Chapter 15

Frame Generator

Frame Generator consists of several tools to automate frame modeling. You can select lines, edges, and points to specify the location of members. Frame Generator derives the selections into a part. This part is called a *skeleton* because it provides the framework for the members. The skeleton part automatically updates when a change is made to the original geometry, which updates the frame member size or position.

Frame Generator gets structural profiles from Content Center. In addition to the structural profiles included in the Autodesk® Inventor® libraries, you can author and publish your own profiles. This capacity is useful for adding profiles of extruded aluminum, plastic, and other materials because the structural profiles in the Inventor libraries are standard steel shapes.

In this chapter, you'll learn to

◆ Work with frame files

◆ Insert frame members onto a skeleton model

◆ Add end treatments to frame members

◆ Make changes to frames

◆ Author and publish structural profiles

◆ Create BOMs for Frame Generator assemblies

Accessing Frame Generator Tools

The Frame panel, shown in Figure 15.1, is on the Design tab of the Assembly ribbon. It has tools specific to Frame Generator plus the Beam/Column Calculator from the design accelerators.

FIGURE 15.1
Frame Generator tools on the Design tab of the Assembly ribbon
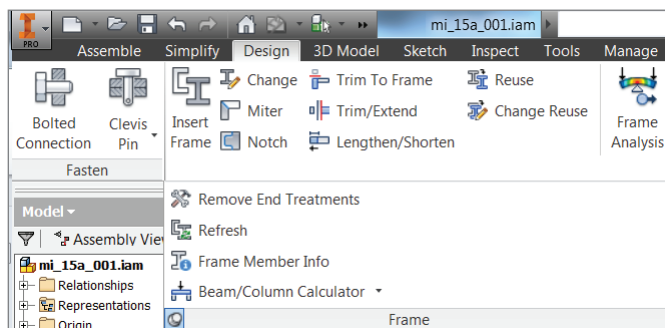
The tools fall into four categories:

Creating Frame Members   Frame members can be created using the Insert Frame and Change tools.

Creating End Treatments   End treatments can be created with these tools:

◆   Miter

◆   Trim To Frame

◆   Trim/Extend

◆   Notch

◆   Lengthen/Shorten

◆   Remove End Treatments

Performing Maintenance   Maintenance can be performed with the Frame Member Info and Refresh tools.

Performing Calculations and Analysis   Calculations and analysis can be performed with the Beam/Column Calculator.

ABOUT FRAME GENERATOR

The Frame Generator application is an add-in that uses the Inventor application programming interface (API). Since the API does not provide access to all the Inventor functionalities, there are some user interface differences between add-ins and core Inventor tools. For example, the edit fields in the core Inventor tools have an extensive flyout menu. The add-ins do not have access to this functionality, so their edit fields are more limited.

## Exploring the Frame Generator File Structure

When you create the first members in a frame assembly, a dialog box prompts you for filenames. Frame Generator creates a subassembly and a skeleton file in the parent assembly. The subassembly does several things. It acts as a container for the skeleton and frame members, isolating them from the assembly solver, and it acts as a filter so Frame Generator tools, such as Frame Member Info, ignore other assembly components. The skeleton file is created in the subassembly and consists of all of the edges and points you select when placing frame members. Each frame member is created as a separate file and saved into a subdirectory named after the frame assembly.

As an example, imagine you create a cube to use as the basis on which to model a frame. You name the cube file **Cube_Frame** and then place this cube into an assembly called **Cube_Frame_Assembly**. Both files are saved in a folder called My_Frames. You could then use the cube to create a basic frame, using each of its edges as a placement reference for the frame members. When you create the frame, you will be prompted to supply the following items:

◆   A new frame filename and location; these are the name and location of the frame subassembly.

◆   A new skeleton filename and location; these are the name and location of the frame skeleton reference.

If you just accept the defaults, you will end up with a folder such as \My_Frames\Cube_
Frame_Assembly\Frame\. In this folder (once you have saved the top-level assembly), you will
find an assembly file named Frame0001.iam and a part file named Skeleton0001.ipt. Special
attributes in the frame subassembly contain references to the parent assembly. This enables the
frame skeleton to maintain references to the other assembly components. Figure 15.2 shows the
default frame subassembly and skeleton file naming.

FIGURE 15.2
Creating a new frame
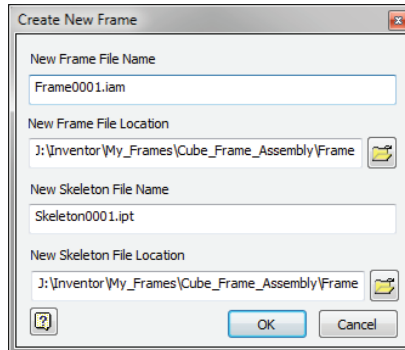
MAKE SURE YOU SAVE

When you click the OK button to create the Frame Generator assembly and place the first mem-
bers, a folder is created in the directory. However, like all Inventor files created in the context of
an assembly, the parts are not written to disk until you save the assembly. This is true when you
create frame members as well, and since it is so easy to create members, it may not be obvious that
you need to save them after the initial creation. Pay attention to the Save reminders and limit the
amount of data you risk losing between saves.

COPYING FRAMES

If you have a completed frame design that you would like to copy, you can use the iLogic Design
Copy tool (see Chapter 20, "iLogic") to do so. Keep in mind that you needn't have used iLogic to
use the iLogic Design Copy tool for copying frames. You can find the iLogic Design Copy tool by
closing all of your Inventor files and then going to the Tools tab and looking on the iLogic panel.

Use the Assemblies list panel in the iLogic Design Copy dialog box to select the frame assembly you
want to copy. All files referenced by the frame assembly will be selected in the other lists. You can
then specify a target folder to copy to and specify a naming prefix or suffix as needed.

Once you start the copy process, the frame files will be copied, and the internal references will be updated.

Note that there are a couple of limitations to be aware of. First, you cannot create more than
one frame subassembly in the same assembly. Another limitation is that you can't use copies
of the frame generated with the assembly Copy tool in other assemblies and maintain Frame
Generator functionality.

Exploring the Anatomy of a Frame Member

Frame Generator initially creates frame members the same length as the selected geometry. When you add end treatments, the length is adjusted to make the member longer or shorter. To accomplish this, the structural profiles are created with a From-To extrusion between two work planes, as shown in Figure 15.3.

FIGURE 15.3

A typical frame member
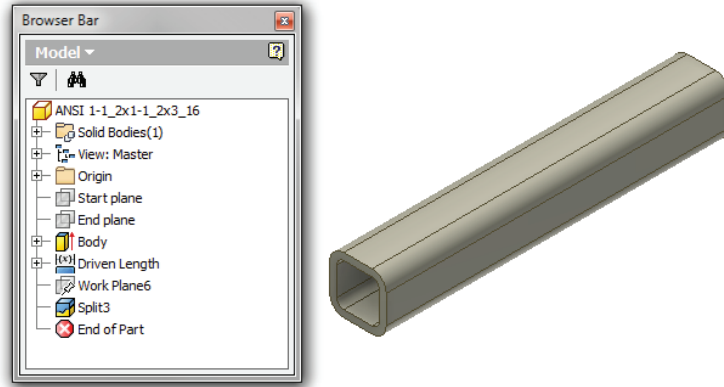
When the part is first created, the start plane is coincident with the XY plane, and the end plane is set to the initial length. When an end treatment is added, the start or end plane is moved to shorten or lengthen the member.

The parameter relationships that control the length are complex. Three parameters drive the length, two parameters are driven by those parameters to determine the length, a reference parameter reports the overall length, and a parameter is used in the BOM, as indicated in Figure 15.4.

FIGURE 15.4

Frame member parameters

Parameter Name	Unit/Type	Equation	Nominal Value	Tol	Model Value	Key	Comment
Model Parameters							
d4	deg	0 deg	0.000000	○	0.000000	□ □	
d7	mm	G_IR	10.000000	○	10.000000	□ □	
d9	mm	G_W	50.000000	○	50.000000	□ □	
d11	mm	G_H	50.000000	○	50.000000	□ □	
d12	mm	G_T	5.000000	○	5.000000	□ □	
d13	mm	-G_OFFSET_START	-0.000000	○	-0.000000	□ □	
d14	mm	G_OFFSET_END + B_L	300.000000	○	300.000000	□ □	
d15	mm	d9 / 2 ul	25.000000	○	25.000000	□ □	
d16	mm	d11 / 2 ul	25.000000	○	25.000000	□ □	
d17	deg	90 deg	90.000000	○	90.000000	□ □	
Reference Parameters							
d18	mm	300.000 mm	300.000000	○	300.000000 □	□	
User Parameters							
G_W	mm	50 mm	50.000000	○	50.000000	□ □	Width
G_H	mm	50 mm	50.000000	○	50.000000	□ □	Depth
G_T	mm	5 mm	5.000000	○	5.000000	□ □	Thickness
G_IR	mm	10 mm	10.000000	○	10.000000	□ □	Radius r1
G_L	mm	d18	300.000000	○	300.000000	□ ☑	Length
G_OFFSET_START	mm	0.00000000 mm	0.000000	○	0.000000	□ □	
G_OFFSET_END	mm	0.00000000 mm	0.000000	○	0.000000	□ □	
B_L	mm	300.00000000 mm	300.000000	○	300.000000	□ □	
MAS	kg/m	6.56 kg/m	6.560000	○	6.560000	□ □	

Table 15.1 lists the length parameters.

TABLE 15.1:   Frame member parameters

| PARAMETER	DESCRIPTION
B_L	The initial length of the member.
G_OFFSET_START	The offset value of the start work plane.
G_OFFSET_END	The offset value of the end work plane.
d13	The parameter for the start work plane. It is driven by G_OFFSET_START.
d14	The parameter for the end work plane. It is driven by G_OFFSET_END.
d19	A reference dimension that measures the overall length of the part.
G_L	The length parameter that is used in the BOM. It is equal to the reference dimension.

Inserting Frame Members

The process for inserting frame members can be broken down into three basic steps. You select the frame member profile (this comes from Content Center), select the placement geometry for placing the frame members, and then adjust the orientation of the frame members.

Specifying a Structural Shape

The left side of the Insert dialog box, shown in Figure 15.5, has a series of drop-down fields for specifying the structural shape.

