Skeleton Modeling Basics

by Neil Munro

Welcome back to our tutorials on working with Autodesk Inventor® software. In this installment, we examine a lesser-known assembly-modeling technique often referred to as *skeleton* or *master model modeling*.

No strict set of rules applies to skeleton modeling, and you can use many variations of the basic technique to suit your specific modeling requirements. The basic process for creating a skeleton-based assembly is as follows:

1. Create a single part model consisting of base sketches for all, or a subset, of the assembly components. Position the sketches to reflect the position of the component in the assembly.

2. Include construction surfaces, work features, and even solid geometry to be used as feature terminations or reference geometry during assembly component modeling. The sketches and other geometry form a skeleton of the final assembly.

3. Edit parameter names in the skeleton part, and export any key parameters that may be required in the individual assembly components.

4. Start a new assembly. Create each part (or subassembly) in-place and select the Assembly XY Plane as the location for the initial sketch for new parts. The common starting plane for each part is important if you are to retain the relationships between the new parts because all new parts use the skeleton part as their foundation.

5. Exit the initial sketch for the new part. Then use the Derived Component tool to import the skeleton part. You can filter out sketches, work features, and surfaces that are not required to create the base feature of the component.

6. Create the base feature of the component from the derived geometry. Add additional features as required.

7. Return to the top-level assembly and ground the component. Optionally you can place assembly constraints between the origin geometry of the component and the top-level assembly to lock the position of each component in the assembly.

8. Repeat the above steps for all components defined in the skeleton model. To make changes to the assembly, edit the skeleton part and then update the assembly to reflect the changes in all components affected by the skeleton part.

Skeleton Part Review

The part file included with this tutorial is the completed skeleton part. We'll examine this part prior to creating the assembly.

[Download](#) (zip - 127Kb)
You can download the files used in this tutorial from here.

> **Tip:** Keep the skeleton model open as you build the assembly. In a complicated skeleton model, you may well want to hide sketches and other geometry to reduce the filtering required when deriving the skeleton model into the assembly components.

1. Open SkeletonBase.ipt The part should match the one shown in Figure 1.

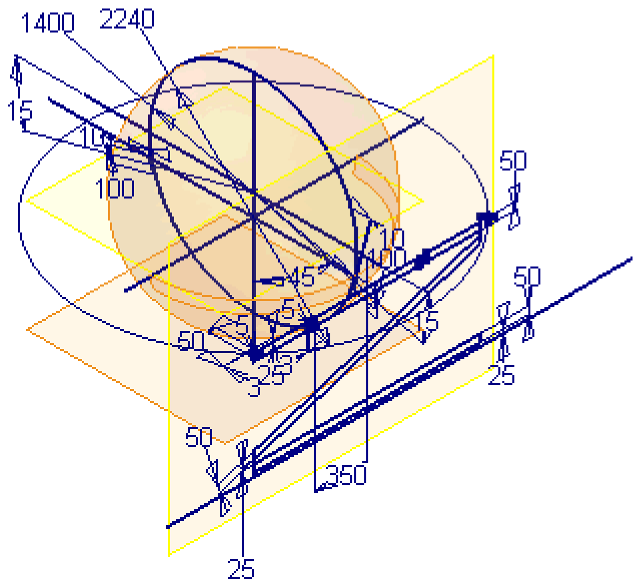


Figure 1: Skeleton part.

The part contains sketches, work features, and construction surfaces that define the basic geometry of a support frame for a spherical container. Skeleton models are particularly suitable for static models, such as frames and other fabricated assemblies. The jumble of geometry may look a bit confusing, so let's look at the individual sketches and other geometry in the part.

2. In the browser, drag the End of Part Marker and drop it just below the STRAP SURFACE node.

3. Click the Look At tool on the Standard toolbar, and then click CIRC STRAP SKETCH in the browser. Your view should match the one shown in Figure 2.

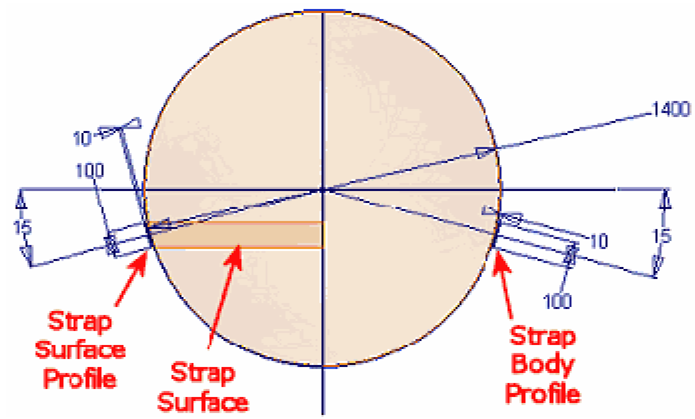


Figure 2: Strap sketch and surface.

The spherical construction surface is the key feature in the part. The diameter of the sphere drives the entire assembly, as you will see later in the tutorial. The shared strap sketch contains two related profiles:

A body profile that you will use to revolve a strap encircling the sphere.

A surface profile used to create the strap surface feature. This feature is used as a termination surface for supports between the strap and the external frame surrounding the sphere. Look for more explanation when the supports are created later in the tutorial.

