Solid Edge with synchronous technology

Steering a new course in 3D design

- Learn from step-by-step examples
- Watch video tutorials
- Discover proven productivity tips

siemens.com/solidedge

Solid Edge with synchronous technology
Steering a new course in 3D design

Table of contents

Prologue

Synchronous technology was born out of the idea that merging the best ideas of direct edit techniques with the best ideas of history-based modeling would deliver unprecedented power and control in editing CAD geometry.

For decades, history-based modeling has dominated the CAD world for reasons we will discuss more later. History-based methods have a lot of power, such as being dimension driven, highly automated, and feature-based, but come with a lot of less desirable baggage such as the need for pre-planning, inflexibility and the fact that it slows down as you add many features or many parts. Meanwhile, direct edit methods have also existed for a long time and have several advantages, but because of its associated weaknesses, it could never compete with history-based modeling.

Enter synchronous technology, which marries the best of both worlds, and brings some new strengths to the table that make it all work together seamlessly. In the chapters that follow, we'll guide you through a step-by-step approach to synchronous technology and share some compelling examples that will help to demystify what synchronous technology is, and how it can revolutionize the way you work.

Visit the companion resource file to obtain parts for the tutorials contained in the following chapters.

Chapter 1

Introduction to synchronous technology

Introduction

Welcome to synchronous technology for the history-based user. In these pages we hope to help history-based users of all stripes understand synchronous technology better. Whether you're an experienced Solid Edge® software ordered user or are coming from SolidWorks® software or Inventor® software, we want to help you understand this revolution in CAD. This is not necessarily a book for CAD beginners. We expect that you will already have an understanding of history-based CAD programs such as SolidWorks, Autodesk, Inventor, PTC Creo® software, or even ordered (history-based) functionality in Solid Edge.

It doesn't do synchronous technology justice to call it simply "direct edit", and the popular notion that it is not parametric is altogether incorrect. Part of the goal of this book is to first properly define synchronous technology, and then to dispel misconceptions through demonstrations of its capabilities, and to help existing CAD users see how application of synchronous methods will help you save time and energy in your everyday CAD and engineering tasks.

For decades, history-based modeling has dominated the CAD world for reasons we will discuss more later. History-based methods have a lot of power, such as being dimension driven, highly automated, and feature-based, but came with a lot of less desireable baggage such as the need for pre-planning, inflexibility and the fact that it slows down as you add many features or many parts.
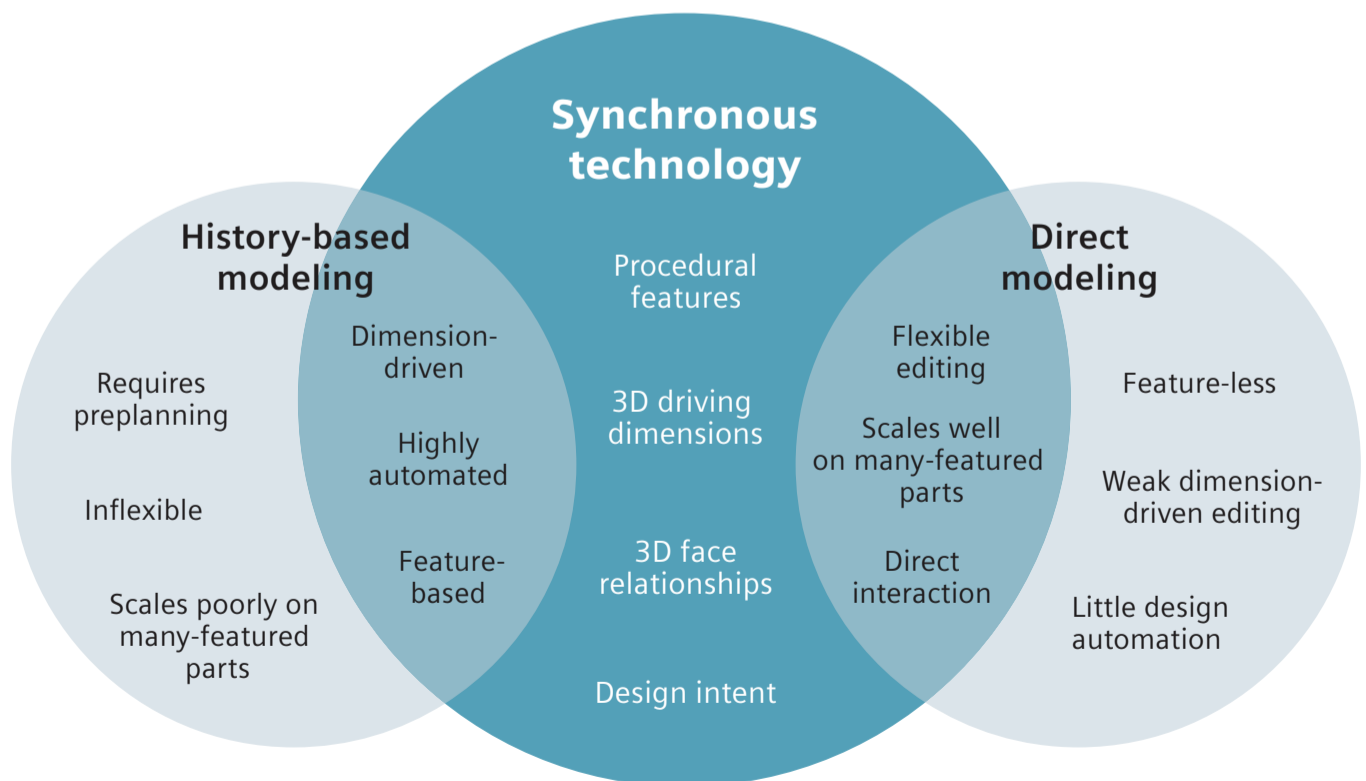
Synchronous technology is the combination of the best functions of history-based CAD and the best functions of Direct Modeling, along with some special exclusive functionality.

Meanwhile, direct edit methods have also existed for a long time and have several advantages, but because of its associated weaknesses, it could never compete with history-based modeling.

Enter synchronous technology that borrows some of the best ideas from history-based and Direct-edit methods, while leaving behind the weaker areas. Plus, synchronous technology brings some new strengths to the table that make it all work together seamlessly.

Why synchronous?

Let's start by understanding the name, synchronous technology. What does it mean? "Synchronous" means that several things are happening together, at the same time, in a coordinated way. This is contrasted with the history-based concept of doing things sequentially.
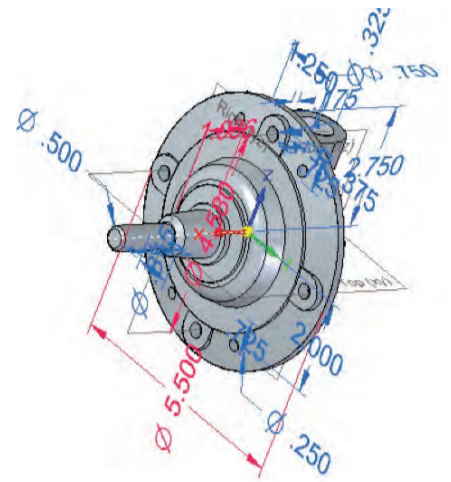
Synchronous technology adds several other technologies on top of the direct edit method such as feature recognition, intent recognition, driving dimensions (PMI), Face Relations, and procedural features.

In synchronous modeling, we make use of PMI dimensions directly attached to the model.

In synchronous technology, the entire model is solved at once. By contrast, in history-based CAD, the model is solved one feature at a time, and each successive feature is added to the body. So in a sense, solving the whole model at once rather than sequentially in parts can also be thought of as "synchronous".
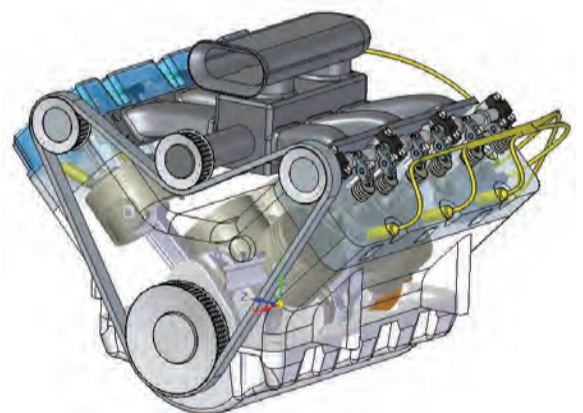
Another level of synchronous behavior in synchronous technology is that Solid Edge enables you to combine both synchronous and history in parts and assemblies. So you don't have to use only one method at a time, although you can if you want. Parts made in Solid Edge can use either synchronous, ordered or both. Likewise, assemblies can have parts in them that are either synchronous, ordered or both.

Let's list the things are happening at the same time in synchronous technology:

- Synchronous solves the model geometry at the same time, instead of feature-by-feature
- Synchronous employs several technologies that work at the same time:
  - Direct edit
  - PMI dimensions (driving dimensions on the solid body)
  - Face relations (work like sketch relations on the 3D model)
  - Procedural features apply parameters to features like patterns, thin walls, ribs, holes, and others
  - Feature recognition (can convert patterns, holes and other features from imported bodies into procedural features)
- Synchronous and ordered methods can both be used on the same models, in the same assemblies

Synchronous assemblies enable us to use both synchronous and ordered parts in the same assembly.

Synchronous technology is like…

History-based users sometimes have a hard time getting their heads around synchronous technology when they see it for the first time. When asked how to describe the software in words, we often resort to analogies. Synchronous technology is to 3D as 2D-only CAD is to 2D line drawings. You may have worked with 2D-only CAD before. To move lines in 2D, you select a set of items and use the appropriate tool to move the selection set. You can move an approximate

amount by just dragging, or a specific amount by keying in a number or snapping to points. This analogy may over-simplify some aspects of the software, but it does help illustrate the select-and-move aspect of editing with synchronous technology.

If you have preconceptions of what you think synchronous technology is, you'll do best to put them aside until you're done reading this book. We think that when you understand what synchronous technology is, what its strengths and weaknesses are, and how it stacks up against the strengths and weaknesses of history-based modeling, you'll come to agree that there should be a place for it in your design process.
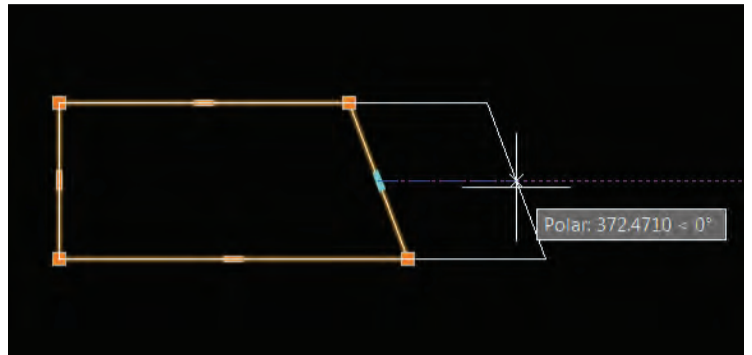
Editing with synchronous technology is a lot like editing in 2D-only CAD: select and move.

You may have become numb to the pain of history

The shortcomings of history-based methods are often overlooked because history-based modeling has been considered the only seriously viable method for decades. But synchronous technology has strengths where history-based modeling has weaknesses, so these methods can and should be used together to make the most robust models possible.

We are not proposing that you should entirely abandon history-based modeling and only model synchronously. We generally like the idea of using synchronous and history for their respective strengths, using them together as appropriate. This allows you to avoid the weaknesses of both history-based and synchronous. Combining modes greatly simplifies the parent/child tangle that can result from inter-associativity between straight history-based parts, especially when designing in-context parts in assemblies. Using these technologies together is really where the biggest value lies.


Many of us have used history-based CAD for so long, we have forgotten how many areas of it are based on difficult concepts to learn.

Anyone who has used 3D CAD in the last couple of decades has dealt with the workarounds and frustrations of history-based modeling. For example, editing features that you made early on in the process causes features that you made at the end to disappear as they are rolled back and then recompute when the edit is complete. This is made necessary by the linear nature of the history tree, but from a design point of view makes no sense, and simply wastes time. We have all used history-based CAD for so long we just accept its shortcomings without question. So in this book, we aim to show that synchronous has an alternate method that enables you to avoid those frustrations. We will then show you techniques where synchronous works best, and some clever ways of using it. Beyond that, we will demonstrate how and when to combine synchronous and history-based methods.

Not everyone starts a believer

In doing the research for this book, we have taken time to examine some of the fears and misconceptions that have prevented people from moving forward with synchronous technology. There are some limited specific situations where synchronous really doesn't fit into certain design workflows, and in those situations we recommend ordered.
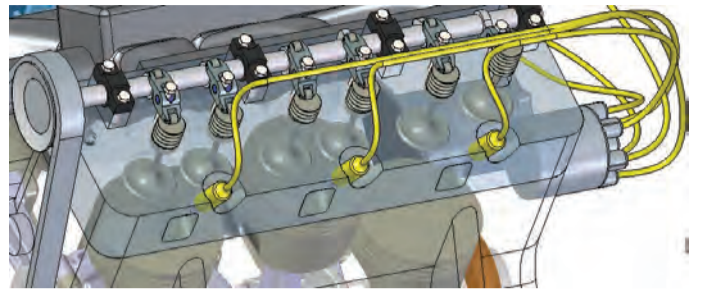
Synchronous technology enables the user to create design intent without very resource intensive techniques such as associative links between parts, but when it's really necessary, we can still create those associative links.

As a product of this research, we also have found that people tend to cling to what they know. You owe it to yourself and those working with you to be open to potential improvements to your process. Even if you start out reading this book as a skeptic, that's ok. Just promise yourself that you will be open to new ideas, and that you will recognize the value in possible new ways of doing things.

As another product of the research, we kept seeing examples where the history paradigm was illogical, or was susceptible to procedural problems that more closely resemble programming bugs than mechanical design problems. The deeper you get into studying history-based modeling, the less you will be convinced that it is the best method for most modeling situations.

Combining synchronous and ordered techniques within a single part is one of the strengths of Solid Edge with synchronous technology.

Weaknesses of history-based modeling

History-based models rely on a recipe of features, executed in a specific order. Synchronous technology, in contrast, is only concerned with the present state of the geometry, without regard for how you might have arrived at that point. Through the evolution of CAD, history-based modeling has been the most prevalent for the last 30+ years. Pro\ENGINEER® software was the product to develop and popularize this method. Synchronous technology was introduced in Solid Edge in 2008, and is gaining traction in the industry.

The necessity for introducing the time/order metaphor for CAD models originally had to do with the computer hardware that was available in the 1980s. It wasn't very fast, so it couldn't handle the typical complexity of a manufactured part all at once, much less the complexity of multiple parts going into an assembly. Thus, CAD developers needed to break up the geometrical information into smaller computable chunks.

This meant that not only did product data have to be broken down into assemblies and sub-assemblies, but the assembly data had to be broken into parts, and the parts into features, and the features into sketches and metadata parameters. Feeding the computer large amounts of information in small chunks was already a familiar method to the people who were charged with solving this problem – programmers. Programmers write computer programs using essentially the same method – by creating a

Working with imported data in synchronous technology makes things possible that simply can't be done with imported data in history-based software.

sequential set of instructions that can be strung together one by one to make a finished product of any size. So to employ yet another analogy, history-based CAD modeling developed into a method that requires the mechanical designer to work (and think) like a programmer.

For decades this abstraction of thinking of our CAD models in a time-based metaphor has seemed acceptable to mechanical designers who work in 3D, even though the metaphor breaks down when we start working in assemblies, and doesn't make much sense when we look at our models in terms of feature importance rather than order in time. To some users, approaching CAD modeling without this artificial metaphor now seems unnatural.

Embracing a new way

It is common to hear history-based users who are contemplating the switch to synchronous technology complain that they feel they will lose some sort of control because of the perception that there is no way to fully constrain the model in synchronous technology. The fully constrained mindset is one that comes from the history-based way of thinking. In synchronous technology, the idea of fully constraining a model works backwards from the history-based method. The part geometry will not change unless one of the four tools tells it to:

1. PMI 3D Dimensions
2. Steering Wheel editing interface
3. Face Relations
4. Design Intent
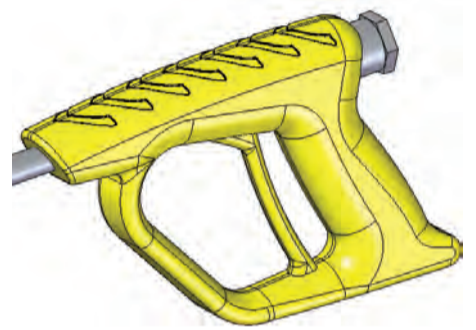
In history-based modeling, the model – either in its entirety or a portion thereof – is recalculated with each change, so the entire model is constantly in danger of changing. With synchronous, there is no reason to assume the model will change, unless you actively change it with one of the tools above.

The most difficult part of history-based users making the switch to synchronous technology is letting go of the programming metaphor, and learning to just be concerned with how the model looks right now.

The zen of synchronous

I used to teach history-based CAD for many years. Every now and then I would come across someone who would completely ignore the feature tree, and just keep building, editing, and cutting away portions of the model. They would wind up with hundreds of features, many of which simply chopped away features created earlier in the tree. This was of course completely the wrong way to handle history-based models, but its how things work in synchronous. So many of us have been ingrained with the idea of "right" and "wrong" ways to model in history. It's very hard to throw away that training, but that's exactly what you have to do to understand synchronous technology. I will wind up saying this many times

Point to the History on this part.

before this book is over – the only thing that matters in synchronous technology is the state of the geometry right now. It doesn't matter how you got there, all that matters is the current state.
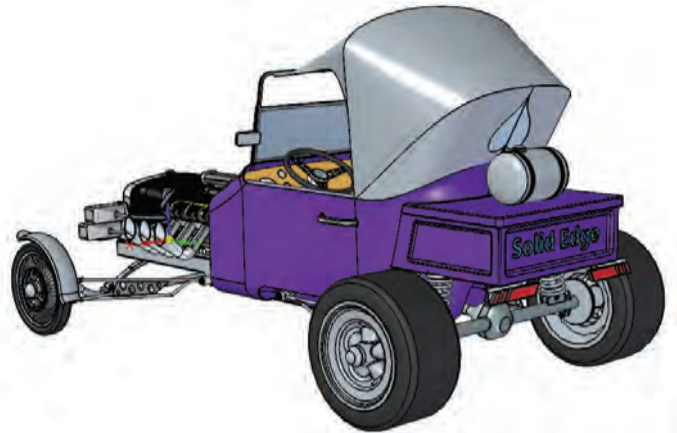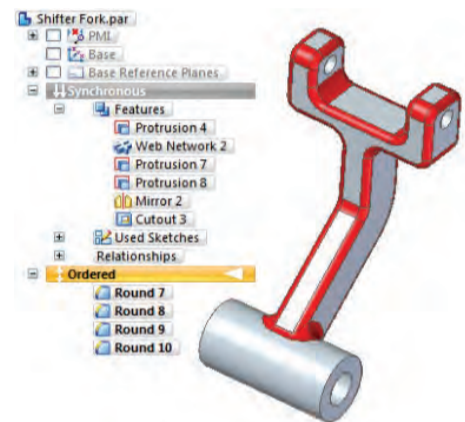
History-based complaints

One step in confronting the fear of change is to outline the problems that we have grown to overlook with the existing history-based paradigm. This is my favorite part, because many people have completely forgotten that they go to great lengths to work around these problems to make history work. Let's examine the difficulties associated with history-based modeling:

1. Design is not linear. There is nothing about design that leads us to think of features in some sort of order, especially not as an executable list of features.

2. Change is inevitable in CAD models. We want our design intent to handle the changes gracefully. But sometimes the design intent itself is what changes. In history-based CAD, changing the design intent itself is similar to rerouting a big expressway through a congested city. It's a complex, frustrating, and ineffective undertaking.

The ability to mix modes in parts and assemblies makes Solid Edge stand out as the most flexible CAD tool around.

3. The concept of rolling back the tree to insert a feature is entirely artificial. You should be able to just add a feature without worrying about order. If a feature exists, you should be able to just edit it without affecting the other features in the model. Daisy-chained parent/child dependencies lead to many unintended relationships in parts with long feature trees. These daisy-chained relationships often lead to daisy-chained failures.

4. History-based modeling looks more like computer programming than anything else. That's because it's a method created by programmers to overcome the limitations of hardware thirty years ago. The concept of history-based modeling does not come from design. It comes from the intersection of hardware computational limitations and programming paradigms.

Synchronous technology as implemented in Solid Edge works best with prismatic parts.

5. Even in history-based software, sketches, assemblies, and drawings don't use the concept of history. If you can work without it in those places, you can work without it in parts as well. In fact, some people compare the way synchronous technology handles 3D geometry to the way most CAD programs handle 2D geometry – select and move.

6. Failures in the feature tree have to be repaired before you can proceed. How much time do you waste on this? In my consulting business, I spent probably 60 percent of my time repairing things after changes because the change I had to make was not foreseen at the time I created the model. Design intent in history-based software requires you to be able to see the future to build design intent into the model, and requires you to see into the past to recall how the design intent was meant to work.

7. The parent/child relationship becomes extremely convoluted on big complex parts, and is even more difficult to unravel when parts become inter-related in the context of the assembly. When you select a base feature to start your part, there are often multiple ways of getting started. In many cases, the feature you pick is completely arbitrary, or doesn't involve factors which become
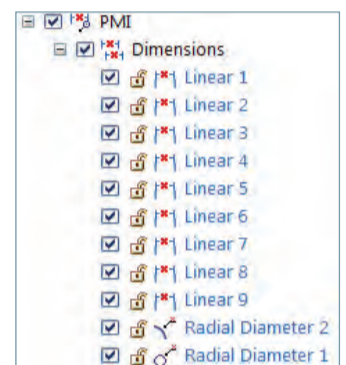
important later on. A week later, however, when you're making edits, you might wish you made a different choice. This kind of foresight/hindsight experience that you encounter almost inevitably in history-based models, does not exist in synchronous technology.

8. Rebuilding and possibly repairing all the features in the tree repeatedly is inefficient and slow. And every time the software does it, it's another chance to do it incorrectly. The only thing that really should matter with your CAD model is the final state. History-based software, however, acts like your model is really itself software that has to be constantly recalculated, and so is constantly in danger of returning errors or unintended changes.

9. Best practice for history-based modelers is built around the limitations of history-based modeling. In history-based modeling, you can have the correct geometry, but your CAD Admin might still declare your model a disaster. In synchronous technology, if the geometry is right, it's right. So has all that effort that has gone into best practices really been necessary?

History-based modeling and the concept phase

History-based modeling has particular weaknesses when it comes to modeling concepts rather than finished models. Many users resort to making what amounts to a practice model during the concept stage. Concept work where the direction of the design intent can and does change very rapidly is inherently incompatible with requirements of history-based modeling, where your idea needs to be fully formed before you start creating sketches. When the design is finalized, these users go back and re-create a more robust well-thought out model where the design intent actually works as intended, essentially doubling their work.

Solid Edge captures much of the model data right in the PathFinder, which makes it easy to find, edit and control.

Synchronous models, on the other hand, only care about the present state of the geometry. Design intent in synchronous is captured in the geometry, PM and applied relations. If faces in the model are parallel, the software recognizes that and maintains that relation until you tell it not to. So as long as the geometry is correct, your concept model is as good as a production model. No more practice models.

Synchronous is: PMI dimensions

Product Manufacturing Information or PMI refers to the dimensions that are placed directly on the 3D model to drive size, orientation, and position. Applying dimensions directly to the 3D geometry rather than indirectly through sketches and defining dialogs gives you a clear and unambiguous way to document and control parts.

Dimensions can be located and turned on/off easily through the PathFinder, or show only the PMI attached to a specific feature. PMI can also be grouped in user-specified groupings.

To be most successful when starting to learn synchronous technology, PMI dimensions should be your first stop when making changes to engineering models. The Steering Wheel (discussed later in this chapter) might seem more attractive and interactive at first, but for stability, clarity, and precision, beginning synchronous users should start with PMI dimensions. They are simpler to control and more direct than other available methods.
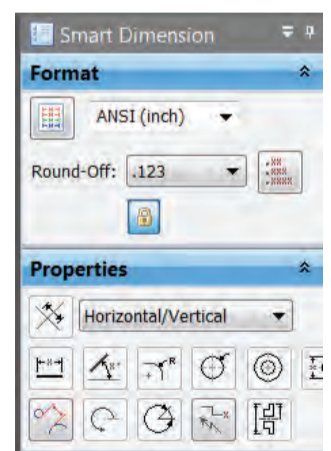
Dimensions connect to faces

Dimensions are created by selecting edges, but they drive faces. Dimensions do not drive edges. Edges always exist at the intersection of two faces. So even though you create dimensions by selecting edges, dimensions always specify a distance between faces.

The dimensions in these examples have been created using the Smart Dimension tool. Smart Dimension can be used to create most but not all the dimensions you will need to place on models in Solid Edge. You need to pay attention to the available options in the Smart Dimension CommandBar, shown here, and the other dimension tools available from the Ribbon.
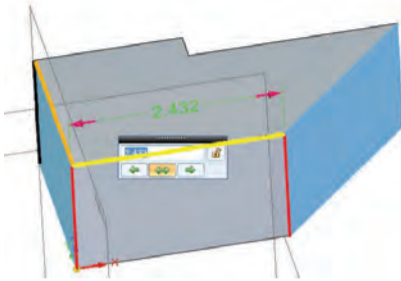
Here we have a simple block. The 2.432 dimension could have been created by selecting either just the edge highlighted in yellow, or both edges highlighted in red. In either case, what the dimension does is specify the distance between the two blue faces at the vertical gray face.
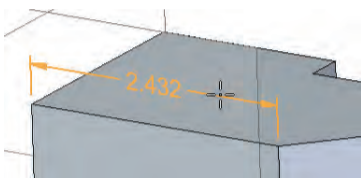
If the dimension had been created between the left red edge and the black edge (or just the orange edge), the dimension would have been different, and the faces it controlled would have been different. The software is smart enough to know which faces you mean to control even though there is the possibility of two different faces for each selected edge.
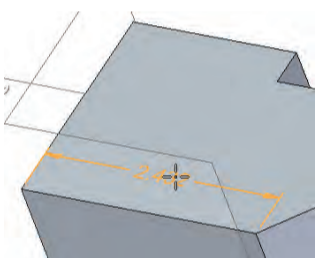
When you edit the dimension, you can make the left side move, the right side move, or move both sides symmetrically based either on the dimension settings or where you select the dimension for editing. This is a really novel idea in driving design intent, and is not available with history-based models. Notice that the face highlighting in the previous image reflects the change direction setting (the face that will move is highlighted). This concept helps you determine the design intent on the fly and will be discussed in more depth in the next chapter.

Notice another thing that isn't necessarily relevant to synchronous technology or the actual function of the dimension. The placement of the dimension changes from parallel to the ZX plane to parallel to the XY plane if you rotate the model while placing the dimension. Whichever plane normal is closest to parallel to your line of sight is the plane that the dimension will be placed on. Once the dimension is placed, it will remain in that orientation.

If you dimension between two edges that are not co-planar, you can rotate the view and get a number of different results. For beginners especially, it's recommended that you keep your dimensioning choices clearer, which will make it easier to control and understand the results the software gives you. Once you gain some solid experience with the system, some of these more complex options will give you additional control, but for now, keeping it simpler will be less confusing.

Synchronous technology and curved faces

Synchronous technology deals with curved faces somewhat differently from planar faces. When you select a cylindrical face, the software displays a radial or diameter dimension that you can use to change the size. This happens whether you have created a dimension on the cylindrical face or not. You may also notice that the Steering Wheel is not displayed to change the size of a cylindrical face, although it can be used to change the position or orientation of a cylinder.

When you place a dimension on a non-planar edge, Solid Edge projects that edge onto the nearest base plane. So if you have an edge that is the intersection of two curved surfaces,

dimensioning the edge will yield a read-only Length dimension (shown here as L 1.580in purple) because the software sees the edge as a complex spline.

If you want to change the radius of a curved face, you need to select the curved face itself. The image to the right shows a part with two curved faces that intersect. The circles used to make the cuts are also shown. The circles have diameter dimensions, and the faces have radial dimensions. The dimensions for the curved faces are assigned by the software if the user doesn't directly specify them. Selecting the curved face brings up the radial dimension.
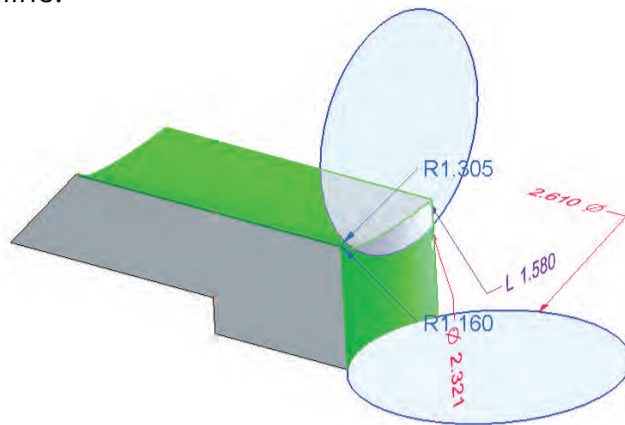
Dimensions are attached to faces, even though they may be created by selecting edges. Working with PMI dimensions attached to the 3D model is a very intuitive way to work, although it may take the history-based user some time to make the transition.

This just helps to demonstrate the relationship between dimensions, edges, and faces. It is relevant to synchronous technology in that you have to understand how dimensions connect to the 3D geometry rather than just working with 2D sketches. The model used to create the previous graphic is provided with the reference materials for this chapter if you want to examine it more closely, called Curved Faces.par.

Live Sections are a little different

In some situations, it makes sense to attach dimensions to Live Sections rather than to faces. The Live Section is essentially a sketch created at the intersection of the model faces and a section plane. Solid Edge automatically creates live sections for revolved features, and the Live Section is placed in the PathFinder where you can turn it on or off. This Live Section can be used to drive the model, unlike other sketches. Live Sections can be used anywhere on any model, but are most effective on revolved features where controlling the profile just from model edges can be difficult.
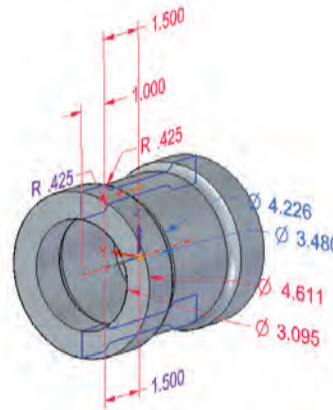
Live Sections help you apply simplified dimensions on revolved parts.

Synchronous is: Steering Wheel, Design Intent, Solution Manager

The Steering Wheel enables you to translate or rotate the selection set along axes or within a plane by using the tool plane. The blue dots help you locate and orient the Steering Wheel, and the arrows help you move along an axis. The tool plane enables you to move the selection within the indicated plane. The ring enables you to rotate the selection. Each of these controls enables you to key in a specific value for the translation or rotation after you have established a direction.

Axis knob
Axis arrow
Cardinal point
Torus
Origin knob
Tool plane
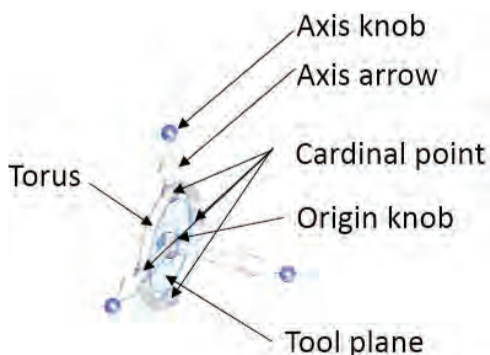
You need to become familiar with the parts of the Steering Wheel and how to use it, as it is one of the primary interface tools synchronous users use to edit parts and assemblies.

Editing done with the Steering Wheel is applied to the selection set – the set of faces you have selected at the time when you invoke the Steering Wheel. In addition to the faces you have selected, synchronous technology uses your Design Intent settings to select additional faces. We will discuss selection sets and Design Intent later in this chapter.

Generally, if the model has faces that are parallel or perpendicular, they aren't that way by accident. You probably intend for them to stay that way. Thus, those relationships are part of the design intent. Synchronous technology can

identify relationships like this quickly and easily, and so it helps you maintain these relationships by adding any faces that meet the criteria of any active option in the Design Intent settings to the selection set. So if you have the Concentric option turned on, any faces that started out concentric will still be concentric at the end of the edit. This is a very different way of looking at design intent, but the big advantage is that you can assign or disable Design Intent at the time of edit. You no longer have to establish it before you know how you are going to edit the model.

Note: because Solid Edge has a tool called Design Intent and a general concept called design intent, this book will refer to the tool with capitals and the concept with lower case.

Also note that Design Intent was previously called Live Rules, so you may see that name used to describe this functionality in older references.
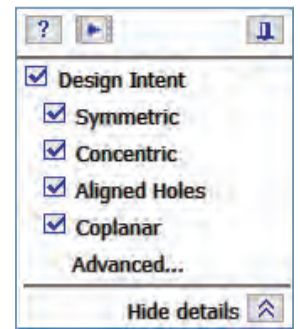
The Design Intent panel enables you to control relationship assumptions made by the software.

Manipulating the Steering Wheel

You can use this section as a tutorial, by following along and just getting the feel for how the Steering Wheel works, and how to control it. You can use the part called Practice With Steering Wheel.par to practice on shapes similar to those in the following figures.

The ability to manipulate the Steering Wheel is key to being able to make the edits you want to make.

The first time you click on a planar face, a single arrow appears. This is the simplified Steering Wheel, and only allows movement in the direction of the arrow (normal to the selected face).

To get the full Steering Wheel to appear, either click and release twice on the arrow, Ctrl-select multiple faces, or use the origin knob (blue dot) to move the Steering Wheel. This enables you to move the select set in the direction of any arrow, rotate about the axis of the torus, or along the plane of the tool plane.

If you click on a cylindrical face, the Steering Wheel appears with only two axes, (2D Steering Wheel) at the end of the cylinder to allow you to move the cylinder. To change the length of the cylinder, click the origin knob, or Ctrl-select the end face of the cylinder and use the normal arrow as for the planar face.

There is not a way to use the Steering Wheel to change diameter or radius values, you must use the PMI dimensions that the software places on cylindrical faces automatically.

If you want to move a selection of faces along an orthogonal direction (parallel to one of the three main axes of the part), then it's easy. You have three axes on the Steering Wheel, so you just move along one of those axes. But when you want to change the angle of a selection of faces, you have to use the torus, and the Steering Wheel origin has to be on the vertex or axis of the angle. The Steering Wheel only has one torus, so you've got a 1 out of 3 chance of the torus being in the right orientation by default.

To reorient the Steering Wheel's torus to be parallel to one of the XYZ directions, Shift-click on the tool plane on the Steering Wheel. There are only three options, so if clicking once doesn't give you the result you need, just click once more. The Steering Wheel will toggle through the three possible orientations.

To orient the Steering Wheel along a direction other than the XYZ directions, drag the origin knob to an edge or face that is either parallel or perpendicular to the desired direction.
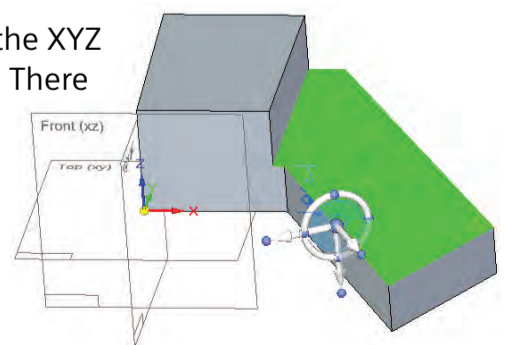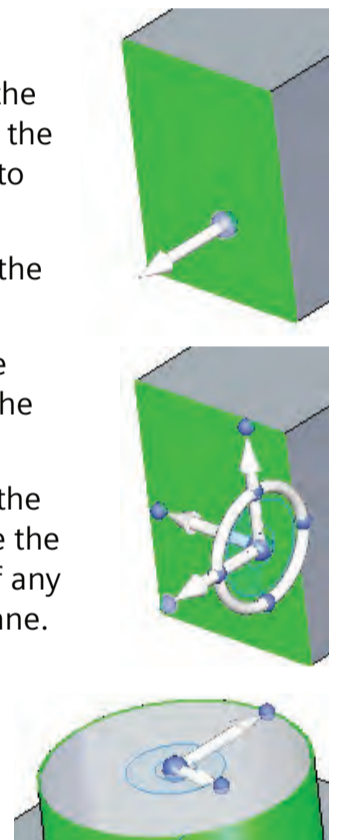
If there is no geometry in the current assembly or part that fits the purpose, Ctrl- click on an axis bearing knob and drag or key in an angle of rotation. If hovering the cursor over the axis knob highlights the wrong axis for rotation, then use Shift instead of Ctrl.

You can also use the surrounding geometry to pick up references to get exact distances or angles without round-off error involved in measuring or keying in values. For example, if you want to copy a set of faces by the length of an existing edge, you can select the faces you need to move/copy, and offset the Steering Wheel by dragging the origin knob to a keypoint (say one end of an edge), then Ctrl-click on an axis and to place the faces, click on another keypoint (say, the other endpoint of the edge).

When you are using the Steering Wheel to move items in a From-To sort of arrangement, you can think of the Origin Knob as being your From reference, and whatever keypoint you select will be the To reference.

Synchronous is: Face selections

Although there is a chapter in this book exclusively devoted to creating selections in synchronous technology, we will go over some of the highlights here.

One of the things that distinguishes synchronous technology from history-based modeling is that changes occur to selection sets rather than to features. In synchronous, features in the PathFinder really are just selections of faces. So in order to make a change such as for example angling a set of faces, you first select the faces, then use the Steering Wheel to make the change.

Rather than selecting a "feature" to edit, in synchronous technology, you select a set of faces to edit. This gives you far more flexibility than being tied to what your CAD tool identifies as a feature.

Selections can be made using common techniques such as:

- Ctrl-selection
- Window/box/fence selection
- Selecting features in the PathFinder
- Selecting a user-defined set of faces
- Using the Selection Manager for automated or intelligent selections

You can also manually create user defined sets if the features listed in the PathFinder don't match up with the edits you want to make.

Selections can also work with dimensional changes. If you have a set of faces selected, and then click on a dimension to make a dimensional change, the selected faces will also move with the faces that move with the dimension.

Selections can be made by individually clicking on faces, using Ctrl-click to select multiple faces, or by fence/window selecting multiple faces at once. If you use Ctrl-click, the cursor changes between 3 states, −, + and +/− . You can toggle these states by pressing the Spacebar.

Design Intent is actually a method for automatically selecting faces. If you are using the Parallel design intent, the software automatically selects any faces that are parallel to your existing selection for use in the edit you are about to perform with the Steering Wheel.

In any case, with this book, we hope to leave you with a correct idea of what synchronous technology is, what it does, and what it does not do. We hope to help you see that the best way to use it is to combine its strengths with the strengths of history-based modeling. Our goal is to help you spend less time wrestling with the artificial process of CAD, remove the frustration, and put time back in your day for things you value most.

Confronting the buzzwords

Converting to a new way of working with CAD data is difficult enough without all of these buzz-words confusing things. Here we define some of the more common words and phrases that tend to confuse new users.

History-based
History-based CAD software relies on the user to build an individual part with a series of fea-tures. The software remembers the features in order, and the model must re-solve these features in order perfectly at every step to move forward. You can think of this method of model building like a computer program. You give the computer instructions for every step, and it executes those steps for every time the model must be rebuilt.

Interestingly, only the part file follows this way of working. The sketch, assembly and drawing do not. Things begin to get complicated when you make references between parts in the assembly, because the references are still bound by history, while the top-level assembly document is not.

History-free
History-free CAD still uses the idea of features, but once the feature is part of the model, the order of the features is irrele-vant. For example, in the PathFinder shown to the right, the features are shown in the order in which they were created, but can also be sorted alphabetically or by type. Features can be thought of as a collection of faces, but can also contain certain types of metadata, such as size for a round or cham-fer, or diameter for a hole, or possibly number of features for a pattern. Synchronous technology uses both history-free and history-based methods. As an overgeneralization, history-free model-ing is sometimes equated with direct edit.
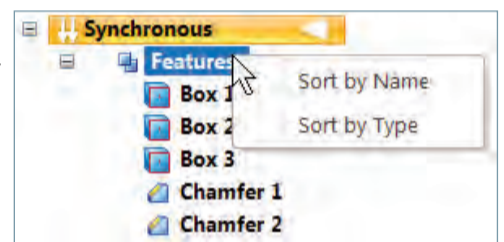
Direct edit
There seems to be no end of experts arguing over finely nuanced definitions and variations of this term, but in general, direct edit is a CAD method where you directly manipulate faces of the model, rather than indirectly edit feature definitions or sketches. The reason edit is part of the name is because how the geometry is made really doesn't matter, and frankly, isn't as interesting as how the geometry is changed. All designers know that you draw something once, but edit it a dozen times or more. Synchronous technology augments the basic direct edit idea with several tools such as PMI, Design Intent, Face Relations, the Steering Wheel, Procedural Features, and others.

Feature-based
Both history-based CAD and synchronous technology are feature-based, although the features work differently for each. Each operation you perform is equivalent to one feature. During the creation of new geometry, features work roughly the same. From a sketch or sketches, you make an extruded feature, or a sweep, or add a round, or chamfer, or pattern. The difference comes when editing. In history-based software, you edit with the same interface that you used to cre-ate, and the software remembers values. In synchronous technology, the features are generally selections of faces. Synchronous technology also has a type of feature called a Procedural Feature, where it will remember some values that go with the collection of faces. Examples of procedural features are rounds, chamfers, patterns, holes and thin wall. Editing these features does not take you back to the original interface or suppress other dependent features, it just selects the faces of the feature, and allows you to change a value such as wall thickness, or hole diameter. Deleting a synchronous feature will delete all the faces of the feature. To get a little ahead of ourselves, synchronous technology uses the set of currently selected faces to edit. This is why a feature remembers the faces used to create it. But this also allows you to change the selection set, and thereby change the design intent. It's almost too easy. The hardest part is for-getting the old history-based definition of a feature.

Parametric

People frequently have the mistaken idea that only history-based software is parametric. Synchronous technology is also parametric. Parametric means that the model is driven by parameters, which can include dimensions, pattern instances, wall thicknesses, hole diameters and depths, and so on. You can drive synchronous parts with a table of values that in turn drive PMI dimensions on the part.

Associative

Associativity in CAD means primarily links between documents. In some cases this is good, such as between a model and a drawing. But in other cases, it is considered onerous overhead, for example when you have links between parts in an assembly. Synchronous technology has tried to lighten the CAD compute and file management burden by limiting links between part files, while still allowing you to make in-context changes.

In history-based CAD, it's possibly too easy to create these links between parts. Synchronous can go either way, with explicit associativity, or with the less permanent associative intent by selection, which you might think of as manual associativity. Synchronous technology gives us functions like Create Inter-part Relationships, Inter-part Copy, and the use of externally linked variable table to drive dimensions. This allows you to work freely, without creating the tangled associative spaghetti that you tend to get with history, but when you need an associative link, the tools are there.

Chapter 1 | Introduction to synchronous technology

Summary

Synchronous technology is a combination of a lot of powerful tools. Some of these tools will be familiar such as dimensions and equations, and some will be new to history-based users, such as the Steering Wheel, and Design Intent on the fly. It is simultaneously an improvement on and a complement to history-based modeling. In many ways, synchronous technology is a completely new way of approaching 3D CAD modeling that can revolutionize the way you work.

Chapter 2

Driving design intent without history

In this chapter we build on the concepts from Chapter 1 and start looking at practical aspects of how the details of synchronous technology actually work.

What is design intent?

Design intent is the set of properties that allow a CAD model to react predictably to change.

This is the same definition for design intent used for history-based CAD software, but it also applies to synchronous technology. The design intent does not imply anything about feature order, or whether features are driven by sketches, because you don't need those things to create design intent. Relationships and dimensions create design intent, and synchronous technology uses both of those control types.

In synchronous technology, the geometry itself captures the design intent. Synchronous technology recognizes existing relationships in the model and maintains those, without requiring duplicated instructions in sketch relations. For example, in a simple rectangular block, three pairs of model faces are parallel, and synchronous technology will recognize this situation without the user needing to specify that the faces are parallel. You can, however, tell the software to stop recognizing the parallel relation when you need to, or manually add a relation that doesn't already exist.

For example, concentric holes are probably not concentric by accident. The software recognizes the concentric condition, and maintains it. This is how synchronous technology is able to apply design intent to imported models with no sketches or "features". The intelligence comes from the geometry and the software just has to read or recognize it. This is why data transfers involving history-based CAD generally lose all of their intelligence – because it is part of the proprietary file format rather than simply captured in the existing geometry, as in synchronous.

Driving geometry size and position synchronously

In synchronous technology, there are four tools that we identified in the previous chapter that you can use to drive geometry size or position: PMI (dimensions), Design Intent (assumed relations), Face Relations (manual relations), and the Steering Wheel. These methods work together with the selection set to make changes.

1. **PMI** (product manufacturing information) refers to driving dimensions that are directly attached to the 3D model and drive the locations or sizes of faces. PMI dimensions are placed using the tools on the Dimension group on the Home tab.

Solid Edge offers a complete set of tools for applying dimensions to models.

Dimensions in Solid Edge® software have several colors indicating function or state:

a. **Red** = Locked. This means that you have to actively edit the dimension to change its value (the system cannot drive or change the value).

b. **Blue** = Unlocked. This means that you can change the dimension manually or the system can change it through other changes in the model. You can use the Lock toggle on the dimension box to toggle dimensions between locked and unlocked states.

c. **Purple** = Read Only or For Reference. Can be a number of things, including Not To Scale or driven by table.

d. **Orange** = Highlighted

e. **Green** = Selected

f. **Brown** = Dangling, or detached dimensions that have lost one of their references
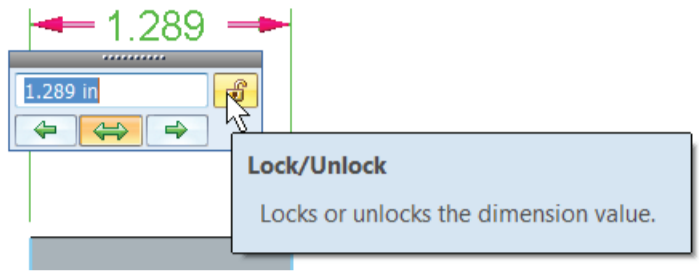
Note that the following only applies to PMI dimensions on the model, not to dimensions within sketches.

Within the dimensions are directional arrows (red 3D cylindrical arrows) that indicate which side of the dimension will move when you make a change: left, right, or symmetric. For example, if you did not yet set the direction of the dimension, and you click on the left side of the dimension to edit it, the left side moves. In the previous figure, notice the position of the cursor relative to the numerals and the resulting red arrows on the dimension.

This method of changing the direction of the dimension represents a major departure from how dimensions work in other CAD software. The dimension tools on their own add greatly to the design intent flexibility that synchronous technology gives you over other methods.

You can permanently set the direction of the dimension in the dimension dialog box, shown above. Once you do this, the side of the numeral that you click on no longer controls the direction of change.
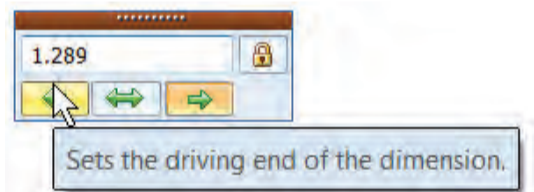
Changing the design intent on the fly with the dimension directional arrows is something you will not find in history-based software.

2. **Design Intent** (formerly called Live Rules) refers to the interface Solid Edge uses to display and enable the user to control the relations captured from the model geometry.

Design Intent essentially adds to your selection set anything that satisfies the enabled options. So if you select a cylindrical face to modify, the default settings will add any faces that are concentric or tangent to the selected face.

Faces selected by Design Intent apply to Steering Wheel edits as well as PMI edits. So the Steering Wheel will move the entire selection set which includes whatever you select manually plus
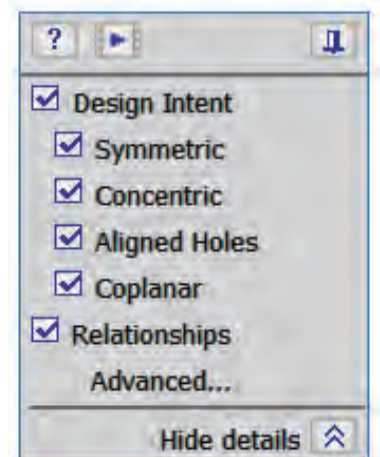
whatever Design Intent selects for you based on the settings. PMI likewise will move the entire selection set plus Design Intent faces for any dimensional changes.

The idea for the two different panels (the gray one above and the colored tiles below) is that for most of your edits, you will not make any changes to the Design Intent settings, or the changes that you make will be to the simple, commonly used settings shown in the gray Design Intent panel. However, there will be those times when you need more detailed control, and for those times, you can access the Advanced Design Intent panel, and get as detailed as you want, including the Solution Manager.

a. The Design Intent panel enables you to turn off the most basic assumed relations. To display more advanced relations, you can display the Advanced Design Intent panel, shown below.

Orange means the option is active, white means it is inactive.

b. The Advanced Design Intent panel enables you to turn off assumed relations, such as keep orthogonal to base planes, coplanar axes and consider sketch planes.

The symbols in the Advanced Design Intent panel are identified using tool tips, but here is a detailed list with the keyboard shortcuts for each listed in parentheses.

	Design Intent Restore (R) – Restore default settings		Consider Reference Planes
	Design Intent Save Relations (F) – Creates persistent relations for Design Intent settings		Consider Sketch Planes
	Concentric (C)		Consider Coordinate Systems
	Parallel (L)		Lock To Base Reference
	Tangent (T)		Same Radius
	Tangent Touching (G)		Keep Orthogonal to Base
	Coplanar (P)		Suspend Design Intent Settings
	Perpendicular (J)		
	Symmetry About Base Planes (S)		Relax Dimensions (D)
	Symmetry about YZ (Y)		Relax Persistent Relations (N)
	Symmetry about ZX (Z)		Solution Manager (V)
	Local Symmetry (Ctrl-Y)		Auto Solution Manager
	Aligned Holes (A) – Coplanar axes of holes, Cylinders, revolved features		Design Intent Options
	Aligned/Coplanar in XY (Q)		
	Aligned/Coplanar in YZ (W)		
	Aligned/Coplanar in ZX (E)		
	Aligned/Coplanar in canted planes (Ctrl-A)		

c. Solution Manager

The Solution Manager is a way to manage failure. When an edit will not work, you would typically adjust the Design Intent settings. If it continues to not work the way you want it to, you would resort to the Advanced Design Intent panel to control the Intent in a more detailed way.

If you still cannot get the edit to work, or if you just want to investigate the reasons why the model is behaving the way it does, the Solution Manager is the way to go. This can be used for either Steering Wheel or PMI edits.

We will do a quick overview of Solution Manager at the end of this chapter to show you that there is always a way to get the solution, and that there are reasons for your model acting the way it does. Below is the classic simplified model often used to demonstrate the functions.

Not changing

Selection set

Error path

Solved correctly

Isolated faces

Rigid set

Grounded

Relaxed
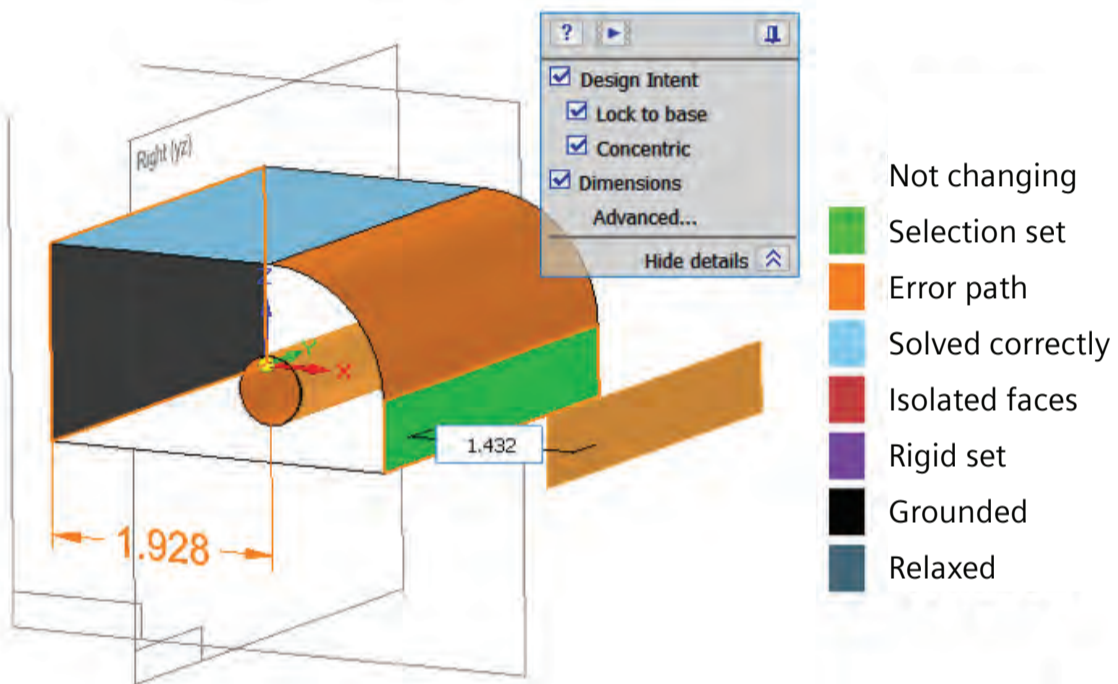
3. **Face Relations**. Manually applied relations that work like sketch relations, but are applied directly to faces in the model.

a. Face Relations are listed in PathFinder.

b. Faces are tracked in PathFinder

c. You can repair/replace by right-clicking a listed face.

4. **Steering Wheel**. You can use the Steering Wheel to move (translate or rotate) faces that are not explicitly defined by any of the previously mentioned tools.

a. The combination of the current selection set and faces selected automatically by Design Intent determines which faces will be edited in a given operation. The selection set in essence replaces the history-based feature, but it is more flexible.

b. The Steering Wheel element that you choose to use (arrow or torus) determines whether the change will be a translation or rotation, respectively.

c. You can move the blue handles to determine the direction of translation or center of rotation.

d. Select the translucent disk to make a planar move.

e. You can move the starting point of the edit (blue dot), and drag to a keypoint to get a From/To type edit, or type in a value, or just drag to make the move.
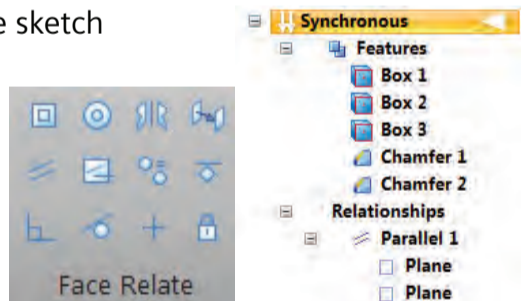
From this information, we can summarize that with synchronous technology:

1. Design intent comes from the geometry.

2. Design intent can be changed on the fly.

3. Since design is not a linear task, the order of features is irrelevant.

Synchronous technology brings together the most powerful tools in history-based techniques with the most powerful tools in Direct Modeling techniques to give you a full range of capabilities to create whatever you need to create.

Tutorials

In this chapter we talked a lot about concepts and thrown around a number of ideas. But in this tutorial, you will see for yourself some of the power of design intent in synchronous technology.

For this, and subsequent tutorials, you can get a 45-day trial of the current version of Solid Edge. This is available through downloading a local installation or streaming the full version over the internet.

Tutorial 1: Using PMI dimensions to change model

This tutorial was created using Solid Edge ST8.
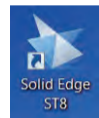
1. Start Solid Edge from your Windows Start Menu or from the icon on your desktop.

2. Click the **Application** button, in the upper left, click **New** and then select a part template with your choice of units.
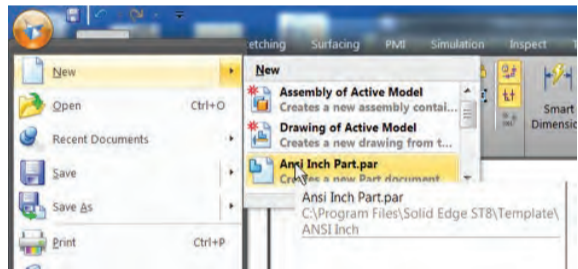
   Use a default template for now. We will set up custom templates for you in a later tutorial.

3. Choose **Home** tab –> **Solids** group –> Box .

   The Box command creates a box primitive without requiring a sketch. For other shapes, you can sketch the shape and use the Extrude command.

4. Move the mouse over the base planes in the center of the display, and when the Top plane is highlighted, press F3 to lock your initial rectangle to that plane.

5. In the CommandBar, set Selection Type to **by Center**.
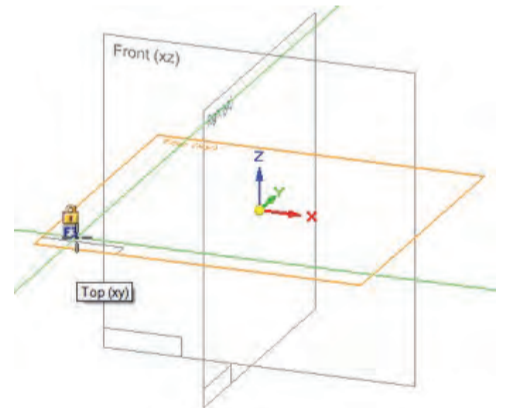
   Note: Your CommandBar may not look like the one shown here. I am using the Maximum Assistance theme, which includes more text in the interface. The default CommandBar is docked horizontally at the top of the graphics window, and uses icons rather than text.

6. Click and release the left mouse button on the origin (Base), then move the cursor away from the origin.
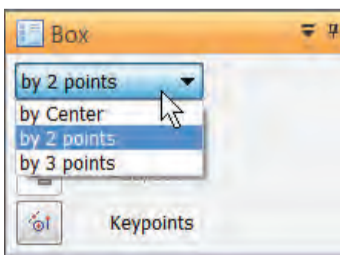
7. Create the box, assigning actual dimensions, in a series of short steps:

    a. Type 5.

    b. Press Tab.

    c. Type 4.

    d. Press Tab.

    e. Press Enter.

    f. Move the cursor so the light blue solid appears in the positive Z direction.

    g. Type 3.

    h. Press Enter.
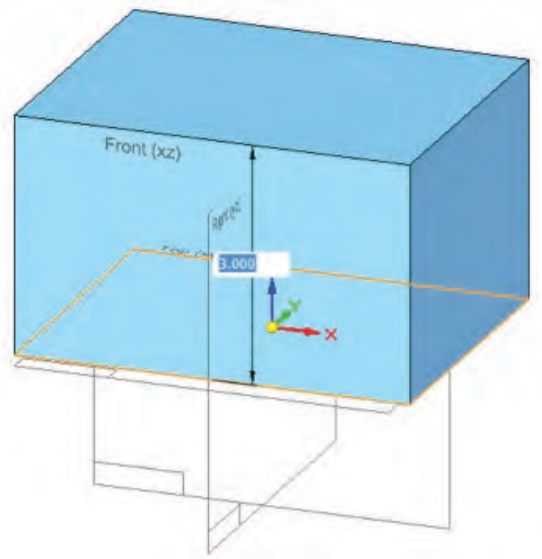
    You could also simply drag with the cursor and click to accept dimensions visually.

    The dimensions placed on the resulting solid are blue dimensions, which means that they are unlocked. You can change these dimensions manually by entering values or using the Steering Wheel. The system can automatically change the dimensions if another dimension changes and requires values to change in order to solve properly.

8. In the images that follow, note that the red arrows on the 5.000 dimension value are pointing in different directions, depending onthe cursor position on the dimension value. When the cursor is over the left or right side of the dimension value, the dimension changes on the left or right side of the block, respectively. When the cursor is in the middle of the number, the dimension changes symmetrically.
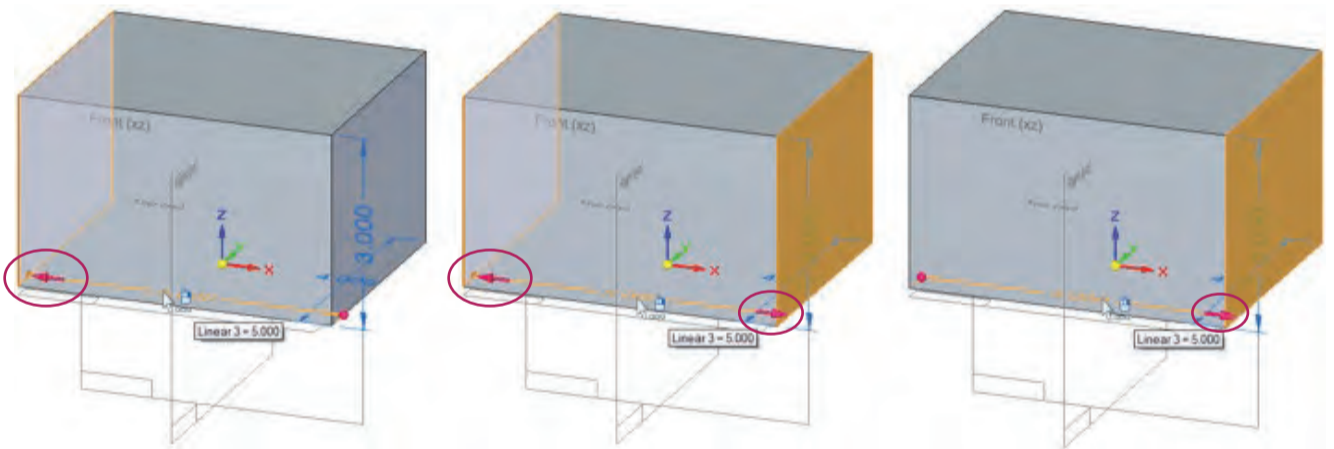
    It may be easier to understand the correlation between the red arrows and the cursor position by trying it: move the cursor from the left of the dimension value to the right. Watch how the red arrows change.

