Autodesk **Skill Builders**

A Skill-building Exercise

Assembly Constraints - Conical Faces

In This Exercise

This Skill Builder demonstrates how to constrain parts that contain conical faces.

Constraining Cones

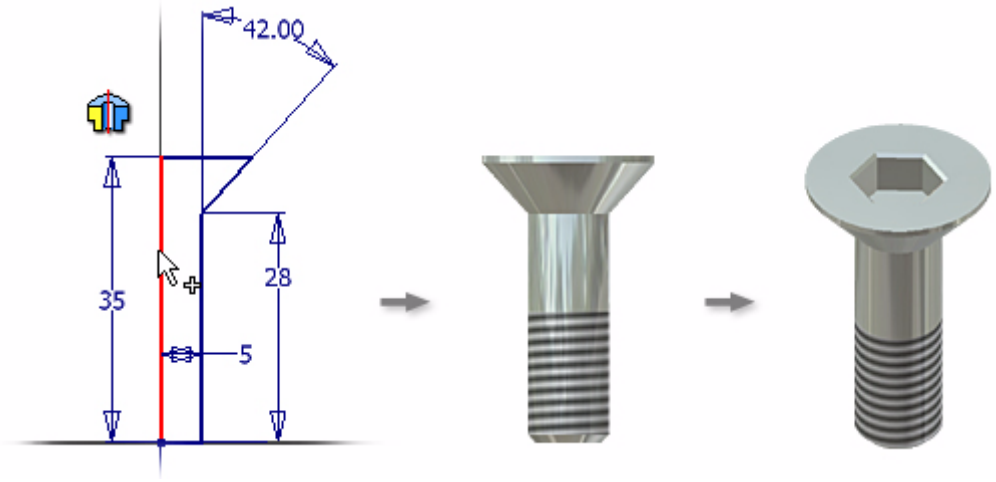
There are numerous applications for parts with conical faces. A countersunk machine screw is probably the most common occurence. Though it may not seem immediately obvious, constraining conical parts is simple and straightforward.

How-to

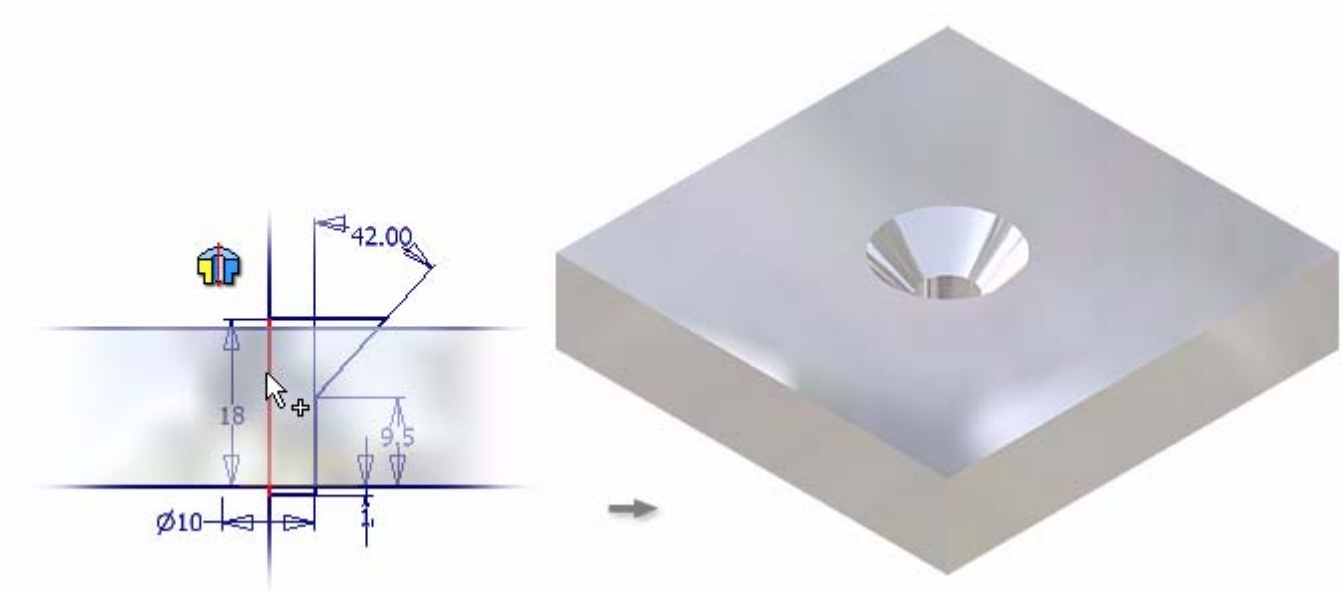
There are essentially two methods you can use to quickly and securely constrain a countersunk machine screw.