Note: The two sketch profiles are linked by parameter values.

4. Drag the End of Part marker below the Work Plane-BOTT of Frame node. Examine the location of the three work planes.

The three work planes below STRAP SURFACE define the vertical extents of the exterior frame. The BOTT of Frame work plane is used as the termination face for the vertical legs of the frame. The sketch for the vertical leg is on the TOP of Frame work plane.

5. Drag the End of Part marker below the VERTICAL LEGS SKETCH node in the browser. Use the Look At tool to reorient your view to a plan view of this sketch as shown in Figure 3.

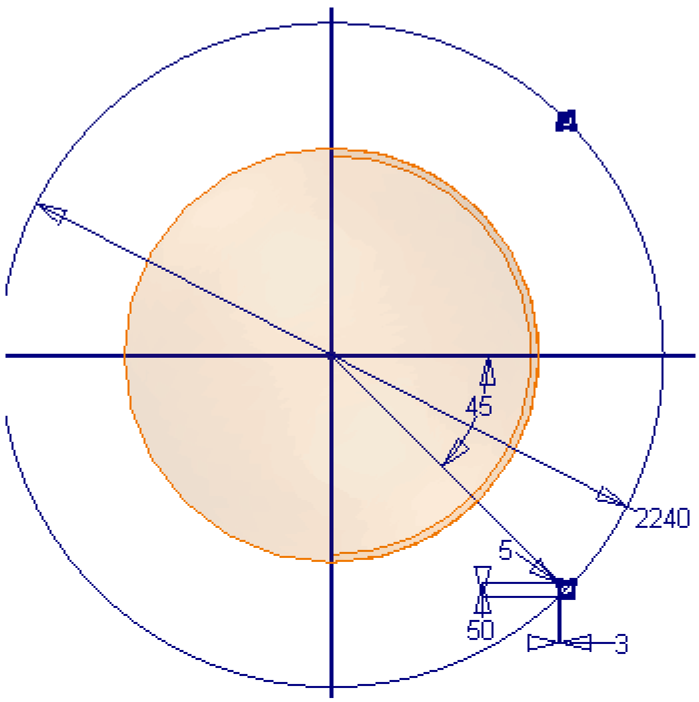


Figure 3: Vertical legs sketch.

The sketch contains a profile for the square tubing leg. The 2240-mm dimension is related to the diameter of the sphere. The size and thickness of the tubing is controlled with user-defined parameters. Figure 4 shows the size of the tubing controlled by the TubeSize user parameter. The user-defined parameters are exported, and thus can be derived into any of the parts based on the skeleton part.

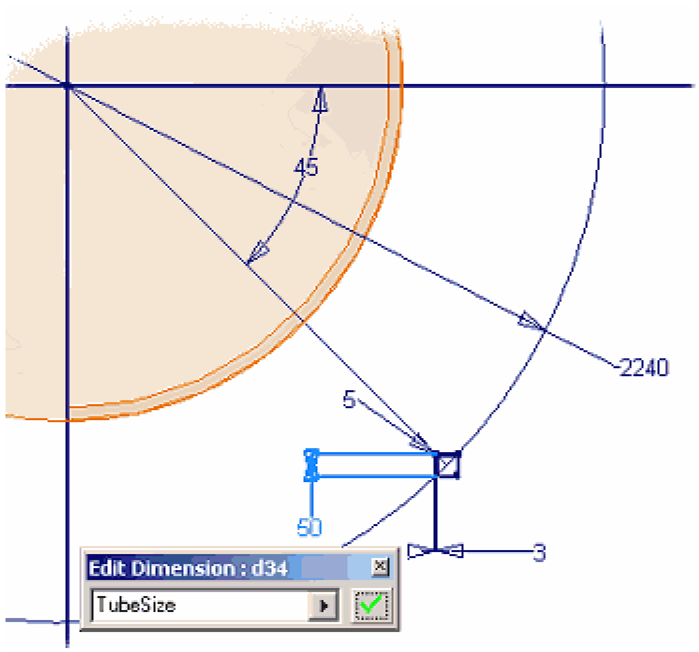


Figure 4: User-defined parameters control sketch size.

6. Drag the End of Part marker to the bottom of the browser. Examine the remaining sketches. Note that all sketches are related through projected geometry, work feature definition, or parameters. The sphere diameter drives all the critical dimensions of the frame.

Create a New Assembly

Now let's build a new assembly based on the skeleton part.

1. Start a new assembly based on the Standard(mm).iam template.

2. From the Assembly panel bar, click the Create Component tool. In the Create In-Place Component dialog box:

- Enter *Strap* as the name of the new part.
- Clear the check box labeled Constrain sketch plane to selected face or plane.
- If your standard part template is inch based, click the Browse tool to the right of the Template list. Click the Metric tab of the Open Template dialog box, and then select the Standard(mm).ipt template. Click OK to return to the Create In-Place Component dialog box.
- Click OK to create the Strap part in the assembly.

The first part in an assembly is automatically grounded and aligned to the assembly origin geometry.

