DESIGN INTENT IN CREO PARAMETRIC

David R. Martin II

DESIGN INTENT

in Creo Parametric

Chapter 3: Building Design Intent in Parts

By David Randolph Martin II

3. Building Design Intent in Parts

A part is a model that consists of features and typically represents an object made out of a single material or composite; manufactured under a defined set of processes; and controlled as a distinct, discrete unit.

We build Design Intent into Part models primarily through the following methods:

- Feature Choice.
- Feature Options.
- Parent-Child Relationships.
- Feature Order.
- Relations.
- Behavioral Modeling.

In addition to these methods of building Design Intent we will examine other considerations, including:

- More artistic methods of creating Industrial Design geometry, such as Freeform Surfacing and Subdivisional Modeling.
- The Direct Modeling Paradigm and how it can be combined with traditional parametric modeling in Creo Parametric.

3.1. Feature Choice

We have numerous different classes of features that we can create in part models, and these different classes contain multiple different types of features. You will often have multiple choices that will create the same result. Which is the best choice?

- The first method that comes to mind?
- The easiest method?
- The fastest one / fewest mouse clicks?

If you chose any of the above... well, you might want to start over reading at page 1. The best choice depends on Design Intent. Let's look at an example to illustrate this choice:

3.1.1. Why Feature Choice Matters

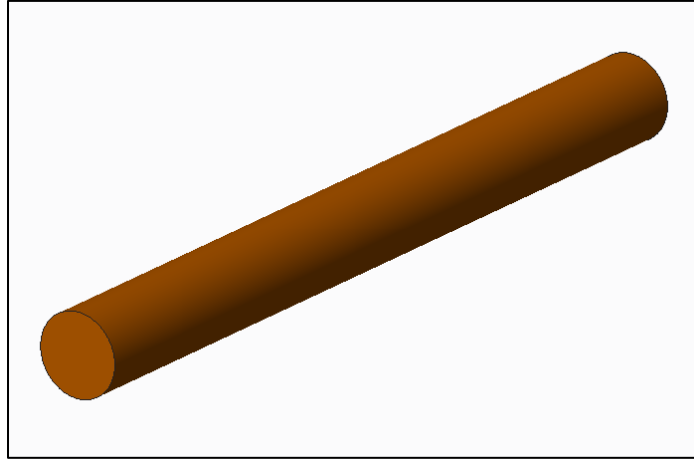Consider the following example: how can you make this cylinder?

Figure 3-1. A simple cylinder part model.

There are numerous methods in Creo Parametric. Some of the more of the obvious methods include:

- Sketch a circle then use the Extrude tool.
- Sketch a rectangle and then use the Revolve tool.
- Use the Blend tool to connect two circular sections.
- Sketch a straight line and then use the Sweep tool.

Which of these methods is the best? That is a loaded question. We don't know which method is the best until we consider how our Design Intent can change over time.

Take the following situations into consideration:

- If the cylinder might change into a shape other than circular, then the Extrude would be the best choice in this situation (see the yellow object in Figure 3-2).
- If the angle of revolution might change to something less than 360 degrees, then the Revolve would be our solution (see the blue object in Figure 3-2).
- If the critical factor is controlling the shape at both ends of the geometry, then we would want to choose the Blend (see the green object in Figure 3-2).
- If the path (trajectory) could change, then the Sweep would best allow us to account for this change in Design Intent (see the red object in Figure 3-2).
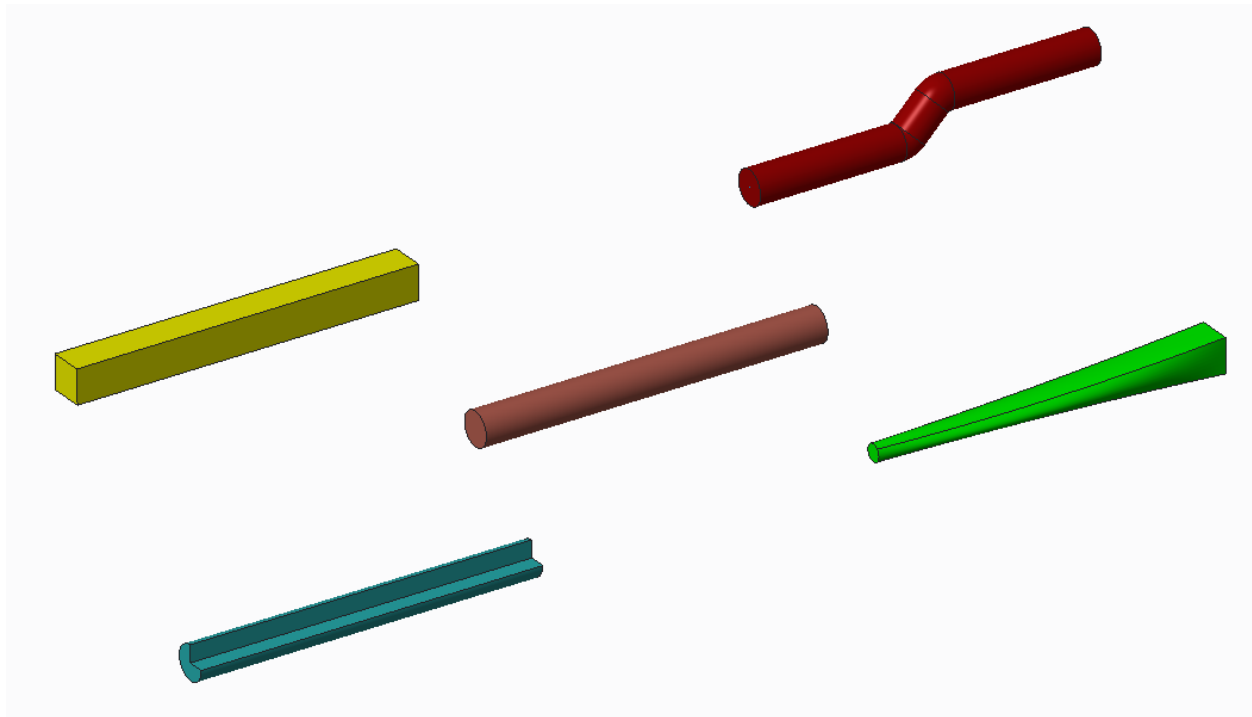
Figure 3-2. Effect of Feature Choice on Design Intent.

Before making a choice consider the most likely scenarios that might change in requirements later on. Let that be your guide when selecting which features to choose.

3.1.2. Classes of Features

There are numerous different classes of features that can be created in part models, including:

- Datum Features: imaginary references that we place in our model in order to create and control other features in our model.
- Sketch-Based Features: these were discussed in the last chapter. These features add or remove geometry from the model by taking sketches and performing operations like Extrude, Revolve, and so on.
- Engineering, or Pick & Place Features: these are features in which the basic geometry or shape is already defined, so the user specifies references to locate the features in the model, dimensions to control the size, and other various options. These include Holes, Rounds, Chamfers, Shells, and Draft.
- Surface Features: these features create a three-dimensional geometry that has no thickness, and therefore no mass. Users can create more organic, ergonomic, and aesthetically pleasing shapes that can be created with traditional Sketch-Based

Features. "Technical" or "parametric" surfaces include Boundary Blends and Fill features.

- Editing Features: these take other features or geometry and manipulate them, such as by trimming, extending, merging, and other operations.
- Feature Duplication: these features take one or more features as inputs, and allow the user to create additional features, which may be copies, mirrors, or pattern instances.

This is by far not an exclusive list. Other classes include Data Sharing Features (discussed in the next chapter in the Top Down Design section), Annotation Features used in Model Based Definition (MBD), and Flexible Modeling Features (discussed later in this chapter).

