

# Solid contains no solid bodies fusion 360 fix

Does Fusion 360 have a solid body?

Fusion 360 can indeed create a solid body from an STL file. Now Fusion 360 did a pretty good job of processing this file and turning it into a solid body. However, you'll see here in the picture that there are still A LOT of triangular surface faces that make up the solid body.

Can a drawing be created in Fusion 360?

"The design contains neither solid bodies nor sketches. A drawing cannot be created" in Fusion 360 The design contains neither solid bodies nor sketches. A drawing cannot be created" The model only contains surface, mesh, and/or T-Spline bodies and no sketches or solid bodies.

Does the design contain solid bodies or sketches?

The design contains neither solid bodies nor sketches. A drawing cannot be created" The model only contains surface, mesh, and/or T-Spline bodies and no sketches or solid bodies. Was this information helpful?

How do I Fix an invalid geometry in fusion?

An example of how to use the Validate feature to repair the geometry is available here: [Fixing Imported Component in Fusion - Stitching and Validating Surface Bodies](#). For parametric modeling, review the timeline warnings and errors to track the invalid geometry and repair it (see: [Resolving Timeline Warning or Errors in Fusion](#)).

How do I report a problem with fusion?

Report bugs and other issues you're seeing on our Support forum. Check the current status of Fusion and other cloud services. While using the Combine, Press/Pull, Extrude, or Boundary Fill feature on solid bodies in Fusion, the following message appears: Error "There was a problem combining geometry together.

I have a model that was created by mirroring a body. Where the two bodies join is a line. I don't want the line there as it is seen in animations. I went into the model and did Modify -> Combine and it works, the two bodies ...

: Fusion 360, "Warning ? " : ?/T-Spline, ? : ...

I need a method to fill a void/cavity inside a design so that it is solid. In this case Extrude/Push-Pull does not seem to provide the solution. The shape inside the body is irregular. Below is a simple illustration of the problem. ...

The Surface Bodies were stitched together = single Stitched Surface Body Thickened each part of stitched Body as F360 could not do all together = individual Solid Bodies (Body57 to Body67) Join Bodies - two ...

## Solid contains no solid bodies fusion 360 fix

Manifold Topology is the way we describe how Fusion 360 recognizes that a shape is a solid. If the shape is open, Fusion 360 will create a surface model. If the shape is closed, Fusion 360 knows that it has a solid. ...

When using Convert Mesh to create a solid body in Fusion, surface bodies are created instead of a solid body and the warning "compute failed" is shown. Compute Failed ...

Follow the steps to verify if this is expected behavior: Right-click the top-level component and select show all components. Right-click the top-level component and select ...

While using the Combine, Press/Pull, Extrude, or Boundary Fill feature on solid bodies in Fusion, the following message appears: Error "There was a problem combining geometry together. If attempting a ...

If your design contains surface, mesh, or sculpt (t-spline) Bodies. Fusion 360 Drawings do not support these body types. Only Solid Bodies (BRep) will be displayed in the Drawings. If that is not the case then please ...

2: Exported to STP file and opened in Fusion again, the two unstitched bodies changed into one unclosed surface. But still a solid. A surface has no thickness but for the sack of manufacturing it is fine. Could be ...

Fusion 360 could really be a great product if a new "super-streamlined for user satisfaction" help system is implemented; this is just as important as good and friendly tech support. I am trying to be supportive of ...

Users reported that a DXF/DWG or SVG created in another program does not have closed profiles after importing into Fusion: The sketch profiles from the imported file are not possible to Extrude or Press/Pull. The ...

In this case, open dwg in Fusion, convert to B-Rep first and then exporting to IGES will work. If your data has 2D drawings, use AutoCAD to export to IGES (Fusion does not support exporting 2D drawings). If your DWG has ...

This study proposes a new set-pair analysis (SPA) method by enhancing existing SPA methods using interval-based fuzzy numbers (IFN), referred to as the IFN-SPA method. ...

Best answer: To join or combine solids or bodies, Do the following: Click Design > Solid > Modify > Combine. How do you connect two pieces in fusion? FAQ How do you combine ...

Most likely there is a gap in the surfaces, which is converting all the faces to surfaces, instead of solid bodies. If you expand the component, you can see it is made up of ...

## **Solid contains no solid bodies fusion 360 fix**

How to stitch surface bodies together to make a solid body in Fusion 360, and validate imported surface bodies to repair broken solids. To create a solid body from a surface body in Fusion 360 Use Stitch to close any ...

You may want to create a hand sketch to explain what this bulk solid geometry is supposed to look like. I don't think Taylor's and my understanding of what you call bulk solid body is what you envision. To ...

Now lets enter CAD into the equation. Fusion 360 is one of the few CAD packages that allows users to covert a quad-mesh into a T-Spline and then NURBS surface or BRep (solid body). A T-Spline is really only an ...

After 3 hours of working with Fusion (and the online docs) and failing to find a solution, maybe someone can give a hint here. I had some experience with 3D construction for visualization purposes, couple of years ...

Web: <https://www.bardzyndzalek.olsztyn.pl>