3. Click the Return tool on the Standard toolbar to exit from the sketch.

4. Delete Sketch1 in the strap part. This step is not required, but makes for a tidier browser since the sketch will not be used.

5. From the Features panel bar, click the Derived Component tool. Browse to the folder containing the SkeletonBase.ipt file and open it. The Derived Part dialog box opens (see Figure 5).

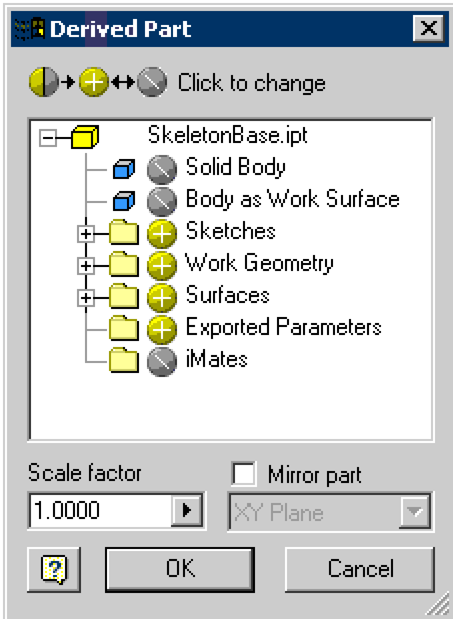


Figure 5: Derived Part dialog box.

6. The strap part requires only the CIRC STRAP SKETCH. Expand Sketches in the Derived Part dialog box. Click the gold icons with the plus signs (+) that are next to the three sketches other than CIRC STRAP SKETCH. Each icon changes to a gray slash symbol, indicating the sketch will not be derived into the new part (see Figure 6).

Figure 6: Eliminate sketches from derived part.

7. Click the icon next to Work Geometry to eliminate all visible work geometry from the derived part.

Note: Sketches, surfaces, and work geometry must be visible in the originating part to be derived into the new part. You can turn off the visibility of items prior to using the Derived Component tool, thus reducing the number of items you have to clear in the dialog box.

8. Click the icons next to Surfaces and Exported Parameters to eliminate them from the derived part.

9. Click OK. The part should match the one shown in Figure 7.

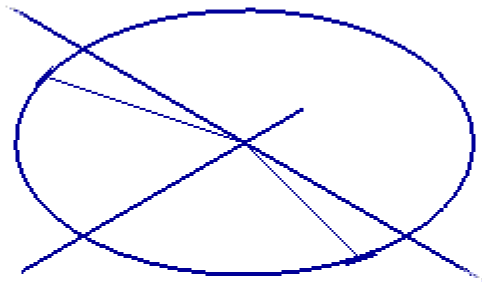


Figure 7: Derived sketch for strap part.

10. From the Features panel bar, click the Revolve tool. Select the closed profile of the strap as shown in Figure 8.

11. Click the Axis tool in the Revolve dialog box, and then select the line highlighted in Figure 8.

12. Click OK to create the strap part.

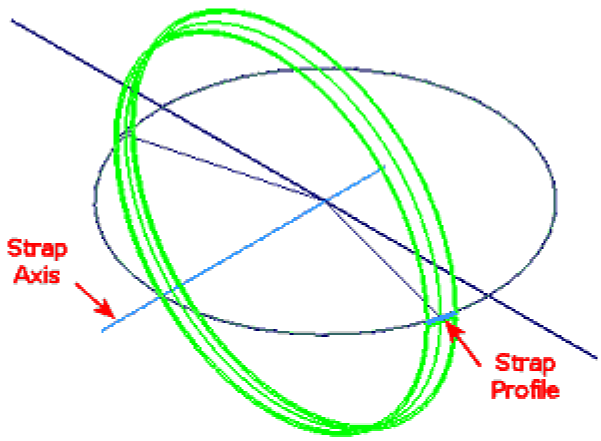


Figure 8: Strap revolved feature.

Tip: You can add features to the part as you would with any part created from scratch. The derived sketch controls the size and position of the base feature in this example. You can use the Derived Component tool at any point in the modeling process, not just as the first feature in a part. For example, a second feature derived from the component might bring in a second sketch, which is then used to add or subtract geometry from the first feature based on the derived component.

13. Click the Return tool on the Standard toolbar to return to the assembly level.

Create a Frame Leg

Next, you create a leg from the skeleton part and then ground it in the assembly.

1. From the Assembly panel bar, click the Create Component tool.

2. Enter *Leg* as the part name and base the part on the Standard(mm).ipt template.

3. Click OK in the Create In-Place Component dialog box.

4. Expand the Origin folder under Assembly1 in the browser. Click the XY Plane node. This aligns the part origin to the assembly origin.

Note: The preceding step is critical to aligning all parts based on the skeleton model. A user-written macro exists that enables you to create derived parts outside the assembly environment and then drag and drop multiple components into an assembly. The macro automatically aligns the components to the assembly origin and grounds them.

5. Exit the sketch in the new part, and then delete Sketch1 to tidy up the browser.

6. Use the Derived Component tool to derive the SkeletonBase.ipt part into the leg part.

7. Eliminate all sketches other than VERTICAL LEGS SKETCH.

8. Expand Work Geometry in the Derived Part dialog box, and eliminate all work features other than Work Plane-BOTT of Frame.

