

How to Create a Model in Rhinoceros 4.0 and Import it into *Examine*^{3D}

Rhinoceros 4.0 (Rhino) is a NURBS based solid modelling package for Windows that can create 3D shapes and surfaces with no limit on complexity. *Examine*^{3D}, developed by Rocscience Inc., is a 3D computer-aided engineering stress analysis package for underground excavations in rock. This Developer's Tip demonstrates how to create a model in Rhino for analysis in *Examine*^{3D}. The process for creating a model in Rhino and importing the surface mesh into *Examine*^{3D} is outlined as follows:

1. Adjust settings in Rhino and create your 3D model
2. Create a surface mesh in Rhino
3. Export your model as a .dxf file
4. Convert the .dxf file to .geo format
5. Import into *Examine*^{3D} and check mesh quality

Step 1: Create your 3D Geometry in Rhino

When you create a new project in Rhino, you are prompted to select a unit template file. Select a unit template that is best suited to your project. Typically, "Large Objects – Meters" is adequate for *Examine*^{3D} projects.

The default settings in Rhino are not suitable for generating quality meshes for *Examine*^{3D}. Change the project settings by selecting File → Properties. Select "Units" from the left pane of the window. Set Absolute tolerance to 0.0001 units, Relative tolerance to 0.001 percent, and Angle tolerance to 0.001 degrees. Hit OK to exit the properties window.

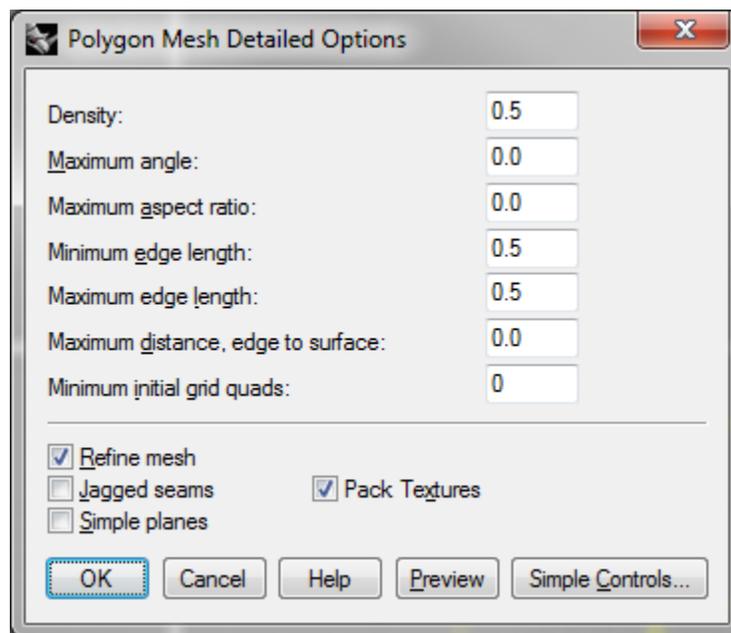
You are now ready to create your model. Creating 3D Geometry in Rhino is easily accomplished through three general methods:

- Create a 2D curve and then extrude the curve. The commands for the 2D curves are under the "Curve" menu. The extrusion command is: Solid → Extrude Planar Curve. Note that you should toggle the "Cap" option to YES. For more complex geometries, you may find it useful to take advantage of the Loft, Patch, and Join Commands. The Loft and Patch commands are found in the "Surface" menu. The Join command is found in the "Edit" menu.
- Create a 3D shape using the commands in the "Solid" menu.

- You can cut and intersect different solids as you please using the appropriate Boolean command, but make sure that the solids are unified. For example, if you wish to have two cylinders intersecting each other, you would first create the two cylinders as separate solids so that they overlap each other. You then use the Boolean command “union” to join/unify them into one cross shaped solid. The Boolean commands are found at the bottom of the “Solids” menu.

