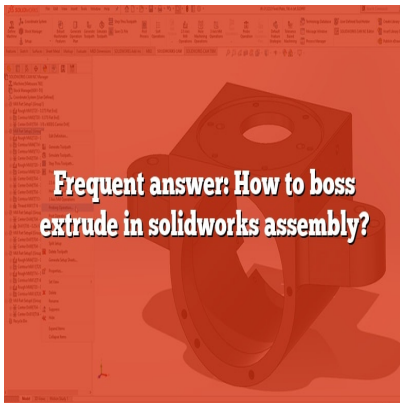




Frequent answer: How to boss extrude in solidworks assembly?

Description

With this article you will have the answer to your Frequent answer: How to boss extrude in solidworks assembly? question. Indeed Solidworks is even easier if you have access to the free Solidworks tutorials and the answers to questions like. Our CAD-Elearning.com site contains all the articles that will help you progress in the study of this wonderful software. Browse through our site and you will find different articles answering your different questions.

The use of parametric design in Solidworks makes it a powerful tool for designers and engineers. The designer can observe the impact of adjustments on neighboring components or even on the entire solution. This makes it quick and easy for designers to identify and solve problems.

And here is the answer to your Frequent answer: How to boss extrude in solidworks assembly? question, read on.

Introduction

Open a sketch on a face or plane, and sketch a profile of the cut. The profile can contain more than one closed contour. Click Extruded Cut or Revolved Cut (Features toolbar), or click Insert > **Assembly** Feature > Cut , then **Extrude** or Revolve. Set the options as needed in the Extrude or Revolve PropertyManager.

Additionally, how do you create an extruded boss feature? To create an extruded **boss**: 1) Sketch a profile for the extruded boss. 2) Select Extruded Boss/Base on the Features toolbar or select Insert, Base, Extrude on the menu bar. Use the spin buttons beside the depth text box to change the value, or type in the value in the text box.

Amazingly, how do you extrude in SolidWorks?

1. Click Insert > Surface > **Extrude**.
2. Select a face:

3. Select other faces to define the extrude as required.
4. Select the end condition.
5. For 3D faces, select a plane, edge, 2D face, or sketch line to define the direction of extrusion .

You asked, how do I change my **boss** extrude? If you enter the wrong distance to extrude to: and exit the **boss**-extrude dialog without correcting it, go to the design tree, right-click on the **extrude** name, and find the "edit feature" button in the box of options that appears above the mouse. The dialog will reopen and you can edit the properties of the extrude.

In this regard, how do you boss **extrude** at an angle in **SolidWorks**? Use the "Sketch Plane" as the start condition. Set the end condition to "Up to Body", select the front face of extrude1 as the reference. Turn on draft and set a 5° draft angle, and tick the draft outward option. The final result is shown below. To create a revolve feature: Create a sketch that contains one or more profiles and a centerline, line, or edge to use as the axis around which the feature revolves. Click one of the following revolve tools: Revolved Boss/Base (Features toolbar) or Insert > Boss/Base > Revolve.

What is the necessary condition for extrude boss command?

The default end condition is "blind", which means we will take our sketch and extrude it a set distance from the sketch plane. After that distance is met, extrusion will stop. So the condition for ending the extrusion is that it will proceed a set distance and then stop. Again, this is known as a "blind" end condition.

What is extrude command in SOLIDWORKS?

Using the SOLIDWORKS Surface Extrude feature, we can extrude the boundary edges of a face of a solid body or edges of a surface body in a customized direction. When we run the Extruded Surface command, two things can happen: Either a sketch is active or preselected or no sketches exists.

How do you swept a boss in SOLIDWORKS?

1. Click New.
2. Click Top Plane and click on Sketch.
3. Click Circle and sketch a circle origin as it center.
4. Exit Sketch.
5. Take a plane normal to spline end (or Click on Front Plane and click Sketch.)
6. Click on View Orientation-Normal To.

How do you extrude an extruded surface?

When you create a boss extrude what parameters you can specify in Solidworks?

Direction 1 Up to Vertex: Up to a point. Up To Surface: Up to a surface/plane/face. Offset From Surface: A specified distance away from a surface/place/face. Up to Body: Up to a body within the model.

How do I change the extrusion dimension in Solidworks?

1. RMB "EDIT FEATURE.
2. RMB "EDIT SKETCH. or.
3. Double click on the feature. The dimensions show up. Double click on each dimension to change.

How do you specify direction of extrusion in Solidworks?

How do you extrude a 3D sketch boss?

1. Click Insert > Surface > Extrude.
2. Select a face:
3. Select other faces to define the extrude as required.
4. Select the end condition.
5. For 3D faces, select a plane, edge, 2D face, or sketch line to define the direction of extrusion .

How do you extrude an angle at your boss?

Why is solidworks extruding thin?

Thin features are an option for most sketch-based features such as Boss Extrudes, revolves, sweeps, etc. Rib is automatically a thin feature because it creates a 3D solid from an open sketch.

How do you use revolve boss?

What is revolved boss?

Revolved Boss or Base is a SolidWorks feature tool which is used to create solid 3D model around an axis. You can revolve the sketch around an axis to add materials to it. You can access revolved boss/base from both command manager and menubar.

What is revolve command in SolidWorks?

Much like the Extrude command, the Revolve feature creates geometry from a sketch. However, the Revolve requires one more ingredient, the Axis of Rotation. Once you have a sketch, the next element

is the center line, axis or internal line, of the sketch. Hereâ€™s a few examples.

What does boss mean in Solidworks?

A boss is a feature that creates the base of a part, or adds material to a part, by extruding, revolving, sweeping, or lofting a sketch, or by thickening a surface.

What is blind extrude in Solidworks?

Blind: Defines the distance or length extrusion with a numerical value. Through All: Through the entire extent of the geometry. Up to Next: Extends the extrusion through the bounding box of your model. Up to Vertex: Extends the extrusion to a defined vertex. Up to Surface: Extends the extrusion to a defined surface or â€¦

Wrapping Up:

I believe you now know everything there is to know about Frequent answer: How to boss extrude in solidworks assembly?. Please take the time to examine our CAD-Elearning.com site if you have any additional queries about Solidworks software. You will find a number of Solidworks tutorials. If not, please let me know in the comments section below or via the contact page.

The article makes the following points clear:

- What is extrude command in SOLIDWORKS?
- How do you extrude an extruded surface?
- How do I change the extrusion dimension in Solidworks?
- How do you specify direction of extrusion in Solidworks?
- How do you extrude a 3D sketch boss?
- How do you extrude an angle at your boss?
- Why is solidworks extruding thin?
- How do you use revolve boss?
- What is revolved boss?
- What is revolve command in SolidWorks?