

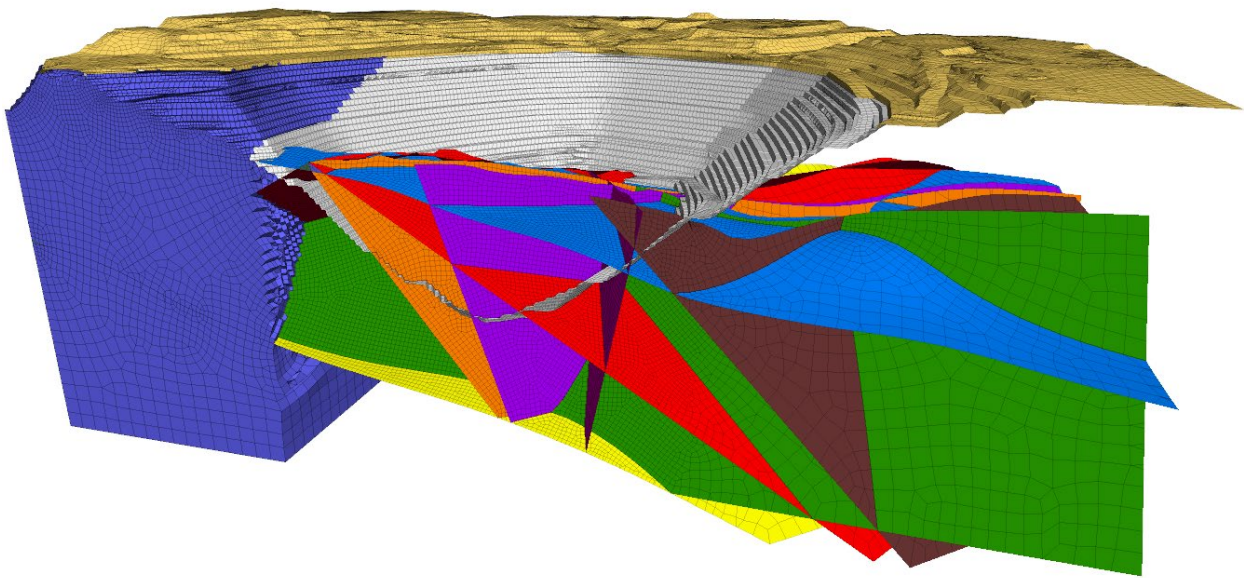
**Griddle™**

Advanced Grid Generation  
for Engineers and Scientists

---

***Griddle 2.0***  
**User Manual & Tutorial Examples**

---



[User Manual - page 2](#)

[Tutorial Examples - page 58](#)



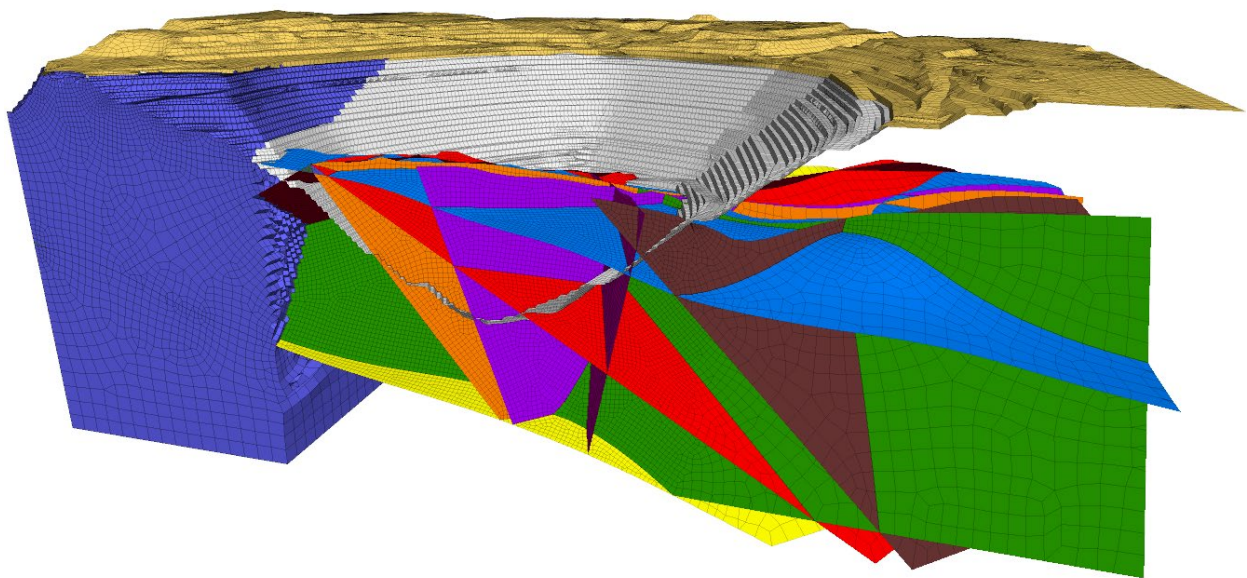
**Griddle™**

# Advanced Grid Generation for Engineers and Scientists

---

## *Griddle 2.0* User Manual

---



©2021

Itasca Consulting Group Inc.  
111 Third Avenue South, Suite 450  
Minneapolis, Minnesota 55401 USA

phone: (1) 612-371-4711  
fax: (1) 612-371-4717  
e-mail: [griddle@itascacg.com](mailto:griddle@itascacg.com)  
web: [www.itascacg.com](http://www.itascacg.com)

## Contents

<b>Contents</b>	<b>2</b>
<b>Introduction</b>	<b>3</b>
<b>Quick Start for <i>Griddle 2.0</i> and <i>Rhino</i></b>	<b>4</b>
<i>First Time Use After Installation</i>	4
<i>Griddle 2.0 Components</i>	6
<i>Griddle 2.0 Licensing</i>	6
<i>Updating Griddle 2.0</i>	8
<b>Brief Introduction to the <i>Rhino 6 and 7</i> Workspace</b>	<b>9</b>
<b>Structured and Unstructured Meshes</b>	<b>13</b>
<b>Using <i>Griddle</i> Tools for Structured Meshing - <i>BlockRanger</i></b>	<b>15</b>
<b>Using <i>Griddle</i> Tools for Unstructured Meshing</b>	<b>19</b>
<i>GInt – Surface Mesh Intersector</i>	20
<i>GSurf – Surface Remesher</i>	23
<i>GVol – Unstructured Volume Mesher</i>	29
<b><i>Griddle</i> Utilities for Working with Surface Meshes</b>	<b>35</b>
<i>GHeal – Tools for Surface Mesh Repair</i>	35
<i>GExtract – Tools for Extraction of Surface Meshes</i>	39
<i>GExtend – Tools for Surface Mesh Extension</i>	44
<i>GExtrude – Tools for Surface Mesh Extrusion</i>	47
<i>Joining Non-Manifold Surfaces</i>	49
<i>Colorizing Objects</i>	50
<b>Group Names Assignment</b>	<b>51</b>
<b>Importing <i>Griddle</i> Volume Meshes</b>	<b>55</b>
<b>References</b>	<b>56</b>

## Introduction

This document describes the use of *Griddle* grid generation tools in conjunction with the *Rhino* CAD system.

*Griddle* offers engineers and scientists both automatic and interactive grid generation capabilities that cover a wide range of volume grid generation needs as well as operations with surface meshes. *Griddle* provides new tools as well extensions to meshing tools available within *Rhino* CAD system.

Detailed tutorial examples of geomechanical applications are provided in *Griddle 2.0 Tutorial Examples* document that can be accessed via *Windows Start Menu* → *Itasca Griddle 2.0*. The files for the tutorial examples can be accessed by clicking on *Griddle 2.0 Tutorial Files* link. The link opens user writable directory `ProgramData\Itasca\Griddle200` (typically on drive "C:") which contains the documentation and the tutorial material. Users can directly work in this directory. A reserve copy of the same material can be found in the `Documentation` subfolder of *Griddle* installation location (typically in `C:\Program Files\Itasca\Griddle200`).

This manual contains images showing snapshots of *Rhino* windows and dialogs. The images were generated in *Rhino 6*, but analogous dialogs and windows can be found in other versions of *Rhino*.



*Griddle 2.0* is a suite of tools for surface- and volume-meshing within the *Rhino* CAD system. ***Griddle 2.0*** is installed as a *Rhino* plugin and is only available for the Windows versions of *Rhino 6, 7, 8*.

The *Griddle 2.0* installer automatically removes all previous versions of *Griddle* installed on the machine, including those for *Rhino 5* and later.

*Rhino* is installed separately from *Griddle* and is used for constructing model geometry.

## First Time Use After Installation

- If previous versions of *Griddle* have never been installed, nothing else needs to be done and *Griddle 2.0* can be readily used. *Griddle 2.0* automatically integrates with *Rhino 6, 7, 8*; the *Griddle* toolbar should be visible within the *Rhino* workspace:



Figure 1: The *Griddle 2.0* toolbar.

- If *Griddle 1.0* was previously installed, additional steps may be needed to remove parts of the older *Griddle* from *Rhino*<sup>1</sup>. After this one-time clean-up, *Griddle 2.0* is ready to be used. The steps below are outlined for *Rhino 6* but they can also be applied to other versions of *Rhino* to fully remove *Griddle 1.0* components.

1. Open *Rhino*. There may be two *Griddle* toolbars: an old one from v1.0 and a new one from v2.0.

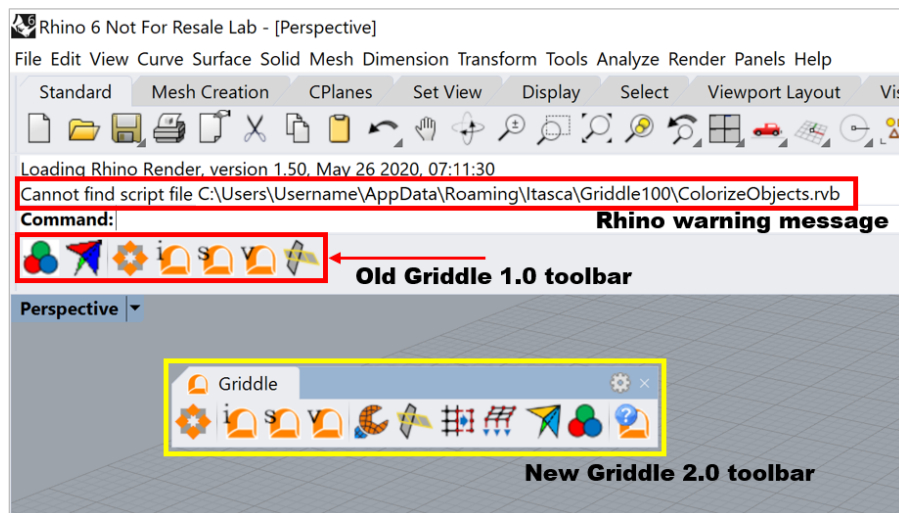


Figure 2: The *Rhino 6* window with *Griddle 1.0* and *Griddle 2.0* toolbars.

<sup>1</sup> *Griddle 1.0* used *Rhino* integration tools and required a multi-step installation process (separate installation of plugins for each command, toolbars, and scripts). *Griddle 2.0* operates differently – it installs all components at once. During the installation, *Griddle 2.0* attempts to remove its older versions from the system, but some parts may remain and should be removed manually.

2. The *Griddle 1.0* toolbar should be removed from *Rhino*. To do this, navigate to the top menu in *Rhino* and click on **Tools** → **Toolbar Layout...** (for *Rhino 5, 6, 7*) or **Window** → **Toolbars...** (for *Rhino 8*). This displays toolbars that are open/loaded in *Rhino*. Find “Griddle100”, right click on it and select **Close** (do not **Save** when offered).

**WARNING: DO NOT CLOSE “Griddle” TOOLBAR - it is used by *Griddle 2.0***  
**ONLY CLOSE “Griddle100” TOOLBAR**

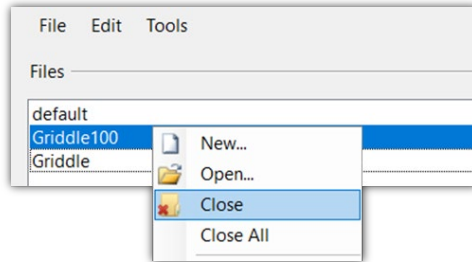













Figure 3: Closing *Griddle 1.0* toolbar from *Rhino*’s Toolbar Layout menu.

3. Figure 2 shows that *Rhino* outputs a warning “Cannot find script file ...ColorizeObjects.rvb”. This message may appear when *Rhino* starts. This occurs because the *Griddle 2.0* installer removes *Griddle 1.0* script files, however, *Rhino* is not aware of that (as these files were originally installed manually). Users may simply ignore this warning or may remove the reference to old scripts in *Rhino* to not have the warning. To do so, use the menu items **Tools** → **Options** → **RhinoScript**. Find “ColorizeObjects.rvb” in the list of scripts and delete it (click on **X**).

## Griddle 2.0 Components

This section provides a brief summary of *Griddle 2.0* components. Each component can be accessed by clicking on an icon on *Griddle* toolbar or by typing a command that is the same as the component name (if *Rhino* is set up to use language other than English, type an underscore before the command, e.g. **\_GInt**).

Table 1. *Griddle* components.

Icon	Component name, Command (English)	Description
	<b>BlockRanger</b>	Structured hexahedral volume mesher that operates on solids
	<b>GInt</b>	Surface mesh intersector for making meshes conformal
	<b>GSurf</b>	Unstructured surface remesher to create unstructured surface meshes with specified parameters
	<b>GVol</b>	Unstructured tetrahedral/hex-dominant volume mesher
	<b>GHeal</b>	A set of tools to identify and fix surface mesh problems
	<b>GExtract</b>	A set of tools to extract pieces of surface meshes based on user-specified criteria
	<b>GExtend</b>	A set of tools to expand surface meshes along a specified boundary
	<b>GExtrude</b>	A set of tools to extrude surface mesh along its boundary to create a watertight domain
	<b>NonManifoldMerge</b>	<i>Rhino</i> command to merge surface meshes and create a single non-manifold mesh
	<b>ColorizeObjects</b>	Assign random colors to objects
	<b>GriddleAbout</b>	Display information about <i>Griddle</i> and check for updates

Detailed descriptions of these components are provided in later sections of this manual.












## Griddle 2.0 Licensing

*Griddle 2.0* requires a license to run all components with full functionality. The licenses can be provided via a local USB key (desktop license), a network USB key (network license) or through online Web licensing (using Itasca Web License portal).

A license can be obtained by contacting [www.itascacg.com/sales](http://www.itascacg.com/sales).

If a license is not present, *Griddle 2.0* operates in demonstration mode. Demonstration mode limits *Griddle* functionality as described in Table 2.

Table 2. *Griddle 2.0* demonstration mode limits.

Icon	Component	Limitations in demonstration mode
	<b>BlockRanger</b>	Saves output volume mesh in VRML format only
	<b>GInt</b>	<ul style="list-style-type: none"> <li>• Functionality to keep meshes separate (<i>OutputMesh=Separate</i>) is not available (all meshes will be merged in the output)</li> <li>• <i>SplitIntersections</i> option is not available</li> </ul>
	<b>GSurf</b>	<ul style="list-style-type: none"> <li>• Number of elements in the output mesh is limited to 5000</li> <li>• Functionality to keep meshes separate (<i>OutputMesh=Separate</i>) is not available (all meshes will be merged in the output)</li> <li>• Element/mesh quality information is not provided.</li> </ul>
	<b>GVol</b>	<ul style="list-style-type: none"> <li>• Number of elements in the output mesh is limited to 10000</li> <li>• Element/mesh quality information is not provided</li> </ul>
	<b>GHeal</b>	Automatic mesh repair, <i>AutomaticHeal</i> , is not available
	<b>GExtract</b>	Only separation of a single surface ( <i>SingleSurface</i> ) is available
	<b>GExtend</b>	Not available in demonstration mode
	<b>GExtrude</b>	Not available in demonstration mode
	<b>NonManifoldMerge</b>	Fully available in demonstration mode
	<b>ColorizeObjects</b>	Fully available in demonstration mode
	<b>GriddleAbout</b>	Fully available in demonstration mode

Note that **BlockRanger** is now integral part of *Griddle 2.0* and therefore it requires a *Griddle 2.0* license. Starting with *Griddle 2.0*, **BlockRanger** will not accept licenses from *FLAC3D* or *3DEC*.