FIGURE 15.5
Frame Member
Selection group
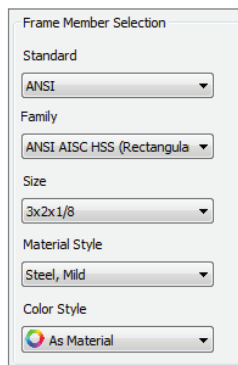
You use the Standard, Family, and Size fields to select the member from Content Center. These fields are progressive from top to bottom, with one updating when changes are made to

the one above it. However, the update behavior varies depending on the field selected. If you select a new standard, the first family is automatically selected. If you select a new family, the size is not automatically selected.

Changing the Orientation

After you have selected the placement geometry, you can change the position and orientation of the member. A thumbnail of the profile is displayed in a grid of radio buttons that control the position of the member, as shown in Figure 15.6. These positions are based on the rectangular bounds of the profile. As a result, the corner positions of a pipe that's 1 inch in diameter are the same as they are in a 1-inch-by-1-inch square tube.

FIGURE 15.6
Orientation group
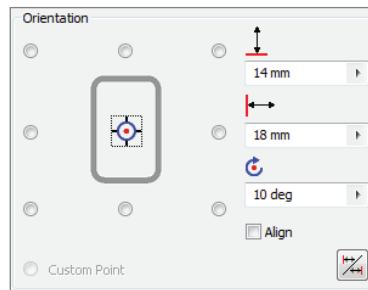
You can fine-tune the placement position by entering values in the horizontal and vertical offset fields. You can also rotate the member. For example, food processing equipment frequently has horizontal members rotated 45 degrees so spilled food doesn't build up on top of square tubing.

The Mirror Frame Member button is shown in the lower-right corner of Figure 15.6. This button is used for profiles that don't have rotational symmetry, such as C-channel and angle iron. The orientation changes affect all the members of a select set. Depending on the geometry, it might be more efficient to use a batch select tool and change the orientation of a few members afterward, or you might want to select only those members that have a similar orientation.

Since structural shapes are extruded, Frame Generator needs a method for determining the extrude direction. When an edge is selected, Frame Generator uses the closest endpoint as the start of the extrusion. Depending on where you select an edge, the same radio button can cause the member to be in a different position. The thumbnail is the view of the profile looking at the XY plane. It takes some practice to get a good feel for the relationship between how an edge is selected and the behavior of the radio buttons. Once you understand this relationship, you will be able to predict the behavior and use it to increase your productivity.

"DO I NEED TO USE 3D SKETCHES WITH FRAME GENERATOR?"

Although you can use 3D sketches to base a frame on, you don't need to do this. You can create 3D solids or surfaces to create a frame base. Once you have a 3D shape, you can then add 2D sketches to the various faces and use those sketches as selection edges as well. So, you don't need to be well versed in the 3D sketch tools to create 3D frames.

When you're using a custom profile with an alternate insertion point defined, the Custom Point control is enabled. This adds another insertion point to the nine standard ones. The custom point is not displayed in the thumbnail image, so you should confirm that the preview is in the expected position relative to the selected edge. Figure 15.7 shows a profile with an alternate insertion point.

Figure 15.7
Profile with an alternate insertion point
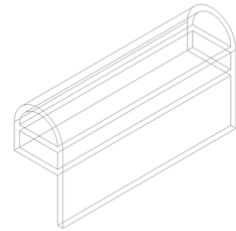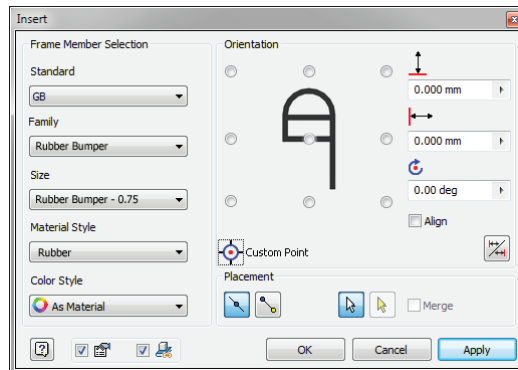
Selecting Placement Geometry

When selecting placement geometry, you can select edges of 3D models and visible sketch lines, or you can select two endpoints. For instance, if you had a cube-shaped base solid, you would use the edges to place vertical and horizontal frame members. To place diagonal cross bracing, you would use the corner endpoints.

Insert Selection Methods

When you use the default Insert Members On Edges option, you can select edges and lines for placement references. Using edges for placement allows you to insert multiple members at once. When you use the Insert Members Between Points option, you select two vertices or endpoints. This method allows you to place only one member at a time. The most common placement method is by selecting lines and edges. This allows the most flexibility in geometry selection and the use of batch select tools.

There are two philosophies for placing frame members. Some people like to place frame members individually, making sure each one is in the correct position and orientation. Others like to place as many members as possible and then edit them as necessary. The method you choose will depend on the type of models you work with, how much effort you put into setting up the skeleton models, and, most importantly, the way you like to work.

Batch Select Tools

Frame Generator has several tools for selecting geometry. Multi Select is the default selection mode. The standard methods for creating a Multi Select selection allow you to select individual edges, use selection windows, and use the Shift and Ctrl keys to add and remove objects to or from the selection. In addition to Multi Select, two other select modes are available in the context menu shown in Figure 15.8; they are Chain Select and Sketch Select.
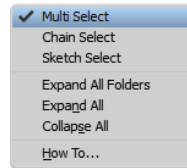
Chain Select automatically selects all lines and edges that are tangentially connected to the selection. Chain Select will not follow past a point that has multiple lines or edges, even if one of them is tangential. For instance, if you have a rectangular sketch profile with rounded corners, you can use Chain Select and select just one of the lines or arcs, and all of the others will be added automatically. By contrast, if you have the same profile in a 3D shape, Chain Select will not automatically select the edges because each edge of the 3D shape has multiple edge intersections.

Sketch Select selects all the lines in a sketch. You can select the sketch in the browser or click a line in the graphics window. For instance, if you have a ladder-shaped sketch, you can use Sketch Select and automatically select the rungs and rails all at once.

The Merge option is enabled when there are connected lines or edges. Merge combines the selections into one member. Although Merge is useful when you want to have one continuous member, you cannot add end treatments to merged members.

Creating a Basic Frame

To create an elementary frame, you'll use a prepared file that has been set up for you. If you have not already downloaded the Chapter 15 files from www.sybex.com/go/masteringinventor2015, please refer to the "What You Will Need" section of the introduction for the download and setup instructions.

The prepared file you'll be using is an assembly file consisting of just two parts and the frame subassembly. One of the parts contains an unconsumed sketch that you will use to place frame members. The frame members will be automatically placed in the predefined frame subassembly. When creating your own frame designs from scratch, you'll be asked to supply the name and locations for the frame subassembly and skeleton files. In this design, you'll add frame members to be embedded in a monument base to support a sign plate:

1. On the Get Started tab, click the Open button.

2. Browse for mi_15a_001.iam located in the Chapter 15 directory of your Mastering Inventor 2015 folder and click the Open button.

3. From the Design tab, click the Insert Frame button.

4. Set the Standard drop-down box to use the standard of your choice.

5. Set the Family drop-down box to use a square tube or square hollow section profile.

6. Set the Size drop-down box to use an 80 mm×80 mm (or 3"×3") section, using a wall thickness of your choice.

7. Select the middle vertical sketch line on the back of the sign plate in the model.

8. Use the radio buttons in the Orientation area of the Insert dialog box to orient the frame member so it matches up with the square cutout in the base.

9. Select the other four vertical sketch lines as well. Note that the two end members are not centered in the cutouts.

10. Holding the Ctrl key, select the two end members to remove them from the selection.

11. Click the OK button to generate the frame members; then click the OK button in the Frame Member Naming dialog box to accept the defaults.

12. Click the Insert Frame button again and click the bottom half of one of the outside vertical sketch lines.

13. Use the radio buttons in the Orientation area to orient the frame member so it matches up with the square cutout.

14. Select the bottom half of the other outside vertical sketch line and notice that the frame member preview does not match the cutout.

15. Using the Orientation radio buttons, you'll notice that none of the solutions allows both members to match the cutouts.

16. Holding the Ctrl key, click one of the frame members to remove it from the selection.

17. Select the sketch line again, this time toward the top, and you'll see that this flips the orientation to provide the correct solution.

18. Uncheck the Prompt For File Name check box (found along the bottom of the dialog box). This suppresses the filenaming dialog prompt and automatically accepts the default naming for the frame members.

19. Click the OK button. Your assembly should resemble Figure 15.9.

FIGURE 15.9
Frame members placed
using Frame Generator
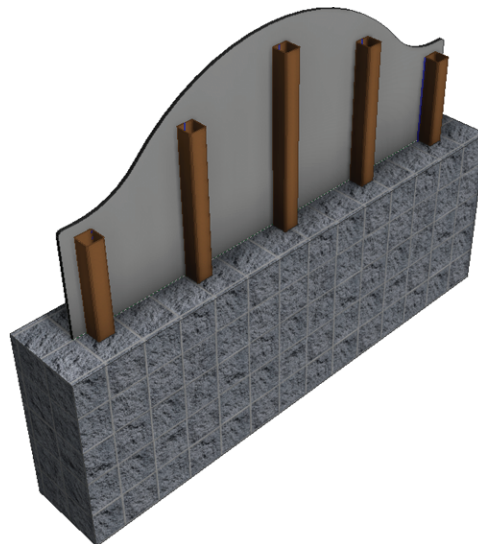
In this simple example you explored the basics of placing frame members, noting the use of the Ctrl key on the keyboard to remove selections. You also noticed that orientation of frame members is sometimes influenced by the location of your lines or edge selections. You can close this file without saving changes and continue to the next section. In the next example, you'll explore more selection and orientation options.


WHERE ARE THE OTHER COMMON SIZES?

You might note that there are several common sizes you use daily that are not listed in the Size drop-down. This is because Inventor includes only the sizes that are ratified to the various structural shape standards. Of course, most mills produce other sizes commonly used in general design. You can add those sizes using the Structural Shape authoring tools.

In the following steps you will explore the offset, rotation, and merge options, as well as the context menu selection options:

1. On the Get Started tab, click the Open button.

2. Browse for `mi_15a_002.iam` in the `Chapter 15` directory of your `Mastering Inventor 2015` folder and click the Open button.

