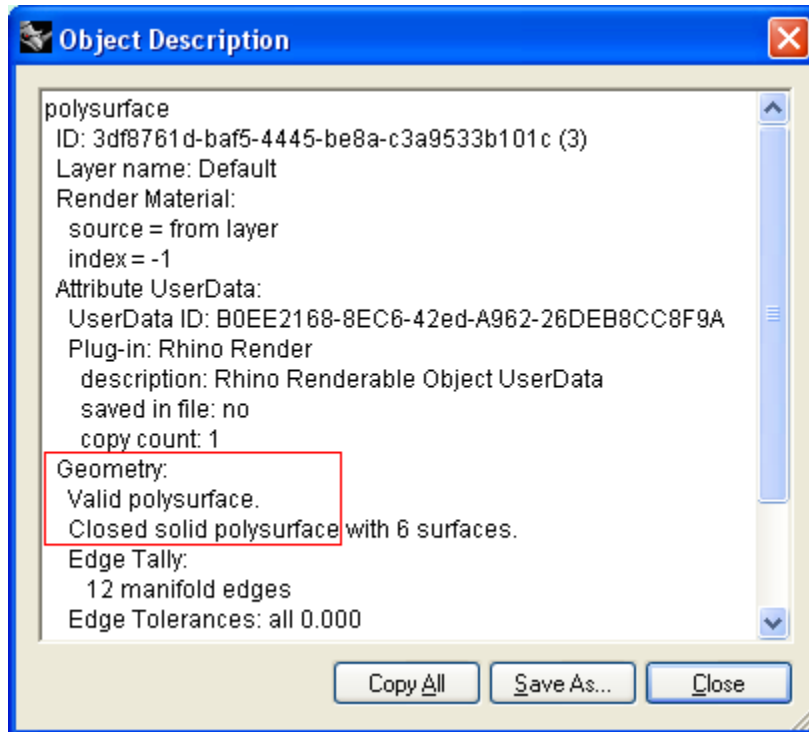


## Models supported by Scan&Solve™

Currently, Scan&Solve™ works with Rhino **solids** only. To determine if a model is a proper solid, use the command “**What**” on the geometry. It should report something similar to the following:



If the model is a proper solid, "Closed solid polysurface..." should appear in the listing as shown above.

If your model is not recognized as solid, you may be able to use the tools under Rhino's [Solid] menu to make it into a solid that Scan&Solve™ can use.

Future versions of Scan&Solve™ are expected to handle other types of models in Rhino, including volumes bounded by collections of surfaces and/or meshes.

If you want to use Scan&Solve™ on a solid with mesh boundary, read below.

## Does Scan&Solve™ work on boundary meshes?

The answer depends on how good your mesh is. If the mesh is good enough, then you can use Rhino command “**MeshToNURB**” to convert the mesh into valid boundary representation of a solid. The mesh polygons become legitimate faces of the solid, and you can then use Scan&Solve™ to perform stress analysis as with any other valid boundary representation.

However, for this to work, the mesh must satisfy several fairly technical conditions, including:

1. Every edge in the mesh must be shared by exactly two polygons or triangles.
2. Every vertex has a single “ring” of faces around it.
3. Mesh faces cannot intersect each other, except at the edges and vertices.
4. Mesh faces (polygons) must be consistently oriented (with normal pointing continuously in the same direction).

These conditions may be verified and corrected using the Mesh Repair and Mesh Edit tools in Rhino.

In addition, currently Rhino restricts the **MeshToNURB** command to meshes with fewer than 20,000 faces.