

Rhino Checks

Use the command bar (at the top) to type in commands – press “enter” key to run.

1. Check the mesh for errors:
 - Select the mesh; command “SelMesh” or “SelAll”
 - Run command “CheckMesh”: a dialogue box will appear, telling you whether it is a “good” or “bad” mesh. If it is a bad mesh it will write a list of errors.
 - BE CAREFUL: the part may be made up of more than one mesh. The mesh check will do each mesh individually, so it is important to scroll down the list to see all of them. Also when selecting the mesh, the number of meshes will be written on the toolbar.
2. Depending on the error, use the following commands to correct
 - CullDegenerateMeshFaces
 - ExtractNonManifoldMeshEdges (when these are selected, delete)
 - SelDup / SelDupAll (when these are selected, delete)
 - UnifyMeshNormals
3. All surfaces and polysurfaces should be closed, to test for any open ones, use the following commands:
 - SelOpenSrf
 - SelOpenCurve
 - SelOpenPolysrf
 - ShowEdges
 - These should be capped / closed individually.
4. When exporting the mesh, a tolerance of 0.01 is recommended.
5. Usually a good, clean file is 10MB or less.