3. From the Design tab, click the Insert Frame button.

4. Set the Standard drop-down box to use the standard of your choice.

5. Set the Family drop-down box to use a flat bar section profile.

6. Set the Size drop-down box to use a 50 mm (or 2-inch) bar, using a thickness of your choice.

7. Right-click in the graphics area and set the selection method to Chain Select.

8. Click any part of the C-shaped sketch and notice that the entire sketch loop is selected; you don't need to click each sketch segment.

9. Set Horizontal Offset to **-100** mm.

10. Set Rotation to **90** degrees.

11. Click the Merge check box to ensure that the result is a continuous piece of rolled flat bar rather than a piece for each segment of the loop.

12. Click Apply to create the frame member.

13. Right-click in the graphics area and set the selection method to Sketch Select.

14. Select any of the segments in the ladder sketch and notice that the entire sketch is selected.

15. Set the horizontal offset to **0** mm and the rotation to **0** degrees.

16. Click Apply to create the frame members.

17. Right-click in the graphics area and set the selection method back to the default Multi Select option.

18. In the Placement area of the Insert dialog box, click the Insert Members Between Points button and then select endpoints of the gap in the C-shaped sketch.

19. Set the rotation to **90** degrees.

20. Click the OK button to create the short length of flat bar and close the dialog box. Figure 15.10 shows the final result.

FIGURE 15.10
Flat bar frame members
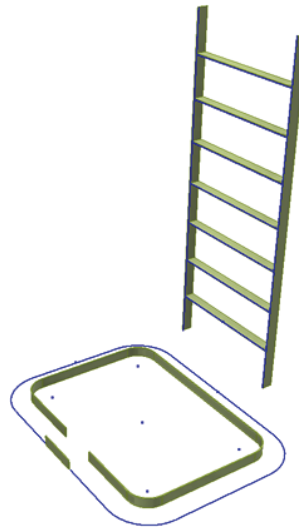
As you can see, the selection options along with the offset, rotation, and merge options allow you to create frame members using just a simple sketch and still get a varied result. You can close this file without saving changes and continue to the next section.

Aligning Frame Members

Frame Generator follows two rules to give a frame member its initial orientation. If you are creating the first member in a selection set, the member is aligned to adjacent geometry or the coordinate system. For the rest of the selection set, Frame Generator tries to align the members to the first selection. These rules work well for most rectangular machine frames. However, if part of the frame is at an angle and there isn't a good reference, Frame Generator might select an orientation that doesn't match your design intent.

The frame in Figure 15.11 has two sketch lines running down from the center point to be used for supports. The members for the base and back have already been inserted, and the two angled supports need to be inserted. When a member is inserted on angled lines, the orientation is skewed, as shown in the inset in Figure 15.11.

FIGURE 15.11
Skewed orientation
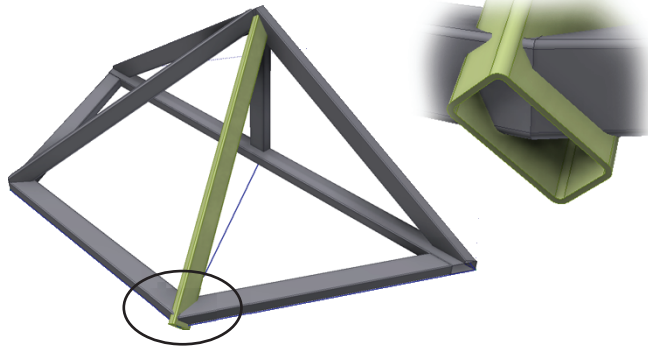
To resolve this, a reference line can be selected first to establish the orientation, and then the angled line can be selected to add the support member and have it follow this orientation. Once the angled line is selected, the reference line can be deselected, and the angled line will hold the proper orientation, as shown in Figure 15.12.

FIGURE 15.12
Using a helper line to establish orientation
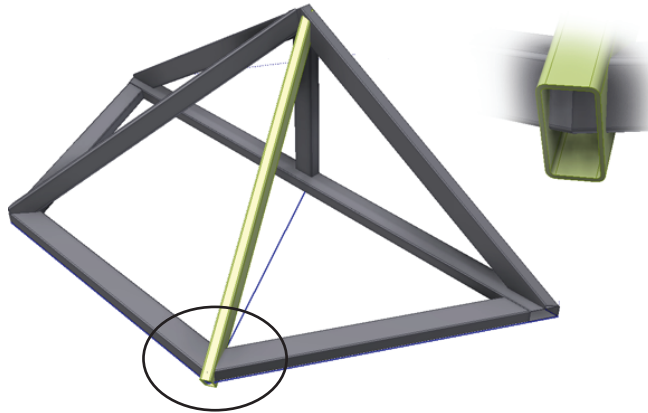
Using reference geometry for angled frame members is a bit of an art. If you regularly create these types of frames, you will develop a feeling for the ways that Frame Generator aligns members, and you will learn when and how you need to add references. To better understand the use of reference geometry for alignment, follow these steps:

1. On the Get Started tab, click the Open button.

2. Browse for `mi_15a_003.iam` located in the `Chapter 15` directory of your `Mastering Inventor 2015` folder and click the Open button.

3. From the Design tab, click the Insert Frame button.

4. Set the Standard drop-down box to use the standard of your choice.

5. Set the Family drop-down box to use a rectangular tube or rectangular hollow section profile.

6. Set the Size drop-down box to use a 100 mm×50 mm (or 4"×2") tube, using a thickness of your choice.

7. Ensure that the placement option is set to Insert Members On Edges and that the offsets and rotation fields are set to zero. Set the orientation to be centered and then click the diagonal line that runs into the frame peak.

8. Click the OK button to create the member.

   Notice how the frame member is coming in at a skewed orientation. To resolve this, you will first delete this member and then place another, but this time you'll use a helper edge to establish the orientation.

9. In the browser, expand `Frame_MI_1503` so you can see the frame members.

10. Hover over each member until you see the one you just created highlighted in the graphics area (it should be at the bottom of the list).

11. Right-click it and choose Delete With Frame Generator.

    Using this option not only deletes the part but also cleans up any end treatments involving other members related to the deleted one.

12. Click the Insert Frame button again. Click the diagonal line that runs into the frame peak first and then select the Align check box.

13. Click the sketch line located in the base of the frame to use as the alignment selection.

14. Adjust the rotation angle by 90 degrees.

15. Click the OK button to create the frame member in the correct orientation.

Keep this alignment trick in mind as you create frames, and you will likely find it to be very helpful. You can close the file without saving changes.

Using the Change Tool

You can use the Change tool to change the standard, family, size, material, color, and orientation of an existing frame member. You can use the check boxes to set the various options to allow or prevent changes. To see the Change tool in action, follow these steps:

1. On the Get Started tab, click the Open button.

2. Browse for `mi_15a_004.iam` located in the `Chapter 15` directory of your `Mastering Inventor 2015` folder and click the Open button.

3. From the Design tab, click the Frame panel drop-down arrow.

4. Select the Frame Member Info button from the list.

5. Click one of the shorter uprights in the frame. The Frame Member Info dialog box displays the information about this part. Click Done to close the dialog box.

6. From the Design tab, click the Change button.

7. Click one of the shorter uprights in the frame again and then select the other one. Notice that the selection allows only one frame member to be selected at a time.

8. Click the Multi Select check box in the Change dialog box and then select the other short upright member.

9. Click the Change Orientation check box and then set the rotation to **45** degrees.

10. Change the Appearance setting to Cadet Blue or something similar and then click the OK button.

This simple exercise demonstrates the use of the Change tool to modify frame members. Keep in mind that if you make a change that requires a new file to be generated, such as changing the size, the old part will be replaced with a new one but the old file will remain in the folder in which it is saved. You can close this file without saving changes.

Adding End Treatments

The end treatments are among the most powerful Frame Generator tools. As you add end treatments, the frame member length automatically updates. The end treatments also carry over if you change the frame member to a different profile. End treatments are listed under the frame member node in the browser and can be accessed by expanding the browser node, as shown in Figure 15.13.
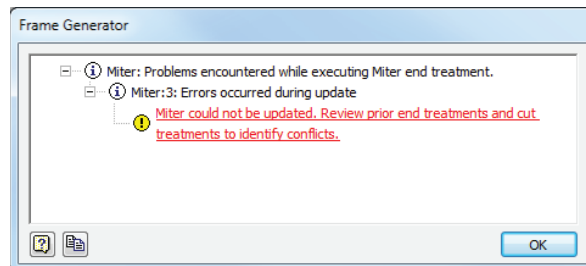
FIGURE 15.13
End treatments in the browser
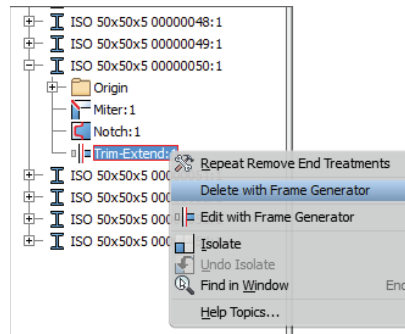
You can delete end treatments from the browser, or you can use the Remove End Treatments tool found on the Frame drop-down list of the Design tab. When you use the Remove End Treatments tool, all end treatments are stripped from the selected frame member.

If a member has end treatments on it already and you attempt to add another, you will need to select the Delete Existing End Treatment(s) check box in the dialog box; otherwise, an error such as the one shown in Figure 15.14 may be created.

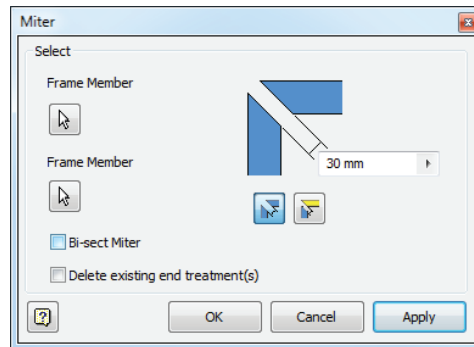**FIGURE 15.14**
Conflicting end treatment error

If you do encounter an error, simply undo the change and reapply the end treatment, this time using the Delete Existing End Treatment(s) option.
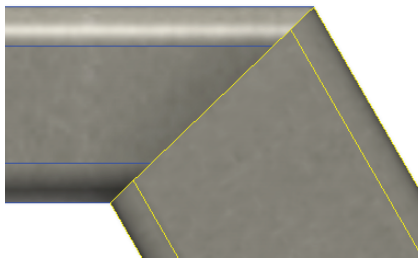
Miter

The Miter end treatment makes angle cuts on two members. Figure 15.15 shows the Miter dialog box. You can miter multiple members by applying the end treatment to each pair of members. You can add a gap between the members by entering a value in the gap input box. The default gap is split between the two members. If you want the gap to be taken from just one member, you can use the Miter Cut At One Side button, and the gap will be taken from the first member you selected. Figure 15.15 shows the gap option buttons and input box.