9. Eliminate Surfaces and Exported Parameters from the derived component. The Derived Part dialog box should match the one shown in Figure 9.

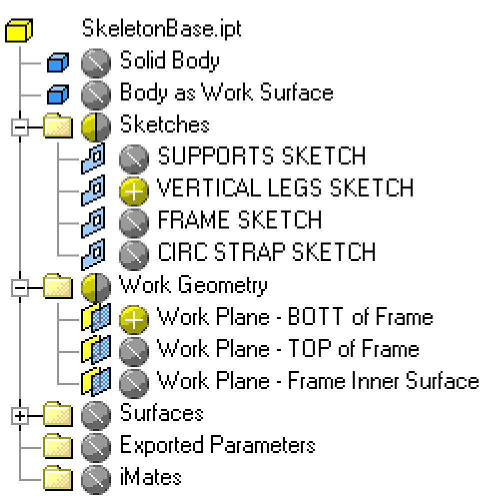


Figure 9: Derived geometry for frame leg.

10. From the Features panel bar, click the Extrude tool. Extrude the tube profile highlighted in Figure 10.

11. From the Extents list in the Extrude dialog box, select To, and then click the derived work plane in the graphics window.

12. Click OK to complete the feature.

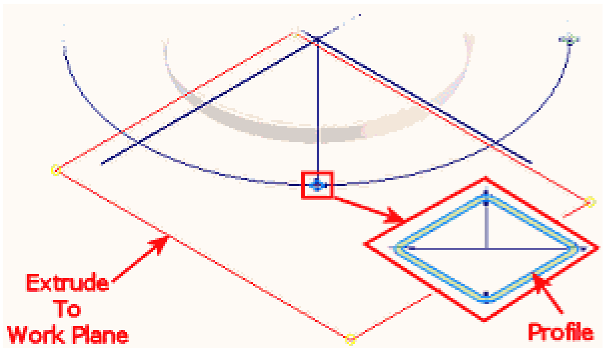


Figure 10: Profile and extent for leg extrude.

13. Expand SkeletonBase.ipt in the browser. Right-click Work Plane1 and click Visibility in the context menu to hide the work feature in the graphics window.

14. Finally, return to the assembly environment. Right-click Leg.ipt in the browser and select Grounded from the context menu that appears.

Create a Frame Subassembly

You now create the three tubes that form one side of the frame. Two of the tubes are identical, so creating them as separate parts might not be the best solution for a parts list or BOM.

Creating a subassembly enables us to look at a slight variation of the workflow described above:

First, you create a subassembly in-place in the assembly. In the subassembly you first create a frame layout from a derived sketch. This sketch is not used to create solid geometry.

Next, you create the two different tube parts in the subassembly, using the same skeleton technique as above.

Finally, you place a second instance of one frame tube, and assemble it using assembly constraints. The second instance of the tube is constrained to the derived sketch in the layout part.

Note: We will incorporate the vertical leg in the subassembly later in the tutorial.

1. From the Assembly panel bar, click the Create Component tool. In the Create In-Place Component dialog box:

- Enter *Frame* as the assembly name.
- Select Assembly from the File type.
- Base the assembly on the Standard(mm).iam template.
- Click OK.

2. Click the Assembly XY Plane in the browser to align the subassembly to the top-level assembly. The Frame subassembly is the active component in the assembly.

3. From the Assembly panel bar, click the Create Component tool. In the Create In-Place Component dialog box:

- Enter *FrameLayout* as the part name.
- Base the part on the Standard(mm).ipt template.
- Click OK.

Since this is the first part in the subassembly, it is grounded and aligned to the subassembly origin. Since the subassembly is aligned to the top-level assembly, any geometry derived from the skeleton model will match the other components.

4. Derive SkeletonBase.ipt into the new part. Eliminate all geometry other than the FRAME SKETCH sketch. Your part should match the one shown in Figure 11.

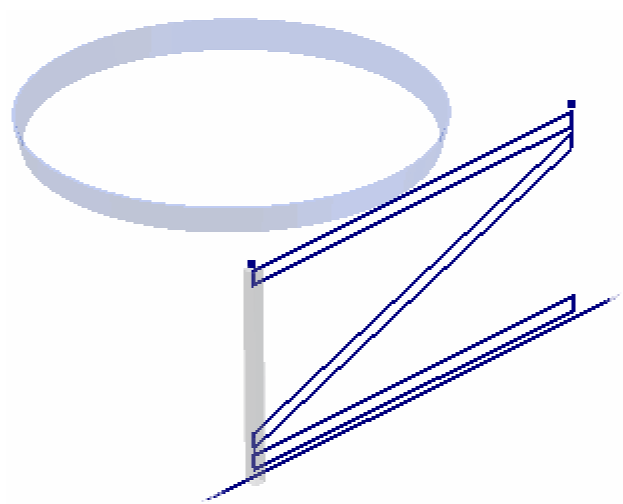


Figure 11: Frame layout part.

5. Click the Return tool on the Standard toolbar to return to the subassembly environment. Ground the FrameLayout.ipt part.