9. When both red arrows are displayed, click the 5.000 value, in the middle.
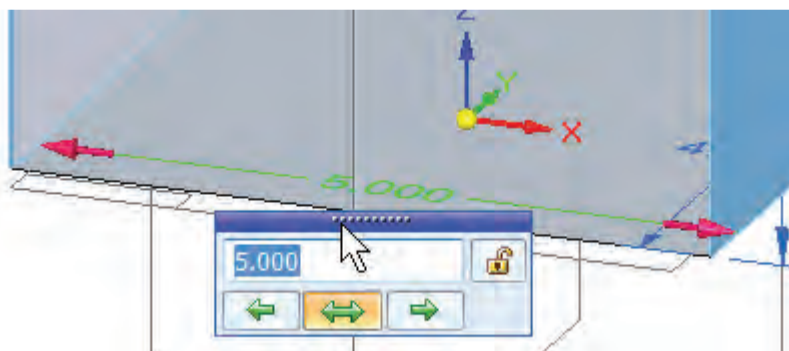
    The display of the red arrows on the screen is reflected by the selected option at the bottom of the value box (double-ended green arrow highlighted in orange).

    If you use your mouse scroll wheel, the gray solid changes symmetrically.

10. Change the dimension value and lock the dimension by doing the following:

    a. Type 7.5.

    b. Click the lock icon to the right of the dimension value.

    c. Press Enter.

    The solid changes size symmetrically and the dimension turns red. Red dimensions are locked, meaning they can only be changed when you manually type in a value.
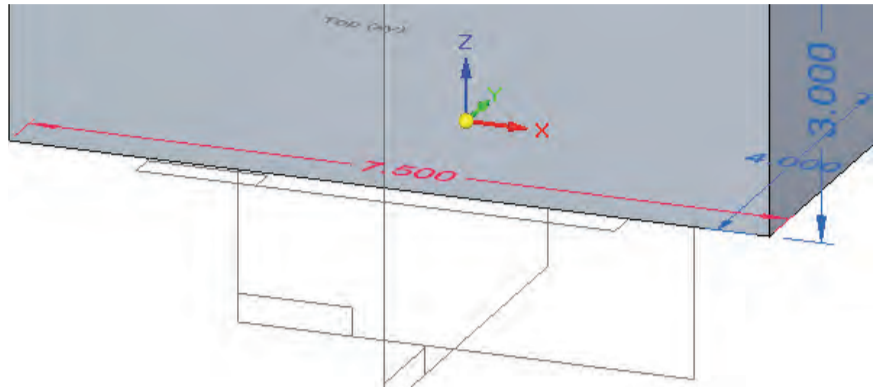
11. Click the vertical 3.000 dimension.

12. To change the dimension so it will change on the positive Z side, click the green arrow that points to the left.

    Do not lock the dimension.

13. Type 3.25 and then press Enter.

14. Click the top face of the box.
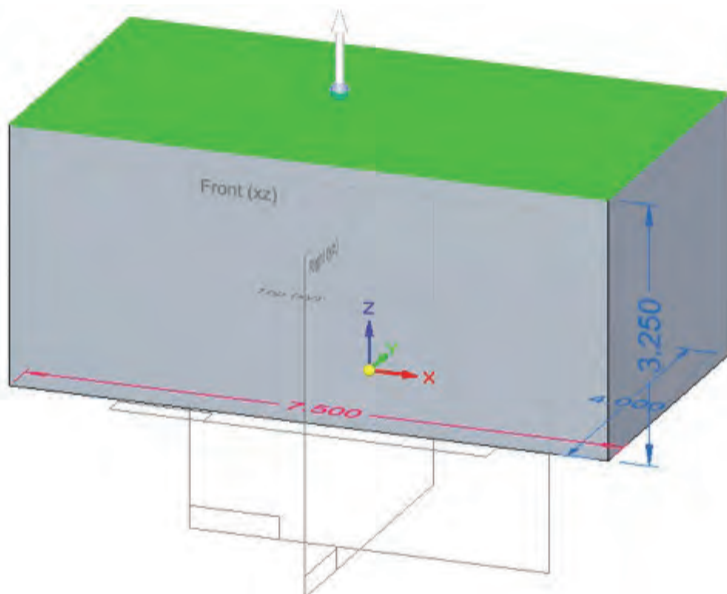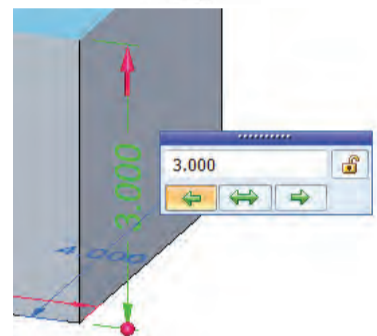
A white arrow is displayed on the top face. This is the Steering Wheel in a simplified form. The software assumes that if you just click a single flat face, you want to move that face.

15. Click the Steering Wheel arrow.

    A value box is displayed on the model. You can just click to get a visual dimensional change or type in a value to get an exact *differential* value. The value that you type in here will add or subtract from the 3.250 dimension.
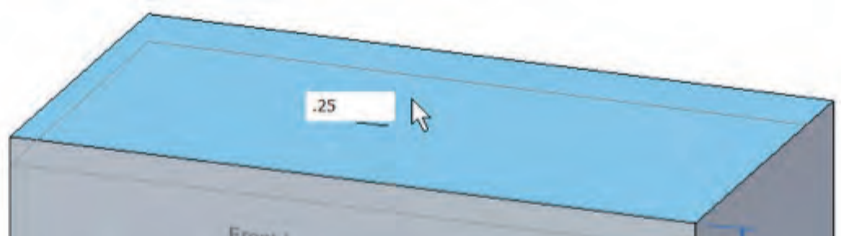
    You changed this value in two ways: directly typing in a new value for the dimension and directly typing in a differential change amount for the dimension.

    This was a blue (unlocked) dimension, which is what allowed the Steering Wheel to change it. The red 7.500 dimension cannot be changed by the Steering Wheel because it is locked.

Tutorial 2: Design intent and solution manager overview

Now you've used simple dimensional changes and been introduced to a simple Steering Wheel change coupled to an unlocked dimension.

You can follow this tutorial either in print or the linked video.

1. Open the file from the folder for this chapter called "tutorial2 start.par".

2. Click on the face highlighted green in the image to the right.

3. Click on the Steering Wheel arrow and drag it back and forth.
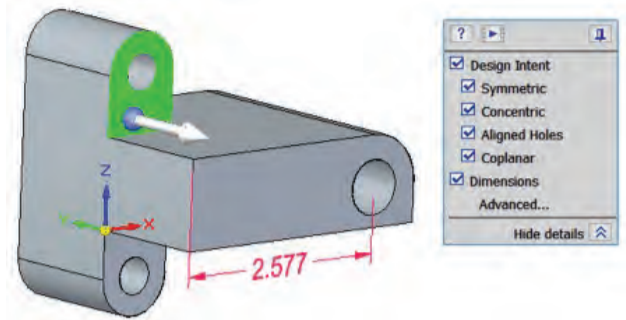
   Notice the lower ear also moves. To prevent this from happening (or to cause it to happen if it's not happening) remove the check from in front of the Coplanar option in the Design Intent panel.

4. Remove about a half inch of material, using the number in the white box as a guide, then click the left mouse button to accept the change.

5. Next let's make a change that you might have to think hard how to do it in ordered, but in Synchronous it becomes easy.
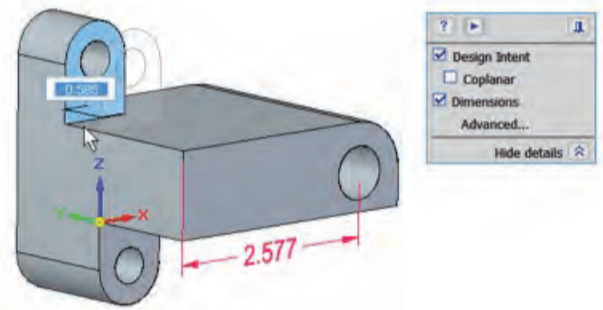
   Let's take the hole on the top ear and angle it in the YZ plane. Another way to think about it is to rotate the hole about a line parallel to the X axis.

   To get started, select the inside face of the top hole, as shown. You may notice that you are getting the simplified Steering Wheel instead of the full wheel shown to the right. To get the full Steering Wheel, click on the big blue ball at the center and just drag it away, then drag it back to the edge of the hole, and it will snap to the center, and display the third axis as well as the torus and the tool plane.
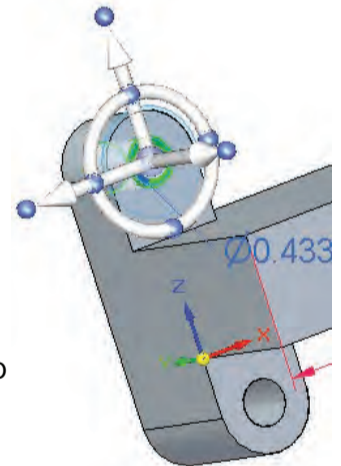
   Also notice that as shown, you cannot get the rotation you want. We want to rotate about X, and the Steering Wheel is currently oriented to rotate about Y. To rotate the Steering Wheel, Shift-click on the tool plane (blue translucent disk coplanar with the torus).

   Note that when you select a hole or cylindrical cut, the Steering Wheel allows you to move or rotate the hole, but you cannot use the Steering Wheel to resize the hole. Selecting a hole brings up a diameter dimension which can be used to resize.

6. Now you can click on the torus and angle the hole as well as the cylindrical outer face together. Play around with this to get the feel for what you can do with the torus. This is a type of edit that you may not even consider doing in ordered modeling. Also remember that it is completely valid to use normal methods to create something and then edit it to achieve an orientation in which it may have been more difficult to create.

7. When you are done, use Ctrl-Z (undo) to return the model to its previous state.
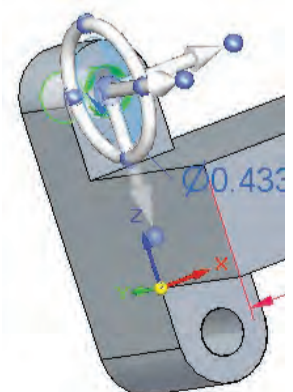
   Ctrl-Z in Solid Edge works to undo until you save the model. You cannot undo functions previous to a save.

8. If you use the Steering Wheel to move in the X direction, you will find that the entire model moves. This is because no faces have been grounded, and the "Lock to Base Reference" Design Intent option is turned off.

   Let's first turn on that option, test it, and then ground some faces.

9. With the hole face selected, click on the Advanced... option on the Design Intent panel.

10. Click on the Lock to Base Reference option so that it turns orange. Now try to move the hole in the X direction again. Notice the end radius on the ear gets bigger as you move the hole in the X direction because the face is tied to the end of the part that was just locked down with the Lock to Base Reference Design Intent option.
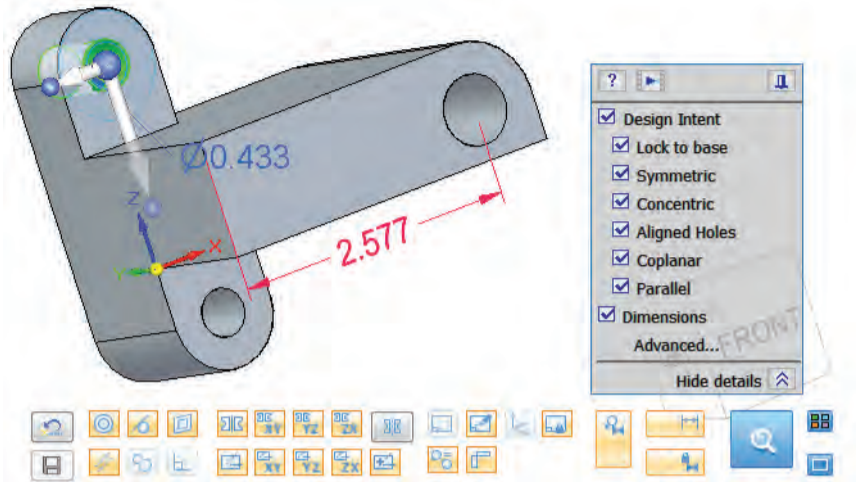
   Next select the tool plane and move the hole in the XZ plane. This is just to get used to editing with the Steering Wheel, and the Design Intent options.

   Finally, return the model to its original state with Ctrl-Z, and turn off the Lock to base option in the Design Intent panel. If you have deselected all model faces, you may have to reselect a face to gain access to the Design Intent panel again, as it only appears when geometry is selected and ready to edit.

11. Next let's lock down some faces, but first we have to reposition the model with respect to the origin.

   Rotate the model so you can see the back face, then drag a box around the entire part, and move the Steering Wheel to the bottom hole, as shown to the right.

   We are going to reposition the part so that the Origin in on the back face, and then lock down all faces that are on the base planes.
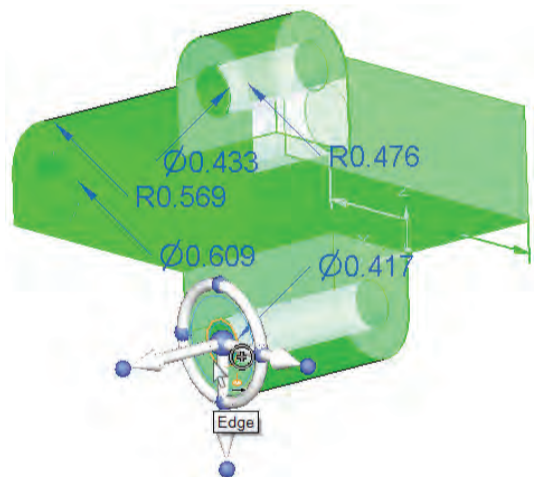
12. Click on the Steering Wheel arrow that is parallel to the hole axis. This enables us to move the entire model along the Y axis.

   Move the cursor to the Origin and click the Origin keypoint, as indicated in the screen shot below. This moves the entire part body along the Y axis so that the back face is on the XZ plane.

   This technique for moving items will be very handy as you use the software.
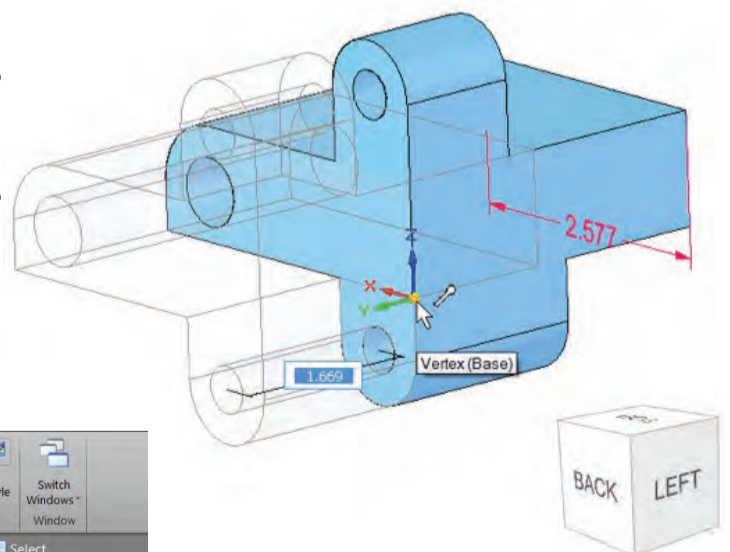
13. Press ESC to clear the selection.

14. On the Ribbon, Home tab, Face Relate group, click on the Ground tool.
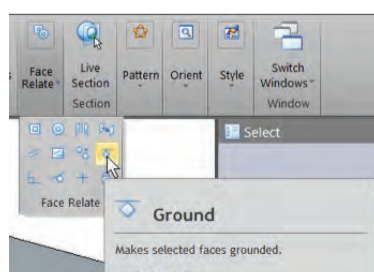
15. Next, select the three faces shown in the image to the right, and click Accept on the Command Bar.

    You will not have to hold down the Ctrl key to select multiple faces while the Ground function is active.
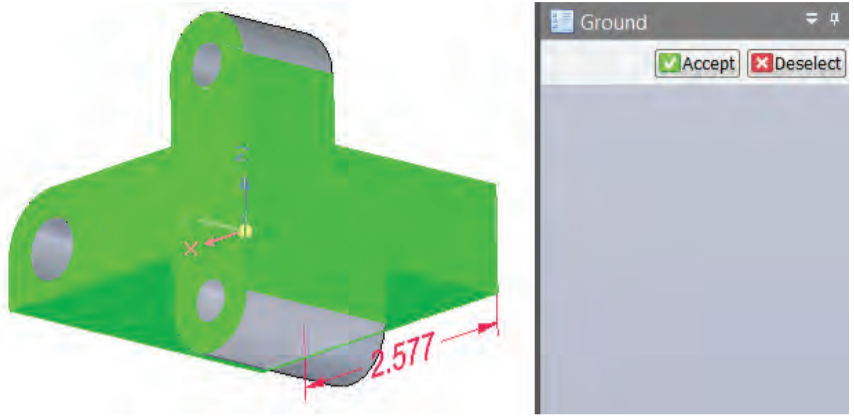
    Note that for these screen captures, I am using the Maximum Assistance theme, which adds more text to the Command Bar and orients it vertically instead of the default horizontal Command Bar. This setting can also be accessed independently from the themes using the Application Menu, Solid Edge Options, Helpers, Command User Interface, Use vertical docking window form.
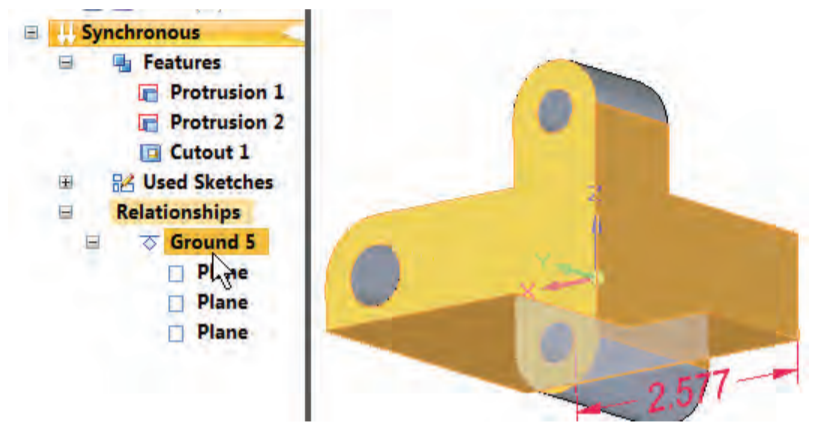
    Note also that the face relationships are stored in the PathFinder, as shown to the right. Very easy to remember what you've applied to the model. Faces highlight when you mouse over the relation.

16. Now we will move on to using the Solution Manager, which was mentioned earlier in this chapter. The Solution Manager helps you sort out which relationships are causing a particular edit to not work as expected. It also helps you visualize different solutions to the problem by analyzing the error path, and relaxing one of the overconstrained links connected to the face you tried to edit.
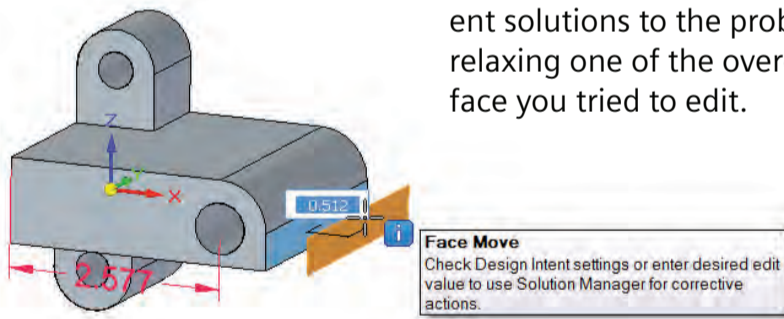
    Select the end face of the model, as shown, and drag the Steering Wheel arrow out from the model. This edit is failing for reasons we are about to research.

    Finding out why things happen is an important way to understanding how things work and what you should reasonably be able to expect your tools to do. A few extra minutes spent researching problems from time to time will build your skills and knowledge of the software significantly.

17. Click to place the face about an inch from the original. Notice how the model again turns colors.

The colors, from top to bottom, are:

Transparent = irrelevant
Green = selected
Orange = error path
Blue = solves correctly
Red = isolated (rules off)
Purple = rigid set
Black = grounded
Slate = rules relaxed
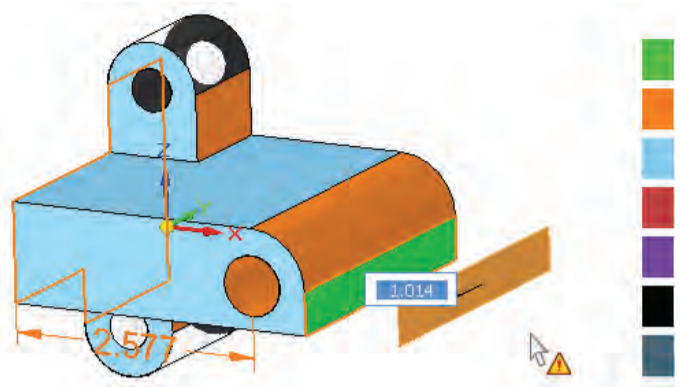
The colors that you are concerned about here are the orange faces. As a history-based user, you can think of all the orange faces being overconstrained. Orange is the "error path" color. Think of the path as a chain where each dimension, face relation, or Design Intent assumption is a link, and you have to decide which link to relax, or break in order to allow your edit to work.
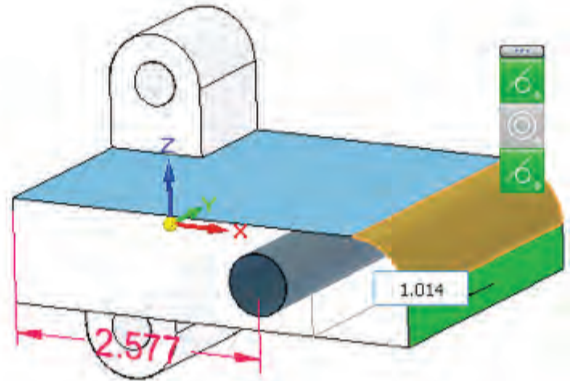
The way to examine the links in the chain is with right mouse button clicks and hovering with your mouse. For example, I can see that the fillet face is part of the error chain because it is orange. Notice that the 2.577 dimension is orange, because it is also part of the problem.

When you right click on the fillet face, the Solution Manager shows the set of relation icons to the right. The yellow triangle means that relation is part of the error chain, so the tangent between the fillet and the top flat face is not part of the problem.

This tells you that the fillet face has three Design Intent assumptions, and hovering your mouse over the icons will highlight the two faces involved in each relation. So hovering over the Concentric icon highlights the fillet face and the hole.

If you click on one of the icons, it will go gray and break that relation temporarily, and show what the model would look like with that relation broken. Clicking on the Concentric relation breaks the error chain, and fixes the edit error.

To put the model back the way it was, click on the gray icon again to reactivate it and show the model in its error state again.

18. Click the right mouse button again to hide the icons, and then right click on the face of the hole. This brings up the concentric icon as well as a dimension icon. If you click the dimension icon, it goes gray, and the Solution Manager turns the 2.577 dimension to an unlocked (blue) dimension, meaning that the edit would drive the dimension. This also solves the problem , but in such a way that the hole moves to match the fillet.
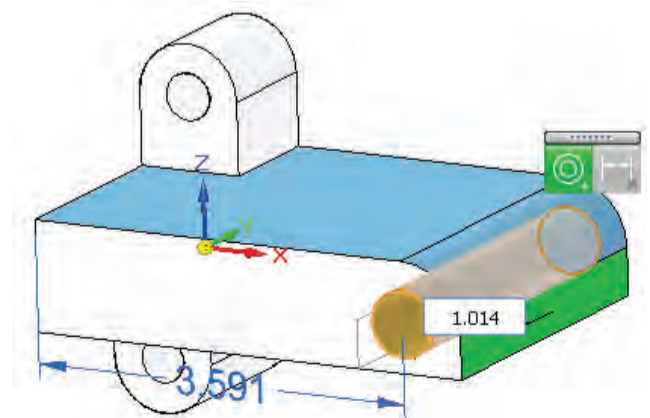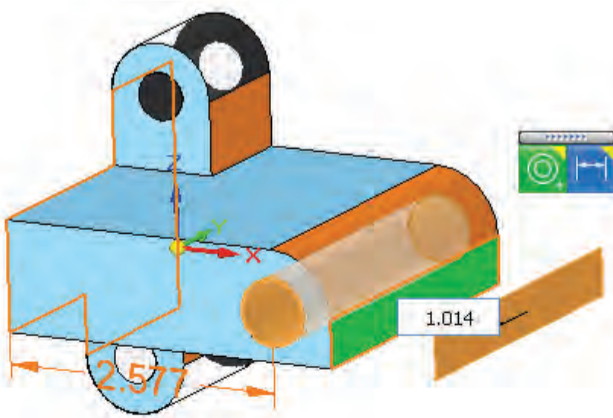
19. Accept this result by clicking on the green checkmark in the Advanced Design Intent.

You may also want to watch the video associated with this tutorial. It does not follow the steps exactly, but shows some additional techniques.

There are a few things here to remember, but with all of the symbols decoded and the ways to access the information spelled out, the Solution Manager is accessible. With the Solution Manager as a safety net, you can feel more secure in exploring more synchronous edits, and feeling better about the idea that synchronous really works backwards from the "fully con-strained" concept. Nothing is going to move unless you allow it, which is the reverse of the basic history-based concept which is everything can move unless you constrain it.

Summary

In this chapter you learned some of the practical details of how synchronous technology edits parts. From PMI to Face Relations to the Steering Wheel and Design Intent with the Solution Manager, you are in control of how your model reacts to change. And changing how it reacts to change is far easier than with history-based modeling. Even troubleshooting overdefined relationships has a clear and understandable pathway to success.

Chapter 3

Introduction to Solid Edge

The Solid Edge® software interface follows Microsoft's standards. Most functions and interface elements will be familiar to most Windows users. You can customize the interface manually or use one of the pre-established Themes that use commonly used groups of settings that different types of users look for.

The Solid Edge workflow, or order of operations is often unique to Solid Edge and doesn't follow conventions from other CAD packages. When learning Solid Edge, it is best not to expect it to work exactly like some other system. The general concepts are usually familiar to users of other software packages, but the details of how you accomplish specific tasks are often different. Terminology is often unique to each system as well.

Solid Edge interface

Let's continue by identifying the major parts of the Solid Edge interface.
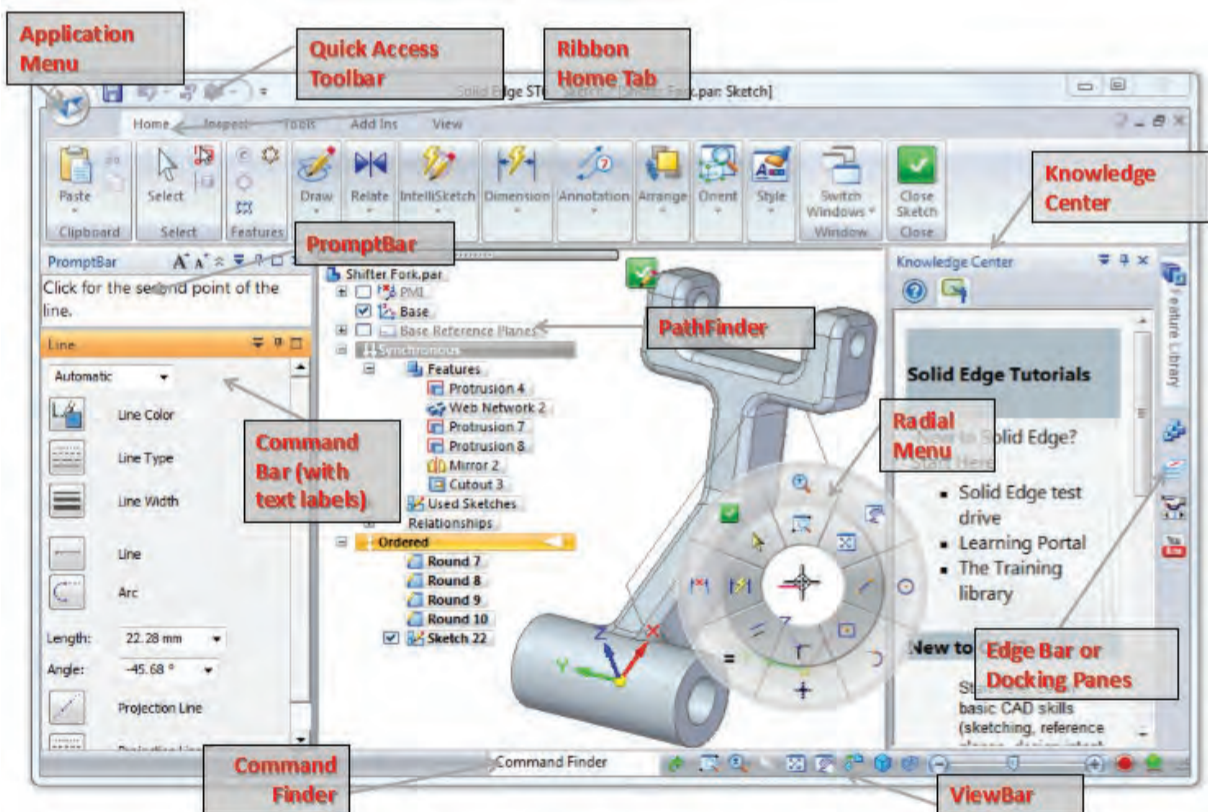
Additional panes can be turned on or off using the View >Show > Panes dropdown list. All of these panes can be docked, or moved around the screen, and even outside of the Solid Edge application window to put them on a separate monitor. Initially, when learning Solid Edge, you will use the Learn Solid Edge pane frequently. The Prompt Bar is important for new and learning users. You will use the Select pane frequently to create and modify features. The rest of the panes come up on an as-needed basis, however I would recommend the YouTube pane for new users who might want to ask questions using video.

Note: The Solid Edge User Community is a great place to ask questions and get answers about the software. You need a login to ask questions, but you can read what other people have written without a login. Give it a try: https://community. plm.automation.siemens.com/t5/Solid-Edge-User-Community/ct-p/solid-edge

Customization

Everyone has a different way of working, and everyone has different desires when it comes to how they use the software interface. Because of these differences, every user has different needs from customization. If you're using Solid Edge eight hours a day, getting things set up properly can save you a lot of time. Some users just want the interface organized in a certain way. Some experts are looking for speed, some who are still learning the software may need to make things easy to find.
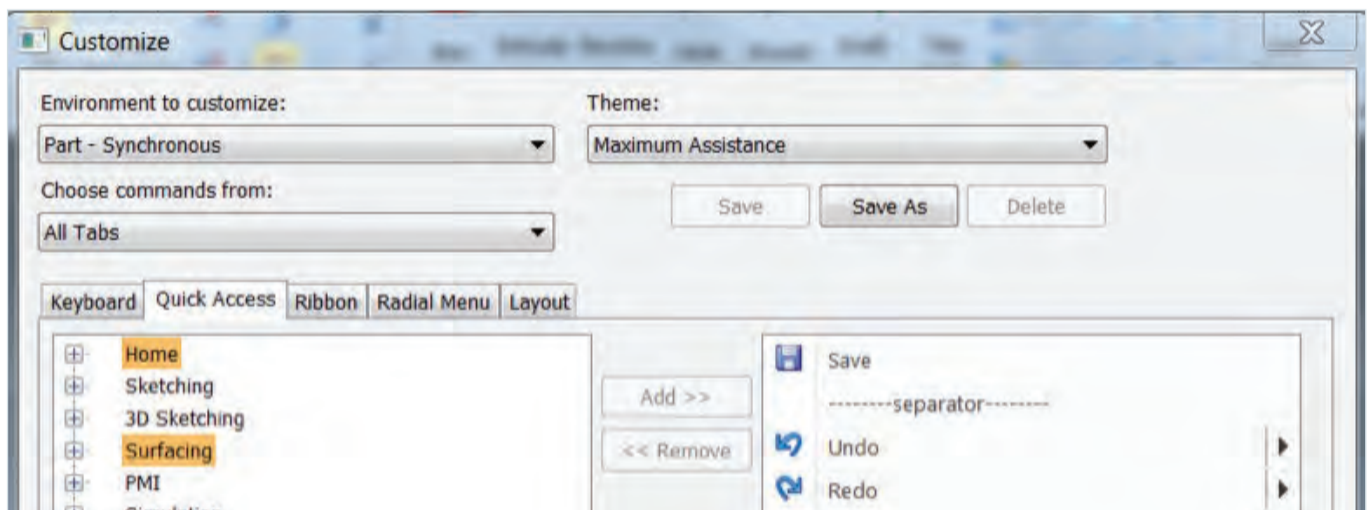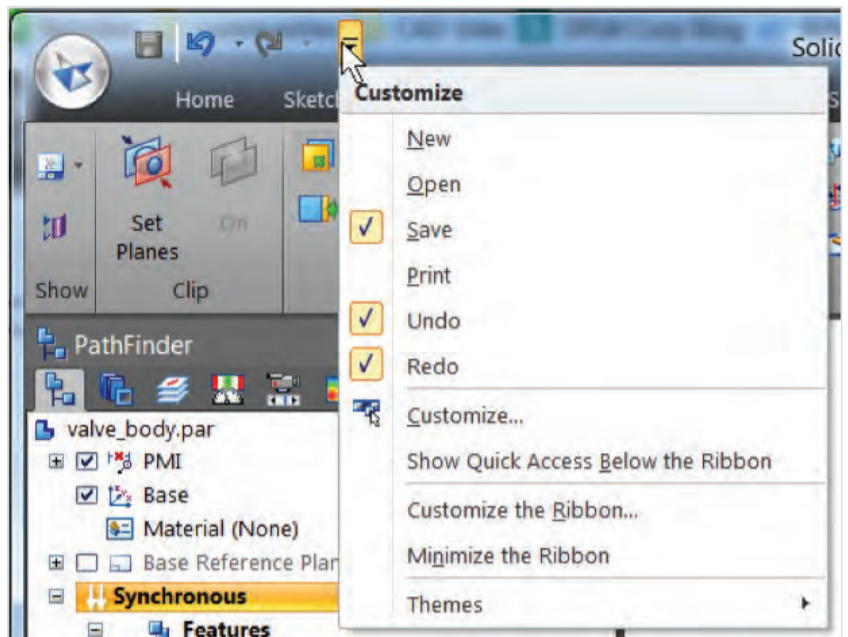
Still other goals for customization might be efficient use of space, total number of clicks to accomplish a task, total amount of mouse travel, or having all the tools you might use available when you need them. Some companies even require that all users use the same interface setup. Whatever your goals, realize that there are probably conflicting ideas that you have to balance out. For example, the default Solid Edge interface settings are very space efficient, but you have to be well acquainted with the interface in order to use it that way. On the other hand, if you use the Command Bar with all of the text labels on it, that makes it easy to find things when you're not sure what you're looking for, but it also takes up a lot of space. Keyboard shortcuts can be accessed with no visual interface, but require keystrokes and memorization. Radial menu requires two keystrokes for every command, and is limited in the number of commands it can contain.

Themes

Just be aware that there are always trade-offs, and each decision requires a balance of potentially conflicting factors. This is part of the idea behind the Solid Edge Themes. Solid Edge Themes are out-of-the-box customizations aimed at the needs of different kinds of users.

Examples of the Themes are:

- Some Assistance – For users who are partially familiar with the interface and workflow, but still need a hint from time to time.

- Maximum Assistance – This is for beginners who are working through tutorials or attending classes, and aren't yet familiar with the Solid Edge workflow and tool icons. (This is the theme used for most of the screenshots in this book).

- Maximum Workspace – This option is primarily for expert users who can identify each icon by sight, and don't need tool tips.

- Balanced – This is the default Solid Edge settings which are most useful for users with more than a year using Solid Edge.

You can set a theme, customize it, and then save the customized theme to a different name. You can even move the theme to other computers or other installations of Solid Edge on the same computer.

Customization is available from the drop down on the Quick Access toolbar at the top left of the screen. The various toolbar areas are listed as tabs in the Customize dialog, and the entire list of commands is available in the window on the left. You can use the Add and Remove buttons between the left and right windows to customize each interface element. Use separators, spacers and the Move Up and Move Down buttons to organize your buttons in a way that fits your workflow or tool search style.

The Radial Menu is sometimes called the Marking Menu. It appears when you hold down RMB. This can be customized for each environment: Sketch, Part, Assembly and Drawing.

Customizing for speed and efficiency

The Keyboard tab can be used to create keyboard shortcuts (hotkeys). Hotkeys are fastest way to use the Solid Edge interface, with the downside being the need to memorize them.

If you're looking for speed and efficiency, the radial menu, shown to the left, is another great option. The radial menu can be accessed by holding down the RMB for a second.
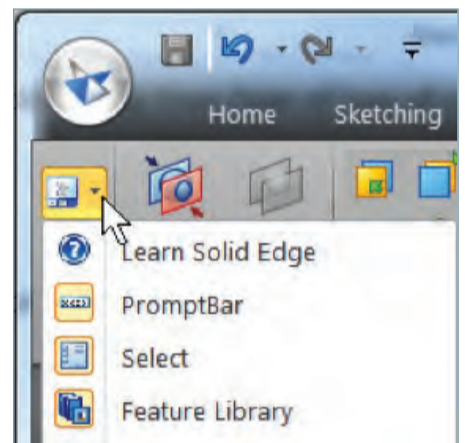
Panes can be docked to the sides or floated inside or outside the Solid Edge application window.

The Ribbon

The Microsoft Ribbon interface has generated a lot of mixed feelings. However, some people believe that the Solid Edge implementation of the Ribbon works better and makes more sense than the original Microsoft Office® software implementation. The Solid Edge ribbon expands and contracts to fit the available space. As it contracts, most of the space is saved by removing the text labels and collapsing grids of icons into dropdown lists.
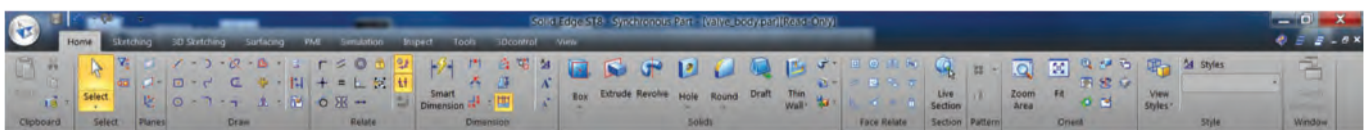
Ribbon for 1920 pixel display.
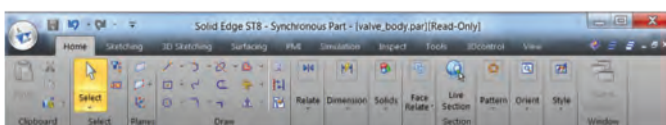
Ribbon for 1024 pixel display.

You can switch between the tabs by clicking on them or by pressing Alt and then pressing one of the keyboard shortcut keys that appear. Notice that the shortcuts are on the Quick Access bar and Application menu as well as the Ribbon tabs.
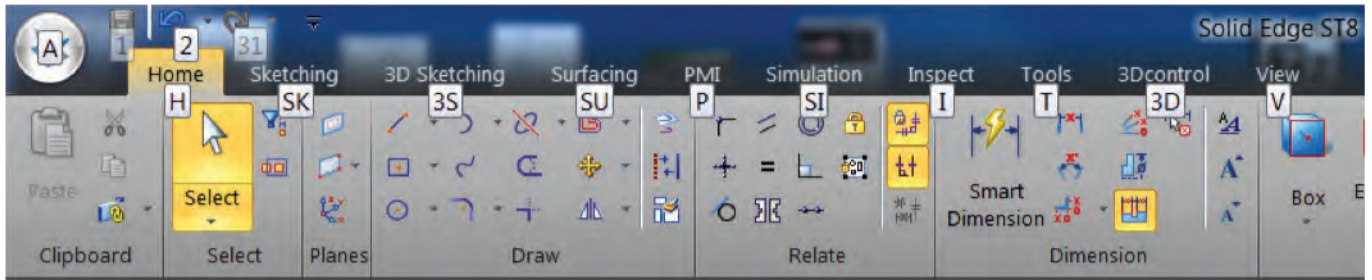
Pressing the Alt key causes shortcuts to the tabs to appear.

ToolTips

One easy way to get help about the interface quickly is to use ToolTips. You can access ToolTips by hovering the mouse over buttons in the interface. Solid Edge has three types of ToolTips, which you can control in Solid Edge Options (Application menu > Solid Edge Options > Helpers> Command Buttons):

1. **Basic**

   Basic ToolTips show the tool's icon, its name, and a brief description of what it does.
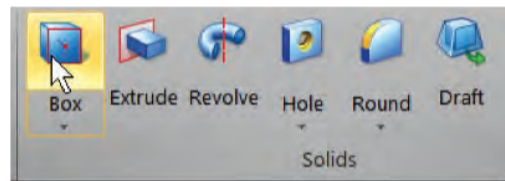
2. **Enhanced**

   Enhanced ToolTips include the basic ToolTips plus some general tips on usage and an image that acts as a visual aid.
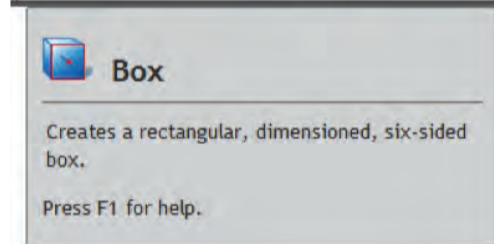
3. **Video clips**

   The Video clips ToolTips looks like the Enhanced, but in place of the static image, there is a brief animated image showing the tool in use.

   For beginning users, the Video clips ToolTips are really a great idea. Very often all you need is that brief description of how the tool works in words, and then a video to show it in action to get you moving. General help often lacks important details like where to find something or if there are any prerequisites for a particular function, and sometimes you can find this sort of thing in the ToolTips.
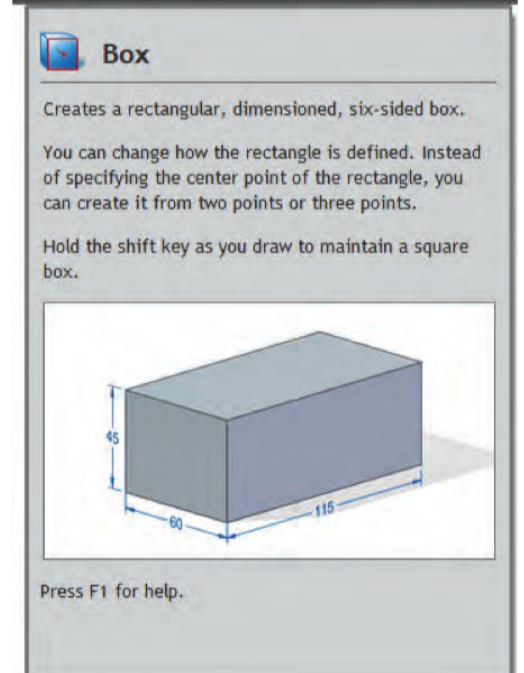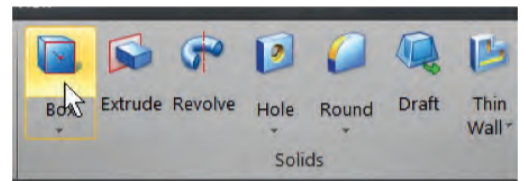
Command Finder

Another type of help that you can make use of for help with the interface is the Command Finder at the bottom of the screen. Just type in the name of the command that you are looking for in the box, and Command Finder pops open a window with a list of related commands.

You can also tell it to look for commands that are outside of the current environment. For example, if you search for center mark, and are in the Part environment, Command Finder might return a list of tools that contain the word "center" or "mark" in their names or descriptions, but

the actual "center mark" tool is found in the Draft environment. If you look at the bottom of the Command Finder window, you will see an option to click for "Show matches outside of Environment", and these matches will be shown in a lighter gray font. If there are not matches outside of the environment or you have already clicked that option, the option will not appear. Choices that are grayed out will not flash to show you where they are, because they cannot be accessed in the current environment.

In the returned list, mouse over your choice of commands, and the Ribbon changes to show you where the command is and the button flashes. When you are done with Command Finder, the Ribbon stays in the configuration where you can find the command you were searching for.

Workflow

Workflow, in this context, means order of operations. Most of the Solid Edge interface works with the "Action => Object" workflow paradigm, where you select the tool you want to use, and then select what you want to use the tool on. An example of this is the workflow to create a sketched extruded feature. The workflow is shown here for both Synchronous and Ordered:

The Command Finder can help you find tools in the interface, but also can help you find equivalent terms from other CAD systems

1. Synchronous

    a. Click sketch tool

    b. Select plane or planar face

    c. Create sketch (click-click method, not click-drag)

    d. Click on the Extrude tool button

    e. Select closed region/chain

    f. Right click to accept extrude region

    g. Set to add or remove material/other options

    h. Drag a distance or key in a value

2. Ordered

    a. Click Extrude (or Cut)

    b. Select existing sketch or a plane to make a new sketch on

    c. Draw sketch (click-click)

    d. Close sketch

    e. Drag or key in distance/set options

    f. Right click to Finish

    g. Click Cancel to avoid making another extrude

You can also use variations of some of these steps, but these are the commonly used workflows for these features.

Other popular CAD packages enable you to select the plane, the sketch or the feature button in different orders, but Solid Edge wants you to follow the process.

I mentioned that the preferred workflow is Action => Object, but there are times when the requirements are the opposite. For example, when you make a mirror or pattern in Synchronous mode, you must have geometry selected before the Mirror or Pattern tools become available. When creating mates, you select the mate tool first, then the geometry.

In general, although there may be a couple of exceptions, Solid Edge requires that you proceed one way or the other, but does not allow you to select tools or items in random order. This aspect of the workflow in Solid Edge can be difficult for SolidWorks® software users to accommodate.

The most important thing to keep in mind when learning a new CAD system is that you will be most successful if you learn the new system in the way it is meant to be used. Trying to force the new system to work like another system generally leads to frustration and failure. Solid Edge has a full complement of tools, but it does not work exactly like SolidWorks. Take this bit of advice from someone with 17 years of SolidWorks experience who changed to Solid Edge: Learn Solid Edge on its own terms. You will not always find a direct comparison between functions in the two systems.

Settings

Most of Solid Edge settings are located in the Solid Edge Options dialog found at the bottom of the Application Menu. Most of the settings in this box are global, meaning they apply to any document opened on the current computer. There are some settings in here that are document specific, however. These settings will travel with the file (or template) with which they are saved.
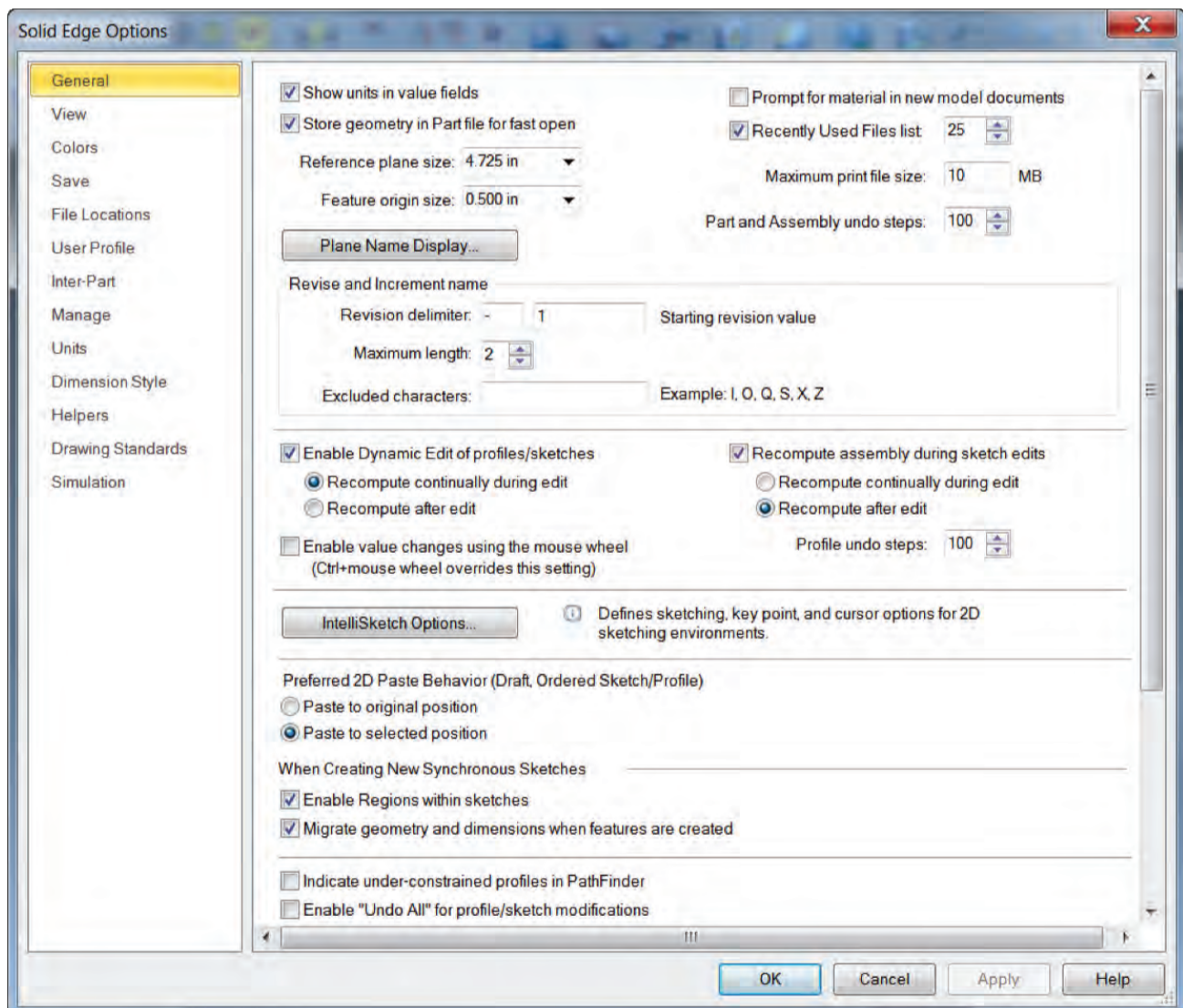
It may be a good idea for new users to browse through the Solid Edge Options dialog to become familiar with some of the settings you can control easily.

This section does not explain every option, but picks out the more used or potentially useful options for more complete explanation.

General

- **Show units in value fields**. Units in value fields can be helpful when you are working with mixed units, but can also get in the way when you don't need to be reminded.

- **Store geometry in Part file for fast open**. The trade-off here is that file size increases.

- **Enable Dynamic Edit of profiles/sketches**. Dynamic Edit in Ordered operations enables you to see the finished feature while you are editing the sketch, and the 3D geometry updates as the sketch changes.

  The Recompute options should be set to "after edit" if you have a slow or old computer or are marginal on some of the system specs such as RAM or video requirements.

- **Profile undo steps**. The level of undo buffer can affect RAM usage, but can be a great convenience. The idea of balance and conflicting desires should be applied here.

- **Intellisketch Options**. Intellisketch is the set of automatic sketch relationships or selection aids attached to the cursor while sketching. These options can also be set in the main interface on the Home tab.

- **When Creating New Synchronous Sketches**. These options should be left at default, which is On. Regions are very convenient, especially in Synchronous mode, and if you took the time to put a dimension on a sketch, it should probably move to the feature when it is created.

- **Indicate under-constrained profiles in PathFinder**. Users of history-based CAD systems tend to be very concerned about under-constrained profiles. With Synchronous, these worries should disappear, but some habits are hard to break.

- **Enable "Undo All".** This is the equivalent of exiting a sketch without saving.

- **When entering Profile/Sketch**. The "Orient the window to the selected plane" option applies to Ordered. The view will automatically reorient normal to the sketch plane.

View
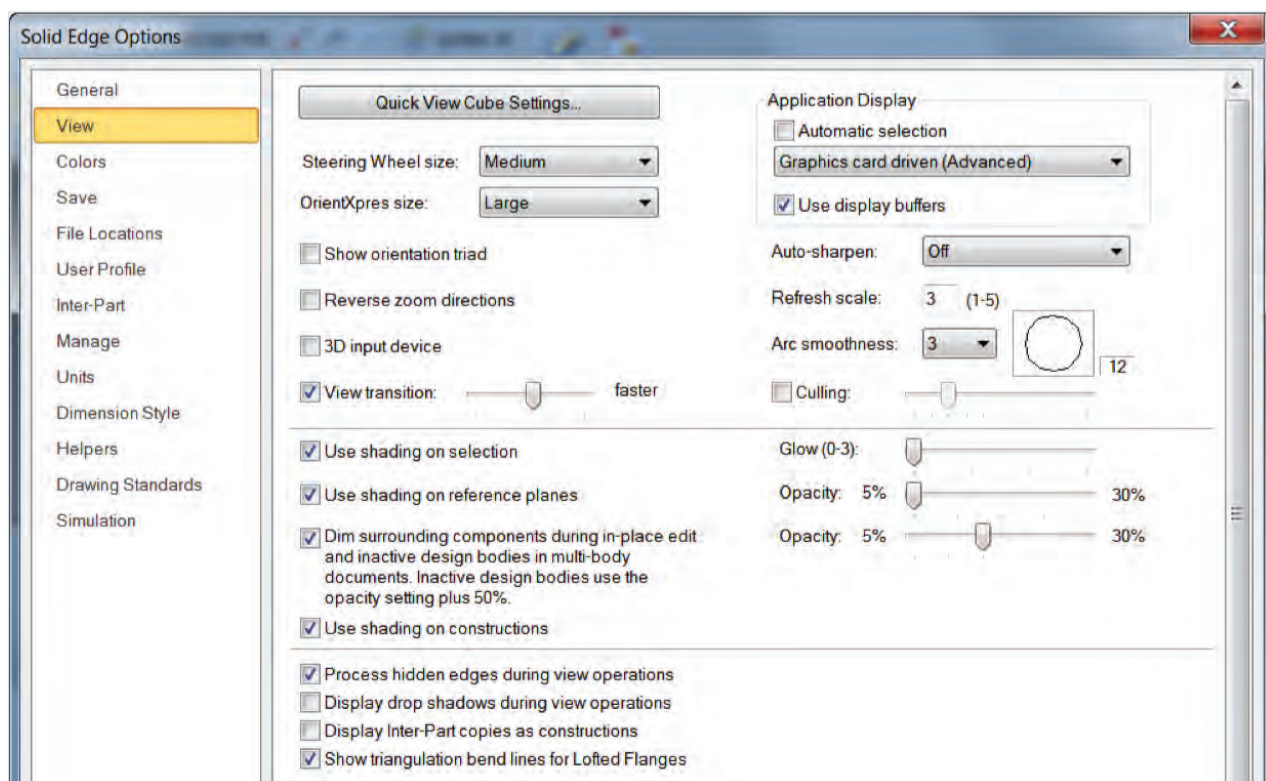
- **Quick View Cube, Steering Wheel and OrientXpress.** The size settings for these visual aids control how visible and selectable the on-screen elements are. Also controls how much space they take up on the screen.

- **Auto-sharpen**. This controls how closely Solid Edge approximates curves. When the arc smoothness is low, circles can look like hexagons. The better the curves look, the harder your video card will have to work to keep up.

- **Use shading on selection**. If this is off, a selected face will only show the edges high-lighted. With it on, the entire face will highlight orange when you mouse over it and green when you select it. The same settings apply to reference planes and constructions (surface bodies).

Colors
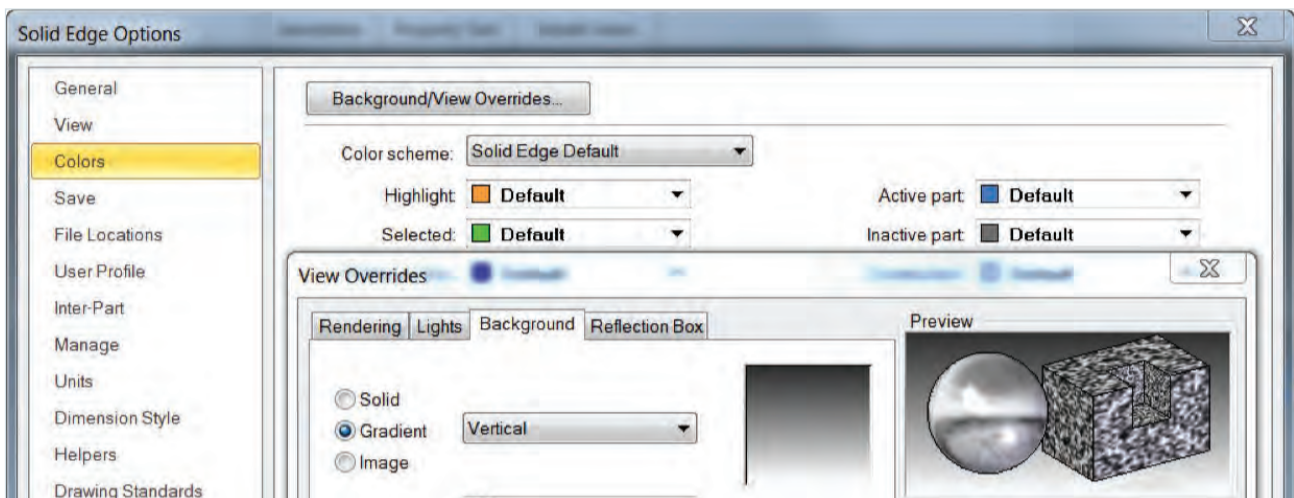
I would recommend that you leave the default colors alone. But notice that you can access the Background/View Overrides from this page. These are document specific settings which you can save with your templates. You can also access those settings by right clicking in blank space inside a part or assembly document.

Save
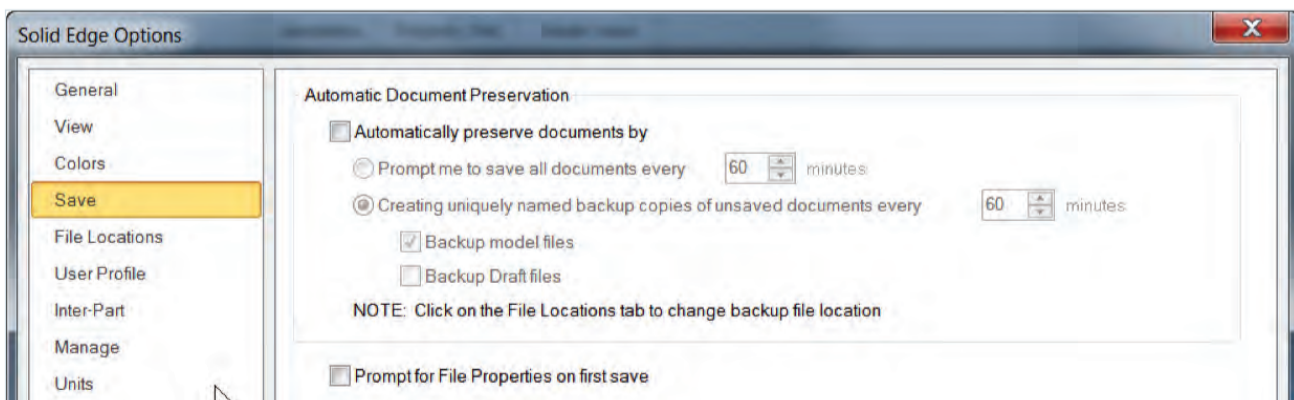
These settings are for automatic save and backup.

File Locations
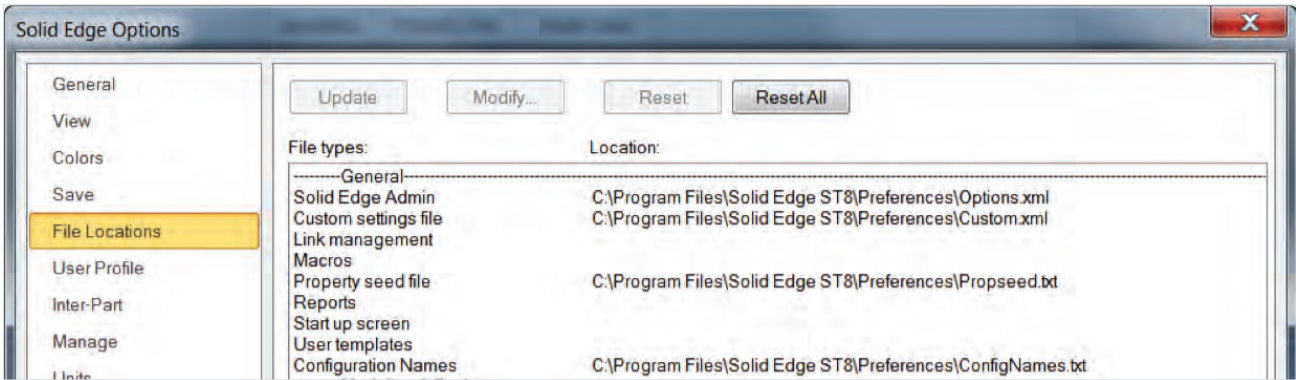
To change one of the file locations, double click on the path and then browse to the new location. Locations that might be important to control shared documents or documents used between versions of Solid Edge are:

- Solid Edge admin options
- Custom settings file
- Macros
- Property seed file
- User templates
- Tolerance tables
- Sheet metal gage file
- Holes database

Inter-Part
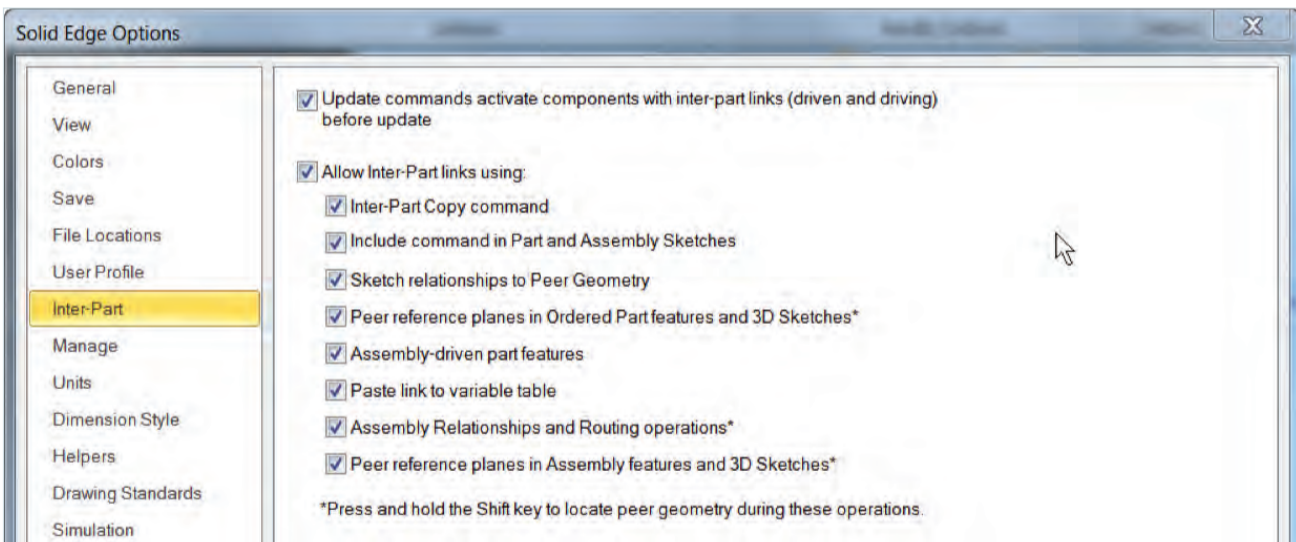
In Solid Edge, you can specify which types of Inter-Part (in-context) relations users can create. These are the relations that create associative links between parts in an assembly.