FIGURE 15.15
Miter dialog box

The default selections cut along an angle, resulting in full-face contact between the members, as shown in Figure 15.16. Bi-sect Miter splits the angle between the members. Figure 15.17 shows that the cut is located where the centers of the two members intersect.

FIGURE 15.16
The standard miter joint has full-face contact

FIGURE 15.17
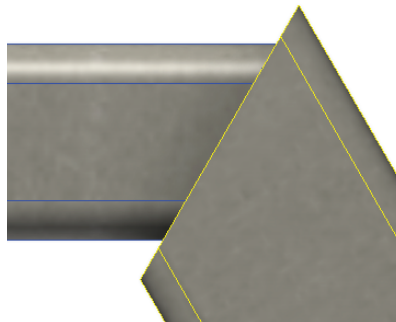The optional bi-sect miter cuts both members at the same angle

Figure 15.18 shows a one-sided gap where the horizontal member was selected first.
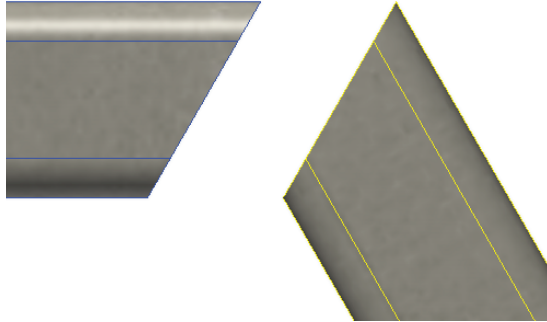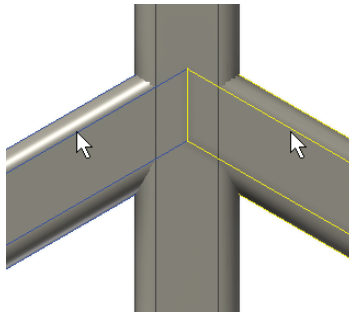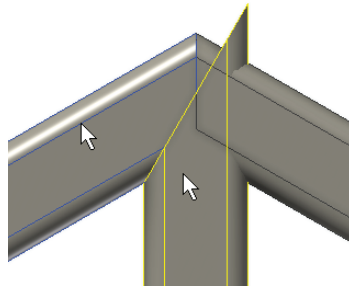
FIGURE 15.18
Miter gap on one
member

Figure 15.19 has a miter end treatment between the two angled members and a vertical member running through them. The vertical member needs to be mitered to fit. Figure 15.19 shows the left member and the vertical member being selected for the first step to create the miter.

FIGURE 15.19
First miter cut

The vertical member still needs a miter to trim the other side, so the miter is repeated with the same settings, this time choosing the member on the right and the vertical member, as shown in Figure 15.20.
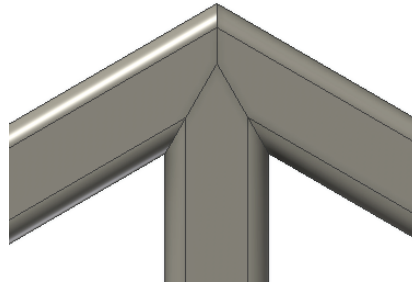
FIGURE 15.20
Second miter cut

The resulting miter, shown in Figure 15.21, requires two cuts on each member. An alternate method that produces a more cost-effective joint can be made using the Trim/Extend To Face tool.

FIGURE 15.21
Resulting miter joint between the three members

You can open the file `mi_15a_006.iam` to explore the Miter tool. Practice creating miter gaps and two- and three-member miters to understand the process.

Trim/Extend to Face

This end treatment is the only one that can trim multiple members at once. When trimming or extending to a face, you select the members you want to trim and then select the cutting face. A separate end treatment feature is created for each frame member. If you edit or delete the end treatment for a particular member, it does not affect the other members. Figure 15.22 shows a typical trim-to-face situation. You would select the vertical member for the member to trim and the top face of the horizontal member as the face to trim to.

FIGURE 15.22
A trim-to-face scenario
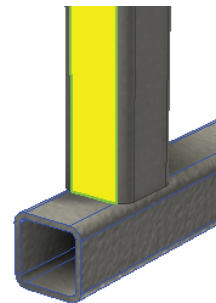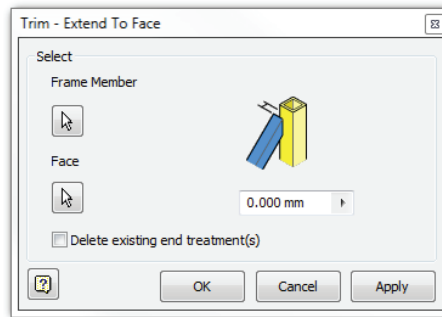
This end treatment can also be used to create miters. Applying miter end treatments to already mitered frame members results in a costly complex joint cut for all three members. Trimming the third member to fit the first two angled members using the Trim/Extend To Face tool results in the less-expensive detail shown in Figure 15.23.

You can open the file `mi_15a_007.iam` to explore the Trim/Extend tool. Remember, you can select multiple frame members to trim to the same face. Experiment with this file as you like— there is no set solution to the final frame outcome.

FIGURE 15.23
Creating a miter
using Trim
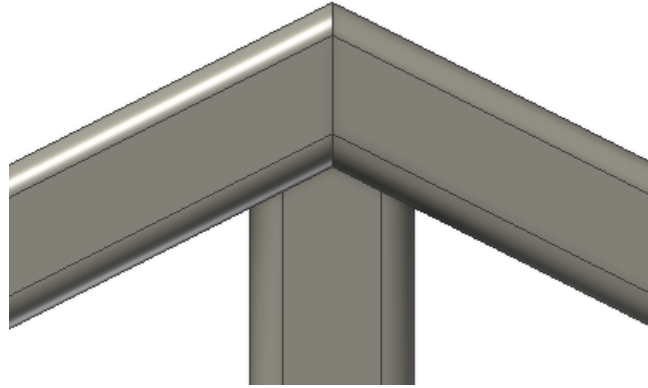
WHICH SIDE IS TRIMMED?

Be aware that when using the Trim tools in Frame Generator, the short end is always the one to be discarded.

Trim to Frame Member

This end treatment trims or extends both members so they are flush. The first selection is made flush to the second selection, and the second selection butts up to the first. Figure 15.24 shows the selections, and Figure 15.25 shows the results.

FIGURE 15.24
Trim To Frame
selections
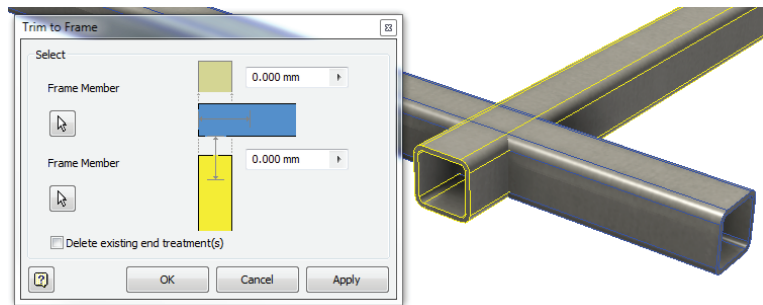
You can open the file `mi_15a_008.iam` to explore the Trim To Frame tool. Experiment with this file as you like—there is no set solution to the final frame outcome. If you cannot get the result you want from the Trim To Frame tool alone, consider the Trim/Extend To Face tool as well. These tools are often used together to achieve a specific result.

Notch Frame Members

The Notch tool cuts one frame member to match the other. It uses the profile to create a cutting surface. By default you can't create an offset, so the cut is an exact match. This is simply

a cut operation, so the frame member is not shortened or extended before the cut. If you have authored a notch profile into a custom frame member library file, you can select the Apply Notch Profile check box; however, if the member does not contain a notch profile definition, the check box is grayed out.
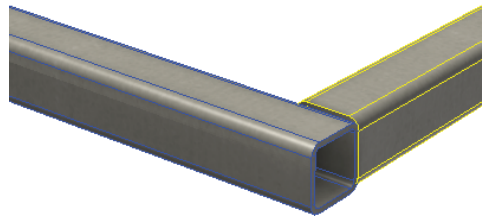
If you were to notch the shorter I-beam shown in Figure 15.26, which extends past the taller I-beam, the extra lump would remain on the left, as shown in Figure 15.27. If the members don't intersect, the notch will have no effect.
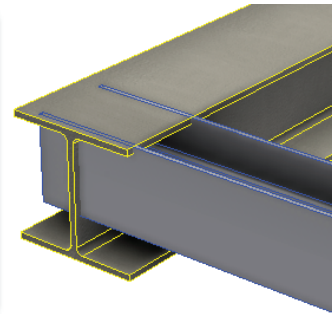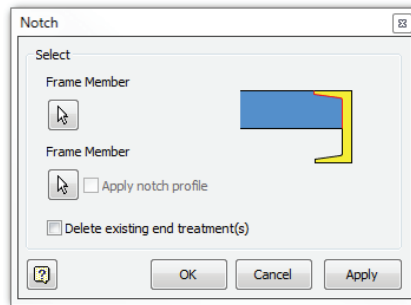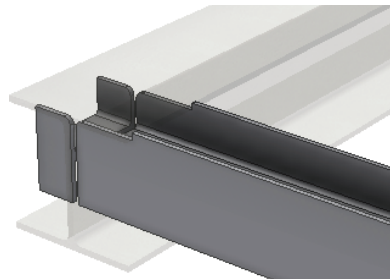
Because of this, it's often best to create a notch as a secondary end treatment. For example, if you had two intersecting frame members as shown in Figure 15.26, it might be best to add a Trim/Extend To Face end treatment first and then use a notch to remove any intersecting material. To explore the Notch tool, open the file mi_15a_009.iam. Use the flanged beams to create notches in the T-shaped members.

