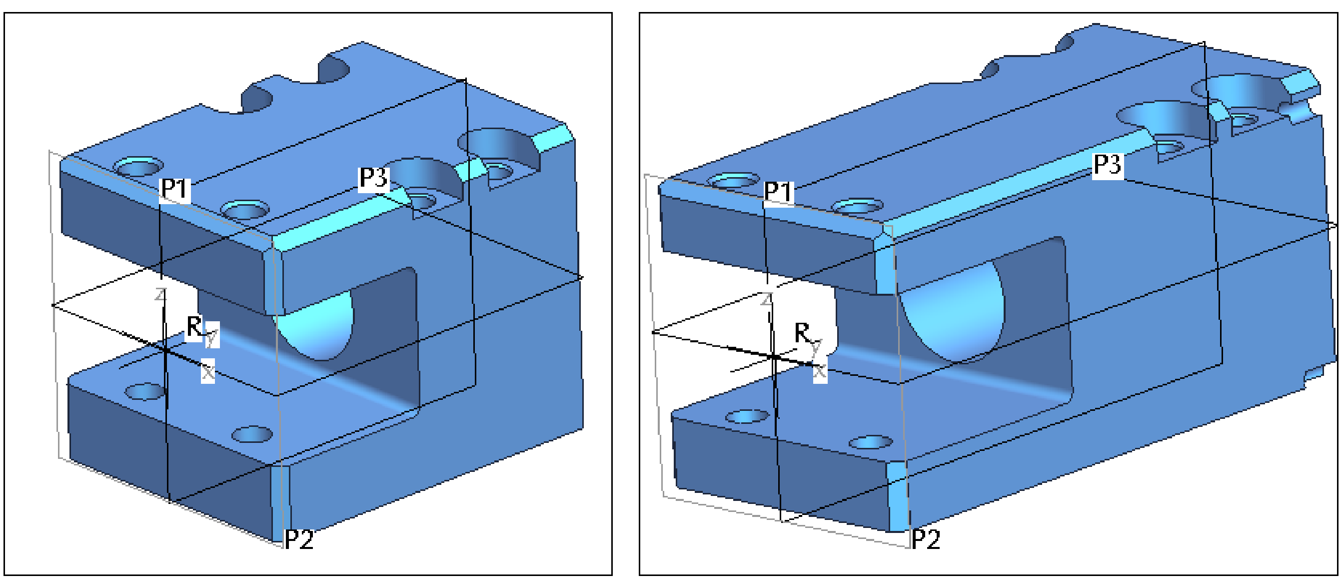


Figure 19 Original Design and Modification

Capturing Design Intent

A valuable characteristic of any design tool is its ability to *render* the design and at the same time capture its *intent* (Fig. 20). Parametric methods depend on the sequence of operations used to construct the design. The software maintains a *history of changes* the designer makes to specific parameters. The point of capturing this history is to keep track of operations that depend on each other. Whenever Creo Parametric is told to change a specific dimension, it can update all operations that are referenced to that dimension.

For example, a circle representing a bolt hole circle may be constructed so that it is always concentric to a circular slot. If the slot moves, so does the bolt circle. Parameters are usually displayed in terms of dimensions or labels and serve as the mechanism by which geometry is changed. The designer can change parameters manually by changing a dimension or can reference them to a variable in an equation (**relation**) that is solved either by the modeling program itself or by external programs such as spreadsheets.

Features can also store non-graphical information. This information can be used in activities such as drafting, numerical control (NC), finite-element analysis (FEA), and kinematics analysis.

Capturing design intent is based on incorporating engineering knowledge into a model by establishing and preserving certain geometric relationships. The wall thickness of a pressure vessel, for example, should be proportional to its surface area and should remain so, even as its size changes.