The critical point is that in design, you will often have to make choices regarding which class of feature you want to use. For example, are you going to make your primary features sketch-based or surfaces? And if so, which one of the types of features in that class will you choose? For example, if you decide to model using surfaces, will you use parametric (sometimes referred to as technical) surfacing, Freeform surfacing (the Style feature within ISDX, the Interactive Surfacing Design Extension), or Subdivisional modeling (the Freestyle feature)?

Here are some of the questions to ask yourself when choosing the class and type of feature to build:

- What are my requirements, design constraints, and goals today?
- What information do I have now?
- What information do I need?
- How might my requirements, design constraints, and goals change in the future?
- What features are most likely to accommodate my requirements?
- Which of these choices are most adaptable to change, preferably without resulting in regeneration failures or high amounts of rework? In other words, which of my feature choices are most robust and flexible?

We will quickly review some examples of some of the different kinds of features in the various classes.

3.1.2.1. Datum Features

Datums provide geometric references in your model so that you can define features. Most standard model templates provide three default datum planes and a default coordinate

system as a starting point. Datums are also used for documentation in Geometric Dimensioning and Tolerancing (GD&T). Types of Datum Features include:

- Planes Plane
- Axes Axis
- Points Point ▾
- Coordinate Systems Coordinate System
- Curves

> By default, an extruded circle and a sketch revolved through 360 degrees and generate axes automatically. Extruded arcs and revolves less than 360 degrees do not create axes. The configuration option *show_axes_for_extr_arcs* will generate axes for both situations.

3.1.2.2.Sketch-Based Features

As discussed in the previous chapter, Sketch-Based features are some of the most widely used features in part models. The base (first solid) feature is often based on a sketch. These include:

- Extrude Extrude : add or remove material normal to the Sketch.
- Revolve Revolve : add or remove material about an axis of revolution.
- Sweep Sweep : move a sketch along the main trajectory. Additional trajectories can be used to control the shape, and the section can be allowed to vary along the main trajectory by a variety of means.
- Blends Blend : connect multiple sections together to create a feature.

Note that in addition to creating solid geometry, Sketch-Based Features can also create surfaces and quilts (non-solid features).

A Note About Internal Sketches

A Sketch does not need to exist prior to creating an Extrude or a Revolve; it can be created within the feature. That is known as an Internal Sketch. When the Sketch is created first it is known as an External Sketch.
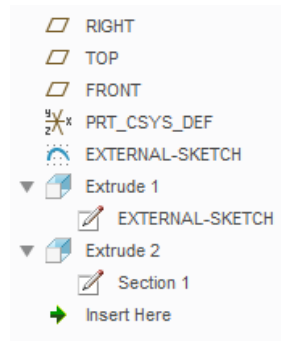
Figure 3-3. External vs. Internal Sketches for features.

External Sketches provide more flexibility, because

- The same sketch can be used to drive multiple features.
- A feature can quickly be changed to use a different external sketch. With an internal sketch, all the entities would need to be deleted and the new sketch created.

An External Sketch can be converted to an Internal Sketch by using the Unlink button in the Placement tab of the Sketch-Based Feature.

Figure 3-4. Unlinking an external sketch from a feature dashboard.

3.1.2.3.Editing Features

These features perform operations on existing features and geometry in our model. They include:

Figure 3-5. Editing features in the part mode ribbon.

- Trim: use one object to reduce the extent of another object.

- Merge: stitch two or more quilts together.
- Extend: increase the boundaries of a surface.
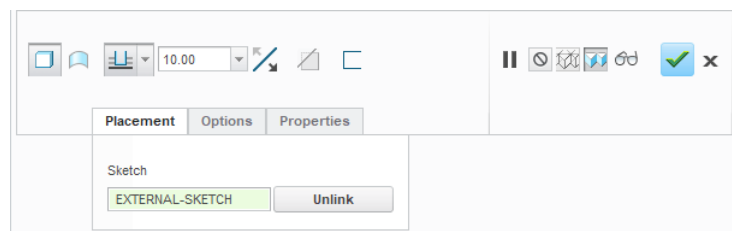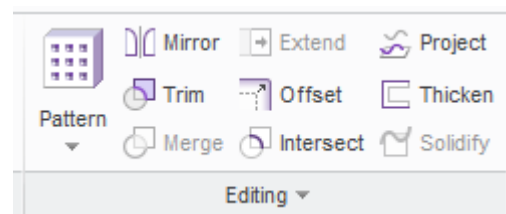- Offset: create new curves or surfaces of moving a parent entity a specified distance in a specified direction.
- Thicken: create solid geometry by offsetting a surface, creating side surfaces, merging the result, and filling in the interior volume.
- Solidify: create solid geometry, either adding or removing material, using a quilt as an input.

3.1.2.4. Engineering Features

Engineering Features sometimes referred to as "Pick and Place Features," are the ones in which the basic shape is already defined. The feature is placed in the model by specifying the location references and attributes regarding size and extent.
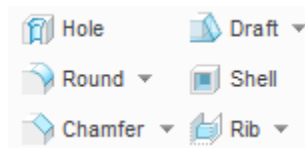
Figure 3-6. Feature tools from the Engineering group on the Model tab.

Engineering features include:

- Hole: generally a circular cut in a model. They include Straight, Standard, and Sketched holes, but they can be tapered, and have countersinks and/or counterbores.
- Rounds: also known as fillets, they break a sharp corner with rounded or conic geometry.
- Chamfer: remove sharp corners from an object, leaving a flat beveled surface.
- Draft: add taper to the surfaces of a model in order to facilitate manufacturing via molds and casts.
- Shell: hollow out a part to a given thickness, optionally removing specified surfaces so as to provide access to the interior volume.

Holes vs. Extrudes

Holes present an interesting choice when it comes to Design Intent. A simple hole can be created by simply extruding a circle as a cut. If the hole could likely change to a slot, then an Extrude would be preferred. But otherwise, hole features provide numerous advantages, including:

- Changing between different dimensioning schemes.
- Easily adding counterbores, countersinks, and exit countersinks.
- Selecting standard holes from preconfigured lists for tapped, clearance, and tapered holes. These can include thread surfaces and automatically generated notes to be used in drawings.
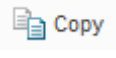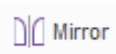- Reducing regeneration time with lightweight holes.
- Searching for holes and placing them with layers for suppression of downstream processes.

3.1.3. Duplication Methods

The power of computers lies in their ability to perform operations over and over again, faster than we can, saving us the time and effort of doing things manually. Creo Parametric takes the same approach with duplicating features, using the following tools:

- Copy Copy and Paste Special Paste Special
- Pattern Pattern
- Mirror Mirror
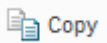
Duplicating features using these methods provide greater control and Design Intent than creating multiple entities in a Sketch (which is a bad practice).

3.1.3.1. Copy and Paste Special

While Copy and Paste saves you from recreating features, it does not build Design Intent in the model because there is no dependency between the original object(s) and the copies. However, Paste Special does allow dependency; a Parent-Child Relationship can be established so that changes to the original are propagated to the duplicate.

However, if it's been a while since you last used this command, I recommend practicing on some sample models first. There are nuances to the command that are not intuitive.

To perform this operation, do the following:

1. Select the feature(s) you wish to copy.
2. Use CTRL-C or the **Copy** Copy command.

3. Click the Paste drop-down menu (just below the Copy command in the **Operations** group on the **Model** tab) and select the **Paste Special** ⧉ Paste Special command.

Here is where things get tricky. The Paste Special dialog box will open:
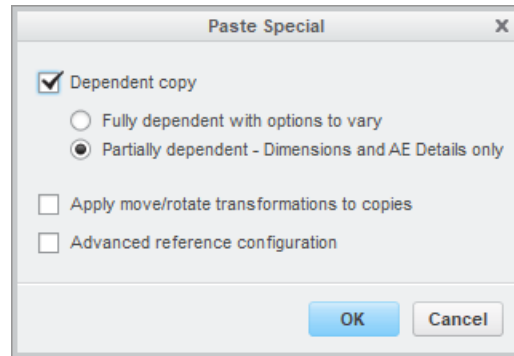
Figure 3-7. The Paste Special dialog box.