> **CREATING ACCURATE NOTCHES**
>
> Notch shapes are driven by manufacturing requirements. Typically, the shape is simplified to reduce cost, and there may be a gap between the members to allow for tolerances. The easiest way to create these shapes is to add a Trim/Extend To Face to make the member the correct length and then add an iFeature to trim the end of the member to the correct shape. You can also define notch profiles in custom frame member definitions.

Lengthen/Shorten Frame Member

You can use the Lengthen/Shorten tool to change the length of a frame member. By default, frame members are initially assigned the length of the reference edge or the distance between selected points. However, you will often need to change the length. Much of the time you will simply use another frame member and one of the Extend or Trim tools. Sometimes, though, there isn't another frame member you can use as a reference, so in those cases you would use the Lengthen/Shorten tool.

When you use the Lengthen/Shorten At One End option, the Extension value is applied to the end closest to the pick point, meaning that if you select a vertical frame member toward the top end and specify an extension length of 100 mm, that value will be applied to the top end. When using the Lengthen/Shorten At Both Ends option, the Extension value is applied to both ends. So if you applied a 100 mm extension value, the frame member would grow 100 mm on both ends for a total of 200 mm. If you enter a negative value, the frame member will be shortened.

> **FRAME END TREATMENTS ARE NOT EXEMPT FROM DESIGN INTENT**
>
> You should spend time planning your frame design to minimize the number of end treatments required because each end treatment is an opportunity for the model to fail if a change is made. For example, if you create a skeleton to the inside dimensions of a frame, you can offset members during placement so the members butt against one another without adding end treatments.

Reuse Frame Members

When you use the Insert Frame tool, Inventor creates a new part file for each new frame member you insert. This allows you to create separate part files that allow separate end treatments. For instance, you might initially insert two identical frame members but then choose to miter both ends of one but only one end of the other. Having two separate part files allows this.

However, there are often times when you might want to reuse the same frame member over and over. In these cases you would prefer not to have separate part files and could use the Reuse tool to ensure that the same part file is being reused as needed. To see the Reuse tool in action, follow these steps:

1. On the Get Started tab, click the Open button.

2. Browse for `mi_15a_005.iam` located in the `Chapter 15` directory of your `Mastering Inventor 2015` folder and click the Open button.

3. From the Design tab, click the Reuse button.

4. For the Source Member selection, click the longer existing member from the graphics area.

5. Click the longer sketch line to use as the Placement selection, but be sure to select it at the end of the line furthest from the existing short member (near the sketched arc).

6. Click the OK button to create the new frame member and notice the direction of the holes.

7. Next click the Change Reuse button.

8. Choose the member you just created and then click the Reverse Member Direction button; then click the OK button to see the change. Notice the direction of the holes again.

9. Click the Miter tool and select the ends of the two original frame members when they come to an intersection and then click Apply.

10. Still in the Miter tool, click the remaining end of the short member and the corresponding end of the new long frame member (even though it is already mitered) and then click the OK button.

11. Click the Reuse button and choose the shorter member for the Source Member selection.

12. Choose the remaining straight sketch line for the Placement selection and then click the OK button to create the new frame member.

13. Click the Change Reuse button, select the new short frame member, set Orientation Angle to 180 degrees, and then click the OK button.

14. Click the Miter tool and select the unmitered end of the original longer member and the corresponding end of the shorter member; then click the OK button to create the miter.

The Reuse and Change Reuse tools offer powerful options for working with repetitive frame members in your frame designs. Keep these tools in mind when working with frame design, and you can likely simplify your frame bill of materials and save yourself some work along the way.

Here are a few things to know about the Reuse and Change Reuse tools:

◆ The Reuse tool is available only to straight frame members and will not work for curved members.

◆ To convert a frame member created with the Reuse tool to a separate frame member file, you can locate it in the browser and then right-click it and choose Break Reuse Member.

◆ When you use Break Reuse Member, you will lose any existing end treatments found on that member.

◆ Frame members created using the Reuse tool might not be selectable for some end treatment and other frame modifications tools.

◆ If you modify the frame base inputs on which a Reuse frame member relies, such as the base sketch, an error message will display, and you will likely need to delete the Reuse member.

◆ You can delete a Reuse member by locating it in the browser and then right-clicking it and choosing Delete with Frame Generator.

◆ In addition to the Orientation Angle options, you might need to use the Align check box in the Reuse and Change Reuse dialog boxes in order to reuse members so that they are placed in the correct orientation.

When you've finished exploring the Reuse and Change Reuse tools, you can close this file without saving changes and continue to the next section.

Maintaining Frames

Maintaining existing assemblies can be time-consuming. Frame Generator provides several tools that help streamline the process of modifying end treatments and determining how frames were originally designed.

Remove End Treatments

The Remove End Treatments tool removes all end treatments from a selected frame member. You can also select multiple members for the batch removal of end treatments. This is handy if you need to change the end treatments on a few members or if you have to rebuild a frame. You can find the Remove End Treatments tool in the Frame drop-down list indicated by the small black arrow on the Frame panel of the Design tab. (See Figure 15.1 earlier in the chapter.)

Frame Member Information

The Frame Member Information tool is used to query frame members. It displays the family and size information, mass properties, and material. This is a useful tool because it quickly gives you information about a member. For example, it can help you quickly determine the difference in the wall thickness of two similarly sized hollow tubes. Since the tool filters for only frame members, you can use it at any level of the assembly. You can find the Frame Member Information tool in the Frame drop-down list indicated by the small black arrow found on the Frame panel of the Design tab.

Refresh

The Refresh tool is a Content Center tool. It checks Content Center for the latest revision of the members in the frame. If a newer version is available, it will prompt you to replace it. End treatments are retained during refresh, but other features, such as holes, are not carried over to the new member. You can find the Refresh tool in the Frame drop-down list indicated by the small black arrow on the Frame panel of the Design tab.

MANAGING FRAME GENERATOR FILES

Frame Generator uses an algorithm to create default filenames. You can rename the files as you create them, when prompted by the Frame Member Naming dialog. But often this slows the design process and requires you to enter information you may not currently know.

Another approach is to let Inventor apply the filenames automatically. To do so, you can uncheck the filenaming prompt check box in the Insert Frame dialog box, and Inventor will create the filenames and save the files to the location specified when the frame subassembly and skeleton files were initially created. It is a good idea to set the frame subassembly and skeleton filenames and locations to use a well-planned naming scheme, whether you specify the frame member names or let Inventor do so.

If you decide to accept the default filenames, it is a good idea to use the BOM Editor to set the part number iProperty for each file to match your standard. In this way, you have a unique identifier for each part member. You can set two identical frame members to have the same part number so they are rolled up together in the BOM. Using part numbers (you can think of them as mark numbers), rather than the filenames, to manage the frame members provides a more flexible and real-world workflow for the typical frame design.

Once part numbers have been defined, you can also set the assembly browser to use the part number rather than the filenames. This allows you to quickly index individual frame parts more easily. Do this by selecting the Assembly tab, clicking the drop-down list on the Productivity panel, and choosing the Rename Browser Nodes tool.

Although Frame Generator can create frames (and therefore a lot of part files) quickly, once the frame is modeled, you should slow down and take the time to manage part numbers. This allows your design to be managed properly in the detailing and revision stages.

You can find a supplemental PDF file called Frame Generator BOM in the `Chapter 15` directory of your `Mastering Inventor 2015` folder.

Performing Calculations and Analysis

Included in the standard Frame Generator tools are two calculator tools: the Beam And Column Calculator and the Plate Calculator. These calculators are design accelerator tools that can do a simple stress analysis. The Beam And Column Calculator, for instance, can analyze a single beam or column, but it assumes a uniform cross section, so it does not take into account holes or end treatments.

If you have Inventor Professional or Inventor Simulation, you can use the Frame Analysis tools that are part of those packages. You can learn more about the Frame Analysis tools in Chapter 17, "Stress Analysis and Dynamic Simulation."

The Beam and Column Calculator

You can find the Beam And Column Calculator tool in the Frame drop-down list indicated by the small black arrow found on the Frame panel of the Design tab. As with other design accelerator dialog boxes, the message pane at the bottom and the calculation results pane on the right side can be opened and closed by clicking the small >> symbols. You can drag the splitter bar (double gray lines) to resize the panes, or you can double-click the splitter bar to open or close the panes.
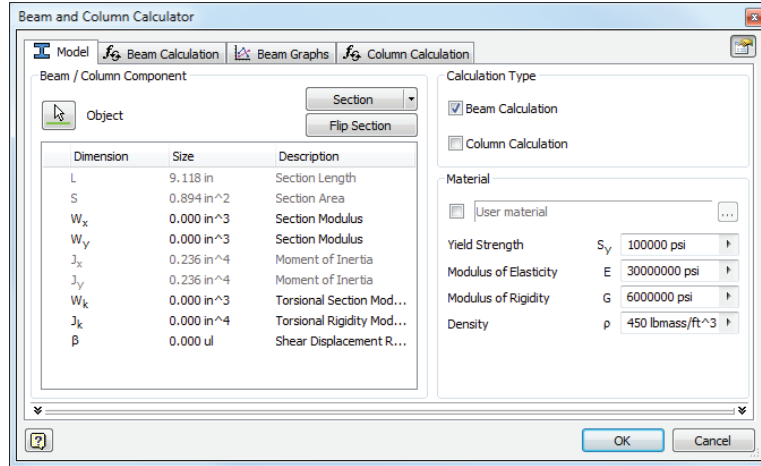
CALCULATING SECTION PROPERTIES

As you explore the subjects in the following pages, you can refer to the conveyor assembly in the file `mi_15a_014.iam`. Take a moment to open this file from the `Chapter 15` directory to become familiar with it. You will use this file to complete the exercise steps presented later.

Select an Object

In this assembly, the Beam And Column Calculator might be used to calculate the loading on the power roller supports. When one of the gold tubes is selected, the calculator automatically loads the section properties from Content Center, as shown in Figure 15.28. Although Content Center has most of the section properties, be aware that some data is missing. You can use several methods for determining the properties.

FIGURE 15.28
The section properties
for a selected object are
loaded from Content
Center

Use the Region Properties Tool

Inventor has a tool to calculate the properties of a closed sketch profile. If you want to use this tool for the section properties of a frame member, you can open the frame member, place a sketch on one end, and project the face. Once you have the profile, select the Inspect tab, and click the Region Properties tool on the Measure panel. Select the profile you want to analyze and click Calculate. The basic region properties for any closed loop are calculated. You can then calculate the rest of the properties based on those results.