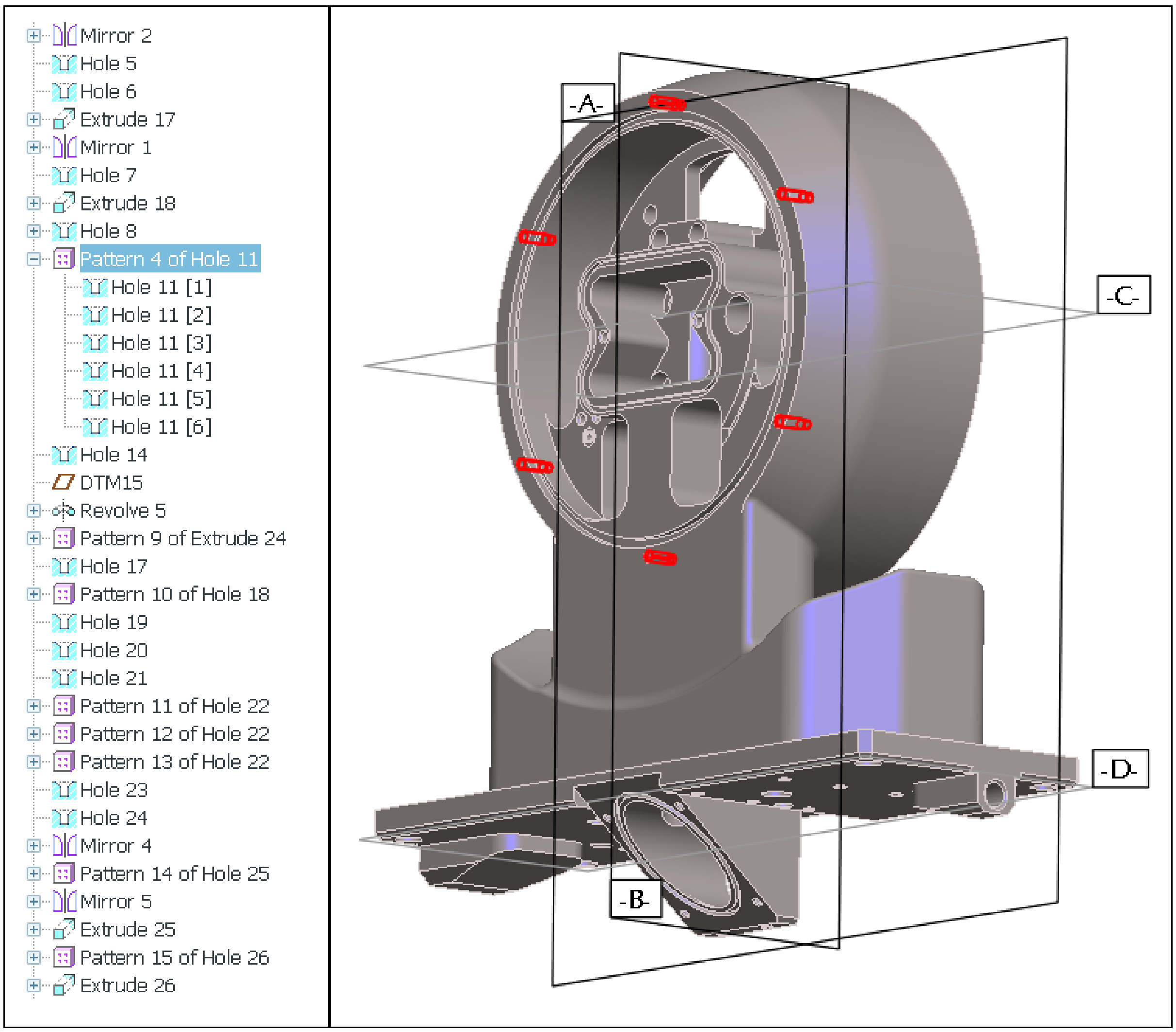


Figure 20 Capturing Design Intent

Parametric designs capture relationships in several ways:

Implicit Relationships Implicit relationships occur when new model geometry is sketched and dimensioned relative to existing features and parts. An implicit relationship is established, for instance, when the section sketch of a tire (Fig. 21) uses rim edges as a reference.

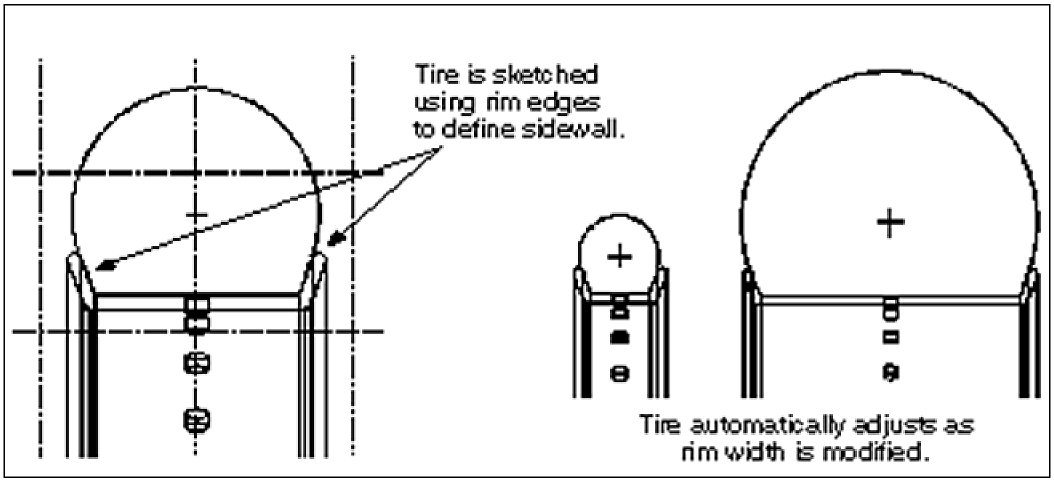


Figure 21 Tire and Rim

Patterns Design features often follow a geometrically predictable pattern. Features and parts are patterned in parametric design by referencing either construction dimensions or existing patterns. One example of patterning is a wheel hub with spokes (Fig. 22). First, the spoke holes are radially patterned. The spokes can then be strung by referencing this pattern.

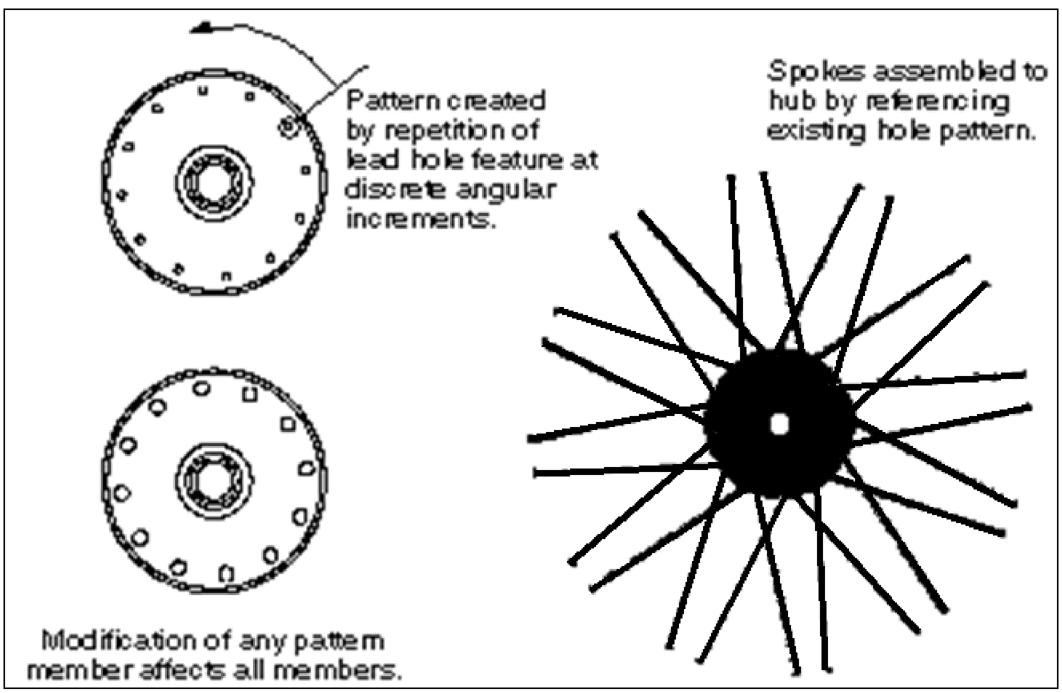


Figure 22 Patterns

Modification to a pattern member affects all members of that pattern. This helps capture design intent by preserving the duplicate geometry of pattern members.

The modeling task is to incorporate the features and parts of a complex design while properly capturing design intent to provide flexibility in modification. Parametric design modeling is a synthesis of physical and intellectual design (Fig. 23).