## Using *Griddle* network license

If a network key is purchased, follow these steps to install *Run-time Environment* on the license server:


1. Do not connect the network USB key to the server machine. Depending on the type of the key obtained, install on the server:

- “*Sentinel Protection Installer 7.7.0*” (or any newer version) for Sentinel SuperPro or similar keys. Make sure that ports 6001 and 6002 are open for UDP and TCP connections.
- “*HASP\_Setup*” for Sentinel HL (green) keys. Make sure that port 1947 is open for UDP and TCP connections.

Both utilities are provided within *Griddle* installation directory in `Tools\Runtime`; they both can be installed on the same machine. The utilities can also be downloaded from <https://www.itascacg.com/software/support/utilities> as part of *ICG Tools* package (navigate to `Runtime` folder).

2. Connect the *Griddle* network USB key to the server.
3. Start *Rhino* with *Griddle* on the user machine (make sure it has stable connection to the license server) and follow the steps described in the next section.


## Changing *Griddle* license location

To change the type of *Griddle* license to be used or location, click on the  icon on *Griddle* toolbar or type the **\_GriddleAbout** command and then click on *Show or Change Griddle License* button. A **License Location** dialog will open (it may take a moment to open as it tests connections to the existing licenses). Users can select desired license type in the dialog, test connection to the license and list license information.

If using *Griddle* network license, enter the IP address of the license server or type `localhost` to use local machine as the server.

## Updating *Griddle 2.0*

*Griddle 2.0* is continuously being improved. Users may expect periodic updates. *Griddle* automatically checks for available updates each time *Rhino* is closed. If updated version is available, a message box will appear offering users to navigate to the download page.

Users may also check for updates manually by clicking on the  icon on *Griddle* toolbar or typing the **\_GriddleAbout** command. Besides displaying technical information about *Griddle* and license information, the **Griddle About** dialog notifies users when updates are available.

Users may also check for updates by navigating to the *Griddle* update page:

[http://www.itascacg.com/ftp\\_pub/griddle/v200/griddle200.html](http://www.itascacg.com/ftp_pub/griddle/v200/griddle200.html)


The installer for *Griddle* updates removes previous versions prior to installing the update.

## Brief Introduction to the *Rhino 6 (or later)* Workspace

### The *Rhino* Command Area, Command Prompt, and Commands

All *Rhino* operations are performed by commands. The command area and command prompt are shown in Figure 4.

The command area is the field (generally on top) where *Rhino* displays system and command output information. Below the command area is the command prompt, where users can type *Rhino* or *Griddle* commands. The location of the command area can be moved by dragging.

When a user clicks on any icon (including icons from the *Griddle* toolbar) or a menu item, a corresponding command is automatically issued and placed into the command prompt. For example, clicking on  from the *Griddle* toolbar issues the command **\_GSurf**, as shown in Figure 4.

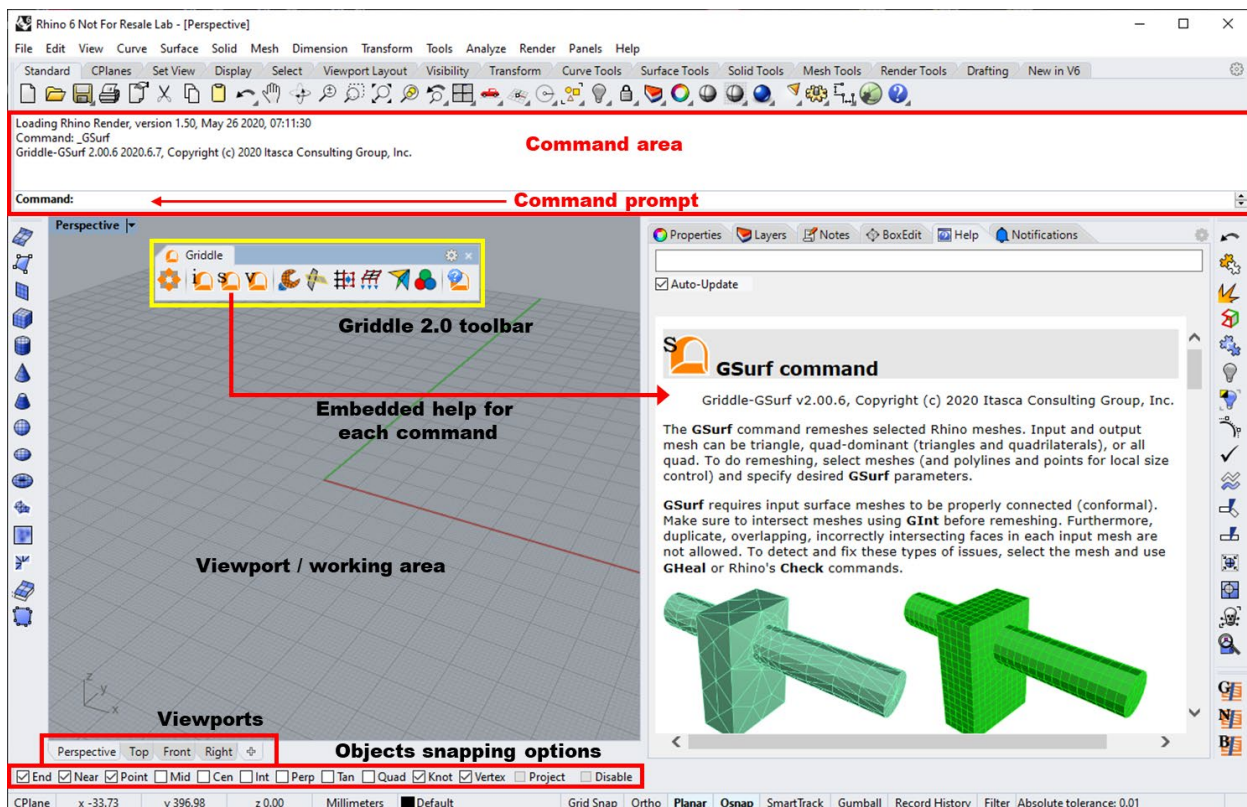



Figure 4: The *Rhino* window featuring the *Griddle* toolbar and the *Help* pane.

Another example is the command **\_CommandHelp**, which opens the *Help* pane. If the Auto-Update checkbox is selected in the *Help* pane, it will display information about currently typed command, including *Griddle* commands, as shown in Figure 4. The same *Help* pane can be opened using **Help** → **Command Help** on the menu. The *Help* pane can be dragged (by grabbing the tab) and placed into any convenient location.



Note that if an underscore is placed before the command name (e.g. **\_Shade**), *Rhino* assumes that the user refers to the English command name (i.e., **Shade**). Therefore, regardless of the language in which *Rhino* is set up, **\_Shade** creates a shaded image whereas the command **Shade** will only be understood if the installation language is English.

Throughout this manual, *Griddle* and *Rhino* functions are mostly referred through commands that are entered or issued in the *Rhino* command prompt. Sometimes *Rhino* functions are referred to by their icon name or through a menu item. The name of an icon is displayed in a tooltip that pops up when user hovers the mouse over the icon. Often, *Rhino* icons have two related functions depending on whether the user clicks the left or right mouse button (e.g., the commands **\_PointsOn** and **\_PointsOff** corresponding to the *Rhino* icon ).

## Viewports, Rotating, and Panning Views

Viewports are the part of the *Rhino* workspace where the user manipulates models. There are four standard viewports: *Perspective*, *Top*, *Front*, and *Right*, as shown in Figure 4. In the *Perspective* viewport, which presents a model in 3D perspective view, the user can rotate the view (right-click and hold) or pan the view using **Shift** and the right mouse button held down. The other viewports provide 2D views that only allow panning. The user may display several viewports simultaneously and may even create custom viewports.

## Object Snapping Options

Two types of snapping are available in *Rhino*: **Grid Snap** and **Osnap**. When **Grid Snap** is enabled, moving the mouse while it controls an object (e.g., dragging or extending an object) will cause the mouse pointer to snap to discrete positions in space corresponding to the X, Y, Z position of grid nodes.

To be able to snap lines, polylines, corners of objects, etc. to existing objects, **Osnap** must be active. Click on the **Osnap** at the bottom of the *Rhino* window to activate it. Next, specify which particular point(s) of an existing object may be snapped to by checking any of the words: **End**, **Near**, **Point**, **Mid**, etc. that appear at the bottom the *Rhino* window, as shown in Figure 4.

Both snapping options can be active simultaneously.

## Units and Tolerances

*Rhino* allows users to specify units when creating a new project. Units determine what is displayed when measuring distance, location, or angles. More importantly, units and object sizes are related to tolerances used by *Rhino*. By default, tolerance is determined based on model units. However, users can specify custom values (see Figure 5).

Tolerances are very important when intersecting or doing Boolean and other operations with NURBS or SubD surfaces or BRep<sup>2</sup> objects or when applying *Rhino* commands to meshes (note that *Griddle's* **GInt**

---

<sup>2</sup> NURBS – non-uniform rational basis spline, SubD surfaces are high-precision Catmull-Clark subdivision surfaces (only in *Rhino 7* and later), BRep – boundary representation;

tool has its own user-specified tolerance). Because these types of operations often do not provide exact (analytical) results but use approximate calculations or representations of objects, using proper tolerances ensures that operations are done as desired and in reasonable timespans.

In all cases, make sure that the model is never too far away from the origin and, if it is, **\_Move** it closer to the origin. Moving objects closer to the origin improves accuracy of operations and helps with the graphics both in *Rhino* and in the analysis software.

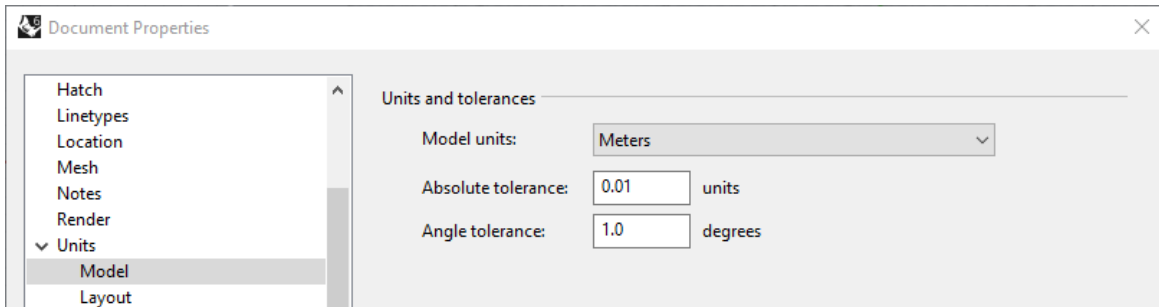


Figure 5: The *Rhino* window to specify model units and tolerances.

When starting a new project, use the appropriate *Rhino* template to specify whether the model will be using “Large Objects in Meters”, “Small objects in feet”, etc. After that create a model or import DXF, STL files, or even other existing 3dm files into the new project. This technique controls the tolerance (instead of using an inherited tolerance when starting from an imported file).

More information on tolerance can be found in the References section (see: Understanding *Rhino* tolerances).

## Orthogonal Restriction of Mouse Movement

Mouse movement can be restricted permanently to the X, Y and Z directions by clicking the word **Ortho** that appears at the bottom of the graphic window — or by pressing **F8**. Mouse movement may be temporarily restricted by holding down **Shift** while moving the mouse.

## Thicker lines

The default representation of curves (1 pixel wide) may be too thin and difficult to see in a working area or when a *Rhino* window is projected on a screen. Figure 6 shows, at left, the default representation. The following procedure shows how to draw curves 3 pixels-wide whenever **Shaded View** is selected (Figure 6, right).

Within *Rhino*, Select the **Tools** → **Options** menus item. In the left pane of the **Rhino Options** dialog, select **View** → **Display Modes** → **Shaded** → **Objects** → **Curves**. In the *Curve Settings* pane, to the right, change the default setting of *Curve width* from 1 to 3 pixels and click **OK** (Figure 7). These changes are saved with the project.

---

NURBS surfaces – surfaces represented by non-uniform rational basis spline; SubD surfaces (only in *Rhino* 7 and later) represented by generalized higher order splines for extra smoothness (Catmull–Clark subdivision surface); BRep objects – objects represented by its boundaries. All these objects have (semi-)analytical description.

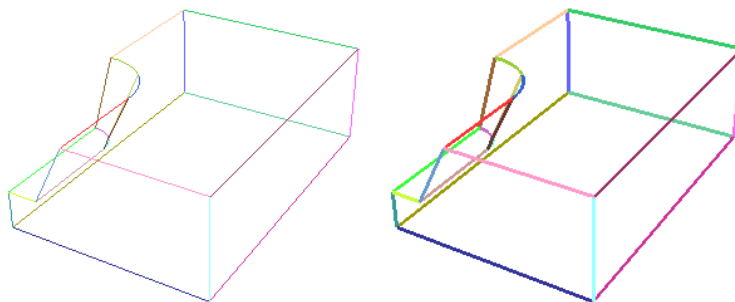


Figure 6: A regular (left) and “thick line” (right) representation.

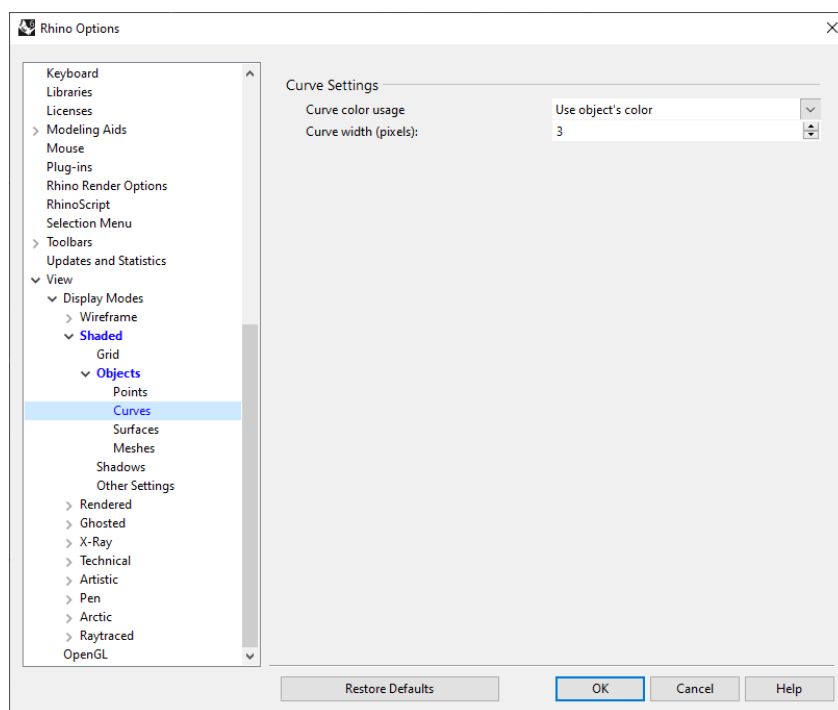


Figure 7: Setting line width to 3 pixels in Shaded View.

## Hiding the Background Grid

Hide or show the background grid in an active viewport by pressing **F7**. To hide the grid in all viewports, go to **Tools** → **Options**. In the left pane of the **Rhino Options** dialog, click on **Document Properties** → **Grid**, and in the Grid properties section, uncheck *Show grid lines* and *Show grid axes*.

## Avoiding Accidental Object Dragging

It is easy to inadvertently drag a highlighted object with the left mouse button. To avoid this, open menu item **Tools** → **Options**. In the left pane of the **Rhino Options** dialog, click on **Rhino Options** → **Mouse**, and in the *Click and drag* section, set the *Object drag threshold* to 100 pixels. Now, dragging an object requires a minimum of 100 pixels of mouse movement before it takes effect.

A number of other customizations and helpful tips/information about *Rhino* can be found in the *Rhino* documentation (press **F1** or navigate to **Help** menu).

## Structured and Unstructured Meshes

*Griddle* creates structured volume meshes using **BlockRanger** and unstructured volume meshes using **GVol**. **BlockRanger** operates on solids<sup>3</sup> represented by BRep or NURBS objects. **BlockRanger** does not operate on closed SubD objects. **GVol** operates on sets of conformal surface meshes (structured or unstructured) which compose watertight (closed volume) domains. After executing **BlockRanger** or **GVol**, output file(s) with volume mesh data are created. The output contains information about nodes, faces, elements, node-face connectivity, node-element connectivity, and element and surface groups<sup>4</sup>. A mesh from such output can be imported into numerical modeling software (e.g. *FLAC3D*, *3DEC*, *ABAQUS*, etc.)

