Understand and remove "Edge is short" notes

Why do *Edge is short* notes appear?

Edge is short notes appear when the system finds an edge on a model smaller than the minimum edge length defined in the system settings (Fig. 1).   (In mm, regardless of unit settings.)

You can change the minimum edge length in the **Model Advisor** tab under **System Settings** (Fig. 2).

An **Edge is short** note also appears if the edge length is smaller than the defined Part Resolution. The Part Resolution is the level of detail you can create in your parts. For example, if you set your resolution to 1.0E-6 mm, you can create edges with measurements as small as 1.0E-6 mm.

Why do *Edge is short* notes matter?

Models containing short edges make it difficult to use the model data in downstream applications like CNC, CAM, or CAE.

How does OneSpace Modeling check for short edges?

Model Advisor (a feature built into OneSpace Modeling) finds short edges and other problems that may prevent manufacturability. Model Advisor looks for:

- **min Edge** (edges that are smaller than the minimum length or smaller than the Part Resolution).

- **Void Shells** (pockets of air or hollow chambers).

- **Knife Edges** (two faces that meet at the same point to form a knife edge).

- **Non-manifolds** (spatial ambiguities that cannot be manufactured).

How do I activate Model Advisor and check for short edges?

Model Advisor automatically checks for short edges:
- When Model Advisor is active (and it's active by default).
- When you load or import a file.
- After a machine or modify command on a model.

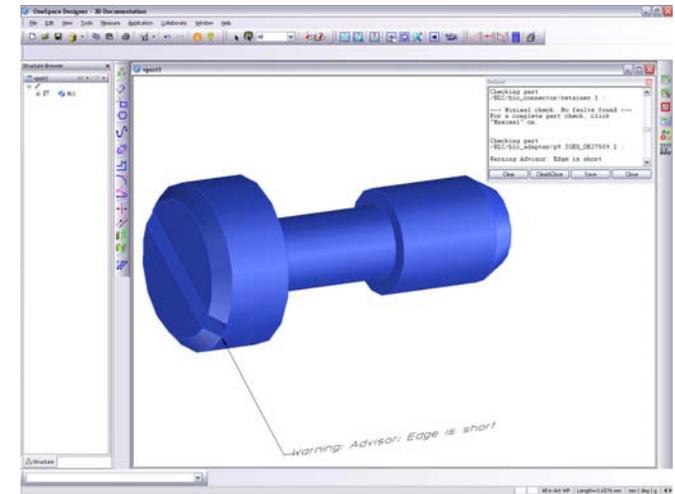**Note:** You can also run Model Advisor using the **Check Part** command (Fig. 3).

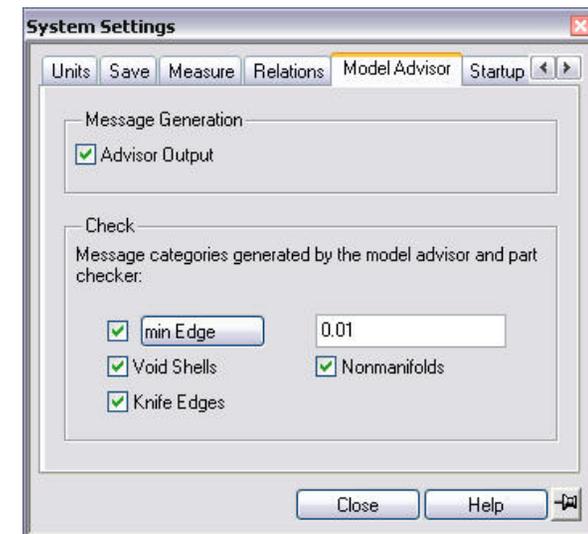

Fig. 1. Edge is short note pops up on screen.

Fig. 2. Model Advisor System Settings menu

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How do I remove *Edge is short* notes from the model?

A short edge can be removed by modifying the geometry. If a model has short edges, you can resolve it using a number of options:

Option 1: Remove short edges by using machining operation, like **"Mill"** or **"Punch,"** followed by a **"Cut"** or **"Align"** command.

Option 2: **"Reintersect"** (Surfacing menu) the short edge and **"Merge"** (Modify 3D menu) the vertices of the short edge.

Summary

Edge is short doesn't signal danger in the design stage. However, it may be difficult for others to use your model later. Short edges can be lost due to different resolution settings in downstream applications, like CNC, CAM, etc. If you use the model for FE Analysis, the short edge can cause unpredictable behavior. And short edges might interfere with mesh creation in FE Analysis and generate false FE results.
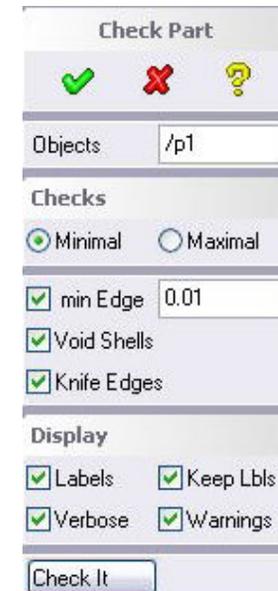
In these cases, follow the options above to remove the short edges.

Fig. 3. Check Part command.