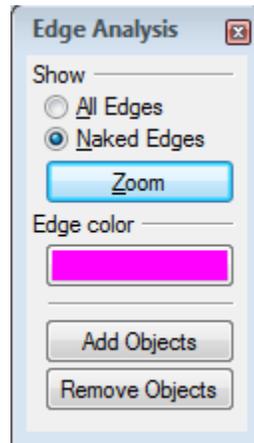
Step 2: Create Surface Mesh

Once your structure is complete, it must be meshed. This is accomplished by selecting Mesh → From NURBS Object and selecting your structure. In the dialog that appears select “Detailed Controls.” Set Density to 0.5 and Maximum and Minimum edge length to 0.5, ensuring all other options are zero. These values should work well for most structures. If you feel your mesh needs refinement, adjust the Maximum and Minimum edge lengths and use the “preview” command until you achieve a more suitable mesh. Select OK when you are satisfied with your mesh.



Before you continue, it’s best to ensure that the mesh is valid. Naked edges (mesh faces adjacent to holes) are acceptable in Rhino but will produce “leaky geometry” errors in *Examine^{3D}*. In the command window, type in “CheckMesh” and evaluate the dialog that appears. If the check informs you that the mesh has naked edges, there is some repair work to be done! In the

command window, enter “ShowEdges” and select your structure. Hit Enter and select “Naked Edges” in the Edge Analysis window.

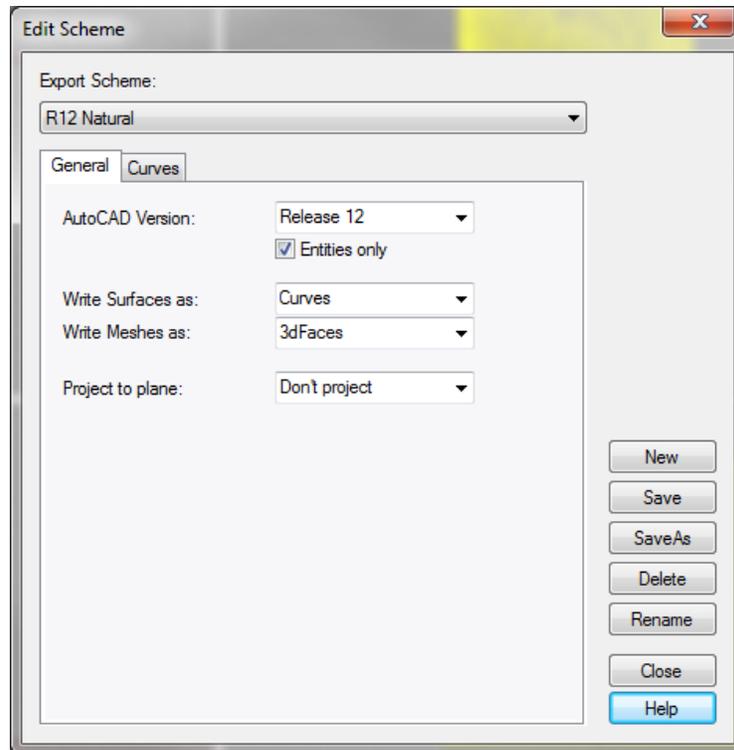


Select Mesh → Mesh Repair Tools → Fill Hole. Select any naked edges to eliminate them. To check that your mesh is non-leaky, select zoom on the Edge Analysis window. If no lines are highlighted, you have successfully eliminated your leaks.

In certain cases the hole in your structure might be too small to use the “Fill Hole” command directly. When this is the case, it is best to delete elements adjacent to the gap and fill in the new, bigger hole. Select Mesh → Mesh Edit Tools → Delete Mesh Faces and pick out faces adjacent to any naked edges. It is best to select elements exclusively from only one side of a naked edge to ensure that your intended geometry is unaltered. Fill in the newly created gaps using the “Fill Hole” command.