Structured meshes are identified by regular connectivity and typically have well-shaped elements (zones). Simple examples of structured meshes are (see examples in Figure 8, Figure 10, and Figure 12):

- a quadrilateral mesh in 2D where each internal node is joined to 4 neighboring quadrilaterals, forming a regular array of elements, and
- a structured 3D hexahedral grid that has each internal node connected to 8 elements.

An unstructured mesh is identified by irregular connectivity. Surface unstructured meshes typically employ triangles and quadrilaterals, while an unstructured 3D volume mesh may contain tetrahedra, pyramids, prisms, hexahedra, and even more complex elements. *Griddle*'s components **GSurf** and **GVol** build triangular, quad-dominant, and pure quadrilateral surface meshes and tetrahedral and hex-dominant unstructured volume meshes, correspondingly.

Using an unstructured mesher typically provides the advantage of generating meshes for geometry of any complexity. It is typically much faster than the operations required to build a similar model with a mapped structured mesher (for cases when it is possible to use a structured mesher).

Two examples below (Figure 8, Figure 9) illustrate the difference between structured and unstructured volume meshes created for the same initial geometry. The model includes a layered domain with several parallel curving tunnels that are intersected by another tunnel.

Figure 8 shows a fully structured volume mesh that was created by decomposing the initial geometry into hexahedral, tetrahedral, and prism-like solids within *Rhino*. After such decomposition, the solids were meshed with **BlockRanger**. Preparing solids from the initial geometry in *Rhino* takes a considerable amount of time and effort. However, the resulting mesh is a high-quality, all-hexahedral structured mesh.

Figure 9 illustrates a fully unstructured volume mesh that was generated using **GVol**. The volume mesh was created from surface meshes of tunnels and rock layers. Manual decomposition into simple primitive shapes/solids is not required in this case. Overall model creation is much faster. The resulting mesh is a good-quality hexahedral-dominant unstructured mesh.

---

<sup>3</sup> Solids are assemblies of surfaces, called polysurfaces in *Rhino*, that have a clearly defined interior and exterior.

<sup>4</sup> Element and surface groups currently are available only for output in *FLAC3D*, *3DEC* and CSV formats.

For some models, like for the one presented in Figure 8 and Figure 9, it is also possible to create hybrid (mixed) meshes, in which the interior of the tunnels is structured mesh (meshed with **BlockRanger**) and the exterior volume is unstructured mesh (meshed with **GVol**). This is possible because tunnels have regular shape, while the overall shape of the model plus tunnel surfaces is irregular. These meshes would have to be created separately and output into separate files. Subsequently, they can be combined into a single mesh within numerical modeling software.

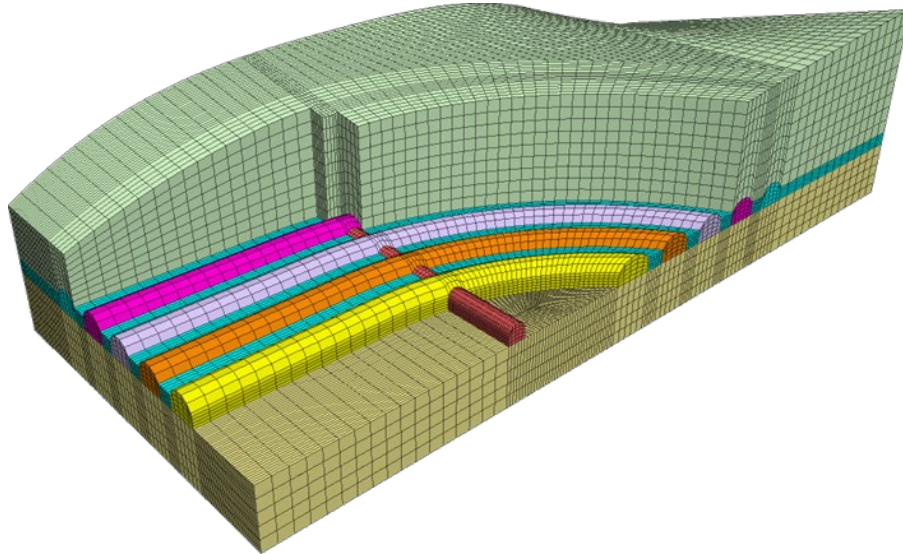


Figure 8: All hexahedral structured *FLAC3D* mesh generated by *BlockRanger*.

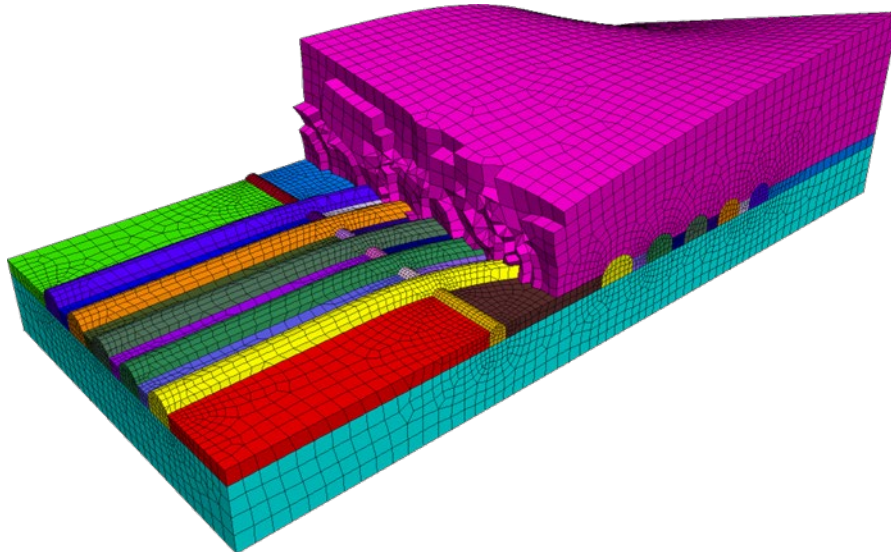


Figure 9: Hexahedral dominant unstructured *FLAC3D* mesh generated by *GVol*.

## Using *Griddle* Tools for Structured Meshing - *BlockRanger*

**BlockRanger** is an interactive all-hex mapped mesher. The *Rhino* command to run **BlockRanger** is **\_BR** and the icon to run it is provided in Table 1.

**BlockRanger** operates only on 4, 5 or 6-sided solids represented by a watertight BRep or NURBS objects (not SubD). It creates high-quality hexahedral (brick) volume mesh within such solids, which can be output in different formats for use in numerical modeling software.

Admissible **BlockRanger** solids (Figure 10) are:

- 6-sided solids (hexahedron-like) composed of 6 surfaces each bounded by 4 curves.
- 5-sided solids (prism-like) composed of 2 triangle-like surfaces bounded by 3 curves and 3 quad-like surfaces bounded by 4 curves.
- 4-sided solids (tetrahedron-like) composed of 4 triangle-like surfaces each bounded by 3 curves.

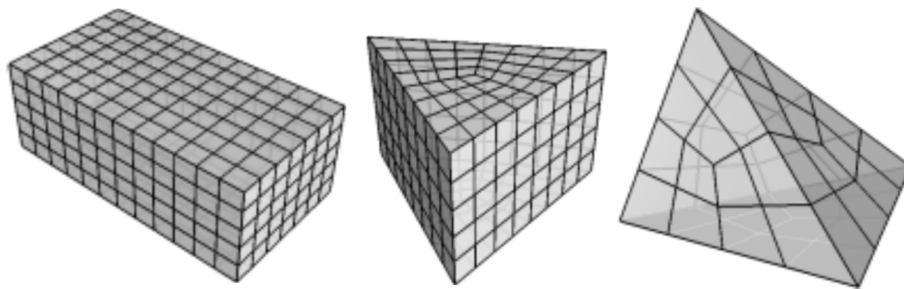


Figure 10: *Rhino* solids recognized by **BlockRanger** with example **BlockRanger** mesh patterns for each.

If the selected solids are contiguous, **BlockRanger** ensures that the resulting grid maintains grid conformity and continuity across block corners, edges, and faces so that no dangling nodes will result.

All solids successfully processed by **BlockRanger** are saved as grid files in a user-specified location or, if *AutoOutputName* option is specified – in the same location as the *Rhino* project. Solids that did not qualify or could not be successfully meshed remain highlighted onscreen. Use the **\_Invert** and **\_Hide** commands make these solids visible and correct them.



### **BlockRanger Options**

**BlockRanger** options consist of two categories: Meshing settings and Output parameters. They are outlined below. If options have predefined or default values, they are provided in square braces.

#### *Meshing Settings*

##### *MaxEdgeLength* [value ≥ 0, default = 0]

Maximum element (zone) edge length in model coordinates. If a default value of 0 is specified, this number is calculated as one tenth of the length of the longest edge in the model.

##### *MinEdgeResolution* [value > 0, default = 3]

Minimum number of elements across each *Rhino* solid edge. Its default value is 3.



**TargetNumElements** [*value > 0, default = 1*]

Use this option to reduce the maximum element aspect ratio in the grid. **BlockRanger** will initially build a grid based on the prescribed *MaxEdgeLength* and *MinEdgeResolution*. In a typical block-structured grid, as **BlockRanger** reduces the maximum aspect ratio, the number of elements (zones) increases. **BlockRanger** stops when the number of elements exceeds *TargetNumElements*.

**GenerateSurfaceMesh** [*None (default), ByModel, ByLayer, BySolid*]

This option specifies if surface mesh should be created on the boundaries of volume meshes. The surface mesh may be composed of external and internal boundary faces of the generated volume meshes:

- If *ByModel* is selected, the surface mesh will correspond to the external surfaces of all meshed solids combined (whole model).
- If *ByLayer* is selected, the surface mesh will correspond to the external surfaces of all meshed solids combined and the boundaries between solids in separate layers.
- If *BySolid* is selected, the surface mesh will correspond to the external surfaces of each solid.

Note that a single surface mesh is generated containing all boundary faces and it is placed in the default layer. The surface mesh is not saved into the output file; it is generated for use within Rhino model only (e.g., to quickly assess generated volume mesh by looking at its boundaries).

**OutputFormat** = [*FLAC3D, 3DEC\_5x, 3DEC\_7x, ABAQUS, ANSYS, NASTRAN, LS-DYNA, VRML, CSV*]

Format in which the resulting volume grid file should be saved.

*3DEC\_5x* stands for *3DEC* v5.0 or v5.2, and *3DEC\_7x* stands for *3DEC* v7.0 or later. When outputting to *3DEC* format, a rigid block is generated for every element. This output is equivalent to *BlockType* = *Rigid* in **GVol**. **BlockRanger** currently does not have capability to output to *3DEC* deformable blocks format.

When outputting to CSV format, several output files will be created: a file with general information, separate files with information about elements, faces, and nodes (a file with internal surfaces will not contain any information as **BlockRanger** does not operate on such objects).

If the license key is not present, **BlockRanger** only allows outputting generated meshes in VRML format (see Demo limits in Table 2).

**FormatType**= [*Binary, Text*]

This option appears only if user selects *OutputFormat* = *FLAC3D* or *3DEC\_7x*. The option specifies the format of the output file: in binary or text (ASCII). Saving and loading (reading) files in binary format is faster compared to the text format but binary files are not human-readable. All other *OutputFormat* choices save in text format only.

**AutoOutputName** [= *N/A (default), UserDefinedName*]

This option specifies a string that will appear in the output filename along with other identifiers. For example, for a *Rhino* project named "RhinoProject.3dm", a *FLAC3D* output in binary format will be named: "RhinoProject\_UserDefinedName\_Binary.f3grid". If user specifies a string in this option, the

**Save As** pop-up dialog will not appear and the output file will be named automatically as described above. If *AutoOutputName* is not specified (= "N/A"), a **Save As** dialog will appear, asking the user to provide the output file name and location. Note that:

- The value for this option is not saved: each time a user clicks on this option, a new string must be typed. To erase an existing string in *AutoOutputName*, click on the option and press **Enter**; this will cause a **Save As** dialog appear when executing **BlockRanger**.
- When using this option, the output file will be saved in the same location as the *Rhino* project file. If a file with same name already exists in that location it will be overwritten.

This option allows running **BlockRanger** in automatic mode without user interaction (i.e., the need to type file name and press **OK / Cancel** in the **Save As** dialog). This is particularly useful when invoking the **\_BR** command from *RhinoScript*, *PythonScript*, or other *Rhino* tools and plugins.

## Local Edge Resolution Control

**BlockRanger** controls local mesh resolution by manually setting individual edge resolution. A short example is provided below. Hands-on information can also be found in the *Griddle 2.0 Tutorial Examples, Tutorial 4: Creating a Structured Mesh with BlockRanger*. Figure 11 shows a model representing a 3D slope. The model consists of 8 solids.

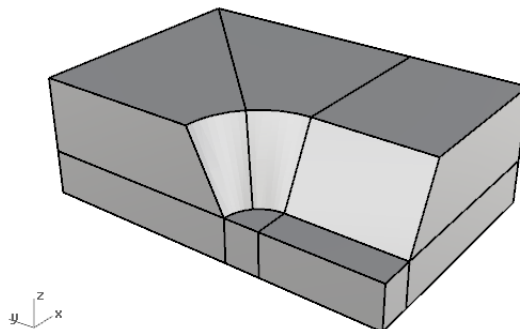


Figure 11: 3D slope example.

Running **BlockRanger** using the default parameters results in the grid as shown Figure 12.

Use the command **\_DupEdge** to duplicate an edge from a *Rhino* solid and set the number of elements for that edge. For example, in Figure 13, the highlighted edge on the right side is duplicated first. Select duplicated edge and navigate to its *Properties* pane (or press **F3**). Enter number 8 in the Name field for the selected edge, which will be the resolution of the edge. A second edge is duplicated using **\_DupEdge** (left side on Figure 13) and the number 5 is entered in its Name field.

If **BlockRanger** is run again, this time making sure to include the edges in the selection with the solids, the resulting grid will be more refined at the location of the duplicated edges (Figure 14). Set the option *GenerateSurfaceMesh=Yes* to visualize the resulting surface mesh in *Rhino*.

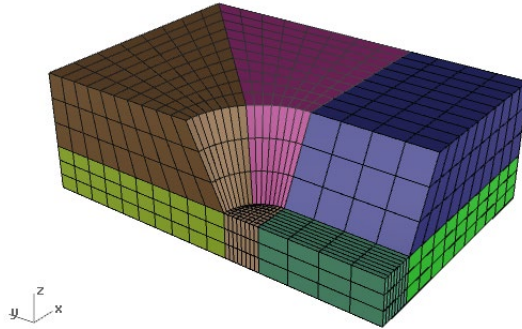


Figure 12: *BlockRanger* generated mesh with default parameter values.

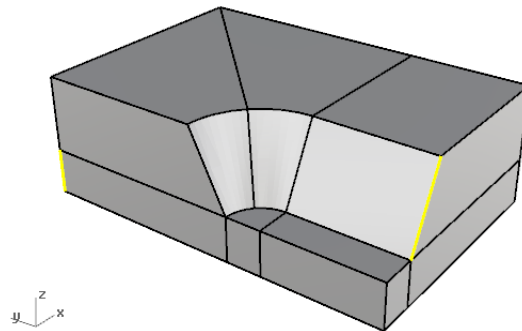


Figure 13: Duplicated edges (highlighted) with a custom number of elements specified.

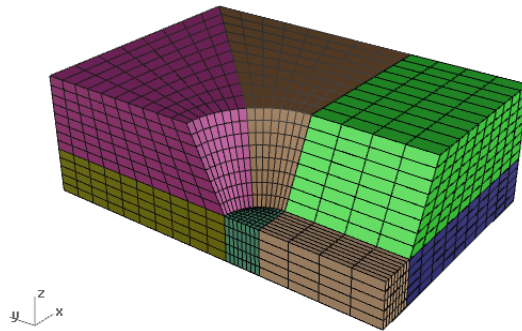


Figure 14: *BlockRanger* generated mesh with specified edge resolution.

Note that local edge resolution control takes precedence over automatically determined number of subdivisions per edge (based on the input parameters) only when it provides a larger number of elements for the edge (i.e., a finer mesh). Otherwise, the automatically determined number of elements is used.

## Using *Griddle* Tools for Unstructured Meshing

Besides the structured volume mesher, **BlockRanger**, which operates on *Rhino* solids, *Griddle* includes tools to work with surface meshes and tools to generate fully conformal unstructured volume meshes. These tools are grouped within the *Griddle* toolbar (see Figure 1, Figure 15, and Table 1).