The first and simplest is to use an Insert constraint, just as you would for a regular capscrew or pin.

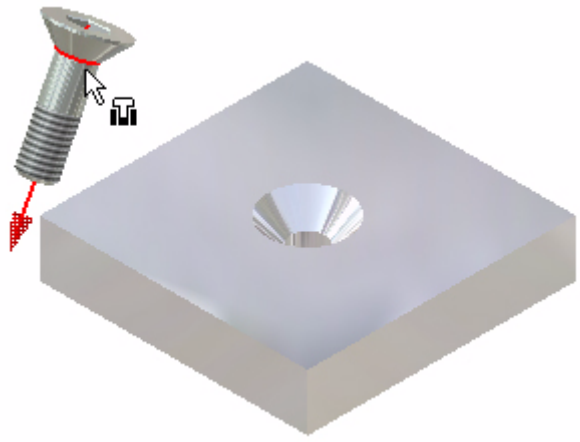
Open or create an assembly that contains a countersink-type machine screw and a part with a mating hole. You may find it convenient to select a countersunk machine screw from the content library and then create a part with a matching countersunk hole. You can also create the screw from scratch by revolving a profile similar to the following illustration.



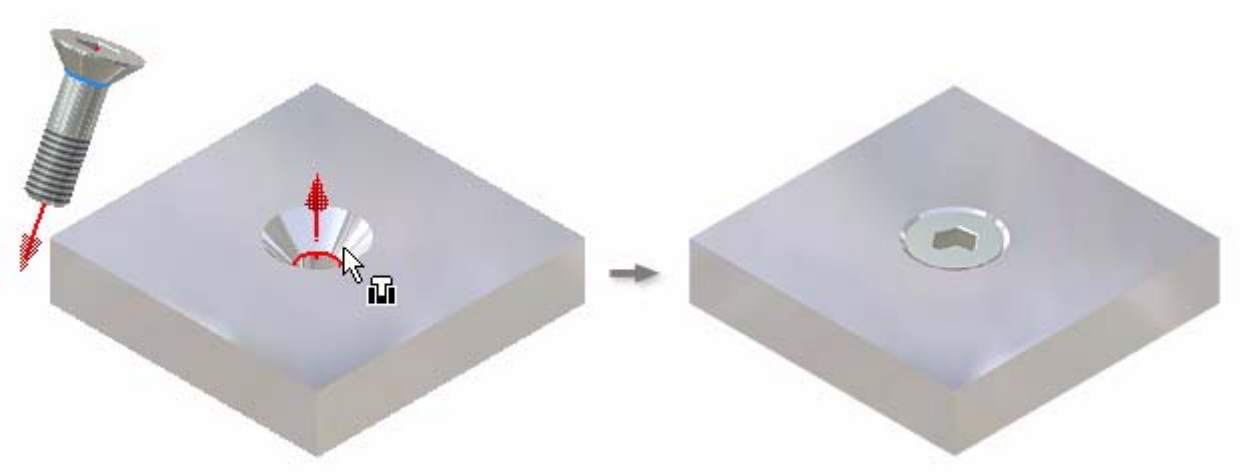
While it certainly isn't necessary to model the screw in detail, it is important that the critical profile dimensions of the screw (primarily the screw diameter and seat angle) match those of the hole.



Once you have the added the screw and the part with the mating hole to an assembly document, press **C** on the keyboard to activate the Place Constraint dialog box. Select the Insert button in the Type field. Pause the cursor over the screw shank near the lower edge of the screw head. When the edge highlights and the insert direction indicator points away from the screw head, select the screw.



