

Solidworks the file does not contain any surface solid data

Does SolidWorks support 3DM files?

Hi @Josh10 ,that Plugin looks quite old (18yrs old at Rhino v3 times) Unfortunately 3DM files are poorly supported by solid modelers. Solidworks does import Rhino geometry (not curves) but you lose all layers hierarchy.

Does SolidWorks import Rhino geometry?

Solidworks does import Rhino geometry(not curves) but you lose all layers hierarchy. More info here This can be ok if Rhino file does contain few objects but can be a nightmare if you have to deal with more elements. The only solid Modeler I found which manages correctly Rhino files converting layers to parts is SpaceClaim.

What is 3D Interconnect in SolidWorks?

SOLIDWORKS introduced the 3D Interconnect import option in 2017. This feature allows you to open proprietary CAD data in its native format, keeping a link to the original part or assembly, and work seamlessly with third-party CAD files inside SOLIDWORKS.

What can you do with non-native SOLIDWORKS models?

You can run "Import Diagnostics" on non-native SOLIDWORKS models, but you cannot heal or fix any bad geometry. Plus, the ability to work seamlessly with third-party CAD files inside SOLIDWORKS.

What happens if I run 'import diagnostics' on a non-native SolidWorks model?

When you run "Import Diagnostics" on a non-native SOLIDWORKS model, it will not allow you to heal or fix any bad geometry. Modifying the geometry could change the face and edge identifiers, which would break the update capability of "3D Interconnect".

Does SolidWorks support STEP and IGES files?

While SolidWorks supports third-party CAD files, it may encounter issues with STEP and IGES files. For instance, the 'Use Part Level Settings' option can cause unwanted results. You can run 'Import Diagnostics' on non-native SolidWorks models, but you cannot heal or fix any bad geometry.

The Rhino file is not properly saved. 2. KeyShot software is not up to date. 3. KeyShot does not support your version of Rhino yet. Here are some things you can try: 1. ...

STL is DEFINITELY a bad choice because it only transfers tessellated data which is nearly useless compared to the original b-rep solid/surface data. "SW2013 introduced the ...

Initial Graphics Exchange Specification. Focused on wireframe but includes other solid modeling data as well. This format does not translate especially well to SOLIDWORKS. Parasolid *.x_t / *.x_b: Native kernel to ...

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Did the component contain any surface / solid bodies? When you imported the STEP file, a SWX assembly document was created, but were the component part files created as ...

Hey guys, I'm having a problem opening this step file. No matter who tries at our office we get the same issue. It says that there is no solid data and opens a blank page. I ...

The SOLIDWORKS software can import and create multiple solid bodies from a mesh file, provided the mesh is closed. If the mesh is not closed, the Import as Solid Body ...

This article describes the procedure to show surface bodies in a drawing of a multibody part that contains both surface and solid bodies. By default, drawing files do not show the surface bodies of a multibody part containing surface ...

If a SOLIDWORKS part or assembly file has multiple configurations, when you save the file as a Composer file, you can choose which configuration to save to the Composer file. ...

Not being able to run "Feature Recognition" on an imported part that was not converted to the SOLIDWORKS file format. A STEP or IGES file opened as an assembly in its native format will not allow individual ...

Each time when a non-SOLIDWORKS file is imported to SOLIDWORKS, it is recommended that the geometry is checked for such topology errors. The first thing that everyone should do is to make sure that ...

By default, drawing files do not show the surface bodies of a multibody part containing surface and solid bodies. The surfaces are considered reference geometry. Thus, the surfaces need to be imported into the model view of the ...

All the options needed to successfully open the step is hopefully ticked (Pic 3). Also, every time I close and say Don't Save, it has already created a Solidworks file ...

I have a part model that is solid geometry. When exporting the part using STEP it generates the file but the STEP file does not contain any geometry when imported. The part does not have any geometry checks and ...

More SOLIDWORKS Tutorials . SOLIDWORKS Assembly Hole Series Explained. Create a SOLIDWORKS Assembly from a Multibody Part. How to Link Values with External Files in SOLIDWORKS. Quickly Create a Radial ...

If users check the "Surface/solid entities" checkbox, SOLIDWORKS will import any 3D geometry it can from the file. Related checkboxes underneath allow users to ask SOLIDWORKS to try and form the ...

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Are your STEP and IGES files imported into SOLIDWORKS acting a little strange? SOLIDWORKS added the "3D Interconnect" import option to the program in 2017. This allows for proprietary CAD data being opened in its ...

The home for the SOLIDWORKS Forum. REAL People, REAL Experiences, REAL Knowledge. ... #3c3e43; background-color: #f7f8f9;">"The JT file could not be imported ...

So I brought in 2 files into the PDM and when I highlight them to see a preview, this message pops up. I followed the directions, but the check box was already checked for ...

After updating the sketch for one of my parts, the model stopped displaying. It only appears in a sort of wireframe mode when I select the solid body, but disappears again upon ...

On all of them I get the same problem "the file does not contain any geometry data"; load this same file in SImplify 3d and opens first time. Now I open in prusa slicer and it ...

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