By default, **Dependent copy** and **Partially dependent; Dimensions and AE Details only** are selected. Some notes about this first option:

- It may be necessary to use this in conjunction with **Advanced reference configuration** in order to locate the copy in a new place.
- If you attempt to edit the definition of the sketch for a copied Sketch-Based Feature, you will receive a warning:
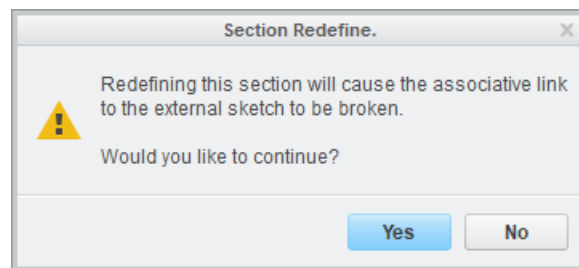
Figure 3-8. Warning for editing definition of a dependent sketch.

- If you right click on a copied Sketch-Based Feature, the command **Copied Feature > Make Section Independent** is available to break the Parent-Child Relationship.

- If you attempt to edit any dimensions in the copy, you will see this warning:
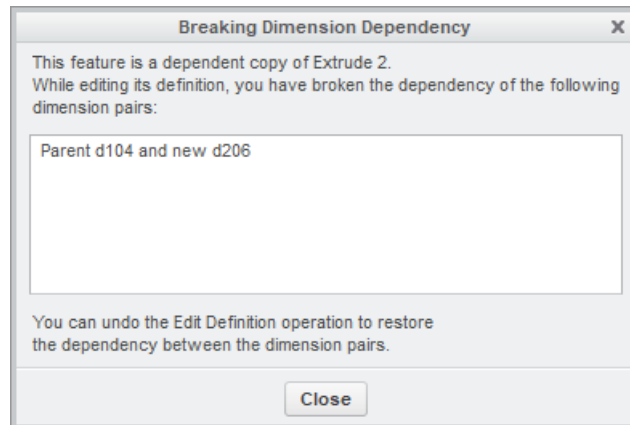
Figure 3-9. Break dimension dependency warning.

Here are some notes regarding the **Fully dependent with options to vary** choice:

- The copied feature(s) will have the word Copied at the start of its name in the Model Tree.
- Edit Definition is not available for these features.
- Any options that are varied can be viewed in the Varied Items table:
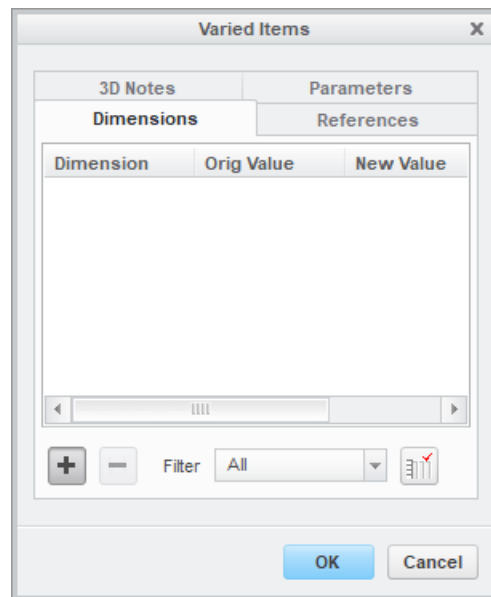
Figure 3-10. Copied feature Varied Items dialog box.

- Right clicking on the copied feature(s) will provide access to the **Copied Feature** commands to access the Varied Items table and control the dependency (see Figure 3-11).
    - o **Break dependence** temporarily suspends the duplicate from updating with changes to the original.
    - o **Remove dependence** permanently stops changes from being propagated.
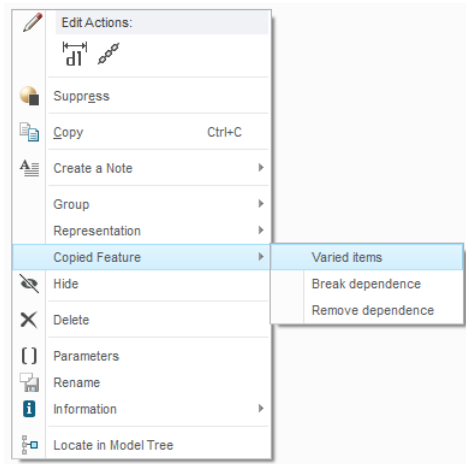
Figure 3-11. Copied Feature right mouse button commands.

Again, I highly recommend familiarizing yourself with the repercussions of the various options before using the command to be aware of its limitations to future changes.

3.1.3.2.Patterns

Patterns allow us to create multiple instances of a feature, or a group of features, using a variety of methods:

- Dimension: define increments for the dimensions used to define the feature(s).
- Direction: create patterns in one or two directions by translation, rotation, coordinate systems, or a mix of these.
- Axis: create a rotational pattern in one or two directions.
- Fill: use an open or closed curve as either a boundary or a trajectory for locating instances.
- Table: fill in a spreadsheet for values to define the instance's location and size.
- Reference: use an existing pattern as the basis for creating a new pattern. More on this below.
- Curve: create a pattern by following a defined path.

- Point: use a Sketch or Datum Point array to define the location of instances. This is much quicker and easier than a Table Pattern.

> Tip: Point Patterns can use Sketcher Coordinate Systems in addition to Points. This is very convenient for modeling cabling connector entry ports and placing forms in Sheetmetal parts.

By default, under the most basic circumstances, all the instances will be created identically to the lead feature of the pattern. In this way, we build Design Intent because changes to the lead feature(s) are propagated to the pattern instances.

However, we can get the instances to vary using a number of techniques:

- The dimensions describing the shape of the feature could be incremented. This increment can be defined using a Relation. This applies to Dimension, Direction, and Axis- Patterns.
- The pattern uses a table that includes feature dimensions in addition to locating dimensions.
- The patterned features could have Sketch References, depth options, or some other Parent-Child Relationship that cause them to vary.

Reference Patterns

Reference Patterns is a particularly strong method of building Design Intent into models with feature duplication. If the original pattern changes, then the pattern that references it automatically updates.

This is even stronger at the assembly level. Let's say we have a pattern of holes in a part. Then we have a series of components assembled to those holes, such as a vibration mount, washers, screws, and nuts. If the pattern of holes changes (we increase or decrease the number of instances or change the spacing between the holes) all the components will update automatically. Additional components will be added or removed if necessary, and the locations updated. This is the essence of Design Intent: our models update when changes happen, exactly as we expect them to, without us having to do anything.

3.1.3.3.Mirror

If our model has one, two, or even three planes of symmetry, we can use this to our advantage by modeling only half, a quarter, or an eighth of our part, and then create one, two, or three Mirror features.

With this technique, the mirrors are always dependent on the original portion, so if you modify a feature, those changes are propagated to the mirrored portions. The process of doing this is as follows:

1. Select the top node in the Model Tree (the line with <model_name>.PRT).
2. Select the **Mirror** ⫿⫾ Mirror command.
3. Select the plane or surface you want to mirror about.
4. Click the middle mouse button to complete the feature.