- **GInt** – surface mesh intersector (command: **\_GInt**),
- **GSurf** – unstructured surface remesher (command **\_GSurf**),
- **GVol** – unstructured tetrahedral/hex-dominant volume mesher (command: **\_GVol**).



Figure 15: *Griddle* tools for unstructured meshing.

All three functions operate directly on *Rhino* surface meshes. The meshing workflow to generate an unstructured volume mesh is summarized in Figure 16.

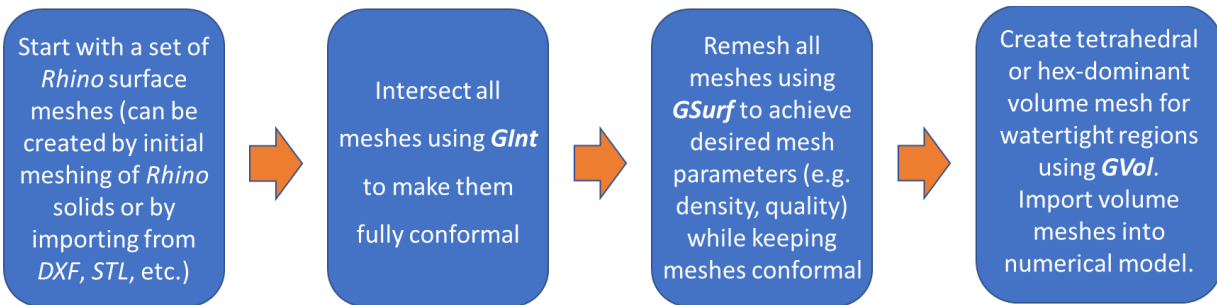


Figure 16: Typical workflow for unstructured meshing using *Griddle* tools.

*Rhino* has a rich set of surface meshing tools that allow users to create (e.g., by triangulation) and edit 3D surface meshes for objects represented by NURBS, SubD, or BRep surfaces and polysurfaces. These meshes, although good for machining and prototyping purposes, are usually not suitable for numerical computations. *Rhino*-generated (or imported) surface meshes often must be properly intersected and remeshed with **GInt** and **GSurf**, respectively. Once a desirable watertight set of surface meshes is obtained, it can be used (along with internal surface meshes) as an input to **GVol**, which fills the interior regions bounded by the surface meshes with tetrahedra or hex-dominant (hexahedra, prisms, pyramids and tetrahedral) elements for the use in numerical programs (such as *FLAC3D* or *3DEC*). The options for **GInt**, **GSurf**, and **GVol** are described below (and in the *Help* pane).

Numerous examples describing the use of **GInt**, **GSurf**, and **GVol** to create unstructured volume meshes are provided in *Griddle 2.0 Tutorial Examples*.

Griddle's tool **Glnt** is a surface mesh intersector that can be applied to improperly intersected surface meshes to make them conformal.

- In conformal surface meshes, edges and nodes of mesh elements (faces) are fully shared between all elements connected to them.
- In conformal volume meshes, element faces, edges and nodes are fully shared between all elements connected to them.

Figure 17, left, shows a simple example of two non-conformal meshes in contact (blue and yellow meshes). After application of **Glnt**, some of the initial faces are (randomly) split in triangular and quadrilateral faces to create conformal contact between meshes.

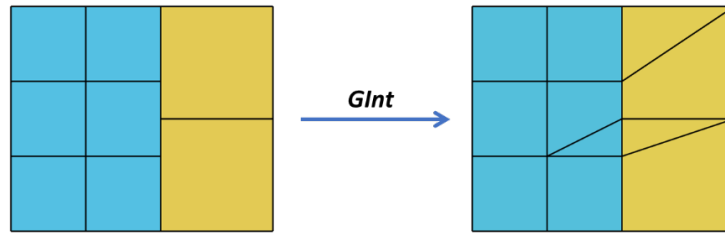


Figure 17: Example of non-conformal (left) and conformal (right) 2D meshes.

Figure 18 shows a more complex example of **Glnt** operation on two meshes intersecting in non-conformal fashion (left image). After application of **Glnt** to both meshes (right image), intersecting faces (darker green and pink) are split into numerous triangular faces to provide conformal contact.

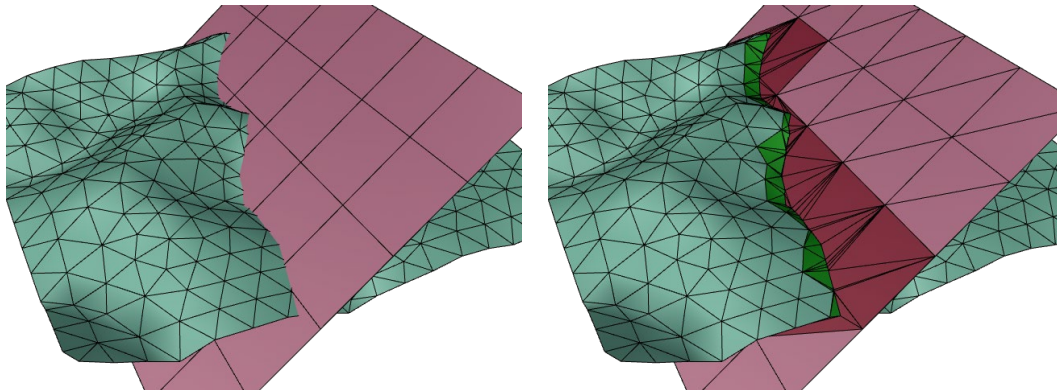


Figure 18: Example of non-conformal (left) and conformal intersecting surface meshes (right) after using **Glnt**.

**Glnt** can operate on multiple surface meshes at once to make the meshes themselves as well as intersections and contacts between them fully conformal. Use of **Glnt** before continuing with meshing is typically needed as Griddle meshing tools **GSurf** and **GVol** require conformal surface meshes at input (see the workflow in Figure 16).

Note that **Glnt** internally operates on triangular meshes only. Therefore, if an input surface mesh contains quadrilaterals, these quadrilaterals are automatically converted into triangles (by arbitrarily

inserting one of the two possible diagonal edges splitting the quad face) prior to **GInt** running (as in Figure 18). **GInt** has an option to restore some of the planar quadrilaterals (as in Figure 17) but they may not match the original ones.

If surface meshes are generated from *Rhino* NURBS, SubD, or BRep objects, it is better first to intersect such objects with *Rhino*'s Boolean functions or merge them with the **\_NonManifoldMerge** command. After that the objects should be meshed (triangulated) with the **\_Mesh** command and then remeshed with **GSurf**. Intersecting/merging of NURBS, SubD, or BRep objects is more accurate than intersecting their mesh representation as meshes always approximate the initial analytical surfaces.

## GInt Options

### *MeshType* [= *Tri* (default), *Mixed*]

This option specifies the type of the output mesh. By default, it is a triangular mesh (*Tri*), however, if user selects the *Mixed* type, the output mesh will contain triangular and quadrilateral faces (elements). Quad faces are obtained by merging neighboring planar or near-planar triangular faces (only if the angle between face normals  $\leq 0.5^\circ$  and if the ratio of the diagonals in the resulting quad is less than 5). The *Mixed* option may be useful to preserve not-intersected quad faces in planar patches of the input meshes. In rare cases using this option may cause the output mesh to lose conformity (if two triangular faces across a previously non-conformal intersection are merged into a quad). This option is equivalent to using *Rhino*'s command **\_QuadrangulateMesh**.

### *Tolerance* [value $\geq 0$ , default = 0]

Tolerance is the most important parameter of **GInt**. It is an absolute distance (in model units) used to determine whether mesh faces intersect each other. If zero tolerance is specified, only those faces are intersected which are in exact contact with each other. Specifying non-zero tolerance allows searching for nearly intersecting faces (i.e., those within the tolerance). On the other hand, a very large tolerance may lead to a distorted output mesh as nodes within the tolerance are merged together.

## AdvancedParameters

### *OutputMesh* [= *Separated* (default), *Merged*]

By specifying *Separated* value, **GInt** will try to associate output mesh patches (sub-meshes) with the distinct input meshes and will place them into the layers they were originally obtained from. Patches that could not be associated with a source mesh are placed into a layer "MISMATCHED\_PATCHES". When the value *Merged* is selected, a single output mesh will be generated and placed in the active layer.

### *SplitIntersections* [= *Yes*, *No* (default)]

If *Yes* is specified, **GInt** will split intersected faces from the rest of the mesh. If the user specifies *OutputMesh* = *Separated*, each set of split faces will be placed into a sub-layer "IntersectedFaces" of the original mesh layer. If the output mesh is merged, a new top-level layer named "IntersectedFaces" will be created, and all intersected faces will be placed in it (as a single merged mesh).



Note that if a high *Tolerance* is used, **GIInt** may not only intersect faces between separate meshes but also may change faces within each mesh internally (mostly due to node merging). In such cases, if *OutputMesh = Separated*, the algorithm for detecting intersected faces may additionally report some non-intersected faces (therefore, *Tolerance* should be as small as possible when using this option).

Extracting intersected faces allows operation on them (e.g., change name/layer, delete layer, etc.) or assigning specific mesh sizes to them (for example, to densify meshes around the intersections).

**DeleteInput** [= *Yes (default), No*]

Specifying *Yes* indicates that the original selected meshes will be deleted. If *No* is selected, the original meshes will remain intact.

**Reset** resets advanced parameters to the default values.

After **GIInt** completes the operation, it outputs a text (ASCII) log file with information about input and output meshes. The log file uses the name of *Rhino* project file (e.g. "RhinoProject.3dm") and adds "\_GIInt.log" to it (e.g. "RhinoProject\_GIInt.log"). The file is saved in the same directory where *Rhino* project is saved.

*Griddle's* tool **GSurf** is a surface remesher that is used to remesh selected surface meshes to desired element size (on average) and type (triangle, quad-dominant, or all-quad). Surface meshes are used as input into the *Griddle* volume mesher **GVol**. The volume mesher uses the surface meshes to determine solid element size and type. Input surface mesh faces will appear as faces of solid elements in the volume mesh; thus, it is important to create good quality surface meshes before sending them to **GVol**. Input to **GSurf** must be one or more conformal (properly intersected) meshes or meshes that do not intersect at all.

The example in Figure 19 shows the result of application of **GSurf** to the meshes presented in Figure 18, right. The left image shows the conformal quad-dominant surface meshes remeshed to element size<sup>5</sup> 5-10m. The right image in Figure 19 shows same meshes but remeshed with local size control specified on the intersected (and extracted) patches of the input meshes (darker green and pink mesh patches in Figure 18, right). The local size for the patches was set to 0.5m while the global element size was set to 5-10m (see sections below on how specify local element size). **GSurf** creates a smooth transition between different element sizes. The gradation can be controlled.

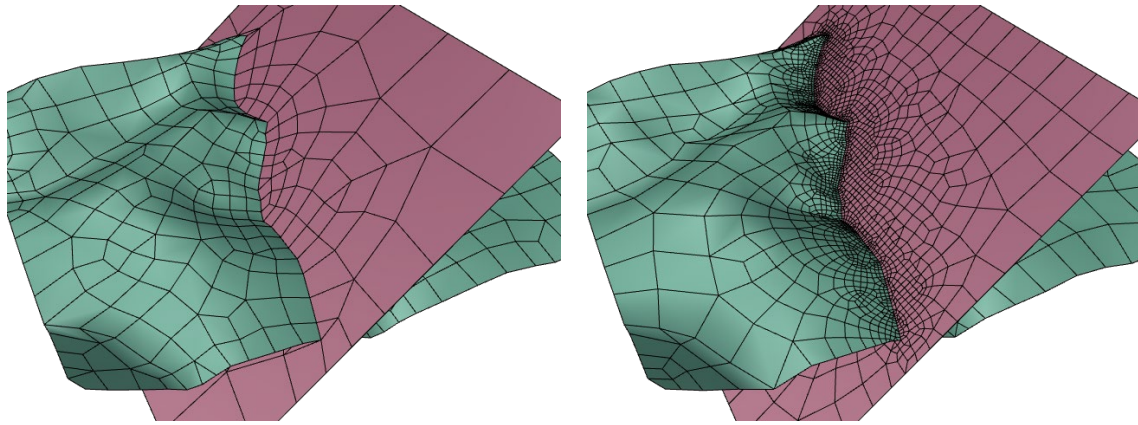


Figure 19: Examples of remeshing with **GSurf** without (left) and with local size control (right).

## GSurf Options

**Mode** [*Tri (default), QuadDom, AllQuad*]

This option allows to choose the type of the output mesh:

- *Tri* produces an all-triangle surface mesh.
- *QuadDom* produces a quad-dominant surface mesh (contains a mix of triangles and quadrilaterals).
- *AllQuad* produces a pure quadrilateral surface mesh. In certain cases, it is impossible to create an all-quad mesh and **GSurf** may show an error. In this case use *QuadDom* mode instead.

<sup>5</sup> In this case, element size is determined by the average value between element edge sizes.

### ***MinEdgeLength, MaxEdgeLength*** [value > 0]

These parameters control the output element size by setting minimum and maximum allowed edge sizes in the final surface mesh. To get uniform sizes, minimum and maximum edge size can be set to the same value. Edge size is specified in model units. It is important to set a non-zero minimum edge length to get a good quality output mesh.

Note that these values are not strictly enforced in the output mesh. However, **GSurf** attempts to achieve provided limits on edge size through multiple remeshing optimization passes (see *Optimization* below).

### ***RidgeAngle*** [value between 0° and 90°, default = 20°]

*RidgeAngle*, specified in degrees, controls the level of detail (the number of ridge lines) in the resulting mesh. The angle between mesh faces sharing an edge is termed a ridge angle (angle = 0° if faces are coplanar and 90° if faces are perpendicular). Ridge lines can be traced through the surface mesh by joining edges of faces that have ridge angles greater than the specified *RidgeAngle*. Using higher value for *RidgeAngle* results in less ridge lines (less detail) included in the final mesh. A lower *RidgeAngle* results in more detail included in the final mesh. Generally, *RidgeAngle* should be kept below 45°. The default value of 20° is a good compromise between mesh size and fidelity.

## ***AdvancedParameters***

### ***MaxGradation*** [value > 0, default = 0.1]

This parameter controls the gradation of element sizes. A value close to 0 leads to a more gradual variation of mesh size (smoother), while higher values lead to more abrupt changes in element size.

### ***Optimization*** [value between 0 and 10, default = 5]

This parameter controls the optimization of the mesh. A zero value makes **GSurf** skip the optimization step; the remeshing speed is the highest in this case, but the quality may be poor. From value 1 on, the optimizer algorithm uses several techniques to improve both the shape quality and the size quality of the elements, such as node smoothing, edge swapping, node insertion, and node removal. Level 5 is usually a good trade-off between quality and speed.

### ***QuadWeight*** [value between 0 and 1, default = 0.75]

*QuadWeight* controls the preference of quadrilaterals vs. triangles in the output mesh. This parameter is used only in quad-dominant meshing mode. If *QuadWeight* = 0, quadrilaterals are never used. If *QuadWeight* = 0.5, quadrilaterals are used only when they improve the quality of the mesh. For values between 0.5 and 1, quadrilaterals are used more even if it leads to a lesser quality mesh. If *QuadWeight* = 1, the minimum number of triangles is used. This parameter is not the ratio between the number of quads and triangles. Furthermore, there is no linear relation between *QuadWeight* and the ratio between number of quads and triangles.

### ***ShapeQuality*** [value between 0 and 1, default = 0.7]

*ShapeQuality* controls the trade-off between shape optimization and size optimization. The default value (0.7) gives a slightly stronger preference to the element shape quality over the size quality. For

example, an elongated surface mesh patch can be remeshed using few elongated triangles or quads (if *ShapeQuality* is close to 0) or it can be remeshed with many almost perfect triangles or quads (if *ShapeQuality* is close to 1).

#### ***OutputMesh*** [= *Separated* (default), *Merged*]

By specifying *OutputMesh* = *Separated*, **GSurf** will try to associate output mesh patches (sub-meshes) with the distinct input meshes and will place them into the layers they were originally obtained from. Patches that cannot be associated with a source mesh are placed into a layer “MISMATCHED\_PATCHES”. When *Merged* value is selected, a single output mesh will be generated and placed in the active layer.

#### ***DeleteInput*** [= *Yes* (default), *No*]

Specifying *Yes* in this parameter indicates that the original selected meshes will be deleted. If *No* is selected, the original meshes will remain intact.

***Reset*** resets advanced parameters to the default values.

## **Additional Edge Size Control Options**

In addition to the global values of min and max edge size specified in the parameters above (which are applied to all selected meshes), local overrides can be made to specify the desired element size for a surface mesh. Local values supersede the min and max edge sizes for the mesh.

