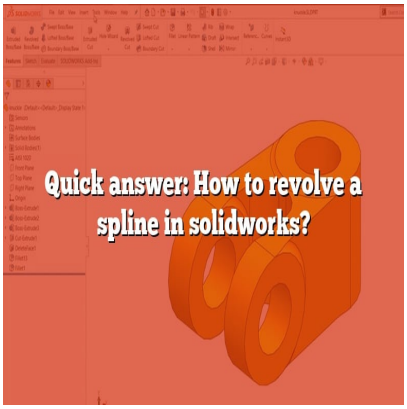


Quick answer: How to revolve a spline in solidworks?

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Description

Starting with this article which is the answer to your question Quick answer: How to revolve a spline in solidworks?.CAD-Elearning.com has what you want as free Solidworks tutorials, yes, you can learn Solidworks software faster and more efficiently here.
Millions of engineers and designers in tens of thousands of companies use Solidworks. It is one of the most widely used design and engineering programs and is used by many different professions and companies around the world because of its wide range of features and excellent functionality.
And here is the answer to your Quick answer: How to revolve a spline in solidworks? question, read on.

Introduction

Also, how do you revolve a spline?

In this regard, how do you **revolve** in SOLIDWORKS? 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.

Likewise, how do I move a spline in **SOLIDWORKS**?

1. Spline points: Select and drag a **spline** point to display the Point PropertyManager.
2. Control polygon handles: Select and drag a control polygon handle.
3. **Spline** handles: In the Spline PropertyManager, under Parameters:

Furthermore, why canâ€™t I revolve in **SOLIDWORKS**? Unless youâ€™re creating a Thin Feature Revolve then SolidWorks can only Revolve a closed profile. This means that you need to have no gaps or breaks around the outside of your profile. Try using an Extruded Boss/Base to check whether your

profile is closed.

1. In the Design workspace, Form contextual environment, select Create > **Revolve** .
2. In the Revolve dialog:
3. Select the Profile to **revolve**. Click Axis and then select the axis to revolve around.
4. Click OK.

How do you use the spline tool on Onshape?

How do you do a revolve?

1. Click Model > Revolve and create a sketch to revolve. This method is referred to as action-object.
2. Select an existing sketch and then click Model > Revolve.
3. Select a datum plane or planar surface to use as the sketching plane and then click Model > Revolve.

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How do you make a revolve?

1. Click Modify | Create Revolve tab Draw panel (Boundary Line).
2. To create a single revolve, sketch a closed loop.
3. To create more than one revolve, sketch multiple, non-intersecting, closed loops.

What is revolve cut in SolidWorks?

â€œ â€“ The revolved cut feature allows us to take a basic shape and revolve it around a center line to create a cut into parts or multiple parts.

How do you move a spline?

1. Select the spline, and then right-click to display the shortcut menu.
2. Click Edit Spline.
3. â—¦ To move a single point on the spline:
4. Click Single Point in the MOVE PNTS menu.
5. Select the point you want to move.
6. Click in the new location to place the point.
7. â—¦ To move range of points on the spline:

What controls the direction and magnitude of a splines curvature SolidWorks?

1. Spline Points. You can drag the spline points around the graphics area to manipulate the shape of the spline.

2. Spline Handles. You can manipulate the spline handles at each spline point to edit and define the weight or magnitude and direction of the spline at that point.
3. Control Polygon.

How do you constrain splines?

You can constrain a style spline by dimensioning line segments in the control polygon. To constrain line segments in a control polygon: In an open sketch, click Smart Dimension (Dimensions/Relations toolbar) or Tools > Dimensions > Smart. Click a line segment in the control polygon.

How do you use the revolve base feature in Solidworks?

1. Create a new 2D sketch.
2. Create all solid sketch geometry on one side of the axis.
3. Choose Revolve Boss/Base from the features menu.
4. Pick the axis you wish to use for the revolve.
5. Specify direction and angle.
6. Accept the Feature.

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How does the revolve tool work?

How do you use revolve Extrude in Solidworks?

1. Open a sketch on a face or plane, and sketch a profile of the cut.
2. Click Extruded Cut or Revolved Cut (Features toolbar), or click Insert > Assembly Feature > Cut , then Extrude or Revolve.
3. Set the options as needed in the Extrude or Revolve PropertyManager.

How do you change the direction of revolution in Fusion 360?

How do you rotate a cut in Fusion 360?

How do you rotate a sketch in Fusion 360?

How do you make a 3D spline on shape?

1. Tap the 3D fit spline tool.
2. Select vertices to create a 3d fit spline.

3. Check the box to create a closed spline.
4. Tap the checkmark.

How do you trace an image on shape?

Conclusion:

I hope this article has explained everything you need to know about Quick answer: How to revolve a spline in solidworks?. If you have any other questions about Solidworks software, please take the time to search our CAD-Elearning.com site, you will find several Solidworks tutorials. Otherwise, donâ€™t hesitate to tell me in the comments below or through the contact page.

The following points are being clarified by the article:

- How do you use the spline tool on Onshape?
- How do you make a revolve?
- How do you move a spline?
- What controls the direction and magnitude of a splines curvature SolidWorks?
- How do you constrain splines?
- How do you change the direction of revolution in Fusion 360?
- How do you rotate a cut in Fusion 360?
- How do you rotate a sketch in Fusion 360?
- How do you make a 3D spline on shape?
- How do you trace an image on shape?

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