# Quick answer: How to join lines in fusion 360?

## Description

The objective of the CAD-Elearning.com site is to allow you to have all the answers including the question of Quick answer: How to join lines in fusion 360?, and this, thanks to the Fusion 360 tutorials offered free. The use of a software like Fusion 360 must be easy and accessible to all.
Fusion 360 is one of the most popular CAD applications used in companies around the world. This CAD software continues to be a popular and valued CAD alternative; many consider it the industry standard all-purpose engineering tool.
And here is the answer to your Quick answer: How to join lines in fusion 360? question, read on.

## Introduction

Amazingly, how do you **join** parts in Fusion 360?

1. Click Design > Solid > Modify > Combine. .
2. From the Combine dialog, select the Target Body.
3. Select Tool Bodies.

Considering this, how do you **join** the edges in fusion? Click or hold Ctrl and click to select one or more edges in the first group. In the Merge Edge dialog, select Edge Group Two. Click or hold Ctrl and click to select one or more edges either in the same body or in a second body you want to be joined to the first. Click OK.

Correspondingly, how do you fill gaps in **Fusion** 360?

1. Press the â€œFill Gapsâ€ button in the â€œSketchâ€ panel.
2. Enter a gap width.
3. Select the fill or join mode.
4. Check the delete double checkbox to delete double **lines**.
5. Check the project geometry checkbox to project the geometry to a new sketch.
6. Press â€œOKâ€ button.

Likewise, how do you join a spline and a line **Fusion** 360? Create a joint between the shaft and the gear housing In the Model workspace, choose Assemble > Joint. In the Joint dialog, set Motion Type to Revolute. Fusion 360 enables the required degrees of freedom for the motion type. Fusion 360 automatically chooses the Z axis, but you can specify another axis if necessary.

## How do you make a hinge joint in Fusion 360?

# How do I connect two edges in Inventor?

1. Click Freeform tab Modify panel Merge Edges .
2. Select Set 1 open edges in the graphics window.
3. In the dialog box, change the focus to Set 2 and then select the Set 2 open edges in the graphics window.
4. Set the Mode. Use To Edge to blend to the second edge selection.
5. Select OK.

# How do you subdivide in Fusion 360?

On the Sculpt contextual ribbon, expand the MODIFY panel and select Subdivide. The SUBDIVIDE dialog box opens and prompts you to select a T-Spline face. Pick the face in the middle of the front surface of the body. If a dialog asks you to upgrade your T-Spline, click OK.

# How would you edit and sculpt a T-spline form in Fusion 360?

1. Issue: How to edit a t-spline after finishing the form in Fusion 360.
2. Solution: With the timeline enabled in Fusion 360: Right-click the form icon in the timeline (purple icon) Select Edit.
3. Products: Fusion 360;
4. Versions:

# How do you make a cavity in Fusion 360?

# How do you use the loft in Fusion 360?

1. In the Design workspace, Form contextual environment, select Create > Loft .
2. In the canvas, select two or more Profiles to loft.
3. In the Loft dialog, select an end condition or continuity for the first and last profiles:
4. Select a Guide Type:

# How do you cut a shape into another shape in Fusion 360?

# How do you connect two sketches in Fusion 360?

There are a couple of ways to combine sketches in Fusion 360. The first is to copy and paste one sketch into another. Select the first sketch and Copy it to your clipboard. Next, edit your second sketch and select Paste.

# How do you combine fusion lines in sketches?

# How do you add a construction line in Fusion 360?

# How do you mate objects in Fusion 360?

# What is a joint origin in Fusion 360?

A Joint Origin defines the geometry that will be used to relate two or more components with joints in Fusion 360. The Joint command creates joint origins within the context of a specific joint. The Joint Origin command creates independent joint origins to facilitate more complex joint locations.

# Are there mates in Fusion 360?

If you previously used Solidworks or Inventor CAD, you may used to the term Constraints or Mates. Fusion 360 Joints are able to do the same thing as a Mate or Constraint but often only require a single Joint to constrain multiple degrees of freedom!

# How do you make a movable joint?

# How do you make a revolute joint?

1. Click Revolute Joint .
2. Click New Mechanism .
3. The Null Offset option is set by default (option button).
4. Select Line 1 in the geometry area.
5. Select Line 2 in the geometry area.
6. Select the planes as shown below:
7. Assign the Angle driven command to the revolute joint if needed.

# Bottom line:

I believe you now know everything there is to know about Quick answer: How to join lines in fusion 360?. Please take the time to examine our CAD-Elearning.com site if you have any additional queries about Fusion 360 software. You will find a number of Fusion 360 tutorials. If not, please let me know in the comments section below or via the contact page.

The article makes the following points clear:

- How do you make a hinge joint in Fusion 360?
- How do I connect two edges in Inventor?
- How do you subdivide in Fusion 360?
- How do you make a cavity in Fusion 360?
- How do you cut a shape into another shape in Fusion 360?
- How do you combine fusion lines in sketches?
- How do you add a construction line in Fusion 360?
- How do you mate objects in Fusion 360?
- Are there mates in Fusion 360?
- How do you make a movable joint?