Local element size for a mesh can be specified in one of two ways:

1. By specifying element size via a hyperlink available through object properties (Figure 20):
  - Select mesh
  - Open the *Properties* pane and click on the ellipsis (...) next to the **Hyperlink** property
  - In the **Hyperlink** dialog choose *Type: (other)* and type in the URL field “elemsize:*NumericValue*”. Click **OK**. No spaces are allowed in the URL field.
2. By setting mesh name in the *Properties* pane to a numeric value representing the required element edge size (see image below).

*Griddle* first checks if element size is specified in the *Hyperlink* field and if nothing is found, it will check the *Name* field for a numeric value. Specifying local element size via hyperlink is preferred for a few reasons: clean syntax; the mesh name can contain digits that may be confused with element size; and, if the mesh name should not be modified.

Figure 20 shows two ways to specify local element size (*Name* field or *Hyperlink*). In this example only the yellow part of the mesh is assigned local element size, after which all meshes are remeshed with **GSurf**. The result of remeshing (bottom image) shows a much finer mesh at and around the yellow part, which corresponds to the provided element size.

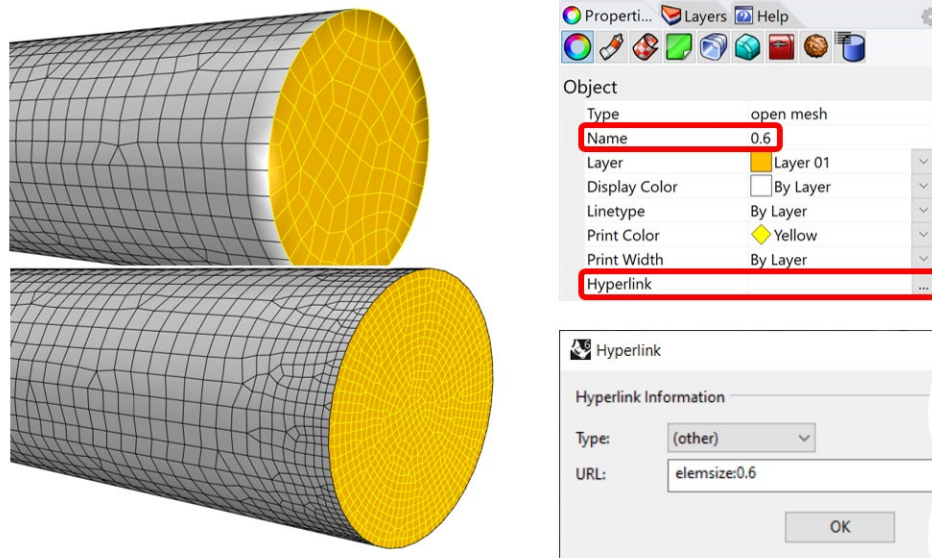


Figure 20: Two ways to specify local element size for surface meshes.

Similarly, local element size can be specified locally at a point, by creating a point (with *Rhino's* **\_Point** command) and assigning a numeric value to the point's hyperlink or name (as shown above). Such points must be coincident with the existing mesh vertices (enable vertex snapping in *Rhino* to snap the new point to a mesh vertex). If no element size is specified at a point, such a point will be treated as a hard node and will become a mesh vertex in the output mesh. Element size around this point will be determined automatically.

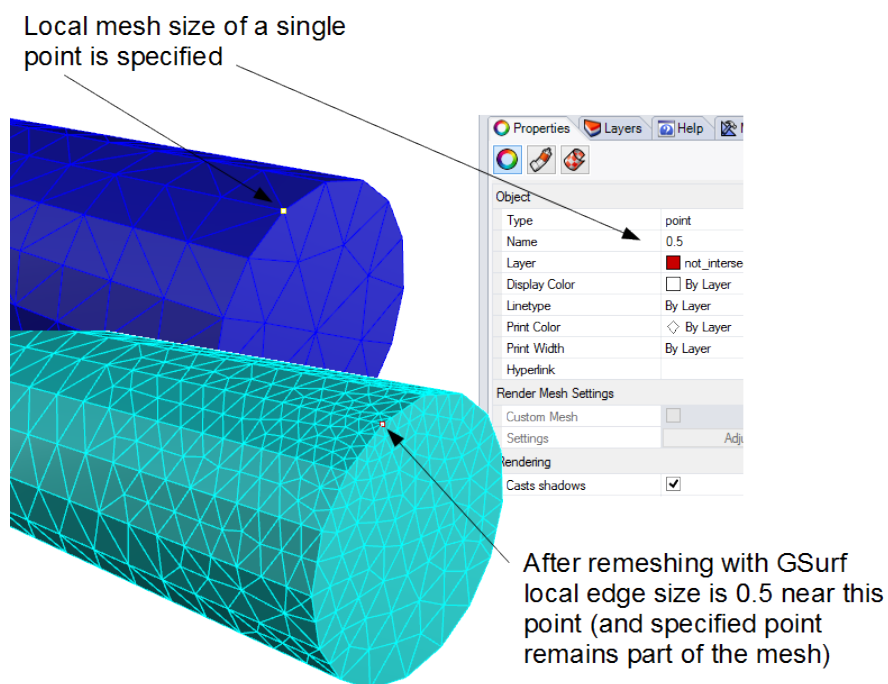


Figure 21: Specifying local element size at a point.

## Hard Edges and Hard Nodes

Hard edges are edges that are preserved in a mesh during the remeshing process. Consider an example in Figure 22 (left), which shows two conformal surface meshes. If one of the meshes (for instance, the red mesh) is changed (for example, refined) while the other is unmodified, the conformity of the meshes will be broken. **GSurf** offers a way to remesh a specific or all meshes while preserving conformity along mesh edges. This can be done by creating hard edges.

1. Use *Rhino's* **\_DupBorder** command on the red mesh, which will duplicate the mesh border (i.e., will create a curve connecting only nodes on the mesh boundary). Delete unnecessary segments of the curve (use the **\_SubCrv** command) leaving only those segments that connect to the green mesh (as shown in Figure 22, left).
2. Select both the red mesh and the remaining part of the curve (polyline) and call **GSurf** with desired meshing parameters (in this case, smaller edge size). **GSurf** will recognize the polyline as a hard edge and will remesh the red mesh, taking the hard edge into account.

The remeshed red mesh will be finer, and both meshes will still be conformal.

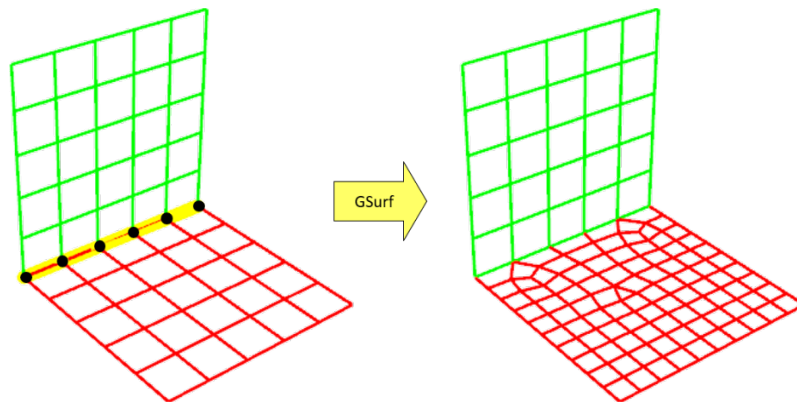


Figure 22: Specifying hard edges.

Similarly, a hard node can be defined to preserve a location of a vertex. Information about hard nodes is provided in the previous section.

## Mesh and Element Quality Information; Output Log

**GSurf** outputs information about input and output meshes, meshing parameters, and element quality information to a log file (ASCII). The element quality information is based on shape quality analysis of each element in the remeshed mesh. The following information is provided.

- **Total area:** The cumulative surface area of all elements in the remeshed mesh in model units.
- **Element min shape quality (QS) value:** Minimum normalized shape quality among all elements in the remeshed mesh. The values of shape quality range between 0 and 1. Detailed information about shape quality calculations is provided below.



- **Max error distance:** Measure of the geometric error between the input and output (remeshed) meshes. The nodes of the remeshed mesh are located exactly on the input mesh surface. However, this is typically not the case for the centroids of new elements, and new nodes may not coincide with the initial nodes. If the input mesh surface is not flat, the remeshed mesh may be some distance away from it. The `Max error distance` parameter is a measure of this maximum distance (*Hausdorff distance*).
- **Histogram QS for (All/Specific) Elements:** Provides information about distribution of shape quality (QS) values. The bins (intervals) of the histogram are of equal size.
  - **Total number of bins:** Number of bins (intervals) in the histogram. The default value is 11.
  - **Total number of counts:** Total number of counts in the histogram equal to the number of elements (all or specific type, e.g., quadrilaterals) in the output mesh.
  - **Number of larger values:** Number of hits (elements) with QS above the largest histogram bin value.
  - **Number of smaller values:** Number of hits (elements) with QS below the smallest histogram bin value.
  - **V max:** Maximum shape quality (QS) value among all values in the histogram.
  - **V mean:** Mean shape quality (QS) value among all values in the histogram.
  - **V min:** Minimum shape quality (QS) value among all values in the histogram.

Next, the information about shape quality values distribution in the histogram is provided:

```
Bin number      -- Bin boundaries --      Hits
```

Shape quality of an element QS is calculated based on the following expressions:

For triangular elements:  $Q_s = 4\sqrt{3} S / (L_{max} P)$ , where

- $S$  is the area of the triangle,
- $L_{max}$  is the length of the longest edge of the triangle,
- $P$  is the perimeter of the triangle.

For 3D quadrilateral elements:  $Q_s = Q_s^{2D} Q_w$

Here,  $Q_s^{2D}$  is 2D shape quality:  $Q_s^{2D} = 8\sqrt{2} S_{min} / (L_{max} P)$ , where

- $S_{min}$  is the minimum area of the four triangles within the quadrilateral,
- $L_{max}$  is the max length of the four sides and the two diagonals,
- $P$  is the perimeter of the quadrilateral,
- and  $Q_w$  is warp quality of the quadrilateral:  $Q_w = 1 - \frac{\text{acos}(\min[\langle n_0, n_2 \rangle, \langle n_1, n_3 \rangle])}{\pi}$ , where  $n_i$  is the normal at node  $i$  (or normal to the triangular face  $i$  of the quadrilateral).

The output log file uses the name of the *Rhino* project file (e.g., “RhinoProject.3dm”) and adds “\_GSurf.log” to it (e.g., “RhinoProject\_GSurf.log”). The log is output to the same directory where the *Rhino* project is saved.

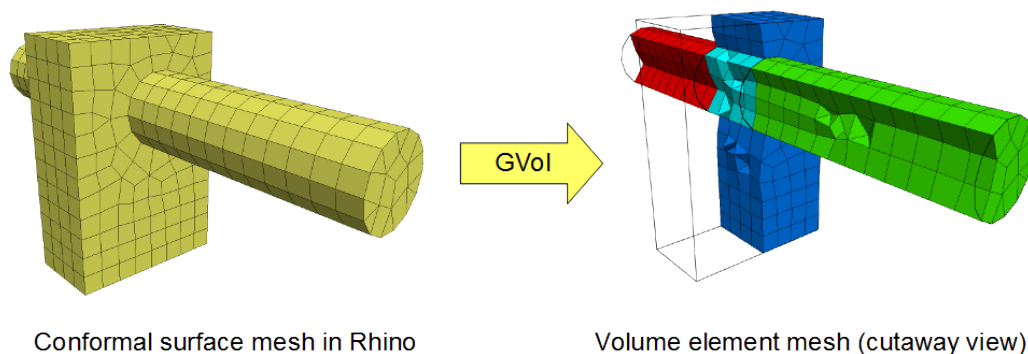
## **GVol – Unstructured Volume Mesher**

**GVol** is *Griddle*'s unstructured volume mesher, capable of creating a tetrahedral or a hex-dominant mesh using selected surface meshes as boundaries (Figure 23). Surface meshes can be composed of triangles, a mix of quadrilaterals and triangles, or all quadrilaterals. The latter two must be used for generating hex-dominant grids. Tetrahedral grids can be generated from triangle, triangle-quad, or all-quad surface meshes (quadrilaterals are arbitrarily split along one of their diagonals into triangles).

There are two important requirements for **GVol** to successfully create a volume mesh:

- a combination of selected surface meshes must form a watertight boundary surrounding the entire volume of interest, and
- all intersecting or connecting surface meshes must be conformal.

Surface meshes can also separate discrete volumes within the larger volume. Surface meshes can also “float” inside the volume of interest or be partially connected to other surface meshes. All surface mesh faces, including “floating” surface meshes inside a volume, are included as “hard faces” in the final volume mesh. This means that all input surface faces will be present as faces of elements in the resulting volume grid.



**Figure 23: Volume mesh creation from surface meshes.**

**GVol** requires input surface meshes to be properly connected. Duplicate, overlapping, or intersecting surface mesh faces are not allowed. Check the mesh for errors with *Griddle*'s **GHeal** tool and fix issues, if needed (note that remeshing may be required after fixing issues). If issues remain after using **GHeal**, consider intersecting surface meshes using **GInt** and specifying a higher tolerance and remeshing all meshes again.

If **GVol** encounters problems during meshing, traces of the problem areas will be placed into a *Rhino* layer “MESHING\_ERRORS” (Figure 24). This layer will contain *Rhino* curves and points, outlining areas of surface meshes that caused problems. Typically, these problems are caused by the presence of naked edges (holes) or non-conformal (not properly intersected) mesh faces.

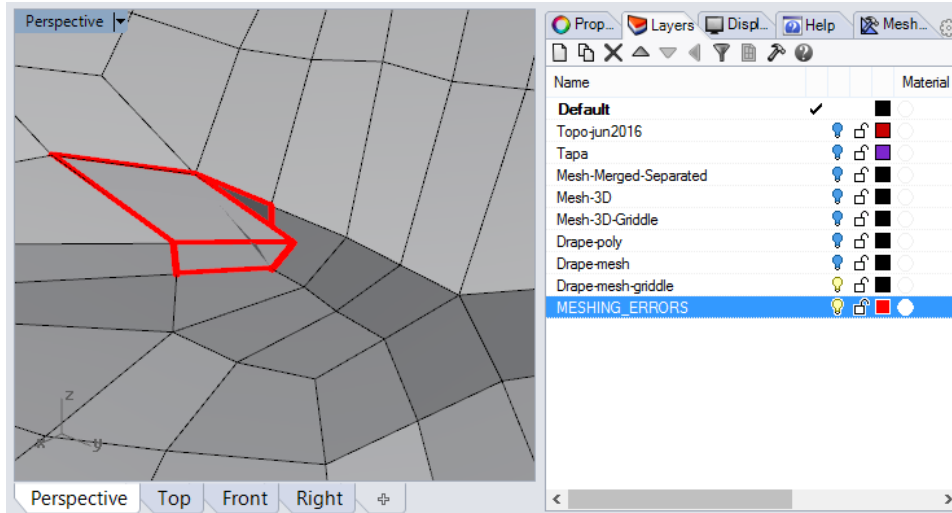


Figure 24: Traces of problem areas placed in “MESHING\_ERRORS” layer.

## GVol options

### MeshSettings

#### Mode [Tet (default), HexDom]

This option specifies the type of the output mesh:

- Tet (default) produces an all-tetrahedral volume mesh. Any quadrilaterals on surface meshes are converted to triangles in this case.
- HexDom produces a conformal hex-dominant volume mesh. Input surfaces must contain quadrilaterals for this option.

#### MaxGradation [value > 0, default = 0.5]

This parameter controls the gradation of element sizes from the size on the boundary to the preferred size defined by *TargetSize* inside the domain (far enough from the boundaries). A value close to 0 leads to a more gradual variation of size while higher values lead to more abrupt changes in element size.

#### TargetSize [value > 0, default = 0]

This parameter specifies the preferred element size inside the domain. The element size tends toward this value as elements get away from the boundaries. The default value of 0 indicates that the element size inside the domain is automatically determined based on the size of the elements on the boundary faces. Make sure that this parameter is meaningful and is not too small compared to the elements size on the boundary faces; otherwise, mesh generation may take a very long time as a large number of small elements will be generated inside the domain.