Units
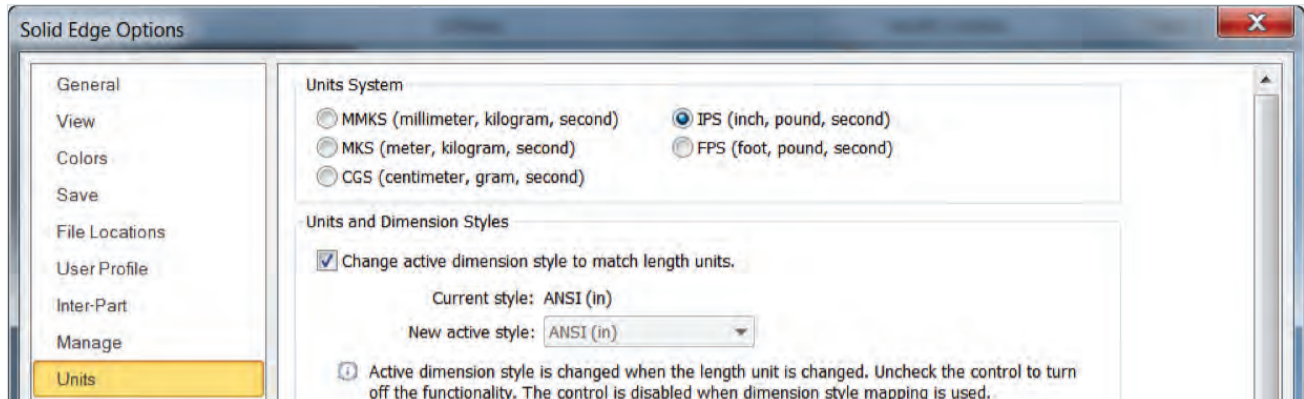
Units are probably better set up as template-driven settings. You also need to be aware of the role Dimension Styles play in controlling units, standards, and dimensioning before deviating too far from the default settings.

Dimension Style
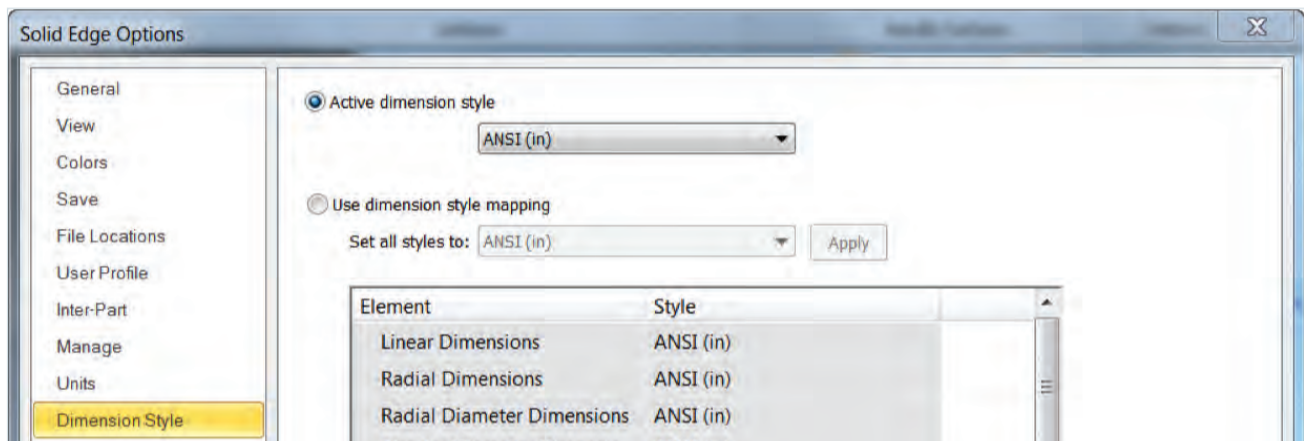
Helpers
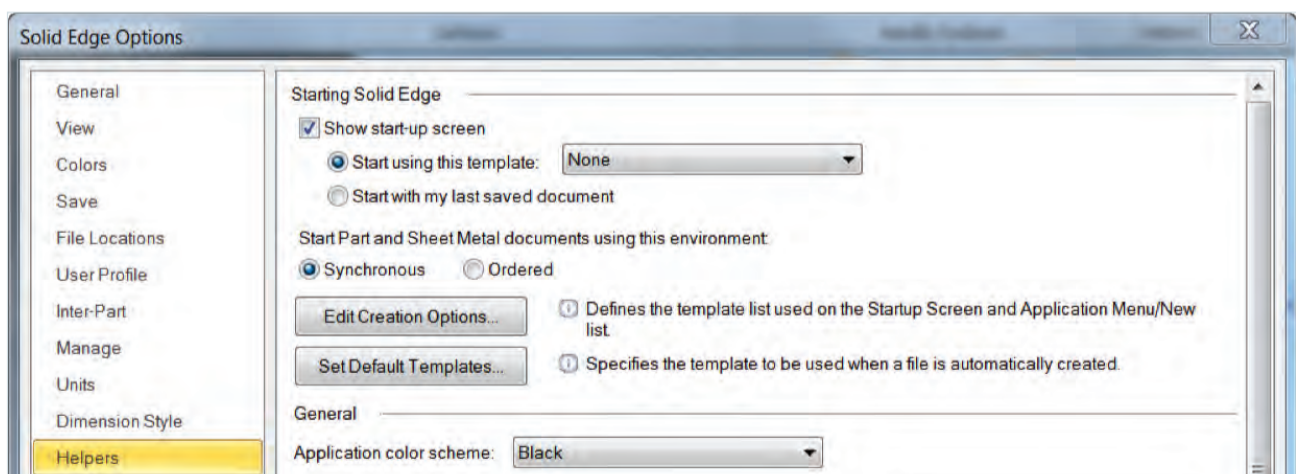
The Helpers page is where you will find many useful interface settings.

- **Start using this template/Start with my last saved document**. When this is set, a new session of Solid Edge will automatically start with a new document using the specified template. This is very useful if you start your day making parts, or if you work on one part for a number of days in a row. Set to None to disable.

- **Start Part and Sheet Metal documents using this environment**. Are you a fan of Synchronous? A hold-out for Ordered? You can always start your parts in the same mode with this setting.

- **Edit Creation Options**. You can remove extraneous templates from the Startup list to increase your efficiency.

- **Set Default Templates**. Most Solid Edge users have specific templates that they prefer for each type of document. This setting allows you to select which templates are used as defaults for each type of document.

- **Show PathFinder in the document view**. When this setting is ON, the PathFinder appears as an overlay in the graphics window. With the setting OFF, the PathFinder is put into a separate dockable pane. In the image on the left below, the setting is OFF, the image on the right has the setting turned ON.
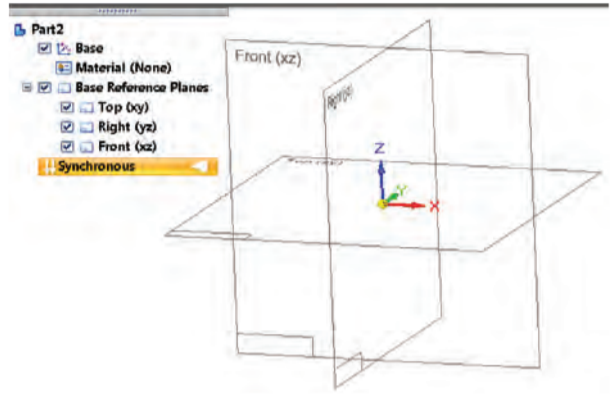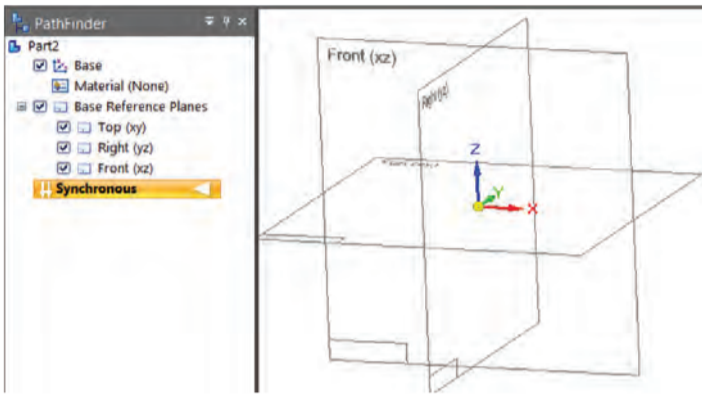
- **Show command tips**. Command tips are similar to ToolTips, but show up right in the graphics window while you are working. They show up as dismissable text bubbles from which you can access additional help. For brand new users these can be helpful. Resetting them and revisiting them may also be helpful to get a refresher on some of the workflow and terminology.

- **Show basic tooltips/enhanced tooltips/video clips**. The display and types of tooltips are discussed earlier in this chapter.

- **Increase buttons on Command Ribbon by 2x**. This shows the relative size difference when using the Increase 2X setting. Notice other text in the interface is also increased, including the PathFinder text.
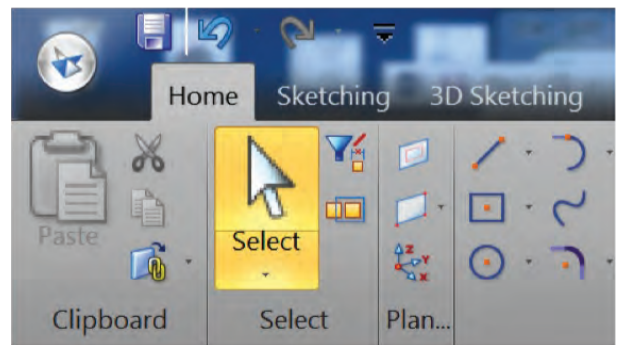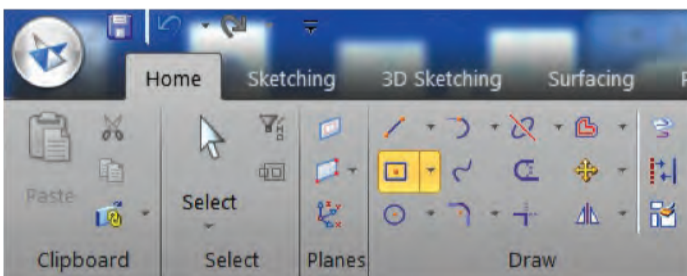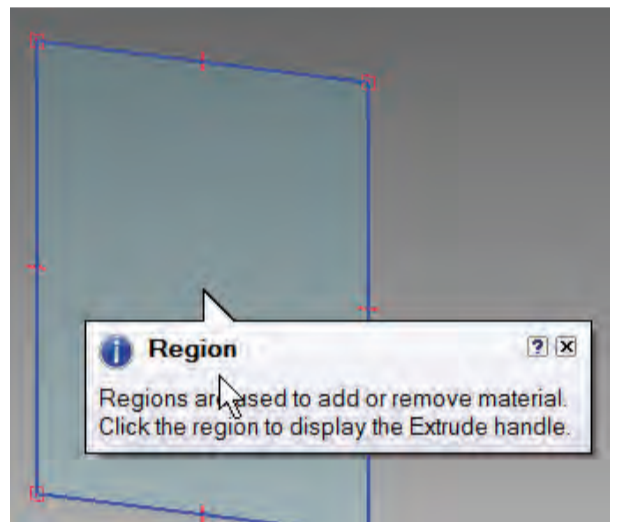
- **Use horizontal toolbar form/vertical docking window form**. This setting refers to the Command Bar, and is one of the items controlled by Themes. The horizontal setup is very compact, but primarily for experienced users. Because you'll have to use ToolTips to know what the various buttons are for. The vertical setup takes up a lot of space, but it contains text so you know what most of the buttons are for. Plus, the vertical setup is going to look more familiar to SolidWorks PropertyManager users. Below are both interfaces for the same Extrude function.

  Again we see the tradeoff between usable graphics space and convenience of knowing what the buttons do.
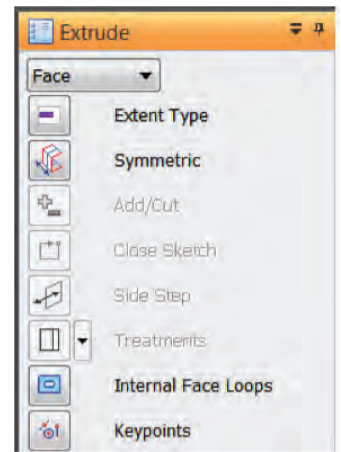
- **Radial Menus**. The radial menu can be accessed by holding the RMB down for longer than the designated period in milliseconds or by dragging the RMB in the direction of one of the commands on the radial menu disk (gesture). With touch interfaces, the gestures are useful. In more desktop oriented setups with mouse and keyboard, the RMB delay may be the preferred method.

- **Advanced Design Intent Panel**. Design Intent Panel was formerly called Live Rules. The basic Design Intent panel is shown here to the right. This is new with ST8, for those of you familiar with previous versions.

  The advanced Design Intent Panel is shown below, and can be rotated vertically. Using this Solid Edge Option.
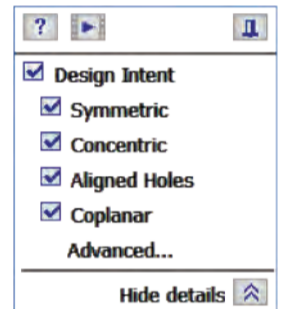
  This is another example of the choices Solid Edge gives users for a sleeker, icon-driven interface, or a more explanatory icon-and-text-driven interface.

- **Document Name Formula**. You can use available document properties to automatically build a filename for your new documents.

A video demonstration exists that goes along with the material in the chapter. Click on this link to view it.

Chapter 3 | Introduction to Solid Edge

Summary

The Solid Edge interface follows Microsoft standards and is highly customizable. You can apply themes to it to make it appropriate for various levels of user. The interface is also flexible enough to be useful on a variety of devices, although you will probably find that as a professional CAD product, it is best used on large monitors.

Chapter 4

Creating geometry

If you are familiar with creating geometry in a history-based CAD program, you won't have any trouble creating geometry using synchronous technology. Creating geometry in the two environments is very similar. It's the editing that holds the biggest differences. The one big difference with synchronous technology is that you will need to leave some history-based ideas behind. Sketches in synchronous are not associative to the features they create. Said another way, once you use a sketch to create a feature, the sketch can be used as reference, but it is no longer tied to the part geometry. This is an idea that a lot of history-based users have a hard time accepting. Just remember this – when it comes time to edit the part, you can edit the part directly, without worrying about steps, or order, or dependency. That is the trade-off.

In this chapter, you will learn about the features, tools and concepts that you can use while creating new geometry with synchronous technology. This chapter also includes a tutorial that steps you through creating a part from scratch.

Two video demonstrations are available for the material in this chapter. Click on the links for the Part 1 video and the Part 2 video.

Primitives

When you are no longer restricted by the constraints of associative sketches, you realize that for the most common shapes – boxes, cylinders and spheres – sketches really don't need to play any role at all. In fact, Solid Edge® software realizes this and provides you with the ability to create primitive geometry entirely without sketches. The three types of primitives you can create in synchronous technology are boxes, cylinders and spheres.

Box

The Box command is located on the Home tab, in the Solids group. The Box dropdown list also contains the Cylinder and the Sphere commands.

You can create a basic box with three clicks. As soon as you select the Box command, the software (or Solid Edge) displays crosshairs on the screen, as if you were sketching. To change the orientation of the crosshairs, roll the mouse over a reference plane or planar model face. To lock the crosshairs to a particular orientation, press F3 when the lock symbol appears next to the cursor.

To use the Box command:

1. From the **Home** tab -> **Solids** group, click the **Box** command.

2. (Optional) Change settings.

3. To create the first corner, click once.

4. To create the diagonal corner of the first face, click again or type values for length and width.

5. To define the length of the box, drag and click or type a value.
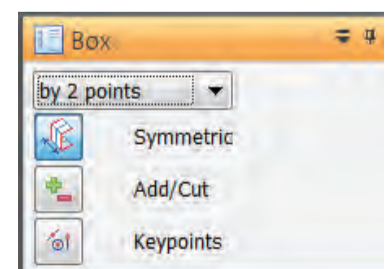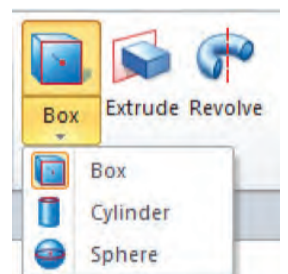
The Box command offers several options:

- Holding down the Shift key between the first and second click creates a box with two equal sides. The sides remain equal through changes.

- The initial rectangle can be created in three ways:

  - By center – Click center of rectangle, drag half diagonal.

  - 2 points (default) – Click one corner, click diagonal corner.

  - 3 points – Click first corner, click adjacent corner to establish direction of sides, click third corner.

- Symmetric – When dragging the depth of the box, the Symmetric option adds (or removes) material equally on both sides of the original plane. When this option is OFF, material is only added or removed on one side of the original rectangle.

- Add/Cut – Solid Edge will generally figure out what you are trying to do, but sometimes you need to specify your intent. The options are: Automatic (default), Add and Cut. You can toggle this setting using the Spacebar.

- Keypoints – Keypoints are very important in Solid Edge, and especially on the synchronous technology side. Keypoints enable you to establish direction, distance, or location by selecting an existing entity that might be the end of a sketch line, the center of a hole, the midpoint of a line, or others.

Solid Edge has four shortcuts for invoking keypoints, which can be easily remembered using the acronym MICE:

M = Midpoint

I = Intersection

C = Center

E = Endpoint

When the sketch cursor is over an entity such as an edge, press M, and then click the mouse to start or end the sketch entity at the midpoint.

Cylinder

The Cylinder command is located on the Box dropdown list. The Cylinder command works just like the Box command, except that the only way to draw the original circle is to click the center, then click the radius.

Sphere

The Sphere command is similar to the Box and Cylinder commands. To create a sphere, you simply click the center point, and a point to indicate the radius, or type a value.

If you don't like the placement or orientation of any of these primitives, you can use the Steering Wheel to move, resize or rotate them. The Steering Wheel is discussed in Chapter 1.

Sketching and sketches

Although the actual act of drawing sketches in Solid Edge is probably going to be very familiar to you, sketches fit into the Solid Edge process differently than in other history-based programs you may have used. We have already seen that sketches are treated differently in synchronous than they are in ordered workflows.

Besides sketching, in this section we will discuss how you can use sketches differently in synchronous and ordered. Since synchronous has no history, a collection of sketches in synchronous behave differently than a collection of ordered sketches. With ordered sketches, you can only make references to earlier sketches from later sketches, but in synchronous, every sketch can reference all the other sketches. Synchronous sketches are like keeping all your sketch information in one big 3D sketch, where there is no concept of order. You can even use synchronous sketches to drive ordered features. This can be very beneficial when you need to have several features referencing one another.

Sketch Organization in Synchronous

Solid Edge synchronous technology sketches are handled somewhat differently from SolidWorks sketches. In SolidWorks, you open a new sketch, and stay in that sketch until you close it. Then you open a new sketch, and stay in it until you close it. You may go back to edit the first sketch. These sketches can be on the same plane, and they have a history-based relation to one another.

In synchronous technology, any sketch entities that are on the same plane are automatically put into the same sketch feature or collector (entry in PathFinder that acts as a folder for items). If you start sketching on a different plane, Solid Edge creates a new sketch for that plane. If you go back and start adding items on the first sketch plane, Solid Edge adds those new items to the original sketch. Synchronous doesn't use the concept of closing a sketch. You simply turn off the sketch tool.

It's as if all sketch entities are in one big 3D sketch, organized by sketch plane. To create features, you can select any available sketch entities from any plane or any sketch. This analogy will help the SolidWorks user make sense of the synchronous sketching scheme.

When editing sketches in Synchronous, you can move from editing items in one sketch to editing items in a different sketch without closing or exiting a sketch. These concepts don't exist in synchronous. Synchronous sketches won't cause a feature to fail because it has too many or not enough or the wrong kind of entities in it. These are limitations or frustrations that SolidWorks users encounter on a daily basis.

Here is the workflow for synchronous sketching:

1. Select a sketch command (such as **Line**, **Rectangle**, or **Circle**).

2. Move the cursor over a plane or planar face.

3. Watch the orientation of the crosshairs – the sketch plane can shift dynamically if you are not actually sketching.

4. To lock the sketch plane, press F3 (or right-click the lock).

5. To start the sketch entity, for example a line, click and release.

6. To end the line and continue sketching, click and release.

7. To quit sketching the line, right-click or press Esc.

To switch sketch planes:

1. To make sure you're not actively sketching, right-click or press Esc.

2. To unlock the sketch plane, press F3.

3. Reorient the cursor over a different plane and watch the crosshairs change direction.

4. To re-lock the sketch plane, Press F3 (or right-click the lock).

The sketches are collected in an area of PathFinder, and can be reordered manually or sorted by name. Items you sketched at different times on the same sketch plane are stored in the same sketch in PathFinder.

IntelliSketch

IntelliSketch is the set of automatic relations assigned to sketch entities as you sketch in Solid Edge. The IntelliSketch options are located on the Sketching tab, in the IntelliSketch group. You can turn the options on or off as you sketch, to help you add relations to your sketches automatically.

IntelliSketch icons attach to your cursor as you move it around the screen. In the image, the symbol on the lower right shows that the cursor is picking up an endpoint, and the tooltip text confirms this.

Sketch tools

Relate – IntelliSketch automatically applies sketch relations as you sketch, but sometimes you need to apply relations manually, and that's when you use the Relate tools. The Relate tools require a special workflow:
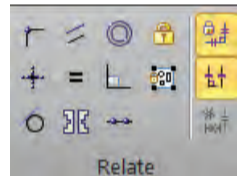
• Select the Relate tool you want to use to create a sketch relation.

• Select the sketch entity that you want to move first.

• Select the sketch entity that you want to remain stationary second.

• You can continue to assign relations until you press another tool button or Esc.

Some tools only require selection of one entity, such as the Horizontal/Vertical tool – just select a line, and it will snap to whichever state is closest – horizontal or vertical.

You can also select two endpoints, which will become horizontal or vertical from one another, such that the one you selected first will move while the second endpoint will remain stationary.
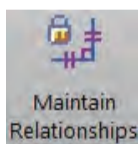
Maintain Relationships – Turning this setting off turns off the automatic application of sketch relationships. Existing relationships are still valid, but new automatic relationships are not created.
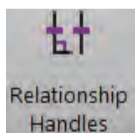
Relationship Handles – This option turns on or off the display of relationship symbols on the sketch entities.
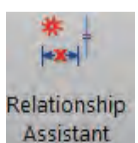
Relationship Assistant – This is only available when the sketch plane is locked. Use it on a selection of underdefined sketch elements. The Relationship Assistant applies IntelliSketch relations and dimensions. When working with this tool, be sure to monitor the Prompt Bar so that you are aware of what Solid Edge is expecting you to select next.

Lock – The Lock tool enables you to lock a keypoint or an entire sketch element in place. If you lock:

The entire element, none of the keypoints on that element can move.<bullet>A keypoint, other keypoints may move. So a line with one end locked may rotate or lengthen.

Rigid Set – A rigid set acts as a single block.
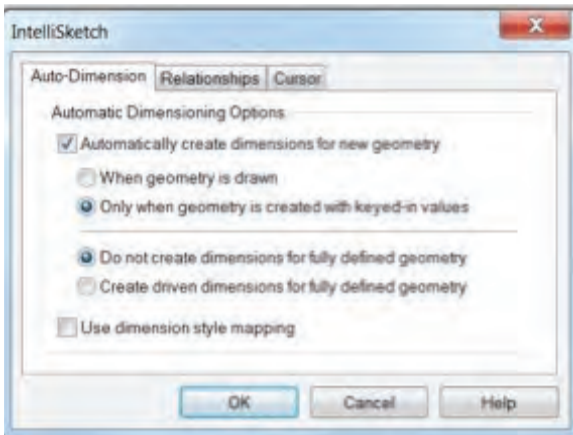
IntelliSketch Options – In the IntelliSketch dialog box you can define which IntelliSketch relations the software adds automatically.
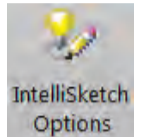
The IntelliSketch Auto-Dimension function gives you a high degree of automation for adding dimensions to sketches. You can use this together with typed-in values to help define new sketches quickly and accurately.

Relationship Colors – Many users making the change from SolidWorks to Solid Edge are very concerned about sketch colors reflecting the fully defined state of the sketch. The Relationship Colors tool does this, and is available in ordered mode sketches.
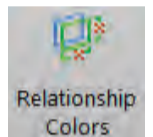
Note: What constitutes "fully defined" is not the same in Solid Edge and SolidWorks. In SolidWorks, for example, if a line is parallel to the X-axis and dimensioned at a distance from it, the line is displayed as fully defined (black). In Solid Edge, the line must also have one endpoint located along the X-axis and have a designated length to be considered as fully defined. Also, the dimensions in Solid Edge must be locked (red). SolidWorks does not have an equivalent for unlocked (blue) dimensions. SolidWorks allows lines that it determines to be fully defined to have open degrees of freedom and move around. Solid Edge does not. In other words, SolidWorks allows the state of the endpoints to be different from that of the rest of the line, such that the endpoints can be underdefined while the rest of the line is fully defined. In Solid Edge, the entire line, including the endpoints, is either fully defined or not.

Features

Solid Edge has two types of synchronous features: synchronous and procedural. The synchronous feature in PathFinder acts as a collector for the faces associated with that feature. Any dimensions associated with the feature will be PMI, attached to the 3D model.

The synchronous features you find in Solid Edge will be of no surprise to history-based users coming from other CAD packages. Both synchronous and ordered modes have extrude, revolve, sweep, loft, and so on.

There are some differences, however, in the synchronous way of doing things. Obviously, editing features are different, but in this section, we are only addressing creation of new features.
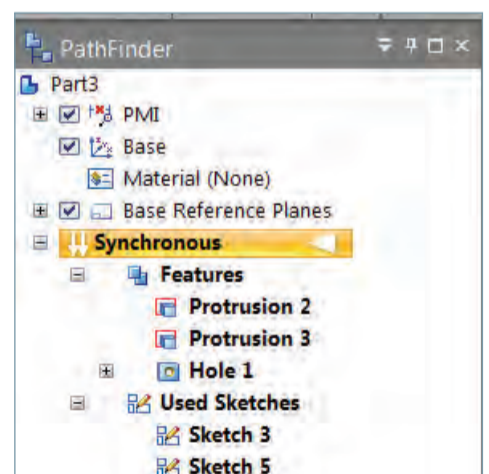
The biggest difference to which history-based users will have to adjust is that the sketches are not used to drive the features. Once the feature is made, the sketch is really just for reference, and gets stored in a collector folder called Used Sketches. To use the sketch again, right-click it and select Restore.

Note: The workflow for synchronous extrude and revolve features is generally:

1. Create a sketch with a closed loop.
2. From the ribbon, select an operation.
3. Set options and distance/angle.

You could also do this:

1. Create a sketch.
2. Press Esc.
3. Select the region (inside closed-loop sketch).
4. From the CommandBar, select an operation.
5. Set options and distance/angle.

Synchronous features are listed in PathFinder, and for simple feature types, such as extrude and revolve, the listed feature is a collector for all the faces associated with that feature.

Although features are listed in PathFinder in order of creation, you can reorder them however you want without affecting the geometry in any way. You can sort features alphabetically or by type. If you click on the name of a feature, the faces associated with that feature will highlight in the graphics area. If you delete a feature in PathFinder, Solid Edge deletes from the model all the faces associated with that feature.

Procedural Features

Procedural features work more like history-based features in that they have certain non-geometrical parameters associated with them, such as the number of instances in a pattern.

The types of procedural features in Solid Edge are:

- Patterns
- Rounds
- Chamfers
- Holes
- Thin wall (shell)
- Rib
- Vent
- Lip
- Sheet metal features

Double-clicking a procedural feature displays parameters that drive that feature. But this has to do with editing, which we will discuss in Chapter 7.

Tutorial – Creating features

In this tutorial, we'll walk you through the creation of the part shown on the right. We'll talk about the process and some related facts. In addition, we'll discuss the decision making process, so you learn why we made certain decisions when creating the part.

Base feature

In synchronous, the base feature, or the first feature you create, doesn't matter as much as it does in history-based modeling. In fact, it doesn't really matter at all to the rest of the model. The determining factor here is simply convenience. You could make this part in a number of ways. I'm going to choose a combination of revolved and extruded features. In the case where you have shape on the front and back, revolved features are easier because you don't have to do protrusions and cuts. Plus, you just sketch it once, revolve the whole thing, and you're done.
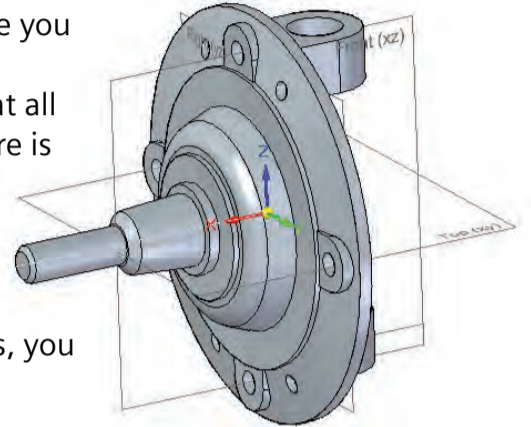
Create the sketch
1. Open a new part using a template with ANSI mm units.

2. Choose **Home** tab -> **Draw** group -> **Line** command.

3. Place the cursor over the edge of the displayed Front (xz) plane, and press F3.

   Pressing F3 (or clicking the lock icon) locks the current sketch to the Front plane.

   Depending on your settings, your display may swivel around, indicating that the view is normal to the sketch plane. The setting that controls this behavior is in the **Application** menu -> **Solid Edge Options** -> **General** -> When entering **Profile/Sketch** -> **Orient** the window to the selected plane.

4. Starting from the origin, create a sketch similar to the one shown at the right.

   The sketch does not have to be exact. Only a few dimensions are placed on it so that you just get some practice with sketching without getting caught up in dimensioning.

   Tips:
   a. You can type in values as you sketch.

   b. Sketching is click (release), click (release).

   c. To end sketching, do any one of the following:
      i. Complete a loop.
      ii. Press Esc.
      iii. Right-click.
      iv. Choose the Sketch command again.

   d. If relationship handles (small red symbols) are not displayed, you can turn them on from the **Sketching** tab -> **Relate** group -> **Relationship Handles**.

   e. To get the tangent arc, once you have finished a line on one side, press A, and Solid Edge will transition to the Tangent Arc command.
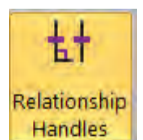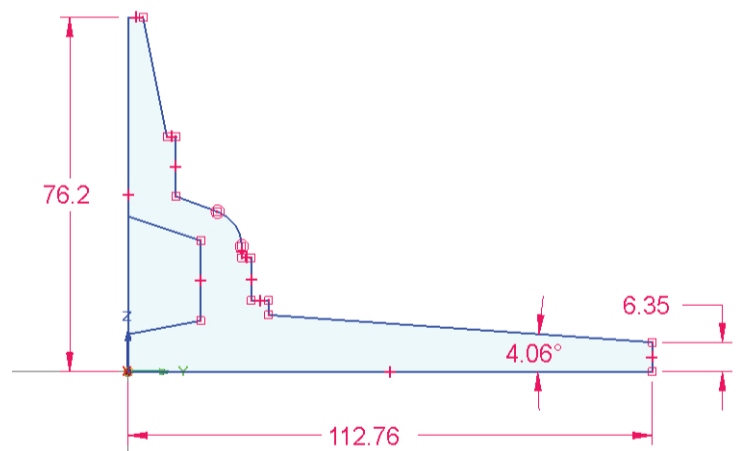
f.   Changing the sketch will not change the solid.

g.   The solid can be edited by changing dimension values.

h.   Red dimensions are locked, and you can change them. Blue dimensions are unlocked, and you, or the system, can change them. Purple dimensions are for reference, driven by various sources.

i.   The cursor gives you IntelliSketch feedback, to indicate when you can pick up a relationship to the end of another line, vertical, or tangent from another entity.
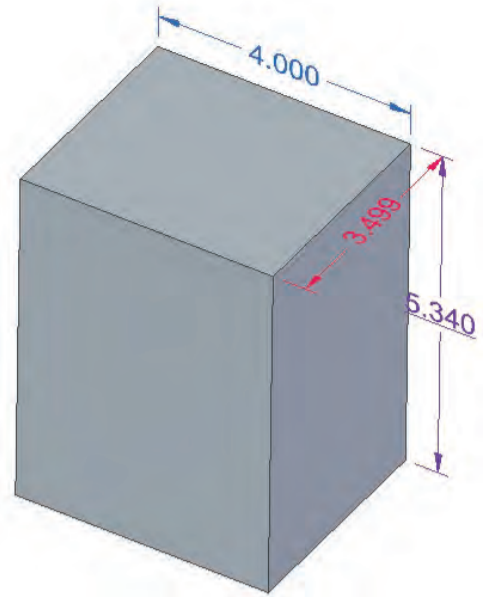
Create a revolved extrusion

1.   When the sketch is complete to your satisfaction and shades an internal area as shown above, click the Revolve command.

2.   Place the cursor over the shaded area that you want to revolve into solid geometry. The shaded area turns orange, as shown.

3.   Click inside the shaded area. The shaded area turns green.

4.   Right-click to accept this selection.
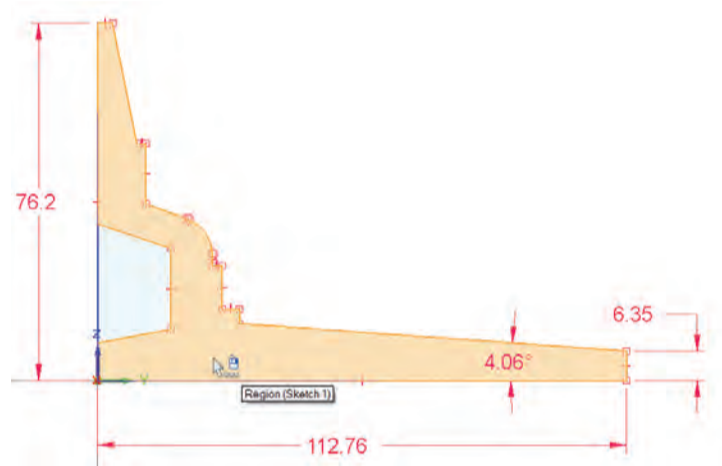
> *Note: Monitor the PromptBar window for messages telling you what the software is expecting you to do next. PromptBar may be in different locations around your graphics window, depending on the theme you selected or your interface settings.*
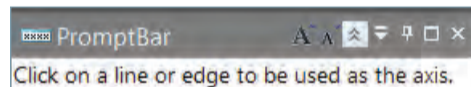
5.   Click the 112.76 mm dimension, as indicated in the PromptBar window.

6.   Type in 360 to make the revolution go all the way around. Or you can use the Extent Type option in the CommandBar to specify 360 degrees.

7.   To complete the revolve feature, press Enter.

Note: Observe what happened in PathFinder:

a.   Solid Edge created new collector folders:
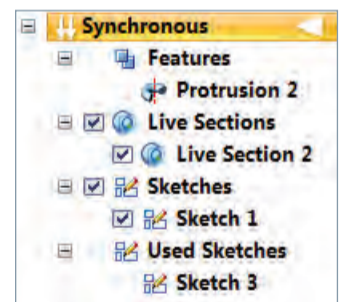
Features – Contains the new Protrusion revolved feature. In Solid Edge, a Protrusion adds material and a Cut removes it. The features listed in this folder can be sorted alphabetically or by type. The order does not matter. In Solid Edge, there are two types of features: synchronous and procedural. A revolve is a synchronous feature.

Used Sketches – Contains sketches that were used to create features. These no longer drive the feature and are just used for reference. You can keep them or delete them.

Sketches – Contains sketches that were not used to create any geometry. Solid Edge accounts synchronous sketches differently than SolidWorks. The rules for the sketch are less rigid in Solid Edge.

Live Sections – Contains sketches that take a cross-section of new revolved features through the plane of the revolved sketch. You can use such a sketch to drive the 3D geometry. You can put dimensions on this sketch and change the solid.
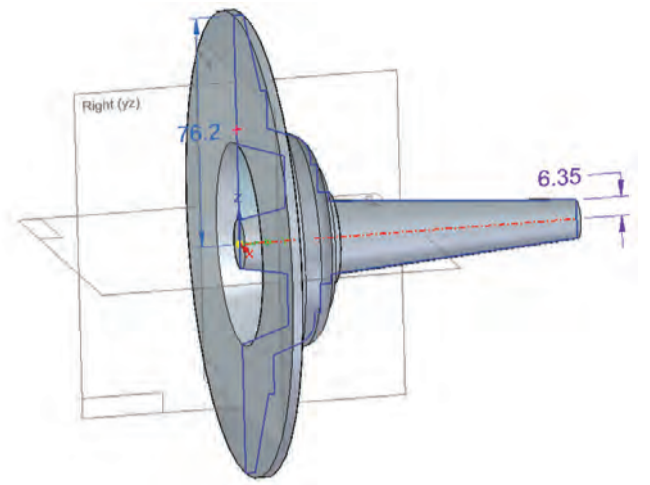
You can turn off the display of Live Sections and Sketches by clearing the check box to the left of each folder.

b. The sketch dimensions are no longer displayed. The sketch dimensions were red, but the remaining dimensions are blue and purple. You cannot change the purple 6.35 dimension, but you can change the blue 76.2 dimension by dragging the Live Section. The system can also change the blue dimension.

Simplify the view

1. Turn off PMI, Live Sections and Sketches folders by clearing the check boxes located to the left of each folder.

2. If the planes and dimensions are in your way, clear the check boxes for the associated folders: Base Reference Planes and PMI, respectively.

Create a rectangle

1. Choose the **Rectangle By Center** command, move the cursor over the plane lock, as shown, and press F3.

2. If the sketch plane does not reorient automatically, click the **Front** face of the **Quick View Cube**.

3. Position the cursor over the disc aligned vertically from the center, as shown. A dotted line shows that the cursor is lined up with the origin.
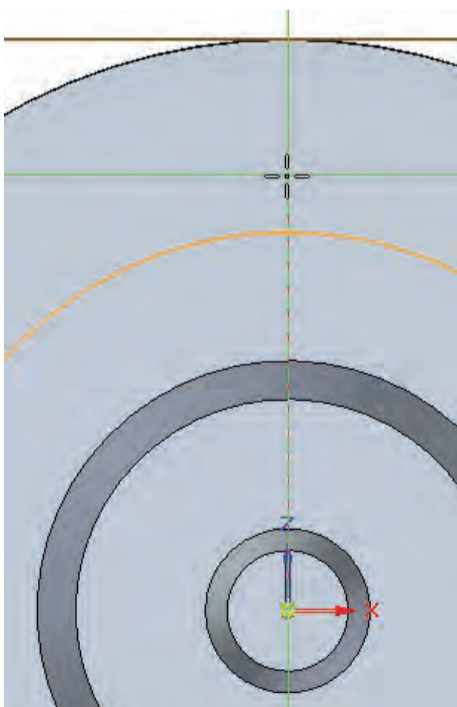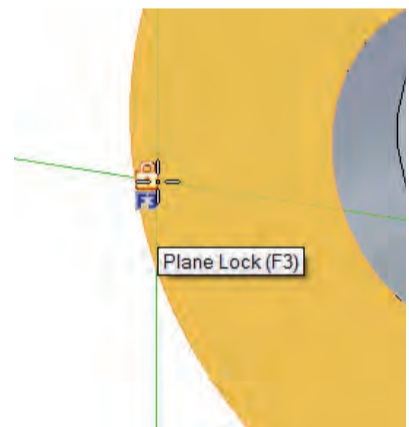
4. Click the center of the rectangle, and then click a corner point to create a rectangle approximately as shown below. The rectangle is centered across the reference plane.

Remember, sketching in Solid Edge is click (release), click (release). If you drag the cursor (holding the left mouse button down), you will create a different type of rectangle that won't be centered on the reference plane.
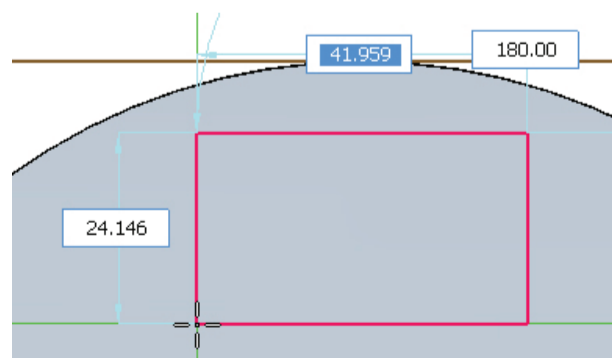
5. Add a **Vertical** relation ⊹ between the midpoint of the line and the part origin.

> *Note: This step isn't really necessary, because if you created the rectangle such that it is symmetrical about the vertical axis, and the solid is created from that, then the Design Intent of synchronous technology will understand this symmetry and maintain it. Design Intent, however, doesn't work with sketches, only with solids. Also I want to show some manual application of sketch relations.*
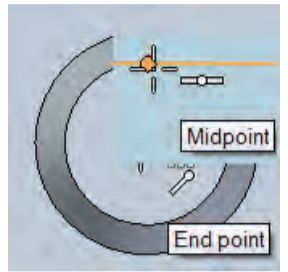
> *So to add the sketch relation, remember Action -> Object. Click the Horizontal/Vertical command (Home -> Relate -> Horizontal/Vertical), then click the midpoint of the lower horizontal line of the rectangle, and finally, click the part origin. Remember that IntelliSketch adds visual feedback to the cursor to help identify what is being selected.*

6. To quit the Horizontal/Vertical command, press **Esc**.

7. It's best to be in a view other than looking square onto the sketch plane. To rotate the view, you can drag the middle mouse button or click one of the corners of the Quick View Cube.

Extrude the rectangle
1. Click inside the rectangle you just sketched.

   The rectangle area turns green and the basic Steering Wheel appears.

2. Position the cursor over the extrude handle (the arrow that comes out of the solid) and move the cursor from side to side.

   The feature draws dynamically as you move the cursor.

3. To create the feature, either click a second time at your intended length, or type in a value.
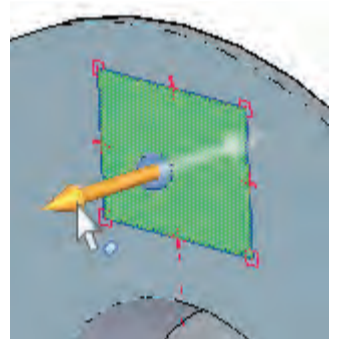
   You created an Extrude feature without invoking the Extrude command.

Round the rectangle edges
1. Choose **Home** tab -> **Solids** group -> **Round** (and then select the two edges as shown).

2. Type in 5 and then press Enter.

3. Use the scroll wheel on the mouse to gradually increase or decrease the radius of the fillet.
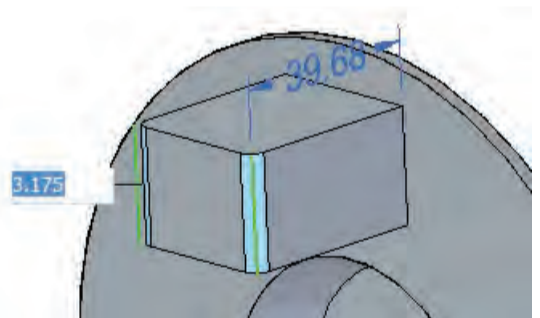
   This mouse wheel setting is available in **Solid Edge Options** -> **General**, above IntelliSketch options, and is named Enable value changes using the mouse wheel.

   > *Note: Solid Edge displays the dimension units in the value fields because the option, Show units in value fields, is set on the Solid Edge Options -> General page.*

Create a circle

1. Choose **Home** tab -> **Draw** group -> **Circle by Center Point**.

2. Hover over the orange face, as shown, and lock the sketch plane by pressing F3.

   Optionally, you can click on the appropriate face of the Quick View Cube to rotate the view normal to the selected face. The setting for making this rotation automatically was discussed earlier, and is found at **Solid Edge Options** -> **General** -> When entering **Profile/Sketch** -> **Orient** the window to the selected plane.

Create a hole

In this procedure, you align the hole on the midplane of the feature. You can align the hole in the same way you centered the sketch for the rectangle.
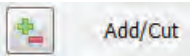
1. Move the cursor over the origin, and then move to approximately the location shown.

2. Sketch the circle by clicking the center and clicking a point on the radius.
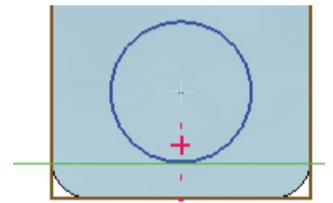
   You should have a dashed line from the center of the circle to the origin.

   Don't worry about dimensions just yet – you can be more precise later. In this chapter we focus on creating geometry.

3. Set the view to a 3D orientation, select inside the circle, and drag the arrow that goes into the material.

   Solid Edge guesses if you want to add or remove material, and you can correct the software using the Add/Cut button on the command bar.
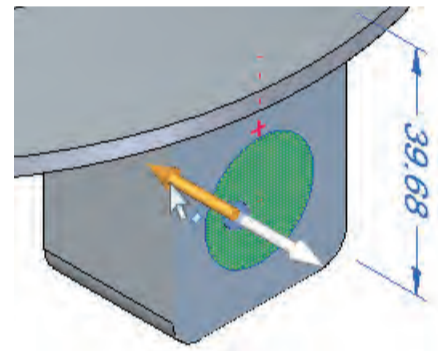
4. Make sure the hole goes all the way through the solid.

Mirror elements

In this procedure, you mirror the boss, round, and cut you made.

1. Reorient the view so you can see the whole feature, and without any tools selected, drag a selection box around the feature.

   The result should look as shown.

2. Choose **Home** tab -> **Pattern** group -> **Mirror** .

   If you're not sure what to do next, read the prompt in the PromptBar.

3. Move the cursor over the origin.

   The three base planes are highlighted, even if they are not shown.
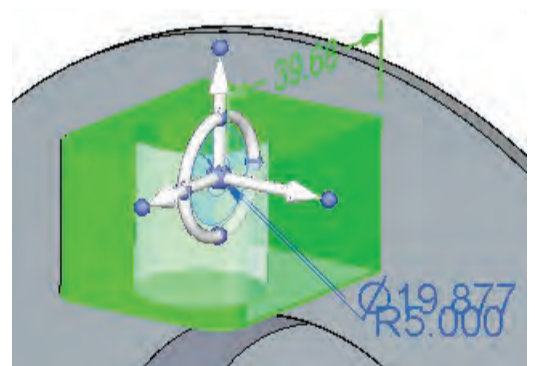
4. Select the appropriate plane to mirror the selected feature, as shown.

    Solid Edge displays the new feature in blue, while the selected feature remains green. If you move the cursor over other planes or flat faces, the blue mirrored feature may jump around.
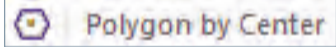
5. Once it is displayed where you want it, click the plane that gives you the correct result.

6. To clear the display, press Esc.

    You mirrored a bunch of geometry that you just selected from the screen, without worrying about what feature listed in the tree made that geometry. It just happens that our selection of faces lined up with three features that we created, but it didn't have to work that way.
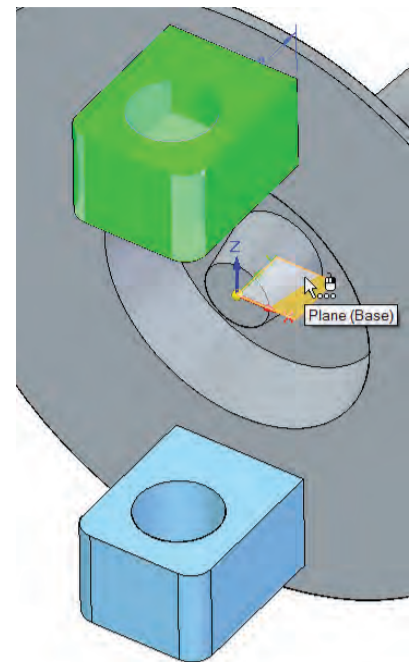
Add a hexagonal section
Before you can add a hexagonal section, you must extend a circular area and eliminate a round.

1. Drag a box to select just the two green faces, as shown.

2. Select the arrow pointing along the axis of the cone, and move your cursor toward the small end of the cone.

3. Type in 15 mm to extend the feature 15 mm from its original location.

4. Press Enter, then Esc.

5. Click the dropdown arrow next to the Rectangle by Center command and select the Polygon by Center command [Polygon by Center] .

6. Lock the sketch plane as shown.

7. Move the cursor to the center of the circular edges, click at the +, and look on the CommandBar to make sure the number of sides is set to 6.

8. Type in 19 for the included radius, and 0 for the angle.

9. Press Enter and Esc.

10. Click inside the hexagon and then drag the arrow up.

11. Type in 15 and press Enter.
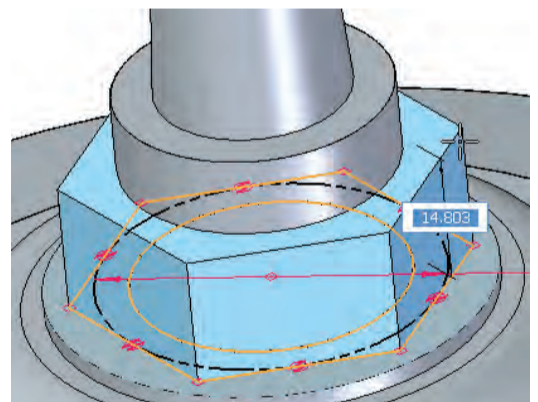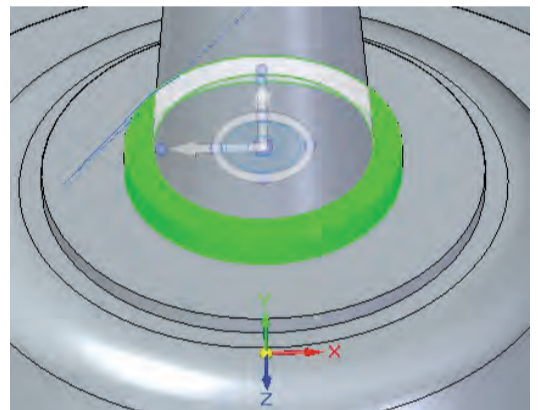
A construction circle with a Ø38 dimension is left over.
You can either delete or turn off the sketch with the check mark in PathFinder, in the Sketches folder. The brown/orange color of the dimension means that is has lost its reference.

Making geometry is making geometry, and aside from a couple of small differences, everything looks fairly familiar to history-based users, right?

Create dimensions

Well, now the fun begins!

1. Choose **Home** tab -> **Dimension** group -> **Smart Dimension**.

2. Place the dimension on the edge of the newly created mirrored feature, as shown. Use the middle arrow setting, and change the value to 40 mm.

3. Click the lock symbol.

   Locking the dimension changes the dimension to red and ensures that the only way that value can change is if you change it.

   Note the following:

1. We applied a dimension to what history-based users would call a child feature, but when we changed it, the parent feature changed in size. Lesson learned? Parent/child dependencies don't matter in synchronous.

2. The arrows at the bottom of the dimension value box determine where the dimensional change will apply: to the left, to the right, or symmetrically. Lesson learned? Synchronous does design intent on the fly. These arrows reflect the red 3D arrows on the dimension.

4. If you forgot to use the center arrow setting, Solid Edge still applied the dimension symmetrically, using the symmetry design intent. Using the Smart Dimension command, make the block stand up off the disk by 45 mm.

5. Click the change arrow away from the main disk and then lock the dimension.

   Note some additional functionality:

   • This time the arrows mean something. The correct arrow to have selected is the one toward the free end of the block. Changing the dimension on the disk side of the block causes the disk to get thicker.

   • The dimension automatically accounts for the round, and goes to the virtual sharp.

   • The base plane closest to the current display determines the plane on which the dimension is displayed. If you delete the dimension, rotate the screen and recreate it, the dimension may appear on a different plane.
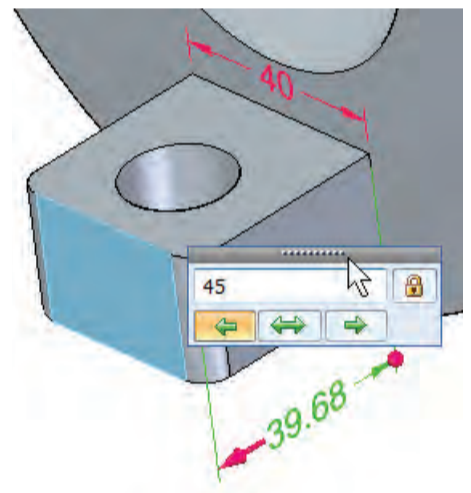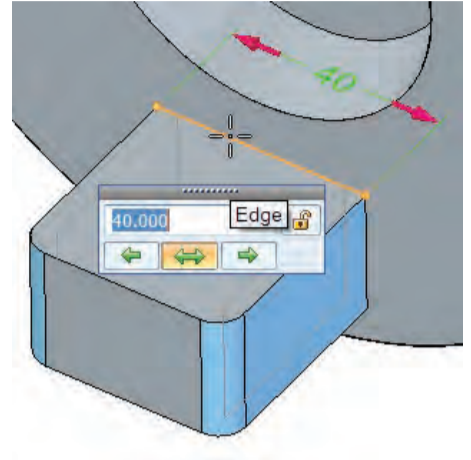
6. Change the thickness of the block to 15 mm.
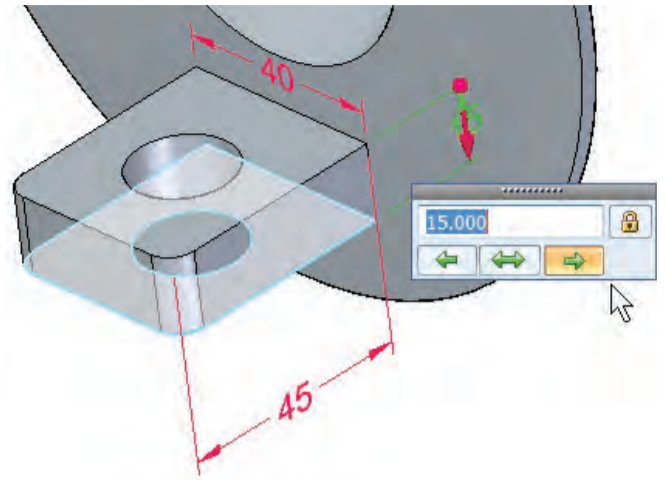
7. Dimension the hole diameter and the distance of the hole from the end of the block, as shown. Be sure to:

   a. Lock each dimension.

   b. Make the change arrows go the right way.
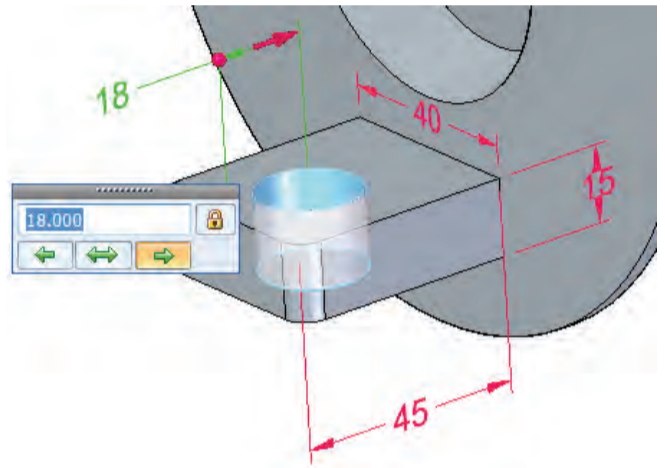
8. Dimension the block from the center of the part to be 45 mm, as shown.

   You can also dimension between the inside edges of the blocks.
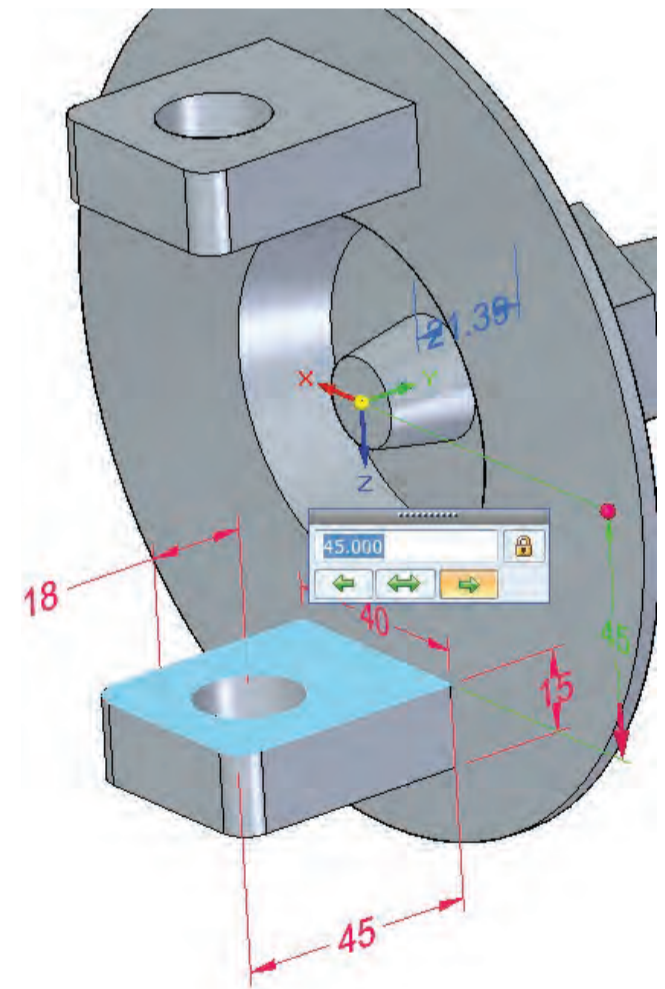
Create live sections

Using Live Sections, you can dimension the sketch just like you would any sketch, and it will drive the model. In synchronous models, live sections are of great help when working with revolved features.

1. In PathFinder, click the check box to the left of Live Sections.

2. Place a 150 mm diameter dimension on the disk edge, not on the Live Section.
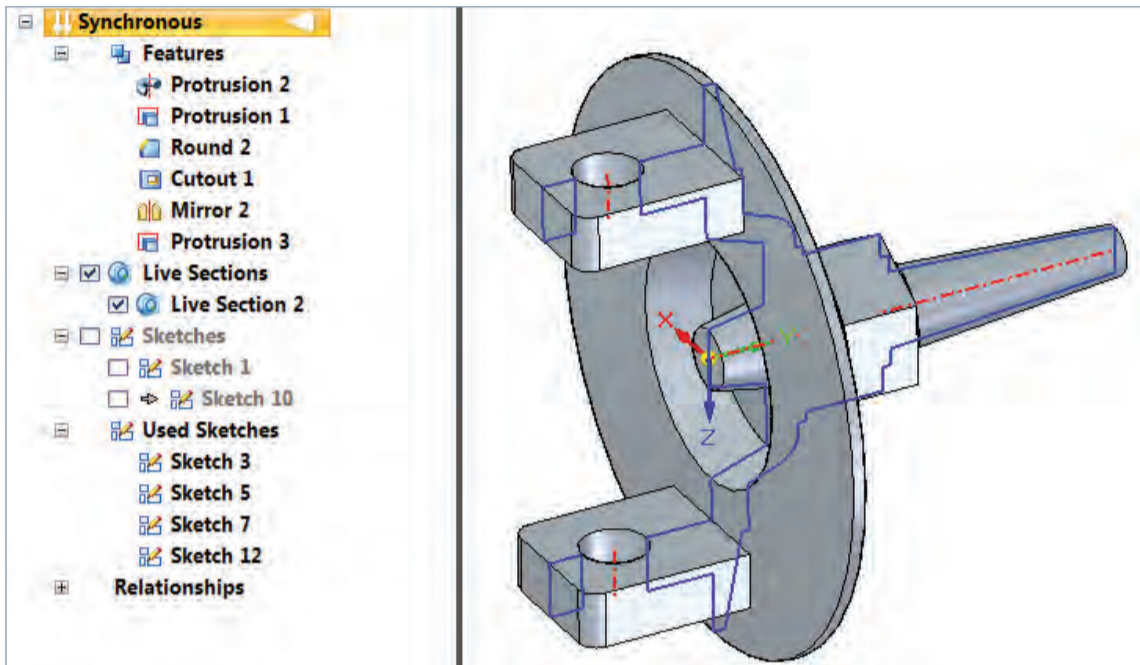
3. Place a 5 mm dimension on the edge thickness of the disk using the short line of the Live Section.

4. Make sure that both dimensions are locked, and the 5 mm dimension arrow is pointed toward the back of the part.

   Note:

   • To move a dimension, drag the dimension line.

   • To move the Ø150 dimension, drag the leader.