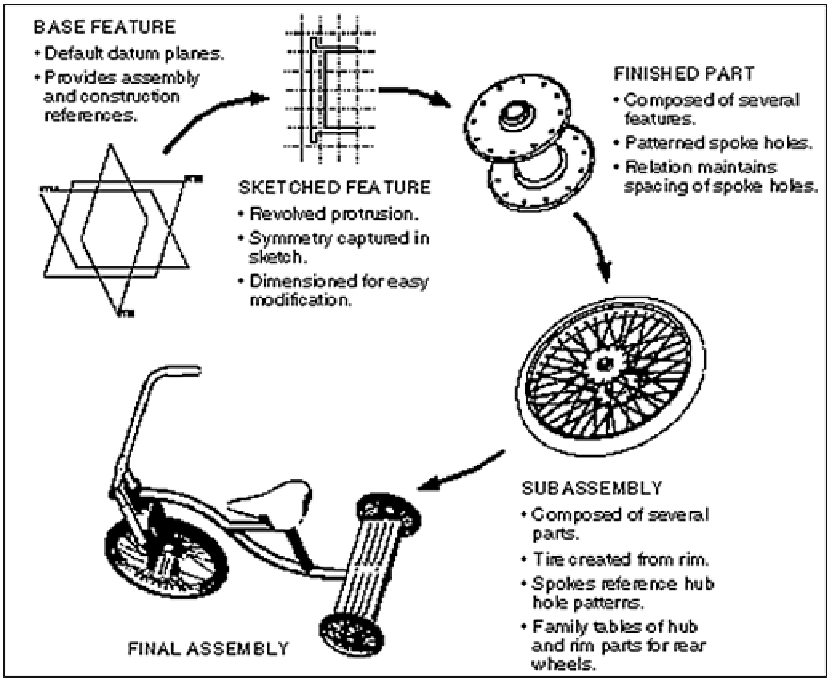


Figure 23 Relations

Explicit Relations Whereas implicit relationships are implied by the feature creation method, the user mathematically enters an explicit relation. This equation is used to relate feature and part dimensions in the desired manner. An explicit relation (Fig. 24) might be used, for example, to control sizes on a model.

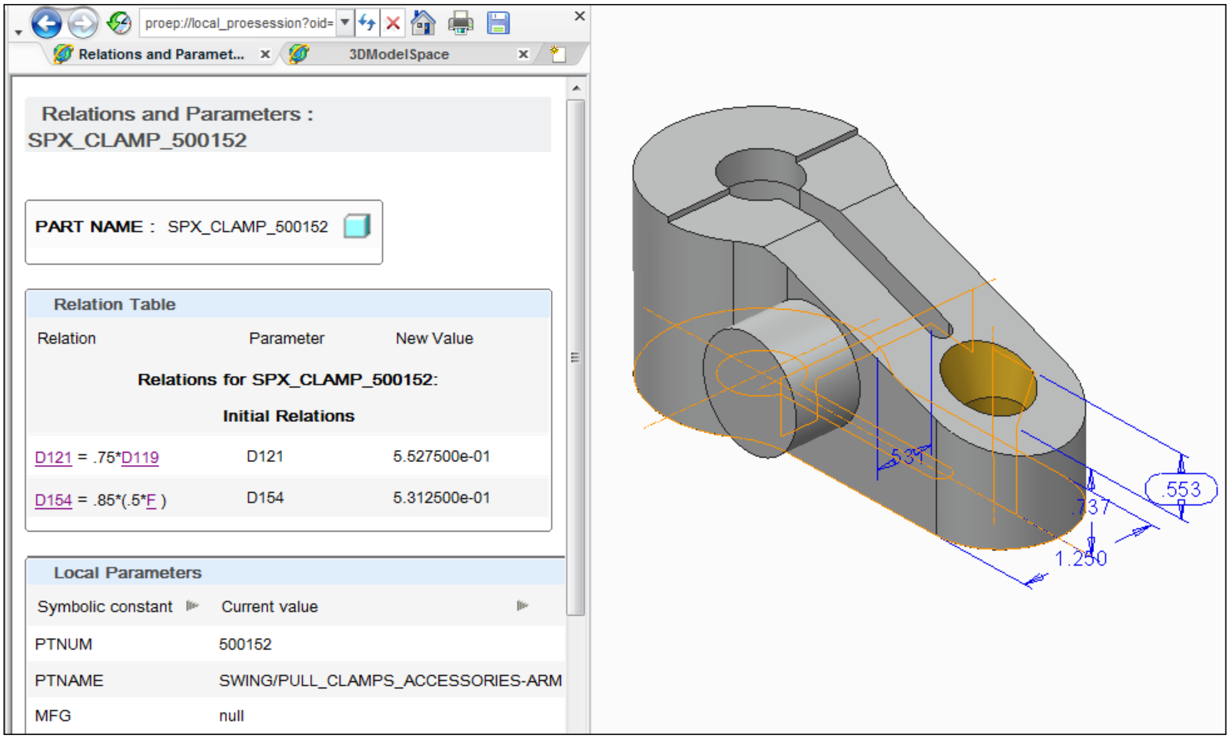
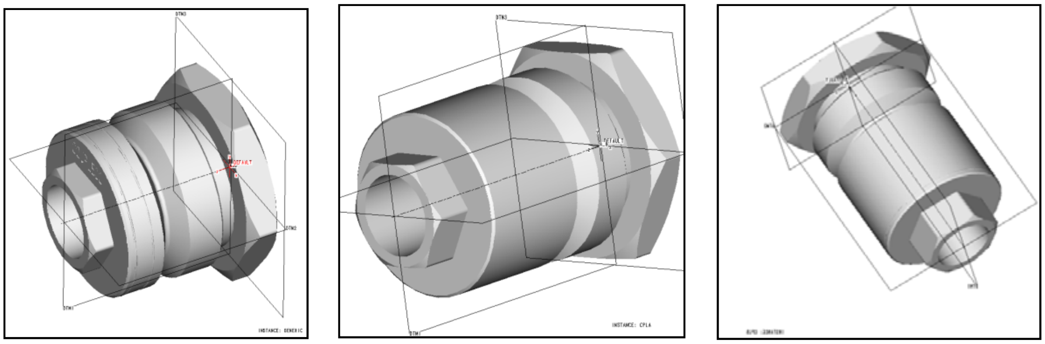


Figure 24 Adding Relations

Family Tables Family tables are used to create part families [Figs. 25(a-c)] from generic models by tabulating dimensions or the presence of certain features or parts. A family table might be used, for example, to catalog a series of couplings with varying width and diameter as shown in Figure 26.



Figures 25(a-c) Family of Parts- Coupling-Fitting

Type	Instance Name	Common Name	d30	d46	F1068 [CUT]	F829 [SLOT]	F872 [SLOT]	F2620 COPIED_G...
	COUPLING-FITTI...		2.06	0.3130	Y	Y	Y	Y
	CPLA		3.00	0.5000	N	N	N	N
	CPLB		3.25	0.6250	N	Y	N	N
	CPLC		3.50	0.7500	Y	N	Y	Y

Figure 26 Family Table for Coupling-Fitting

Assemblies

Just as parts are created from related features, **assemblies** are created from related parts. The progressive combination of subassemblies, parts, and features into an assembly creates parent-child relationships based on the references used to assemble each component (Fig. 27).

