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UG HEAL GEOMETRY FUNCTION

Until there is a universal data translation format and tolerance standard we are always looking for good tools that assist in data translation issues. Whether getting a clean data set ready to export, or trying to fix errors from a bad data import, every tool helps.

A new addition to UG as of v17 is the Heal Geometry function. It is still a 50/50 gamble that it will do anything to help when you have disappearing surfaces in your model, but when you're scrambling to get a clean data-set, what's the worst it could do? We have had reasonable success with it and if you need another method to try on a bad model, it can't hurt.

OVERVIEW:

Heal geometry corrects some/most types of geometry and topology problems either A) just before exporting part data or B) just after importing part data.

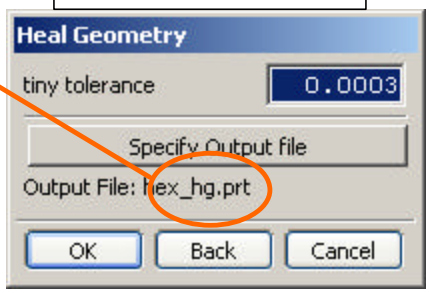
- It can be used on solids and sheet bodies only.
- Unparameterizes the resulting bodies.
- You set the tolerance to correct geometry SMALLER THAN and export a new part file which by default adds the suffix "_hg.prt" for 'heal geom. Part'. The operation filters the data for tiny errors that match your tiny tolerance and tries to fix them.

TO RUN IT:

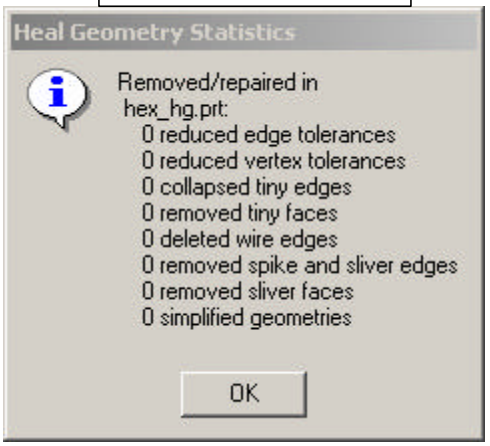
Under **FILE – EXPORT – Heal Geometry**. Dialog box comes up.

1. Specify the tiny tolerance
2. Specify new output file name and path
3. Click OK to execute
4. See the status in the cue line. A window pops up telling you what UG was able to fix or not.

Export Dialog Window



Output report is created



CAVEATS:

- Good thing to try if you are having problems getting good data during import or export to another CAD system.
- If you have surfaces that don't show up in shaded view, this may help.
- As in life, it's not a guaranteed process, but when you're scrambling to get good data it's one more tool in the CAD Arsenal.
- C an also be used in conjunction with exporting a parasolid, then trying the heal geom.
- Just remember it is NOT associative or parameterized.