   • To select a dimension to delete it, select the leader or right-click the dimension and select Delete. Only select the dimension value to change it.
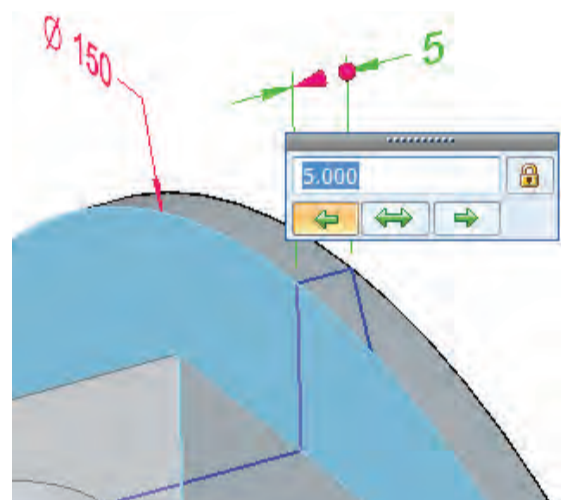
5. Place a dimension on the large end of the conical face and assign it to be 21 mm.

6. Choose **Home** tab -> **Dimension** group -> **Angle Between** .

7. Place a 92 degree dimension, as shown. Make sure the change arrow is correct, so that the side that is already angled will be the one that moves.

8. In PathFinder, navigate to **PMI** -> **Dimensions**, where are all of the dimensions used in this part.

9. At the bottom of the list, turn on all the dimensions you just created on the Live Section:

    a. Select the first dimension you created in Step 3.

    b. Shift-select the last dimension in the list.

    c. Right-click and select Group.
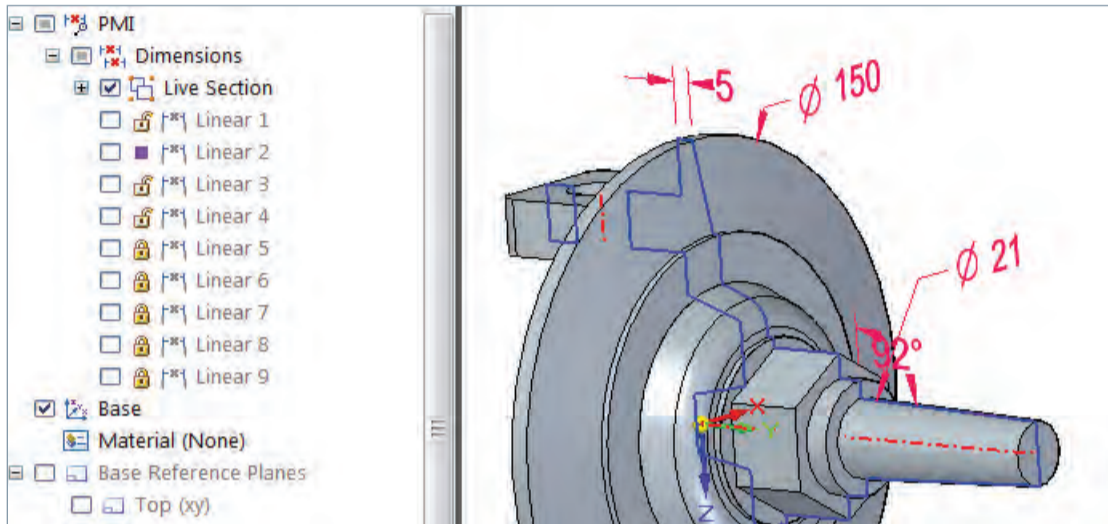
10. With the group collector selected, press F2 and rename it Live Section Dimensions.

    Now you can quickly turn this set of dimensions on or off as a group.

11. Save the file, sit back, and enjoy your handiwork. Try to change a few dimensions, and see how simple it is to drive a model through PMI dimensions directly attached to the solid geometry.

Two video demonstrations are available for the material in this chapter. Click on the links for the Part 1 video and the Part 2 video.

Summary

Synchronous technology offers tools for geometry creation that are familiar to history-based users, while giving more editing flexibility. Where most synchronous features simply contain geometry, procedural features also remember some metadata such as number of instances in a pattern, or type of hole. There is no better way to learn than by doing, so this chapter is dominated by a tutorial with various creation and editing steps.

Chapter 4 | Creating geometry

Chapter 5

Selection and re-use

In synchronous technology, you can change the size, position, or orientation of whatever you have selected. If you want to make a change, it doesn't matter how the geometry was created, or how or if it is grouped into features, it just matters which faces are selected. You could have faces created as portions of various features, different types of features, imported, native, faces inserted from a different model, and so on.

You can also save selections as part of a group, or user defined set, and each time you create new geometry, Solid Edge® software remembers the faces associated with that particular set. There is also a feature type called Procedural Feature, which works in some ways like history-based features, with metadata associated with the feature, but still without any associated order, sketch or dependencies.
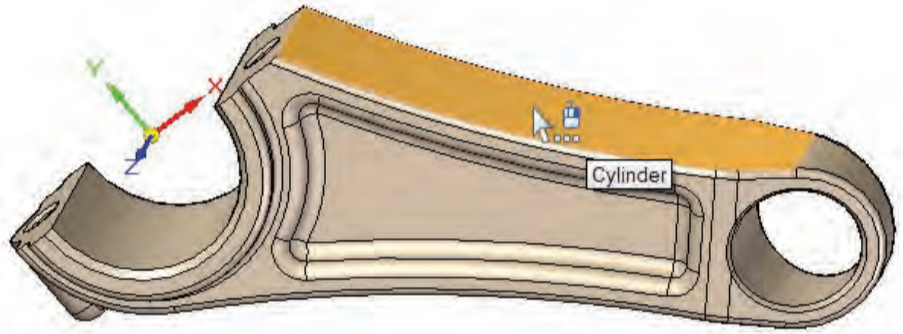
All of this talk of saved sets of faces can make one think of re-using, or copying sets of faces to be used in other places.

So for these reasons, this chapter lumps together the topics of selection, selection methods, saved and re-used selections.

Selection methods

Since the selection of faces is so integral to how synchronous technology works, we're going to spend some time talking about how to select what you want to change.

There are multiple methods for selecting faces:

1. Selecting individual faces
2. Using Ctrl-Select
3. Using Selection Manager
4. Fence/Window/Box selection
   a. Using the Overlapping setting
   b. Using the Selection Filter
5. Automatic addition to manual selection through Design Intent settings

Note: Sometimes, users who are new to synchronous want to be able to make changes using edges or vertices, since these are also recognized entities that can be selected. An edge is just the intersection of two faces, and a vertex is the intersection of three or more edges. So the common denominator in all of these is the face.

You cannot make changes in synchronous technology by using edges. Remember that even when you select edges to create dimensions, the dimensions are assigned to faces. Edges are commonly removed by rounds, chamfers, or other features, and lost reference entities tend to cause a lot of downstream problems when making edits to the geometry. Faces are more stable references.

Selecting individual faces

Selecting individual faces is easy enough. Just point and click. Right? Well, most of the time. But what if you have already selected something, and you want to deselect it or select something else?

Deselecting is also fairly simple, although it may not work the way you might anticipate. To deselect a face, you can press Esc, Ctrl-Select, or just select another face.

If a command is already active, and you are selecting at the prompting of the Command Bar, you may have to use the Deselect button in the Command Bar. This works mainly for selections such as edges or sketch entities in a command like the Sweep where multiple selections are often necessary. In general, however, if nothing else is active, Esc or selecting something else will deselect the current selection. There is also a Clear Selection option in the Select dropdown menu located at **Home** -> **Select**. Clicking on an already selected face will not deselect it unless you have the Selection Manager active, which we will discuss shortly.

Recognize also that there is a difference between a face being *highlighted* and a face being *selected*. Orange is the default color for highlighted, meaning your cursor is over it, and if you click right there, that is the face that will be selected. Green is the selection color.

Note: It is generally considered best practice to avoid standard interface colors when assigning part colors. Standard interface colors in Solid Edge include orange, green, light blue, and secondarily, any of the colors from the Design Intent interface.
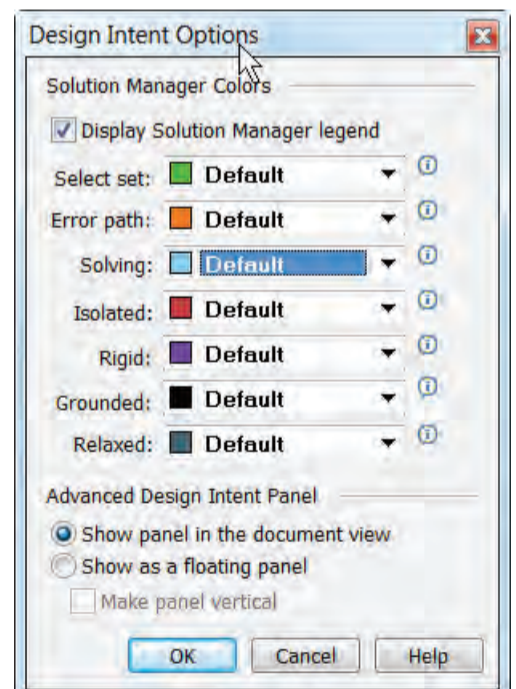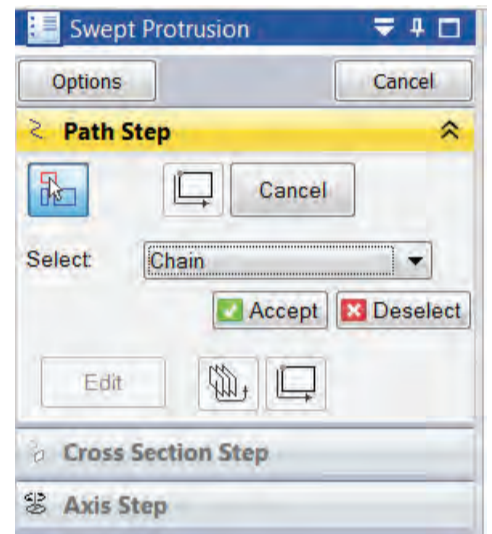
QuickPick

The QuickPick tool enables you to select hidden or overlapping faces, or other entities that may be more difficult to select directly. When you hover your mouse over a surface and delay for more than a second or two, the mouse icon with three dots will be added to the cursor. This is prompting you to right click for the QuickPick selection.

The QuickPick list box names each entity that is directly under your cursor at the time. This can include hidden faces (on the far side of the part), faces of internal holes, or other types of entities, such as features or bodies that a simple click might be too ambiguous to identify. As you move your cursor over each item in the list, it will highlight in the graphics window, and clicking the item in the QuickPick list will select the actual item.
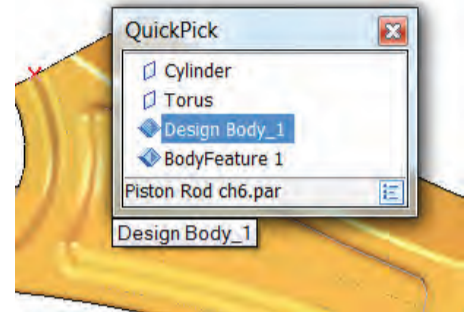
It should also be mentioned here that tool tips appear on the cursor and in the QuickPick list identifying the type of face that is about to be selected. You see that the Cylinder tool tip is displayed on the cursor in a previous image, and in the QuickPick it identifies that cylinder and a torus. The symbols used for QuickPick help you identify the type of entity listed.

QuickPick can help you select faces, edges, curves, planes, features, bodies, sketches, sketch entities, coordinate systems, parts in assemblies, and more.

Using Ctrl-Select

Ctrl-Select is another Windows standard method that you should be familiar with. However, Solid Edge has its own take on making multiple selections easier to use. After you use Ctrl-Select to pick the second face, you don't need to hold down the Ctrl key for subsequent selections. Solid Edge adds the plus and minus symbols to the cursor, clicking on unselected items adds them to the selection set, and clicking on selected items removes them from the selection set.

Using the Fence/Window/Box selection method

Many older CAD systems, particularly the 2D systems, work on a similar "select and edit" principle used in synchronous technology. Selecting multiple items is generally handled by a method alternately called "fence", "window", or "box" select. In Solid Edge, "box" select refers to a 3D volume, and "fence" select refers to a 2D rectangular area.

Fence selecting all faces within a rectangular area will select all faces within the selection fence regardless of depth into/out of the screen. So faces obscured by other faces, as long as they are completely within the rectangular selection fence will be selected. So the orange selection fence in the image below left results in the selection of hidden faces shown below right.
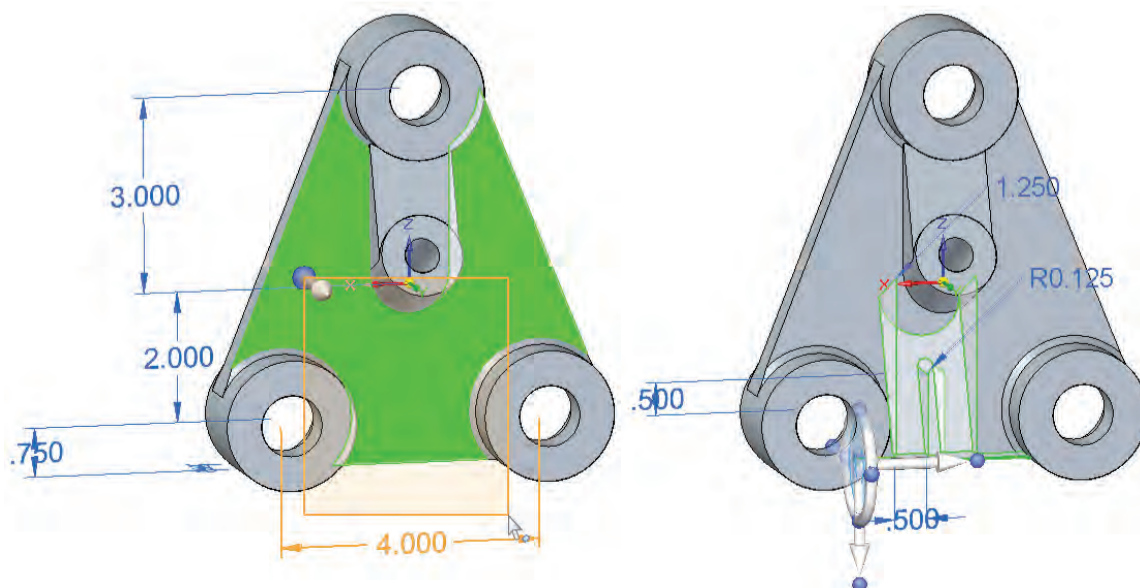
This concept and the application of it should be pretty straight forward, since all computer users make use of fence selections in applications outside of the CAD realm on a daily basis.

Techniques that you can use along with the fence select method involve pre-orienting the screen to help you get the desired selection set more easily. For example, the view orientation and selection fence shown to the right might make it easier to get the selection from the images above.
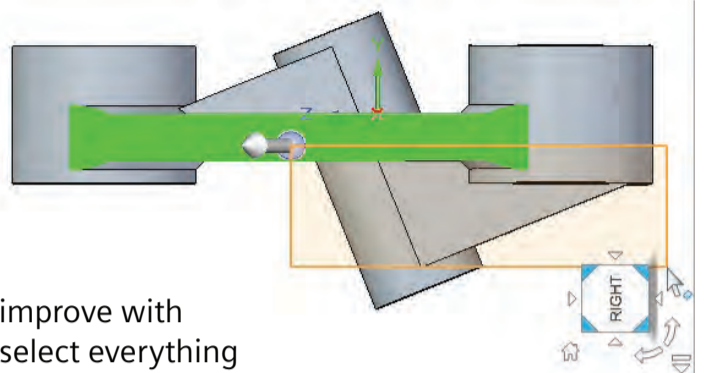
In any case, this is one of those things that will improve with practice. And remember that you don't have to select everything all at once. You can use Ctrl-selection as well as multiple fence selections to get the final selection set that you need for a particular edit. You may also find it easier to perform multiple edits from single selections. In the end, it's up to what you find easier in a given situation.

Using the Overlapping setting
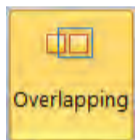
The Overlapping setting can be found on the Select group on the Home tab. Using this setting will select any face that is completely or partially within the selection fence. So the selection depicted in the screen shots below would result from using the Overlapping setting.
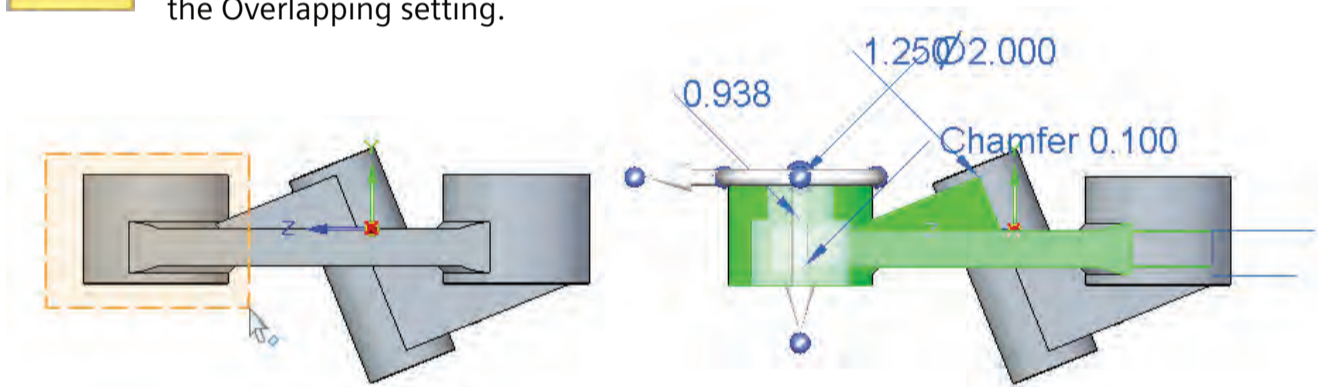
The Overlapping setting is turned off by default, and is not commonly used. Still, it's something that's available, and you should be aware of its existence in case it might come in handy.

Polygon Fence

In draft mode, you can use a polygon fence for more intricate selections, but this option is not available in part or assembly mode. A polygon fence is an irregularly shaped polygon that you would use to select the items inside or intersecting the polygon. To create one, I show the example of a sketch on a draft document. In the Select Command Bar, activate the Polygon Fence tool, then click near what you want to select. Each click creates a new corner of the polygon. Keep clicking to create corners until you have the items inside the polygon that you want to select, and everything else outside the polygon.
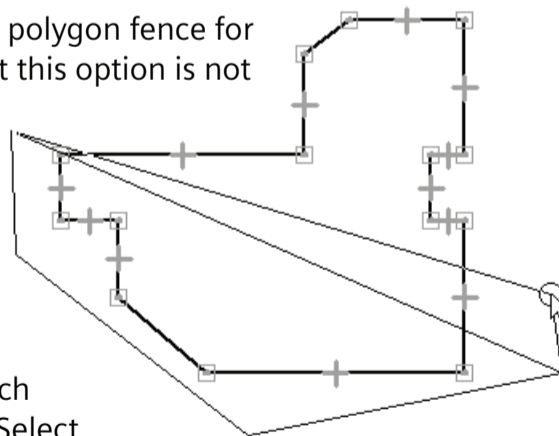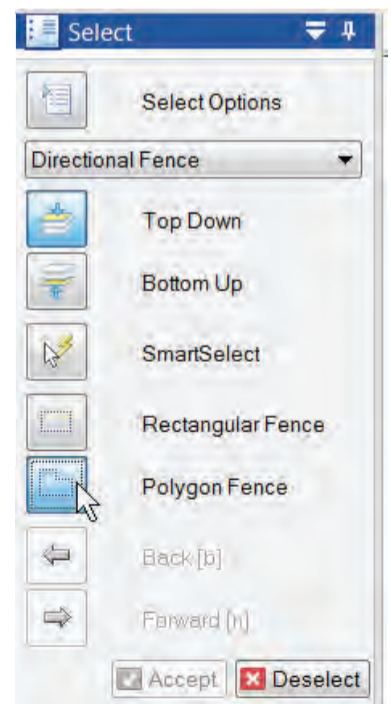
Using the Selection Manager

The workflow for using the Selection Manager has changed slightly within the last few years, so if you go searching for YouTube videos on the topic, you may find obsolete methods. You can access the Selection Manager from the Home tab, Select group, on the Select drop down list. Or if you prefer keyboard shortcuts, Shift+Spacebar will also get you there.

The Selection Manager enables you to make selections based on geometrical criteria, such as all concentric faces within a box, or all connected faces. It sounds very similar to the way Design Intent works, but Selection Manager requires more interaction from the user. Design Intent starts with a lot of automated rules turned on by default, where Selection Manager requires the user to actively select the rules.

The Selection Manager dialog.

> *During the development of synchronous technology and its related tools, two of the product managers half-jokingly referred to the Selection Manager and Design Intent as Finders and Keepers respectively.*
>
> *"If for example you want to select all coplanar faces prior to an action, you would use the Selection Manager – a "finder" operation. If you had already started an operation and wanted to exclude coplanar, you use Design Intent as a "keeper" operation." – Chris Dayton, Product Planner for synchronous technology*

After you have activated the Selection Manager and selected a face to start from, the Selection Manager dialog appears to help you use some automated decision making rules to select groups of faces.

One of the very attractive aspects of the Selection Manager is that you can limit the selection to faces that match the criteria and are within a 3D volume that you specify.

One way in which you might use the Selection Manager would be to turn off the Design Intent options, and simply pre-select all the faces you want to change using the criteria and tools in the Selection Manager. You can also use the Selection Manager to deselect faces, which might be important in more complex operations.

Overall, the Selection Manager is a tool you don't see demonstrated as much as Design Intent, but for some people, it might offer more security in that it enables you to pre-select the faces that the Design Intent/Live Rules would otherwise select for you automatically.
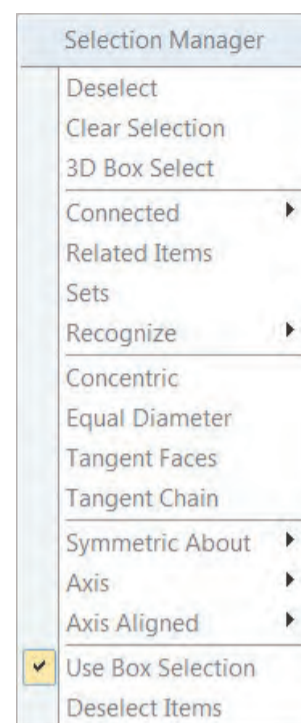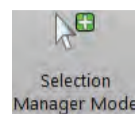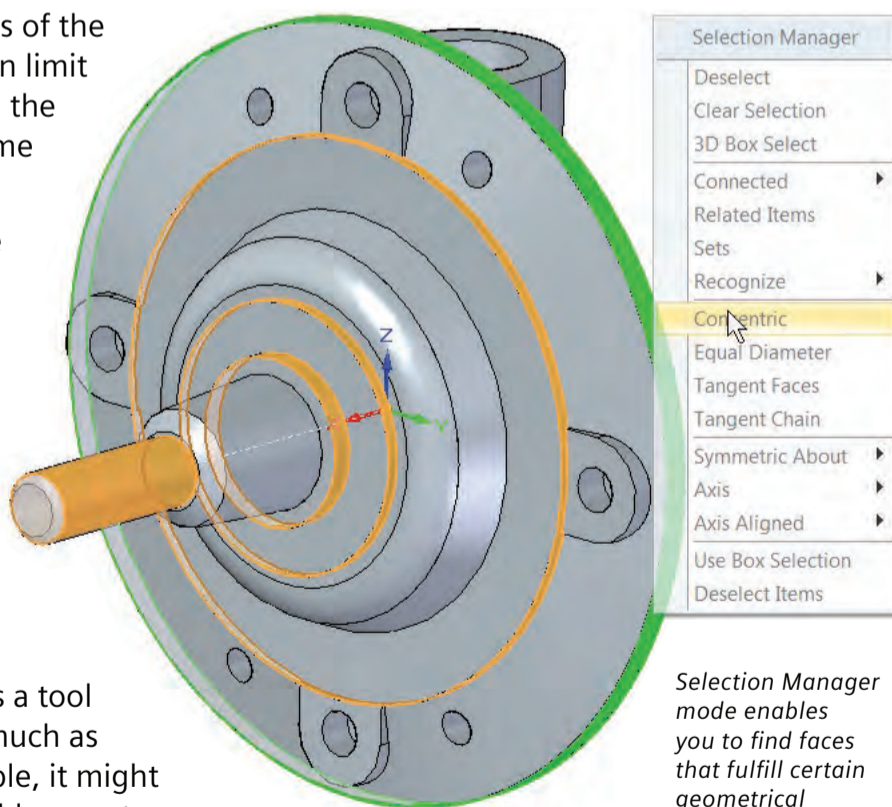
Selection Manager mode enables you to find faces that fulfill certain geometrical conditions.

In the image shown above, Selection Manager was invoked using Shift + Spacebar, and the green cylindrical surface was selected. This brought up the Selection Manager dialog, and the mouse hovered over Concentric, which highlighted the other concentric cylindrical faces in orange. To exit the Selection Manager and proceed with the operation, such as move or copy, press the Spacebar.

Re-using geometry in synchronous technology

When you are not limited by the way features are defined, you have much more flexibility to re-use existing geometry where the same geometry exists again. This is one of the key, underlying tenets of synchronous technology, so you may want to read that first sentence again. In standard history-based CAD, you have to worry about two things when re-using geometry: First, is all of the geometry you need part of a single feature, or would you need partial features? And second, is the geometry you want to re-use dependent upon something else?

> *When you are not limited by the way features are defined, you have much more flexibility to re-use existing geometry where the same geometry exists again.*

You really shouldn't have to worry about things like relationships in re-used geometry. You should just be able to see some geometry you want to copy, and copy it. Whether it goes in the same part, or in a different part, copy the faces from here to there, position them, and incorporate them into the model. Done.

Sometimes you may need to create a pattern that doesn't fit available tools nicely. You don't have regular recurring groups, or the pattern goes between parts. Or maybe you have a particular shape that you make frequently, like a port cavity that you might need to add to any sort of part at a particular time. You should be able to pull a cutout from a library without having to create some sort of dummy material for it to be cut from.

Let's show some examples of copying geometry, and also saving geometry into a library for later re-use.

Selections in assemblies

Selections in assemblies follow the main concepts outlined here, but also have more types of data to deal with. For that reason, selections in assemblies are handled in the Assemblies chapter rather than here.

Tutorial – Copying geometry

1. Open the Spindle.par part from the folder for this chapter. This part was built natively in Solid Edge ST8.

2. Box select the lug and the hole at the top, and then Ctrl-Select the hole to the right. You could probably get all of this in a single box select, but practicing different methods is a good thing. At the bottom of the screen, you should get a prompt that says "7 items are selected". The lug itself has 4 faces, the hole on the lug has 2, and the separate hole has 1, for a total of 7.
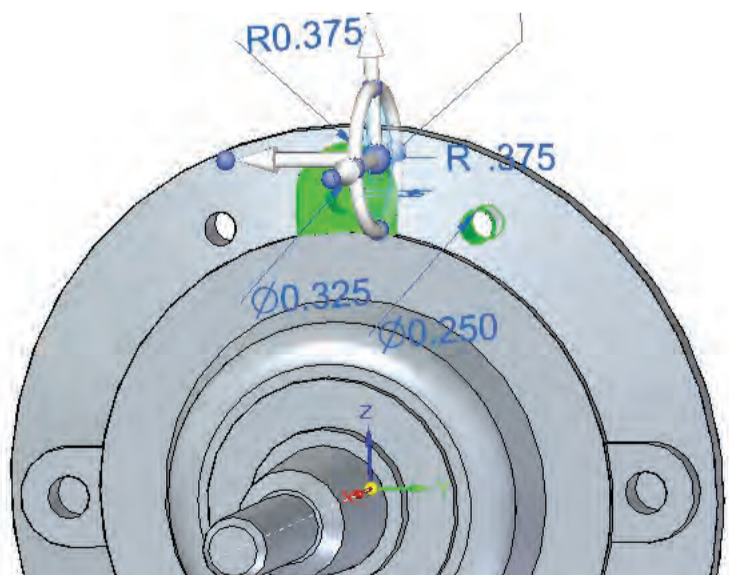
Make sure that you don't accidentally get any of the faces from the back side of the model.

3. Now drag the big blue ball (origin) of the Steering Wheel, and drop it on any of the circular edges that is concentric with the main axis of the part, so that the Steering Wheel torus goes around the axis. The part should now look like the image on the right.

    Notice we have selected what could be a portion of a feature. That hole to the right could be the "child" of a mirror or of a pattern, or part of some other situation. But we can make a copy of it without worrying about any of that.

4. Holding down the Ctrl key, click and drag clockwise on the Steering Wheel torus. Drop the left mouse button, release the Ctrl key, key in 45 to the value box, and press Enter, then Esc to deselect the newly copied faces.

    If Solid Edge opens up the Solution Manager at this point, just turn off the Design Intent options, and things should land in place. Don't forget to reinstate the Design Intent settings when you are done.

    If the resulting geometry shows up as blue rather than gray, it copied the faces as surfaces because you got an extra face, or landed the faces in a place where they could not reattach to the model, Press Ctrl-Z (undo) and try it again.

5. Next let's try something a little different. Click on the Selection Manager Mode tool, located on the Select drop-down list. Your cursor will get a green circle with a white plus sign on it. Next, click on the cylindrical face highlighted as shown in the image to the right.

6. From there, the Selection Manager appears. Select **Recognize -> Feature**, and the spindle will highlight green. Press the Spacebar to get out of Selection Manager mode, but still retain the selection.

7. Use the Quick View Cube to go to the Back view K (or use Ctrl-K). Use Shift-Left Arrow or the curved arrow on the Quick View Cube to rotate the view so that the spindle is point straight up.

    If necessary, move the Steering Wheel such that one arrow is pointing in the Z direction. Hold down the Ctrl key and tug the Steering Wheel arrow in the positive Z direction.

    Note: If you make a mistake and forget to push the Ctrl key before clicking on the Steering Wheel arrow, Solid Edge will think you are trying to move the spindle rather than copy it. You can just press Esc, and it will retain the selection for you to try again.
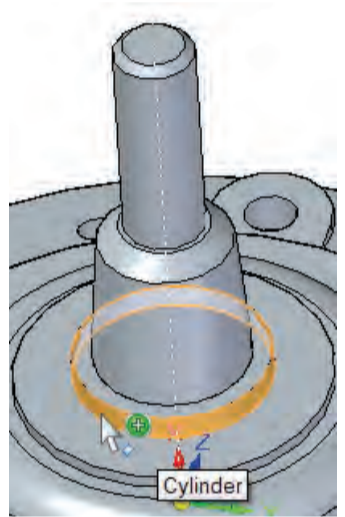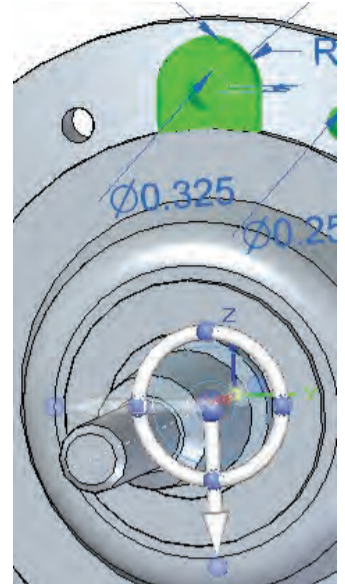
8. Drop the mouse buttons and keyboard keys, and key in 1.3 and then Enter, and Esc.

> *Note: If the copied spindle does not turn gray and merge with the rest of the part, and instead turns blue, that means it has been copied as a surface body. You can find the new body in the PathFinder, probably identified as "Face Set X", right click on it, and select Attach. This does a number of things automatically, including extending and trimming the surface body, and then merging it with the existing solid body. You could do all of this manually in Solid Edge, but it would be a tedious process.*
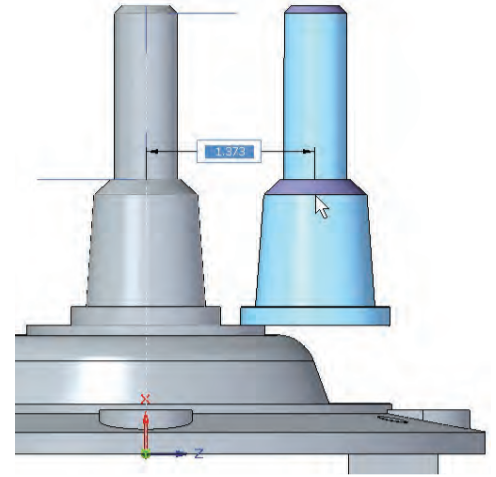
This type of edit produces a result that doesn't make much sense from a mechanical design point of view, but it does demonstrate a technique that you will need frequently.

Copying geometry from one part to another

9. One last demonstration. Drag a selection box around the spindle as shown. You should have 6 items selected.

10. With the selection still active, move the Steering Wheel to the bottom of the spindle selection such that the torus is around the axis of the spindle, and press Ctrl-C (copy) on the keyboard.

11. Now open the Piston target.par part from the folder of files for this chapter, and press Ctrl-V (paste). Use the Steering Wheel arrows to position the spindle so that it is fully contained on the rod, as shown. Click to finish the positioning.

12. Now find the Face Set in the PathFinder, right click on it, and select Attach. This will merge the spindle onto the piston rod.
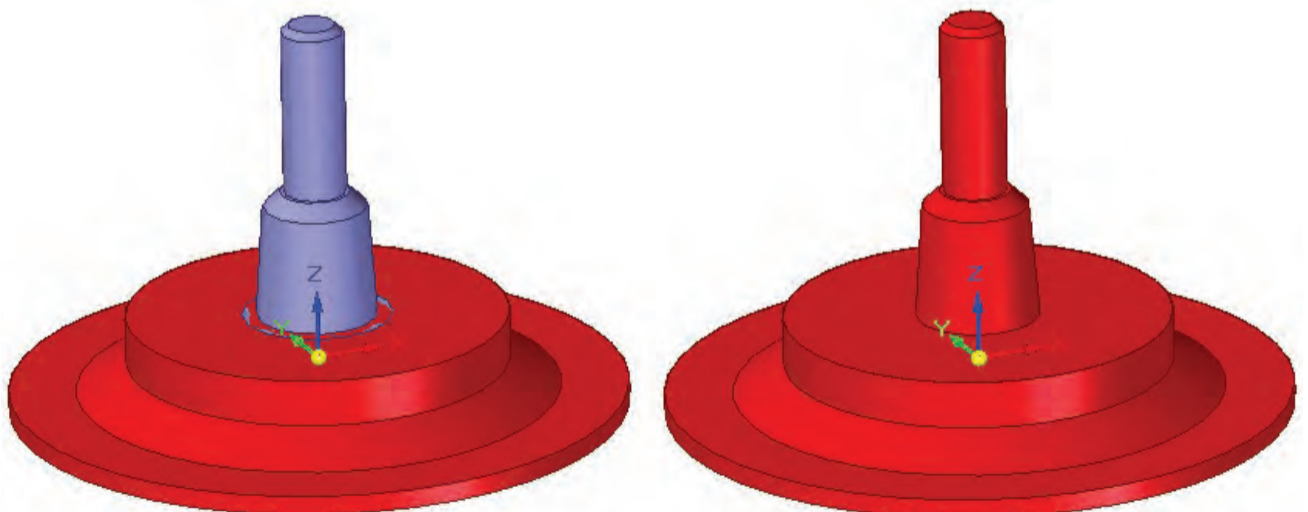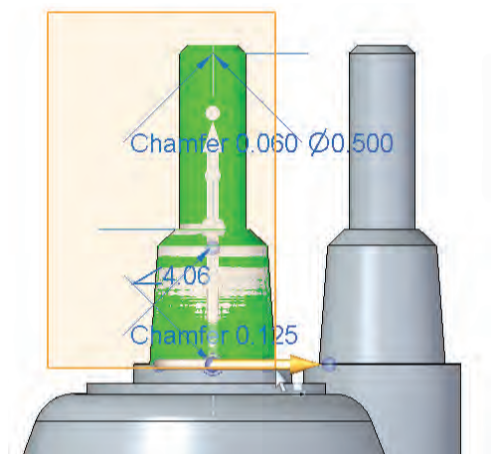
A brief video accompanies this chapter.

Summary

Synchronous technology changes are driven by face selections. In synchronous, the selection is the feature. The software supplies several ways to select what you need to get, including the automated methods of Selection Manager and Design Intent.

Because there are no restrictions on "parent/child" dependencies, or feature order in the tree, you can re-use any geometry you can select.

Chapter 5 | Selection and re-use

Chapter 6

Working with imported data

In the history-based CAD world, it is common to refer to imported data as dumb because the metadata necessary to drive history-based features is never transmitted along with the geometry. This is true whether the transfer is between two entirely different CAD programs or between different versions of the same program. You lose all of the intelligence, or design intent in a data import.

Some history-based CAD programs have features that are meant to look like direct-edit functionality, but they in fact create additional history-based features. This is not direct edit, and just compounds the problems associated with sequentially rebuilt history-based features, namely a big time lag associated with simulating direct edit functionality. Don't be fooled by this, you are still paying the history-based rebuild tax unnecessarily.

However, in synchronous technology, the design intent comes from the geometry itself, not from metadata stored with the geometry. If there are parallel faces, the chances that they are parallel by accident are very, very low. The software assumes this is intentional, and retains that relationship as design intent.

Synchronous technology can even recognize things like patterns, and holes in imported parts, and can create editable synchronous features. This is not some program that rebuilds the existing geometry with history-based features: it just assigns properties to the existing imported geometry. So if you created mates to holes in a pattern, and then run feature recognition to assign properties to those holes, the mates will continue to reference the original holes.

Reading native data

Solid Edge also has tools for import, such as direct import from SolidWorks, Inventor, and PTC Creo. To read these files directly, you must have Solid Edge installed locally, and the other program installed in a place where it is accessible on the network. Later in this chapter, we show the steps for migrating a SolidWorks assembly into Solid Edge.

The advantage of using the migration tools as opposed to simply importing data is that with the migration tools you get some assembly mates and some procedural features that come across into synchronous, and all native features translate as face sets (synchronous features).

Translation data types

One of the first things to address is the type of data being translated. The best type of data to send or receive for Solid Edge users is Parasolid. Parasolid files can come with a couple of different file extensions, but the most popular are "*.x_t" (text-based transmit file) and "*.x_b" (binary transmit file). Parasolid also has
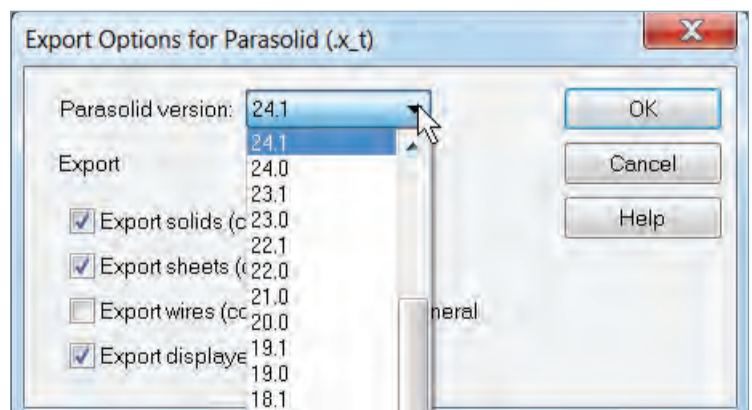
various versions, so if you are sending data to an older CAD program, make sure that it can accept the data that you write out.

Other common translation formats, in decreasing order of desirability are:

- STEP
- VDAFS
- ACIS
- IGES

Future version

Another big problem with history-based systems is the future version problem. This is the situation where you get native data from someone else, but you have an older version of the software than the sending system, so you can't read the data. You're likely to encounter this problem with most, if not all CAD packages. The solution, generally, is to get a Parasolid file instead, but as mentioned before, all of the intelligence is lost.

You can also run into the future version problem with Solid Edge, and the solution is the same, but the results are different. As you know, when this happens with history-based systems, you lose assembly mates and the ability to edit parts (all you can do is add features). It's as if the import came from some completely different CAD system.

Solid Edge with synchronous technology, however, can immediately edit all prismatic parts as if they were native. In addition, if the imports have patterns or holes, you can run recognition routines and actually make it native data. So with Solid Edge, Future version problems become an inconvenience instead of a complete change of direction.

Data migration

Solid Edge has three data migration tools. These tools pull native data from Inventor, Pro/E, or SolidWorks and create Solid Edge files. For this to work, you have to have Solid Edge installed on a local machine, and the other program installed on a computer that is accessible through the network.

The input for this process is a folder full of native parts and assemblies. The output is a set of Solid Edge parts and assemblies. Assemblies will bring across mates that use planes or axes. Parts will be synchronous, with the faces divided into user-defined sets of faces that correspond to the original SolidWorks features. This enables you to delete or move faces by feature.

You can find the Solid Edge migration tools in the Windows Start menu, under the Solid Edge entry. You must set up the paths for the various requirements, such as where the templates come from and where the finished Solid Edge files go.

Data Migration tools work better than using Parasolid, because you get synchronous features and assembly mates. In addition, it's automated, so if you have a large number of assemblies to convert, you can set it to unattended.
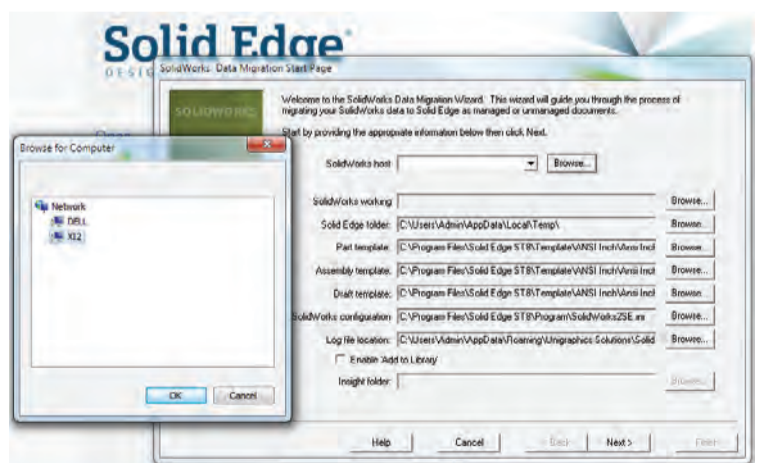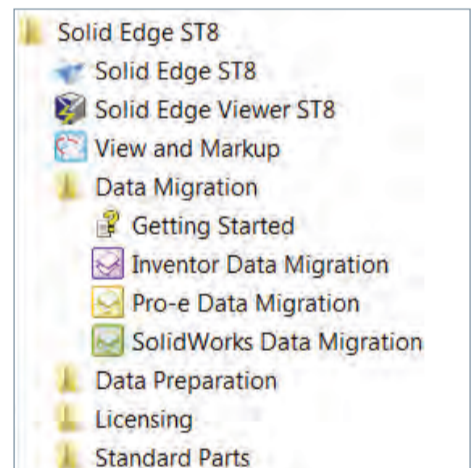
Migration tools translate parts and assemblies, and allow you to set up various folders for the use of the tool.

These images show, clockwise from the upper left, the original assembly in SolidWorks, the migrated assembly in Solid Edge with several of the mates that came over connected to the block casting part, and to the right, the synchronous features that came over to Solid Edge on the block casting part.

When you import an assembly through Parasolid or other formats, each part is grounded in place. But when you use the migration tools, many of the parts will get enough mates to allow for motion without further intervention. The quality of the end result depends in part on the quality of the data you start with, and also on the fact that only mates that apply to planes or axes can be recreated in Solid Edge.

Getting this sort of information with a CAD translation is something that history-based users have given up on long ago.
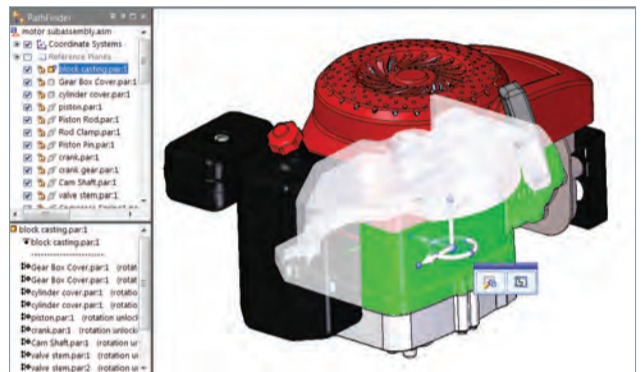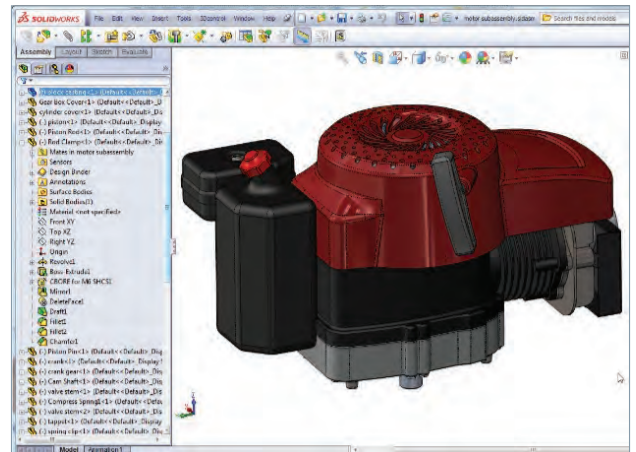
SolidWorks Associative Drawing Migration

At this writing, this tool is heading into Beta testing, but by the time this eBook is published, the tool should be available to the general public. It should be available as a part of ST8.

Prior to the introduction of the SolidWorks Associative Drawing Migration tool, we relied on DXF translations for drawings. These were not associative, so that changes to the parts were not reflected in the drawings. With the drawing migration tool, the drawings migrated to Solid Edge from SolidWorks will be associative to the migrated 3D models. This of course will make switching to Solid Edge to take advantage of synchronous technology that much easier.
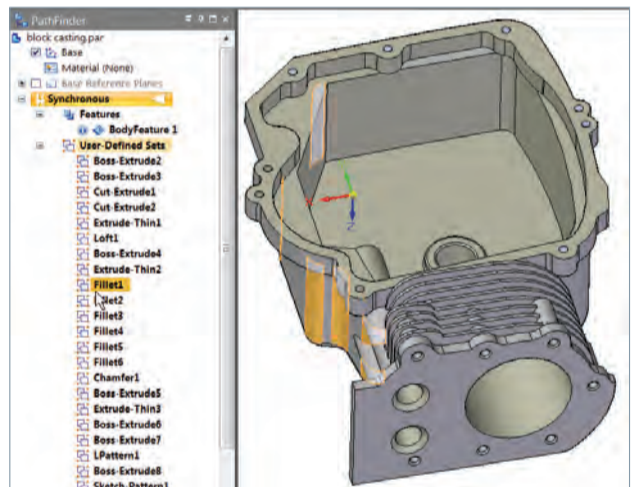
The SolidWorks Associative Drawing Migration tool will work with principal, section, auxiliary, and detail views. You will be able to use shaded, partial sections, cropped and broken views. Dimensions and annotations recreated in Solid Edge will be connected to the drawing geometry. Tables and additional geometry will also be available for reference.

Similar to the migration tools for parts and assemblies, you need a version of SolidWorks 2012 or later as well as your Solid Edge license to make this work.
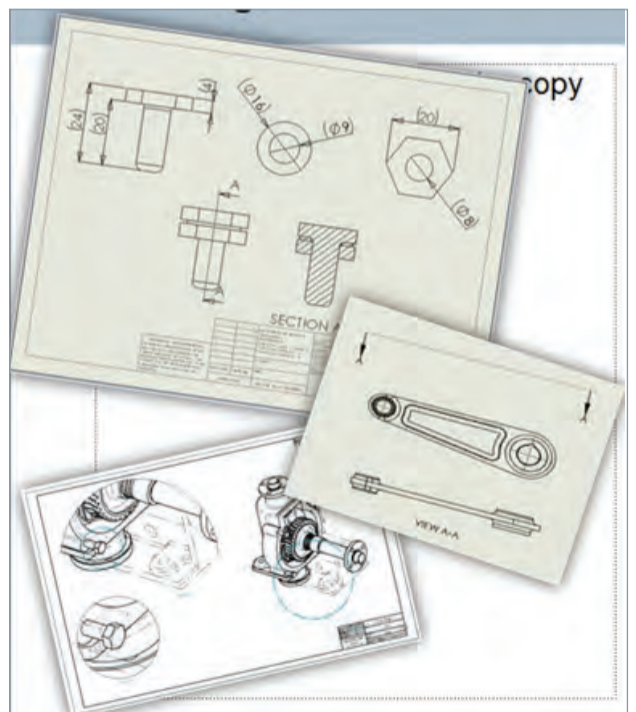
Import/export options

When you import or export files to/from Solid Edge, a button labeled Options becomes active in the dialog box. The Import options, shown below, enable you to handle some of the typical issues with transferring bodies and surfaces, as well as individual parts within an assembly.

Export options that enable you to set parameters are also available. The export options are different for each type of export format. If you are having difficulty getting a good transfer of geometry between two packages, these options are one place you should look.

In the end, successful knitting of all surfaces into clean, healthy solids is imperative for edits in synchronous technology to work. The small entities mentioned in the Geometry Inspector section can also derail certain synchronous edits, so merging smaller slivers into larger faces of the same type is crucial to getting successful synchronous edits. Remember from the more conceptual chapters that consuming topology is easy, but recreating it is not.
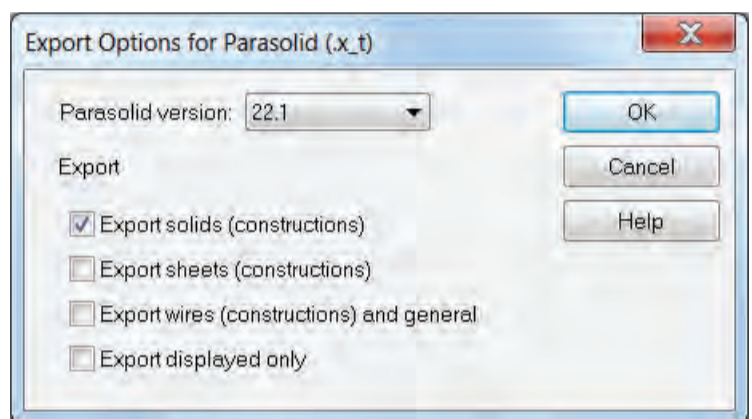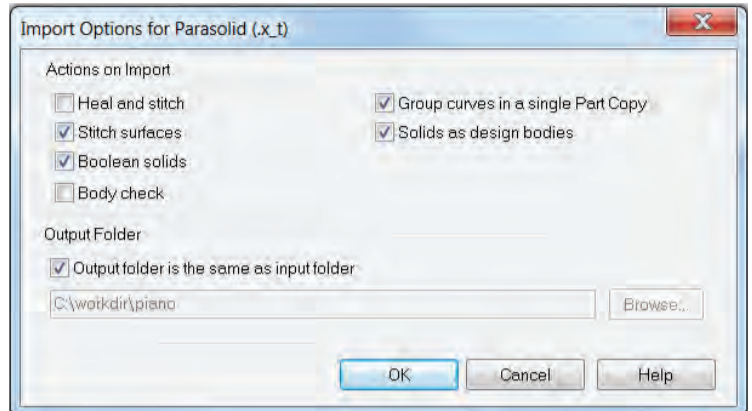
Feature recognition tools

The recognition tools in Solid Edge can be found on the Home tab, Solids group and Pattern dropdown list, under their respective feature names for Holes, Rounds/Chamfers, and Patterns.

Solid Edge has three recognition tools that will be valuable if you receive a lot of translated parts:
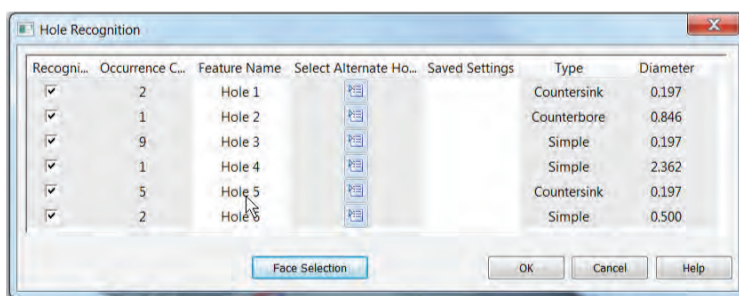
1. Hole Recognition – Recognized hole features allow you to change the diameter, move, or delete multiple holes that were recognized as the same diameter all at once. This enters into PathFinder a collector for the general hole type and individual features for individual holes.

2. Chamfer Recognition – Chamfers can be recognized, edited, identified, and deleted in the same way that holes can.

3. Pattern Recognition – Before running pattern recognition, it's a good idea to first run one of the other recognition tools. There is a separate tool for recognizing hole patterns vs. patterns of general features. Hole patterns also have the ability to recognize patterns of patterns. The advantage of recognizing patterns is the ability to change the number of features in a pattern. Holes can generally be moved or deleted in groups, even if they are not part of a pattern, just by selecting the collector in PathFinder for that type of hole. Alternately, you can Ctrl-select, or fence select, features that are not grouped into a single collector.
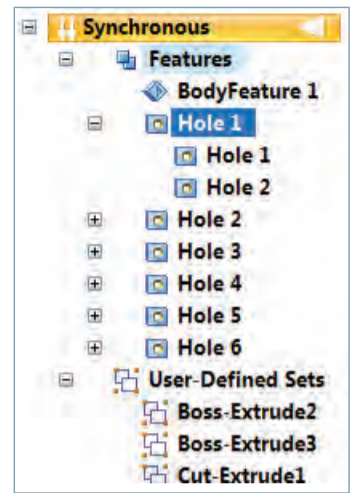
Geometry Inspector

When geometrical data is imported, sometimes errors occur. Errors will be more common if you use IGES or ACIS translation formats. Parasolid is the most reliable format to and from Solid Edge.

Note: When doing translations of parts or assemblies, best practice recommends that you round trip any file you save out. This just means that any file you write, you should also try to read back in. You should also recommend this to anyone sending you data. This guarantees that at least the package that wrote the data can also read it, which will help eliminate translation errors. Units are a common source of problems, as are multi-body parts and surface bodies.
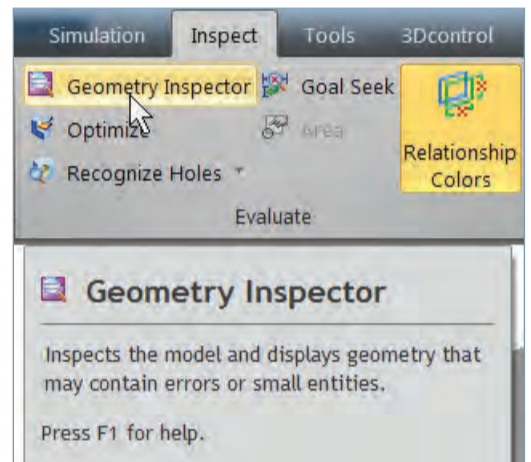
When you receive translated data, you should start out by running some diagnostics on the imported file. Diagnostics are especially important if you find the model behaving strangely, such as not allowing you to select certain faces, faces won't accept mates, unexpected parts of the model appear black or transparent, or you see spikes or other anomalies.

The Geometry Inspector command is located on the Inspect tab, in the Evaluate group.

Geometry Inspector is looking primarily for faults in the geometrical data, and optionally for small entities. Small entities are frequently where you find a lot of geometrical errors on imported data, and can make some functions such as creating Thin Wall features, offsets, or rounds difficult to create. You can specify how the Geometry Inspector should define small within the interface.

To use the Geometry Inspector, open the part you want to check and then click the Geometry Inspector button. Set the options to look for either Faults or Small entities, and then click Go.
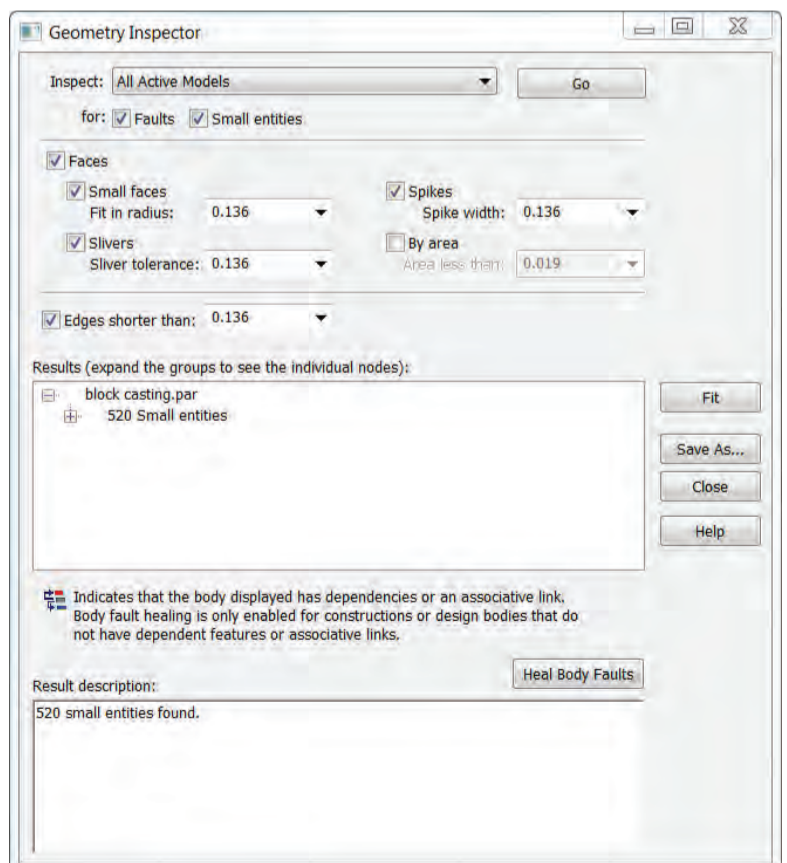
If there are no faults, the Result description box will state that. Also, faults or small entities will be highlighted on the model in the graphics area. If you find faults, you should first click the Heal Body Faults button. This automatically tries to fix the faults. If you use this a couple of times, and the Geometry Inspector still reports faults, you may need to manually repair the fault. There are many ways in which you might go about this, including using surface modeling to dissect and reassemble the faces around the fault or simply cutting out a solid chunk and remodeling over it. Rounds are often the source of errors, and deleting a few faces around a fault may be all it takes. This is an area that requires advanced experience, and is really beyond the scope of this book. Besides, the tools needed are not that different from the tools you would need in history-based software.

Editing imported parts

Editing imported parts is only different from editing native synchronous parts in one respect: There are no procedural features in PathFinder. For that reason, there will be no editing section in this chapter. Everything that would be said will also be said in Chapter 7: Editing with synchronous technology. Many of the parts we use in that chapter will be imported, just to make things simple.

This chapter has a brief video to accompany the text.

Summary

Working with imported data is one of the strengths of synchronous technology because the data can be edited immediately after import, or it can be massaged with recognition routines for additional usability. In addition, the migration tools take parts and assemblies to another level by adding synchronous features and mates. With synchronous technology, there is no such thing as dumb geometry.

Chapter 7

Editing with synchronous technology

We have already talked about the biggest differences between history-based modelers and synchronous technology being that the method of editing has nothing to do with how you originally model the geometry. When you switch from history-based modeling to using synchronous technology, many of the habits you learned in history-based methods become not only obsolete, but also counterproductive in many cases. For these reasons and more, as you learn how to make edits with synchronous technology, you should approach it as something completely new. Don't expect it to work like history-based CAD software, with all of the best practice rules that you have become used to, because it is different. Expecting synchronous to work like history will lead to frustration. It's different; it works differently; it has different strengths and weaknesses.

Note: A brief video goes along with the content of this chapter.

Assume everything is possible

You should start with the assumption that everything is possible in synchronous technology. Synchronous, of course has limitations, but there are things that are possible in synchronous that will surprise you. Things that you would not even imagine doing in history-based software, such as controlling a "parent" from a "child" feature, editing only part of a "feature", rotating an entire feature out of the plane in which it was created, and so on. If you give up the idea of the limitations of history-based CAD that you have learned to live with, the realm of what is possible with synchronous will open up for you, and you will be much more successful with the switch.

Learning the capabilities of a system is one thing, but in order to maximize your effectiveness, you also need to understand what you can and cannot do with it. This might sound like a contradiction of the previous paragraph, but we are preparing you for a big shift in how you approach editing geometry.
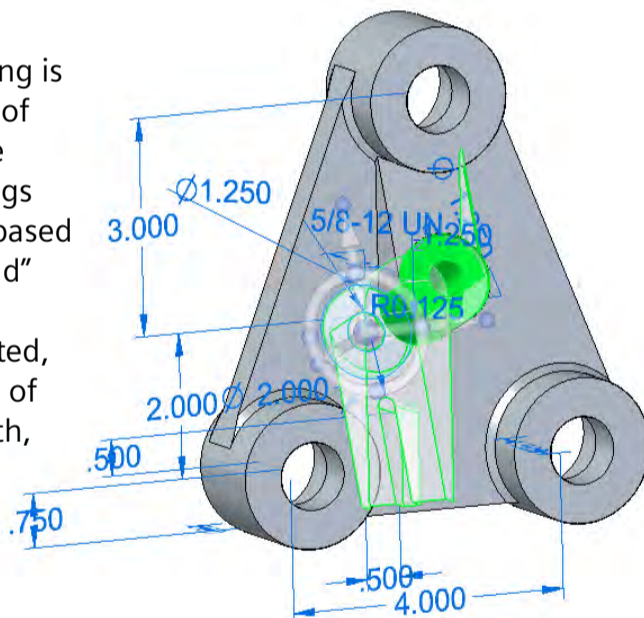
Limitations of editing with synchronous technology

Once you understand the boundaries of a system, it's easier to learn what it can do. Synchronous technology has two main limitations that you need to know about before you start modeling:

1. Synchronous technology as implemented in Solid Edge® software can only edit prismatic geometry. Spline-based general NURBS surfaces can be created and can exist in synchronous technology, but they can only be edited through history-based techniques.

2. Synchronous technology can consume topology while editing, but cannot re-create it (during the edit process) once it has been consumed.

Both of these statements deserve further comment, and the next section is devoted to a more complete explanation. The statement about "topology" will also be demonstrated in a later section of this chapter.

Prismatic versus spline-based general NURBS

For the previous statements to make much sense, we have to define some of the terms.

The word "prismatic" denotes combinations of the following types of geometry:

- Planar
- Cylindrical
- Conical
- Spherical
- Toroidal (doughnut shaped, with double curvature)

This means any shape you can create by extruding or revolving any combination of straight lines and arcs is considered "prismatic" (sometimes also referred to as "analytic"). These are all readily editable by synchronous technology.