One or more features in the model can also be mirrored, following the same general procedure. By default, it is set to **Dependent** using the **Partially dependent; Dimensions and AE Details only** option. These settings can be changed from the **Options** tab in the dashboard:
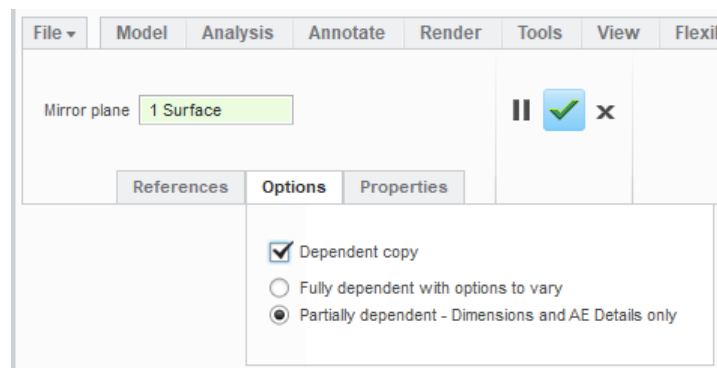
Figure 3-12. Dependency for a mirrored feature.

See Section 3.1.3.1 on Copy and Paste Special for more information regarding the dependency options.

3.2.   Feature Options

The dashboard used to create various features contains numerous additional attributes and options that can be specified to define your feature. There are too many features to cover all the options, but we will discuss some examples.

3.2.1. Depth and Angle Options

The typical depth options for an Extrude or a Hole are:

- Blind: a numerical value is entered.
- Symmetric: a numerical value is entered, but half the depth is applied on each side of the Sketch Plane.
- To Next: the feature extends until it intersects the very next surface.
- Through All: the feature goes through all geometry that exists in the model. This option is appropriate for removing material only.
- Through Until.
- To Selected.

The angles for a Revolve feature include:

- Variable: similar to Blind, a numerical value is entered.
- Symmetric.
- To Selected.

Blind, Symmetric, and Variable are based on pure numbers; they extend the depth or angle specified. Unless those values are controlled by Relations (more on those later in this chapter), they don't update for changes made to the model.

The other options can update the resulting feature with changes made to their references or geometry prior to them in the model. For example,

- If geometry appears in the path of an Extrude or a Hole set to To Next, the feature will terminate.
- If geometry appears in the path of an Extrude or a Hole set to Through All, it will be intersected.
- If the reference chosen for Thru Until or To Selected moves, then the Extrude or Hole will update its depth, and a Revolve will update its angle.

Depth and angle options build Design Intent into our models.

A Note About To Selected vs. Thru Until

These depth options fundamentally do the same thing. However, Thru Until has a critical distinction: the feature must fully intersect the reference selected. If that reference gets changed such that the feature no longer fully intersects it, the feature will fail.

Therefore, To Selected is a much more robust option when compared to Thru Until. To Selected should always be used instead of Thru Until (unless you would want a potential regeneration failure triggered by a change to the reference, but I can't fathom a situation like that).

3.2.2. Other Examples

All features have various options that can be selected during creation and edit definition. If you are a new user, take a minute or two to read the various choices in the dashboard and the right mouse button menu. If you don't use those options today, hopefully, they will stick in your brain when you have a more complicated problem that you are trying to solve.

We'll take a look at some options for a few features. Think about how those choices will result in the feature reacting to change:

Rounds

- Is the radius driven by a curve or a point / vertex?
- Is it a full round where the removed surface determines the size of the feature?
- Constant or variable radius. For variable radius rounds, are you locating the radius handles using length ratio or a point / vertex?

Draft

- What object are you using as the Draft Hinge?
- If you are splitting the Draft? What are you using to split it?
- Are you using variable draft angles? If so, are you locating the handles using length ratio or a point / vertex?

Shells

- Are you applying a non-default thickness to any surfaces?
- Are you excluding any surfaces from the draft?

Patterns

- Can you change from a General Pattern to Identical or Varying for better regeneration performance, or will the pattern become more likely to fail?
- For Curve Patterns, do you want to control the spacing or the total number of instances?

- For Fill, Point, and Curve Patterns, do you want the instances to follow the shape of some other surface?

Your model's Design Intent is determined by all the choices you make, especially those that tie your feature to other features. This leads us right into our next section.

3.3.   Feature References: Parent-Child Relationships

Parent-Child Relationships have been mentioned a few times before in this book. They are critical because they determine how our models are parametric, how changes are propagated from one feature to other features.

Parent-Child Relationships in Creo Parametric work in the following manner:

- Whenever you create a new feature, you have to select existing features in the model as references, in order to determine things like location, shape, and size.
- The new feature you are creating will be the child of the existing feature.
- The existing feature will be the parent of the new feature.
- In Creo Parametric, children are absolutely dependent on their parents. Changes to the parents will be propagated to the children. And if you want to delete or suppress a parent, by default, Creo Parametric will want to delete or suppress the child.

Parent-Child Relationships are also created in assemblies. When we assemble a new component to an existing component, that new component is the child and the existing component is the parent. Parent-Child Relationships can also be created in assemblies via External References, which will be discussed in Section **Error! Reference source not found.**.

Table 1. Parent-Child Relationships

Feature Class	Feature	References
Datum	Planes and Axes	Location references, display size reference
	Coordinate System	Origin location reference, axes directions references
Sketch	Sketch	Sketch Plane, Orientation Reference Plane, Sketch References
Sketch-Based	Extrude	Sketch, Depth Options
	Revolve	Sketch, Angle Options
	Sweeps	Trajectories, section plane control reference, x-vector / direction references, sketch placement point
	Blends	Sketches, boundary condition references
	Ribs	Sketch
Engineering	Rounds	Edges and surfaces to define placement; curves, vertices, and points to define radius
	Chamfers	Edges, surfaces, and corners to define placement
	Holes	Placement surfaces, location references, depth options
	Shells	Surfaces to be removed, excluded surfaces, and non-default thickness surfaces
	Draft	Draft surfaces, draft hinge, pull direction, split references
Editing	Mirror	Objects to be mirrored, mirror plane
	Trim	Quilts, trimming object (curve, quilt, plane)
	Merge	Quilts

Given that Parent-Child Relationships build Design Intent, how do we use them to build better models?

- Choose references carefully. The more stable they are, the better. By stable, I mean that they are either not likely to change, or will change in very predictable ways that aren't disruptive.
- The fewer the references, the better.
- Surfaces are better choices than edges.
- If you can use Advanced Selection Methods or Intent References (see below) that's even better.
- Default datum planes are the most stable features in your model. They can make great parents. However, go with the logical choice for how you want changes to be propagated in your model.

3.3.1. Advanced Selection Methods

The basic methods of selecting references in Creo Parametric include:

- Left-clicking on entities in the graphics area (or Model Tree when appropriate).
- Using the CTRL key to perform a multi-select.
- Using Query Select (tapping the right mouse button quickly) or Pick From List.

These methods allow you to select explicit references; that is, you are picking a specific surface or edge. However, for certain kinds of features, you can use advanced selection methods, either using the SHIFT key or the Details button in various dashboards. These advanced selection methods are a little more robust than selecting explicit references.

The types of edge chains are:

- From-To chains.
- Tangent Chains.
- Boundary Chains.
- Surface loops.

The types of surface sets are:

- Solid surfaces.
- Loop surfaces.
- Seed and boundary.

For information on how to construct these, refer to the Creo 3.0 or Creo 4.0 Quick Reference Card.

3.3.2. Intent References

There is an even more robust method of selecting references for features like Rounds, Chamfers, and Draft. Rather than selecting specific edges or surfaces, you can select the edges or surfaces associated with a given feature.

For example, let's say you want to apply rounds to the edges of a feature. If that feature is a brick, you could select the top four edges one by one. But supposed the feature changes shape, and instead of a brick, it becomes an extruded racetrack. It's still four edges, but the edges are different. Or maybe it changes to a circle (one edge to pick), or an octagon (eight edges), or some n-sided shape, even including internal loops.

In such a situation, the round would fail to regenerate after the change.

If, however, we picked the Intent References, the round would not fail to regenerate, because we are selecting the edges associated with the feature, not specific explicit edges.