The region properties are calculated with respect to the sketch origin. Depending on the profile, you may have to edit the sketch coordinate system to locate the sketch origin at the center of the profile.

Use the Section Button

Another option for calculating section properties is to use the Section button in the Beam And Column Calculator. When you click the button, a list of geometric shapes appears. When you select a shape, a dialog box like Figure 15.29 appears and allows you to enter dimensions. The calculated properties assume sharp corners and constant thickness, so the results won't be accurate for profiles with tapered flanges but may be good enough for many applications.

FIGURE 15.29
Calculating sec-
tion properties for a
rectangle
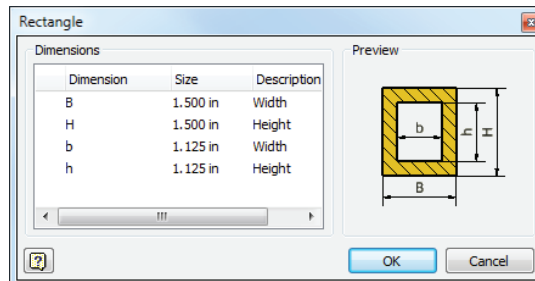
The Flip Section button is used to change the orientation of the x- and y-axes. The z-axis is always in the direction of the extrusion. Gravity is always in the negative y-axis direction, so it is important to make sure the calculation coordinates match the assembly coordinates. If the beam is at an angle, you have a couple of options for handling gravity. You can place a copy of the beam horizontally in the assembly. If you want to ignore the effect of gravity, there is an option on the Beam Calculation tab to turn the gravity load off.

Both Beam and Column calculations are available. The Beam calculations focus on deflection based on loads and supports. The Column calculation checks for buckling. You can select Beam, Column, or both calculation types. The Calculation tabs are turned on and off based on the selections.

The default material properties do not correspond to a material style and are not linked to the style library in any way. Instead, these materials provide you with a starting point and an example of the required properties. You can enter properties for a particular material, or you can select one of the generic materials listed. When you check the box, a dialog box appears with materials such as gray cast iron, steel, and aluminum. These properties can be used for initial calculations, but for more accurate results, you should enter the properties for the particular alloy you are using.

The following steps use the power roller supports found in the conveyor assembly `mi_15a_014.iam`. Follow these steps to enter the member data into the dialog box:

1. Click the Beam/Column Calculator button to start the calculator. Recall that this button can be found in the drop-down list indicated by the small black arrow on the Frame panel of the Design tab.

2. Select the lower, gold-colored support for the power roller.

3. If necessary, click the padlock icon for the Section Length row in the table, thereby unlocking it for editing. Change the value to **12 inch**.

4. Click the Section button and select Rectangle from the drop-down list.

5. In the Rectangle dialog box, enter the tubing dimensions. Enter **1.5 inch** for the outside dimensions (B and H) and **1.125 inch** for the inside dimensions (b and h). Then click the OK button.

6. Select both the Beam and Column calculation check boxes in the Calculation Type area.

7. Click the check box next to the Material field to launch the Material Types dialog box. Select Steel and click the OK button.

8. Note the Yield Strength of 44000 psi.

9. Leave this dialog open because you will use it in the next set of steps, where you'll work with loads and supports.

The coordinate system alignment is correct for this example. So in this case, gravity could be ignored, but having the correct orientation simplifies adding the loads and interpreting the results.

The dialog box should look like Figure 15.30. Note that all the section properties except Shear Displacement Ratio are calculated. This property is optional for the calculations. Comparing the calculated values with the original ones, Section Area and Moments Of Inertia are close but higher.

FIGURE 15.30
Model data entered
into the Beam And
Column Calculator
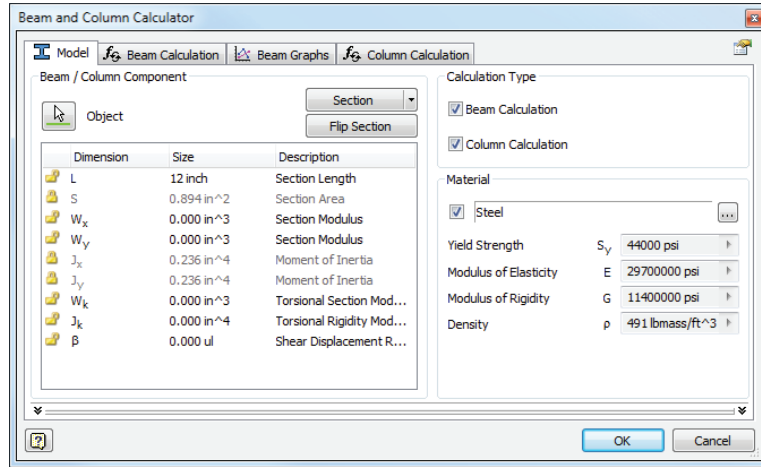
THE BEAM CALCULATION TAB

The Beam Calculation tab, shown in Figure 15.31, has controls for defining the loads and sup-
ports for beams and columns as well as the calculation options. The Engineer's Handbook (hid-
den away in the Power Transmission panel drop-down list; it's called Handbook) contains the
equations used in the calculations. You might want to review those equations before using the
calculator.

FIGURE 15.31
The Beam
Calculation tab
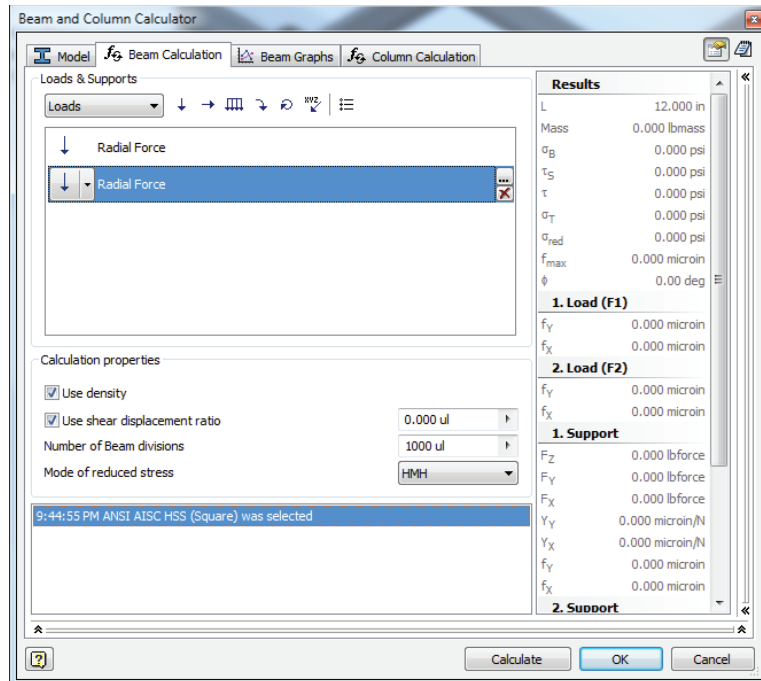
Loads & Supports

The Loads & Supports area contains controls along the top and a browser pane for adding and removing loads and supports for the frame member. All of the controls can be accessed by using the buttons at the top or by right-clicking and choosing them from the context menu in the browser pane area.

The drop-down menu switches the browser between Loads view and Supports view. The controls, shown in Table 15.2 and Table 15.3, change depending on whether Loads or Supports is set to current.

TABLE 15.2:   Loads buttons

BUTTON	DESCRIPTION
↓	Adds a force.
→	Adds an axial force.
(distributed force icon)	Adds a distributed force.
↱	Adds a bending moment—a single twisting force perpendicular to the z-axis.
↻	Adds a torque load—a twisting force around the z-axis. Two equal and opposite torque loads are required.
(XYZ icon)	Adds a combined load—any of the forces added at the same point on the beam.

TABLE 15.3:   Supports buttons

BUTTON	DESCRIPTION
☰	Displays the Options dialog box
(fixed support icon)	Adds a fixed support
▲	Adds a free support
(beam end icon)	Fixes one end of the beam

The Options dialog box, shown in Figure 15.32, gives access to visibility controls for the 2D and 3D previews. By default, the loads and supports dynamically update to maintain the same size as the view scale changes. You can turn off the automatic update and set a static scale value. The Options dialog box is the same whether it is launched from the Loads control or Supports control.

FIGURE 15.32
Loads & Supports
Options dialog box
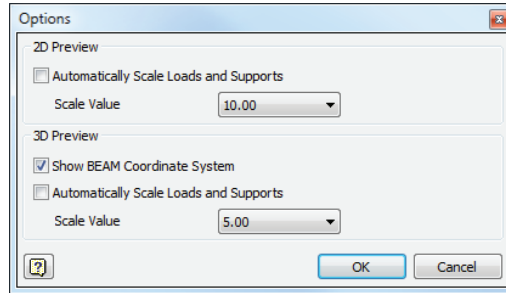
Each load or support can be edited in the browser by double-clicking or by clicking the browse (…) button. A Properties dialog box displays controls for specifying the location, size, and direction of the force.

Calculation Properties

The Calculation Properties group, as shown in Figure 15.33, has four controls that adjust how the calculations are made.

FIGURE 15.33
The Calculation
Properties group
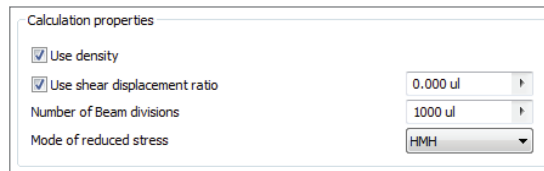
The Use Density check box adds gravity as a load. This is selected by default. The Use Shear Displacement Ratio check box is used when calculating the twist angle caused by torsional loads. The value is determined by the profile shape. It is also called the *form factor of shear*. This check box is selected by default.

The default setting for Number Of Beam Divisions is 1000. Increasing the number of divisions can result in improved accuracy for longer beams. You should experiment with different values to see whether the number of divisions causes a significant change in the results.