Splines are general curves with continuously changing curvature, with the exception of ellipse, hyperbola or parabola. We don't need to get too deep into the technical definitions here. In Solid Edge, splines are called "curves" in the sketch tools. You can create geometry from splines/curves in synchronous, but you cannot edit those faces directly. You can, however, make synchronous sketches that drive history-based features. There is an example of this later in this chapter. It sounds mind-blowing, but it is not only possible, but highly useful at times.

The only methods available for editing spline-based complex geometry are moving/revolving, replacing or deleting the face, or of course cutting it off of the model or consuming it in other geometry.

For this reason, when you run into complex or spline-based geometry, we recommend that you use an ordered feature for that type of work.

Creating topology

For our purposes in this book, we will take "topology" to mean "the number, type and connectivity of faces". The types we refer to are simply the prismatic types mentioned above. So a box has topology consisting of six planar faces, where each rectangular face is connected to four other rectangular faces at the edges. A cylinder's topology consists of a single cylindrical face with two planar faces normal to the axis.

Spline-based faces are also topology that you may see in Solid Edge, but this discussion is about topology that can be edited directly using synchronous methods, so the spline surfaces are not discussed here.

In synchronous, you can eliminate the top flat face by applying the rounds on the top corners, but you cannot go backwards through the process to get the flat face back. This works much like 2D editing of the same type of shape.
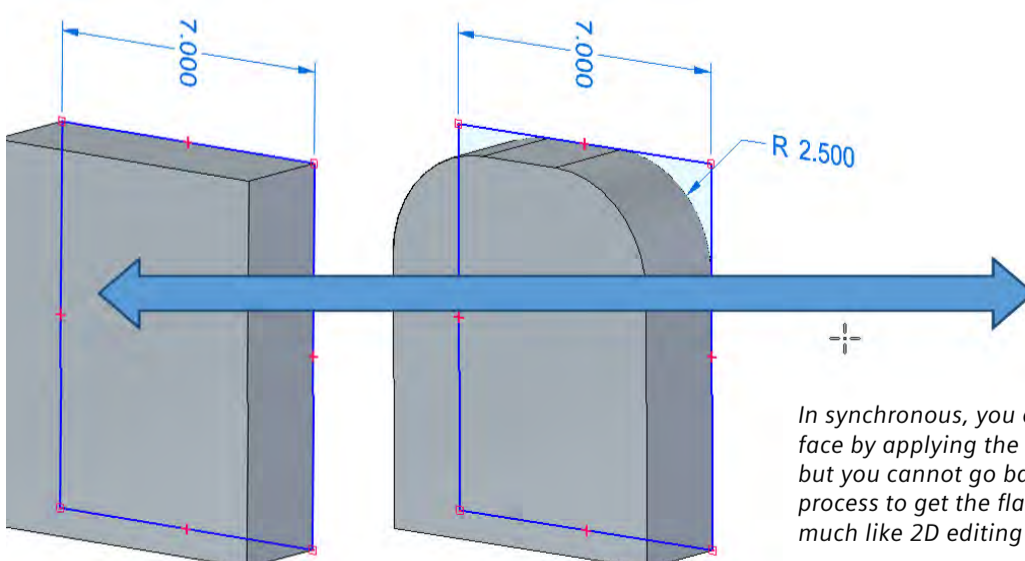
Deleting topology happens frequently enough. One example would be when you put a full radius on the end of a part that encompasses two edges. For example, the image on the previous page, on the right, shows a block, and then rounds applied that eliminated the flat face on the top to make a tombstone shape. In this situation we say that "topology (the flat face on top) has been eliminated by the round". Synchronous can create this situation, but it can't go through this process in reverse: that is, starting with the tombstone shape, and recreating the deleted topology by simply changing the radius dimension. Going through the process in reverse will fail in synchronous. If you can still use Undo, you could get the flat back, or you could simply cut another flat, reassign the tangents and correct the length. (The preferred method here would be to apply the rounds in Ordered mode to the rest of the part which is in synchronous mode.)

Synchronous technology is analogous in many ways to 2D sketching. If you were to make this same change in a 2D sketcher, the flat would be eliminated when the radius becomes half of the width, but again, you cannot go backwards through the process. In 2D, you cannot automatically recreate a flat by simply making the full radius smaller. It's the same idea in 3D. Since we are already familiar with this problem and its workarounds in 2D sketching, dealing with it in 3D should not be a problem.

To get the flat on top back, you could:

- Use Undo if still available
- Re-extrude the original rectangle
- Cut off the round on the top and resize the block to full size
- Cut off a portion of the round leaving a flat face on top, and use Delete Face to remove the rest of the round
- The real solution to this issue in synchronous is to make the box in synchronous, but make the rounds in ordered. This way if the rounds consume a face, you can get the face back. Use the strength of synchronous when editing the box (synchronous strength) and use the concept of ordered features for the rounds (which play to ordered strengths).

Using the Steering Wheel

The Steering Wheel gets a lot of demo time, but you need to know that the Steering Wheel isn't the primary tool for editing in synchronous technology. In fact, for people just learning it, the Steering Wheel isn't even the first option we like to turn to.

The Steering Wheel is used for three things:

1. **Directional control** – The Steering Wheel is easy for changes along or about the three main axes, but you can also use it to precisely angle faces, or move components in an assembly at an angle. Once you invoke the change by pulling the arrow or torus in a specific direction, a value field appears which you can use to make your motion very precise.

2. **Differential control** – Sometimes you only know that you want to move a face by a certain distance. The Steering Wheel is good for this kind of edit, because the value field that appears only represents the change in size, position or angle.

3. **Approximate visual edits** – This is the part that many people get hung up on. People tend to complain about synchronous technology not being good at "precision". This perception seems to come when CAD users are getting their first look at the Steering Wheel, and only see the tug-and-pull operations changing parts. If tug-and-pull were the only way synchronous made edits to parts, then I would agree – it would need more precision. But combined with the driving PMI and differential control operation, synchronous technology has as much precision dimensional control as any other CAD modeler.

The main message here is that the Steering Wheel is a visual interface device that is capable of precise dimensional control as well as approximate visual editing.
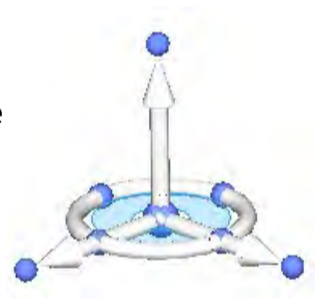
Tutorial: Imported air filter housing

Let's consider this part, shown below. This is called "air filter.par", and can be found in the materials for this chapter. It is an imported part, created in SolidWorks, brought to Solid Edge using the Migration Tools. It is what most history-based users would call a "dumb" part. The previous chapter dealt with imported parts specifically, but we will continue some of that work here because imported and native parts are not that different when it comes down to it.
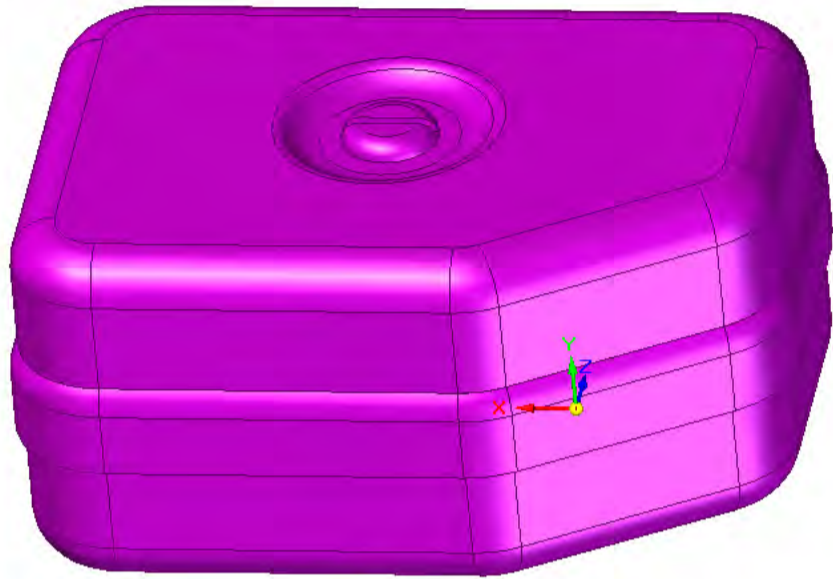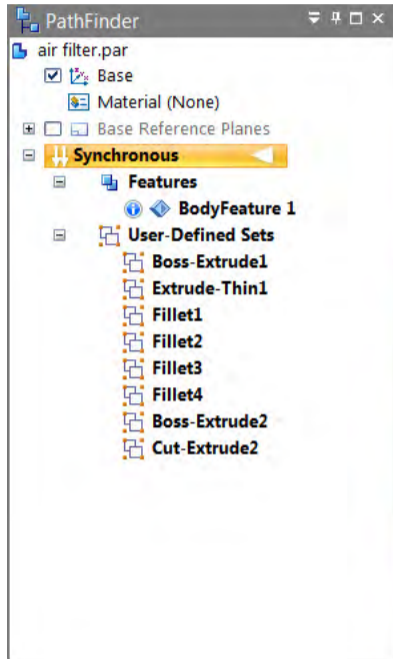
Apply dimensions

Let's start with some dimensions. You'll notice that all of the edges you would want to dimension have been removed by rounds. Most history-based CAD administrators who make best practice rules for their users will make a rule that says something like "don't apply dimensions to edges created by fillets". The reason for this rule is that dimensioning to an edge created by a fillet makes the fillet the driving feature, and we all know how often fillets just disappear.

> *Note: Synchronous technology PMI dimensions drive faces even though you select edges to create them.*

This rule is a good practice if you use software that can't automatically compensate. Solid Edge, however, understands what you are looking for when you dimension to a round, and handles it correctly automatically. When you place the dimensions you'll see it highlight missing edges or vertices. Plus, remember that although you select edges when placing dimensions, Solid Edge is really using the edges to identify faces, and is actually dimensioning faces.

Changing units

This part has been imported to a template that uses inch units. If you prefer another unit system, it is best to change it before you start applying dimensions. To change to millimeter dimensions, go to the **Application** menu -> **Solid Edge Options** -> **Units** -> **Unit System** and select MMKS or your preferred unit system.
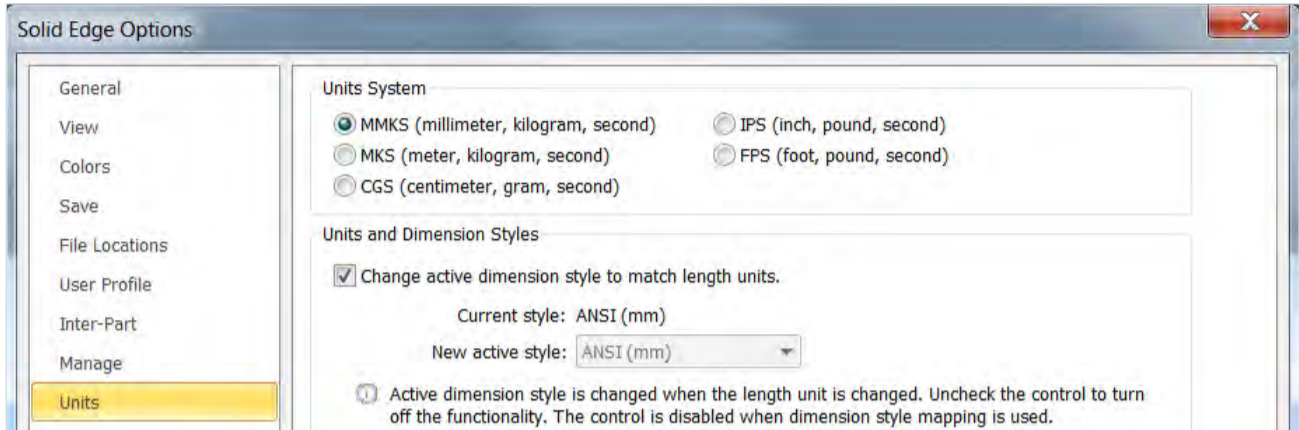
This will set the units for any new dimensions that you place, but will not change any existing dimensions. To change existing dimensions, select the dimensions from the PathFinder PMI collector (via Ctrl- or Shift-select), right click on the selection and pick Properties from the RMB menu. Then select the units from the options for Linear and Angular dimensions.
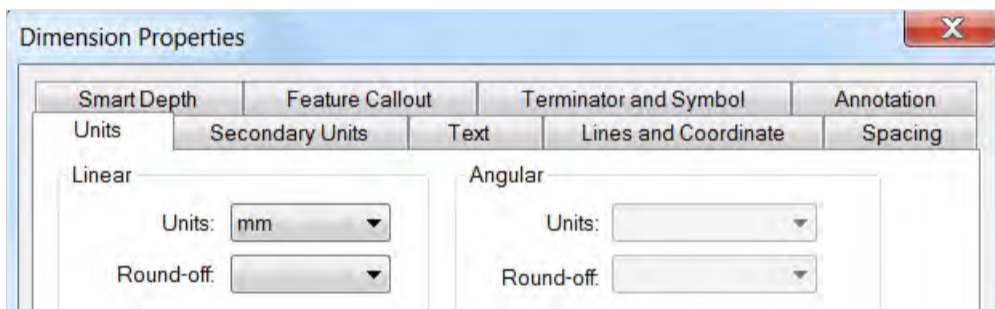
Note: The Units page of the Solid Edge Options controls the active dimension style, which can be set separately on the Dimension Style page of Solid Edge Options.

Dimensions to rounded edges

1. Go to the Home tab, and select the Smart Dimension tool. Create the dimensions shown on this part.

   Note: If you are getting results from your dimensions that you don't expect, try to spin the part to look orthogonal to the view in which you want the dimension to lie. You can use a 3D controller, your middle mouse button or the Quick View Cube in the lower right corner.

Change dimensions

2. Click on the value (number) of the 2.755 dimension. This brings up an editing box, and you can use your scroll wheel to change values or key in numbers. When you use the scroll wheel, the geometry updates automatically. When you key in numbers, you have to press TAB to get it to update while still keeping the value box open. If you press Enter, the part will update, and the box will close. Remember the trick with the TAB key, as this workflow will come in handy when testing various values.

Notice that this dimension works for values between about 1" and 4.75". Outside of that range, the dimension will fail due to the rounds colliding with one another. Notice what happens when the dimension fails. An orange dimension icon shows up in the graphics area, and other attaching dimension may turn a brown color, which means that the dimension has lost one or both references (this condition is called "dangling").
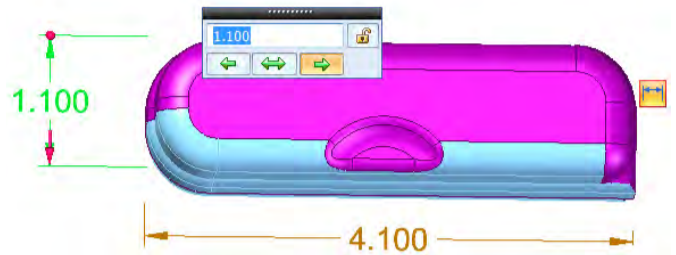
Consumed topology will return within an edit, but not after you have pressed Enter.

You need to recognize and know how to react to failures in synchronous technology, just as in history-based software. In this case, you can press ESC to go back to the pre-edit value, key in another value or scroll until you hit a value that works.

Note: This is one of the places where you need to be careful. If you are changing dimensions, and a face in the model is consumed, and then you use Enter exit the Modify function, you will not be able to get that face back by changing dimensions. You will have to use Undo or one of the more advanced methods discussed later.

Change fillet radii

3. Click on some of the round faces on the model. Notice that for every fillet/round face you select, Solid Edge gives you a radius value. Notice also that in the PathFinder, the User-Defined Sets contain four fillet features. If you click on one of those, the faces associated with that feature are highlighted.
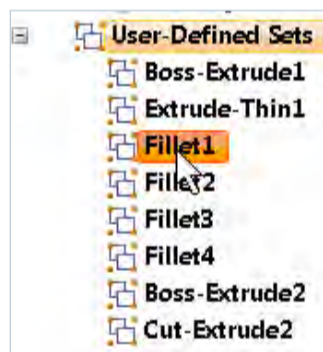
Click on one of the light blue highlighted faces shown in the image above, and then click on the radius value that appears.

What's happening here is that changing the radius for any of the 0.500" rounds will change the radius for all of the rounds with the same radius, not just the rounds created in the same feature. The icons to the right of the value in the modify box enable you to change just the selected radius, or all the faces with the same radius. Change the value up and down, and see what the limits are, and what it looks like when you push it to failure.

Reattach dimensions

4. You might notice that the dimensions are connected to the main body of the filter, not the offset edge that goes around it. To reconnect the dimension, click on some part of the dimension other than the value (so, the dimension or witness lines), and notice that you get light blue and red handles. The light blue handle is to control the offset of the witness line from the part, and the red handle controls the connection point. To reconnect the connection point, move the cursor over the red handle until it changes to a "+", hold down the Alt key, and drag the handle to a new entity.
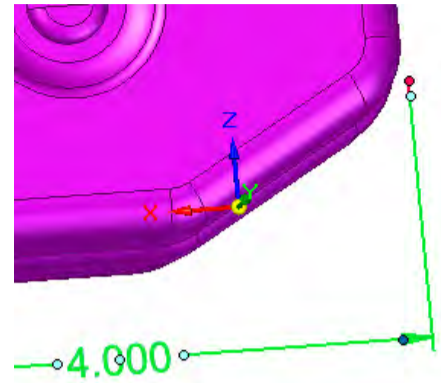
The dimension should increase by 0.050" when attached to the outer side of the lip. Since this needs to be done twice, it will increase to 4.100".

Move origin to helpful location

5. You might notice that the origin is not in a helpful location. It should be located in one of the corners or maybe the center of the screw location. In history-based software, the sketches determine the location of the part, so if you wanted to move the entire part with respect to the origin, you would have to move several sketches. In 2D, you just move the drawing, view, sketch or whatever you are working on. Synchronous again in this situation works more like 2D than like history-based CAD.
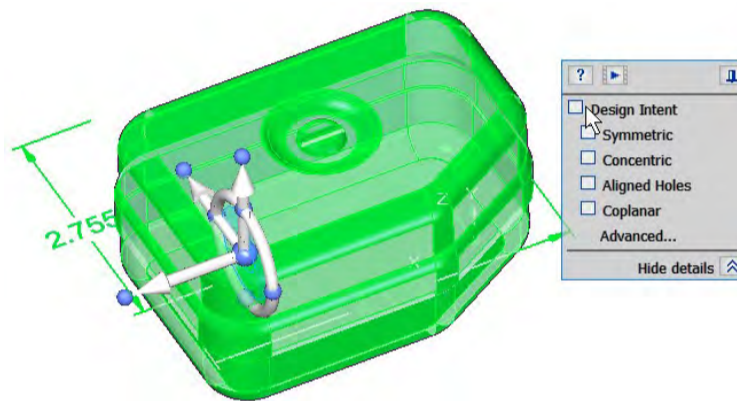
To move the origin to a helpful location, we simply move the entire part. Here's how to do this:

6. Make sure all tools are clear (you're in select mode), and drag a selection window around the entire part.

    As a warning, it is important to keep the entire part active as the selection for this entire process. Extraneous clicks may clear the selection, and you will need to start again.

7. All of the geometry should turn green, and the Steering Wheel appears. You will also get a little gray window called Design Intent. Since we aren't changing the shape or size of the part, we can turn this off. So remove the check next to Design Intent. Using the Move Origin of the Steering Wheel (big blue dot), drag the Steering Wheel to the vertex shown. The view orientation is the same as the previous image, it is just zoomed in.

    Next, Shift-select the arrow parallel to the 4.000 dimension (X axis) and move it to the left, then key in .3, and press Enter. This moves the entire Steering Wheel to the combined distance of the radius and the lip, so it is at the furthest extent in the X direction of the model.
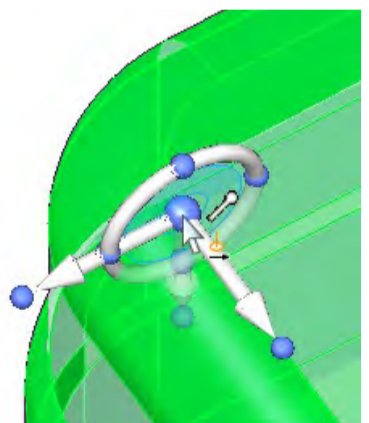
    Single click in white space to bring back the entire Steering Wheel. The selection will remain active.

8. Now, Shift-select the arrow parallel to the 2.755 dimension (Z direction), move the cursor away from the center of the model, key in .55, press Enter, and click in white space again, and you should have something that looks like the image to the right. The Steering Wheel is now positioned where you want the part origin to be.

   To move the part such that the part origin is in that location, we can do it in two steps.
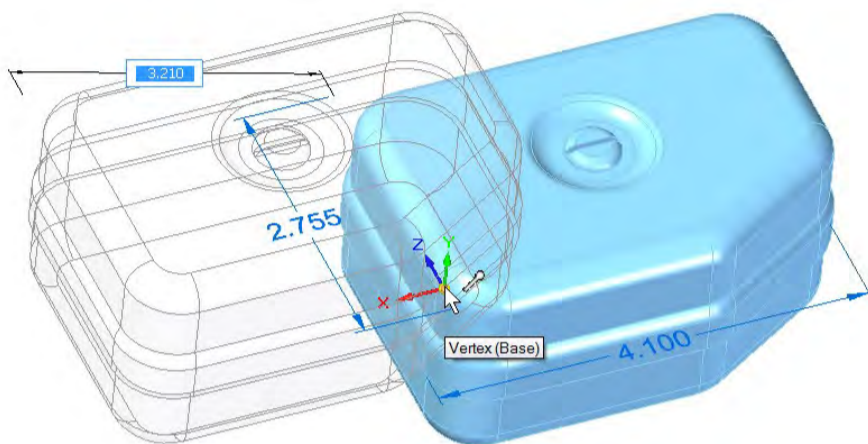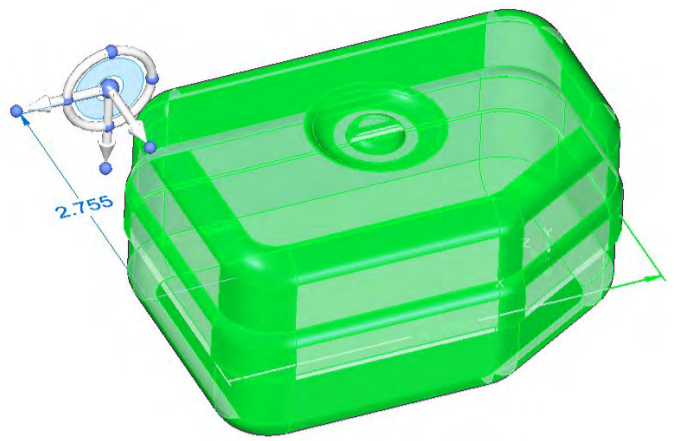
9. First, click on the tool plane of the Steering Wheel. This moves the part in plane, even if you select something out of plane. So your second click should be on the part origin, which is ghosted but still visible against the part. When you start the move, the part origin will be colored normally again, and be easier to see.

10. Finally, click the arrow parallel to the Z axis, and then click on the part origin again. This should put the part origin at the corner of the virtual bounding box of the part. You can now press ESC to clear the selection.

Remove radii

By now, you have read several times in this book that fillets and rounds should be treated differently while modeling using synchronous technology because they can consume faces, and cause errors.

In most cases, you will be creating your own rounds, so you can create them in ordered mode, to follow best practice, and get the most editable models possible.

But in the current situation, we have an imported part, and we didn't have a choice about the rounds. Still, this situation can be handled by the tools in synchronous technology. We just need to first remove, and then reapply the rounds.

> *Note: You don't have to work this way with all imported models. You can leave imported rounds on the part and continue to work that way. Sometimes, however, you may want to have more control over the round features, and this part of the demonstration shows that even though this is an imported part, you still have complete control.*

Some rounds are easy to remove, and others will require more effort. Let's start with the .25 R rounds on the top and the bottom. These are very easy.

1. Just click on the Fillet3 feature in the Pathfinder, and press Delete on the keyboard.
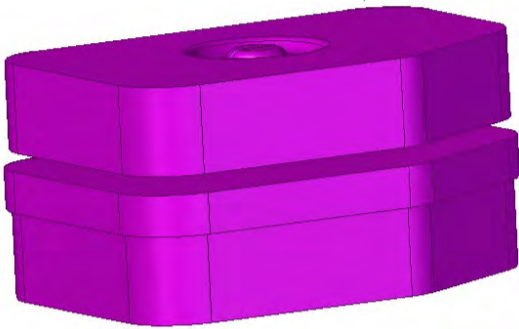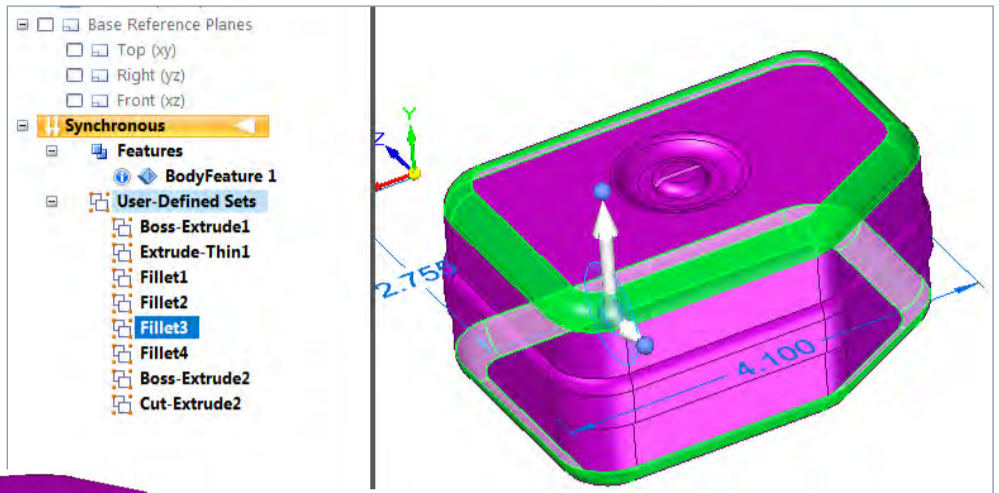
2. Now select Fillet4 and press Delete. Notice first that Fillet4 was not a complete round, and that there are no faces to extend to take its place. This is part of the "consumed topology" problem. Deleting the feature has done something that was unexpected: it cut the model in half.

There are two ways to deal with this:

3. First, we could simply extend the cut face of the lower half of the model up to the upper half. To do this, select the cut face, click the arrow in the direction you want to move the face, and then click an endpoint on the upper half of the part. That's easy.

4. The second method is less easy, but because we didn't expect this to happen in the first place, it is the method you should learn, because there won't always be an easy way out of an unexpected situation.

5. Press Ctrl-Z (undo) until you get the Fillet4 feature back.
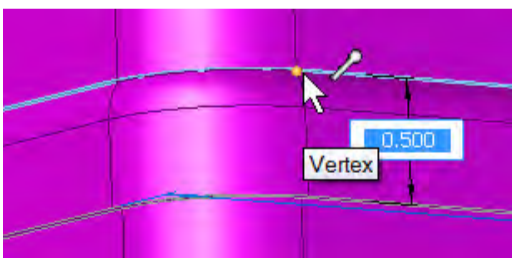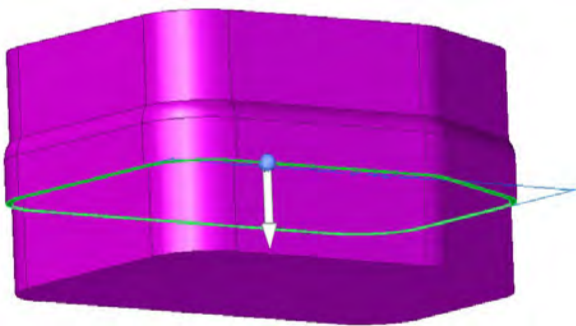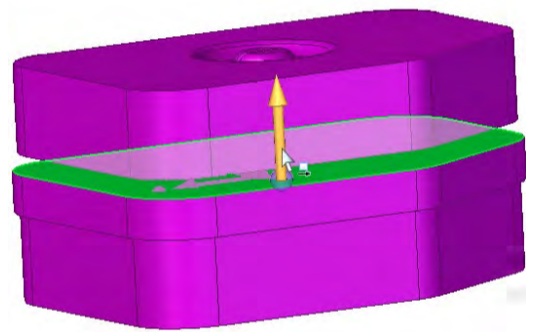
6. Now rotate the model so you can see the flat underside of the lip around the midsection of the part, and select that flat face.

7. This shows the simple Steering Wheel. Hold down the Ctrl key, and click on the arrow, and then move the cursor to the top edge of the Fillet4 feature that we want to remove, and click a Keypoint, as shown in the image.

8. This has essentially copied the flat face (as a surface body) to the location where we want to recreate a flat face that was there before the partial round Fillet4 consumed it.

9. Next, click in white space to clear the selection.

10. To incorporate the flat surface in the solid model and remove the round faces, we are going to use the Replace Face function. Find Replace Face on the Surfacing tab, Modify Surfaces group.

Chapter 7 | Editing with synchronous technology
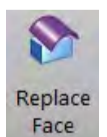
11. The Replace Face workflow goes like this. Invoke the command, select faces that you want to get rid of. Then select the face that you want to use instead. The CommandBar helps you step through the workflow. Remember that the screenshots in the book are taken using the Most Assistance theme, which makes the CommandBar look different (more words).

12. So select all the faces of the partial round. You can press Ctrl for the second face, but you'll see a special cursor, and don't need to hold down Ctrl for the third or subsequent face selections.

13. When all the faces of the round are selected, click on the Accept button in the CommandBar.

14. Next you will need to zoom in to select the offset surface we copied.
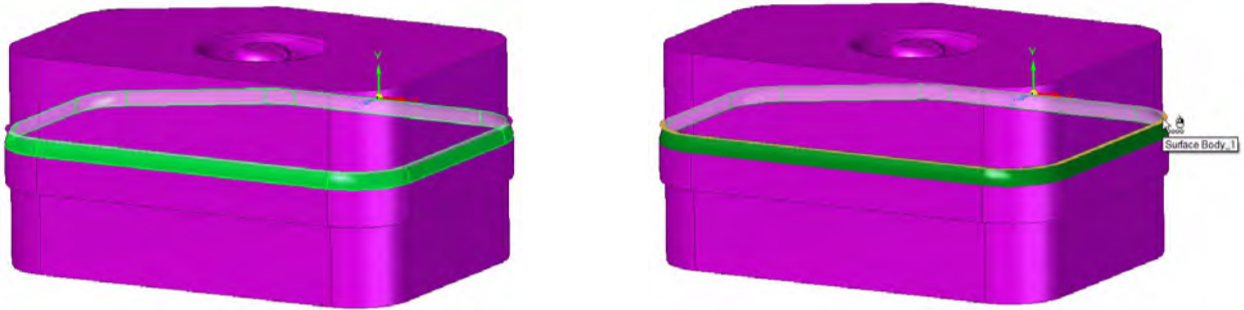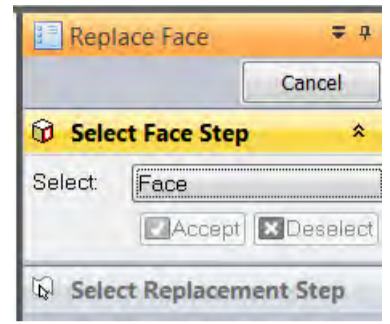
15. The result is a lip that goes around the part with flat faces on top and bottom. Press ESC to finish the command.

16. The remaining fillets could be deleted by just selecting the Fillet1 and Fillet2 features in the Pathfinder, but in the line of demonstrating techniques, select the 5 round faces with the .55 radius around the lip, and press Delete on the keyboard.

17. Notice that when you delete all 5 faces together, the Fillet2 feature in the Pathfinder is also removed.

18. Now click and delete Fillet1 in the Pathfinder.

19. Next, delete the green face from the screw hole, and finally, the yellow face from the screw hole.

   Note: The result of leaving a chamfered face around the hole is somewhat unexpected, since all the faces that could be extended to patch the deleted faces have been consumed by the rounds. This function should have failed, but obviously Solid Edge has added some magic behind the scenes.
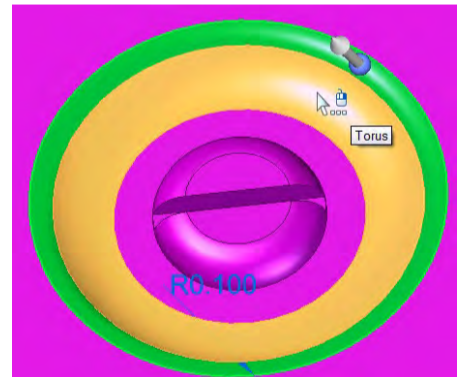
20. The result is a conical face. When you click on a synchronous conical face, an angle dimension is displayed, but you can't use the angle to make either 0 or 90 degrees, it has to be something in between. This is because the surface would change type from a cone to either a cylinder or a planar surface.
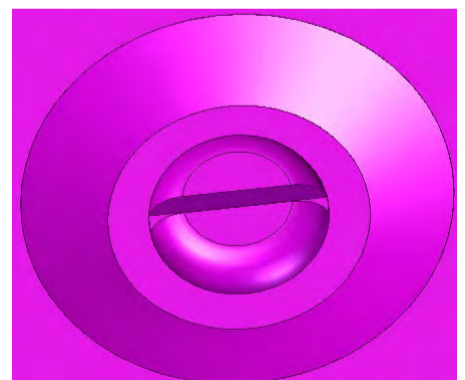
21. If the goal is to make this a counterbored type hole, the way to handle it is with a Draft feature, applying a draft of 0 degrees.
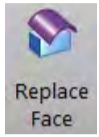
The Draft tool is found on the Home tab, Solids group. The first face (Draft Plane) to select is the main planar face of the part. The "Draft Faces" in this case is just the conical face created by deleting the rounds. Enter 0 to the value box.
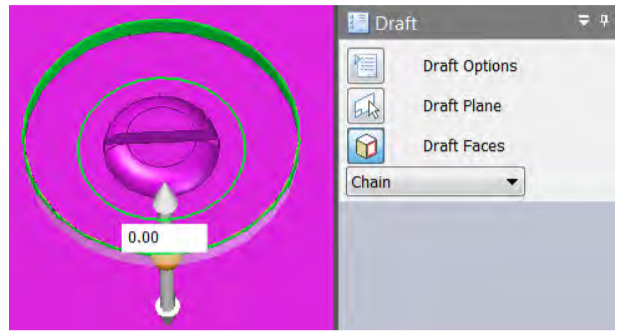
22. The rest of the rounds can be removed by selecting the features in the PathFinder, and deleting them.

Note: It would have been much easier to move the origin after the rounds had been removed, but the difficult method was demonstrated to make sure your skill set was more complete. Also, you could have placed a reference coordinate system, and used the Steering Wheel to locate it.

Remove screw head

1. There are several ways to remove the screw head. You could fence select all the faces on the screw head and press Delete on the keyboard. Or, you could use the Detach command, which will enable you to reattach it later on, if you want to. You could also cut away the screw head at the same time you cut a hole for the screw to go into. Since we also need a screw hole, let's use this last technique.

2. On the Home tab, in the Draw group, click on Project to Sketch. Then bring your cursor over the big flat face opposite the screw head, and press F3 on the keyboard. Finally, click on the outer edge of the screw head, and Solid Edge will place a sketch circle on that sketch plane.
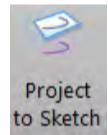
Press ESC to cancel the Project to Sketch command, and click inside the projected circle, which will give you a green highlighted region.

3. The goal here is to create a hole that is offset from the bottom (where the projected circle is) and go through the top (where the screw head is), and remove the screw head in the process. We will then make the hole smaller so the screw head doesn't go down inside the hole.
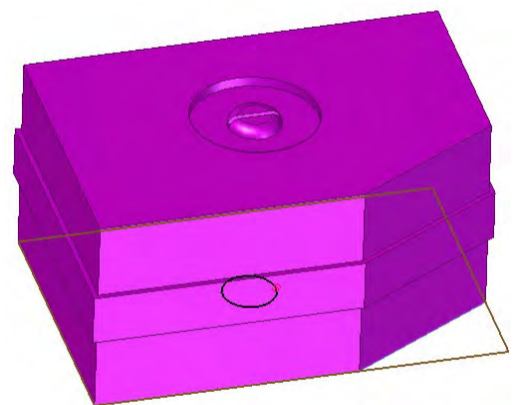
4. The process that we will go through could be done more simply, but these are techniques you should know how to use, so they will be demonstrated here.

5. To start cutting from an offset of the projected circle, we have to move the projected circle. In the PathFinder, expand the Sketches collector, and select the sketch with the circle in it. When you do this, the circle highlights, and the Steering Wheel appears. This enables you to move the sketch with the Steering Wheel.
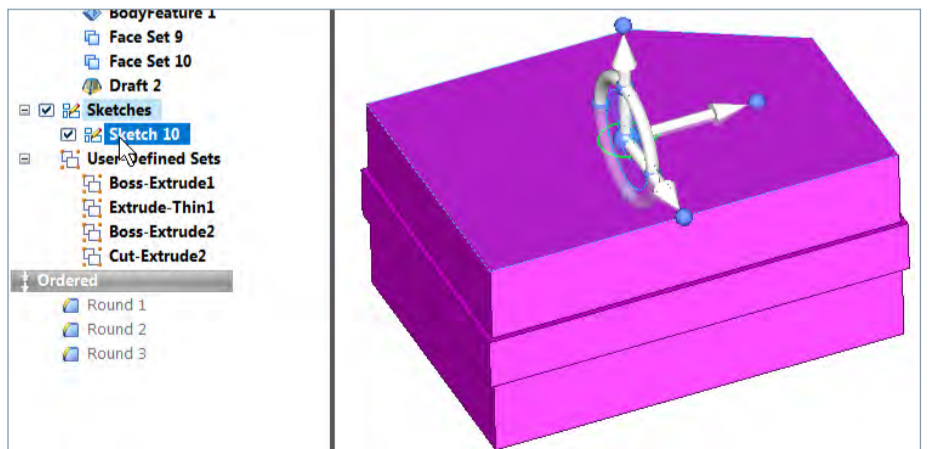
Note: You could also select the sketch by using the RMB QuickPick method, which has not been covered to this point. QuickPick appears as a mouse icon after you hover over an entity for a second or more. When you click the right mouse button with the QuickPick icon showing, it activates a small list of items that are under the cursor in its current position.

QuickPick

- ○ Circle
- Region (Sketch 10)
- Region (Sketch 10)
- Plane
- Plane
- Plane
- Sketch 10
- Design Body_1
- BodyFeature 1

air filter.par
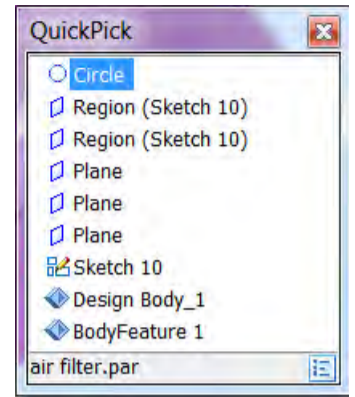
6. Click the arrow pointing vertically in the image above, and drag it down into the part. Key in .375.

7. Notice that this moves the sketch without moving any faces or planes. If there were other sketch items on that plane, they would also move.

8. From here, the sketch is buried in the block, so you can't just select the region inside the circle as easily as you could when the sketch was on the flat face. So we have to use QuickPick again to select the region.

9. Hover the cursor over the interior of the circle until you get the QuickPick icon, then click the right mouse button. Bring the cursor down the QuickPick list until it highlights the correct region inside the circle, and select that item. The Steering Wheel appears.

10. Now drag the Steering Wheel arrow through the rest of the part until it comes out the other side.

11. When you flip it over to look at it, the hole should have removed the screw head, but you will also see that the hole does not go all the way through the part.

12. Next we need to make the hole smaller. The radius of cylindrical faces of all types are only controlled by dimensions. There is no visual dragging like you find for flat faces. So when you click on the cylindrical face of the hole, a diameter value appears, in addition to the depth dimension which is already there.

13. Click on the dimension, and change the value to .375.
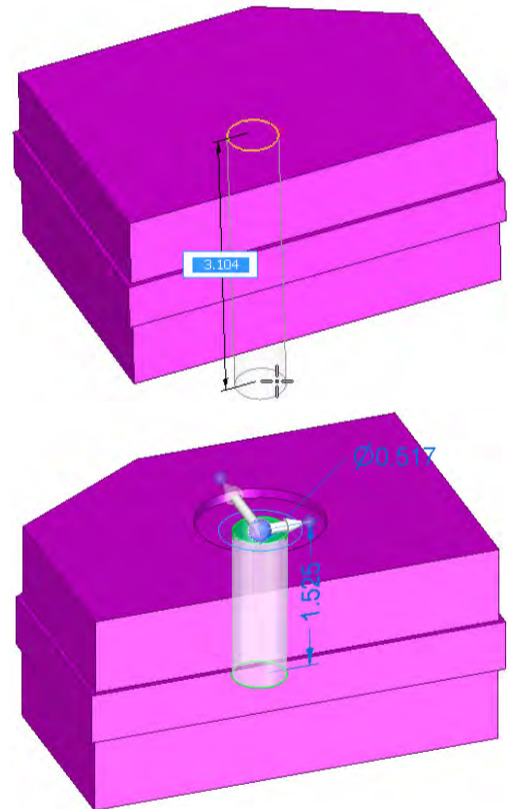
Shifting into ordered mode

1. Now we need to put the rounds back on the model, but these rounds need to be applied as ordered features, so let's shift into ordered mode.

2. Right click on the synchronous header in the Pathfinder, and select Transition to Ordered.
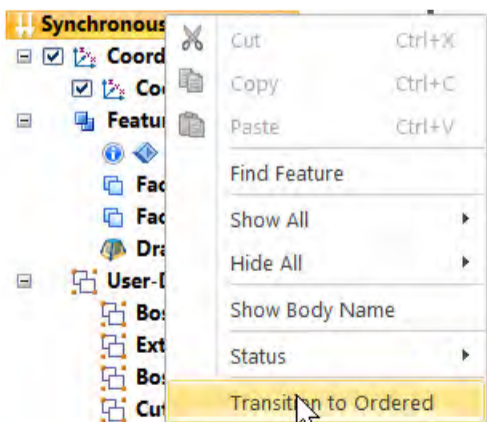
Note: When you combine synchronous and ordered modes in the same part, the synchronous body always comes first, and then the ordered features are applied on top of that. You can make synchronous edits to the synchronous body while in ordered mode. To add more synchronous features, you have to go back to synchronous mode. You can also push ordered features to synchronous, but this is a one way trip. Synchronous features cannot be promoted to ordered features.

3. The ordered features will show up under the ordered header after all the synchronous items. New features will be listed in the ordered area.

Replacing the rounds

1. Usually, the recommendation would be to apply the rounds in order from biggest to smallest. In reality, it's more difficult than that to write best practice type rules that work 100% of the time. Best practice must always be accompanied by common sense. In this case, we need to deviate from the best practice order slightly.

Round

2. Create a Round feature using a radius of 0.500" for the edges shown to the right in orange.

3. Create a second Round feature using radius of 0.550" on the edges shown to the right in blue.

4. Create a third Round feature using radius of 0.250" on the edges shown below in green.
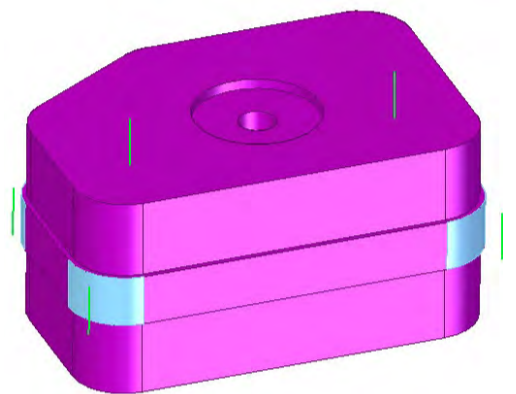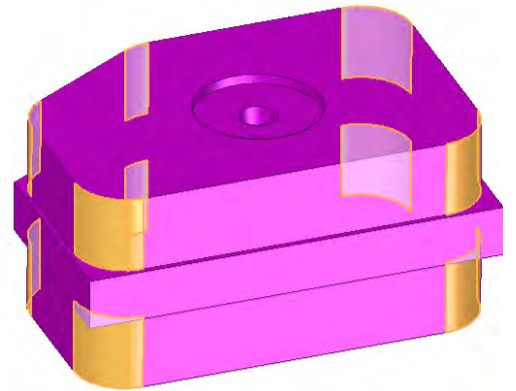
Making synchronous changes in ordered mode

The Solid Edge PathFinder helps you keep your model information accessible and organized. One of the items you can find quickly is the PMI collector. You can turn all or some of the dimensions on or off at will. You can group dimensions together so they can be toggled always at the same time. You can even select, delete, control properties, and do many other things to your PMI dimensions from the PathFinder.

The dimensions Linear 5-9 are locked dimensions that maintain the width of the lip around the part. The 4.100 dimension drives the outside of the lip while the 2.755 dimension drives the inside. This is not necessarily the way you should do it, but it demonstrates that you can drive either side, and there is no such limitation as the "parent/child" difficulties you run into when figuring out how to drive a history-based model. (The model has been saved at this point for your convenience with the name Air Filter Intermediate 4.par in the data for this chapter.)
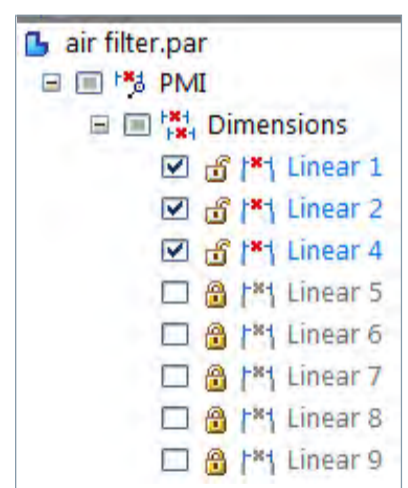
5. Turn on/off dimensions until you just have
    these three dimensions showing on the
    screen, for the overall part dimensions.

6. In previous instructions, you learned how
    to reattach dimensions to new geometry
    (by selecting the dimension first, then
    Alt-dragging the red handles). This time
    we just want to change values.

7. Click on the 2.755 value, and the modify
    dialog appears. Use the scroll wheel on
    your mouse to change the value, or key in
    a new value.

8. Since these dimensions are blue, it means that
    you can change them via the modify box or by using
    the Steering Wheel. So click on a flat face,
    and give it a tug.

9. Notice that if you click on a face belonging
    to a round feature, the toolbar for making
    changes to ordered features appears.

Chapter 7 | Editing with synchronous technology
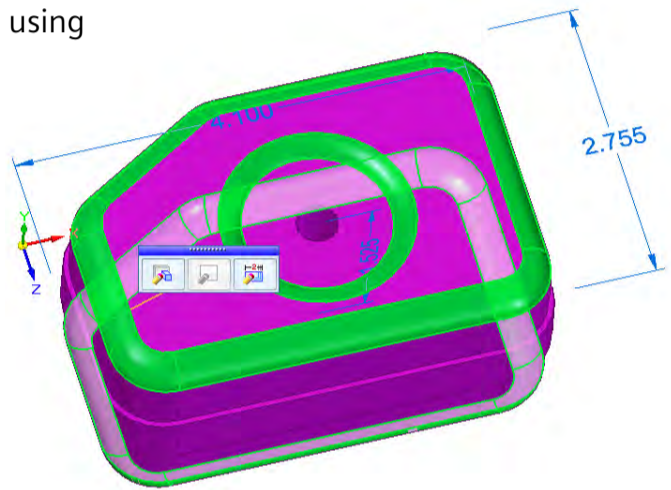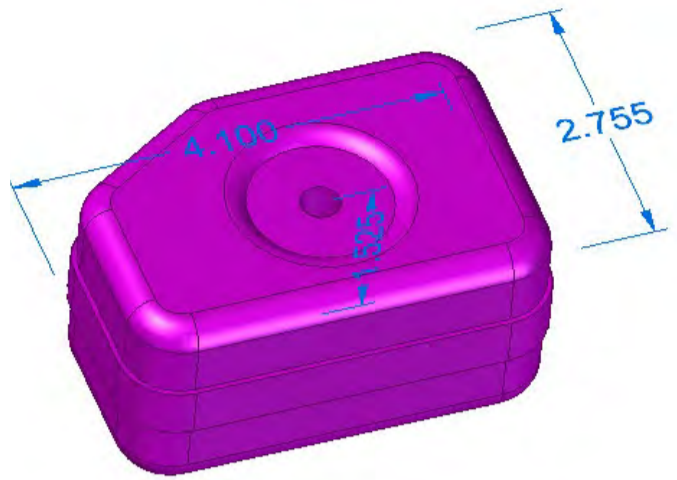
Summary

The main idea that you need to take from this chapter is that although the methods may also be used independently, synchronous and ordered methods work together very well. The synchronous method relieves you from the pain of parent/child history-based entanglements. The ordered method is used primarily where there may be danger of features consuming faces which might need to be replaced later on such as rounds, or detail features that require sketch editing such as extruded text.

You will find that there is very often more than one way to do things, and you always need to be familiar with multiple methods, because one method is not going to work for everything.

Practicing using the Steering Wheel can be important for being able to make different kinds of edits. Also, multiple methods for selections, whether from the PathFinder features, fence select, Ctrl-select or QuickPick, you will need each of these in various situations as you continue to learn and use Solid Edge.

Making yourself familiar with the idea of driving the 3D model directly using locked and unlocked PMI dimensions opens an entirely new world for CAD users ingrained with the history-based mindset. Making the conceptual leap to synchronous technology enables you to regain control over your design intent.

There is much more to synchronous technology, but the simple concepts demonstrated in this chapter paint a picture of compelling functionality to add to your current toolbox. The beautiful thing here is that synchronous technology does not require you to give up your current expertise in history-based modeling, synchronous will rather enhance that, building on the areas where history-based modeling is weakest, to give you a broad, full-function set of tools for every situation.

Chapter 8

Synchronous sheet metal

Sheet metal parts in Solid Edge are a different file type than general Solid Edge parts. Sheet metal parts have the extension *.PSM. This chapter assumes that you are familiar with the sheet metal manufacturing process and you have some previous experience with designing sheet metal in CAD.

> *Note: A <u>brief video</u> goes along with the content of this chapter.*

Solid Edge, like most 3D sheet metal modelers, expects you to build sheet metal parts flange by flange and add features as you go. An alternative workflow is also available where you start with a flat sheet and add bends. The design process for sheet metal does not resemble the manufacturing process – that is, you do not start from a sheet, make cuts to it, and then add bends. Instead, you create the final shape, and then from the finished shape the software calculates the flat pattern and bend lines.

You will be more successful and happier with the software if what you expect it to do is in line with what the software actually does. In general, Solid Edge sheet metal is intended for straight brake press type operations, but can also perform some stamping, lancing, and basic forming. The material will remain a constant thickness with constant radius bends, and all sheared faces are perpendicular to the face of the sheet material. Solid Edge can provide a flat pattern for sheet metal parts and can calculate bend allowances accurately using multiple methods.

Some situations in real-world sheet metal work are not allowed for in CAD sheet metal, such as corners where bends overlap, or overlapping flat patterns. Deep draw, progressive die operations, and heavy forming can be modeled geometrically, but Solid Edge is designed to deal with the special considerations required by these processes.

Solid Edge sheet metal and templates

Solid Edge sheet metal templates are usually created from a dedicated sheet metal part, saved with the same extension. Solid Edge sheet metal parts can in turn be created from templates saved in either synchronous or ordered mode. General Solid Edge parts and imported parts can also be converted to sheet metal under some conditions which we will explore later in the chapter.

All of the sheet metal parts I have made in this chapter were created from synchronous sheet metal ANSI inch template. You should have sheet metal templates in all of the drafting standards you will use, as well as in synchronous and ordered.

Synchronous sheet metal can be created from closed-loop sketches (Tab) or open sketches (Contour Flange). As with parts in the rest of synchronous, the initial sketches are used only to create the first feature, and are not linked to the solid geometry after that.

Base features in sheet metal

The Tab is most commonly used as the first, or base feature. With this method, you have to draw the profile of a panel and extrude it to the thickness of the sheet metal part. The Tab tool is on the Home tab, Sheet Metal group. You can enter the material thickness when extruding the feature, or later, in the Material Table.
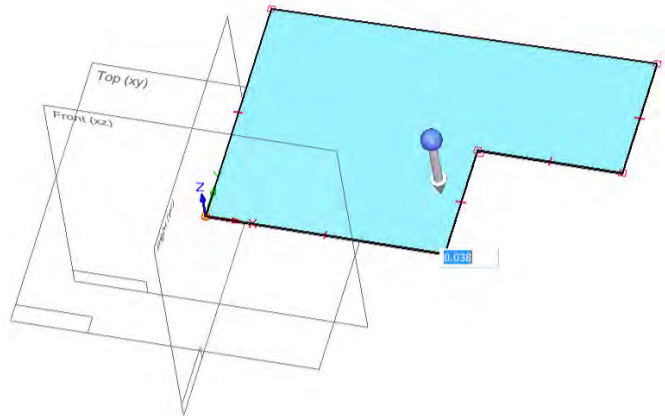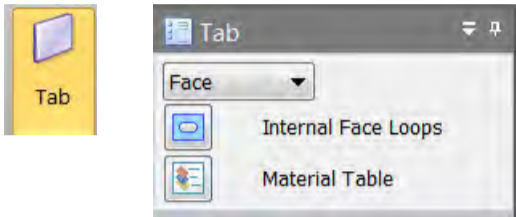
To start with a Tab, sketch the outline of one of the panels. Solid Edge automatically extrudes it to the material thickness set in the Material Table.

Solid Edge Material Table

You can use the Material Table command to assign the material, thickness, default bend radius, method for calculating bends (gage table, neutral factor (K value), or a custom formula. This is the same Material Table command that you use to assign materials in non-sheet metal parts also. The Material Table is available from the sheet metal CommandBar and from the **Application** Menu -> **Properties** -> **Material Table**.
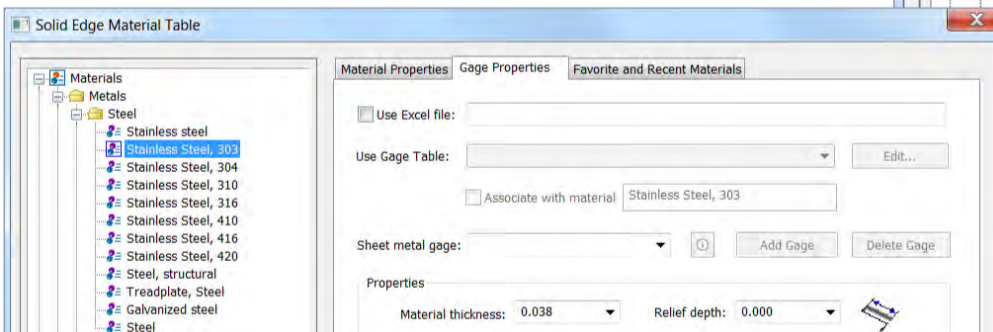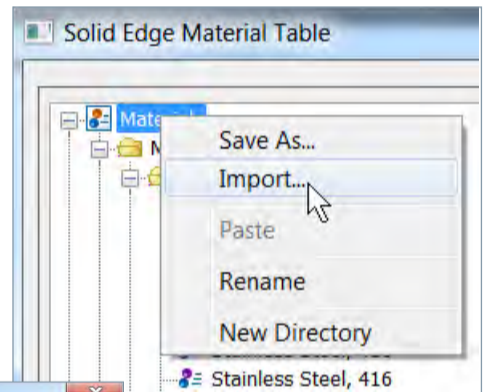
The Solid Edge Material Table can be saved out and imported as an Excel file format. This contains all the properties for materials from sheet metal to simulation.

Solid Edge installs several materials. If your installation doesn't include materials, in the panel on the left of the table, right-click the top-level materials, and then select Import. Solid Edge searches for a *.mtl file or an Excel worksheet. In the folder with material for this chapter, there is a sample Excel worksheet, called Material.xlsx, that shows how Solid Edge expects the material information to be formatted.

The Solid Edge Material Table can be saved out and imported as an Excel file format. This contains all the properties for materials from sheet metal to simulation.

Getting started with a contour flange

The second method of creating the initial feature in a synchronous Sheet Metal part is the Contour Flange, using an open sketch, highlighted in orange in the image.
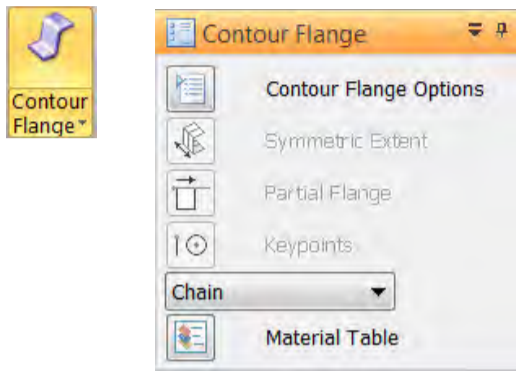
The Contour Flange Command Bar.
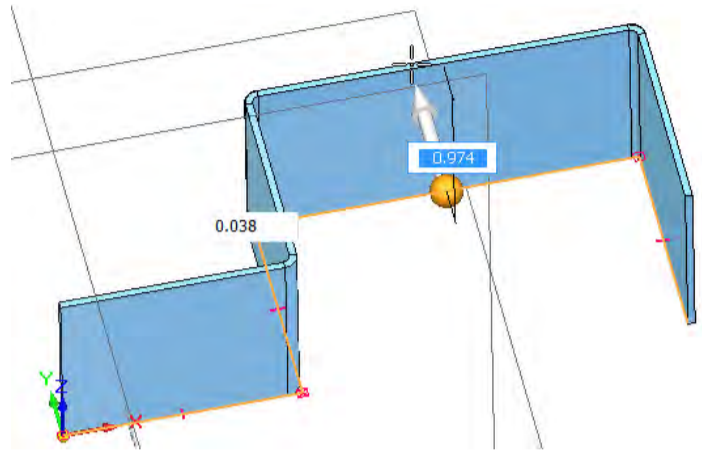
To start with a Contour Flange, you need an open sketch profile. Leave the bend corners sharp in your sketch because the software will add the bends for you based on the Material Table properties for your selected material.

Adding flanges

To add a flange onto an existing panel or flange, simply select the thickness face and pull the short arrow of the Steering Wheel. There is no toolbar button for a simple flange.
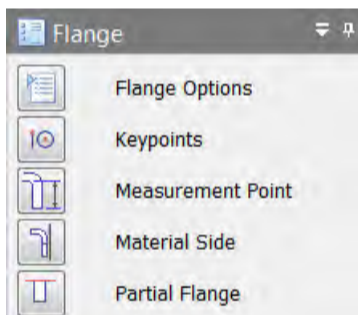
The Flange Command Bar.

You don't have to initiate a command or press a toolbar button to get this to happen, just click on the face, and pull the arrow. The result, and the accompanying Command Bar are shown below.

Setting the length of the flange can be done in one of three ways:

1. Position the cursor on one side of the original edge so the new flange is displayed on the side you want it on, and then click (visual placement).

2. Instead of clicking in the previous step, key in a number for the length and press Enter to accept it.

3. Instead of clicking in empty white space in the first step, click on a keypoint, such as the end or midpoint of an existing flange.

To place the flange at some angle other than the default 90 degrees, press the Tab key, which will highlight the box that displays the angle measurement (90.00). With your cursor on the correct side to denote direction, enter the correct value and press the Tab key again to keep the value, and go back to editing the length of the flange.

You can select multiple edges to create multiple flanges in a single operation.
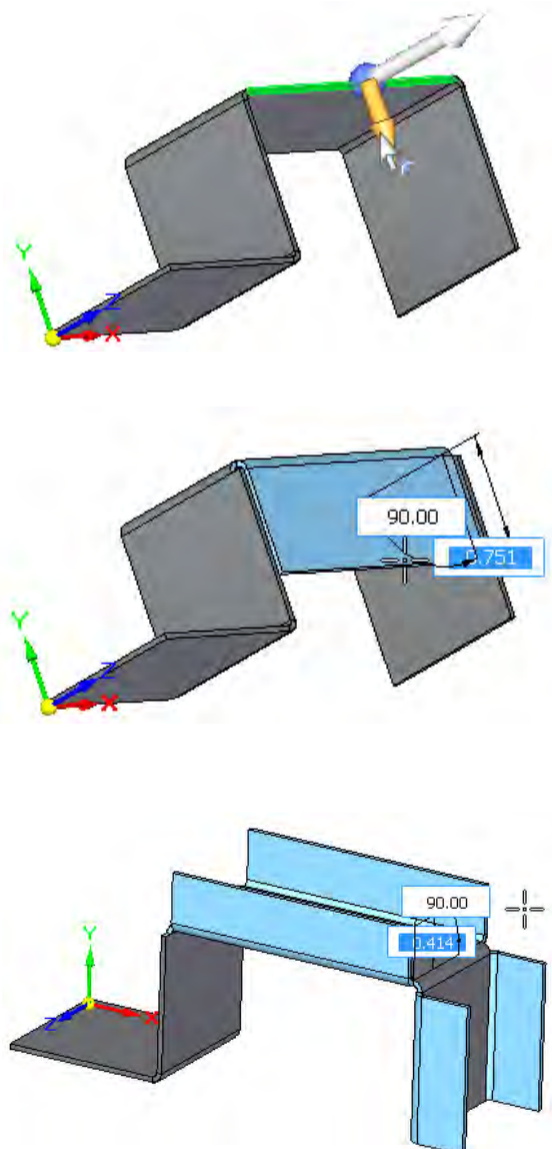
Contour Flange

A contour flange is a flange that can be sketched to include multiple flanges with bends and can also go around multiple thickness faces of another flange. To start a contour flange, you must have a sketch with some geometry in it. If you don't have any sketches available, the Contour Flange command will be grayed out. To activate the button, the sketch doesn't need to be appropriate for the contour flange, it must simply exist and not be empty.