#### Optimization [value between 0 and 10, default = 5]

This parameter controls the optimization of the mesh. A zero value makes the mesher skip the optimization step. The speed is the highest, but the quality may be poor. From value 1 on, the

optimizer algorithm uses several techniques to improve both the shape quality and the size quality of the elements, such as node smoothing, edge swapping, node insertion and node removal. Level 5 is usually a good trade-off between quality and speed.

**ShapeQuality** [*value between 0 and 1, default = 0.7*]

This parameter controls the trade-off between shape optimization and size optimization. The default value (0.7) gives a slightly stronger preference to the element shape quality over the size quality. For example, an elongated volume region can be filled with few elongated elements (if *ShapeQuality* is close to 0) or it can be filled with many almost perfect elements (if *ShapeQuality* is close to 1).

**IniErrorCheck** [= *Yes, No*]

*IniErrorCheck* specifies if an error check for input surface meshes should be executed to detect potential problems with naked edges and/or clashing faces. The error check is executed before volume meshing begins, and if problems are found, a warning message will be shown. The initial error check can be turned off by setting this option to *No*. *Griddle* remembers this option even after *Rhino* is closed, therefore if *IniErrorCheck* is set to *No*, next time *Rhino* is launched, the initial error check will not run. To re-enable the check, set it to *Yes*. It is recommended to keep *IniErrorCheck* = *Yes* to be able to see warnings about input mesh problems. Changing this option does not affect other error checks during and after volume mesh generation.

**OutputFormat** [= *FLAC3D, 3DEC\_5x, 3DEC\_7x, ABAQUS, ANSYS, NASTRAN, LS-DYNA, VRML, CSV*]

*OutputFormat* specifies format in which the resulting volume grid file should be saved.

**FormatType** [= *Binary, Text*]

This option appears only if the user selects *OutputFormat* = *FLAC3D* or *3DEC\_7x* (i.e., *3DEC* v7.0 or higher). The option sets how to save the output file: in binary or text (ASCII) format. Saving and loading (reading) files in binary format is much faster compared to the text format, but binary files are not human-readable. All other *OutputFormat* choices allow saving in text format only.

**BlockType** [= *Rigid, Deformable*]

This option appears only if user selects *OutputFormat* = *3DEC\_7x* (i.e., *3DEC* v7.0 or higher). It specifies if each element of the output volume mesh should be represented as a rigid block (*BlockType* = *Rigid*) or as a deformable zone (*BlockType* = *Deformable*) in *3DEC*. In the latter case, watertight volumes in the model (closed volumes separated by surface meshes) will be represented as rigid blocks filled with deformable zones.

**AutoOutputName** [= *N/A (default), UserDefinedName*]

This option sets a string that will appear in the output filename along with other identifiers. For example, for a *Rhino* project named “RhinoProject.3dm”, a *3DEC* rigid blocks output in binary format will be named: “RhinoProject\_UserDefinedName\_Rigid\_Binary.3dgrid”). If the user specifies a string in this option, the **Save As** dialog will not appear, and the output file will be named automatically as

described above. If *AutoOutputName* is not specified (N/A), the **Save As** dialog will appear in order to obtain the output file name and location.

- Note that the value for this option is not saved: each time user clicks on this option, a new string must be typed. To erase the existing string in *AutoOutputName*, click on the option and press **Enter** without typing anything; this will cause the **Save As** dialog appear when executing **GVol**.
- The output file will be saved in the same location as the *Rhino* project file. If a file with same name already exists in that location, it will be overwritten.
- This option allows running **GVol** in automatic mode without user interaction (i.e., the need to type file name and press **OK** / **Cancel** in the **Save As** dialog). This is particularly useful when invoking the **GVol** command from *RhinoScript*, *PythonScript*, or other *Rhino* tools and plugins.

## Important Notes

If *OutputFormat* = *3DEC\_7x* and *BlockType* = *Deformable*, block joints will not be created along floating surface meshes located (fully or partially) within watertight volumes that will become rigid blocks (Figure 25). Such floating surface meshes (meshes with naked edges) do not split or define separate rigid blocks and currently cannot be represented as joints in *3DEC* (though, they can be represented as interfaces in *FLAC3D*). To check for the presence of surface meshes with naked edges, set *IniErrorCheck* = *Yes* and/or use the **\_GHeal** command.

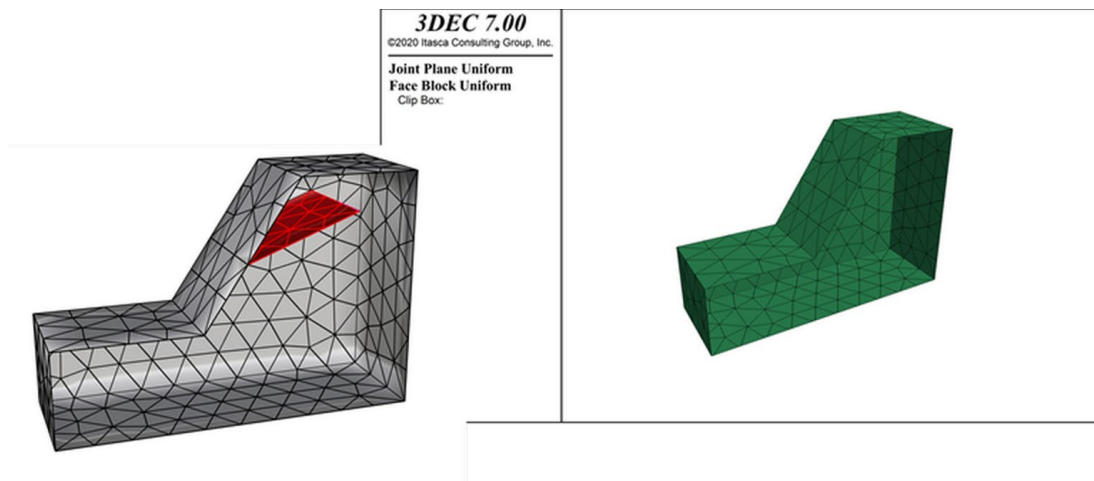


Figure 25: *Rhino* model (left) with floating surface (red) and *Joint* plot in *3DEC* representing the rigid block (right). The red surface does not become part of the rigid block (or a joint) in *3DEC*.

## Mesh and Element Quality Information; Output Log

**GVol** outputs information about input and output meshes, meshing parameters, and element quality information to a log file (ASCII). The element quality information is based on shape quality analysis of each element in the volume mesh. The following information is provided.

- **Meshed volume**: The volume of all elements in the output mesh in model units.
- **Number of meshed subdomains**: Number of closed volume (watertight) subdomains present in the meshing volume.

- `Missed (unenforced) faces`: For the case of a Hex-dominant volume mesh, **GVol** may be unable to use some constrained (boundary) quadrilaterals from the input surface meshes to generate 3D elements adjacent to them (for example, warped or very poor-quality quads). Mainly this is because it would lead to degenerate 3D elements or elements with very poor shape quality. In these cases, the constrained quadrilaterals are replaced by two triangles, which leads to generation of adjacent tetrahedrons (or sometimes pyramids). The parameter `Missed (unenforced) faces` shows how many unenforced (split) quadrilaterals are encountered. To enforce more boundary face quadrilaterals, increase the parameter `Optimization` level. Other entities that cannot be enforced, such as nonconformal faces, are also included in this parameter.
- `Element min shape quality (QS) value`: Minimum normalized shape quality among all elements in the volume mesh. Shape quality value ranges between 0 and 1. Detailed information about shape quality calculations is provided below.
- `Histogram QS for (All/Specific) Elements`: Provides information about distribution of shape quality (QS) values. The bins (intervals) of the histogram are of equal size.
  - `Total number of bins`: Number of bins (intervals) in the histogram. The default value is 11.
  - `Total number of counts`: Total number of counts in the histogram equal to the number of all/specific elements in the output mesh.
  - `Number of larger values`: Number of hits (elements) with QS above the largest histogram bin value.
  - `Number of smaller values`: Number of hits (elements) with QS below the smallest histogram bin value.
  - `V max`: Maximum shape quality (QS) value among all values in the histogram.
  - `V mean`: Mean shape quality (QS) value among all values in the histogram.
  - `V min`: Minimum shape quality (QS) value among all values in the histogram.

Next, the information about shape quality values distribution in the histogram is provided:

```
Bin number      -- Bin boundaries --      Hits
```

Shape quality of an element QS is proportional to the minimum normalized Jacobian of the element. Element shape quality typically ranges from 0 for a degenerated element to 1 for a perfect element, however negative QS values may potentially be reported if warped elements (for which min Jacobian is negative) are present in the mesh. **GVol** typically avoids creating such elements either by splitting boundary quadrilaterals (if they are close or connected to the degenerate element) or by generating several simpler topology elements (e.g., tets, pyramids) in place of the warped element.

For tetrahedral elements:  $Q_s = 6\sqrt{6} V / (L_{max} S)$ , where

- $V$  is the volume of the tetrahedron,
- $L_{max}$  is the length of the longest edge of the tetrahedron,
- $S$  is the total area of the four faces of the tetrahedron.

Note that the Jacobian of a linear tetrahedral element is equal to six times its volume.



For other 3D elements:

$$Q_s = \alpha Q_w^{min} \cdot J_{min} / L^3, \quad \text{where}$$

- $\alpha$  is the scaling factor to scale to 1 for a perfect element (different for each element type),
- $Q_w^{min}$  is the minimum warp quality of the quadrilateral for elements containing such faces (see expression for  $Q_w$  in the **GSurf** Section);  $Q_w^{min} = 1$  for triangular faces,
- $J_{min}$  is the minimum of the Jacobians calculated at all the vertices (equals to the determinant of the mixed product of 3 edge vectors at each vertex); for pyramids, the minimum Jacobian at the apex is computed as the minimum of the 4 Jacobians when taking each 3 edge vectors out of 4,
- $L$  is the square root of the sum of the square lengths of all edges (or the edges used to calculate  $J_{min}$  for pyramid).

Note that the approach to shape quality calculation in *Griddle* may differ from shape quality metrics calculation in Finite Element (e.g., *ABAQUS*, *ANSYS*, etc.) or Finite Volume (*FLAC3D*, *3DEC*, etc.) software.

The output log file uses the name of the *Rhino* project file (e.g., “RhinoProject.3dm”) and adds the following to the filename: “\_GVol” + format type (Text/Binary, if applicable) + Output Format (e.g., “.f3grid”, “.inp”) + “.log” (e.g. “RhinoProject\_GVol\_Binary.f3grid.log”). The log is output to the same directory where the *Rhino* project is saved.

## Griddle Utilities for Working with Surface Meshes



### **GHeal – Tools for Surface Mesh Repair**

**GHeal** is a *Griddle* utility that can be used to identify certain surface mesh problems and fix them. In automatic mode, **GHeal** attempts to fix such issues as: patch mesh holes, patch (mend) cracks (elongated fissures on the external boundary of a surface mesh), clean clashing faces, remove small protrusions (a small number of mesh faces attached to the main mesh in non-conformal way), split non-planar quadrilateral faces, align mesh normals, and clean (cull) redundant nodes and detached faces. Each of these operations are referred to as **GHeal** functions. In certain cases, running all these functions at once helps cleaning up and fixing a mesh. In other cases, only specific functions should be executed to clean up a mesh, as some other functions may distort the mesh or provide undesirable results. In general, **GHeal** functions (executed collectively or individually) often identify and clean mesh problems more efficiently than user efforts to fix a mesh manually or by using available *Rhino* tools.

**GHeal** also has a *Custom* operation mode which calls *Rhino*'s command **\_MeshRepair**. This command allows for more precise repairs of a mesh, e.g., patching a single hole where the user must select the boundary of the hole; **GHeal**'s *AutomaticHeal* operates on the whole mesh. It attempts to find all issues and repair parameters automatically. Refer to *Rhino* help documentation for more information about the **\_MeshRepair** command.

### **GHeal Options**

#### **ShowErrors**

This option searches selected surface meshes for naked edges and clashing faces and plots traces of such edges and faces.

**Naked edges** are face (or surface) edges that are not joined to other faces (or surfaces). In other words, only a single face (surface) shares/contains such edge. Mesh boundaries and discontinuities contain naked edges. They can also be found along disjointed faces and may not be visible (faces may appear to be connected that are not).

**Clashing faces** are pairs of faces which are partially overlapping or intersecting in a non-conformal way.

When clicking on this option, traces of naked edges are placed into the *Rhino* layer "NAKED\_EDGES" and traces of the clashing faces are placed into the layer "CLASHING\_FACES". Examining polylines placed in these layers reveals potential mesh issues.

Note that surface meshes within a watertight domain can have naked edges, for example, be partially connected to domain boundaries or "float" within the volume (also see information on **GVol**). However, the presence of naked edges on domain boundary meshes due to holes, disjointed faces, or

other discontinuities, as well as presence of clashing faces, usually indicates problems with surface meshes. Such problems must be addressed before volume meshing.

### **CustomHeal**

This option calls *Rhino*'s command **\_MeshRepair**, which checks meshes and fixes problems manually (one by one). Refer to *Rhino* help for the description of the **\_MeshRepair** command.

### **AutomaticHeal**

*AutomaticHeal* mode contains two options: *IssuesToFix* and *Parameters*. *IssuesToFix* specifies types of issues to work on and *Parameters* specify corresponding function parameters. They are described below in detail.

#### **IssuesToFix**

This option specifies types of issues (functions) that **GHeal** can operate on in *AutomaticHeal* mode. Any combination of the *issues* can be selected for **GHeal** to work on (by default all are selected). Every selected function is executed in the sequence it appears in the menu.

**All** and **None** options allow to select all or deselect all issues in the list.

#### **FillHoles**

This function fills all holes within the selected meshes (Figure 26). The function uses the parameter *MaxHoleArea*, which is the maximum area of a hole to be filled. A hole will be filled if (i) the area of the hole is smaller than *MaxHoleArea* and (ii) the area of the hole is smaller than half of the total area of the entire mesh. Remeshing may be needed after filling large number of holes.

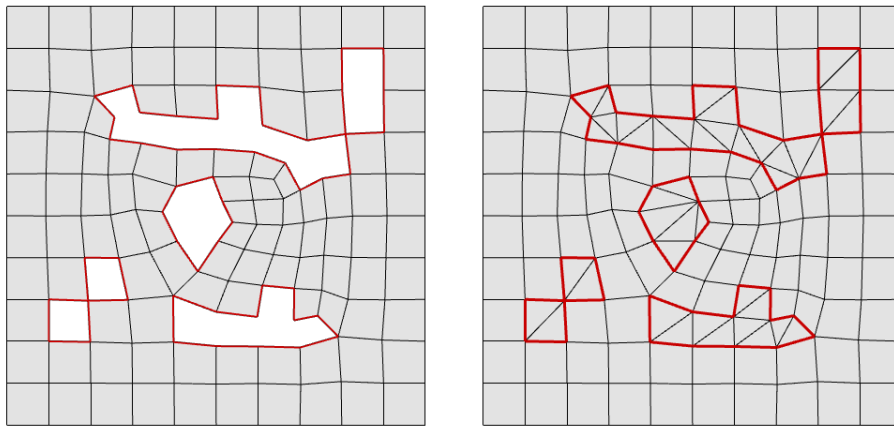


Figure 26: **GHeal**'s function **FillHoles** fixes various holes in the surface mesh. Red lines show naked edges (left) and are provided for reference only in the right image.

#### **MendCracks**

This function mends ("stitches") all cracks on the edges of the selected meshes (Figure 27). A crack differs from a hole by that it is connected to the external boundary of the mesh. The function uses the parameter *MaxCrackOpening*, which is the maximum distance between the edges of a crack.

Crack “stitching” starts at the tip of the crack and proceeds as long as (i) the opening angle at the crack tip is  $\leq 90^\circ$  and (ii) the maximum distance between the edges of the crack is smaller than *MaxCrackOpening*. Remeshing may be needed after fixing large cracks.

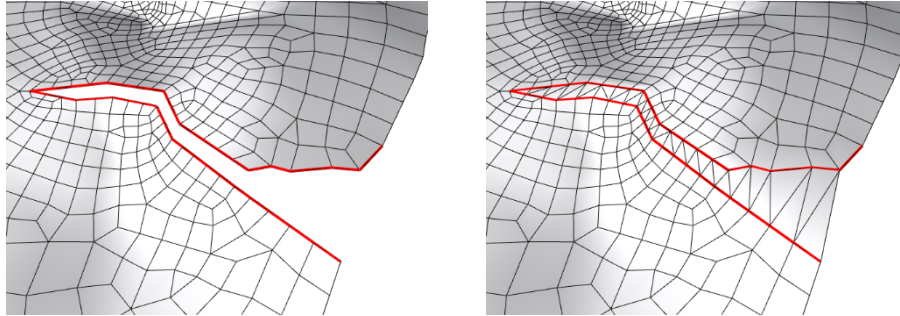


Figure 27: *GHeal*’s function *MendCracks* “stitches” a crack on the boundary of a surface mesh. Red lines indicate naked edges (left) forming the crack and are provided for reference only on the right image.