The way you reach the Intent References is via Query Select (tapping the right mouse button over the reference until the selection appears) or Pick from List.
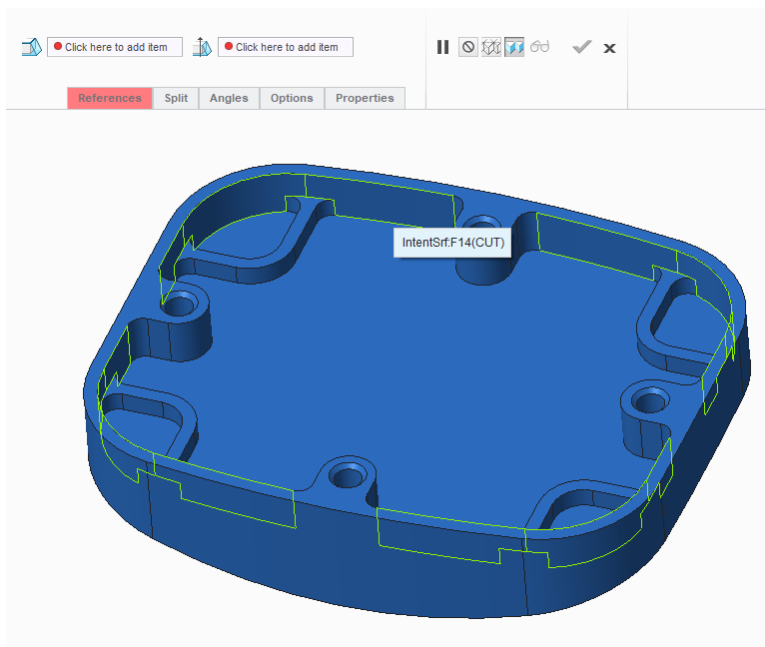

Figure 3-13. Using Query Select to pick the Intent References of a feature to create a Draft feature.

Selecting Intent References when possible is definitely a healthy practice for building robust design models.

3.4.   Feature Order

Creo Parametric retains the history of the model. Parts are regenerated in the order in which they appear in the Model Tree. Changing the order of these features can change our Design Intent.

Consider the following examples:

- A Through All cut removes material from all features that appear after it is in the Model Tree. If a feature is moved above the cut, the cut will not affect it.
- Features with a depth or angle option To Next will have different geometry results depending on which features appear before them in the model.
- Rounds and chamfers propagate to tangent edges by default, so if the geometry is added or removed prior to the feature, it can affect the extent of the round or chamfer.

In assemblies, a component can only be assembled to reference components that appear after it in the Model Tree. If you Edit Definition of a component, all the components that appear after it in the Model Tree will be temporarily suppressed, thus making them unavailable for selection.

Because order and history have such an effect on our models, we need methods to change the order. We do that through **Insert Mode** and **Reorder**.

3.4.1. Insert Mode

Sometimes we want to add features or components at an earlier point in our model's history, in other words, higher up in the Model Tree. You'll notice at the bottom of the Model Tree is an arrow that says Insert Here:
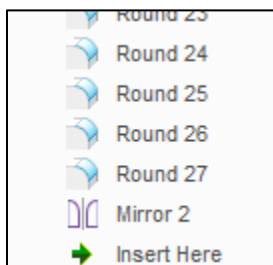

Figure 3-14. Insert Here arrow at the bottom of the Model Tree.

You can simply drag the arrow up the Model Tree until you reach the point where you want to add more features. The features after the Insert Here arrow will be temporarily suppressed in

the model, and a note in the lower left corner of the graphics area indicates that you are in Insert Mode.

> It is a bad practice to save models in Insert Mode. ModelCHECK examines for this condition.

Insert Mode can also be used in assemblies. However, in order to see the Insert Here arrow, Features must be displayed in the Model Tree.

Dragging the Insert Here arrow can be a pain in the neck in a model that contains a lot of features. As an alternative, you can perform the following steps:

1. Right click the feature after which you want to add features, hold down the right mouse button, and select **Insert Here**.
2. Create the new features you want.
3. Right click on the Insert Here arrow and select **Exit Insert Mode**.
4. When prompted to resume features that were suppressed when activating Insert Mode. Click **Yes**.

While in Insert Mode, a note reading "INSERT MODE" will appear in the lower left corner of the Graphics Area.

3.4.2. Reorder

The quick and easy way to reorder features is by simply dragging and dropping them in the Model Tree. Be aware that you will not be able to reorder a feature before its parents or after its children.

Sometimes it's not always easy to drag and drop the features where you want them to go. In those situations, click **Operations > Reorder**, which will open the following dialog box:
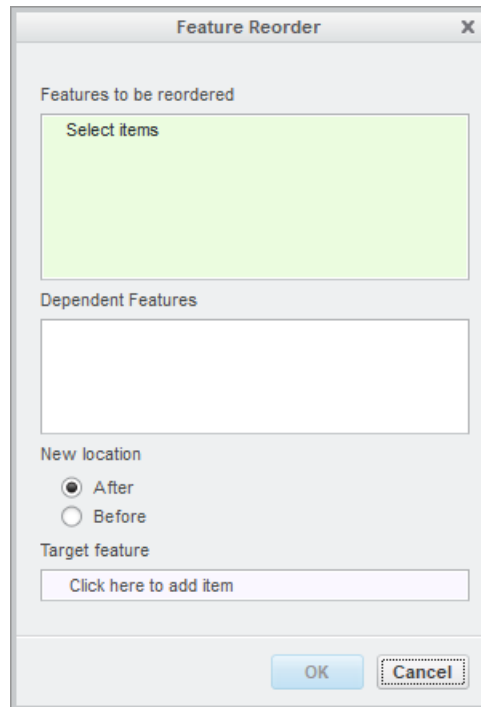
Figure 3-15. Reorder dialog box.

The process is as follows:

1. Select the features you want to reorder using the CTRL key.
2. (Optional) If desired, change the **After** radio button to **Before**.
3. Select in the Target feature collector or RMB **Target Feature**. Then select the feature you want to reorder after or before, depending on which radio button is selected.
4. Click **OK**.

3.5. Relations

Relations are simply mathematical expressions that relate some dimensions and parameters in your model to other dimensions in your model. Relations add Design Intent to your model because they increase the level of automation.
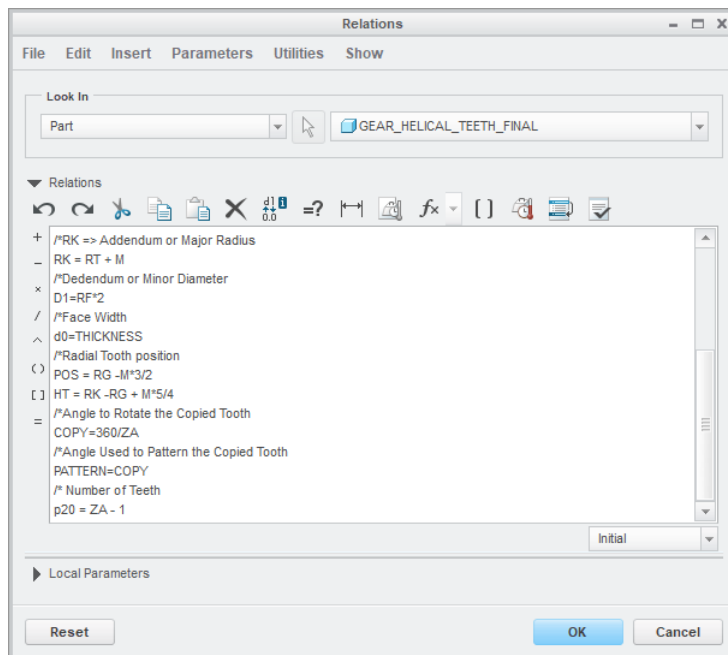
Figure 3-16. The Relations dialog box.