For example, the sketch on the right could be used to start a contour flange, and the resulting flange might look as shown next.
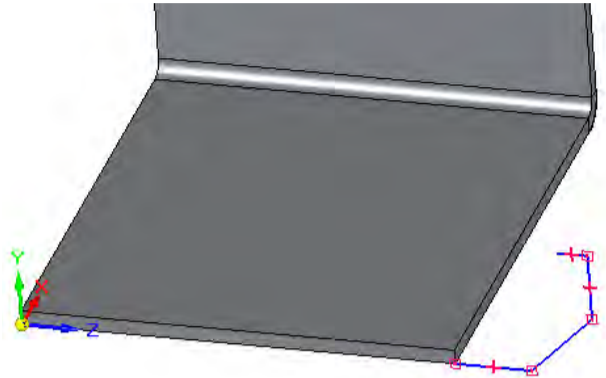
To create a contour flange:

1. Create the sketch on a plane perpendicular to the thickness face you want the contour flange to run along. Make sure the sketch is connected to either the top or bottom edge, as shown in the previous example.

2. Click the Contour Flange command. (Follow the Prompt Bar prompts).

3. Select your sketch as a chain of sketch entities.

4. Pull the Steering Wheel arrow in the direction along the face where you want to place the contour flange.

5. Select additional adjacent thickness faces to run the contour flange around corners. Solid Edge will automatically miter corners and add bends according to your sheet metal defaults for this part.
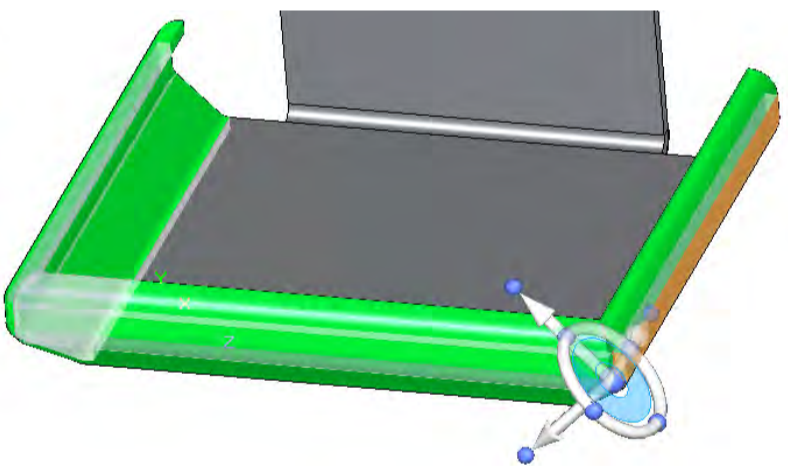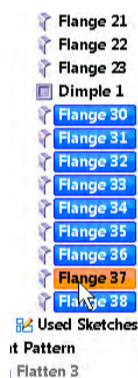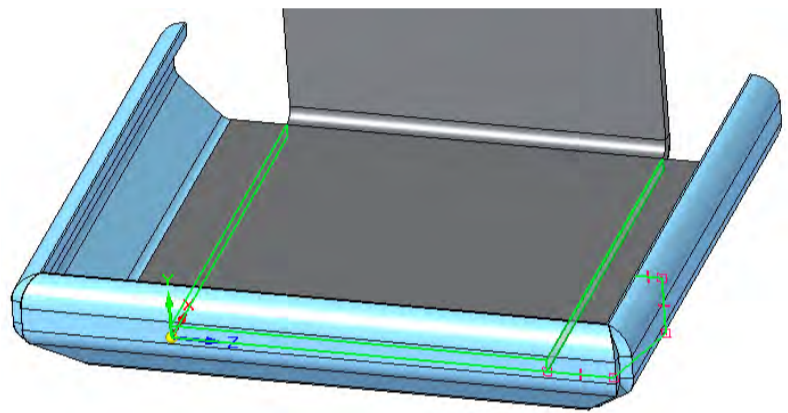
The Contour Flange command automatically makes a set of regular flange features to match your sketch and selected thickness faces. Editing must be done through the normal flange editing process, with the steering wheel and PMI.

For example, the contour flange created above turns into Flanges 30-38 in the section of the PathFinder shown to the left. For complex edits to complex contour flanges, we recommend that you simply delete the existing flanges and create a new feature.

Hem

A hem feature folds the sheet metal part back on itself according to the selected settings.

The hem type can be changed from the Hem Options, on the command bar. Hems can be taken around adjacent edges and mitered, if necessary.

Hem

Closed Hem.

S Bend.

Curled Hem.
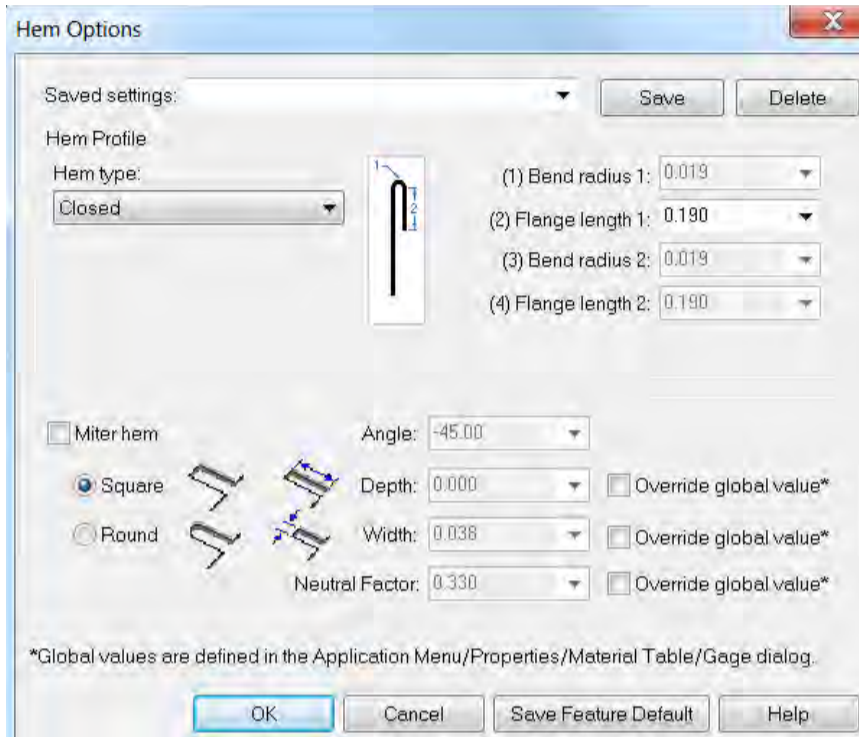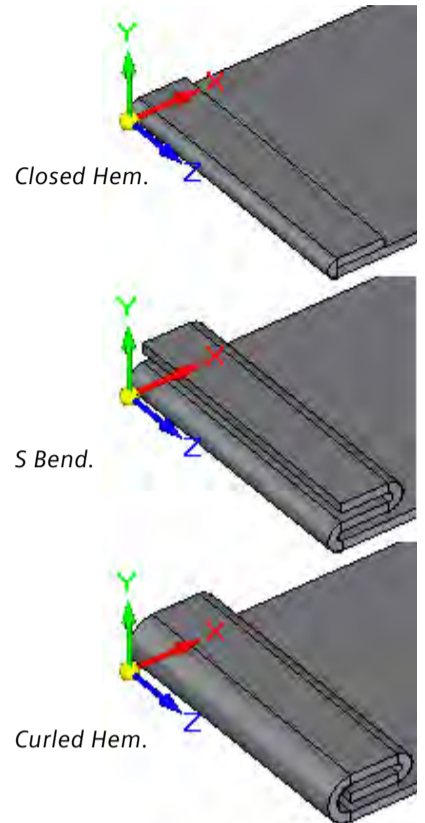
To edit a hem, select the hem from the PathFinder, or select a face of the feature in the graphics window, then click on the hem's name displayed as an annotation. This makes the command bar available.

To delete a hem feature, you can follow the same procedure: select the hem in the PathFinder or the graphics window and press the Delete key.
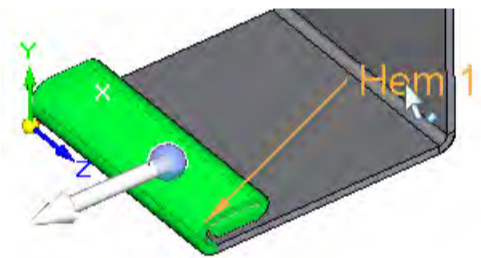
> *Note: You can use regular flanges to go around the three outer sides of a U shape with a flat bottom, but to get those three flanges around the inside, the flanges must automatically miter the corners to prevent overlapping. As a workaround, you can use the contour flange with a single line drawn to the inside, and this will create three flanges to the inside, and automatically miter them correctly. This example is saved in the folder of materials for Chapter 8, called CH8 3 mitered flanges.psm.*
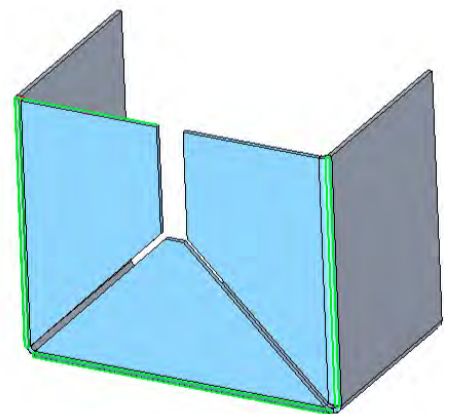
Bend

A bend feature can be added to a Solid Edge sheet metal part simply by drawing a line across a flat face. The line must be drawn before you can activate the feature.
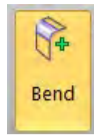
You would use a bend, rather than a flange, if you wanted to make use of existing material rather than creating more complexity for the flat pattern.
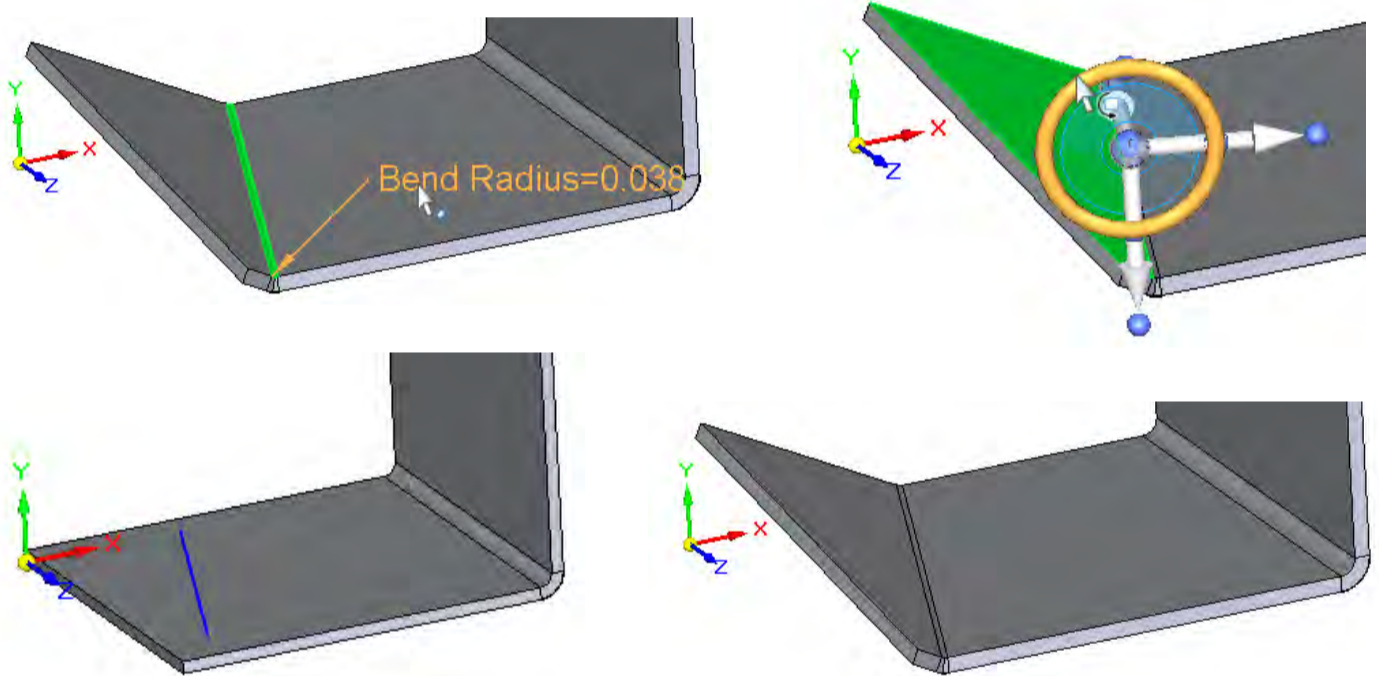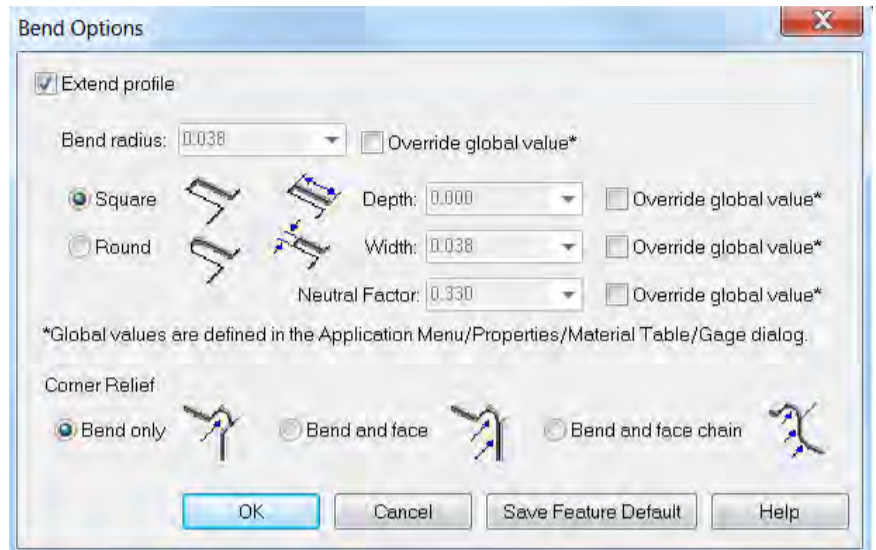
Bends are not listed in the PathFinder, but the flanges created by bend are listed. All bends are edited using the same techniques, not just sketched bends created with the Bend command. To edit a bend, select one of the cylindrical faces of the bend, and the software will display the annotation, Bend Radius = X. You can click on this annotation to edit the bend radius. To edit the bend angle, click on the flange affected by the bend, and move the Steering Wheel to the axis of the bend. This enables you to use the torus to change the bend angle (angle the flange).

Jog

A jog is a series of two bends in opposite directions that create a step in the sheet metal part. You could create a jog by simply making two simple flanges that bend in opposite directions, and in fact, that is the end result. When you look at the PathFinder, you will not see a jog feature, but rather two new flanges.

The Jog feature works much like the bend, requiring a line in a sketch. In fact, the Bend Options available from the Jog command bar are the same as the Bend Options available on the Bend command bar (shown above).
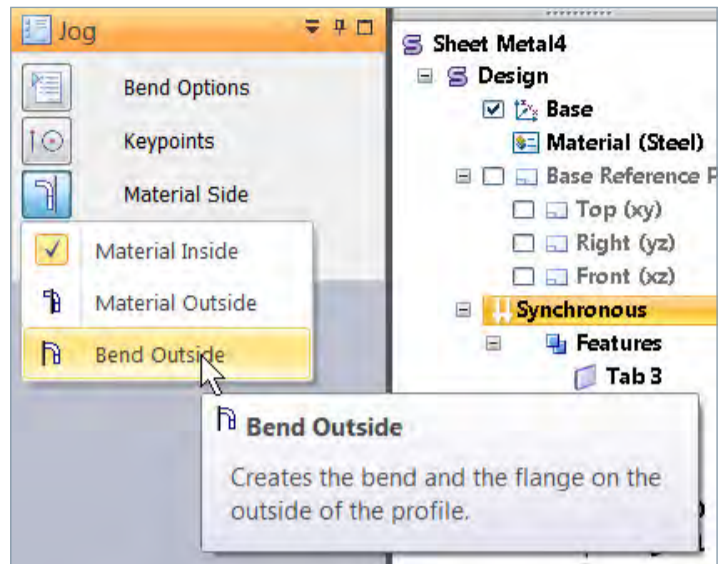
To create a jog:

1. Draw a line where you want the jog to start. The Material Side setting on the command bar lets you control the relationship of your sketch line to the bend.

2. A simplified Steering Wheel displays on the drawn line. The side of the arrow you click on will move up or down.
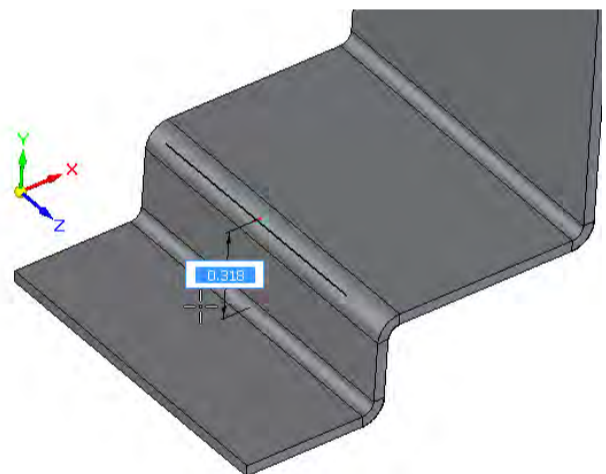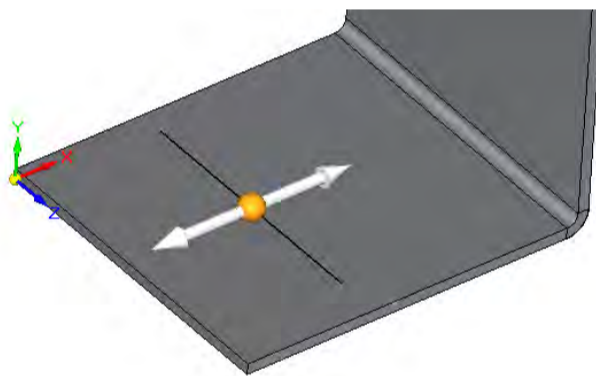
3. Specify the height of the jog step.

This operation adds material to the flat pattern. If you wanted to create a jog from the existing material of this flange without lengthening it at all, you would have to create two bend features.

Bend reliefs and corner reliefs

A bend relief is necessary when a bend ends in the middle of a flange. For example, the orange faces in the image to the right are bend relief faces.
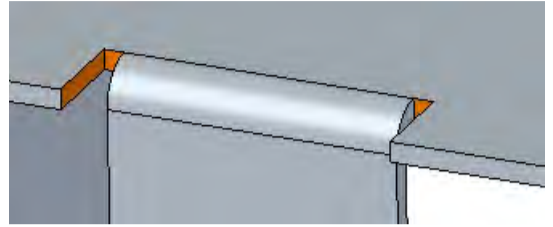
The default values for all the reliefs in a part are found in the Material Table, on the Gage Properties page (**Application Menu** -> **Properties** -> **Material Table**).

You can also access settings for individual bends by clicking the bend in the graphics window, then clicking on the bend radius, and the Bend Options, shown next, available on the command bar.

Solid Edge cannot accept a relief width of 0, but you can enter very small numbers, such as 0.0001". CAD programs that allow a relief type of tear are simply doing this behind the scenes with a very small non-zero number. Solid Edge actually does this with the Corner Relief command, but not with the Bend Relief command. This is one of those situations where it is possible in real-life manufacturing, but CAD cannot work with zero-thickness situations.
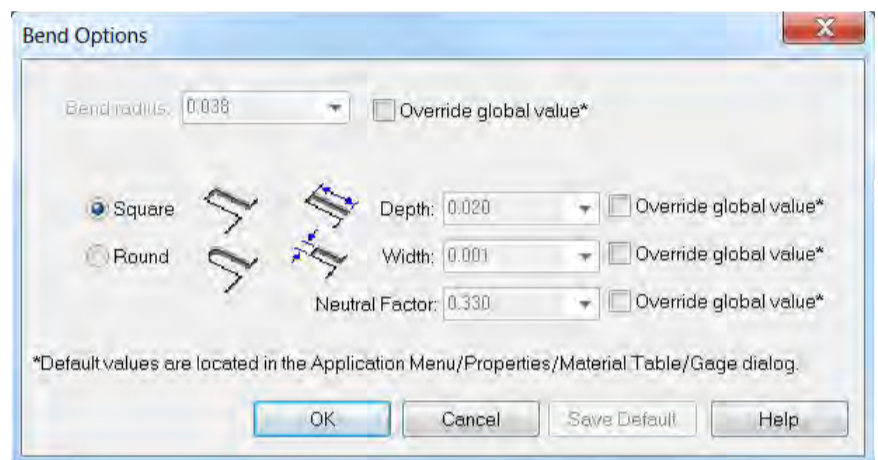
Corner relief

A corner relief comes into play when you have two or more bends that intersect at a corner. The effect becomes exaggerated with thicker material. Again, real-life manufacturing can have a section of two bends that overlap, and it naturally makes a rather complex area that cannot be easily accomplished in CAD.

A typical corner where two bends come together looks as shown to the right. This corner has no bend relief or corner relief. Sometimes your design calls for a tighter corner than this. The intent of this book is not to tell you how to design sheet metal parts – just how to use the available tools.
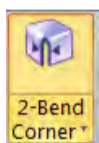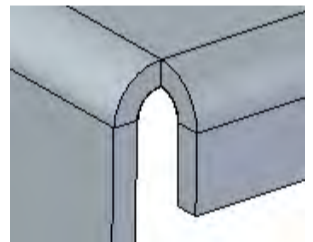
One way to tighten up this corner is to use the 2-Bend Corner command. This enables you to bring the inside edge of the two flanges together until they almost touch.

The drop-down list at the bottom of the command bar is the Corner Treatment setting. This controls how the small area where the two bends would overlap is shown. Again, this may look different in a real-world bend, and may not match the flat pattern, but it is shown simplified in 3D just because it would be more complex to show what actually happens when the bends collide.
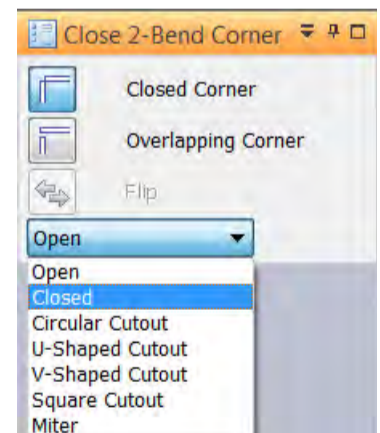
The Closed Corner option looks like this:
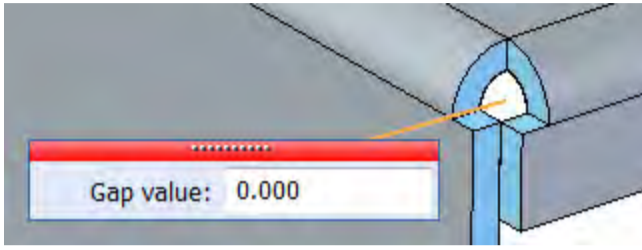
Closed Corner with Open treatment.
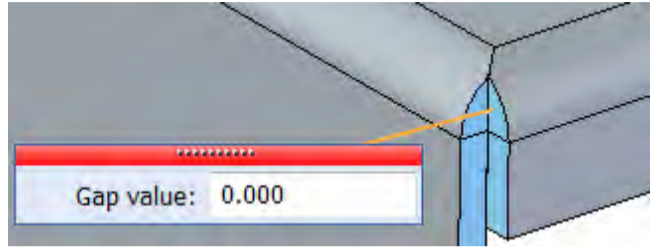
Closed Corner with Closed treatment.
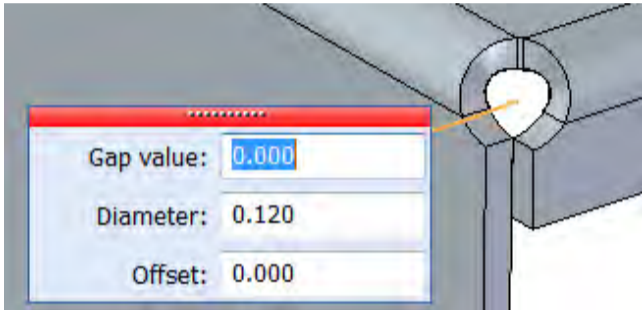
Closed Corner with Circular Cutout treatment.
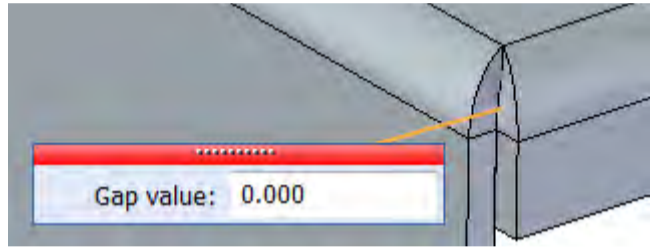
Closed Corner with Miter treatment.

The Gap value shown is 0.000", but the gap, as mentioned earlier, cannot be a true zero gap between the two edges. The value shown is only three places of the actual value, which is 0.00011". If you zoom in very close, you can measure the gap to verify this. This is a little workaround that makes it work for CAD and also makes it work for your drawing dimensions and tolerances.

The Overlapping Corner option makes one flange overlap the other. The treatment conditions are all for the bend corner area, and are the same between the Closed and Overlapping flanges. The Overlap ratio can vary from 0 to 1, where 0 is like the Closed Corner option, and 1 is complete overlap, shown on the right.

The part used to create these screen captures is available in the folder of materials for this chapter.
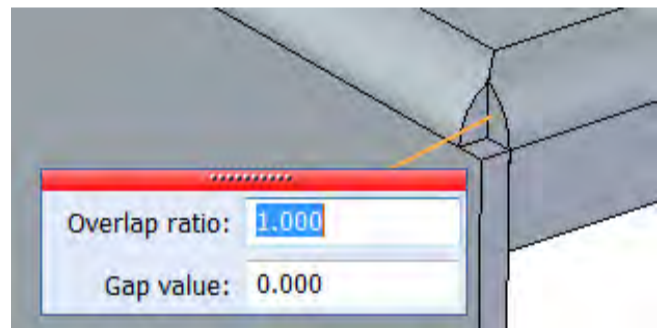
Overlapping Corner with Closed treatment.

Creating a cut feature

You would expect a cut feature to work largely the same in sheet metal as in normal modeling. And it does. With two exceptions:

- The cut does not create a feature in the PathFinder. It is just geometry, with no special properties after it is created, so it can be edited synchronously. The only thing you need to do with this information is to be aware of it so that when you create a cut feature, and it doesn't show up in the tree, you'll know that it's not a mistake.
- The shear is always perpendicular to the sheet face, even when the cut feature hits the face at an angle. In the following image, notice that the cut faces on the angled face on the left are cut through normal to the sheet face, rather than cut at an angle.

This behavior is controlled by the first option in the command bar, Face Normal Cut Types.
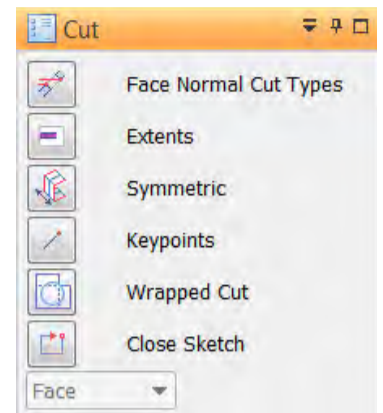
Wrapped Cut

Another cut type worth noting on the Cut command bar is the Wrapped Cut. You can use this command to make a cut that goes across bend lines.

To make a wrapped cut:

1. Start with a sketch on your 3D part that represents where you want the cut made.

2. Select one face as the sketch plane.

3. Sketch as if you are sketching on the flat pattern, even though you're sketching on the 3D part.

4. Click the Wrapped Cut button on the Cut command bar, and Solid Edge automatically recognizes that the cut goes across the bend, and flattens the bend temporarily to complete the rest of the cut.

5. When you right-click again to accept, the temporarily flattened bend re-forms, and your cut is complete.

This is one of the situations in which designing sheet metal in 3D becomes a little confusing. The hole is punched in the flat, and then bent. Solid Edge makes dealing with the situation automatic and easy as long as you know what to expect.
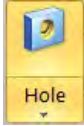
The part file used to make this example is saved in the folder for the Chapter 8 materials with the name CH8 wrapped cut, angled cut, psm.
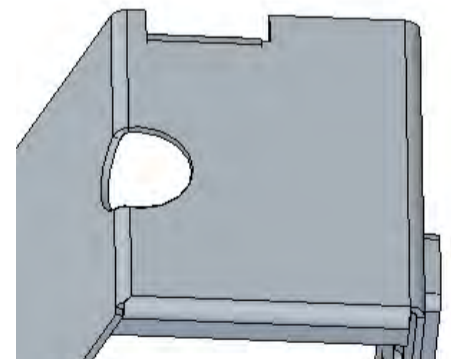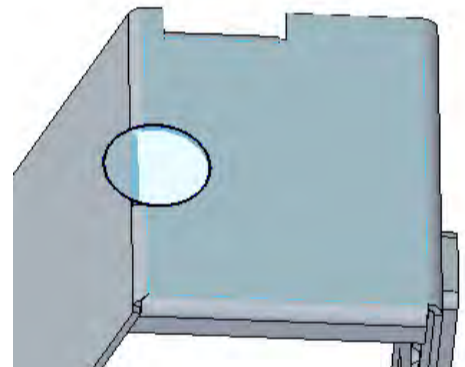
Note: Other features can take advantage of this type of technique of going across bends, but require the Unfold feature to do it. Unfold, as of this writing, is only available in ordered mode.

Holes, slots and library features

To place a hole on a sheet metal part:
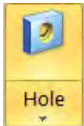
1. Click the Hole button on the Sheet Metal group.

2. Place the hole to accept the default hole type, or:

   a. Using the drop-down list next to the Hole Options button on the command bar, select a saved hole.

   b. Using the Hole Options button on the command bar, specify a new type of hole and optionally save it for quick access from the drop-down later.

If you place multiple holes in a single command, those holes will be listed together in the PathFinder. If you make the same hole later during a different session of the Hole command, it will be put into a separate Hole collector folder. This enables you to edit all the holes created together at the same time, for example to change diameter. In the image on the right, notice that for Hole 1, there is a collector, and then indented is the feature. Note that the collectors and the features can have the same names.

If you want to separate one instance of a hole from a group of holes, just right-click that instance in the PathFinder or graphics window and select Separate. Solid Edge will move the instance to its own new Hole collector folder. After you separate a hole from a group of holes, you cannot reconnect it. Likewise, if a hole has all of the same properties as other holes, but is in a different collector, it cannot be grouped with the other similar holes. This is motivation to get holes grouped correctly from the beginning.
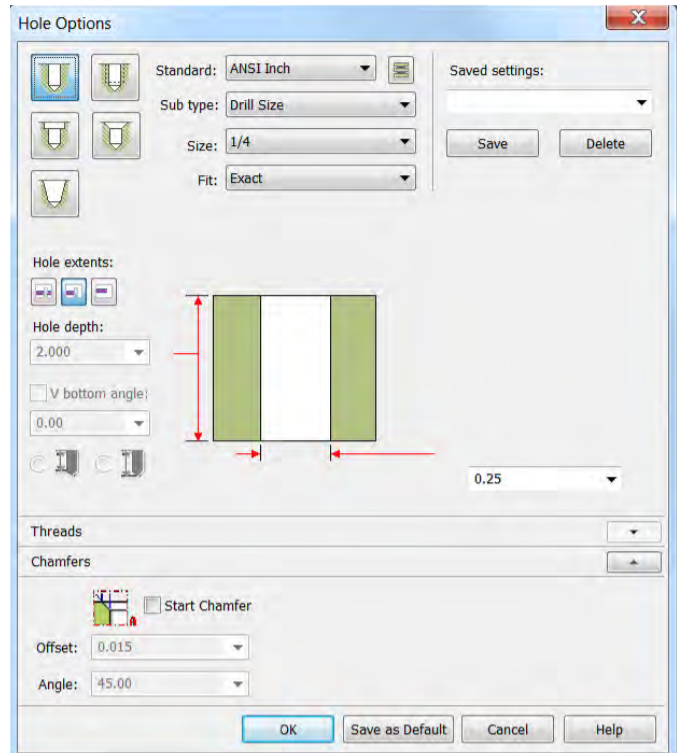
Note: If you are having trouble placing a hole, press F3 to unlock a previously locked plane. This is a common source of error that more experienced users deal with instinctively.

The slot feature requires an associative centerline sketch chain with tangency.

To create a slot:

1. Create an open sketch with no sharp corners. This sketch represents the centerline of the slot.

2. Click the Slot button in the Sheet Metal group, under the Hole drop-down list.

3. Select any element of the centerline sketch, understanding that the entire tangent chain will be selected.
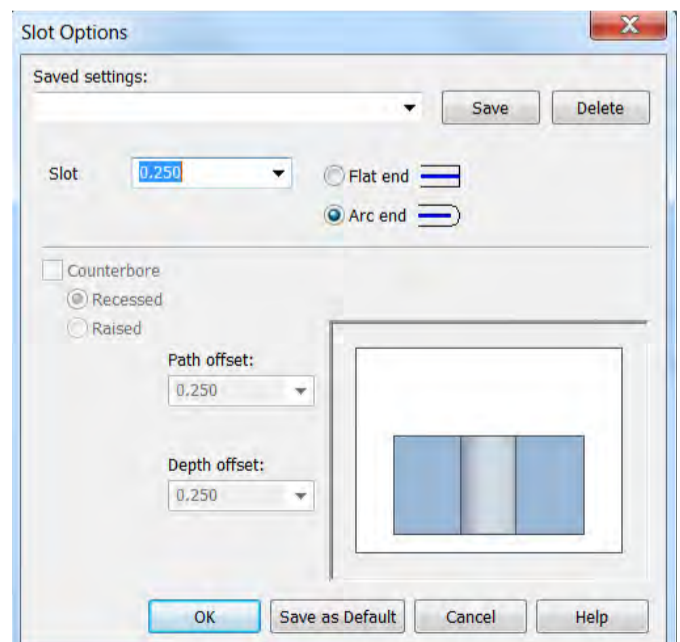
4. Make sure that the Steering Wheel arrow is pointing in the direction of material removal.

   Note: The Counterbore settings are not accessible for sheet metal parts, only for general parts. To sketch a line, then tangent arc, you can use the A key to switch to tangent arc. Be sure to look for the IntelliSketch icon for the location where the second line is tangent to the arc. This will be a big help in sketching open loop tangent chains.
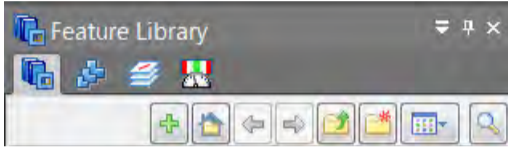
The Feature Library in Solid Edge can contain ordered and synchronous, but each individual folder can only contain one type of features.

The library, as installed with the default Solid Edge installation defaults, is probably not the configuration you will want to work with in your production environment.

To start a new area as a library, use the New button to create a new folder where you want to put the new library, and start by copying and pasting features directly into the Feature Library panel.

You can also just start using an existing folder as a library just by using the Library Feature tools to browse to it. To get some practice with this, follow this workflow:

1. In Windows Explorer, copy the Feature Library folder out of the material for this chapter, and paste it in an easily accessible area on your computer.

2. Show the Feature Library in the Solid Edge interface:

   On the **View** tab -> **Show** group -> **Panes** drop-down list, click the Feature Library button.

3. Use the tools on the Feature Library to browse to the folder you just copied to your computer.

4. Make sure you have a synchronous sheet metal part open, and double-click the synchronous Features folder in the copied Feature Library.

   Note: You can use the folder drop-down list to assist in browsing.

5. Right-click in the white space in the synchronous Features folder, and select Preview. This displays a small window at the bottom of the pane that shows a small preview of the selected feature.

6. Click on the Louver1.psm feature, and drag it into the graphics window. Make sure to release the mouse button inside the graphics window.

   a. When you release the mouse button from the drag, the feature will be attached to your cursor.
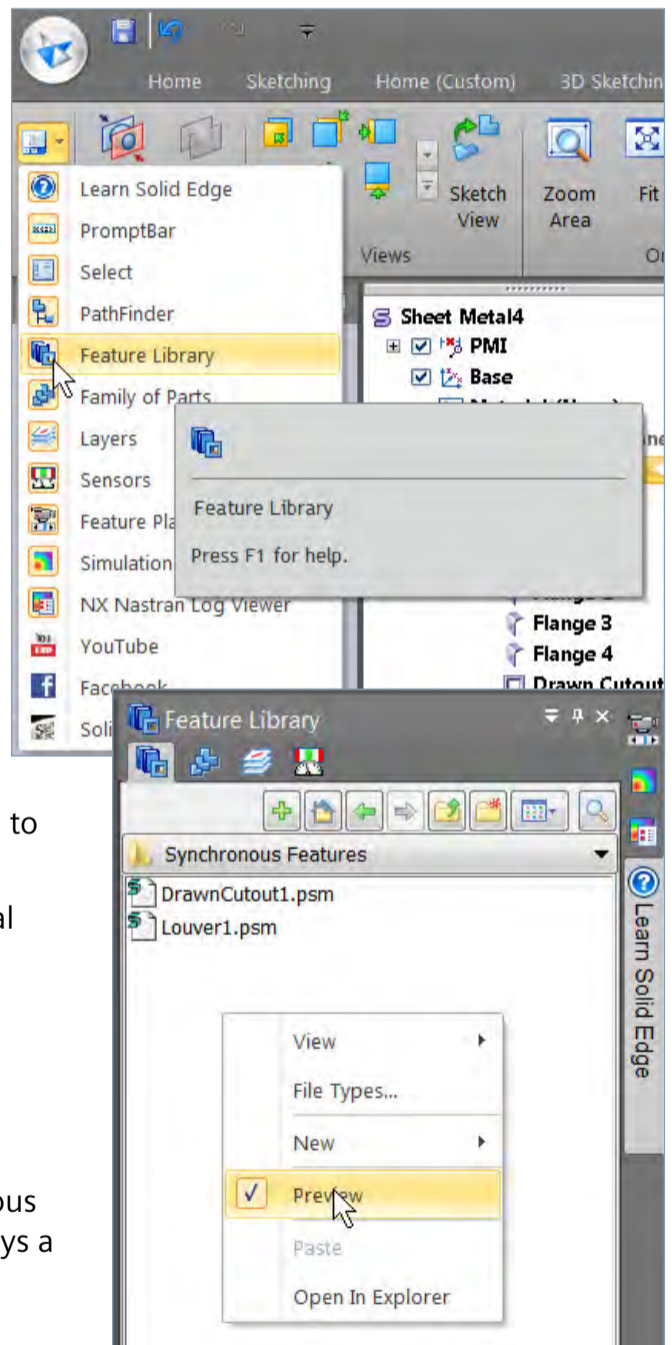
b. Place the cursor over a flat face of the sheet metal model, and press the F3 key. Move the cursor to another flat face, and press the F3 key again. When you have a place where you want to put the Louver feature, click the left mouse button.

c. Press the Esc key to exit the function.

Notice that the Louver feature is pasted to the model as a different color. This is actually a surface model copy of the library feature. It is not a solid yet.
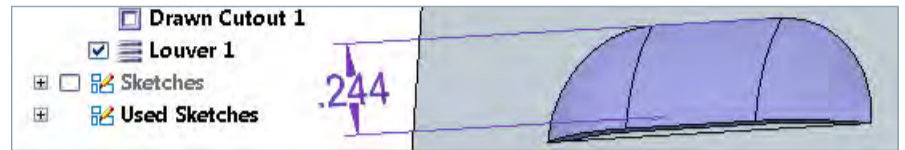
Before attaching the Louver.

d. Right-click the Louver feature either in the graphics window or the PathFinder, and select Attach. This will permanently attach the surface model onto your solid model, and the faces of the louver will turn the same color as the rest of your solid model. When the faces of the feature are merged (attached) into the solid, there will no

After attaching the Louver.

longer be a checkmark in front of the Louver1 entry in the PathFinder. The checkmark gives you the ability to hide the Louver when it is still just an unattached body.

Note: This is the same technique that you use when you copy and paste features within a model. You can think of the Feature Library like an extended clipboard. You use copy and paste to get items into the library, and the drag-and-drop operation is essentially another paste operation. The Attach function merges a set of copied faces into the rest of your solid model.

Deformation features

Some features in Solid Edge sheet metal are not just straight brake press operations. These are generally any of the features under the Dimple drop-down list in the Sheet Metal group (Home tab in the Sheet Metal environment).

Further, all of these *deformation* features (except for Louver, Gusset, and Emboss) are different from other Synchronous features in another way – they are all driven by associative sketches. We have spent a lot of effort in this book trying to get history-based users to forget about associative sketches, so we have saved this topic for near the end of the book when you probably are comfortable with the idea of leaving sketches behind, to reintroduce the concept for these few specialty sheet metal features.

Deformation features do not handle the stretching or thinning of material due to deformation. The volume of the CAD part will be greater after the deformation operation than before. In the real world, the material thins slightly, and the volume remains the same.
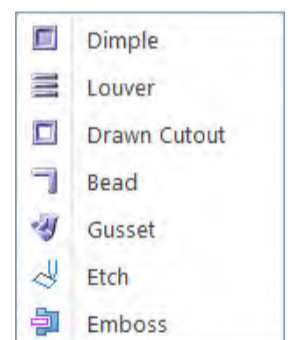
Creating a dimple

In Solid Edge sheet metal, a dimple is simply a sketched shape that is stamped into the part a certain distance, leaving the back side of the part with material removed. For example, this barbell shape with two connected circles was sketched onto the part, and raised a certain distance from the surface, with the material remaining constant thickness on the other side.
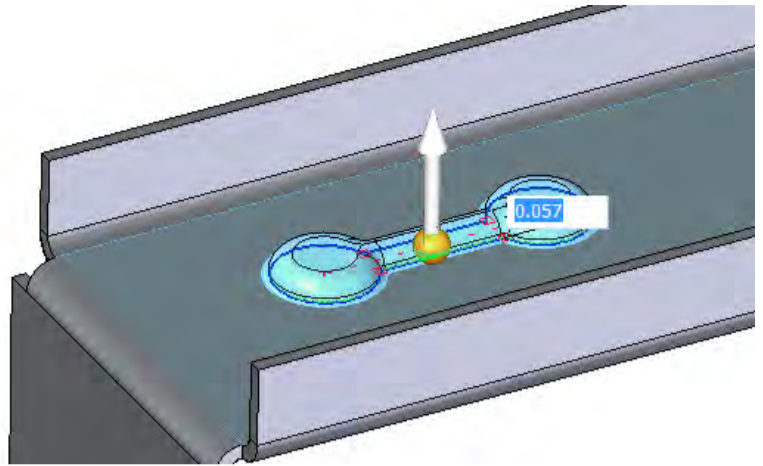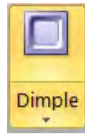
Editing a dimple

To change the shape of the feature, you can edit the underlying sketch, or sketch it again. Let's walk through editing the dimple feature. At the end of the process we will alter the sketch to change the shape.

Selecting the dimple either from the PathFinder or the graphics window highlights all the faces of the feature, and displays the Steering Wheel. You can use the Steering Wheel to move or

Select the feature and use the Steering Wheel to move it within the face.

rotate the entire feature on the face. Remember that the arrows move in one direction only, but the disk (tool plane) allows planar motion.

Selecting the Dimple feature and then clicking on the Dimple 1 annotation that appears allows you access to the Edit Profile link and the modify box for the depth of the dimple feature.

Creating a bead

A sheet metal bead feature is similar to the dimple. You start with a sketch. The bead sketch is required to have all tangent entities (no sharp corners).

To create a bead:

1. Create a sketch on a sheet metal part with no sharp corners. It can be an open or closed loop.

2. Click the Bead command.

3. Select the sketch elements, loop or chain.

The bead feature requires that you start with an open or closed loop with no sharp corners.

4. Use the Steering Wheel to determine the side of the part to which the bead will add material.

5. Use the Bead Options from the command bar to specify the cross-sectional shape and the end shape of the bead.

6. Right-click to accept result.

Editing the bead is the same as editing the other deformation features. Select the feature, click Bead name, click Edit Profile, make changes to profile sketch, and then use the Exit Sketch button and right mouse button click to accept changes. To edit the Bead Options, select the feature and click the Bead name, then use the command bar to access Bead Options.
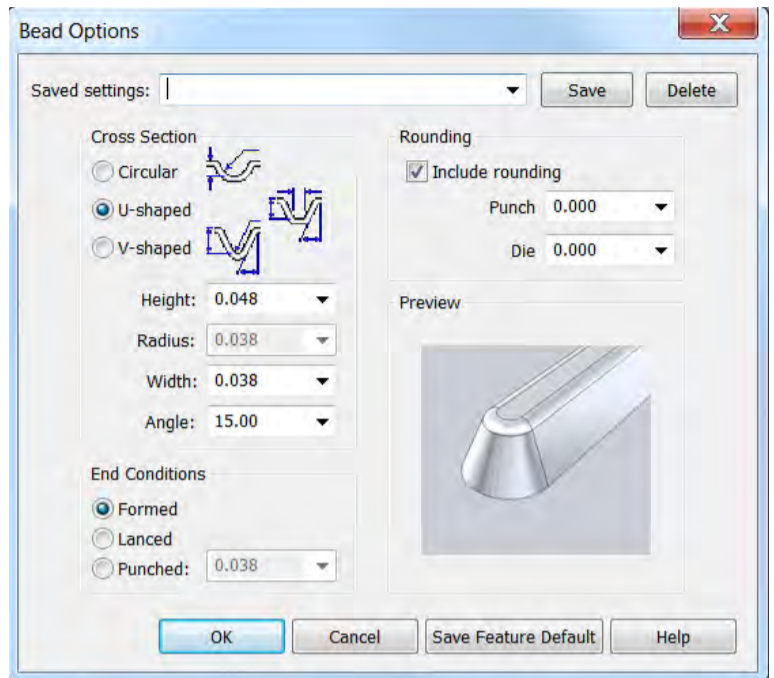
Creating a drawn cutout

A regular cutout feature just cuts through the part, but a drawn cutout pushes some material through with the cut. This feature does not regulate how much material you add as part of the draw part of the operation, and like the other deformation features, it does not thin any deformed material. For these reasons, it is entirely possible for this cut feature to actually add mass to your part – which is to say that it may make geometry that is entirely impossible to manufacture without welding on extra material.
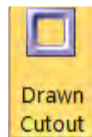
Editing the drawn cutout is the same as editing the other deformation features.
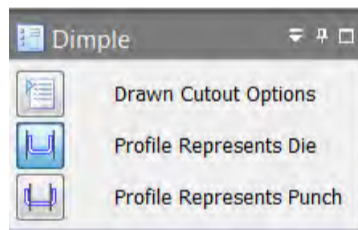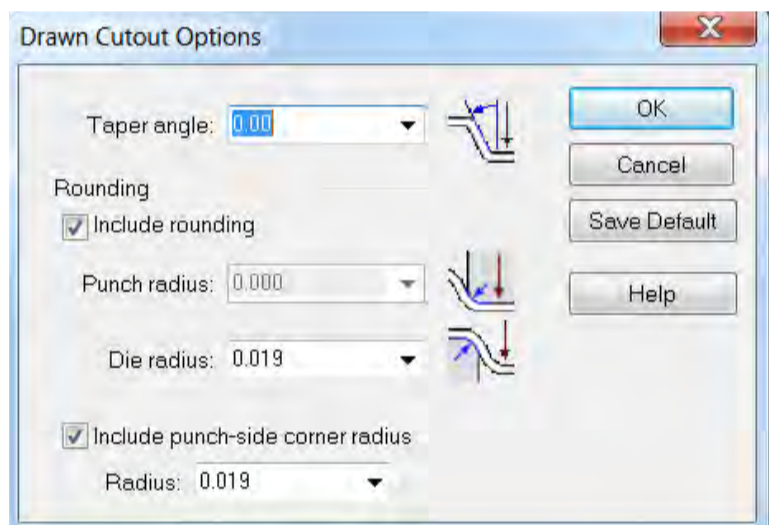
The Drawn Cutout command bar is mislabeled as Dimple.

Creating a gusset

To create a gusset feature, you must first have a sheet metal model with at least one bend in it.

To create a gusset:

1. Click the Gusset button on the Sheet Metal group.

2. Click the Gusset Options button on the command bar.

The sketch can represent the die (female) or the punch (male).
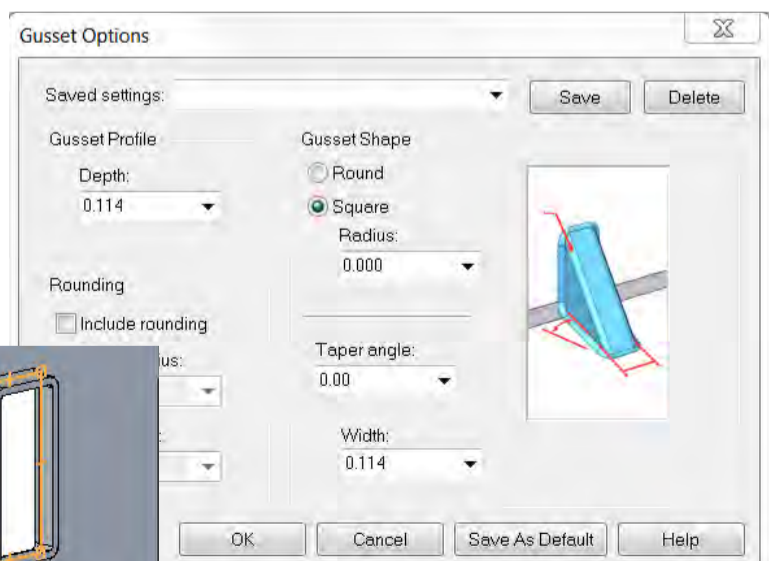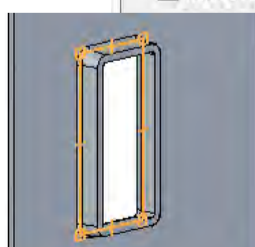
3. Set the options for your gusset, and save the settings or recall the settings from the drop-down list if this will be a set of gusset settings you use frequently.

4. Use the Patterning drop-down list to select whether you want a Single gusset, or to pattern with Fit, Fill or Fixed method.

5. Click on a location for the gusset along the inside of a bend.

6. Use the dimension that appears, to locate the gusset along the bend, or the box that accepts the number of gusset instances.
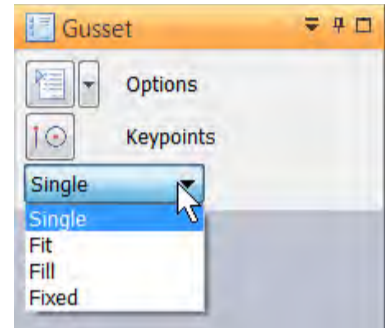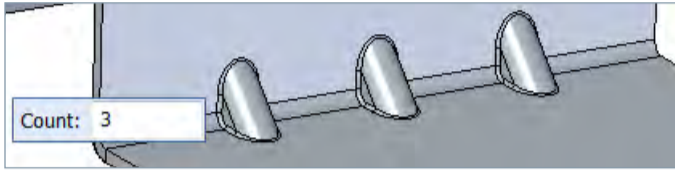
Editing a gusset is similar to editing other deformation features:

1. Select the feature from the graphics window or PathFinder.

2. Click the name of the Gusset feature that appears in the graphics window.

3. Change the pattern number, dimension or the Gusset Options in the command bar.

Creating an etch feature

Etch enables you to show an etched shape or text on the surface of the sheet metal part. This does not violate the constant thickness requirement for sheet metal. You can use standard sketched shapes or insert a text profile. The Text Profile command is on the Sketching tab, Insert group.

Most of the functions in this tool are self-explanatory, except how to rotate the Text Profile. To rotate text:

1. If the Etch feature has already been created:

   a. Select the Etch feature in the PathFinder.

   b. Use the Steering Wheel torus to rotate the feature.

2. If the Text Profile has been created, but the Etch has not:

   a. Click on the name of the Etch feature that appears as an annotation in the graphics area.

   b. Select the text profile in the graphics window. Solid Edge displays the Edit Definition command bar.

   c. Click the arrow to the left of the word Modify, and select Move. Solid Edge displays the Steering Wheel in the middle of the Text Profile.

   d. Shift+click on the tool plane of the Steering Wheel until the torus is on the same plane as the Text Profile (you may have to Shift+click on the tool plane twice).

   e. Use the torus to rotate.
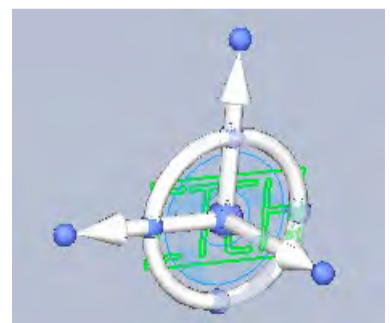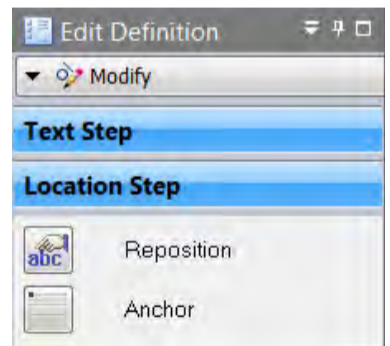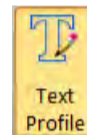
Creating a louver feature

The louver feature can be placed freely on a flat face of a sheet metal part without a sketch. You may need to set the Louver Options first to make the louver fit on the piece with which you are working.

As with other deformation features, you can use the Steering Wheel to reposition or rotate the feature. You can dimension, copy and pattern it as you do other features.
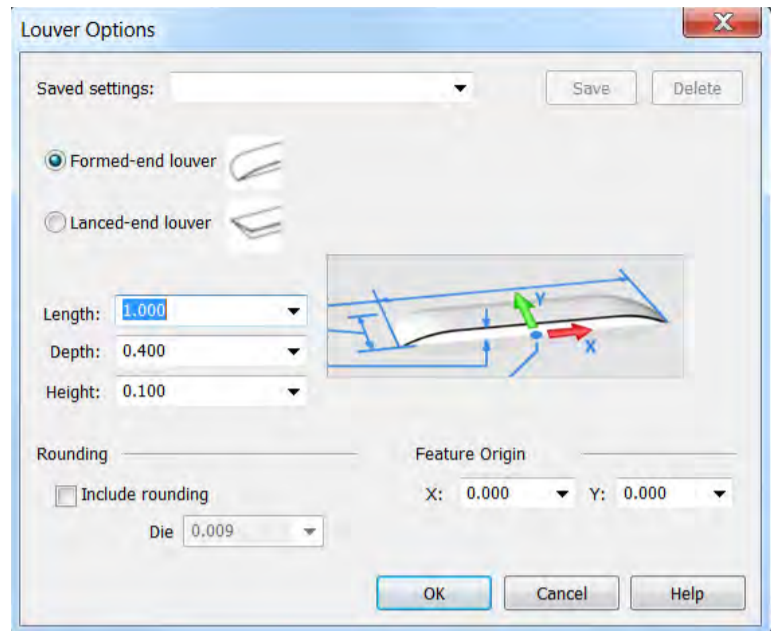
Working with Flatten and Flat Pattern

The Flatten mode is on the Tools tab, in the Model group. This is used to generate a flat pattern for the model. This is useful when working on a sheet metal part to check and make sure your flat pattern does not have any undesirable traits, such as overlapping areas in the flat. It is also how you generate flattened geometry for use in drawings and computer controlled sheet metal manufacturing methods.

To avoid confusion, it should be mentioned that the Flatten tool creates the Flat Pattern collector in the PathFinder, and the Flat Pattern button creates the Flatten entry.

The work flow for creating a Flatten entry for a sheet metal part that has no Flat Pattern collector is as follows:

1. Create a sheet metal part in either synchronous or ordered with at least one bend.

2. Click on Flatten in the **Tools** -> **Model** group (this creates the Flat Pattern collector).

3. Use the Flat Pattern CommandBar to control the following:

    a. The Flat Pattern Treatment is the settings used for a single Flatten feature.

    b. Select a face that will face "up" in the flat pattern ("up" is considered the positive Z direction). The Prompt Bar prompts you for this selection.
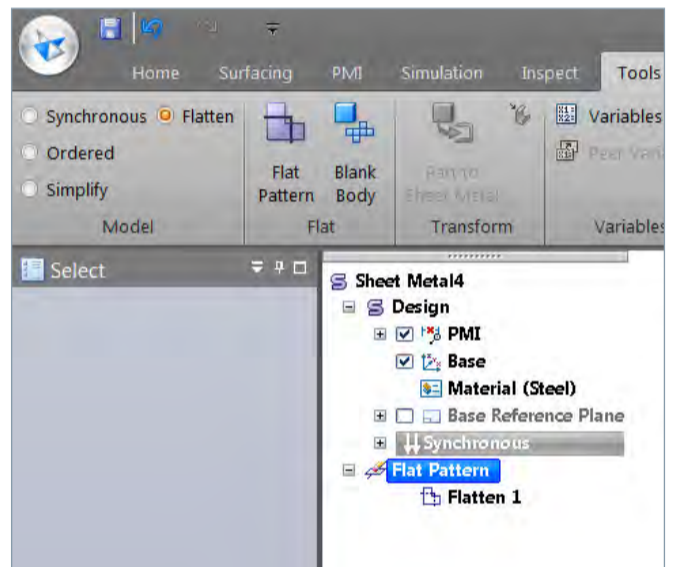
c. Select an edge that will specify the X direction and the origin.

   i. Hovering the mouse over an eligible edge (eligible means that it does not create a conflict with the Z direction) causes the edge to highlight orange and display a dot on one end, as shown in the figure.

   ii. The edge will be reoriented to the X axis, and the dot will be located at the origin end.

To shift back into synchronous mode where you can continue working on building your model, simply click after the Flat Pattern has been created.
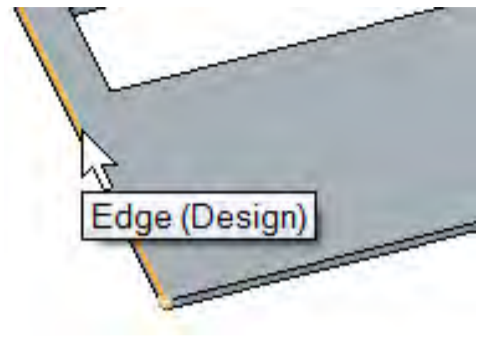
Edge selected for X orientation has an orange dot on one end which specifies which end of the edge will be placed at the origin.

Transforming general models to sheet metal

Solid Edge uses a different file type for sheet metal parts because there are several requirements and operations which are specific to sheet metal parts: flat patterns, constant thickness requirements, special set of tools, and so on.

Sometimes you may change your mind part way through a design cycle, and decide to make some parts using sheet metal techniques, where you had previously used general modeling techniques to make a machined, cast or even fabricated part.
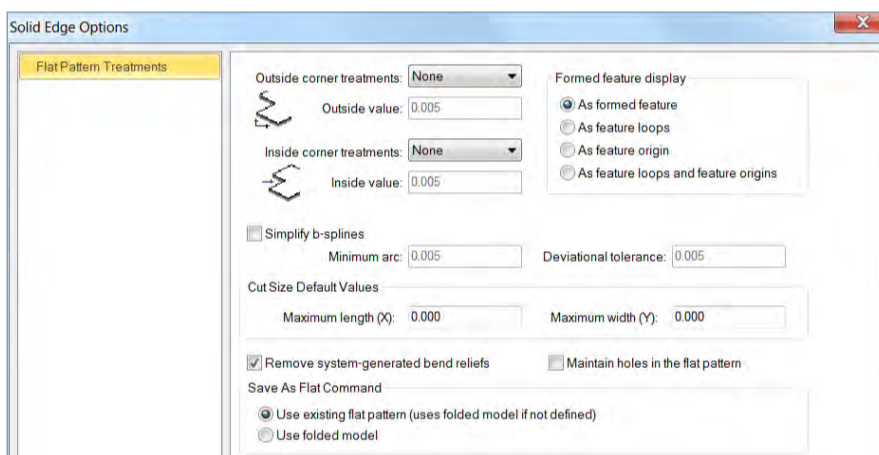
If you decide to change a model over to a specialized sheet metal part, it still has to follow the sheet metal rules. The result must have a constant thickness, with the ability to be flattened without deforming any material.

You can do this type of operation with either native Solid Edge parts or imported data, as long as all the requirements are met.

Look at the part called CH8 transformation.par in the folder that contains the working data for this chapter. This is a solid block of material, made just the way we have made previous general synchronous parts in this book.

To make the transformation to sheet metal, here are the general steps required. (Your specific part may require more steps depending on the complexity of the part).

1. Start with a general synchronous part, native or imported.

2. Use the Thin Wall command to make sure it has constant wall thicknesses.

   a. The Thin Wall command enables you to remove a face, which is generally for making plastic parts, but in making a sheet metal part you may have to remove multiple faces in order to make the part flattenable.
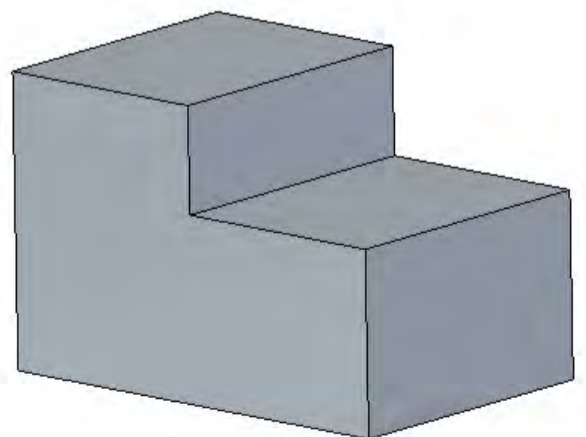
b. There will probably be multiple ways to flatten the part, some may be more awkward than others or more efficient from a manufacturing point of view.

c. Use a thin wall thickness value that matches the sheet metal thickness you require.