### *FixClashingFaces*

This function fixes clashing mesh faces (faces which intersect or overlap) within each selected mesh (Figure 28). *FixClashingFaces* uses logic similar to *GInt* but intersects/splits faces locally in each mesh. *FixClashingFaces* will not fix non-conformal and overlapping faces from different intersecting meshes. To properly intersect distinct meshes, use the mesh intersector *GInt*.

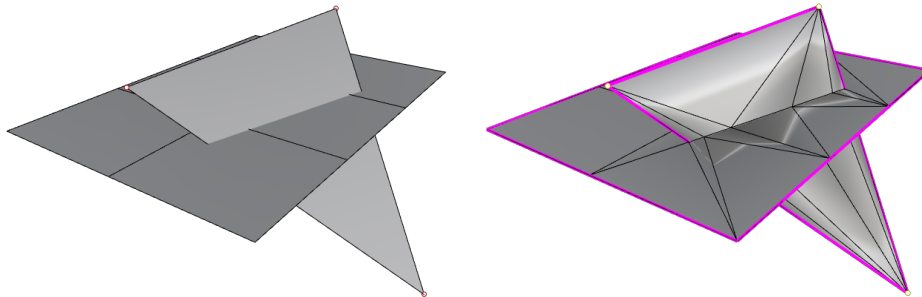


Figure 28: *GHeal*’s function *FixClashingFaces* locally intersects/splits clashing faces.

Note that in certain cases functions *FillHoles* and *MendCracks* may introduce clashing faces if newly built faces intersect with existing faces. Using *FixClashingFaces* together with these mentioned functions helps resolve this problem. Remeshing may be needed after fixing clashing faces.

### *RemoveProtrusions*

This function removes mesh faces (patches) protruding from the main surface of the mesh. An example of such protrusion is shown in Figure 29. The function uses the parameter *MaxProtrusionPerimeter*, which is the maximum perimeter of the protruded mesh patch. The protruded mesh patch will be removed if it is joined to the rest of the mesh in non-manifold manner and if (i) the total perimeter of the protruded patch is smaller than *MaxProtrusionPerimeter* or (ii) there are two or less faces in the protruded mesh patch.

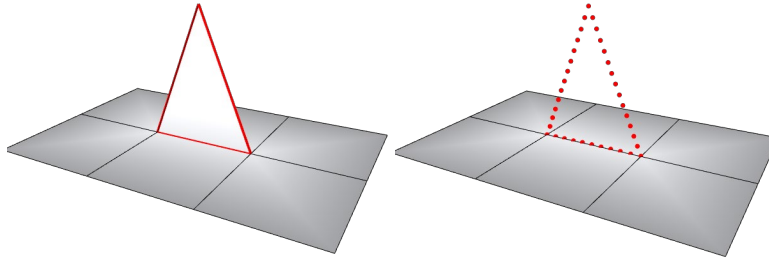


Figure 29: *GHeal's* function *RemoveProtrusions* removes a single face which “sticks out” from the main mesh.

### *SplitNonPlanarFaces*

This function splits non-planar quadrilateral faces into pairs of triangular faces. The non-planar quadrilaterals are divided along the short diagonal.

### *CullRedundancies*

This function culls unused mesh points and degenerated faces and combines points that are too close to each other. The function uses the parameter *MaxNodeDistance*, which is the maximum distance between two mesh nodes below which the nodes will be considered coincident.

### *AlignNormals*

This function aligns normals in each component of the mesh and it works on compound meshes consisting of joined/welded sub-meshes. This function is more robust than *Rhino's*

**\_UnifyMeshNormals** command when working with compound meshes.

### *Parameters* [= *Automatic* (default), *Custom*]

This option allows specifying parameters for the functions selected in the *IssuesToFix* menu.

*Parameters* = *Automatic* automatically calculates and uses optimal parameters for the functions listed above. *Parameters* = *Custom* allows user to manually specify parameters for *GHeal's* functions. Possible options are:

#### *MaxHoleArea* [value > 0, default = 1]

This parameter defines, in model units, the maximum area of a hole to be filled.

#### *MaxCrackOpening* [value > 0, default = 0.01]

This parameter defines, in model units, the maximum distance between the edges of a crack in model units.

#### *MaxProtrusionPerimeter* [value > 0, default = 0.1]

This parameter defines, in model units, the maximum perimeter of the protruded mesh patch in model units.

#### *ToleranceMultiplier* [value 0.001-1000, default = 0.1]

This parameter defines the multiplier for the minimum edge length found among all clashing faces. Clashing faces are intersected in each cluster using *GInt* algorithm, and intersection tolerance is calculated as *tolerance* = *multiplier* x *minEdgeLength*.

#### *MaxNodeDistance* [value > 0, default = 0.001]

This parameter defines the tolerance or the maximum distance between two mesh nodes below which the nodes will be considered coincident. Model units are used.



## GExtract – Tools for Extraction of Surface Meshes

**GExtract** is a *Griddle* utility that allows extracting parts of surface meshes based on user-specified criteria. As surface meshes approximating real engineering and geological structures may be rather complex, there is often a need to extract (or separate) parts of the meshes either to remove them or to assign specific properties in preparation for volume meshing. **GExtract** has five different modes of operation to assist with that.

For the first two modes described below, **GExtract** extracts faces based on a break angle and whether to stop the extraction at non-manifold edges (a common edge that is shared by more than 2 faces). In these modes, the command is similar to *Rhino*'s command **\_ExtractConnectedMeshFaces**, except the **GExtract** utility allows for a consistent manner of selecting faces of non-manifold meshes and stopping only if the angle exceeds the break angle and/or at non-manifold edge.

### GExtract Options

#### SingleSurface

In this mode **GExtract** extracts single manifold sub-mesh containing a selected face. When using this mode, user first must select a mesh face from which the rest of the selection will propagate (Figure 30). After that a dialog will be shown (Figure 31, Figure 32) that requests criteria for stopping selection. When the user clicks **OK**, all the selected faces will be separated as a new mesh (mesh properties will match the original properties).

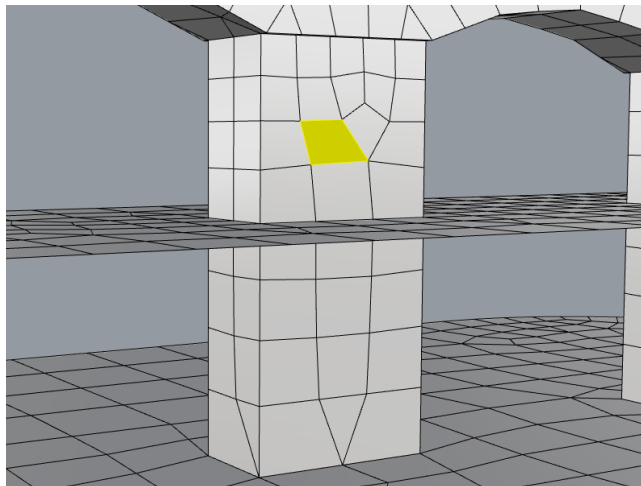


Figure 30: Selecting an initial face for **GExtract**'s *SingleSurface* extraction mode.

The **Non-Manifold Extract Faces** dialog sets two criteria for stopping selection:

- By a break angle (*Max break angle*: 0-180°) or
- when the selection hits a non-manifold edge (*Break at non-manifold edges*).

The first option specifies the angle between joined faces (in degrees). Specifying 0° will only select joined faces that are coplanar with the face picked. If the second option is disabled, **GExtract** will



propagate its selection of faces at non-manifold edges to those faces having the smallest angle with the currently processed faces (i.e., **GExtract** will follow the smoothest path at non-manifold edge).

The second option allows stopping the selection at non-manifold edges. Figure 31 and Figure 32 show the results when breaking at non-manifold edges is enabled or disabled.

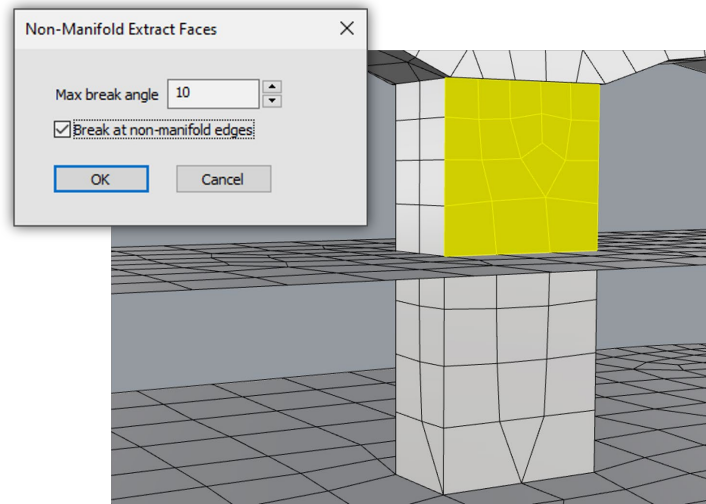


Figure 31: Face selection in the case where *Break at non-manifold edges* is enabled.

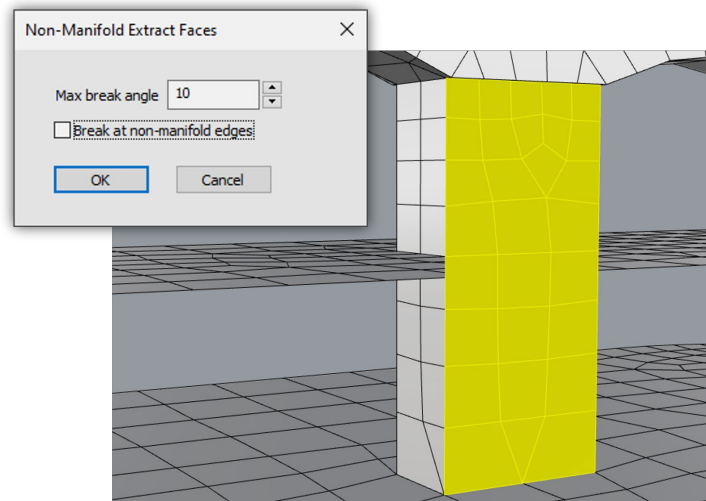


Figure 32: Face selection in the case where *Break at non-manifold edges* is disabled.

### **AllSurfaces**

This mode extracts all possible manifold sub-meshes from selected surface meshes. This option specifies break angle only (0-180°). The extraction of sub-meshes occurs when either angle between neighboring faces exceeds the specified break angle or if a non-manifold edge is encountered. Note that initial faces are picked automatically; the user does not need to pick any faces.

The example in Figure 33 and Figure 34 shows the use of *AllSurfaces* mode with break angle 60°. A single non-manifold conformal mesh (Figure 33) is split into multiple manifold conformal meshes (Figure 34). Afterwards, the extracted meshes are assigned different colors using the **ColorizeObjects** tool.

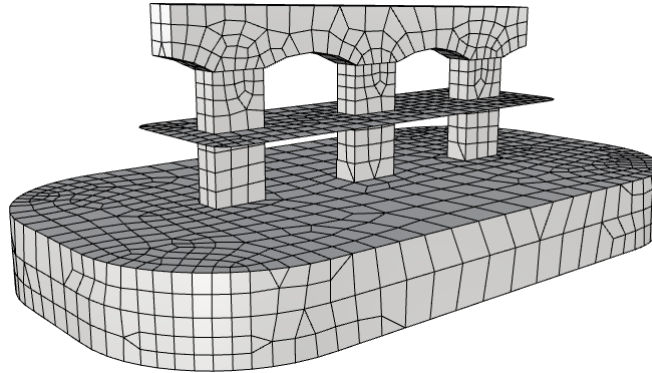


Figure 33: Initial single non-manifold conformal mesh (some internal meshes are not visible).

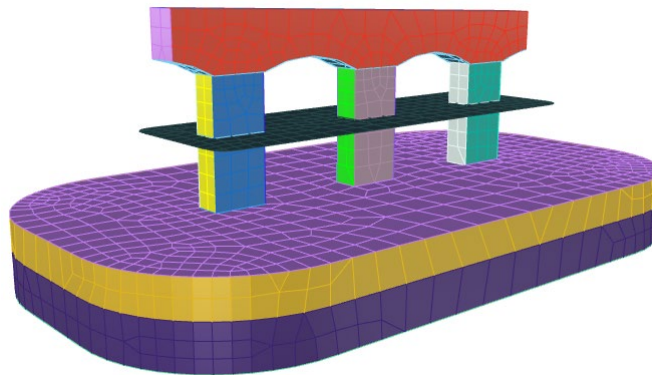


Figure 34: The result of application of *GExtract* with *AllSurfaces* mode to the initial mesh.

When using *AllSurfaces* mode, the angle between faces is calculated using the angle between face normals. Therefore, it is important that all mesh normals are aligned to have proper mesh extractions. If mesh extraction leads to unexpected results, align mesh normals using *GHeal*'s *AlignNormals* function or *Rhino*'s **\_UnifyMeshNormals** command (it may not work properly for compound/joined meshes).

### ***BoundaryFaces***

This mode extracts only those faces that are attached to naked edges (i.e., mesh boundaries). Extracted faces are combined into a separate mesh that is assigned the same layer and name as the original mesh but a different color. This option can operate on multiple meshes simultaneously. An example of extraction of boundary faces from two meshes is shown in Figure 35.

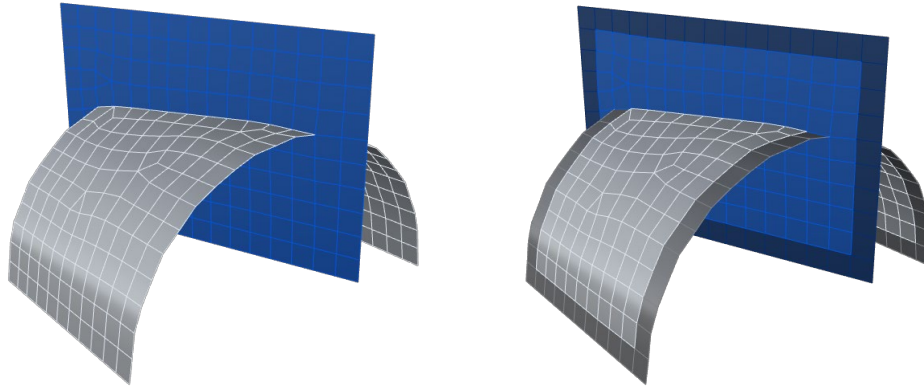


Figure 35: Extraction of boundary faces using *GExtract's BoundaryFaces mode*.

### *NonManifoldFaces*

This mode extracts only those faces that are attached to non-manifold edges if any are present in the mesh. Extracted faces are combined into a separate mesh that is assigned the same layer and name as the original mesh but a different color (Figure 36). The option can operate on multiple meshes simultaneously. This option may be used to refine a mesh around non-manifold edges by using the surface remesher **GSurf** (for example, (i) assign a custom element size to the extracted mesh and remesh everything or (ii) specify hard edges along extracted mesh boundaries using the **\_DupBorder** command and then remesh only the extracted mesh).

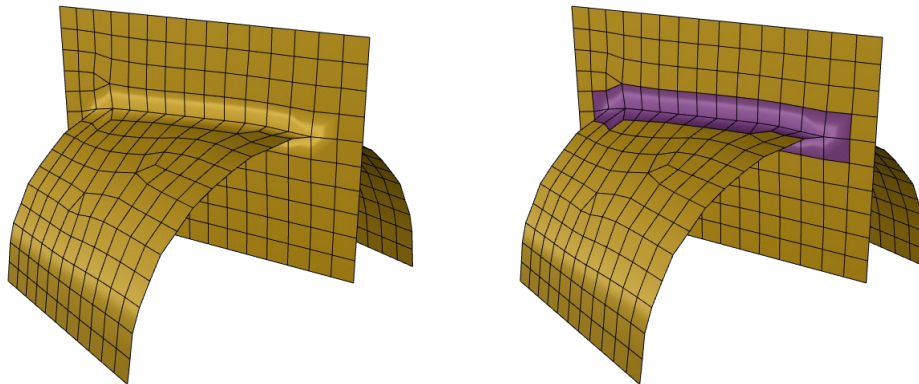


Figure 36: Extraction of faces attached to non-manifold edges using *GExtract's NonManifoldFaces mode*.