Some situations in which you could use Relations include:

- You want your model to change proportionately. For example, if the length changes, the width should update.
- You have a pattern, and you want the number of instances to update with changes to the model dimensions.
- Your model has mounting locations near the edges of the part, but you don't want to dimension those locations from the edges. You want them dimensioned from a mid-plane or relative to some other reference, but you also want them to update with size changes.
- You have a Family Table, and the features in instances should update with changes to Family Table dimensions.
- You have a template part for generating custom models. For example, a part model uses Pro/PROGRAM or some other input methods to generate bearings, gears, springs, or wave washers.

There are three main types of Relations that can be written:

- Equality: evaluate an expression and assign the result to a dimension or parameter.
- Comparison: evaluate if something is less than, or equal to, or greater than, or equal to some value or expression. If the result is false, a warning appears in the Message Area during regeneration.

- If – Then – Else: evaluates a condition and determines the actions to follow depending on whether it is true or false.

Some other capabilities in Relations include:

- A variety of functions are available including, trigonometric, logarithmic, string editing, mass properties, significant figures, and even cable harness information for use in Relations.
- Parameters used in Relations can be assigned units. By default, Relations are unit sensitive, but this can be turned off.
- Relations can extract values from a Datum Graph feature for evaluations that can't be driven by an explicit equation.

Although assemblies aren't discussed until the next chapter, Relations can also be built between components. For example, you can control the length and width of a lid to change with the size of the opening that it covers.

Healthy Practices for using Relations include:

- Always precede Relations with a comment line; any line starting with a forward slash and an asterisk (/*). It's frustrating to open up a model with numerous Relations, and then spend significant time figuring out what they do in the model.
- Rename dimensions to make the model user-friendly. See the preceding Healthy Practice.
- Use spaces and blank lines for readability. You may have eagle eyes, but not everyone you work with does.
- Avoid writing Relations in Sketches. This was mentioned in the previous chapter. Relations written at that level can be hard to detect and can leave your model with a permanent yellow regeneration status.
- Do not write Circular Relations where something ends up ultimately being defined by itself. Although Relations are regenerated in order, just like features, Circular Relations can cause errors and result in unpredictable geometry.
- As mentioned above, be aware of the limitations of Relations written at the assembly level. For better alternatives, see the section on Top Down Design in Chapter 4.
- Most of all, use them! They are a powerful tool for automating how your model reacts to changes.

3.6. BMX: The Ultimate in Feature Based Parametric Modeling

BMX stands for the Behavioral Modeling Extension. The two main parts of this module are Datum Analysis Features and Feasibility / Optimization Studies. The basic concept behind the workflow is as follows:

- Datum Analysis Features perform calculations on your model. These calculations can be basic (e.g., length, distance, volume) or advanced (e.g., mass, center of gravity, surface quality). The results of these calculations are parameters and even datum features (e.g., coordinate system at the center of gravity).
- Feasibility Studies allow you to modify model dimensions within a specified range in order to drive these calculations to a desired value (e.g., the distance between the center of gravity and my rotation axis should be zero). An Optimization Study adds a goal minimizing or maximizing some other quantity in your model to the Feasibility Study, so as to find the best possible result over the solution space. For example, we may want to minimize mass or maximize internal volume.

In other words, we are going to generate the values we want to track, and then get Creo Parametric to make those parameters have the values we want them to have.

Types of Datum Analysis Features include:

- Measure:
  - o Distance
  - o Area
  - o Length
  - o Diameter
  - o Angle
  - o Volume
- Mass Properties for models and cross sections:
  - o Mass
  - o Surface area
  - o Center of gravity
  - o Inertia (e.g., inertia tensors, principle moments of inertia, and radii of gyration)
- Geometry:
  - o Inside and outside radius of a surface
  - o Surface curvature and surface normal
  - o Gaussian and section curvature

- o Dihedral angle
- o Deviation
- o Surface slope
- Relations: calculate parameters based on the parameters in your model and/or parameters generated from Datum Analysis Features.
- Excel: pass dimensions and parameters to an Excel spreadsheet, perform calculations on them, and receive the results back.
- Prime / Mathcad Analysis: similar to an Excel Datum Analysis Feature, except this passes values to and from a Mathcad worksheet instead of an Excel spreadsheet.
- User Defined Analyses: create a custom analysis using a special kind of datum point called a Datum Field Point that is calculated over the domain of a surface or an edge.
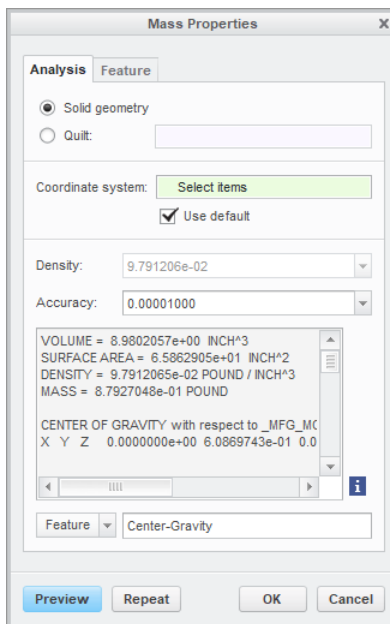
Figure 3-17. Creating a Datum Analysis Feature for a model's mass properties.

Once these values are calculated, we can then set up a Feasibility / Optimization Study, consisting of the following:

- Design Constraints: one of more parameters driven to be equal to, less than, or greater than some specified value.
- Design Variables: model dimensions and parameters that are allowed to vary between a minimum and maximum value to meet the Design Constraints.
- Goals (Optimization Studies only): some quantity to be maximized or minimized over the solution space.

Figure 3-18. The dialog box for defining Feasibility and Optimization Studies.

Since these are features in your model they can regenerate whenever the model changes (Since they can take time to process, you might not want them to regenerate all the time).

That's why BMX is the ultimate in feature-based parametric modeling: your part updates automatically to find the optimal solution when other things in your model change.
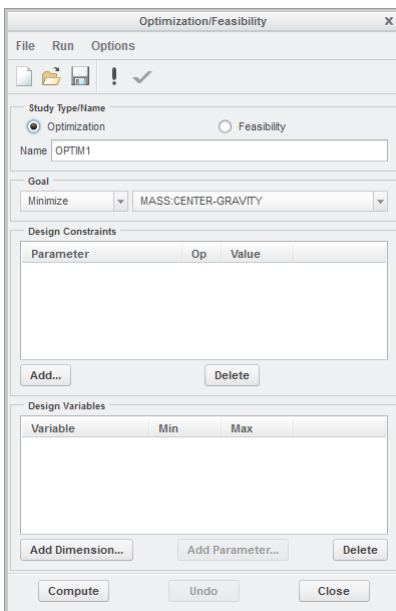
3.7.   Other Considerations

This book focuses on the core modeling paradigm behind Creo Parametric: feature-based, history-based parametric modeling. It is not the only modeling paradigm available in CAD. It has its strengths and its weaknesses. To PTC's credit, they address the limitations of feature-based, and history-based parametric modeling by incorporating: (1) surface modeling techniques that aren't hindered by the requirements of parametric modeling, and (2) direct modeling capability independent of a model's features and history.

In this way, Creo Parametric models become hybrids of the various techniques.

3.7.1. Industrial Design

Industrial Design (ID) is the art of creating concepts for the shape of products that are visually and artistically appealing. ID is an important, and often early, part of the design process for

products like automobiles, consumer electronics, appliances, personal healthcare devices, and so on.

Parametric surface features such as, Boundary Blends and Swept Blends don't lend themselves easily to these more artistic and aesthetically pleasing pursuits. It's very difficult to define the explicit underlying curves for boundaries and trajectories. Also, once created, it's arduous to modify dimensions to get the shapes tweaked into what "feels" right.

