

Geometry Check Messages

The geometry check process warns you of potential problems that could occur after you modify the part, although none of the potential problems would cause the feature to fail. The messages in this document are displayed when Pro/ENGINEER detects a geometry check.

The following table lists the geometry-specific messages and suggested actions.

This feature is defined with dimensions that are very small compared to the rest of the part.

Change the part accuracy.

The surfaces being merged do not exactly match (surface merge only).

This feature removes small highlighted edges and creates imprecise vertices. Redefine the surfaces so there are no small edges.

The given radius is too large to round edge

No. nnn. (round only)

- Reduce the round radius so the edge can be rounded.
- Define a relation so the proper round radius is automatically maintained.
- Redefine the round feature so it no longer references the edge.

This feature references an edge (No. nnn) that is not present in the geometry (round only).

An intervening feature was modified so it removed the edge. Redefine the round feature so it no longer references the edge.

This feature references an edge (No. nnn) that was already rounded by another feature. Feature No. nn, ID mm was redefined so that it rounded this edge (round only).

Redefine one of these features so the edge is only rounded once.

This feature references an edge (No. nnn) that, due to modification of the part, was not created during last regeneration (round only).

The feature that originally created the edge has been deleted, or, this edge was originally created by feature ID nnn (round only).

Redefine the round feature so it no longer references the edge.

The beginning or ending position of the section is very close to, but not touching the highlighted edge (vertex).

- Redefine feature attributes to ... (or depth to ...).

...Or...

- Redefine the feature so the section starts or stops at the highlighted edge (vertex).

This feature creates a tiny edge. The edge is highlighted on screen.

This feature was sketched on a drafted plane, which might be responsible for the problem.

A previously existing edge was almost completely trimmed away, creating a tiny edge.

This feature was sketched on a drafted plane, which might be responsible for the problem.

The system could not construct the intersection of the part and feature.
Redefine the feature so it intersects the part.

This feature contains self-intersecting surfaces.
The highlighted surface has overlapping geometry.
A side surface of this feature is very close to, but not touching, the highlighted vertex.
Align the highlighted entities to the highlighted vertices.

This feature cuts (leaves) tiny piece(s) from an existing edge(s).
Redefine the round feature so it passes through the highlighted vertex.

The highlighted surface contains a singularity near the indicated point (where the surface isolines in the u or v direction become tangent or anti-tangent). This surface may cause problems with future operations such as creating an offset surface.
Exclude or delete the surface and recreate the geometry in Pro/ENGINEER.
See the Interface Guide for detailed information on creating solids from imported surfaces.

Accepting the requested changes in the highlighted edges can lead to misalignment near the indicated points (for misalignment during Zip Gaps).

- Reject the requested changes. Unselect the **Join Surfs** attribute, then select a longer set of chains on both sides of the indicated points. Choose **Zip Gaps**, then reselect the **Join Surfs** attribute.
...Or....
- Align the surface boundaries near the indicated points using the option **Edit Boundary**.
...Or...
- Delete that portion of the geometry and recreate it.
See the Interface Guide for more information on creating solids from imported surfaces.

Accepting the requested changes in the highlighted edges can lead to misalignment near the indicated points (for misalignment during Fix Tangency).

- Reject the requested changes, unselect the highlighted chain of edges, and continue performing **Fix Tangency** on the remaining selected edges.
...Or...
- Delete that portion of the geometry and recreate it.
See the Interface Guide for more information on creating solids from imported surfaces.