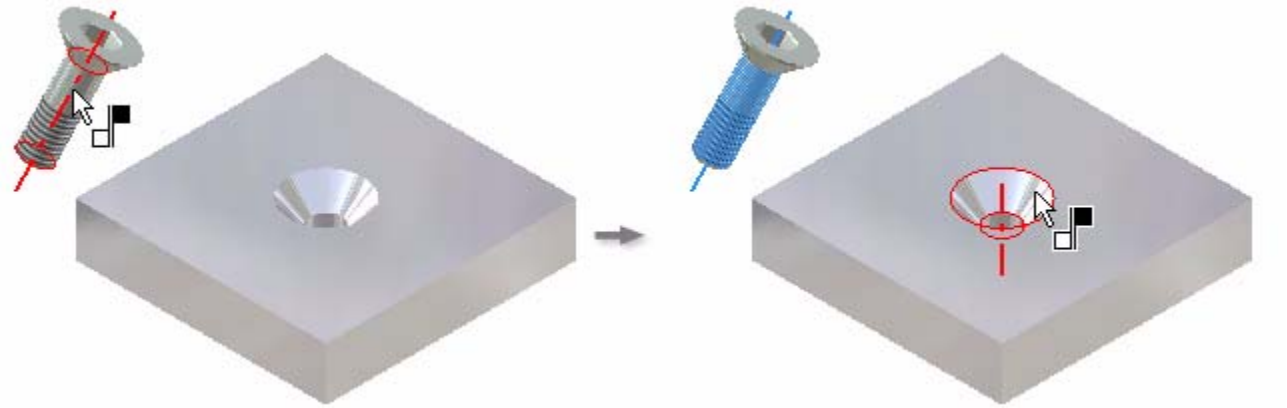
Pause the cursor over the lower edge of the conical face of the mating hole. When the lower edge highlights and the insert direction indicator points up and out of the hole, select the face. Click OK in the Place Constraint dialog box.



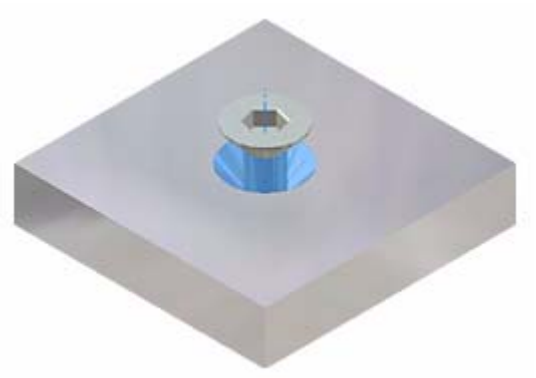
With just one constraint you have removed two degrees of freedom from the screw. For all intents and purposes it is fully constrained.

The second method uses mate and tangent constraints to achieve essentially the same results.

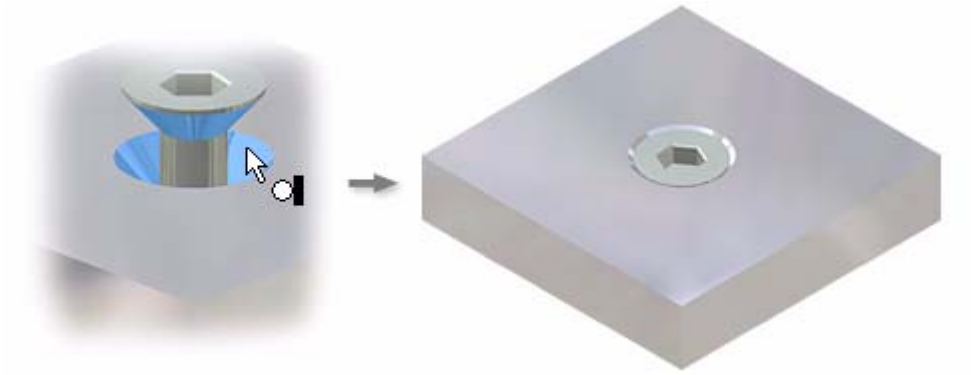
Delete the Insert constraint and move the screw away from the hole. Once again, activate the Place Constraint dialog box. Select the screw shank. Alternatively, you could select the conical face of the screw head. Either selection will create the needed axial mate. Select the hole bore or the hole conical face.



Click Apply.



Select the Tangent button in the Type field. Select the Inside button in the Solutions field. Select the conical face of the screw. Select the conical face of the hole. Click OK.