Over quite the past few years, Creo Parametric (and its predecessor versions) have moved beyond the traditional, highly defined features into more artistic techniques for Industrial Design. We will discuss two of those techniques: Freeform Surfacing (via the Style feature) and Subdivisional Modeling (via the Freestyle feature).

3.7.1.1.Freeform Surfacing

Parametric surface features, mentioned at the beginning of chapter 2, require explicitly defined inputs in the form of curves. Constructing curves, especially three-dimensional curves in Standard Mode can be done with a variety of tools, such as:

- Curve Through Points: which requires explicitly defined points, such as offset from a coordinate system.
- Sketches: which are only two-dimensional, and aside from the spline, require highly defined geometric entities (although intermediate points can be dimensioned, a spline requires dimensions only on the endpoints).
- Curves from an equation: which require x, y, and z to be defined in terms of an independent variable t.
- Curves created from an editing tool such as Project, Wrap, Intersect, Offset, and so on , require some edge, pre-defined curve, or pre-defined surfaces as input.

In other words, defining surfaces using standard parametric modeling can be tedious, difficult, time-consuming, and nearly impossible in some situations. And often we want to design by "feel," by pushing and pulling points, curves, and surfaces to see the resulting shape immediately.

This is what is known as Freeform Surfacing and is created by using the Style feature in the Interactive Surface Design Extension (ISDX).

A Style feature is known as a "super feature" because it contains multiple surfaces and curves. The Style environment accordingly contains a Style tree listing these different entities. But this

is where Style has a significant advantage over standard parametric modeling. In Standard Mode, when you edit the definition of an entity, all the subsequent entities in the Model Tree are temporarily suppressed. You cannot see the effect of your change immediately. However, in the Style environment, you can edit the definition of a curve or surface, and all the other entities remain visible and update as you're working, providing immediate feedback.
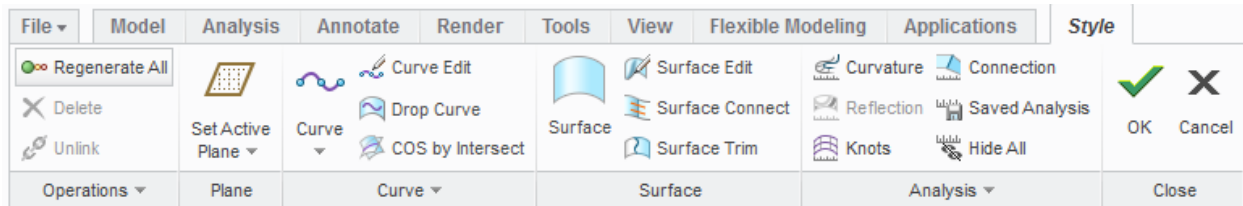

Figure 3-19. The Style toolbar.

The kinds of curves that can be created in the Style environment include:

- Free Curves (3D).
- Planar Curves (2D).
- Curves on Surface.
- Radial Path Planar Curves (curves located on a "soft plane" normal to a curve).
- Drop Curves.
- Proportional Copies.

These curves can be manipulated and controlled. For example, the endpoint conditions can be defined, and internal points can be snapped to geometry.

The kinds of surfaces that can be created include:

- Boundary Surfaces (3 or 4 sides).
- Loft Surfaces.
- Sweep Surfaces.

Surfaces can have G0 (connected), G1 (tangent), G2 (curvature continuous), or G3 (matching acceleration) continuity at their boundaries. Within Style, there are editing tools like Trim and Mirror.

Style curves and surfaces can generate dimensional controls if desired. These dimensions can then be used like other Standard Mode dimensions. They can be edited and used in Patterns or Family Tables.

Parametric surfaces can use free-form curves as inputs, and similarly, surfaces created in ISDX can use parametric curves as inputs. Editing operations from Standard Mode can be applied to Style geometry.

3.7.1.2.Subdivisional Modeling

Another technique for designing complex, organic, and aesthetically-pleasing surface models is so that they resemble modeling with clay. We start out with a blob of matter, and then push and pull it, refining the shape as we go. It is a very artistic, intuitive method in which we either can't or don't want to define a full set of dimensions and parameters.

This is known as Subdivisional Modeling. You start off with this lump that has a control mesh surrounding it. You can break up the mesh (subdivide it) and then push and pull the edges and surfaces of the mesh to create the desired shape.
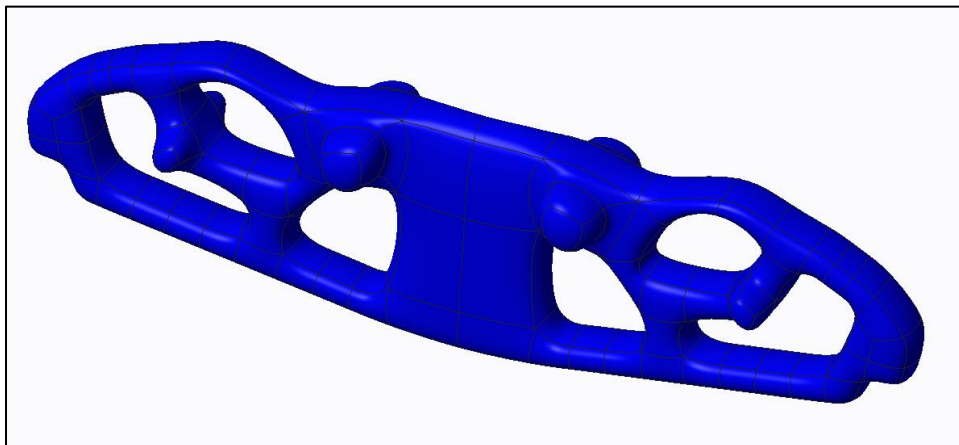
Figure 3-20. Freestyle feature.

These kinds of models are created within the Freestyle command. You start by creating an initial shape known as a Primitive. These can be 2D like circles, rings, squares, and triangles. Or they can be 3D like spheres, cylinders, cubes, and tori (donuts).

Since this is more of an artistic approach, there is no standard uniform. But the kinds of operations you can perform to achieve your desired geometry include:

- Splitting mesh surfaces and edges.
- Dragging, rotating, and scaling mesh faces and edges.
- Extruding faces.
- Connecting mesh faces.
- Creating hard and soft creases.

- Mirror the mesh about a plane.

> Creo 4.0 adds the ability to have multiple objects in the same Freestyle feature.
> To facilitate manipulation, a Freestyle Tree is also added.

3.7.2. The Direct Modeling Paradigm

So far, we have concentrated on feature-based, history-based parametric modeling, the basis for design in Creo Parametric. This modeling paradigm is characterized by the following:

- Our part models consist of features.
- These features are driven by dimensions and parameters.
- These features have parent-child relationships.
- Our part and assembly models are history dependent; the order in which objects are regenerated affects what geometry is generated.

However, there are alternative paradigms, and these paradigms have benefits at different stages of the design process.

In the early stages, we may want to iterate numerous concepts for shape and function, without having to define all the explicit dimensions and parameters that drive the model. We want to generate ideas and geometry quickly.

In the later stages, once our parts have all their geometry and our assemblies have all their components, we may want to implement design changes without having to understand the models' histories and Design Intent.

The direct modeling paradigm allows you to create and modify geometry without regard to the history of the model and without fully dimensioned and parameterized features.

For Creo Parametric models, there are two ways of using Direct Modeling: the Flexible Modeling tab within Creo Parametric and the Creo Direct interface.

3.7.2.1.Flexible Modeling

The Flexible Modeling tab within Creo Parametric provides two major advantages: (1) the ability to modify part models without regard to the history or Design Intent; and (2) the ability to modify imported geometry that otherwise couldn't have been changed.