6. From the Assembly panel bar, click the Create Component tool. In the Create In-Place Component dialog box:

- Enter *HorTube* as the part name.
- Base the part on the Standard(mm).ipt template.

- Click OK.

7. Expand the Origin folder under Frame.iam in the browser. Click the XY Plane node. This aligns the part origin to the subassembly origin.

8. Exit the sketch in the new part, and then delete Sketch1 to tidy up the browser.

9. Derive SkeletonBase.ipt into the new part by:

- Eliminating all sketches other than the FRAME SKETCH sketch.
- Eliminating Work Features and Surfaces from the derived part.
- Not eliminating the Exported Parameters from the derived part. You will use one of the parameters from the skeleton part to control the extrusion distance of the base feature in the new part.

10. From the Features panel bar, click the Extrude tool. Select the profile highlighted in Figure 12.

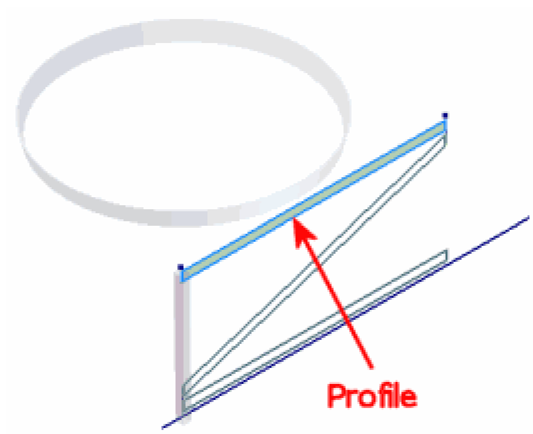


Figure 12: Extrusion profile.

11. In the Extrude dialog box, click the arrow next to the edit box containing the 10-mm default extrusion distance. Select List Parameters from the pop-up menu. Click TubeSize in the Parameters list.

12. Click OK in the Extrude dialog box to create the base feature.

13. Add a 5-mm fillet to the four long edges of the new part.

14. Shell the part, removing the two end faces to create a hollow structural tube.

15. Click the arrow next to the Thickness edit box in the Shell dialog box and select List Parameters from the context menu that appears.

16. Click TubeThickness in the Parameters list.

17. Click OK.

Your part should match the one shown in Figure 13.

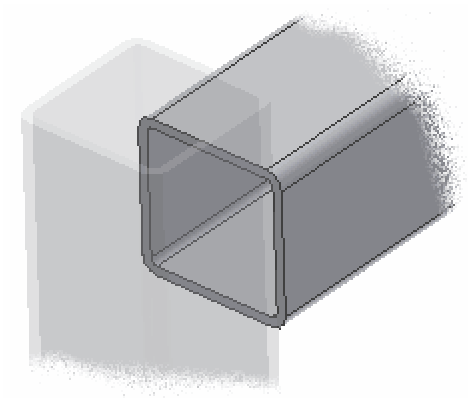


Figure 13: Horizontal tube part.

Note: With some additional work geometry in the skeleton model, you could create the end profile for the horizontal tube, similar to the vertical leg tube. This eliminates the need for additional features to be added in the part, but the additional complexity of the skeleton model may outweigh that advantage.

Create a Diagonal Tube

Using the same steps used to create the horizontal tube, create a diagonal tube in the frame subassembly.

1. From the Assembly panel bar, click the Create Component tool. In the Create In-Place Component dialog box:

- Enter *DiagTube* as the part name.
- Base the part on the Standard(mm).ipt template.
- Click OK.

2. Expand the Origin folder under Frame.iam. Click the Assembly XY Plane in the browser to align the part to the subassembly origin.

3. Exit the initial sketch and then delete Sketch1 to tidy up the browser.

4. Derive the FRAME SKETCH sketch and Exported Parameters from SkeletonBase.ipt into the new part.

5. Extrude the profile highlighted in Figure 14 by first linking the extrusion distance to the derived TubeSize parameter.

6. Add the 5-mm radii to the long edges of the part.

7. Shell the part and link the shell thickness to the derived TubeThickness parameter.

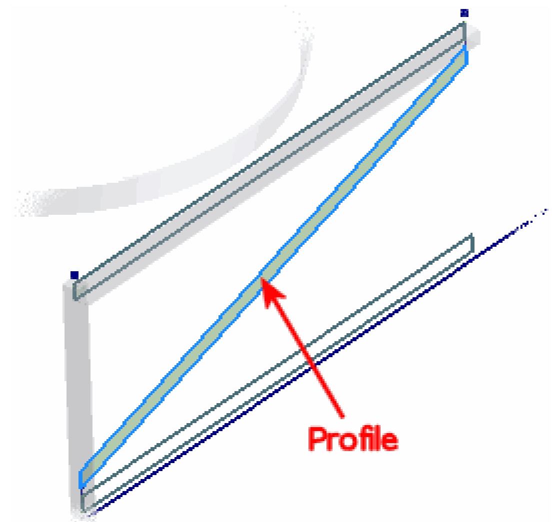


Figure 14: Diagonal tube profile.