The *Assembly* functionality is used to assemble existing parts and subassemblies.

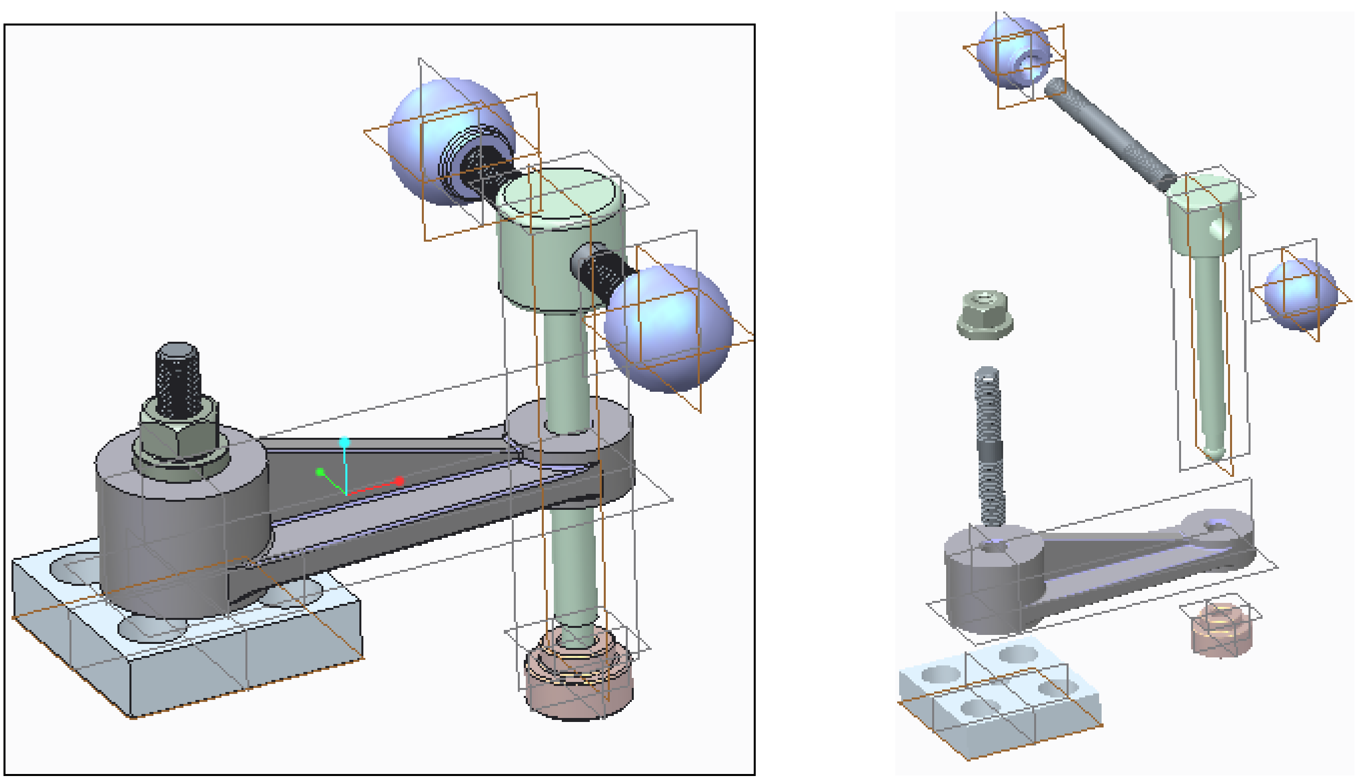


Figure 27 Clamp Assembly and Exploded Clamp Assembly

During assembly creation, you can:

- Simplify a view of a large assembly by creating a simplified representation
- Perform automatic or manual placement of component parts
- Create an exploded view of the component parts
- Perform analysis, such as mass properties and clearance checks
- Modify the dimensional values of component parts
- Define assembly relations between component parts
- Create assembly features
- Perform automatic interchange of component parts
- Create parts in Assembly mode
- Create documentation drawings of the assembly

Just as features can reference part geometry, parametric design also permits the creation of parts referencing assembly geometry. **Assembly mode** allows the designer both to fit parts together and to design parts based on how they should fit together. In Figure 28, an assembly *Bill of Materials* report is generated.