Mode Of Reduced Stress has two options for modeling the stress distribution. The Huber-Mises-Hencky (HMH) method is based on the maximum-energy-distortion criterion, and the

Tresca-Guest method is based on the maximum-shearing-stress criterion. The HMH method is the default selection.
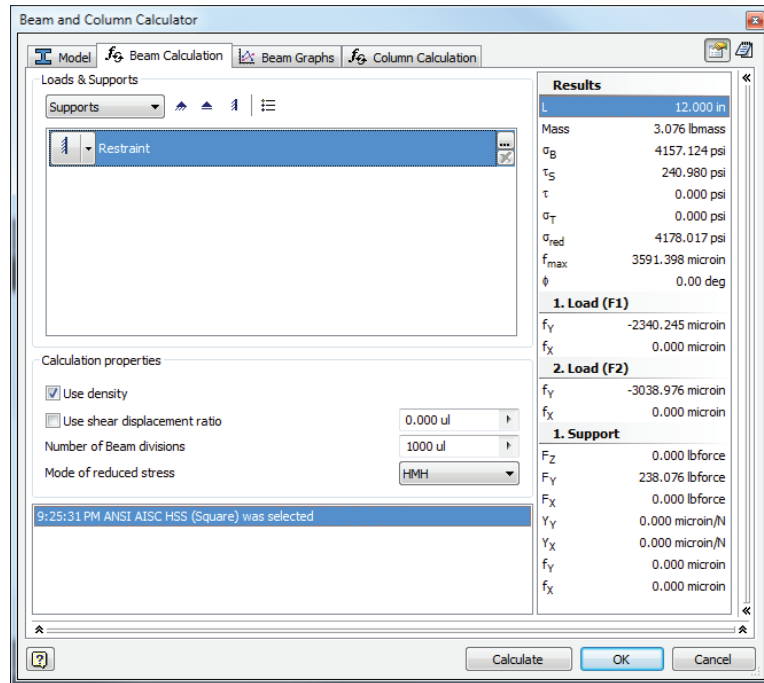
Results

The Results pane on the right side of the Beam Calculation tab updates when you click the Calculate button. Warnings will appear in the lower pane if the calculation indicates that stresses are too high.

For the conveyor example, the support will be welded to the frame at one end and unsupported at the other. The power roller weighs 150 pounds, and the torque is 40 pound-feet. The torque causes the power roller to twist between the supports. The edge of the flat is 1.5 inches from the center of the power roller. This means the reaction force at that point is 320 pounds. Both the weight and the reaction force are split between the two sides. Follow these steps to explore the results and calculation workflow:

1. Ensure that the first load type is set to Radial Force.

2. Click the … button to add a force for the power roller weight.

3. Enter **10** (inches) for the distance and **75** (pounds-force) for the force. This is the maximum distance for the power roller. Click the OK button to close the Radial Force dialog box.

4. Click the Add Torque button to add a torque reaction force.

5. Enter **11.12** (inches) for the distance and **160** (pounds-force) for the force and then click the OK button.

6. Click the drop-down list to switch from Loads to the Supports browser.

7. Delete the Free support by selecting it and then clicking the red X button.

8. Click the drop-down arrow for the Fixed support and select Restraint.

9. Leave the Use Density option checked.

10. Deselect Use Shear Displacement Ratio since you don't have a value for that property.

11. Click the Calculate button and compare the reduced stress to the yield strength of the specified material.

12. Leave this dialog open and use it to explore the next topics.

The dialog box should look like Figure 15.34. Note that the forces are displayed in the graphics window. If you hover over a force arrow on the model, a tool tip displays the information. You can drag the force to a different position, or you can double-click the force to display the properties dialog box.

FIGURE 15.34
Loads & Supports
data entered into
the calculator

THE BEAM GRAPHS TAB

On the Beam Graphs tab, the Graph Selection pane allows you to select the results you want to display. The selected graph displays in the bottom of the Graph area. At the top of the Graph area is a schematic of the beam, supports, and loads. You can drag the supports and loads to different positions to update them. If you double-click a support or load, the properties dialog box displays so you can directly edit the data. The Calculate button is not available on the Beam Graphs tab, so you need to switch back to the Beam Calculation tab to update the results.

The Beam Graphs tab is primarily intended for reviewing results. Twenty-two graphs are available on the tab. This example is a pretty simple analysis. You should experiment with other loads (torques and bending moments) and support types and then view the results on the graphs.

THE COLUMN CALCULATION TAB

The Column Calculation tab checks for column buckling. In the Loads area, you enter the axial load and the safety factor and select a coefficient for the end loading conditions. When you click the … button, a dialog box appears with four end conditions. If you have different end conditions, you should enter the proper coefficient from a reference book.

You shouldn't need to enter any data in the Column area. The length, section area, and least moment of inertia are carried over from the Model tab. The reduced length value is calculated by multiplying the length by the end coefficient.

For example, imagine that during transport, the frame shifts and the power roller supports slam into the trailer wall. The power roller was removed during shipping, so the supports take all the force from the impact, estimated at 4,000 pounds evenly distributed across the four supports. Set the axial load to 1,000-pound force, and click Calculate to determine whether the supports will buckle. Figure 15.35 shows the results.

FIGURE 15.35
Column calculator results
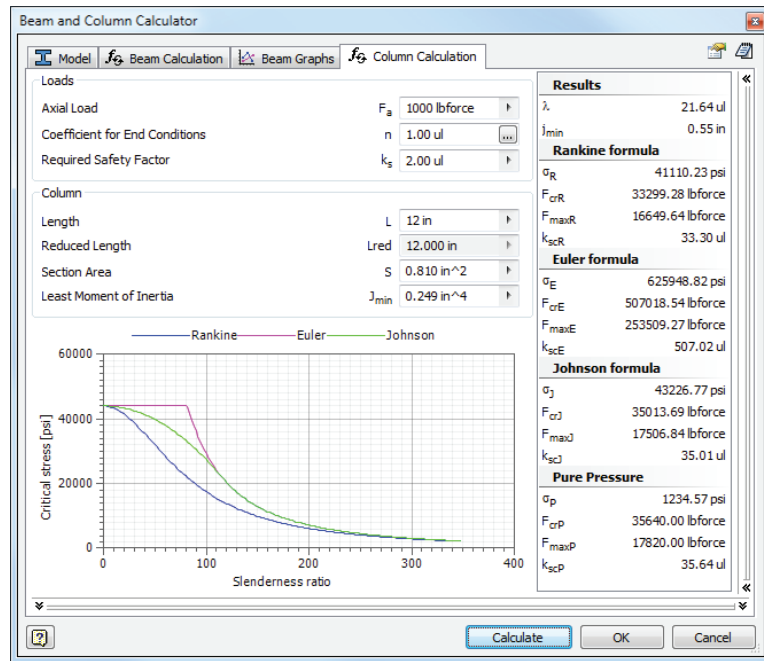
HTML RESULTS

When you click the Results button in the upper-right corner of the dialog box, an HTML page appears with all the data, calculation results, and graphs. You can save or print this file for your records.

FILE NAMING

The File Naming button, found next to the Results button in the upper-right corner of the dialog box, allows you to specify the calculation's filename. When you click the OK button to exit the Beam And Column Calculator dialog box, a subassembly file is created in the assembly file.

This subassembly file is just a container for the calculations, allowing you to access them again at any point. A browser node is created for the calculations. To edit the results, simply right-click the browser node and choose Edit Using Design Accelerator.

Publishing Frame Members

Frame Generator's frame member library is integrated with Content Center. The authoring and publishing process is similar to that used to create any custom Content Center files. Since Frame Generator requires specific modeling techniques, the authoring process will make some changes to the model and the parameters to ensure that all of the required parameters are included.

Authoring a Part

The authoring process for a frame member is similar to component authoring. The Structural Shape Authoring tool, located on the Part-Modeling panel bar, is used to prepare the part for publishing. The tool identifies the geometry used for placement, sets the parameters, and modifies the part so Frame Generator can use it.

This example uses a rubber bumper iPart file like might be attached to frames to guard against impact. In the iPart table there are parameters to control the dimensions and three sizes defined. If this were a true frame member profile, the engineering properties (moments of inertia and so on) might need to be calculated, and therefore that information would be set up as well. Since this isn't a load-bearing part, those properties aren't required. To explore the authoring process, follow these steps:

1. On the Get Started tab, click the Open button.

2. Browse for `mi_15a_020.ipt` located in the `Chapter 15` directory of your `Mastering Inventor 2015` folder and click the Open button.

3. From the Manage tab, locate the drop-down list on the Author panel and then click the Structural Shape button.

4. When the authoring tool starts, everything is blank. Once a category is selected, the dialog box will update with the appropriate controls. Because this is an unusual part, select the Other category, as shown in Figure 15.36.

   Frame Generator looks in the Structural Shapes category only, so you have to select one of the standard categories or create a new one in the Content Center Editor. You can't add a category through the authoring tool, so you have to add the category to Content Center before authoring. See Chapter 7, "Reusing Parts and Features," for more information on Content Center.

   Since there is only one extrusion in the part, the base feature is automatically selected. The default base point is indicated at the center of the profile. For this part, the inside corner of the flanges is the natural insertion point.
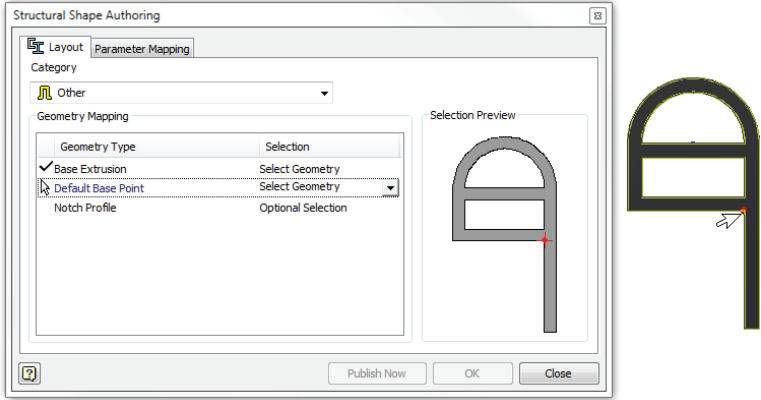
5. Click the drop-down list, choose Select Geometry, and then select that point in the model, as shown in Figure 15.37. Note that you may need to use the ViewCube® to look at the end view of the part to see the insert point selection.
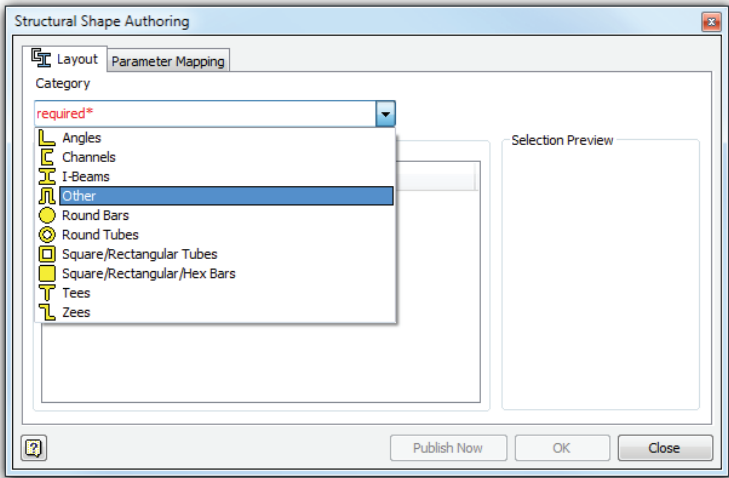
The Notch Profile option is used when you are authoring frame members and you want to include a predefined notch profile to be used with the Notch tool. In this case, there is no notch profile.