8. Return to the Frame subassembly level and ground the DiagTube.ipt part.

Add a Second Horizontal Tube

To complete the frame subassembly, you add a second occurrence of the HorTube part. You then constrain this occurrence to the other frame members and the frame layout sketch.

1. With the Frame subassembly active, drag HorTube.ipt:1 from the browser and drop it in the graphics window.

2. From the Assembly panel bar, click the Place Constraint tool. Click the Flush solution in the Place Constraint dialog box.

3. Add a Flush constraint between the two faces highlighted in Figure 15.

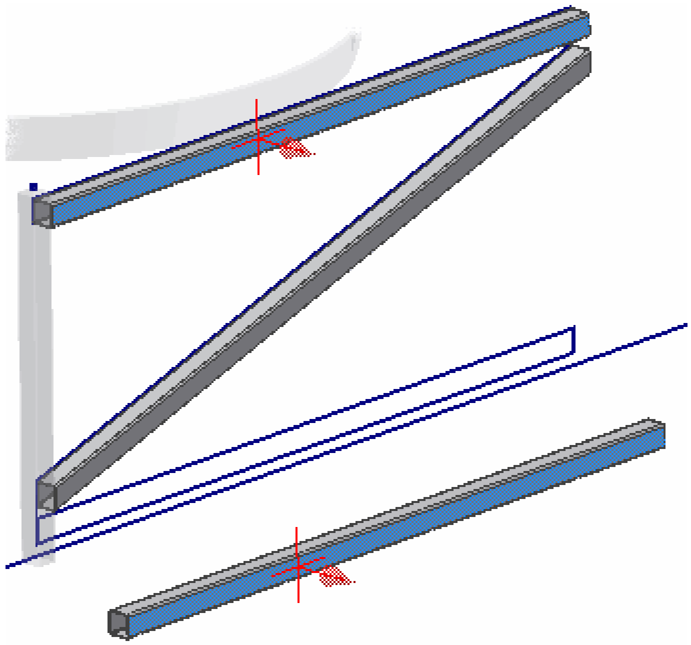


Figure 15: Face/Face flush constraint.

4. Add a second Flush constraint between the end faces of the two tubes as highlighted in Figure 16.

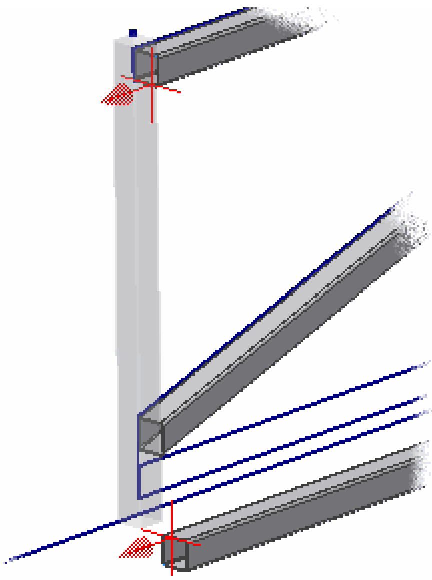


Figure 16: Second Face/Face flush constraint.

5. Click the Mate solution in the Place Constraint dialog box.

6. Click the Face highlighted in Figure 17 and then click the visible sketch edge, which is also highlighted. The edge is geometry in the derived sketch in the FrameLayout part.

7. Apply the constraint and then click Cancel to close the Place Constraint dialog box.

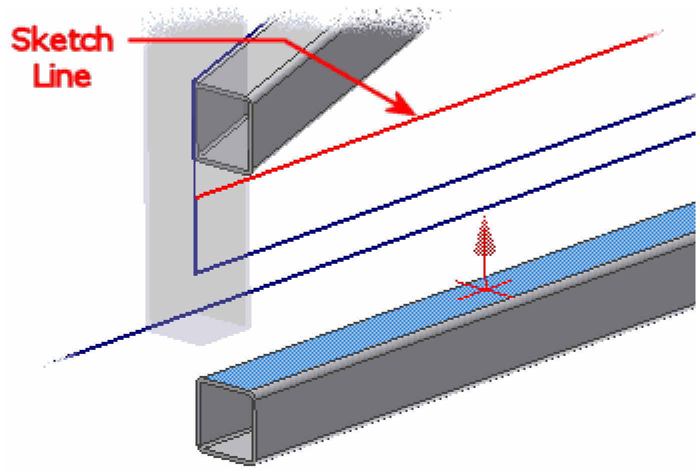


Figure 17: Face/Line mate constraint.

8. Turn off the visibility of the FrameLayout part in the subassembly.

Derived Surface from Skeleton

Finally, you create a left-hand and right-hand support to connect the frame to the strap. The supports use a derived construction surface as the termination surface for an extrusion.

1. With the Frame subassembly active, create a new component named SupportLeft in-place in the subassembly. Select the subassembly XY Plane as the initial sketch plane. Exit the initial sketch and delete Sketch1 to tidy up the browser.

2. Derive the SkeletonBase part into the new part by:

- Eliminating all sketches other than SUPPORTS SKETCH from the derived part.
- Eliminating Work Features and Exported Parameters from the derived part.
- Expanding Surfaces in the Derived Part dialog box.
- Eliminating all surfaces other than STRAP SURFACE from the derived part.