3. Once you have the solid part hollowed out with the Thin Wall command, it still cannot be flattened because the corners are all attached.
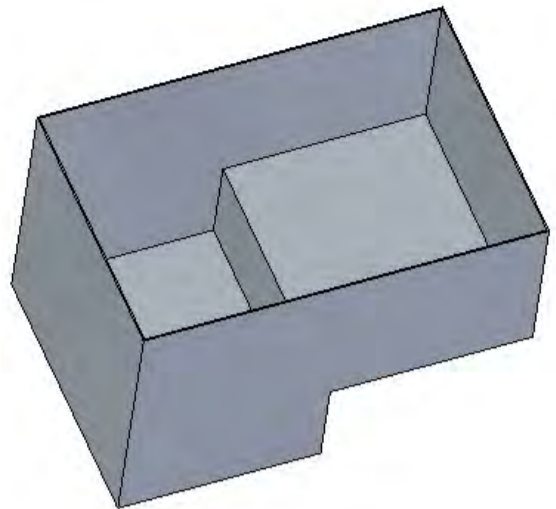
a. Click **Tools** -> **Transform** -> **Thin Part** to Synchronous Sheet Metal tool.

b. Select an exterior face to use as a reference for flattening the part. The software may return an error message concerning Rip Edges. Notice the model cannot be flattened yet because the edges are all *welded* together.

c. Select the Select Rip Edges Step, and select the model edges as shown next. You can select interior or exterior edges, but whatever you do, you should be consistent about it.
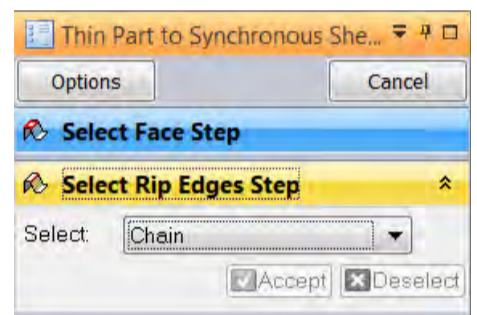
The green face is highlighted.
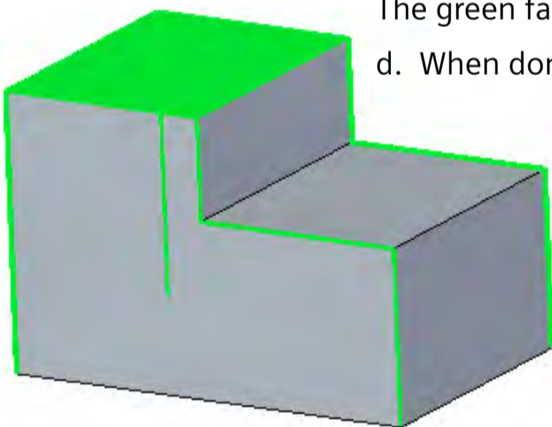
d. When done, click Accept.

e. Flatten the model by going to **Tools** -> **Model** -> **Flatten**, and following the Prompt Bar instructions:

i. Click on a face to be oriented upward in the flat.

ii. Click on an edge to define the X-axis and origin.
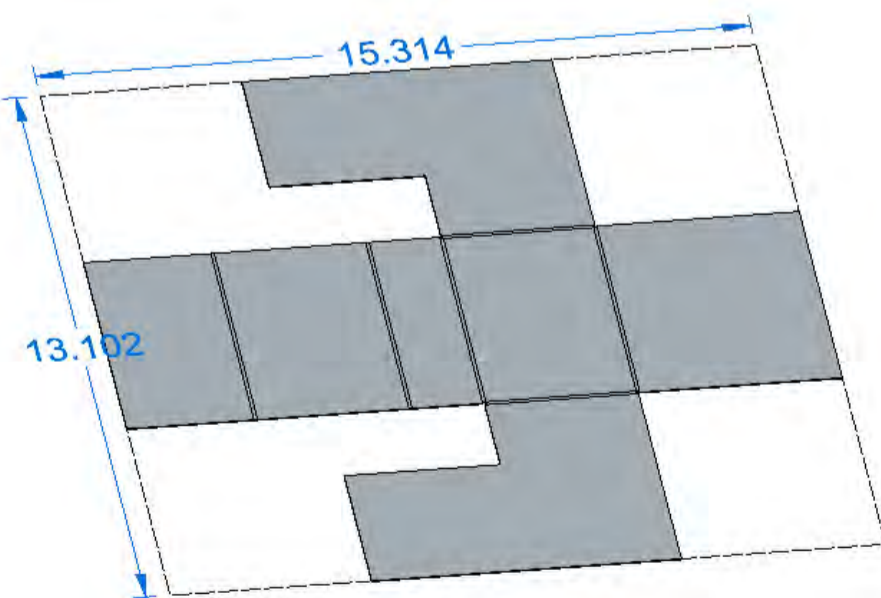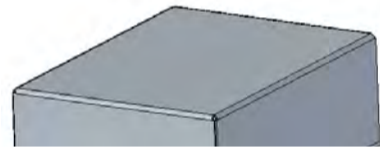
The completed model is saved under the name of CH8 transform3.

When to shift to ordered mode

Synchronous sheet metal functions were developed later than most of the other tools in synchronous technology. For that reason, some sheet metal functions that have existed for years in the ordered environment have not yet been developed on the synchronous side. Two of those functions are deformation features across bends and Lofted Flange. Both of these tools work in Solid Edge, but do not yet work in synchronous mode. Therefore, if you need either of these features, you should model those parts in ordered.

Summary

Solid Edge synchronous sheet metal is powerful and incredibly easy to use. From click-and-pull flanges to gussets that you just click-and-place, building manufacturable sheet metal parts from the ground up is easy. When combined with synchronous assembly techniques, the ease of use combined with the power is astonishing. The interface is meant to get out of your way until you need it, and the available options are well thought out and show a deep understanding of sheet metal manufacturing processes.

Chapter 9

Synchronous assemblies

Synchronous assemblies may be a strange title for this chapter, because in truth, there is no such thing as a synchronous assembly. All Solid Edge assemblies use the same file extension, *.asm. You only need one assembly template. There are no settings or options in the assembly document that make it either synchronous or ordered. Solid Edge® software assemblies can contain synchronous parts, or ordered parts or both. The phrase synchronous assemblies is sometimes used when synchronous techniques are used to edit synchronous parts within a Solid Edge assembly.

Note: There is a <u>video</u> that accompanies the material for this chapter.

Synchronous technology used in assemblies takes the power that we have already demonstrated in synchronous parts, and multiplies it by allowing you to edit multiple parts at once. For example, you can select faces of three parts, move them up to a face of another part. Grab a hole that goes through multiple parts, and move it – without setting anything up beforehand or making any relationships between external files. You can directly use faces or edge outlines from other parts in the assembly to make new features in the current part – without costly in-context or interpart links.

Even more power comes from the ability to reuse data. In individual parts you can reuse any set of faces you can select, without regard for which faces belong to which features or what dependencies those features are involved in. In assemblies, you can replace a part with a copy of itself or another part, and change it on the fly in the context of the assembly. Some of this capability has nothing to do with synchronous technology, but it is part of Solid Edge assemblies, and powerful nonetheless. Solid Edge assemblies have a lot of power whether you use ordered parts, synchronous parts, or mix and match methods and part types.

Editing in the assembly

Editing multiple parts at once in the assembly without any prior setup is one of those things that history-based users really need to see in action. We are going to use the assembly shown to the right to demonstrate. You might recognize it as the imported motor from the chapter on imported data. We've hidden a few parts because the parts you really want to see here are the light blue piston rod, the gray clamp and the orange crank shaft. We will also use the green cam shaft and gear part.

These parts are not created specifically for the edit that we are going to make – they came to us from an import. Ideal parts would have the rounds created as separate ordered features. The
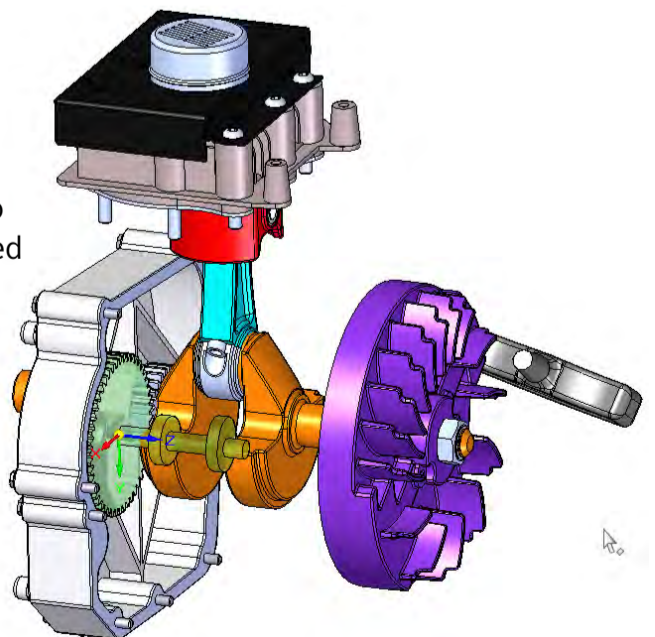
reason for that is to avoid consuming topology, as was discussed in a previous chapter. The worst that can happen when the rounds are ordered features are that an individual round feature might fail, as opposed to the entire synchronous edit failing.

The edit that we are going to perform will involve three separate parts, and at the end maybe we'll show off a little bit and include a fourth part.

Before we step through the edit, there are a couple of assembly related settings you need to know.

Part and Face Priority

On the Home tab, under the Select command are two options, called Face Priority and Part Priority. In the assembly, when you click on a part in the graphics window, your meaning is ambiguous to the software. You might mean to click on the part itself to do part-level operations such as Open or Replace, or you might mean to click on the face to do face or feature-level operations such as edit.

When working in the assembly with parts, you can specify your intention using these two options. Your cursor will highlight things differently, depending on the option you choose. With Part Priority, when you hover over a part, the entire part is highlighted when your cursor hovers over it. With Face Priority, when you hover over a part, only the face is highlighted, as shown on the right.

Activation

The second setting you should know about when working in assemblies, is the Activate option. When you activate a part, Solid Edge loads all of the data (as opposed to only loading the visual data) and makes it available for editing. You cannot edit an inactive part in the assembly, so before you continue, you will have to activate all the parts you want to work with. The Activate tool is in the **Home** tab -> **Select** group, and the icon looks as shown next.

You can also activate a part or multiple parts by selecting them (in the graphics window or in the PathFinder), and selecting Activate from the ribbon, or from the shortcut menu.
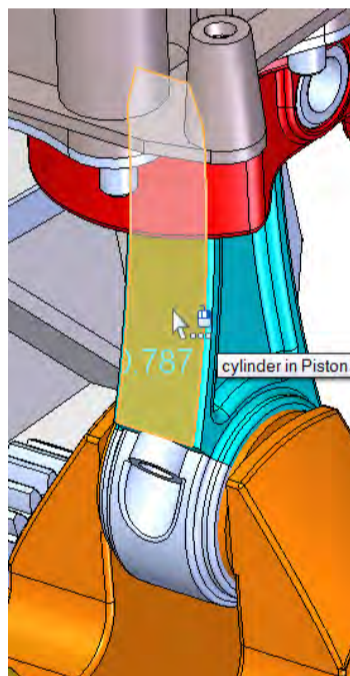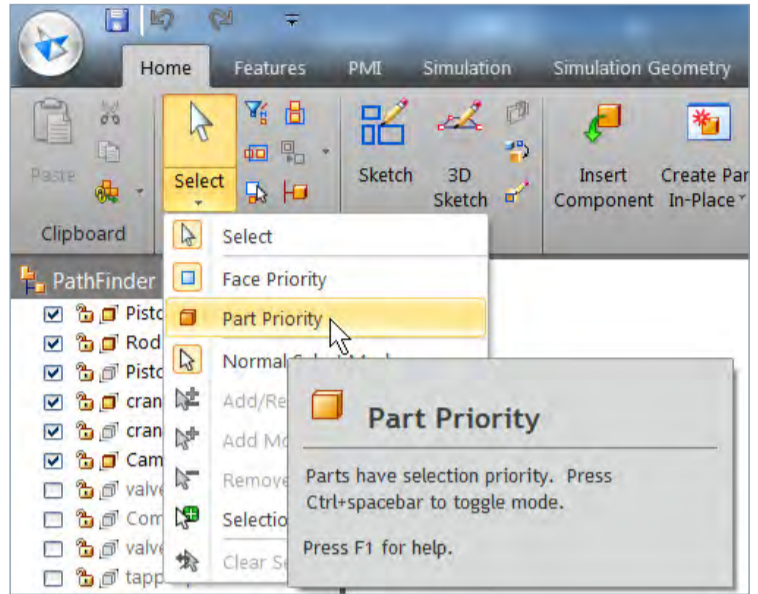
Face Priority helps you select faces and make edits when working in an assembly.
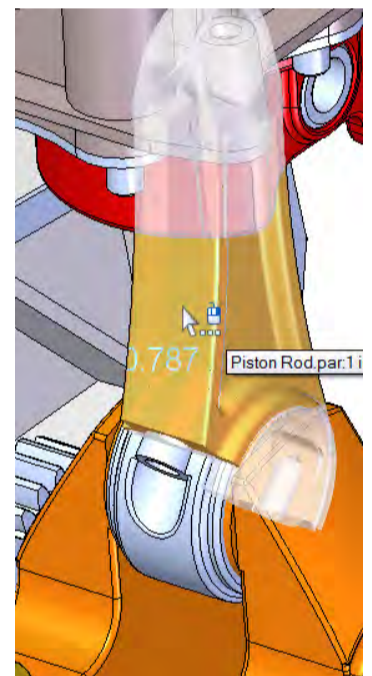
Part Priority helps you select parts for part-level operations within an assembly.
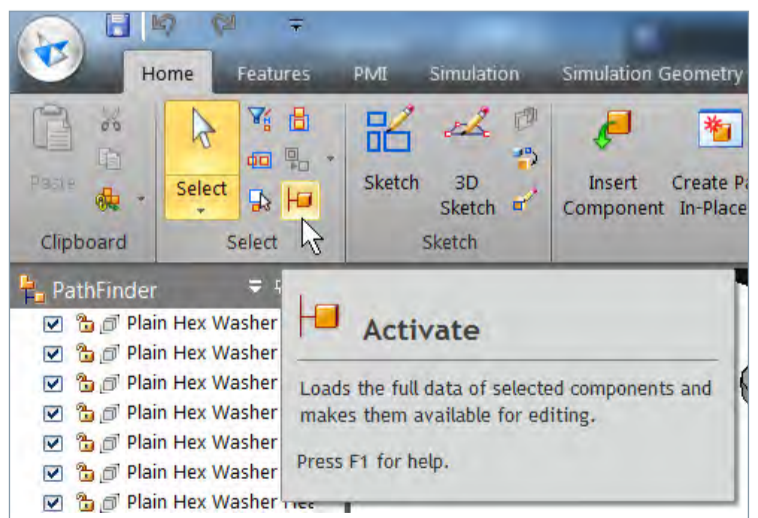
Tutorial: Editing parts in the assembly

The rest of this chapter is going to demonstrate some common synchronous techniques that you can use in assemblies. The assembly called motor subassembly ch9.asm is available in the materials for this chapter. It already has the parts colored or hidden in the state that you need for this tutorial/demo.

1. Open the assembly called motor subassembly ch9.asm from the folder containing material for this chapter.

    Use the document type filter in the lower right of the Open dialog to find Top Level Assembly Documents.

2. Switch to the Right view (Ctrl-R). You may need to use Shift+right arrow to orient it as shown.

3. Start by activating the crank (orange), piston rod (light blue) and the piston clamp (light gray). To do this, Ctrl+select all three parts in the PathFinder, right-click, and select Activate.
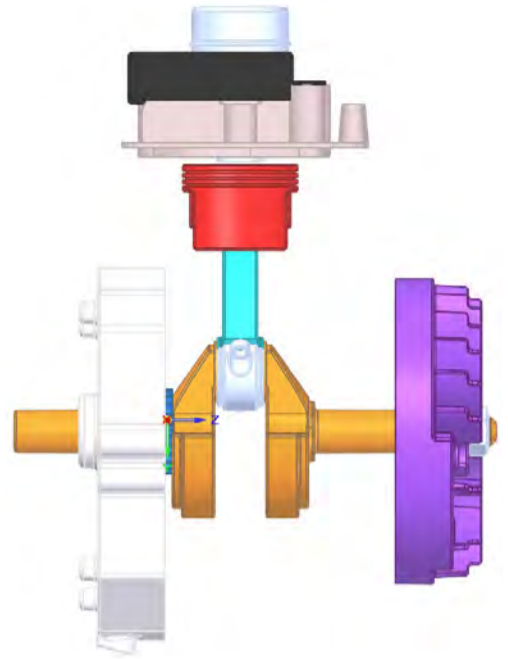
    *Make sure the selection is set to Face Priority. This setting is found on the **Home** tab -> **Select** group, under the drop-down list for Select. The Face Priority setting enables you to make face selections in the assembly in order to make part edits. The opposite of Face Priority is Part Priority, which you would use if you wanted to make part-level changes such as replacing a part.*
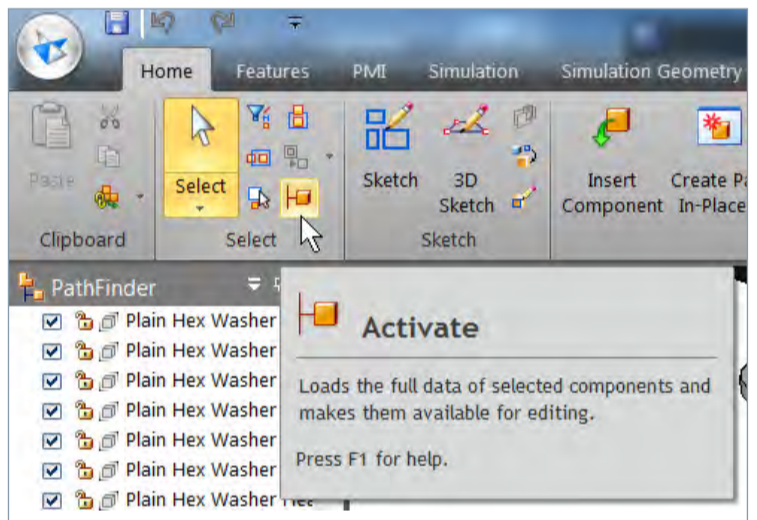
4. Zoom in, and drag a selection window (fence) as shown in the image to the right. This selects every face that is completely within the window.

    When you make the selection, the gray Design Intent panel is displayed next to the selected faces. You can move the panel out of the way, but make sure that everything is checked. The best way to do this is to click on the Advanced option in the Design Intent window, then in the Advanced interface, click the icon on the top left, to Restore all previous Design Intent settings.
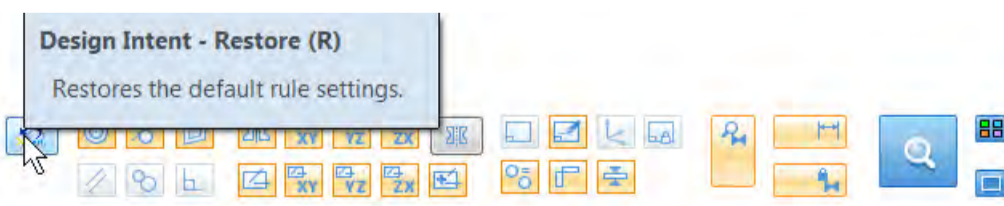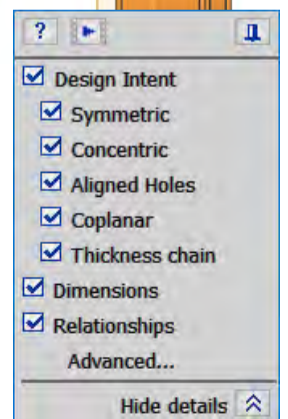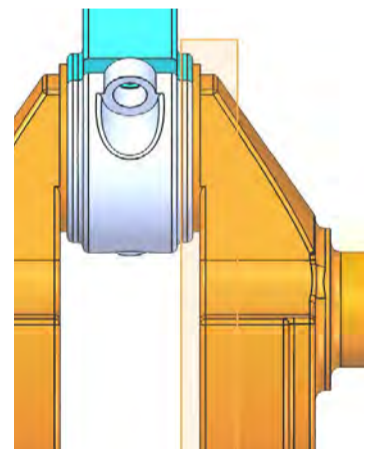
5. Make sure that the pushpin icon in the upper right corner of the Design Intent dialog is vertical when you move it, so that it stays where you put it.

    The Steering Wheel should also appear, as shown next. Click on the horizontal arrow. As you move the cursor, notice that multiple parts are resizing symmetrically. If you move the faces too far in either direction, the edit will fail momentarily, and then start working again when you move the cursor back into range.
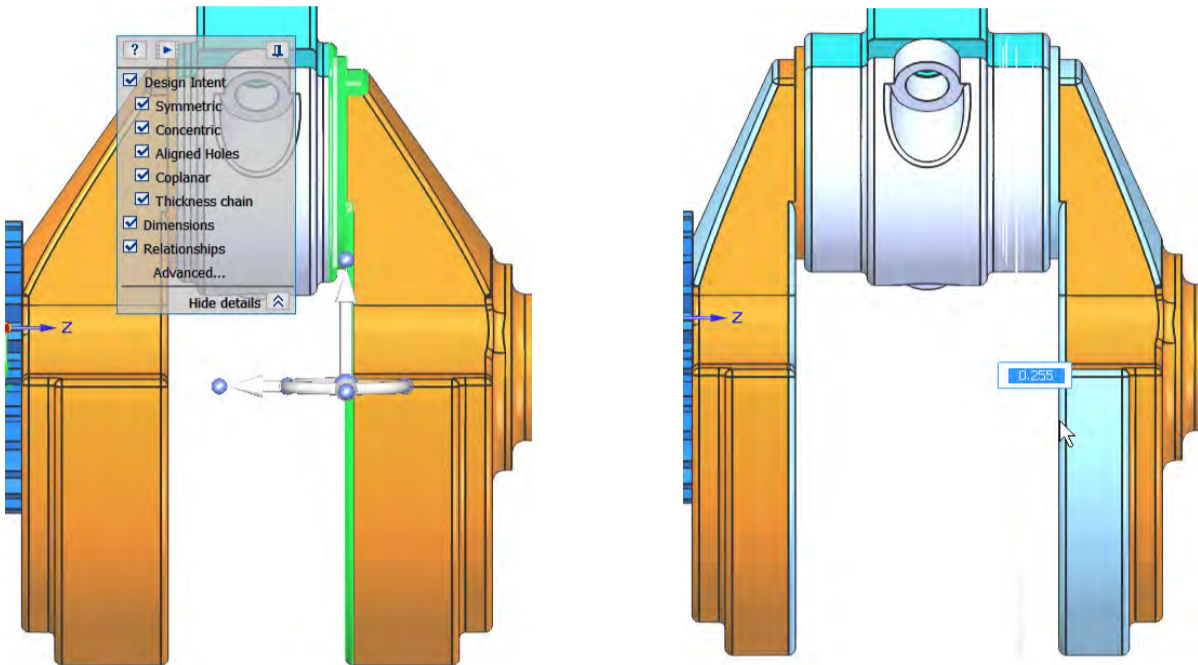
Some of the faces that are moving are being changed because you selected them, and are moving the steering wheel. Any face that is moving (or that is highlighted in light blue) that you did not select is being changed because of a Design Intent setting. In this case, we see that the predominant Live Rule being used is Symmetric, but there are others, such as Tangent, especially for controlling the rounds.

6. Move the cursor to the right of its original position, and enter .25. This changes all three parts without previous planning, or extensive in-context relationship building. This is how synchronous technology deals with a dumb imported assembly.

7. Next, reposition the model on your screen so you can see the outside of the white casting (Gear Box Cover.par), and both sides of the orange shaft.

8. Click on the left end of the orange shaft. The diameter Ø0.866 dimension appears.
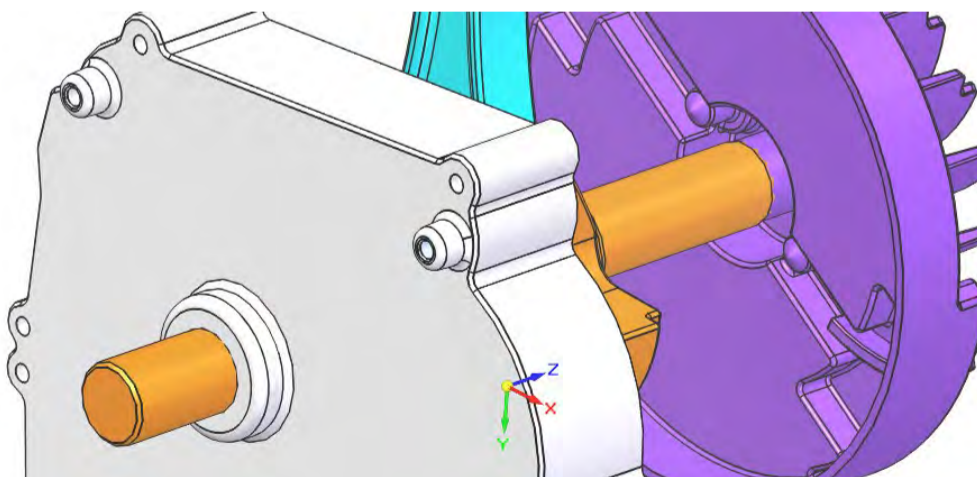
Note: Synchronous uses dimensions for diameters of cylindrical and conical faces, only using the Steering Wheel to position or angle cylindrical faces.

9. Select the inside diameter surface of the white casting part. You can't see that surface, so to select it, place your cursor where the surface would be if you could see it, wait for a second or two for the QuickPick cursor to appear, then right-click.

    Solid Edge displays the QuickPick list. Hover over items down the list and watch the entities highlight in the graphics area. Select the entry that highlights the inside diameter of the hole in the casting. This should display a diameter Ø0.874 dimension.

10. Change Ø0.874 to Ø0.870.
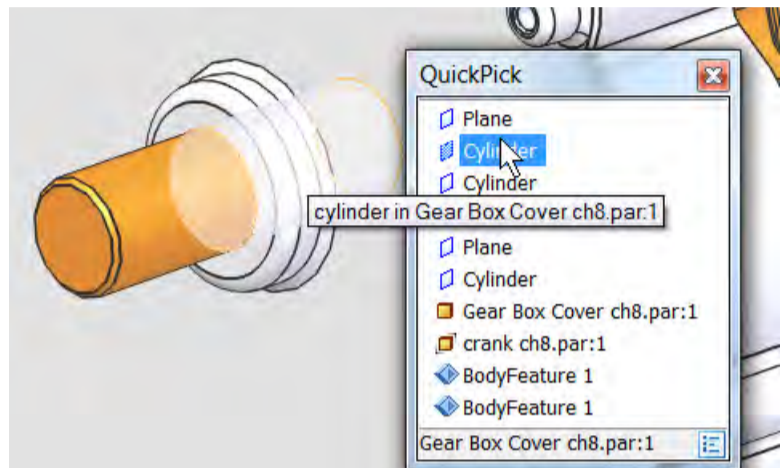
When you change the size of the orange shaft on one side, it also changes on the other side. If this was not intentional, you could turn off the Concentric Design Intent, but remember that this also controls your ability to move the cylinders within the plane. Ideally, the size and location aspects of cylinders would be unlinked by separating the Concentric Design Intent into two separate controls.

When changing diameters in an assembly, you must change each diameter individually, you cannot change multiple diameters at once unless they are the same size. You can move the position of multiple holes or cylinders at once, but you must change sizes individually.

Cylinder location is another issue.

11. Zoom up on the Rod Clamp part, so you can see the holes in both the gray clamp and the blue rod.

12. Click the inside of the clamp hole and move it along the highlighted arrow shown in the image to the right. Notice that the hole, the ear that surrounds it, and the counterbore face, all move, along with their symmetrical counterparts, on the other end of the part. This is due to the Design Intent settings. You can get the same sort of edit in a part file.
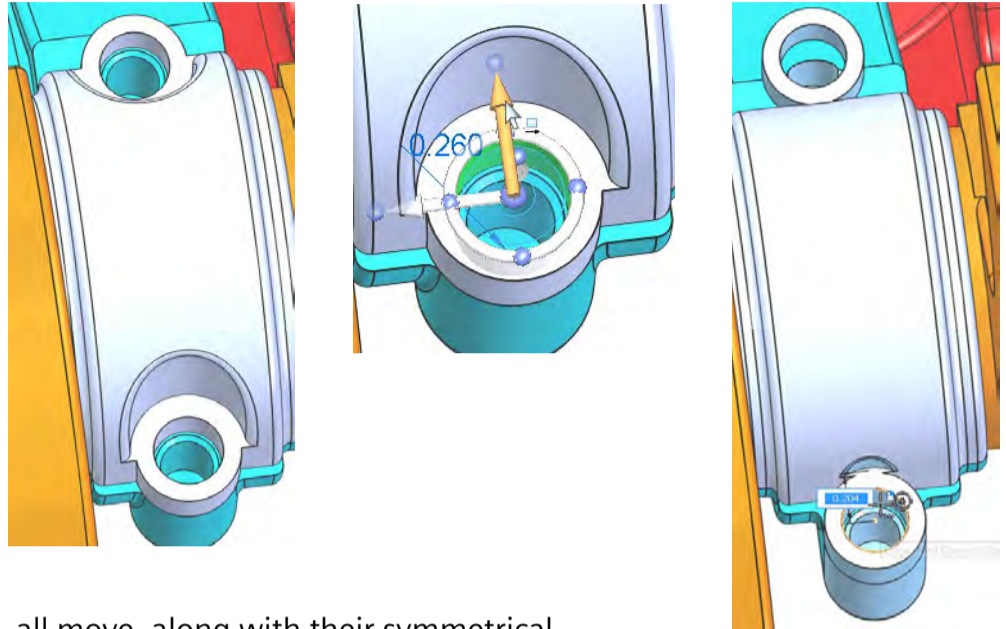
13. Press ESC to clear the selection.

14. Now select the same face again, and continue by Ctrl-selecting the inside face of the hole in the rod (blue part). Features on both parts move and the symmetrical elements on both parts are also moving.

⚠️ You can move the holes far enough that certain features begin to fail. When that happens, Solid Edge displays a red exclamation point in a yellow triangle which warns you that something that is supposed to be changing is not actually doing so, probably because of some geometrical condition.

Note: To move both holes on the Rod part using the Symmetrical Design Intent, along with the rest of the holes, there was some simple editing done to the part. Originally, the holes were on a plane that was 45° from the base planes. This doesn't allow for symmetry to be recognized. If you rotate the entire part 45° about the origin, however, the holes are now symmetrical on either side of a base plane, and the Design Intent of Symmetry can apply.
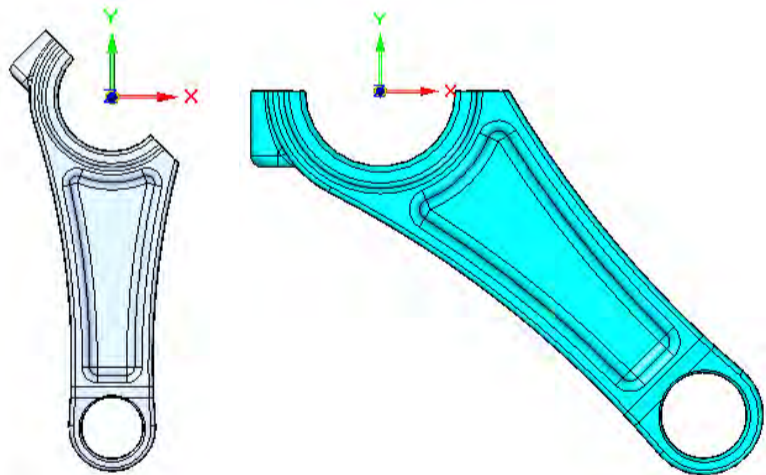
As an alternative, you could create an angled plane and establish a persistent Symmetry face relation between the holes in the part. This demonstrates the parametric capabilities within synchronous technology. These relations in the part would allow the assembly-driven edits to work. Another alternative would just be to edit the second hole in the Piston Rod separately. This isn't as elegant or automatic, but practically speaking, it would take less time than setting up the relationships.

Tutorial: Creating a part in-context

One of the most important tasks users run
into in assembly modeling is the ability to
create new parts in the context of the assem-
bly. Let's take a look at how this works with
synchronous technology.

1. First, orient the assembly on the screen so
   you can see the inside of the Gear Box
   Cover casting, as shown on the right.

2. Next, choose **Home** tab ->
   Assemble group -> **Create
   Part In-Place** command to
   create a new part in the
   assembly.
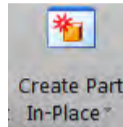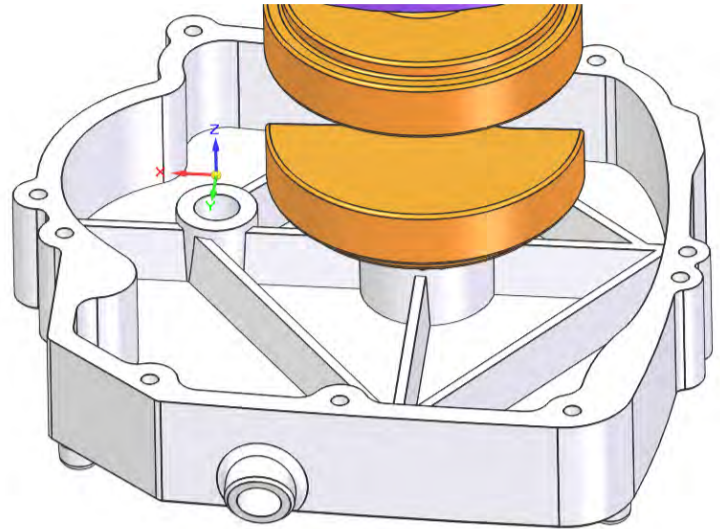
   When you click the Create New Part In-Place command, the command bar appears. Make sure
   to use the Ground and Edit In-Place options. The options for Ground and Origin are repeated
   in the Options dialog, which you also need to take a look at before getting more involved in
   this operation. The Options dialog is shown next.
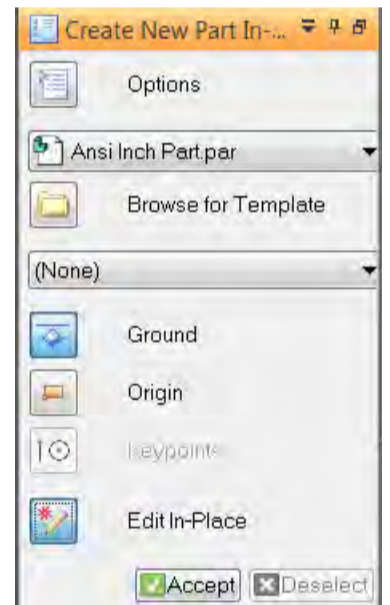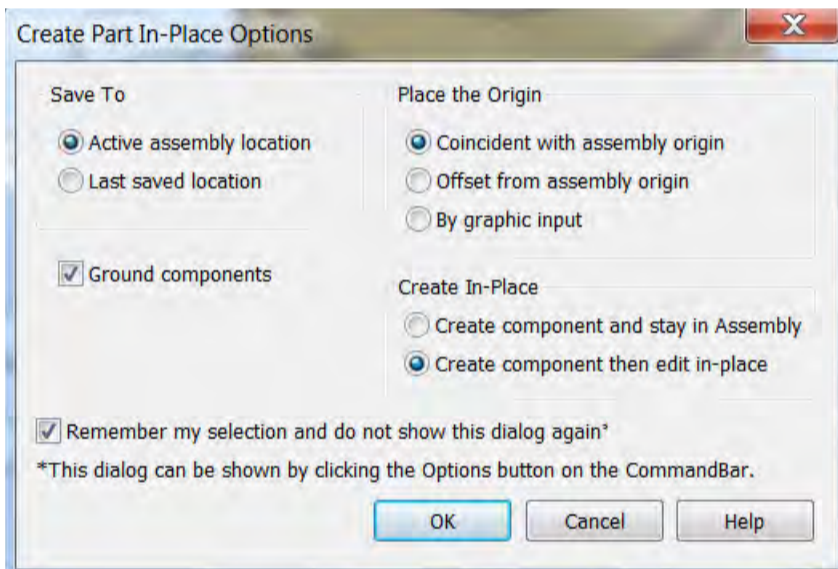
3. Specify a template to use for the new part as well as a material for the new part.
   (Material is listed in the drop-down list labeled (None), in the image above.)

4. Click Accept.

5. Name and save the new part in the directory that corresponds to the Save To setting in the
   options. Name the part GasketX.PAR.

   After you name and save the new part, depending
   on your settings, the rest of the assembly display
   will change.

   The display may turn gray and wireframe. You can
   proceed with the rest of the steps. If all the other
   assembly parts simply disappear, it means that you
   must adjust a setting.

Solid Edge assumes that you want to start sketching a new feature, which may or may not be the case. You already know how to sketch features, so we will skip over that.

6. To create a gasket on the face of the Gear Box Cover, first click the Extrude command, then hover over the flat face highlighted in yellow, as shown above.

7. Click the face, right-click to accept, and adjust your settings to create a gasket 0.080" thick in the appropriate direction from the face. This part was created without any relationships to the Gear Box Cover. You can edit the new gasket without first breaking any relationships. Test this by moving faces of the new gasket. If you want to move holes on the Gear Box Cover and holes on the gasket at the same time, you will need to use the Close and Return tool to go back to assembly editing mode, and then swtich back to Face Selection mode.
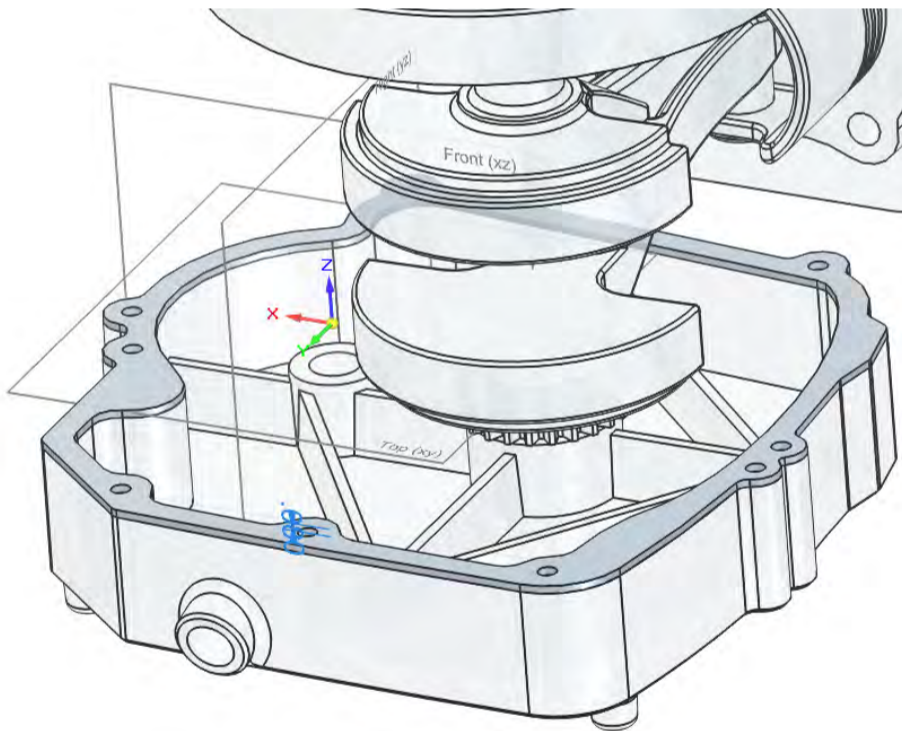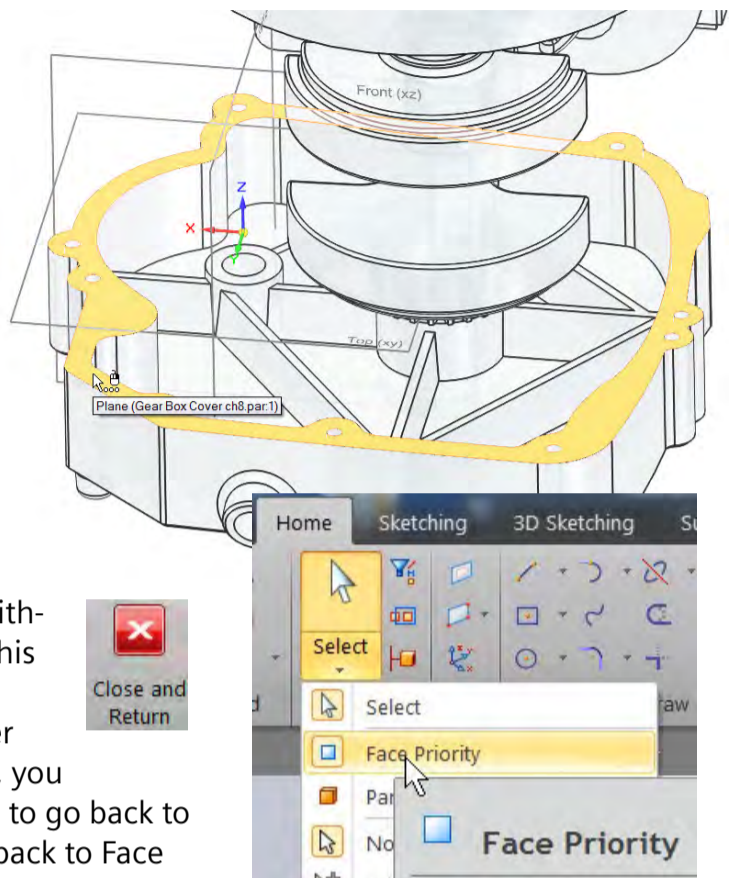
Rundown of non-synchronous functionality in assemblies

After reading some of the other chapters, you might have concluded that assemblies don't seem to have a lot of synchronous-specific functionality. And it's true, they don't. Assemblies, by design, must be agnostic-able to work with both sides. But there is a lot more than it might seem on first examination. Assemblies are where everything else comes together: sheet metal and general parts; synchronous and ordered; associative and non-associative; mates and motion; interferences and gaps; file management and visualization.

While there is a lot of functionality in assemblies, I will discuss three critical areas, just to familiarize the reader who does not have a background in Solid Edge. Those areas are Visualization, Mates, and certain File Management functionality.

Visualization in Solid Edge

Visualization is one of the most underappreciated areas of any CAD package. Users spend more time rotating, sectioning, changing transparency, wireframe states, and generally just looking at the model than any other single function.

Using the PathFinder

You should practice with some of these options to get used to using them.

As you hover over items in the PathFinder, Solid Edge will highlight them in orange in the graphics window. When you select items in the PathFinder, they are highlighted in green in the graphics window.
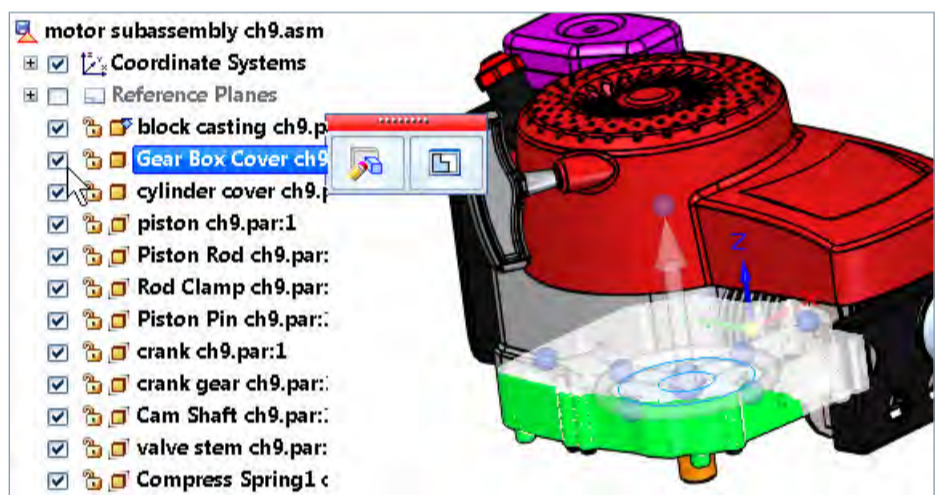
You can turn items on and off using the checkmarks to the left of each item in the PathFinder. It's easy and fast.

Right-clicking the items displays shortcut menus relevant to the different items.
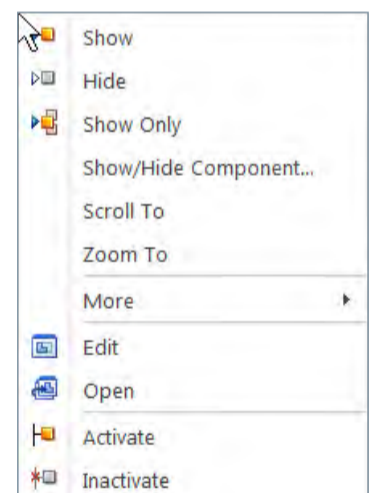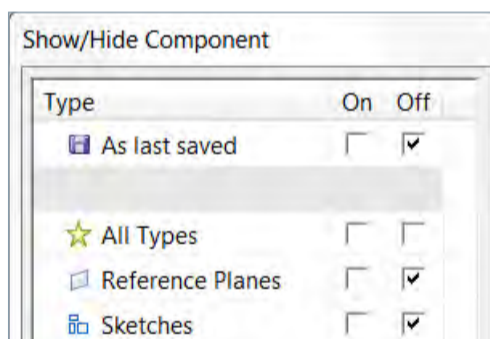
The Show and Hide options perform the same function as the checkmarks to the left of each item in the PathFinder – showing and hiding individual items.

The Show Only option hides everything except the item you clicked on.

The Show/Hide Component option displays a dialog box where you can turn various types of options on or off for the specific item from which you launched it.

The Scroll To option allows you to scroll to a part in the PathFinder, and the Zoom To option allows you to zoom to a part in the graphics window.

You can multiple select parts in the PathFinder using Shift and Ctrl select.

Using the mouse and the Quick View cube

I highly recommend using a 3D controller tool like 3DConnexion's products. They are easy to use, powerful, programmable and they free up your mouse hand for other things.

If you prefer, or are constrained to using the mouse, here are the tools, also found at the bottom right of the Solid Edge window:

Rotate view: middle mouse button drag

Zoom: scroll wheel, or Ctrl+middle mouse button drag, or slider in the lower right corner

Zoom to area: Alt+middle mouse button drag

Fit screen: middle mouse button double-click

Pan: keyboard arrows or Shift+middle mouse button drag

Tools on the View tab
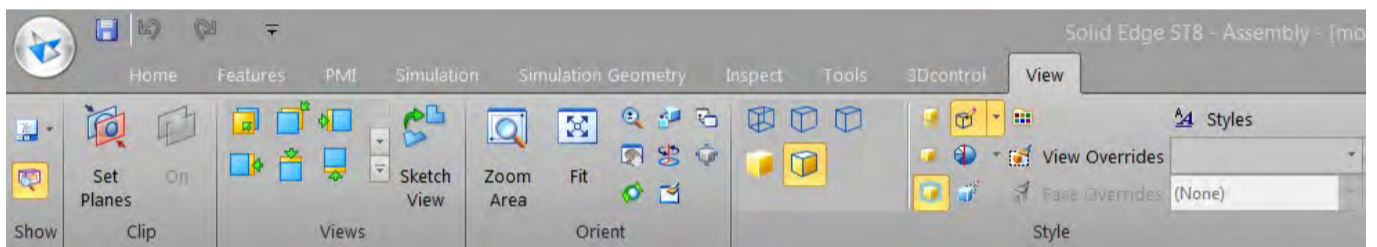
Clipping planes

You can set clipping or section planes to get an interior view of your model.

To set planes:

1. Click the Set Planes button.

2. Select a reference plane (both cuts will be parallel to this plane).

3. Place first clipping plane.

4. Place second clipping plane (if you only want one cut, you can push this one off the model).

5. Turn clipping on or off using the On button.
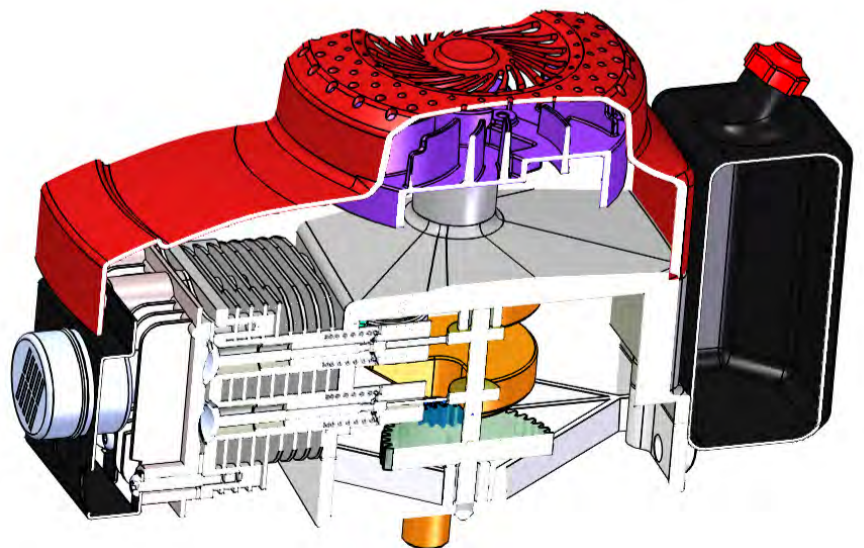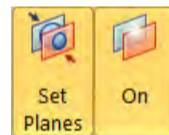
View styles

The following view styles are available:

Wireframe.   *Visible and Hidden Edges.*   *Visible Edges.*

Shaded.   *Shaded with Visible Edges.*
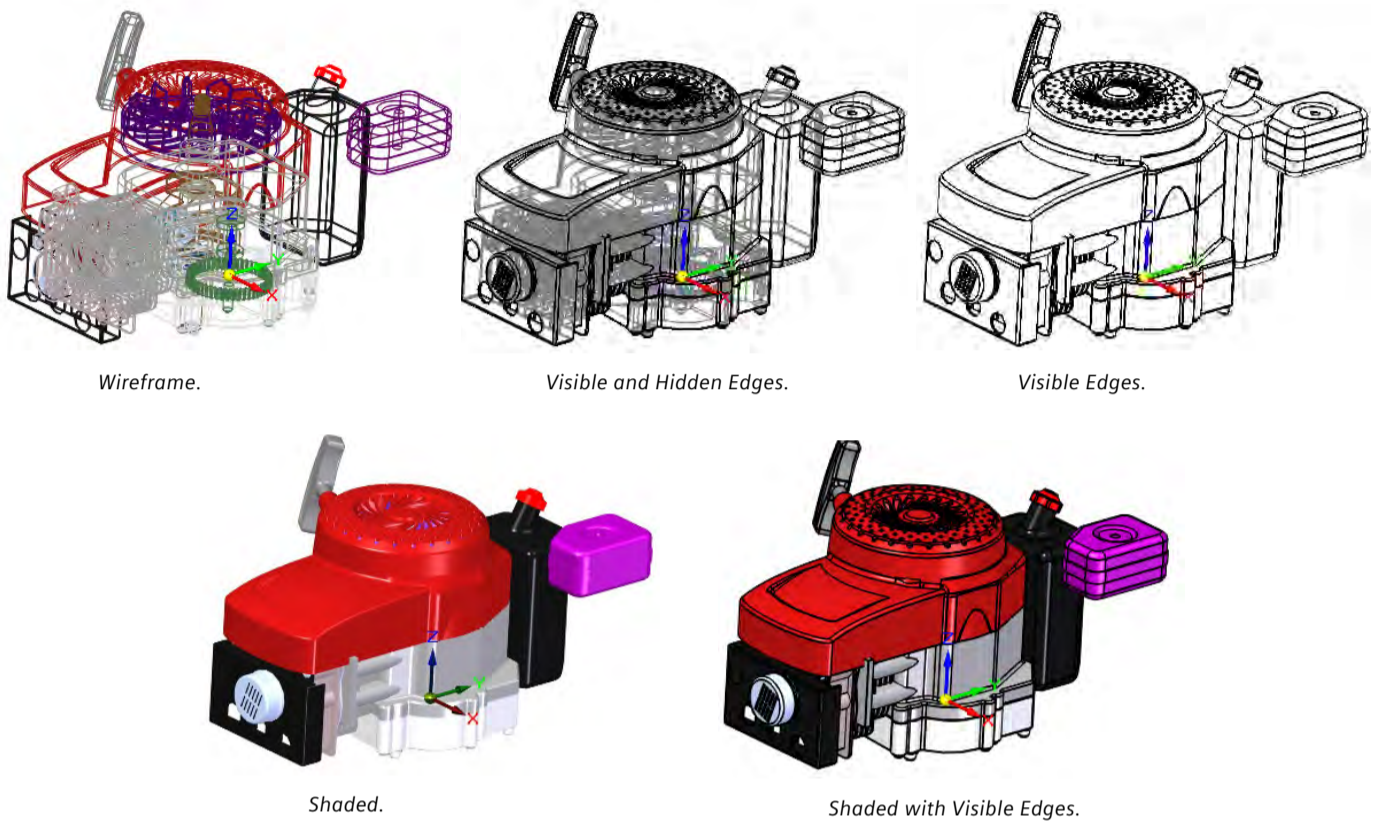
View style settings

The view style settings are:

- *Floor Reflection*: assembly sits on a shiny floor
- *Floor Shadow*: assembly casts shadow on the floor
- *High Quality Rendering*: enables bump maps, gives slightly better specular reflections
- *Edge color selection*: you can use either a single color for all model edges or use each model's edge color
- *Sharpen*: a higher setting may be slower to display
- *Perspective*: use View Overrides to change amount of perspective

PMI Section

PMI Section is on the **PMI** tab -> **Model Views** group. You can establish multiple PMI sections which are tracked in the PathFinder, and can be turned on or off individually as you work.

To create a PMI section:

1. Click the Section button.

2. Select an assembly plane or activate a part to select or create a new plane.

3. Draw a section profile, and exit sketch. Often this is just a rectangle larger than the model to cut everything.

4. Select the side of the profile that you want to cut away.

5. Identify how far you want to cut.
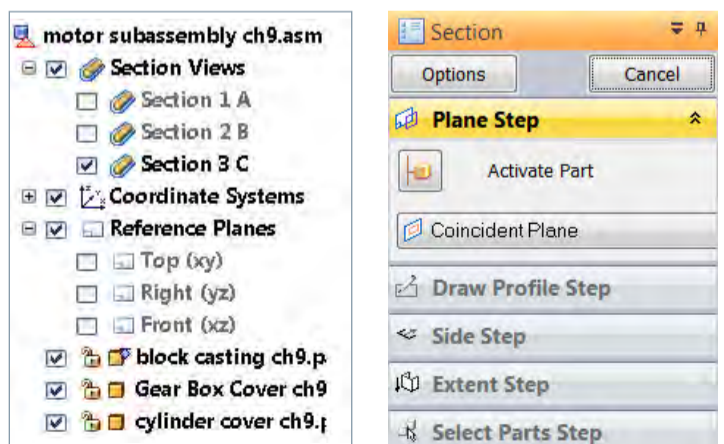
6. Assign which parts to cut:

    a. Cut all parts

    b. Selected parts

    c. Unselected parts
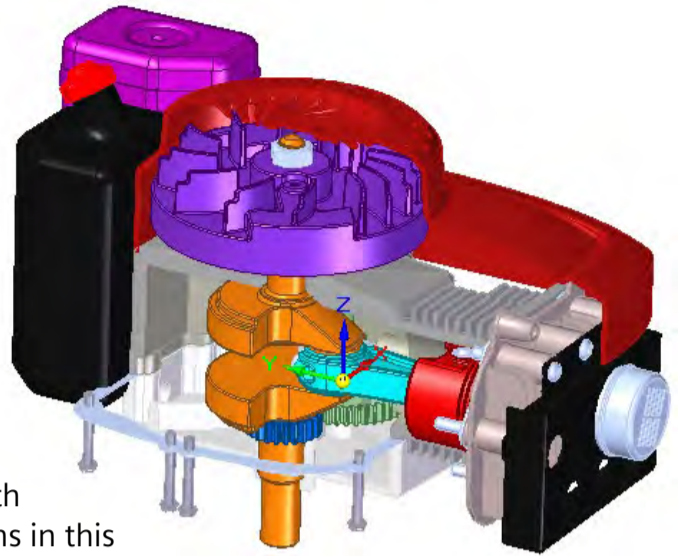
Exploded assemblies

Sometimes the best visualization is an exploded assembly. In Solid Edge, you can create exploded assemblies with the ERA (Explode, Render, Animate) environ-ment. Renderings these days are usually done with the Keyshot plugin, and we don't cover Animations in this book, but we can cover exploded assemblies.

To get to ERA, go to the **Tools** tab -> **Environs** group.

You have two options: Auto Explode and Explode. I used Auto Explode to create the image to the right, so let's give that a try.

As you work through this and any other unfamiliar functionality in Solid Edge, be sure to keep an eye on the PromptBar, which will give you the next step, or what the software is expecting you to select next.

To explode an assembly:

1. Click the ERA button.

2. Click the Auto Explode button.

3. On the Select Step of the Auto Explode command bar, set it to explode either Top Level or Subassembly (the Subassembly setting will keep the subassemblies together as a single unit).

4. To move to the Settings Step, click Accept.

5. The Automatic Explode Options help you deal with subassemblies, but our sample assembly does not have any subassemblies.

6. Automatic Spread Distance enables you to control how far apart the individual parts are spread, or allow the software to control that. In the above image, I allowed the software to control the spread automatically.

7. Click the Explode button, then click Finish.
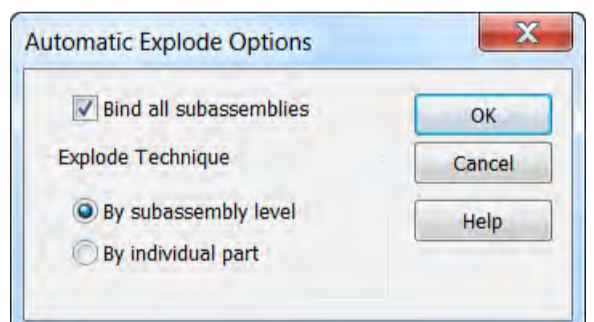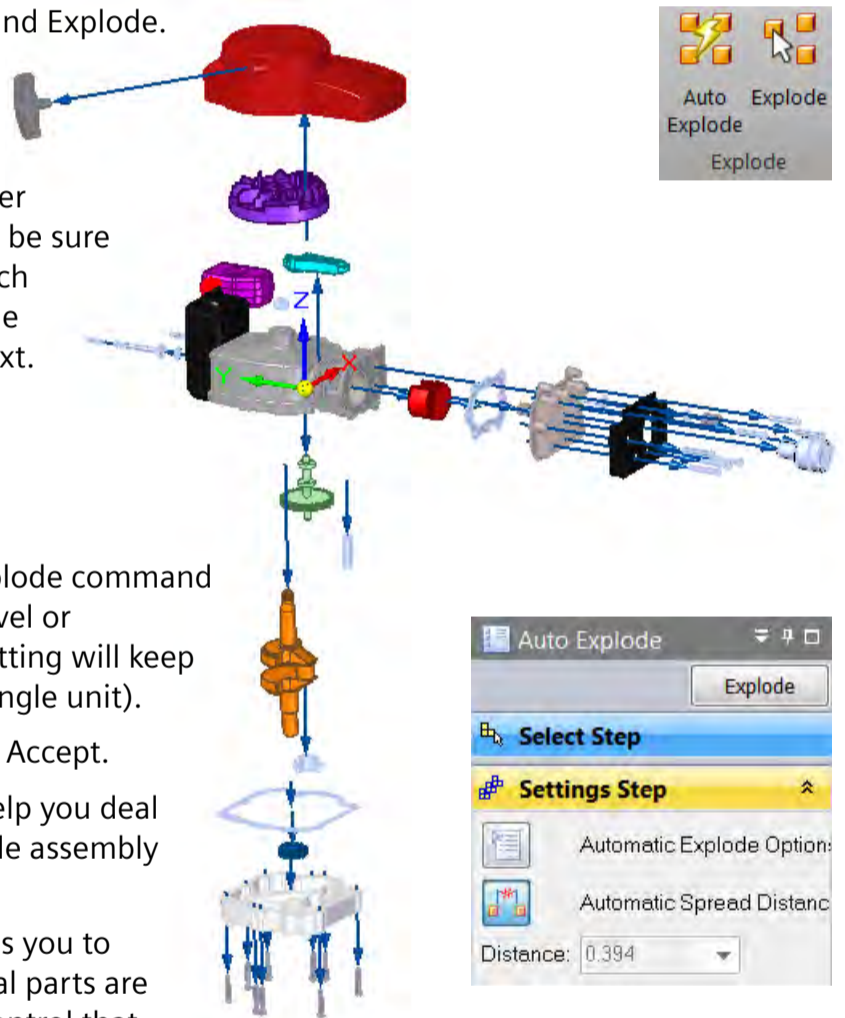
The explode animation is recorded and controlled from the Explode PathFinder, which is only available in ERA mode.

You can click on the exploded parts in the graphics window and edit the explode distance if you want to override the automatic results.
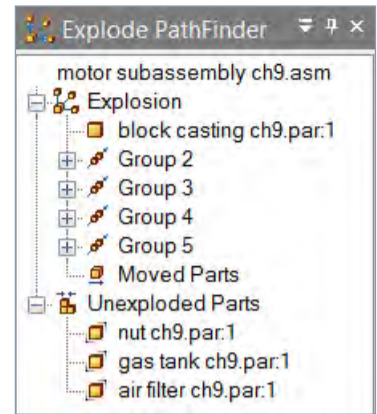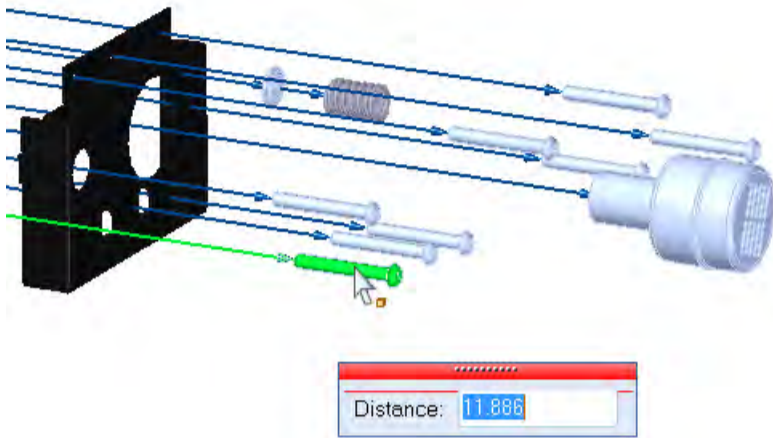
Creating and managing mates in Solid Edge

If you are familiar with other 3D CAD programs, you are probably familiar with mates. Mates are relations between parts that position the parts and allow them to move in constrained ways. There are many mate tools in Solid Edge, but in the following quick tutorial, we are just going to review a few basic mates. If you want details on assemblies, mates and motion, you can access the training and Help tutorials supplied with the software.