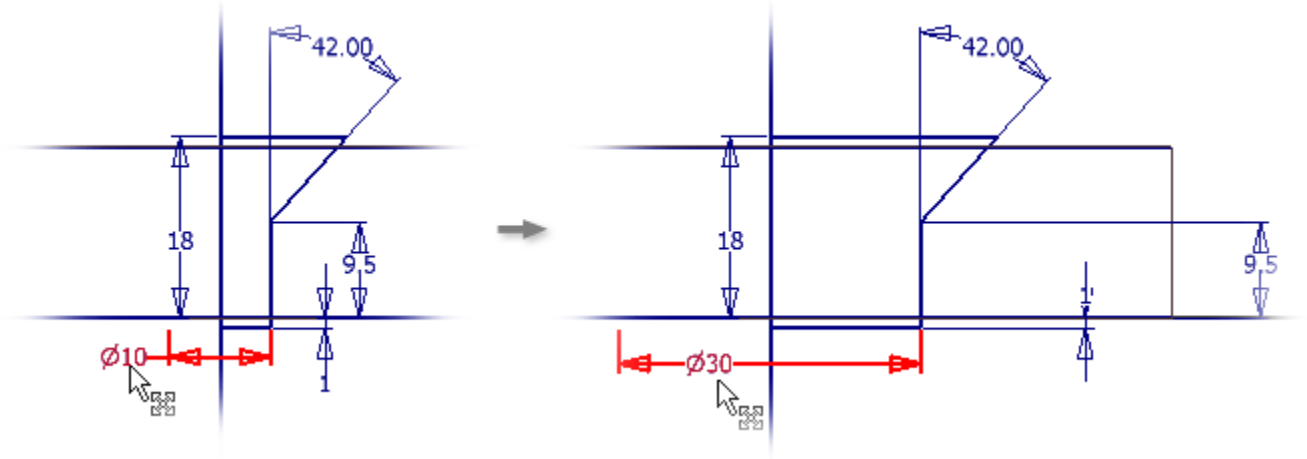


The axes are mated and the conical faces are tangentially constrained.

Some applications of parts with conical faces may require that the axes of the relevant parts are parallel but not colinear, and that the conical faces are tangent but offset. For example, in a design variation of a conical-type rock crusher, the spindle does not share an axis with the crusher bell, but must rotate within the confines of the bell. In addition, the spindle face also must be offset from the bell face by a variable distance, derived from the desired size of the crushed rock.

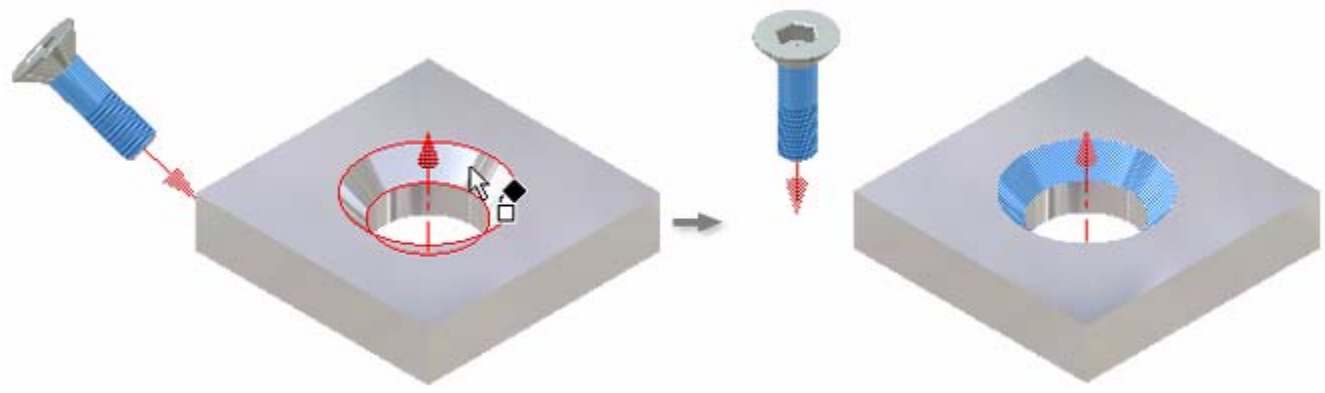
You can recreate the following example using the parts you used for the previous exercises. Imagine that the screw is the spindle of the crusher and that the hole is the bell. For better visibility, the following illustrations show the parts upside down, relative to the actual configuration of the crusher in the field.

Delete all previous constraints and move the screw away from the hole. Edit the sketch of the hole and triple the diameter of the hole.