3. Click OK to complete the feature. Your assembly should match the one shown in Figure 18.

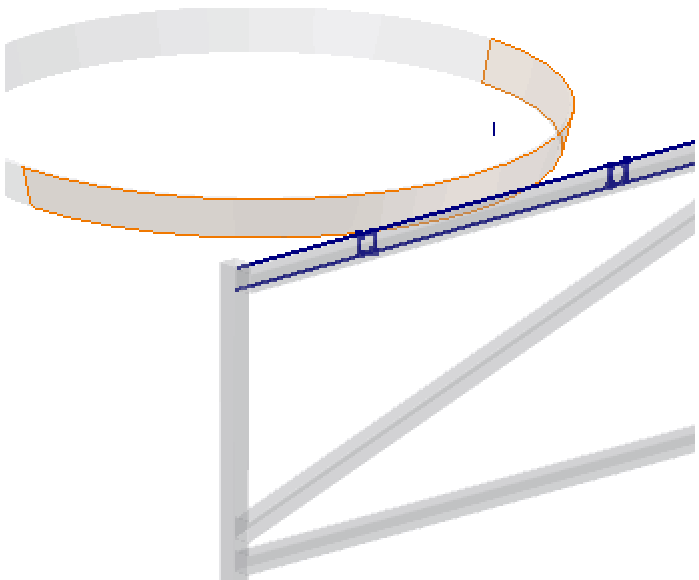
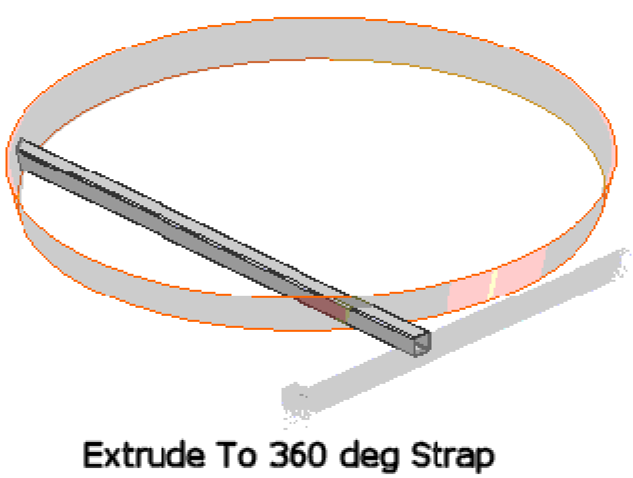


Figure 18: Derived sketch and surface for support.

You might be asking why the separate strap surface is even required. Why not create a solid body of the strap in the skeleton, and then derive the solid body as a solid or surface body into the support part?

Notice that the strap surface feature is a single 180-degree revolved surface that matches the outer surface of the strap. The support tube profile will be extruded to this surface. If the termination surface provides more than one solution, the maximum termination will always be the result. See Figure 19 for an example.

In Autodesk Inventor 6, you can choose a minimum or maximum solution for an extrusion termination.



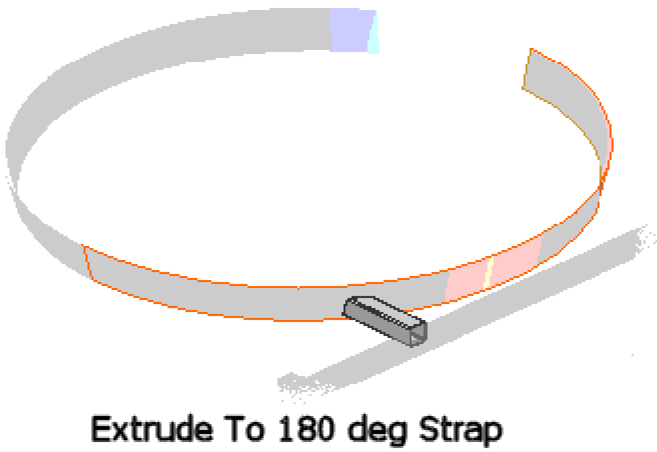


Figure 19: Extrude To termination examples.

4. From the Features panel bar, click the Extrude tool. Select the profile highlighted in Figure 20.

5. From the Extents list in the Extrude dialog box, select To.

6. Click the derived surface highlighted in Figure 20.

7. Click OK to complete the feature.

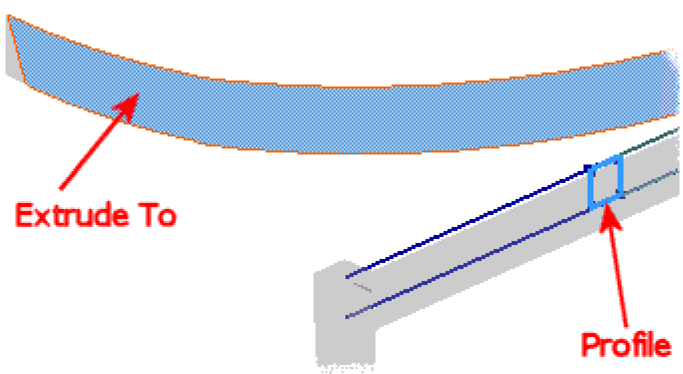


Figure 20: Extrusion profile and termination surface.