6. Click the Parameter Mapping tab.

7. Set the Base Length row to use the Length parameter by clicking the Please Select cell in the table and then choosing Length from the list.

The Parameter Mapping tab has one required field: Base Length. This is the parameter for the extrusion distance. Since this is an iPart, when you click in the field, the

iPart properties are listed, as shown in Figure 15.38. If this were a regular part, the Part Template Parameters dialog box would display a browser tree, as shown in Figure 15.39. The rest of the parameters are optional. They are mechanical properties of the profile necessary for calculating loads with the Beam And Column Calculator.

FIGURE 15.38
Specifying the length parameter for an iPart
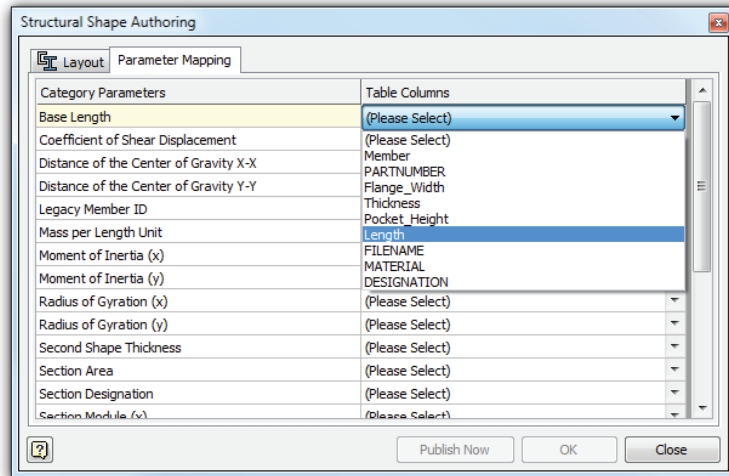
FIGURE 15.39
Specifying parameters for a regular part
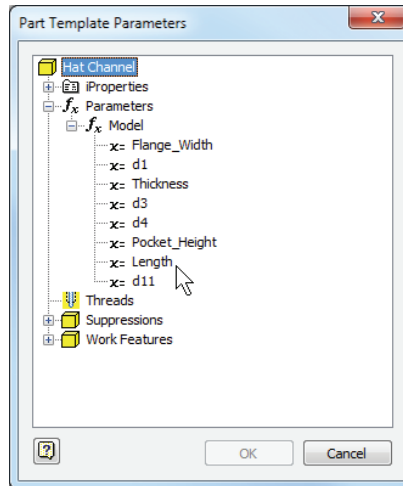
When the geometry and Base Length parameter are mapped, the Publish Now and OK buttons are enabled.

8. Click the OK button to update the part with the authoring changes.

Clicking either the Publish Now or OK button will update the part and close the dialog box. Publish Now will also launch Content Center publishing wizard. Once the 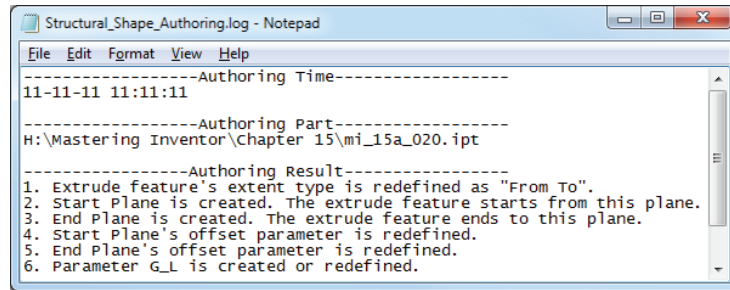part is updated, a dialog box displays with information about the changes. A log file that lists the changes to the part, as shown in Figure 15.40, is created in the project directory.

Figure 15.40
Log file for an authored part


```
Structural_Shape_Authoring.log - Notepad
File  Edit  Format  View  Help
------------------Authoring Time------------------
11-11-11 11:11:11

------------------Authoring Part------------------
H:\Mastering Inventor\Chapter 15\mi_15a_020.ipt

------------------Authoring Result----------------
1. Extrude feature's extent type is redefined as "From To".
2. Start Plane is created. The extrude feature starts from this plane.
3. End Plane is created. The extrude feature ends to this plane.
4. Start Plane's offset parameter is redefined.
5. End Plane's offset parameter is redefined.
6. Parameter G_L is created or redefined.
```

9. Click the OK button in the Structural Shape Authoring notice dialog box.

10. You can close the file without saving changes or use this file to explore the publishing options.

If you inspect the part after authoring, you will see that the browser and parameters have been updated. The details of the model and parameters are discussed in the section "Exploring the Anatomy of a Frame Member" earlier in this chapter.

Test Before You Publish

Before you publish the member for your team to use, be sure to test it in every conceivable situation. Even a seemingly well-constructed frame member can cause issues after end treatments are applied.

Publishing a Part

The publishing process uses the Publish Guide Wizard. Since the part was authored, most of the publishing information has already been added to the file. If you aren't familiar with publishing to Content Center, review Chapter 7.

These publishing steps are important for Frame Generator:

◆ When you define the family key columns, the Length parameter must be set as a key column.

◆ In the Family Properties pane, the standard organization is used to categorize the member during insertion. If you leave this field blank, the category selection will be Unknown.

◆ In the thumbnail image pane, a special thumbnail is displayed. Since the thumbnail is used as the orientation image in the dialog boxes, it is important to use the thumbnail that Frame Generator creates.

Frame Assemblies and BOMs

By default, Frame Generator parts are set to calculate their base quantity from the length of the part. This differs from standard parts that calculate their base quantity from the number of parts in the assembly. When parts are authored for use in Frame Generator, the Base Quantity setting is automatically set to the length. To get a part count in the parts list for your assemblies, you will want to configure them to read the item quantity rather than the base quantity. You can use both columns if that fits your needs, or you can have the BOM display one quantity and the parts list display the other.

You can also use the BOM expression builder to create a description built off the Stock Number property and GL parameter (Length). You can use the CUTDETAIL parameters in expressions to show the end treatment information for each member. Note that you can copy the expression in one cell of the BOM, multiselect the rest of the column, and then right-click and choose Paste to set the expression to all of the parts in the BOM. Figure 15.41 shows the BOM Editor listing the base quantity and the item quantity along with an expression being built to fill out the description.

FIGURE 15.41

Using the BOM expression builder
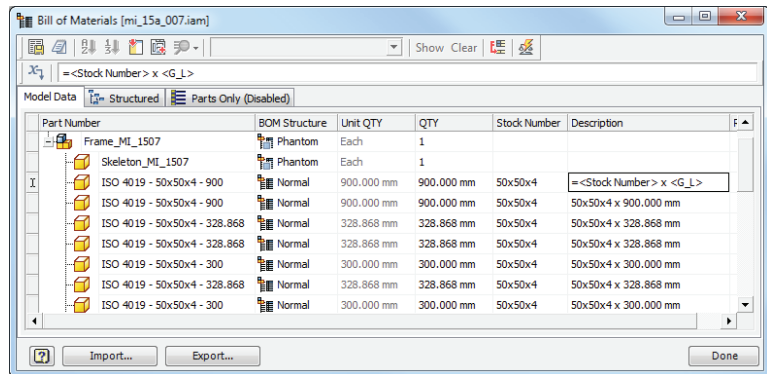
It is a good idea to let Frame Generator name the frame members for you and use the Part Number property rather than the part name to manage the parts. Once you've set an expression as just described, you can sort the BOM by that column to group the like items, set one of the part numbers, multiselect the items, and choose Paste from the context menu to set them all the same. If you want the items to be merged, leave the Part Number Merge Settings options selected. The Part Number Merge Settings button is in the top right of the BOM Editor dialog box.

In the drawing environment, use a Parts List style similar to the default Material List style found in the standard templates. This parts list is already configured to group by stock number and material, as well as sum the values of each unique stock number type. Figure 15.42 shows a parts list placed using the Material List style to roll up the lengths.

FIGURE 15.42

A parts list using the Material List style

MATERIAL LIST		
TOTAL QTY	STOCK NUMBER	MATERIAL
548.64 cm	3x3x1/4	Steel, Mild
2194.34 cm	2x2x1/4	Steel, Mild
121.92 cm	1-1/2x1-1/2x3/16	Steel, Mild

The Bottom Line

Work with frame files.   Frame Generator puts all the members at the same level in the assembly.

Master It   You have a frame that is built up in sections that are welded together. How do you document the manufacturing process?

Insert frame members onto a skeleton model.   Frame Generator builds a skeleton model for the frame from the selected lines and edges.

Master It   Since Frame Generator builds its own skeleton model, you don't have to build a master model before you start creating the frame. What would you reference in your assembly to use as a frame skeleton?

Add end treatments to frame members.   Frame Generator does not support end treatments on merged members.

Master It   Let's assume you are building a stairway and the handrail has curved sections. How would you approach the curved handrail so that its ends can be treated?

Make changes to frames.   Inventor provides detailed frame-member information.

Master It   You need to determine the size and wall thickness of the tubing and make it either thicker or larger. How do you do that?

Author and publish structural profiles.   Frame Generator uses structural shapes from Content Center.

Master It   How would you add custom aluminum extrusions to Content Center so Frame Generator can access them?

Create BOMs for Frame Generator assemblies.   Frame Generator has special parameters for frame members.

Master It   How do you add the profile dimensions and the length of your frame members to the Description field?