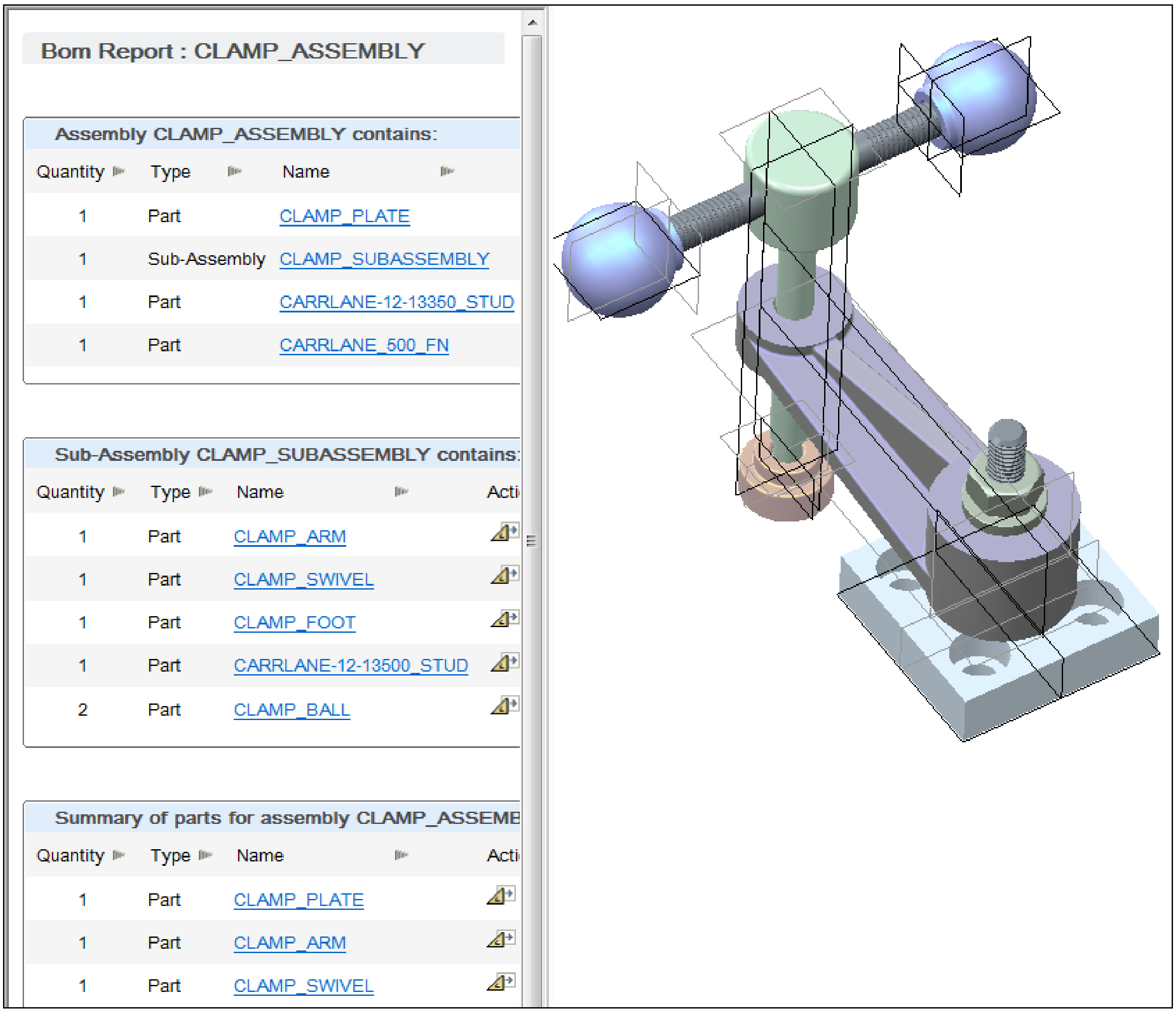


Figure 28 BOM Report

Drawings

You can create drawings of all parametric design models (Fig. 29). All model views in the drawing are *associative:* if you change a dimensional value in one view, other drawing views update accordingly. Moreover, drawings are associated with their parent models. Any dimensional changes made to a drawing are automatically reflected in the model. Any changes made to the model (e.g., addition of features, deletion of features, dimensional changes, and so on) in Part, Sheet Metal, Assembly, or Manufacturing modes are also automatically reflected in their corresponding drawings.

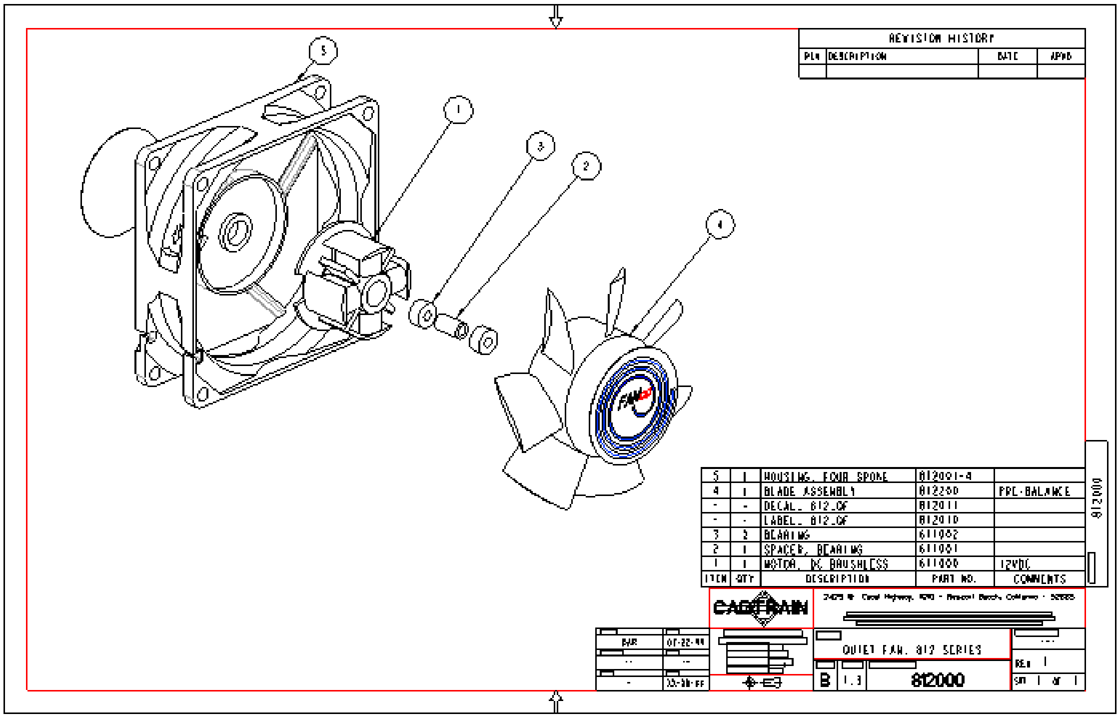


Figure 29 Ballooned Exploded View Assembly Drawing with Bill of Materials (BOM) (Courtesy CADTRAIN)

The **Drawing** functionality is used to create annotated drawings of parts and assemblies. During drawing creation, you can:

- Add views of the part or assembly
- Show existing dimensions
- Incorporate additional driven or reference dimensions
- Create notes on the drawing
- Display views of additional parts or assemblies
- Add sheets to the drawing
- Create draft entities on the drawing
- Balloon components on an assembly drawing (Fig. 30)
- Create an associative BOM

You can annotate the drawing with notes, manipulate the dimensions, and use layers to manage the display of different items on the drawing. The module **Pro/DETAIL** can be used to extend the drawing capability or as a stand-alone module allowing you to create, view, and annotate models and drawings. Pro/DETAIL supports additional view types and multi-sheets and offers commands for manipulating items in the drawing and for adding and modifying different kinds of textural and symbolic information. In addition, the abilities to customize engineering drawings with sketched geometry, create custom drawing formats, and make numerous cosmetic changes to the drawing are available.