8. Return to the Frame subassembly level, and then ground the SupportLeft part.

9. Repeat the above step to create a right-hand support named SupportRight. Derive the same sketch and surface into the new part, and then extrude the other tube profile in the sketch to the termination surface. The Frame subassembly should match the one shown in Figure 21.

Note: Expand the derived SkeletonBase feature under the SupportLeft and

SupportRight parts, and turn off the visibility of Derived Suface1 to tidy up the browser.

Figure 21: Frame subassembly.

Complete the Frame Subassembly

The vertical leg belongs in the Frame subassembly. Because there are no assembly constraints, you can freely drag components between subassembly levels without fear of breaking the assembly structure.

1. Drag Leg.ipt:1 in the browser, and drop it below FrameLayout.ipt:1, as shown in Figure 22.

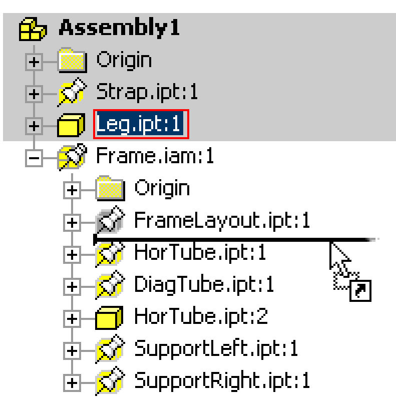


Figure 22: Drag-and-drop assembly restructure.

2. The leg part is ungrounded after its demotion into the subassembly. Right-click Leg.ipt and select Grounded from the context menu that appears.

3. If the Frame subassembly is not the active component, right-click Frame.iam in the browser, and select Edit from the context menu that appears.

4. From the Assembly panel bar, click the Pattern Component tool. In the browser or graphics window, select all parts in the subassembly except FrameLayout.ipt.

5. In the Pattern Component dialog box:

- Click the Circular tab.
- Click the Axis Direction tool under Circular Placement.
- Expand the Origin folder under Frame.iam and click Y Axis.
- Enter *4* in the Count edit box.
- Enter *90* in the Angle edit box.
- Click OK.

6. Return to the top-level assembly. Ground Frame.iam if you have not already done so. The assembly should match the one shown in Figure 23.

Note: Component colors have been changed for clarity in Figure 23.

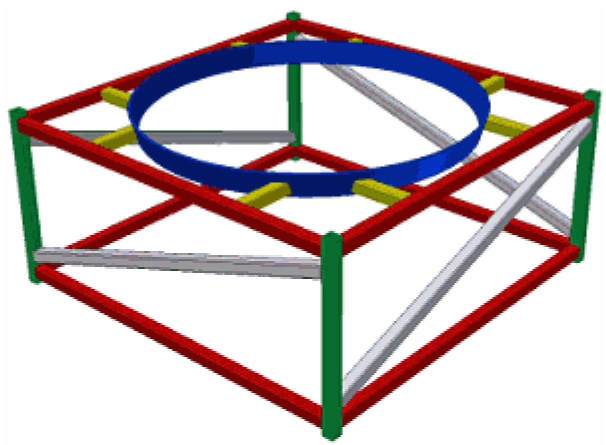


Figure 23: Completed assembly.

Assembly Update

All changes to the assembly components are controlled by the skeleton part. So let's change the sphere diameter in the skeleton part and examine the changes in the assembly.

1. Open SkeletonBase.ipt or activate its window if the file is already open.

2. Click the Parameters tool on the Standard toolbar. Scroll down to the User Parameters area in the Parameters dialog box.

3. Enter *900* in the Equation cell of the SphereDiameter user parameter.

4. Click Done.

5. Save the file.

6. Return to the assembly file.

7. Click the Update tool on the Command bar. The assembly changes to reflect the smaller sphere diameter. Your assembly should match the one shown in Figure 24.

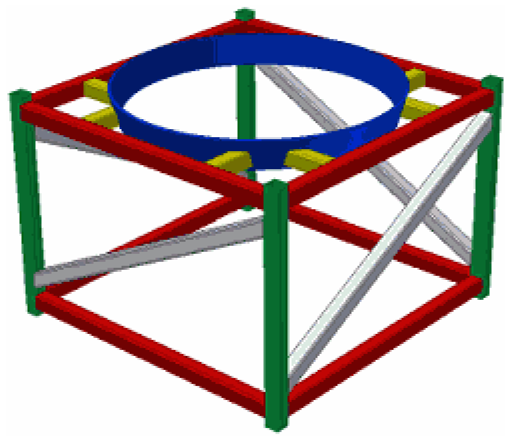


Figure 24: Updated assembly.

Summary

Skeleton modeling is an efficient and versatile technique for building assemblies with Autodesk Inventor software. The application of this technique is limited only by your imagination. The lack of assembly constraints and absence of adaptive relationships can improve the performance and robustness of assembly updates. Setting up a skeleton or master model takes some planning, but the ability to easily control an assembly from a single source can be worth the effort.

Tune in next month when we will introduce the new features and enhancements coming soon in Autodesk Inventor 6.