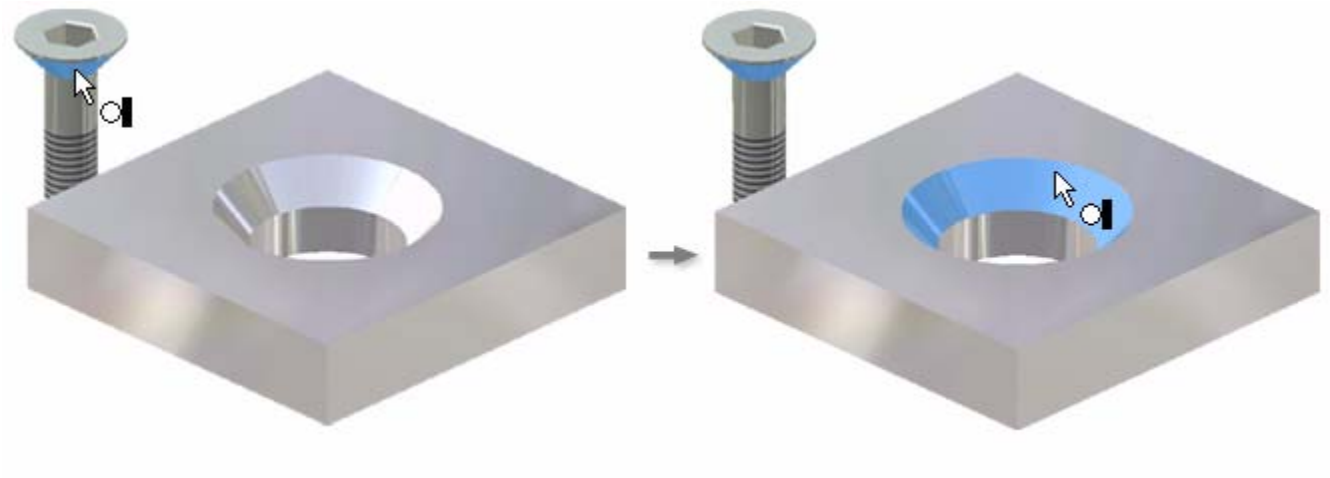


Return to the Assembly environment.

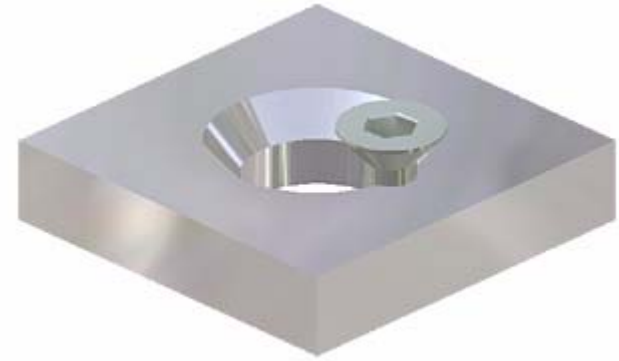
Activate the Place Constraint dialog box. Select the Angle button in the Type field. Select the shank of the spindle. Select the bell. Click Apply. The axes angles are constrained at 0 degrees.



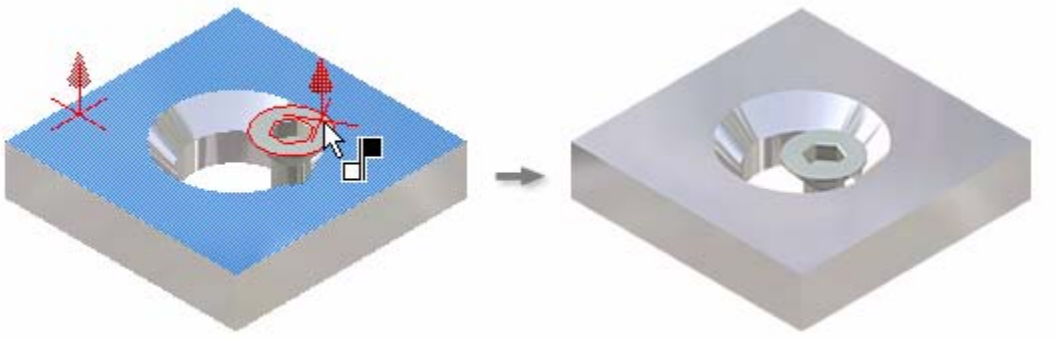
Select the Tangent button in the Type field. Select the Inside button in the Solutions field. Select the conical face of the spindle. Select the conical face of the bell. Specify a suitable distance in the Offset field.



Click Apply.



Select the Mate button in the Type field. Select the Flush button in the Solutions field. Select the bottom of the spindle. Select the flat face adjacent to the bell. Click OK.



The spindle is now tangentially constrained to the bell and free to track around the conical face of the bell.