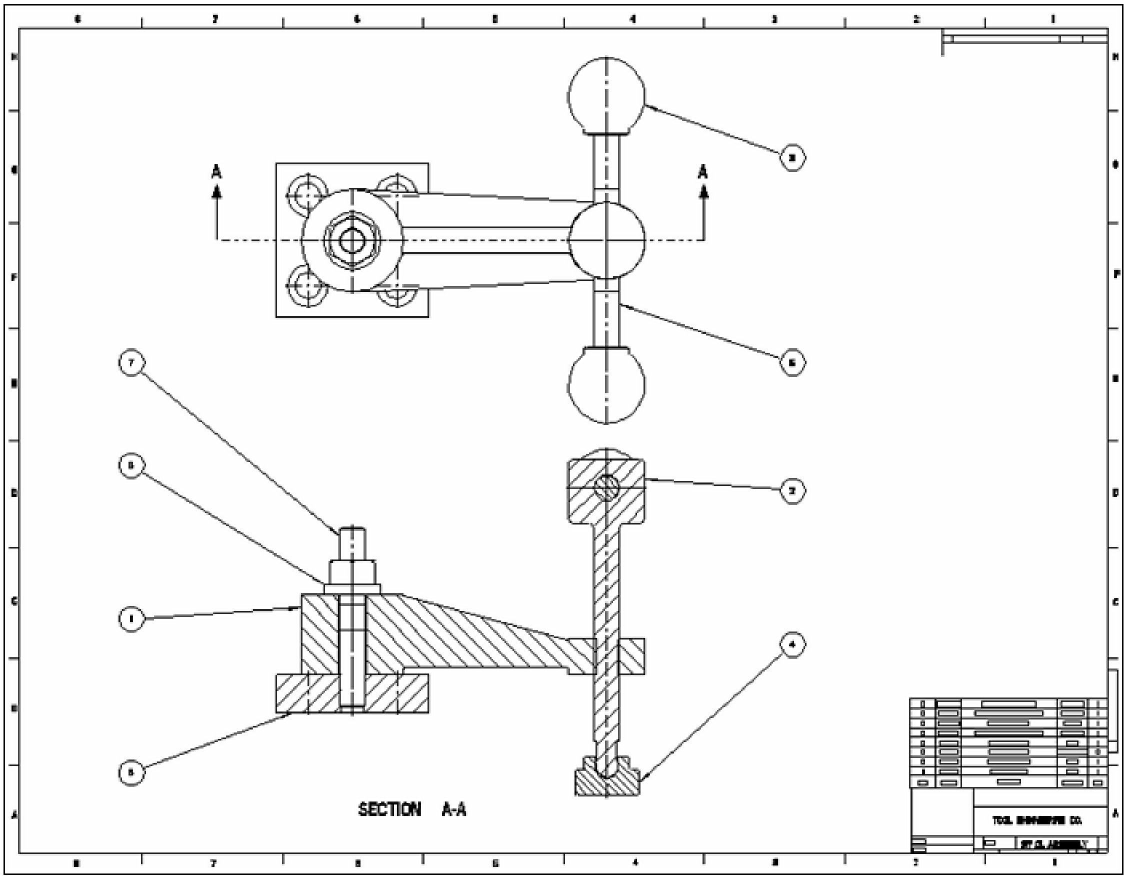


Figure 30 Ballooned Assembly Drawing

Drawing mode in parametric design provides you with the basic ability to document solid models in drawings that share a two-way associativity.

Changes that are made to the model in Part mode or Assembly mode will cause the drawing to update automatically and reflect the changes. Any changes made to the model in Drawing mode will be immediately visible on the model in Part and Assembly modes. The model shown in Figure 31 has been detailed in Figure 32.

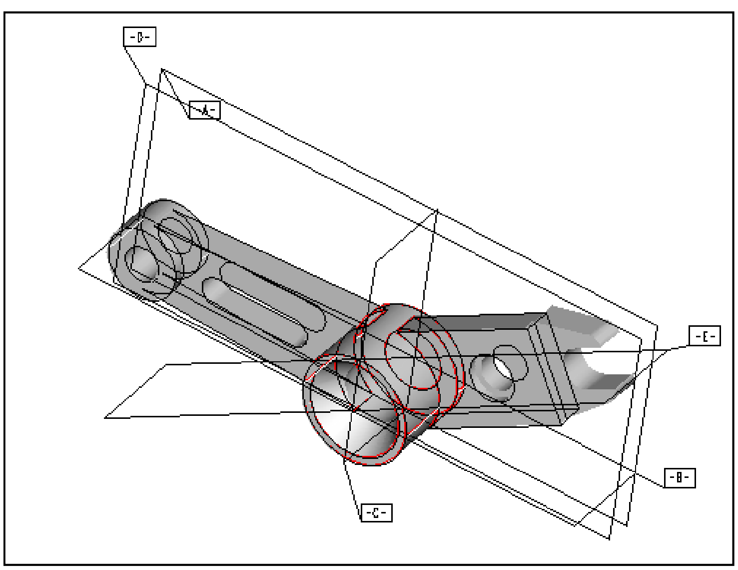


Figure 31 Angle Frame Model

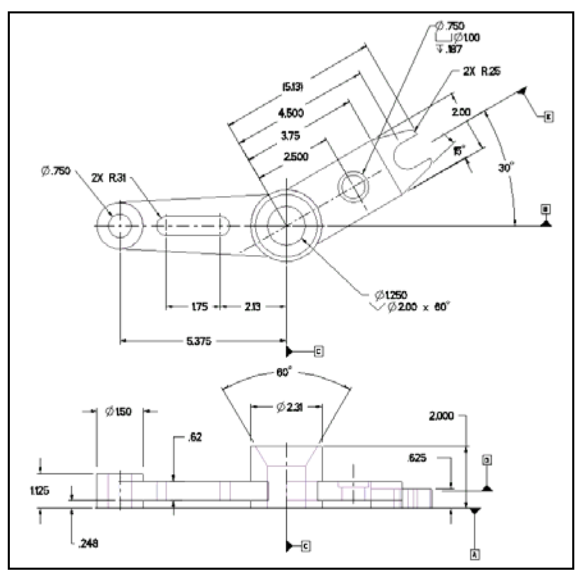


Figure 32 Angle Frame Drawing

Flexible modeling

Flexible modeling allows you to experiment with your design without being committed to changes. You can make explicit modifications to selected geometry while ignoring pre-existing relationships. With this new capability within Creo Parametric you can utilize some basic Creo Direct capabilities. The ribbon in Figure 33 shows the new interface. Design changes are done directly on the model without parent-child relationships being imbedded (Fig. 34).