Step 3: Export Structure as .DXF

With your structure meshed correctly, it is time to prepare the file for import into *Examine^{3D}*. To accomplish this you must first export your file from Rhino as a .dxf. Select File → Save As and change the file type to “AutoCad drawing exchange file – (*.dxf).” Enter a file name and select “Save.” In the dialog that appears select “Edit Schemes.” Select the “R12 Natural” scheme in the menu at the top of the window and make the following changes: set “AutoCad Version” to “Release 12”, toggle the “entities only” option ON, and set “Write Meshes as” to “3dFaces.”

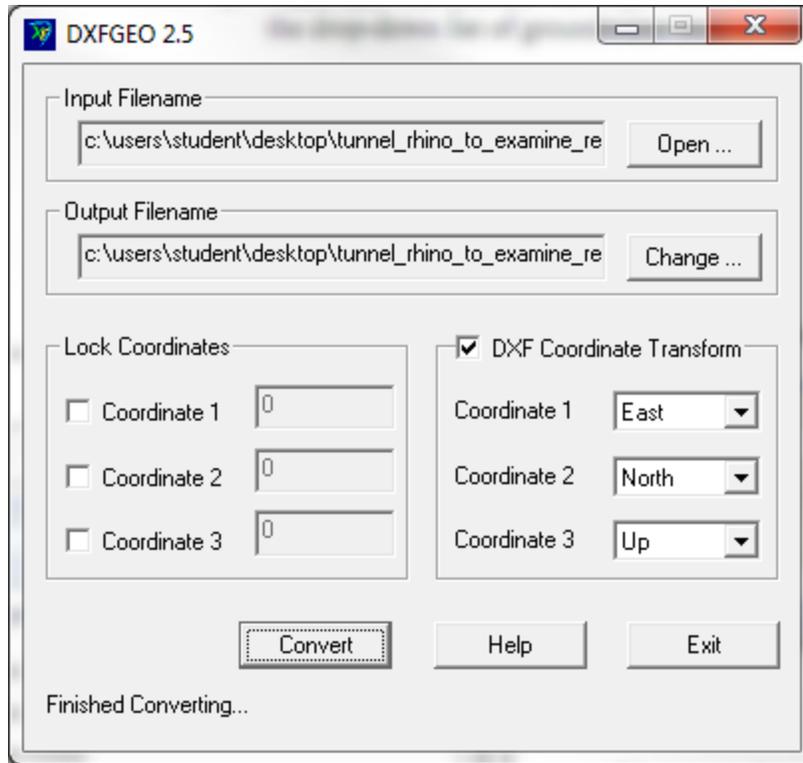


Save this new export scheme by selecting “SaveAs.” Click Yes in the dialog that appears and enter “Examine3D” as the scheme name. Select OK and close the “Edit Scheme” dialog. Select “Examine3D” in the “Export AutoCAD File” dialog and hit OK. Your file now exists as a .dxf!

Step 4: Convert .DXF to .GEO Format

Your new .dxf file can be easily converted to .geo format for import to *Examine^{3D}* using the DXFGEO utility conveniently located in your Examine 3D folder (Program Files → Rocscience → Examine 3D → DxfGeo.exe).

In the DXFGEO window, open the .dxf file you created. Toggle “DXF Coordinate Transform” ON and change the coordinates as follows: set Coordinate 1 to “East,” set Coordinate 2 to “North,” and set Coordinate 3 to “Up.” Select Convert to create your .geo file.



Step 5: Import into *Examine*^{3D}

You are now ready to import your model into *Examine*^{3D}. Open the Modeler in *Examine*^{3D} and select File → Append File. Open the .geo file you created and your model will appear. You're not done just yet however; the quality of the mesh must be checked to ensure it is suitable for analysis.

Select Toolbox → Object Check. Hit Enter to proceed through the check and take note of whether the geometry is leaky. If the geometry is NOT LEAKY, congratulations! You have successfully imported your model into *Examine*^{3D}!

If the geometry is LEAKY, return to Rhino and repair your model. Use the “CheckMesh” command and try to resolve any listed errors using the methods provided in the “CheckMesh” help window.

Examine^{3D} / Rhino Tutorial

For a detailed step-by-step tutorial on how to create a model in Rhino and import into *Examine*^{3D} [click here](#).