1. From the material associated with this chapter, open the assembly called CH9 mates assembly.asm.

2. On the Home tab, click the Insert Component button.

3. The Parts Library pane appears
   (if it does not, go to **View** -> **Show** -> **Panes** -> **Part Library**).

4. Use the Up One Level button and/or the folder drop-down list to browse to the folder where the materials associated with this chapter are kept (ch9 Editing in Assemblies).

5. From the Parts Library, click and drag the part called crank ch9.par into the graphics window and drop it in white space. The part should remain transparent.

6. Click on the cylindrical face, as shown.

7. Click on the inside face of the hole in the large gray block casting part, as shown.
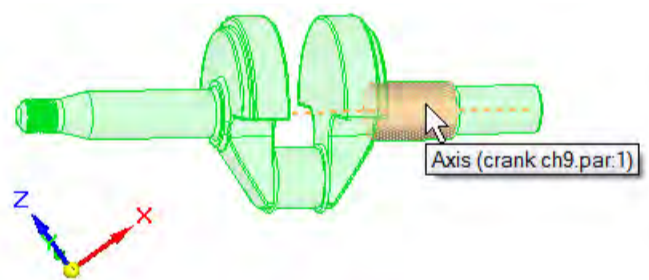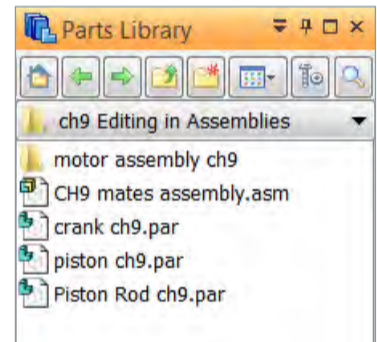
   The crank shaft should fly into place in the casting. The threaded end of the shaft should be pointing away from the casting. If the crank shaft went in backwards, press the F key to flip its orientation 180°. Even if the part went in the correct way, practice using the F key just to see what it does.
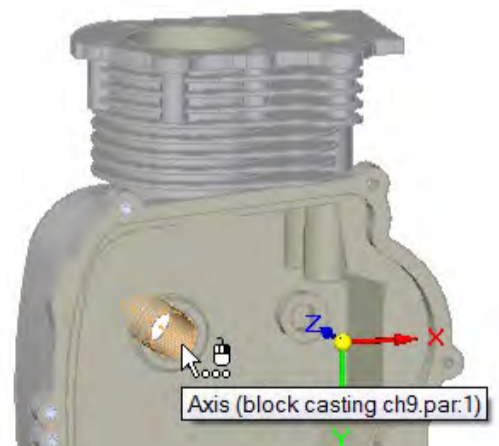
8. Next you push the crank shaft into the casting until it rests against the bearing face. The two faces you must select are indicated in the image on the right. The face on the casting will be easy to select, but the one on the crank shaft is hidden.

    To select the hidden crank shaft face:

    a. Hold your mouse over the hidden face until the cursor changes to a mouse with three dots. This option is called QuickPick, and we use frequently in Solid Edge to select hidden or alternate items.

    b. Right-click to display a list of items available at that location. Move your mouse over the items in the list until the correct face highlights. In the list, click the item corresponding to the highlighted face.
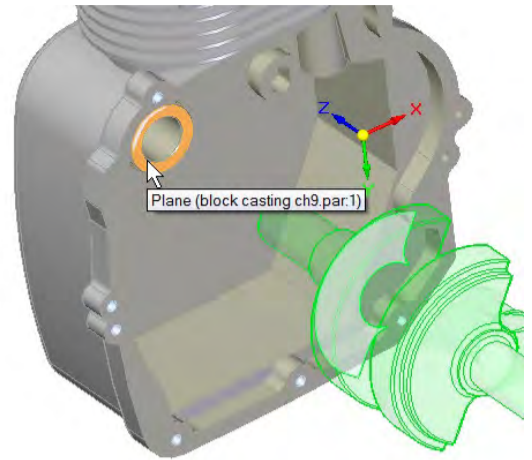
9. Click on the bearing face in the casting if you haven't already. The crank should fly into place inside the casting cavity.
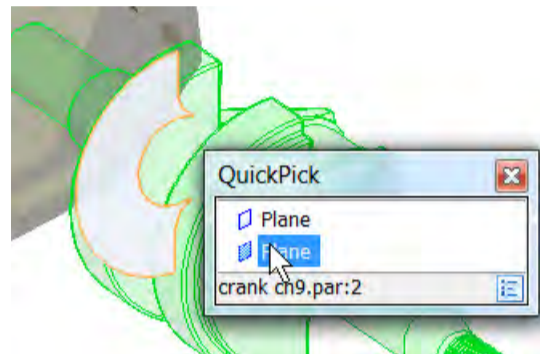
10. Right-click or press the Esc key to complete mating the crank in place. You may notice that we only created two mates, so the part should be able to move.

11. Click the Drag Component button, and click and drag the crank part. It should rotate about the shaft, but not move side to side.

12. From the Parts Library, click and drag the piston ch9. par into the graphics window and drop it in blank white space. It should turn transparent green.

13. Select the outside cylindrical face of the piston, as shown.

14. Click on the corresponding face of the casting.

15. If the part goes in backwards, press the F key to flip it. The part also may get stuck or interfere with the crank part already mated. We only need one mate with this part because the next part we put in will help to constrain this one.
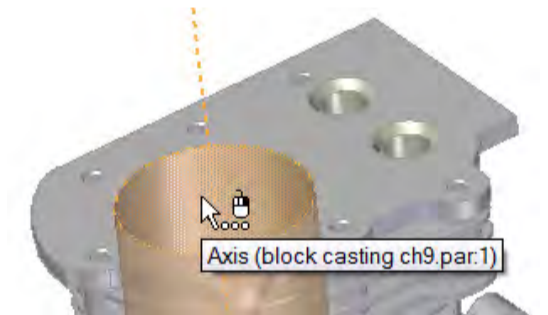
16. When you are done, right-click or press the Esc key.

17. Using the Drag Component tool, drag the piston up so you can still get to the holes to locate the piston rod part.

18. You may want to pre-position the view so you can see both the hole in the red part and the short cylinder in the orange part. Drag the Piston Rod ch9.par into the graphics window like the other two parts.
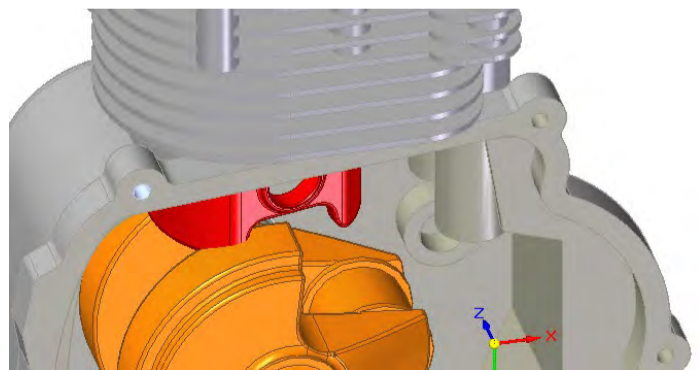
19. Click the small hole in the piston rod and the hole in the red piston part.
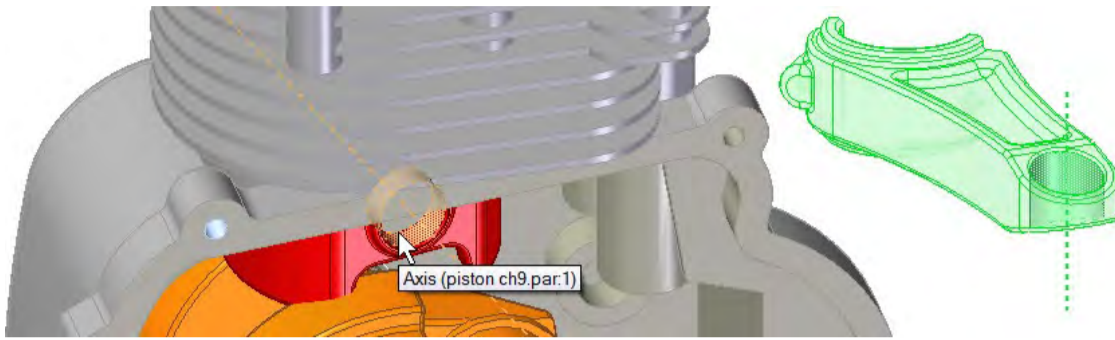
20. Click inside the half cylindrical face of the rod and the short cylindrical face of the orange part.

21. Push the rod into the crank. Don't worry that the parts have to interfere to get where they are going, once they are together it will be correct.

    To push the rod in, select the small flat on the rod, in the lower left of the image, and use QuickPick to select the hidden flat face as shown on the crank shaft. When this part flies into place, your task is finished.
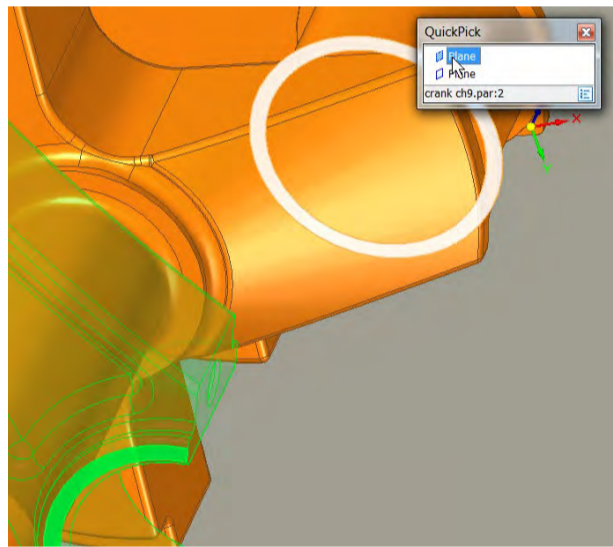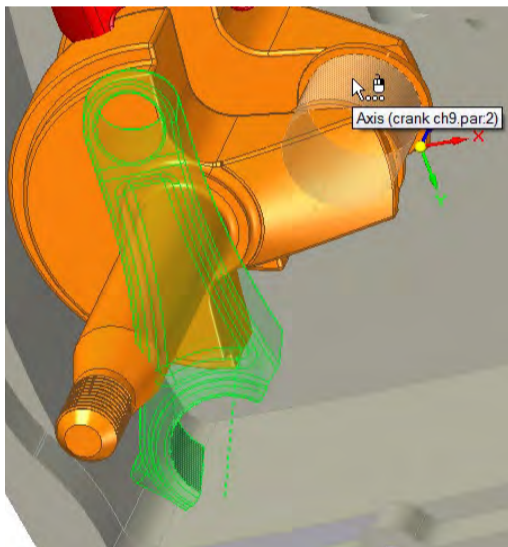
22. Right-click or press the Esc key.

23. Use Drag Component again to rotate the crank shaft, and watch the rod drive the piston up and down in the cylinder.

Simple file management tools in Solid Edge

Solid Edge has a lot of power to manage the files associated with assemblies, but I just want to point out a few.

On the Modify group, right next to the Drag Component button is the Replace Part drop-down list. The other commands on this list are:

- *Replace Part*
  Choose the command, then select a part to replace, and browse for a new part to use in its place.

- *Replace Part with Standard Part*
  When you choose this command, Solid Edge displays the Standard Parts interface in the place of the browse window. Standard Parts is a library of nuts, bolts and other hardware.

- *Replace Part with New Part*
  Use this command if you want to start a brand new part in the place of an existing one so you can work in-context and reference other parts if necessary.

- *Replace Part with Copy*
  Choose this command if you want to copy the part by doing a Save As and be able to rename the part.
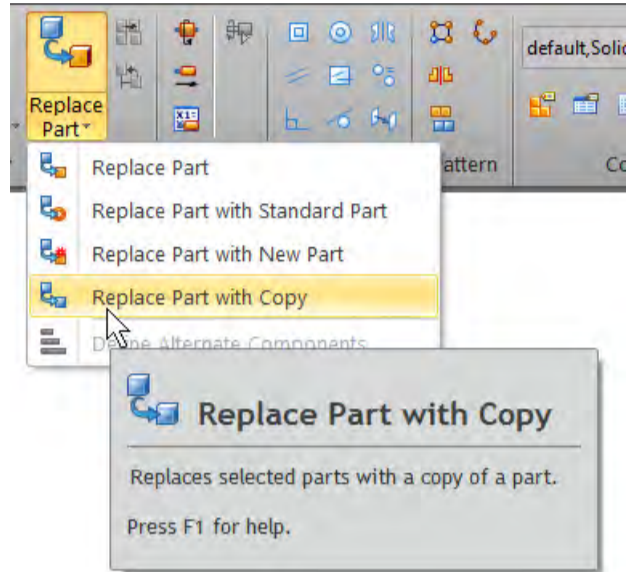
These four commands are fairly simple, but as often as you might need to perform these operations, they are sure to save you a lot of time and clicks.

The final command here is a combination of View and Markup and Revision Manager. These are powerful tools to help you work with your Solid Edge assemblies. From the Open command, you choose which you want to use:

- If you want to rename or reassign parts in an assembly, use the Revision Manager.
- If you, or someone without full-blown Solid Edge, want to view an assembly, use View and Markup.
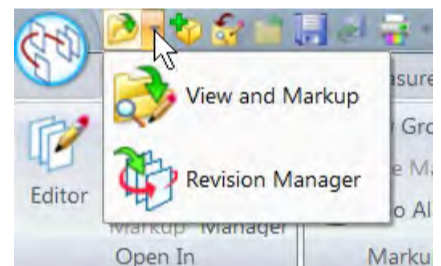
Summary

Solid Edge assemblies don't take sides when it comes to ordered vs. synchronous. You can have and edit both types of parts in any Solid Edge assembly. So there is nothing inherently synchronous about any assembly aside from how you edit the parts. You can edit synchronous parts synchronously in any Solid Edge assembly.

This book is really just focused on synchronous technology, but if you are not familiar with Solid Edge, the functionality in the assembly environment independent of synchronous technology is worth noticing.

Chapter 10

Synchronous, parametrics and associativity

Somehow, even the few CAD users who know there is some alternative to history-based CAD tend to believe that the only alternatives are non-parametric. By now, this book should have proved that the Synchronous method is indeed parametric. This chapter will offer more details and examples of techniques that use parametric design within synchronous technology.

A topic we haven't discussed as much, however, is associativity. For the purposes of this book, we define associativity as links between documents that drive geometrical changes. In general, with synchronous technology we consider it a goal to avoid most associativity. Associativity leads to things like order-driven rebuilds, parts with links to assemblies and other parts driven by in-context associations, and complex relationships that have to be understood to avoid problems with file management. Examples of functions that use associativity are Part Copy, Inter-Part Copy, and external linked tables. In addition, you saw the sheet metal functions with associative sketches, in chapter 7.

Still, associativity is sometimes necessary to automate certain changes, especially changes that involve multiple parts. Master model techniques, where you drive all of the main parts in an assembly from a single part, are possible with synchronous, and in this chapter we will demonstrate some of those techniques.

Note: A <u>brief video</u> goes along with the content of this chapter.

Parametrics

Parametric control over models is crucial to many design professionals. Synchronous gives us several tools to control geometry with parameters:

- PMI dimensions
- Variables and formulas
- Face relations

Variables and formulas

If you are coming to Solid Edge from another CAD package, it could be that the definitions of items may be different between the two systems. One of the things to be aware of is that in Solid Edge, dimension names are a subset of variables. Each dimension has a name, whether the default name (usually in the form of V208 or similar) or one assigned by the user. That name is a variable. Other variables stand alone, such as Density. So some variables are connected to dimensions and some are not, as shown in the following Variable Table.


The Variable Table lists all of the dimension variables, and all of the formulas, links, or other controls associated with those variables. We will revisit the Variable Table shortly.
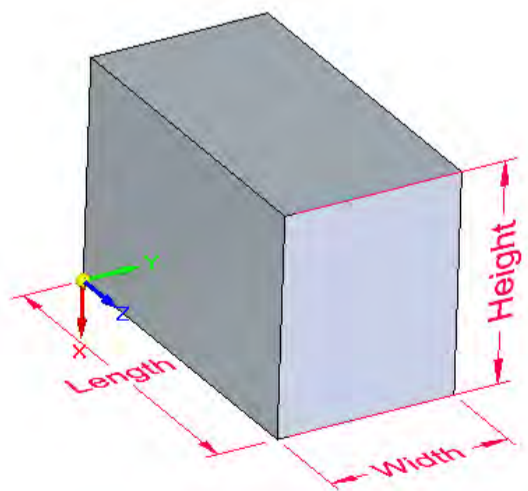
Type	Name	Value	Rule	Formula	Range	Exp...	Exposed ...	Comment
Dim	Height	3.602	🔒 Paste L...	@'C:\workdir\tbucket\tbucket ra...		☐		
Dim	Length	4.514	🔒			☐		
Dim	Width	2.443	🔒			☐		
Var	PhysicalProperties_Density	0.000	Limit		[0.000;)	☑	Density	
Var	PhysicalProperties_Accuracy	0.990	Limit		(0.000...	☑	Accuracy	

You should already be familiar with how dimensions on the 3D model work with synchronous, so in this chapter we can focus on the use of variables, writing expressions and equations, and the use of tables. The first stop on this adventure is variables.

To see and change a dimension name, double-click the dimension arrow or select it, right-click the arrow, and then select Edit Formula.
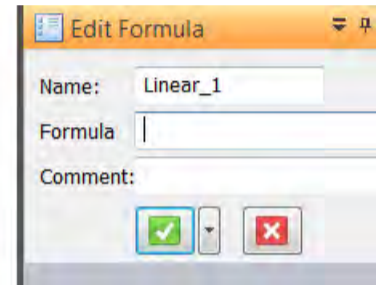
Note: Dimensions must be locked to be driven by a formula. The Formula box will be grayed out if the dimension is not locked.
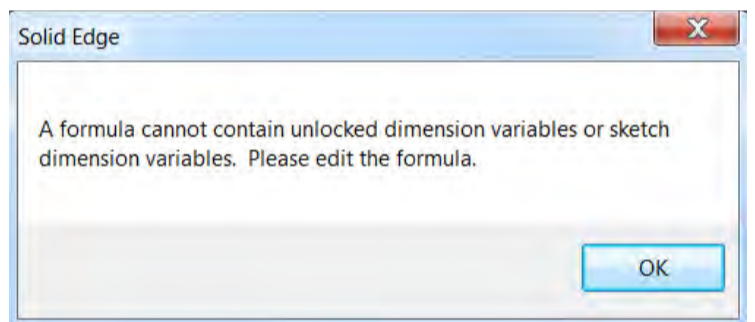
If you are commonly writing formulas or using variables, you should name the dimensions you use with names that will help you remember what they are for.

To write formulas, use dimension names, variables, numbers, and standard mathematical operators. Do not start formulas with an equal sign (=). Save that for Excel. If you are unsure about the abbreviations for some of the allowed functions, check out the Function Wizard, which is available from the drop-down list on the OK (green check) button in the Edit Formula dialog box.
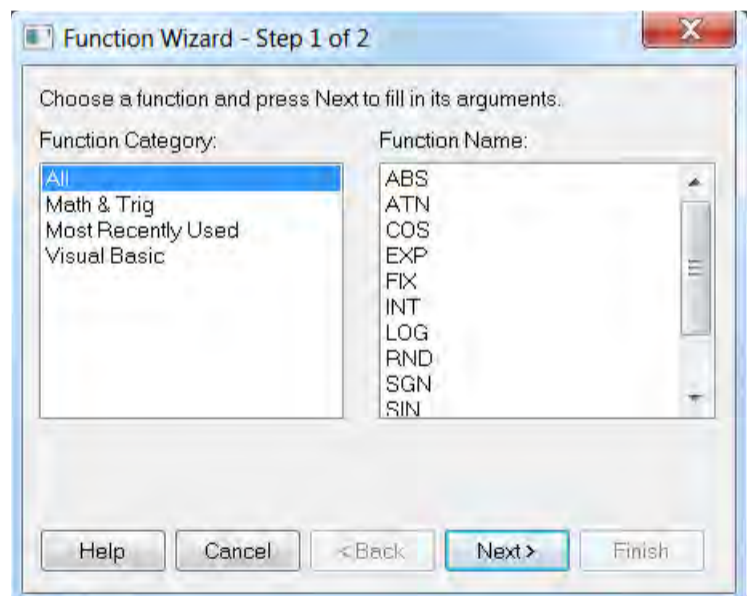
The Function Wizard does not have the operator symbols, such as +, -, *, /, so when you need these operators, you must key them in.

Also, when writing a formula, Solid Edge will not allow you to use unlocked dimensions or sketch dimensions. A dimension that is driven by a formula will be displayed in purple, to remind you not to try to edit the value directly. Also, a dimension or variable that is driven by a formula cannot be used to in turn drive another dimension or variable. Nesting formulas can cause logical loops or nested dependencies.
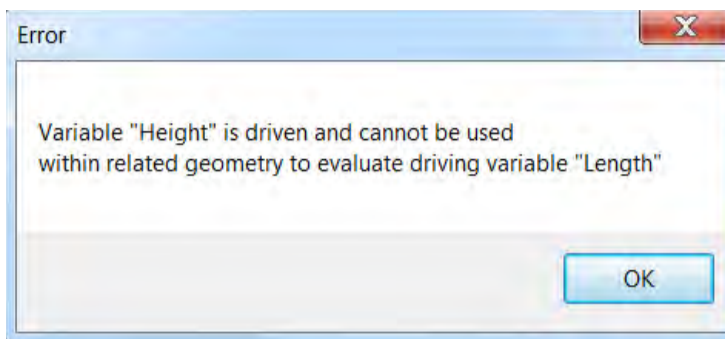
To create a formula, make sure the cursor is in the Formula field of the Edit Formula command bar, and click on an eligible dimension in the graphics area or on the PathFinder. Enter number values and operator symbols or select function names from the Function Wizard. You can also enter anything, but to make sure you get the syntax and spelling correct, letting Solid Edge fill in the boxes may be the best option.
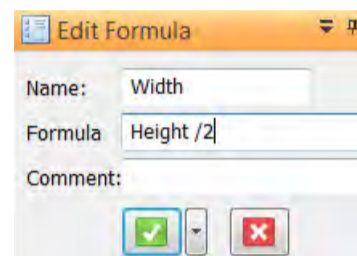
> *Note: By right-clicking on a dimension, you can select Show All Values, Show All Names, or Show All Formulas. The last two will show all names or formulas instead of values (not in addition to values).*
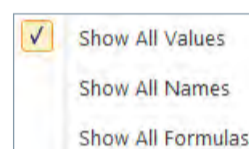
Variable Table

If you use more than a few named variables, they can become difficult to track, especially if you have multiple people accessing the same models. For this reason, Solid Edge provides a table where you can control all your named dimensions and variables. This table encompasses more than just tracking variables and formulas. It also has a button for the Function Wizard and a very powerful Variable Rule Editor.
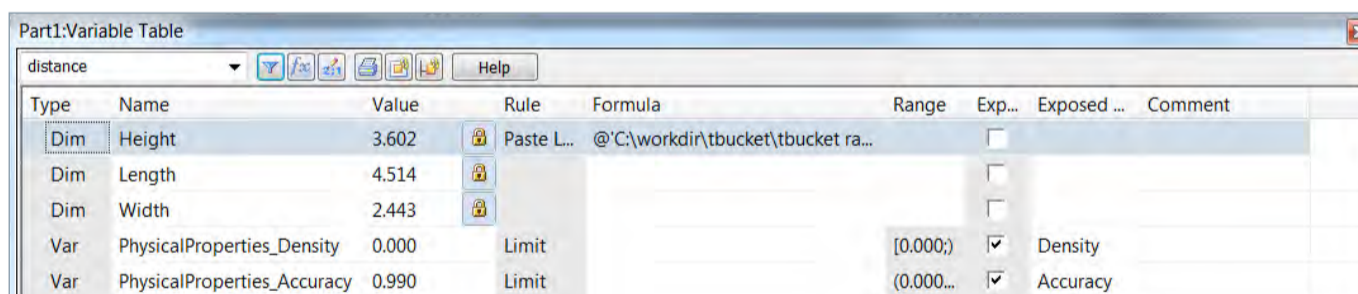
The Variable Rule Editor enables you to control variables and named dimensions as follows:

- Formula
- Visual Basic routine (external *.bas file)
- Link to assembly and part variables (via Peer Variables)
- Links to Excel
- Discrete list of values
- Max and Min limits
- Sorting by type

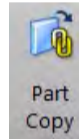There is a great deal of power packed in the variable and formula functionality in Solid Edge.
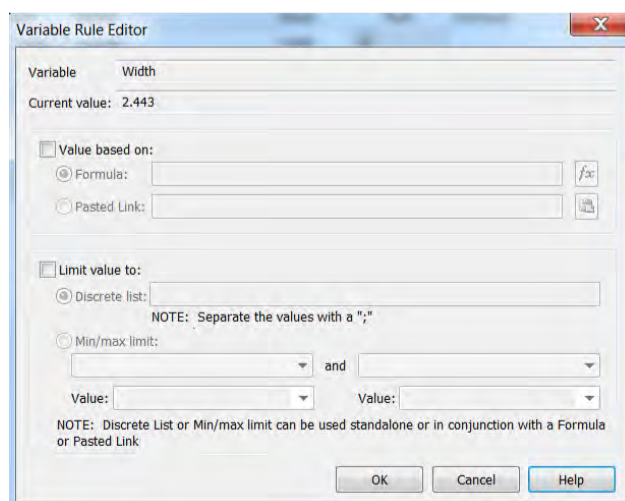
Associativity

Synchronous technology spends a fair bit of effort trying to give the user all the functionality of associative functions, without the negative entanglements of associativity. Links between files complicate file management and slow down CAD work in general. But still, the Solid Edge developers recognized that there is no good alternative for some of the things CAD users need to do with their models, and to have a fully robust CAD modeler, they needed to add associativity to synchronous technology. These synchronous functions are also associative:

- Part Copy
- Inter-Part Copy
- Shared External Tables
- Copied Variable Links

Part Copy

The Part Copy command enables you to take one part and use it as the starting point of another part. In synchronous mode, Part Copy creates a dead link; that is, changes to the copied part will not propagate to the parent part or vice versa. You can, however, make synchronous changes to the copied part. If you need a live associative link, you must insert the part in ordered mode. It doesn't matter how you made the original part – it only matters if the Part Copy feature was created in ordered or synchronous mode.

The Part Copy command is on the **Home** tab -> **Clipboard** group, in the Part Copy drop-down list.

You can use Part Copy when you want to mirror or scale a part, and is especially useful if you have multiple solid or construction (surface) bodies that you want to manage. For an overview of some of the functionality available, see the Part Copy Parameters dialog shown above.

Because of the ability to parse out design and construction bodies, Part Copy is a particularly useful tool when working with master model files, where the parts of an assembly are driven by splitting a single part file into multiple bodies.

Inter-Part Copy

Inter-Part Copy gives you the ability to copy faces between parts in the context of an assembly. You can find Inter-Part Copy on the Part Copy drop-down list on **Home** tab -> **Clipboard** group. It is only available when you are editing a part in the context of the assembly.
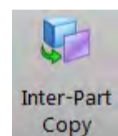
You can use Inter-Part Copy to copy a face (as a surface) from one part to another to extrude up to the surface, use the surface to cut, thicken the surface, attach the surface to a solid, and so on.

To use this command, you must first open an assembly with at least two parts. Activate the parts you want to have involved in the associative inter-part link. Then right-click the part you want to contain the link, and select Edit.

　　　Chapter 10 | Synchronous, parametrics and associativity
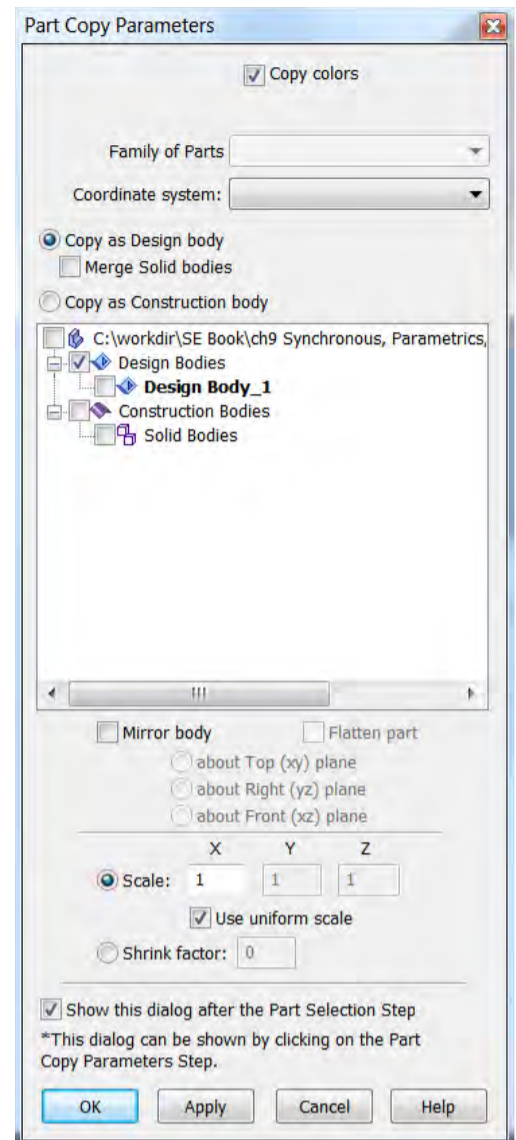
Note: In Solid Edge, if you already have a window open with the part you want to edit in the context of the assembly, Solid Edge will open that window, rather than allowing you to edit the part with all the other assembly parts visible. So if you go to edit a part in-context, and the software keeps displaying a part window, close down the part window, and then right-click and Edit.

When you use Inter-Part Copy, Solid Edge first asks you to identify the part from which you want to copy geometry, then asks you to select the type of geometry, and finally asks for you to select the geometry itself. The Inter-Part Copy command bar is shown on the right.

For example, here is a part where one face was copied from an adjacent part, thickened to a solid, and then additional features were added.

Note the chain link symbol in the PathFinder entry, signifying that there is an associative link between this part and another part. Also, note the Links collector, at the top, which shows you exactly where the link points.
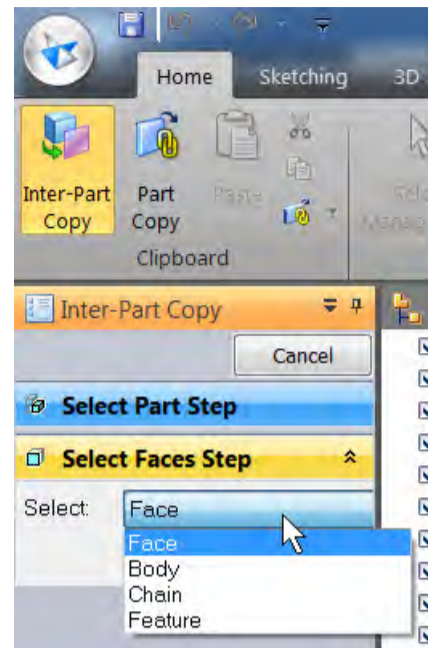
Note: It is possible to get too much of a good thing. Inter-part associations are very powerful, but they also slow down your models and introduce file management complications. This is true with all CAD software. It is also possible to get the links so intertwined that you wind up with looping relations.
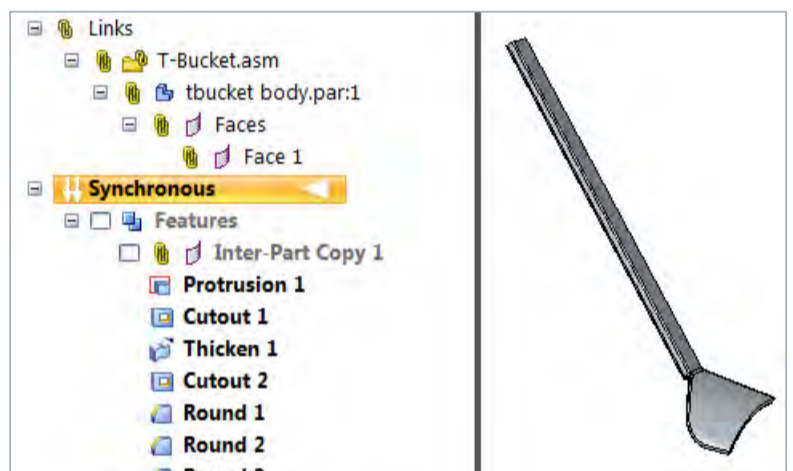
For some people, one of the attractions to synchronous is that it is designed to avoid associative links where possible.

Inter-Part Manager

To invoke an overused cliché, "with great power comes great responsibility". Inter-part links are very powerful, but can also get you into trouble. The Inter-Part Manager gives you the ability to see and manage all of the inter-part links in an assembly. By right-clicking the links, you can access functions that enable you to update or break existing links.
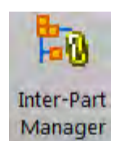
Inter-Part Manager is on the **Tools** tab -> **Assistants** group. It is available in assemblies and also in parts with inter-part links.

Shared external tables and copied variable links

These topics were discussed briefly in the previous section of this chapter on parametrics, but also apply to associativity. These are methods, along with Part Copy are ways to use associativity without employing an assembly. The links are direct, part to part, without requiring an assembly to create some sort of spatial relationship between the parts.
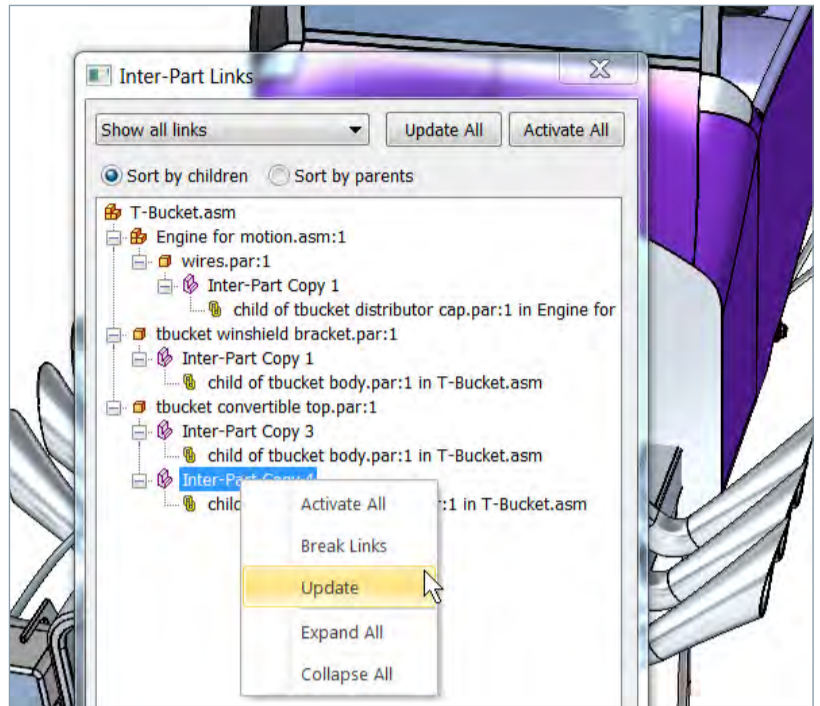

Tutorial: Synchronous parametrics and associativity

In a previous chapter, you created a gasket in place for the assembly shown on the right. In this tutorial you will make adjustments to some parts:

Change the location and size of the main bearing surface of the gear box cover to accommodate the crank shaft. Because we can't move the crank shaft, the hole and bearing face have to move.

Move some bolt holes, but move the shaft hole first.

Move the shaft hole in two ways so you can compare the advantages of the two methods.

1. From the folder with materials for this chapter, open the assembly called motor subassembly ch10.asm.

2. Flip the assembly over so you are looking at the outside of the gear box cover.

3. Make sure your selection is set to Face Priority (under the Select drop-down list).
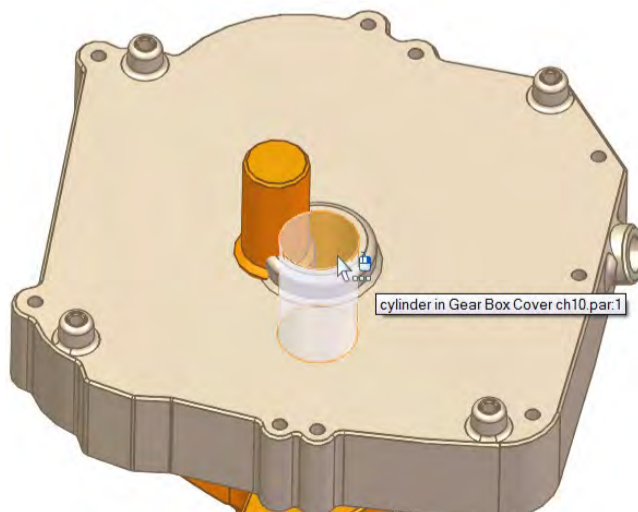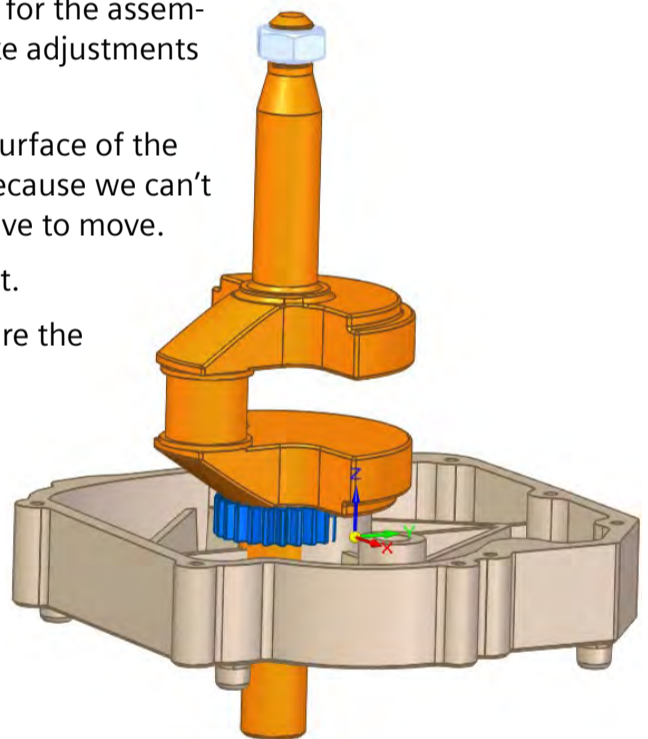
4. Click on the inside face of the hole in the cover, highlighted as shown. Make sure all of your Design Intent options are turned on.

Note: Design Intent does not create relationships – it only assists you in selecting faces. In this case, it is selecting all of the concentric and tangent faces, and adding them to the inside face of the hole in the cover that you selected.

5. Hover over the tool disk, and click.

    The tool disk is the flat washer-shaped surface on the Steering Wheel. This enables you to move the selection set anywhere in the plane of the tool.

6. Hover over an edge of the shaft (orange part), wait for the cursor feedback telling you it has snapped to the center of the shaft, and then click.
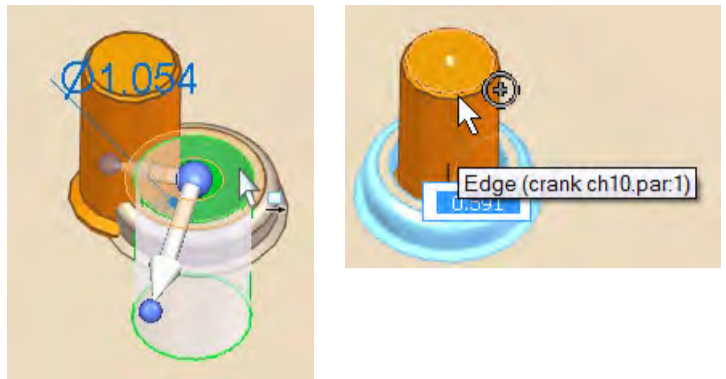
    This was the easy way. You just made an in-context move of a feature in one part to a feature in another part without creating any associative links between the parts. Congratulations!

7. Press Ctrl+Z to undo the move. we're going to make the move associatively now.

8. To make sure you don't have any open windows of the gear box cover part, press Ctrl+Tab, holding down the Ctrl, and releasing the Tab. If a window is open, switch to that window and close it. If you're still holding the Ctrl button, you can switch to another window by pressing the Tab key. After you closed any cover part windows, return to the assembly.
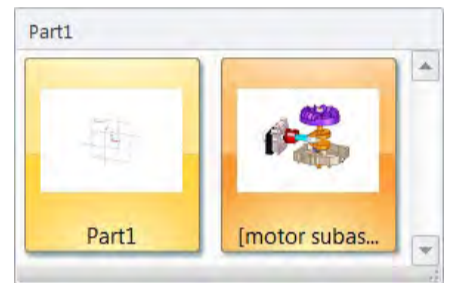
9. Right-click the cover part and select Edit. If the software displays the individual part window, repeat step 6 (close the cover part window) and try it again.

    If the rest of the parts in the assembly disappear, but you are still in the assembly document, go to the **View** tab -> **Show** group (far left by default), and click the Hide Previous Level button. This makes it easier to work on assemblies with multiple levels of subassemblies, but in this case, it is just getting in the way. When you turn this off, the other parts in the assembly should be gray and transparent.
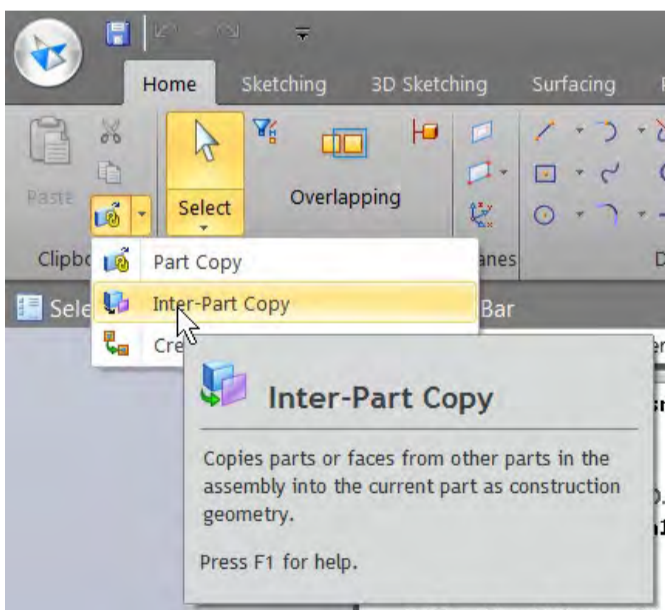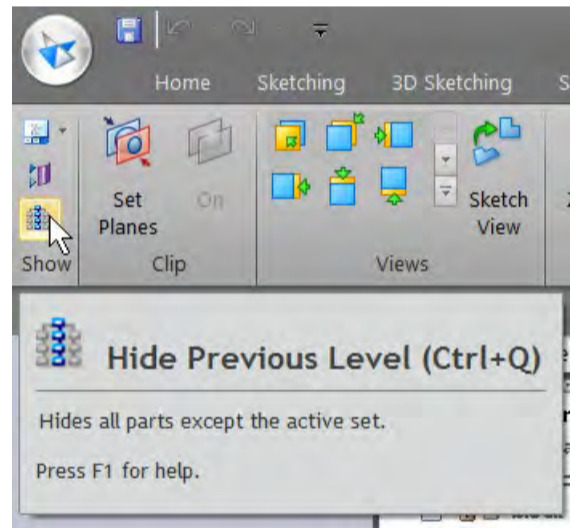
10. Use the Inter-Part Copy command to copy the following faces into the cover part:
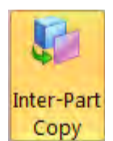
11. Copy faces for the part of the shaft that is near the gear box cover, as shown on the right. You can box-select faces. Use the Quick View Cube to orient the view, and then drag a box to select the faces.

12. Copy the end face of the crank gear part.

    Inter-Part Copy works while you're editing a part in the context of an assembly.

    To copy:

    a. Click the Inter-Part Copy command.

    b. Select the part from which you want to copy faces.

    c. Select the faces you want to copy.

13. Click Accept.

14. Exit part edit mode by clicking the X in the upper right corner of the graphics window.

15. To keep the changes, save the assembly.

16. Right-click the cover part and click Open.

    Solid Edge opens the cover part in its own window.
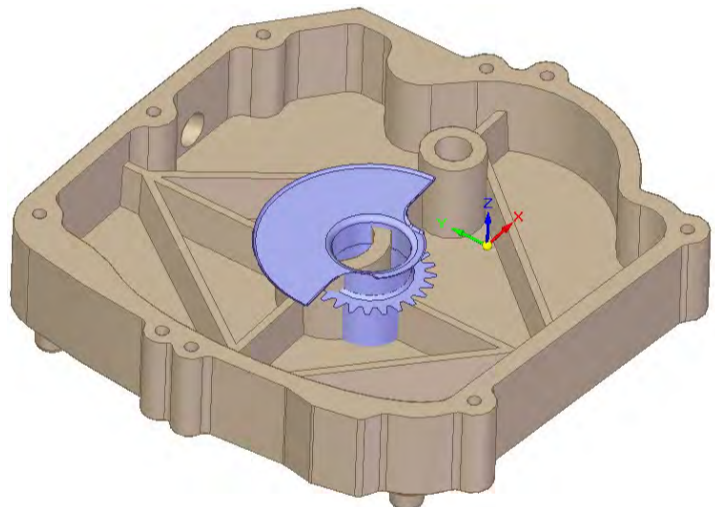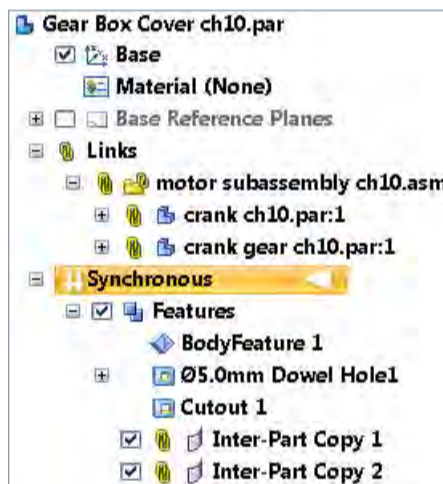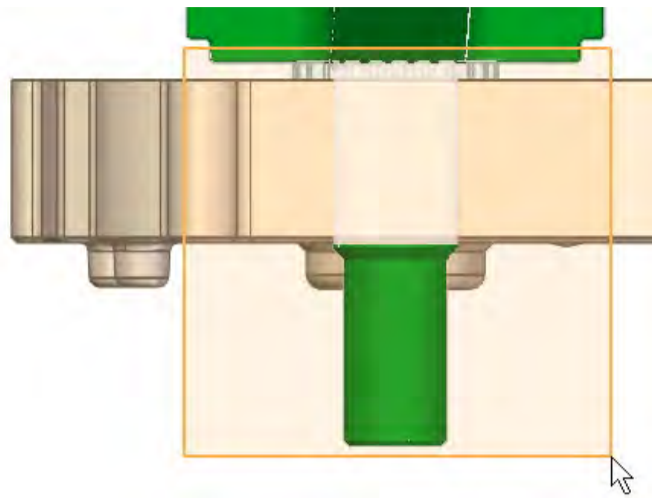
17. Verify that the cover part now has the surfaces from both the shaft and the gear, as shown. The blue surfaces are the copied ones.

    If you open the cover part without saving the assembly, the software informs you that you have unsaved changes, and you may lose the changes if you don't save them.

    You may notice some new items in the PathFinder: the Links collector contains the associative links in the Features area and two new Inter-Part Copies.

    In the assembly PathFinder, the Gear Box Cover part has a new chain link symbol that denotes the associative link.

   Now that you have the links, you can continue with the edits.

18. Move the hole with the bearing surface the same way you did earlier in step 5, using the tool plane.

    Next you put a little clearance between the end of the bearing surface and the gear on the inside.

19. Rotate the part so you can see the inside, then click on the end face of the bearing surface as shown:
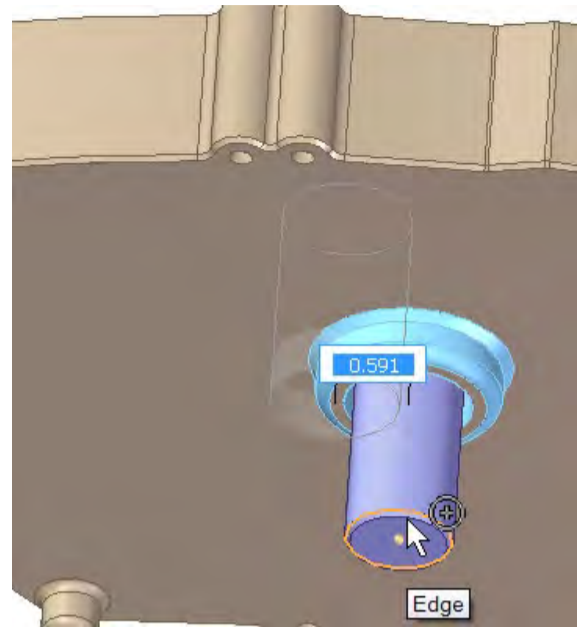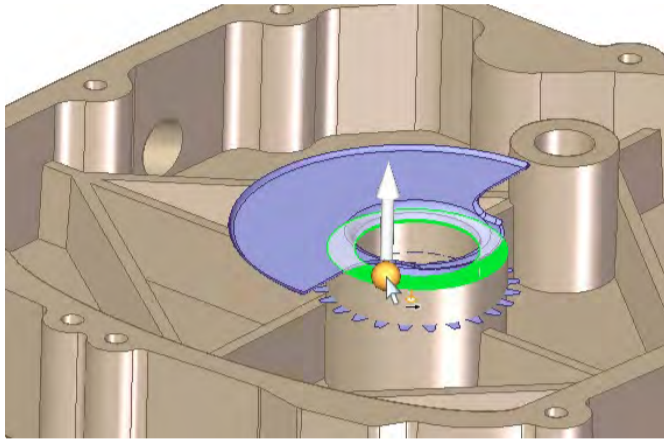
20. Shift+Click on the vertical Steering Wheel arrow, and move the cursor up slightly. Release the Shift key, if you haven't already, enter .05, press the Enter key, then click in white space.

    This series of steps has put the steering wheel 0.05" above the bearing face, and when you move the Steering Wheel to the face of the gear, it will leave a gap of 0.05" between the bearing face and the gear. This is a technique that is commonly used in synchronous edits to create gaps or overlaps.
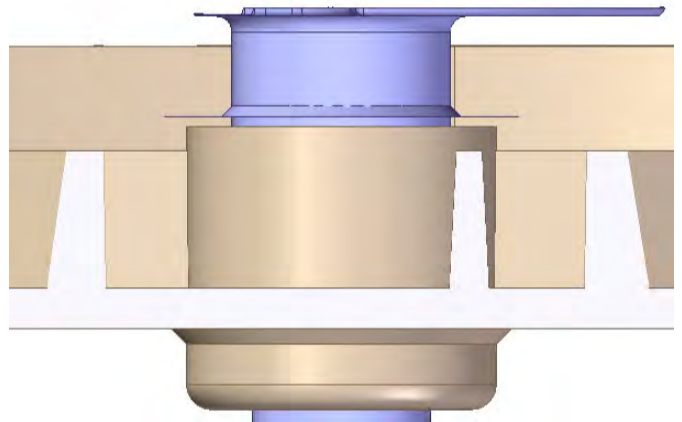
21. Click the vertical arrow once more, and then click any end point on the face of the gear. Using clipping planes, you can see that the move was successful.

    Next, you shape the inside of the hole on the cover to match the shaft.

22. Hide both of the Inter-Part Copy features in the PathFinder.

23. Click the Replace Face button on the **Surfacing** tab -> **Modify Surfaces** -> **Replace Face**.

24. Follow the instructions in the Prompt Bar. The first face you select should be the inside of the hole on the cover part, then right-click to accept.

25. From the PathFinder, show the Inter-Part Copy feature copied from the shaft, and select it in the graphics window.

    As soon as you click the Inter-Part surface, the operation is complete.

26. Click Cancel or press the Esc key.
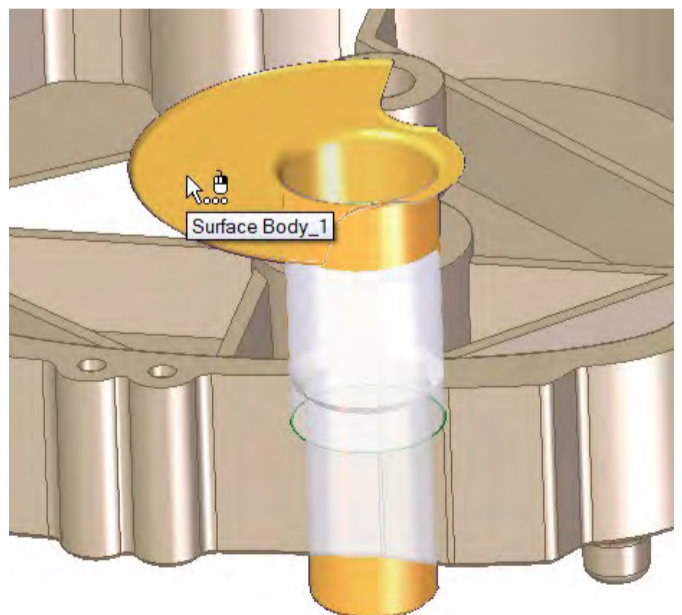
27. To create a bit of clearance to allow the shaft to rotate, click on the inside larger diameter face, then click on the dimension that appears and change it from 1.063 to 1.066.

   Note: Synchronous does not have a steering wheel method for changing radius or diameter values, so those values are changed using dimensions.
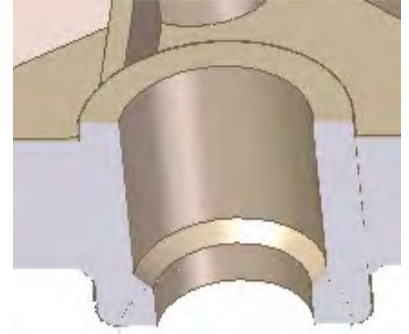
   Next you must move the oil drain because it doesn't touch the bottom face.

28. Select the outer cylindrical face and the chamfer face.

29. Move the Steering Wheel to a model edge or face so that it points straight down, not down at an angle.

   Tip: Move it by dragging the origin – blue ball in the center.

30. Click the arrow pointing down, move the cursor down, and enter 0.215.

31. To display the assembly window again, press Ctrl+Tab.

32. Use the PathFinder to show the Gasket and Block Casting parts again. Turn off the crank and the crank gear.
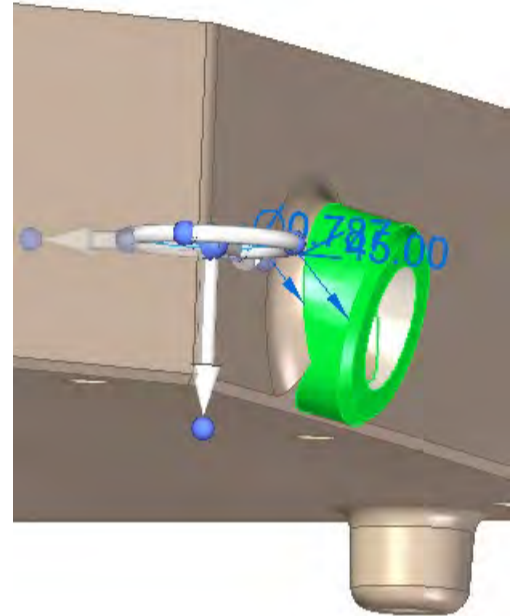
33. Use the Quick View Cube to display the Top view, and zoom in on the area shown next.

34. Switch to Select Face Priority, right-click the top-level assembly name in the PathFinder, and then click Activate. Solid Edge activates all parts in the assembly.
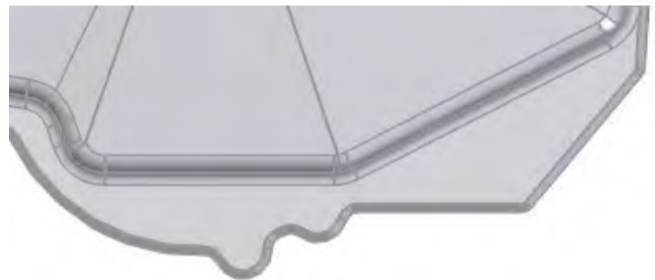
35. Drag a selection box as shown, to select all the faces of the bolt lug nearest the center of the shaft. When selected it looks as shown next.
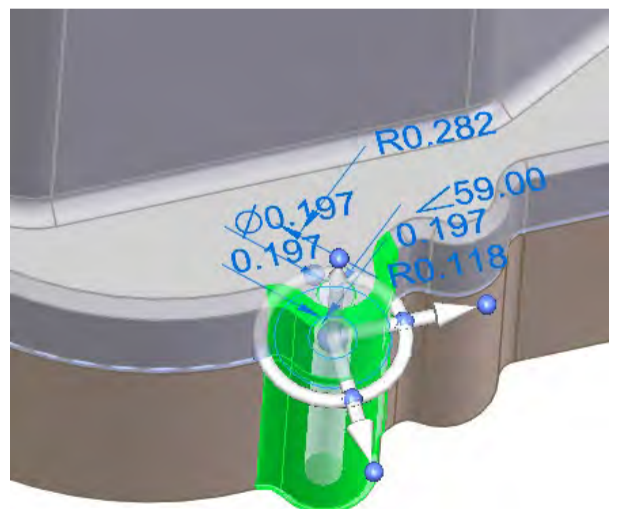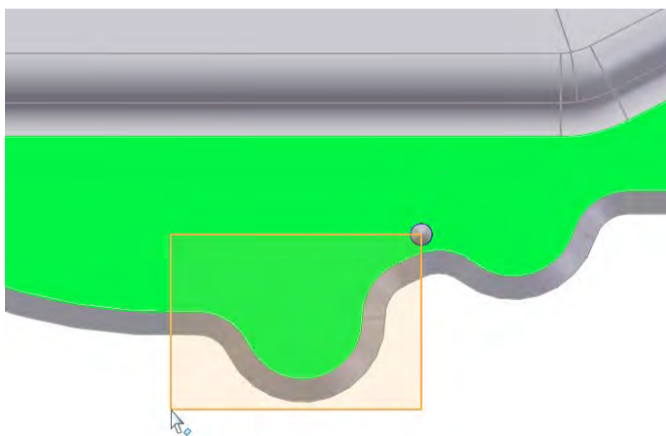
36. Move the Steering Wheel to a circular edge, such as the one near the cursor in the image to the right, will cause the Steering Wheel to snap to the center of the arc. In this position, use the torus to rotate the selection 45° clockwise.

This has moved features of three imported parts together, with no previously established design intent or associativity.

Chapter 10 | Synchronous, parametrics and associativity
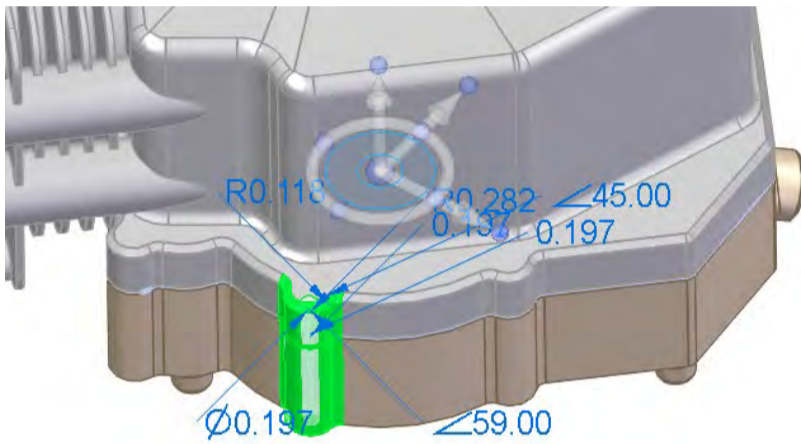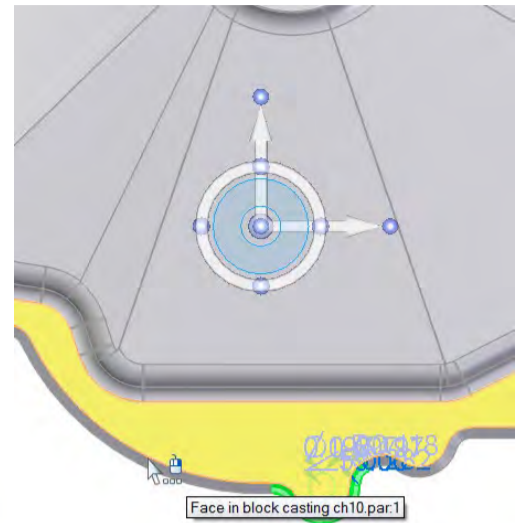
Summary

Synchronous technology makes extensive use of both parametrics and associativity. Parametrics, as in history-based software, is driven by relationships and dimensions. In synchronous, the parametrics are attached to the actual 3D model rather than the 2D sketches. Associativity is something that synchronous technology was designed to minimize, but links between parts in the assembly can still be achieved with Inter-Part Copy functions and even tables driving dimensions from multiple parts.

Chapter 10 | Synchronous, parametrics and associativity

Solid Edge with synchronous technology
Steering a new course in 3D design

About the author

Matt Lombard is a mechanical engineer with over 20 years of experience working on range of designs from manufacturing fixture design to complex plastic assemblies. Matt has worked in roles from associate engineer to engineering services manager, and independent design consultant for companies ranging from independent inventors to Rubbermaid Commercial Products. During that same period he wrote his initial blog, Dezignstuff.com, which engaged the technical side of the CAD community, and several books ranging from a general CAD desk reference to surfacing and CAD Administration, including printed books and video series. Since joining Siemens PLM Software in 2013, Matt has worked as the community manager for the Solid Edge online community where he writes a technical blog and oversees the operation of the Solid Edge Community Forum.

Acknowledgements