



# Quick answer: How to rotate a reference plane in solidworks?

## Description

With this article you will have the answer to your Quick answer: How to rotate a reference plane in solidworks? 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 Quick answer: How to rotate a reference plane in solidworks? question, read on.

## Introduction

Considering this, how do you move a reference plane in Solidworks? Move the plane by dragging the edge of the plane. Copy the plane by selecting a plane in the graphics area. Then hold down the Ctrl key and, using the edge, drag to a new location.

Subsequently, how do you offset a reference plane in Solidworks? Offset Planes To create an offset plane, select the Reference Geometry drop down on the CommandManager and choose the Plane option. Once the option to create a plane is open, select a face or another plane and set a distance for the offset.

You asked, how do I change the sketch reference plane in Solidworks? To change the plane, right click on the sketch in the Feature Manager and select the Edit Sketch Plane Icon. When you do, the Edit Sketch Plane command opens. To change the plane, right click in the window and clear the selections.

Also know, how do you rotate an axis view in Solidworks? Rotates the model view in part and assembly documents. Click Rotate View (View toolbar) or View > Modify > Rotate then drag the pointer . Drag with the middle mouse button.

## How do you create a reference plane at an angle in SolidWorks?

## How do you rotate the boss base in SolidWorks?

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. Revolved Cut (Features toolbar) or Insert > Cut > Revolve.

## How do you rotate a part in SolidWorks?

## How do I move origin in SolidWorks?

1. Go into sketch mode and then choose the tool option.
2. As soon as you choose the tool option go to "sketch tools".
3. Now from the drop-down choose the "Align" option.
4. Choose the Align Grid/Origin option.
5. Now select the point where you need to move the origin.

## How do you rotate a sketch in Solidworks?

1. In an active sketch, click Modify Sketch. (Sketch toolbar) or Tools > Sketch Tools > Modify.
2. In the dialog box, do one of the following: Type a value for Rotate to set the rotation value and press Enter. Rotate a sketch in the graphics area with the pointer: Press the right-mouse button.

## How do you change the plane of an object in Solidworks?

## How can I change my plane?

To change planes for a connecting flight, start by asking a flight attendant on your flight or reviewing the Departures board in the airport to find your departure gate number. Then, if your luggage hasn't been checked through to your final destination, pick it up from baggage claim and go back through security.

## How do you rotate a 180 degree in SolidWorks?

1. To snap, right-click the selected ring and choose Snap while Dragging.
2. To rotate in pre-set increments, right-click the selected ring and choose Rotate 90° or Rotate 180°.
3. To type specific increments, right-click the center sphere and select Show Rotate Delta XYZ Box.

## How do I change the axis orientation in a SolidWorks assembly?

1. Open a model.
2. Click View Orientation (View toolbar).
3. In the Orientation dialog box, click the Up Axis flyout and select an option: Apply Y-up views. Uses the Y-axis as the up orientation for views. Apply Z-up views. Uses the Z-axis as the up orientation for views.

## How do you make a left plane in SolidWorks?

## How do you make a plane perpendicular in Solidworks?

## How do you make an angled plane on shape?

## How do you make a reference plane?

1. On the ribbon, click (Reference Plane). Architecture tab Work Plane panel (Reference Plane)  
Structure tab Work Plane panel (Reference Plane)
2. To draw a line: On the Draw panel, click (Line).
3. To pick an existing line: On the Draw panel, click (Pick Lines).

## How do you revolve a cylinder?

## Why can't I revolve SolidWorks?

Unless you'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.

## Final Words:

I sincerely hope that this article has provided you with all of the Quick answer: How to rotate a reference plane in solidworks? information that you require. If you have any further queries regarding Solidworks software, please explore our CAD-Elearning.com site, where you will discover various Solidworks tutorials answers. Thank you for your time. If this isn't the case, please don't be hesitant about letting me know in the comments below or on the contact page.

The article provides clarification on the following points:

- How do you create a reference plane at an angle in SolidWorks?
- How do you rotate a part in SolidWorks?
- How do I move origin in SolidWorks?
- How do you change the plane of an object in Solidworks?
- How can I change my plane?
- How do I change the axis orientation in a SolidWorks assembly?
- How do you make a left plane in SolidWorks?
- How do you make a plane perpendicular in Solidworks?
- How do you make an angled plane on shape?
- Why can't I revolve SolidWorks?