The basic workflow for Flexible Modeling is as follows:

1. Click on the **Flexible Modeling** tab.
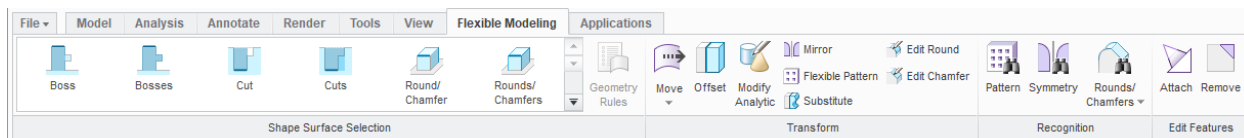
2. Select an initial surface from the model.
3. Select the additional surfaces you want to modify, using the Shape Selection tools, Geometry Rules, or right mouse button menu.
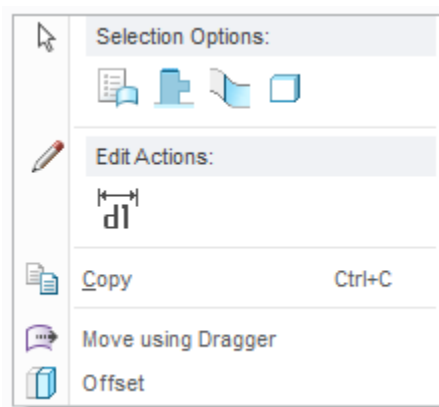

Figure 3-21. Flexible Modeling Right Mouse Button Menu

4. Apply an action, such as **Move using Dragger**, **Move by Dimension**, **Move using Constraints**, **Edit Round**, **Edit Chamfer**, **Modify Analytic**, **Offset**, and so on.

Flexible Modeling has some other powerful tools, including:

- Pattern Recognition: so you can take advantage of Reference Patterns in both part and assembly modes.
- Symmetry Recognition: so changes to an object are propagated to its mirror.
- Round and Chamfer Recognition: so that geometry is treated as such when changes are made to adjoining geometry.
- Remove: similar to the same command in Standard Mode for removing geometry from the model.
- Attach: to trim or extend quilts to solid geometry.

Operations performed in Flexible Modeling do create features, and like other features, you can use Edit and Edit Definition on them.

3.7.2.2.Creo Direct

Creo Direct is a separate interface that creates and uses the same part and assembly models as Creo Parametric.

In part mode, the 2D sketching and 3D tools appear in the same ribbon interface. It contains basic tools only for the quick creation and modification of geometry. The Model Tree contains Datums (Planes, Axes, Coordinate Systems, and Points), Quilts, and Sketches, but no features; in direct modeling, you're not creating features, you're creating and modifying geometry. Accordingly, the Edit and Edit Definition commands do not exist; you modify geometry similar to those commands in Flexible Modeling.

Another nice advantage in Creo Direct is that a Sketch can contain multiple entities, including overlapping entities. When creating an Extrude or a Revolve, you can choose which entities from the Sketch are used. In this way, the same Sketch can be used to create multiple different features. This is a much simpler workflow than in Creo Parametric.

In assembly mode, the relevant commands are Insert, Position, and Move; assembly constraints as in Creo Parametric do not exist. One of the major advantages of direct modeling, though, is that you can perform modifications, like Moves, that affect multiple components.

When those same models are opened in Creo Parametric, the features are exposed in the Model Tree. If a model was modified in Creo Direct, the View Changes dialog box opens. Changes are color coded, and the user has the option of accepting or rejecting them. In assemblies, the parametric behavior is suspended, but it can be restored.

3.8.   How Design Intent Gets Defeated

Some of the ways in which users intentionally or inadvertently defeat Design Intent when designing part models include:

Making choices based on the wrong criteria: In working with users new to Creo Parametric, I've encountered users who have the wrong goals in mind when approaching modeling. The wrong goals include:

- Speed: what is the fastest way to build a part or assembly?
- Simplicity: what methods will result in the fewest number of features or shortest Model Tree?

Getting locked into a standard process: People like standard workflows. It reduces uncertainty and gives us a better sense of confidence. But there is no standard template when it comes to part modeling. You can't say "always start off with an Extrude as the base feature" and "rounds must never appear in the top quarter of the Model Tree." There are healthy practices to follow for sure, but these are not dogma. If you want to be innovative, you should be aware of the guidelines, but know when to diverge from them.

Relying on too much non-editable geometry: Part models that start off with import features can be edited using Direct Modeling techniques. However, tools in the Flexible Modeling tab work better with more prismatic geometry. Models that start off with Data Sharing Features like Copy Geometry and Shrinkwrap that cannot be traced back to their source components (due to renaming, data loss, data ownership changes, etc.) can be difficult or even impossible to modify later.

Multiple entities in a sketch: This is probably the least recommended method in Creo Parametric in order to duplicate features. Patterns, Mirrors, and Copy & Paste Special provide much more flexible modification opportunities to accommodate future changes.

User-unfriendly models: One of the downsides of Design Intent is that you must have a good understanding of a model's Design Intent in order to make changes to it. Complex models are difficult to change when users don't do the following:

- Make simple features with a minimum number of Parent-Child Relationships, especially External References.
- Rename features and dimensions.
- Use Local Groups to organize the Model Tree.
- Use spaces and comment lines in Relations.

Saving models with fundamentally bad practices: Models should never be saved (long-term at least, or into data management systems) with any of the following issues:

- Regeneration failures.
- Suppressed objects.
- Insert Mode.
- Circular references.

Overriding dimensions: This is a particular pet peeve of mine and one I believe that should be expressly forbidden at all companies. It is possibly one of the worst practices that can be implemented in a model.

For whatever reason, sometimes people want a dimension reported on the drawing to differ from the values in the model. For similar reasons, they either cannot or will not change the model. This is a technique (which I will not explain because I detest this technique so much) to allow a user to specify a value other than that in the model.

This technique is bad because companies will manufacture the drawing, but are unaware of potential form, fit, or interference issues in the assembly. Similarly, viewables in data management systems like Windchill will be incorrect. While this may be faster or easier for the individual, it's bad for the company and the product.

3.9.   Summary

- When selecting what types of features to place in your model, consider what changes are likely to happen. This builds the flexibility in your model to adapt to change and lessen the possibility of having to start over from scratch.
- The choices that you select for the various options in a feature also build Design Intent, especially if they involve selecting other entities in your model.
- Whenever you create a new feature in your model and select existing geometry as references, you are creating a Parent-Child Relationship between the existing entities and the new feature. Changes in our models are propagated via Parent-Child Relationships. Therefore, choose the references for new features carefully.
- Use Intent References when possible. By referencing a feature as opposed to specific surfaces or edges, Intent References are more robust (less likely to fail) than explicit references.
- The order in which our features appear in the model affects what geometry is created. Use Insert Mode and Reorder (drag and drop) to change your Design Intent as necessary.
- If you want dimensions or parameters to change automatically when other dimensions or parameters change, write Relations in your model between them.
- Datum Analysis Features can perform calculations on your model and generate parameters and other datum features as results. Feasibility and Optimization Studies can then change dimensions in your model automatically to meet your design constraints. This is the ultimate in feature-based parametric modeling, and in building Design Intent at the part level.

- Tools like the Style and Freestyle features provide more artistic, design-by-feel approaches when parametric modeling for surfaces would be too tedious, time-consuming, or restricting.
- The Flexible Modeling tab provides tools for modifying models when you cannot effect the change through the model's Design Intent. This is useful for complex models and imported geometry. Direct modeling complements parametric modeling.

Thanks for reading. If you would like to read more of "Design Intent in Creo Parametric," you can find us on Amazon.com. If you like what you read, please write a review. If you didn't like it or disagree, please feel free to reach out to me.

Dave Martin

dmartin@creowindchill.com

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