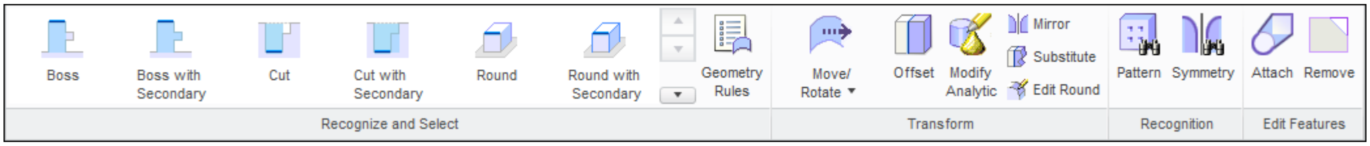


Figure 33 Flexible Modeling Ribbon

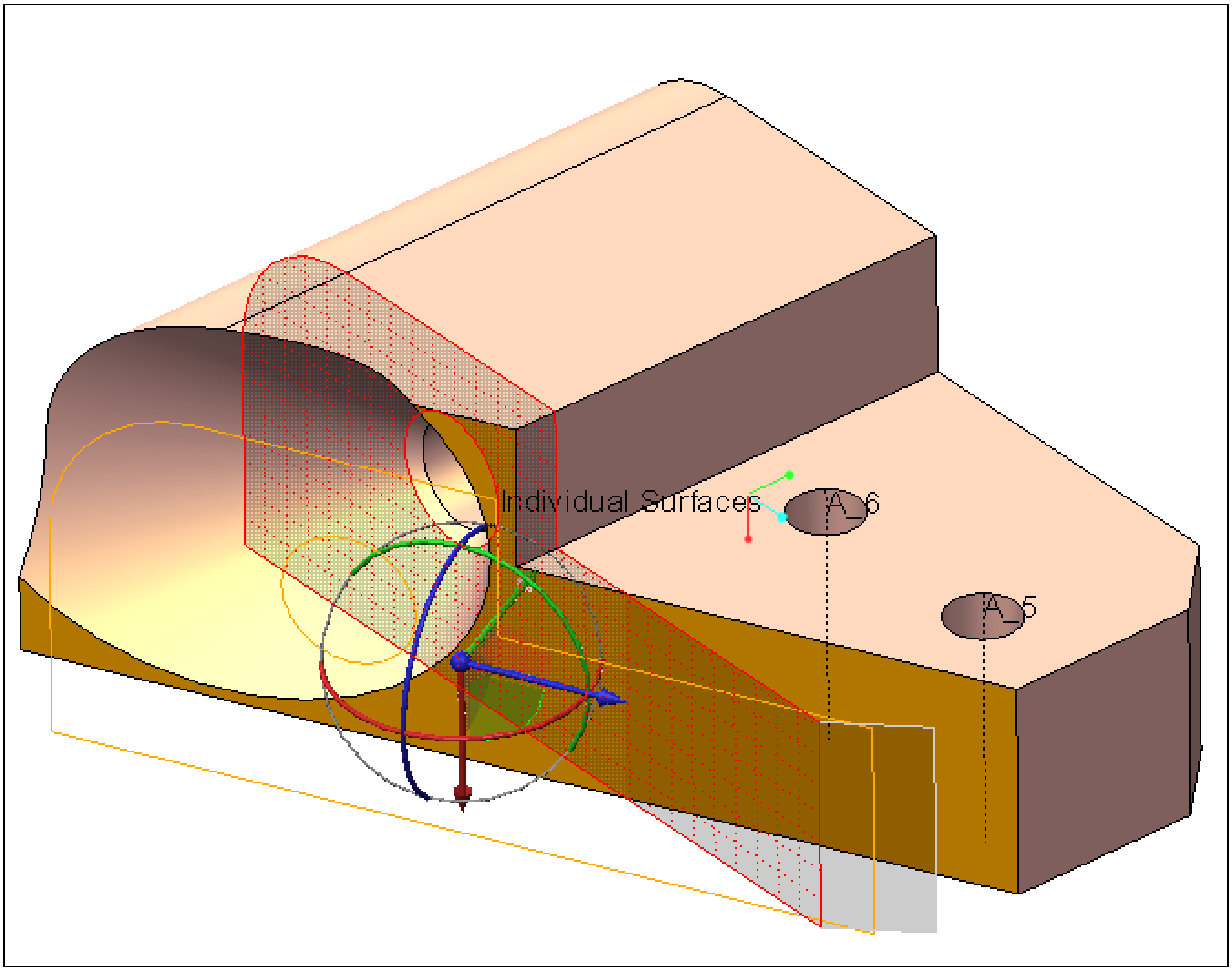
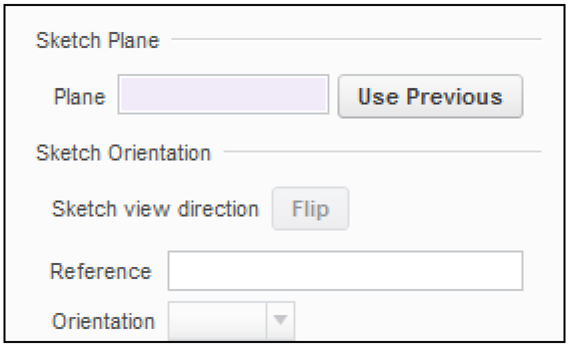


Figure 34 Design experimentation using Flexible Modeling

Using the Text

The text utilizes a variety of command boxes and descriptions to lead you through construction sequences. Also, see the downloadable Creo Parametric Quick Reference Cards at www.cad-resources.come. The following icons, symbols, shortcut keys, and conventions will be used *(command sequences are always in a box)*:

Click: **View** tab > Display Style > Shading With Edges > Standard Orientation > **Model** tab > Revolve > press **RMB** > Remove Material > press **RMB** > **Define Internal Sketch** >



> Sketch Plane--- Plane: select **FRONT** datum from the Model Tree FRONT > Reference: select **TOP** datum > Orientation: **TOP** > **Sketch**

Commands:

- > — Continue with command sequence or screen picks using **LMB**
- **Line Chain** — icon (with description) indicates command to pick using **LMB**

Mouse or keyboard terms used in this text:

- **LMB** — **L**eft **M**ouse **B**utton
  - or "**Pick**" — term used to direct an action (i.e., "Pick the surface")
  - or "**Click**" — term used to direct an action (i.e., "Click on the icon")
  - or "**Select**" — term used to direct an action (i.e., "Select the feature")
- **MMB** — **M**iddle **M**ouse **B**utton (accept the current selection or value)
  - or **Enter** — press **Enter** key to accept entry
  - or ✓ — Click on this icon to accept entry
- **RMB** — **R**ight **M**ouse **B**utton
  - *Click* (toggles to next selection)
  - *Press and hold* (provides a list of available commands)

Shortcut Keys:

Keystroke	Action
ALT	•Disables filters temporarily during selection when the pointer is in the graphics window. • Activates KeyTips when the pointer is on the Ribbon.
CTRL+A	Activates a window.
CTRL+C	Copies the selection to the clipboard.
CTRL+D	Displays a model in standard view.
CTRL+N	Opens the New dialog box.
CTRL+O	Opens the File Open dialog box.
CTRL+R	Repaints a window.
CTRL+S	Opens the Save Object dialog box.
CTRL+V	Pastes the clipboard contents.
CTRL+Y	Performs a single step redo operation.
CTRL+Z	Performs a single step undo operation.
ESC	Performs one of the following operations depending on the modeling context: • Cancels an operation • Clears the selection in the graphics window • Cancels a dragger operation • Closes the Live Toolbar • Cancels the currently active tool
F1	Opens Help for the current context.
G	Hides the current guide in 2D mode.
L	Locks or unlocks the current guide in 2D mode.
P	Shows or hides precision panels in 2D mode temporarily.
S	Switches between the Line and Arc Creation modes in 2D mode.
TAB	Pre-highlights a set of surfaces for selection based on shape selection rules. Refer to the Creo Parametric Flexible Modeling Help for more information on shape selection rules.

Text Organization

- **Lesson 1** has you complete two simple parts, an assembly, and a drawing using default settings.
- **Lesson 2** introduces Creo Parametric's interface and embedded Browser.
- **Lesson 3** provides uncomplicated instructions to model a variety of simple-shaped parts.
- **Lessons 4** and **5** involve part modeling using a variety of commands and tools.
- **Lessons 6** through **12** involve modeling the parts, creating the assembly [Fig. 35(a)], and documenting a design with detail and assembly drawings [Fig. 35(b)].
- **Lessons 13-18 (Download)** Instructions to model and document parts using more advanced features and techniques. Available as a download to minimize the size and cost of the text.
- **Projects (Download)** Provides over 100 pages of project drawings as a PDF download so as to minimize the size and cost of the text.

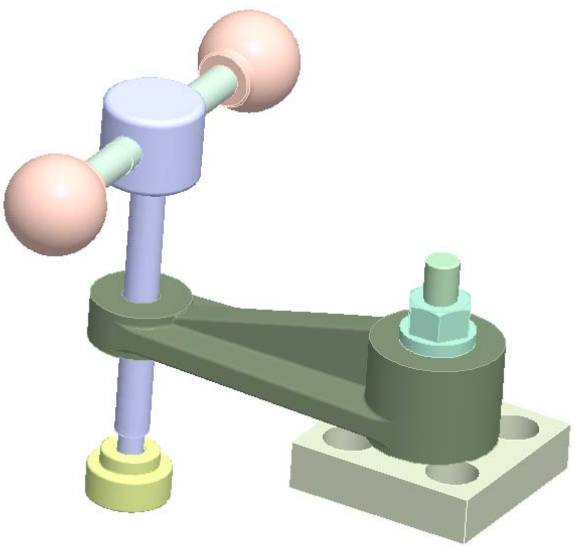


Figure 35(a) Clamp Assembly

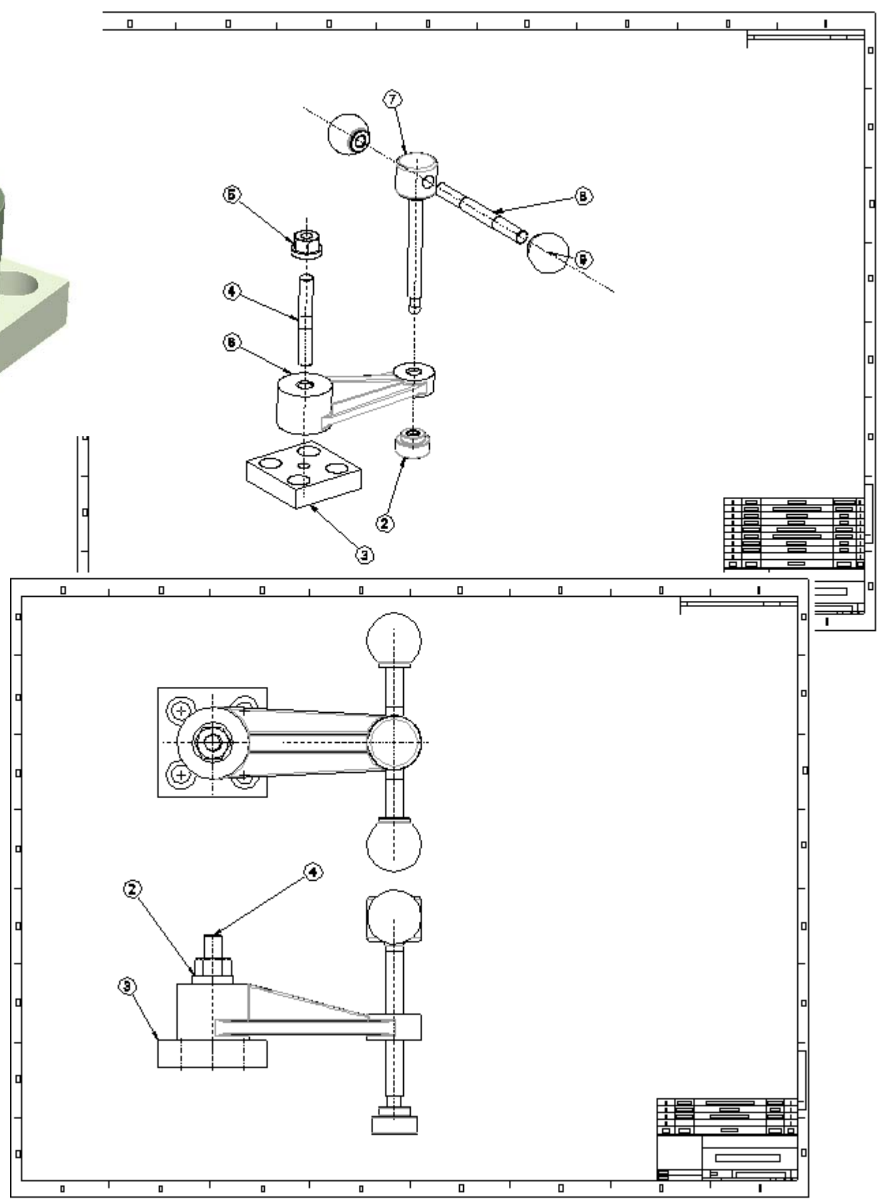


Figure 35(b) Clamp Assembly Drawing