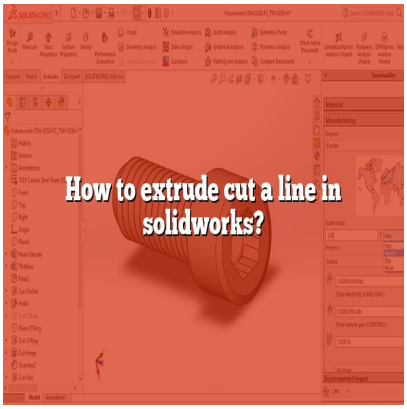


How to extrude cut a line in solidworks?

Description

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If your question is How to extrude cut a line in solidworks?, our CAD-Elearning.com site has the answer for you. Thanks to our various and numerous Solidworks tutorials offered for free, the use of software like Solidworks becomes easier and more pleasant.
Indeed Solidworks tutorials are numerous in the site and allow to create coherent designs. All engineers should be able to meet the changing design requirements with the suite of tools. This will help you understand how Solidworks is constantly modifying its solutions to include new features to have better performance, more efficient processes to the platform.
And here is the answer to your How to extrude cut a line in solidworks? question, read on.

Introduction

Hold Ctrl and select a face to be the starting point for the **cut**. Click Convert to **Cut** (2D to 3D toolbar), or click Tools > Sketch Tools > 2D to 3D > Cut, and click in the graphics area to establish the direction of the cut. The **Cut**-Extrude PropertyManager appears.

Moreover, how do you extrude a cut along a **line**?

Amazingly, how do you extrude a cut in **Solidworks**? 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.

Beside above, how do you extrude a cut surface?

Quick Answer, how do you **extrude** a tube in Solidworks? Click **Extrude** (2D to 3D toolbar), or click Tools > Sketch Tools > 2D to 3D > Extrude, and click in the graphics area to establish the direction of the extrusion. The Base-**Extrude** PropertyManager appears.

How do you do a sweep cut in Solidworks?

What does the cut extrude feature do?

Extruded cut feature removes material from the part. The End Condition, Draft, and Thin Feature options are same as described in Extruded Boss/Base. To create an extruded cut: 1) Sketch the profile for cut. 2) Click Extruded Cut on the Features toolbar or select Insert, Cut, Extrude on the menu bar.

What is boundary cut in SolidWorks?

A boundary spans two or more profiles in one or two directions. A boundary can be a base, boss, cut, or surface. Boundaries follow the profiles specified in Direction 1 (blue) and Direction 2 (purple). The Boundary Surface feature can create surfaces that span the extents of Direction 1 and Direction 2.

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What is Loft cut in SolidWorks?

A lofted cut is a feature that removes material between two sketch profiles. To complete a lofted cut, you must create a minimum of two separate sketches on your part that you wish to cut between. Once you have created your two sketches, you are ready to cut!

How do I cut a part in SolidWorks?

Right-click the sketch and select Edit Sketch. (Sketch toolbar) or Tools > Sketch Tools > Trim. Click in the graphics area next to the first entity, and drag across the sketch entity to trim.

How do you split a surface in SolidWorks?

1. Open install_dir sampleswhatsnewpartssplit.
2. Click Split (Mold Tools toolbar) or Insert > Features > Split.
3. In the graphics area, select Surface-Extrude1 for Trimming Surfaces .
4. In the PropertyManager, under Target Bodies, click Selected bodies.

How do you trim a surface in SolidWorks?

To trim a surface: Create two or more surfaces that intersect at one or more points, or create a surface that intersects with a plane or has a sketch on its face. Click Trim Surface on the Surfaces toolbar, or click Insert > Surface > Trim.

How do you use the Extrude tool in SOLIDWORKS?

Click Convert to Extrusion (2D to 3D toolbar), or click Tools > Sketch Tools > 2D to 3D > Extrude, and click in the graphics area to establish the direction of the extrusion. The Base-Extrude PropertyManager appears. Edit the properties in the Base Extrude PropertyManager.

What is a blind cut SOLIDWORKS?

Blind: A specified distance. Through All: Through the entire part in one direction, normal to the sketch plane. Through All â€“ Both: Through the entire part in two directions, normal to the sketch plane. Up To Next: Up to the next surface of the part.

What is fillet in SOLIDWORKS?

Fillet/Round creates a rounded internal or external face on the part. You can fillet all edges of a face, selected sets of faces, selected edges, or edge loops.

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What is extrude thin?

Thin-features are features that extrude in 2 directions: the first direction is perpendicular to the sketch, which is normal for any 3D sketch-based feature, and the second direction is offset from the original sketch giving the sketch a thickness to be extruded. Figure 5.9 Process of Creating a Thin Feature.

How do you extrude a sketch along a path in Solidworks?

Can you extrude cut a 3D sketch in Solidworks?

You can create extruded surfaces from models that include 2D or 3D faces and knit the extruded surfaces to surrounding features. Click Insert > Surface > Extrude. Select a face: To extrude from a 3D face, select a 3D face.

How do you sweep multiple lines in Solidworks?

How do you create a loft in Solidworks?

1. Go to: File > New > Part.
2. Go to Tools > Sketch Entities > Spline.
3. Create a 3D Sketch.
4. Create a shape with the Spline tool.
5. Go to Tools > Sketch Entities > Spline or click on the Spline icon.
6. To define the shape of the Loft you can add (multiple) Guide Curves.

Conclusion:

Everything you needed to know about How to extrude cut a line in solidworks? should now be clear, in my opinion. Please take the time to browse our CAD-Elearning.com site if you have any additional questions about Solidworks software. Several Solidworks tutorials questions can be found there. Please let me know in the comments section below or via the contact page if anything else.

The article clarifies the following points:

- What does the cut extrude feature do?
- What is boundary cut in SolidWorks?
- What is Loft cut in SolidWorks?
- How do you use the Extrude tool in SOLIDWORKS?
- What is a blind cut SOLIDWORKS?
- What is fillet in SOLIDWORKS?
- What is extrude thin?
- How do you extrude a sketch along a path in Solidworks?
- Can you extrude cut a 3D sketch in Solidworks?
- How do you sweep multiple lines in Solidworks?

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