### *SurfacesWithinSolid*

This mode extracts pieces of meshes that are located within specified closed volume solids (e.g., a box, cylinder, etc.) Extracted mesh pieces have the same properties as the original meshes but are assigned a slightly different color. After the operation completes, extracted mesh pieces are selected so it is easier for the user to do other operations with them (e.g., hide, delete, move, remesh, etc.) It is advisable to keep the solids in a separate layer(s), so they can be easily hidden while keeping the extracted meshes selected (for example, to hide the solids, switch layers containing them to 'off'). The option can operate on multiple meshes simultaneously and can use multiple solids. This option may be used to change element size in extracted meshes by using surface remesher **GSurf** (for example, (i) assign custom element size to extracted mesh parts and then remesh everything or (ii) specify hard

edges along the extracted mesh boundaries **\_DupBorder** command and then remesh only the extracted mesh parts).

Figure 37 shows an example of extraction of sub-meshes located within a cylinder (the solid). The cylinder is represented as BRep object, and it is not a mesh. First, user must select all meshes to process and then select the solid. After **GExtract** finishes operation, the extracted meshes are highlighted (top-right image). If the cylinder was placed into a separate *Rhino* layer, it can be easily hidden from the view by turning off the corresponding layer. The extracted meshes stay selected (bottom-left image). At this point user may change properties of the selected meshes or do other operations with them.

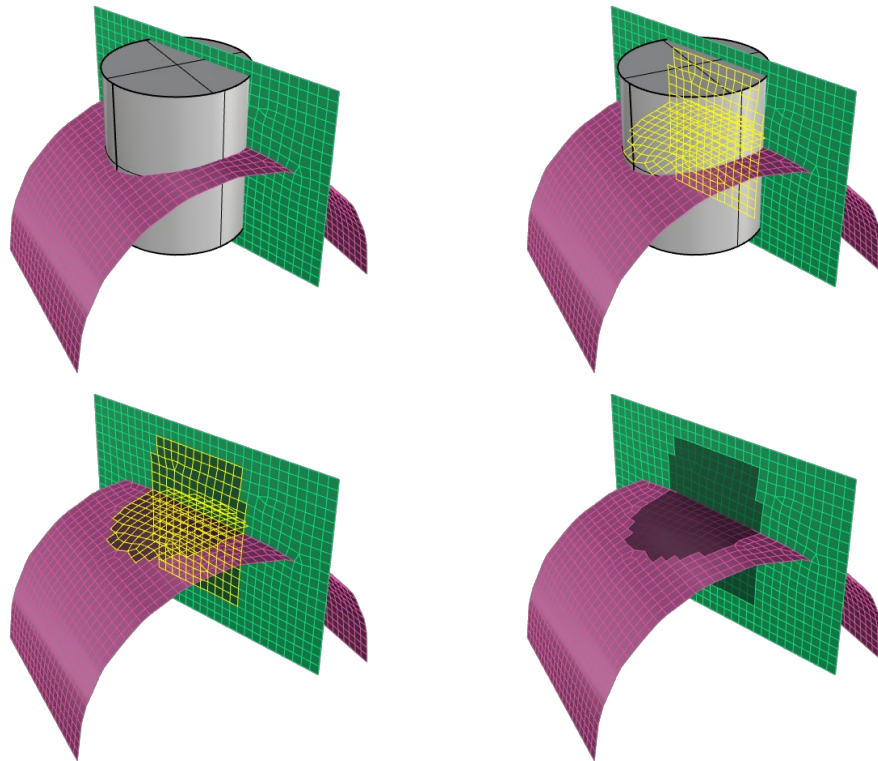


Figure 37: Process of extraction of mesh faces located within a solid by using *GExtract's SurfacesWithinSolid* mode.



## **GExtend – Tools for Surface Mesh Extension**

**GExtend** is a *Griddle* utility that allows extending a surface mesh by adding new faces to it. New mesh faces are created along a selected part of the mesh boundary or along the whole boundary. After that they can be joined to the original mesh or kept as a separate mesh (the original mesh is not changed).

**GExtend** has several options when building new faces: (i) building new faces in the average tangent direction calculated using orientation of the initial mesh faces at the selected part of the boundary, (ii) building new faces along user-specified vector, and (iii) building new faces in “free” direction (in this mode user can drag-and-drop the boundary piece on the screen to set the final position).

One of the important parameters of **GExtend** is the extension length, which specifies edge size for newly built faces (along the extension direction). The extension length should be comparable to the size of faces in the initial mesh at the selected part of the boundary. If the extension length is significantly larger than the typical initial face/edge size, the extended mesh may be of poor quality or even be invalid due to the requirement of preserving average tangent.

After using the **\_GExtend** command, remeshing of the extended surface mesh may be necessary to improve mesh quality and to achieve desired element size.

**GExtend** has three modes of operation: *ExtendSelectedBoundary*, *ExtendAllBoundaries*, and *FreeExtend*. For the first two modes, the command extends the boundary by a given distance while the third mode allows the user to do “free” extension of the selected boundary by moving it to any location.

When extending a mesh along a part of the mesh boundary, all mesh boundaries are highlighted in purple for easier selection (Figure 38). With boundaries highlighted, the user selects a desired region as shown in Figure 38. Additional pieces of the boundary can be added (or removed) by pressing and holding **Shift (Ctrl)** and the left mouse button. Make sure to select a continuous piece of the boundary as **GExtend** operates on single continuous segment of the boundary.

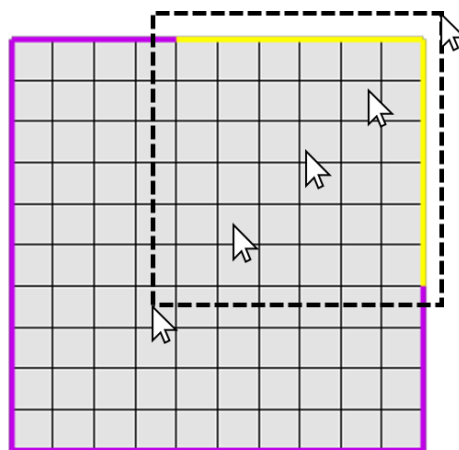


Figure 38: **GExtend** highlights mesh boundary in purple and allows selecting a piece of the boundary (in yellow).

## GExtend Options

### ExtendSelectedBoundary

In this mode, **GExtend** extends the mesh along the selected part of mesh boundary by a given distance (Figure 39). Three parameters should be specified for the operation to begin:

#### ExtendLength

This parameter specifies, in model units, the extension length. It is the same as the edge length of newly built faces in the extension direction.

#### MeshType [= Merged (default), Separated]

This parameter specifies if the newly built (extended) mesh patch should be merged with the initial mesh or kept separate.

#### Direction [= LocalTangent (default), AlongVector]

This parameter specifies the extension direction along which new faces are constructed. If *LocalTangent* is selected, new faces are constructed along average local tangent vector calculated at the nodes of the selected boundary part. If *AlongVector* option is used, the extension direction can be specified by typing the coordinates of vector's starting and ending points. Alternatively, as the command shows all mesh vertices, the user may select any point as the start and/or the end of the extension vector (in general, any two points can be chosen to specify the extension vector).

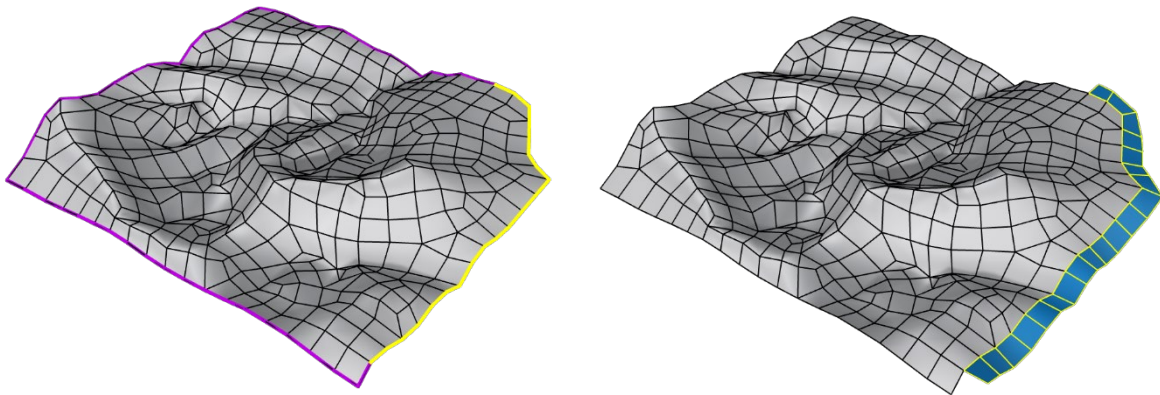


Figure 39: Extension of a mesh along part of its boundary using *ExtendSelectedBoundary* mode.

### ExtendAllBoundaries

This mode expands the initial mesh(es) along all its boundaries by a given distance (Figure 40).

Multiple meshes can be selected and boundaries of each mesh will be extended. The options in this mode are the same as in *ExtendSelectedBoundary* mode.



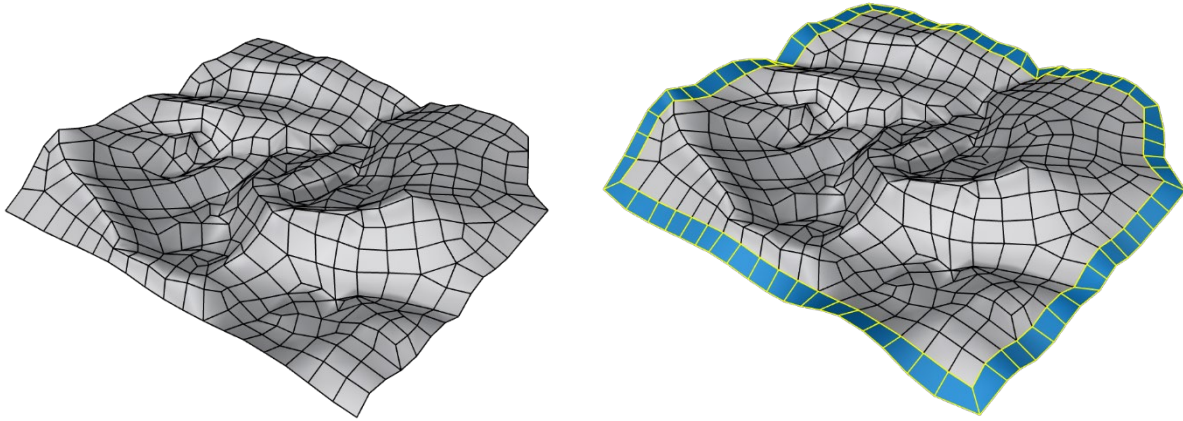


Figure 40: Expansion of a mesh along all its boundary using *ExtendAllBoundary* mode.

### *FreeExtend*

This mode extends a mesh at a selected part of the boundary to a custom position by moving (dragging and dropping) the selected part (Figure 41). The movement starts at a node in the middle of the selected part and stops at a point where user “drops” it. The extended part of the mesh is merged with the initial mesh.

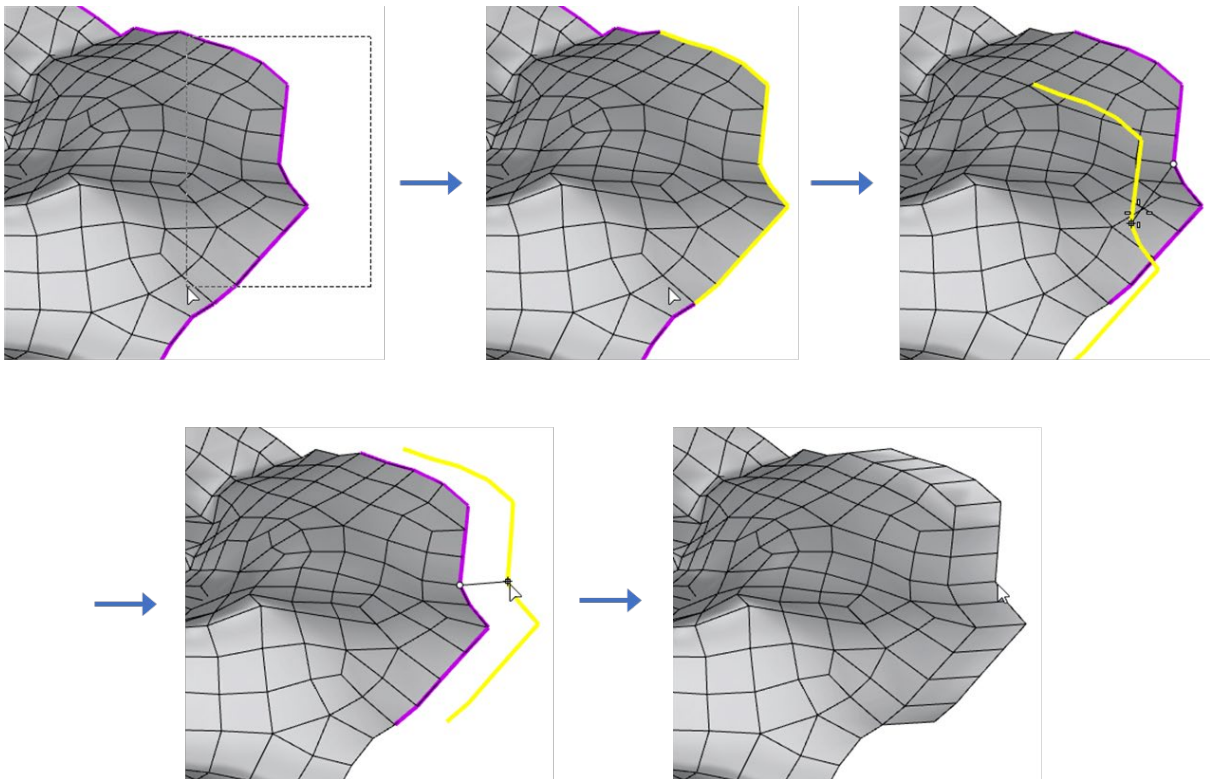


Figure 41: Process of selecting and dragging a part of a mesh boundary using *FreeExtend* mode.



## **GExtrude – Tools for Surface Mesh Extrusion**

**GExtrude** is a *Griddle* utility that allows extruding a surface mesh to a bounding surface to create a watertight mesh domain. The extrusion is done along the mesh external boundary and follows these steps.

1. Project each node on the boundary of the initial surface mesh onto the bounding surface (using shortest distance).
2. Connect the projected nodes to each other and to the corresponding original nodes to form side surfaces.
3. Mesh the side surfaces.
4. Fill the area within the projected boundary on the bounding surface with a mesh.

This approach of mesh extrusion onto a bounding surface is designed to quickly create a watertight modeling domain from a given surface mesh (e.g., a topographic mesh). It works well when the bounding surface is planar or nearly planar. The approach is not designed for exact mapping of the initial surface mesh onto a bounding surface and it does not guarantee that mesh projection will exactly conform to the bounding surface if it is non-planar. The approach requires that the initial surface mesh is simply connected—meaning it cannot contain holes, naked edges, or internal boundaries (check mesh quality and fix issues with **GHeal**, if needed).

**GExtrude** has three basic options: *ExtrdMeshType*, *MeshOutput* and *MeshMode*. Several additional options vary depending on the *MeshMode*.

### **GExtrude options**

#### ***ExtrdMeshType*** [= *Tri* (default), *QuadDom*]

This parameter specifies the type of surface mesh used in the extruded mesh: *Tri* produces an all-triangle surface mesh, *QuadDom* produces a quad-dominant surface mesh (contains a mix of triangles and quadrilaterals).

#### ***MeshOutput*** [= *Merged* (default), *Separated*]

This parameter specifies if the extruded mesh should be merged with the initial mesh into a single mesh object or kept separate.

#### ***MeshMode*** [= *Unstructured* (default), *Structured*]

This parameter specifies the mode of the output mesh. When it is set to *Unstructured* mode, two additional parameters will display:

#### ***MinEdgeLength*, *MaxEdgeLength*.**

These parameters control the resulting minimum and maximum edge size in the final surface mesh. To get uniform sizes, minimum and maximum edge size can be set to the same value. Edge size is specified in model units. It is important to set non-zero minimum edge length to get a good quality output mesh.

Figure 42 demonstrates how the command extrudes a surface mesh onto a bounding surface when *MeshMode* = *Unstructured*.

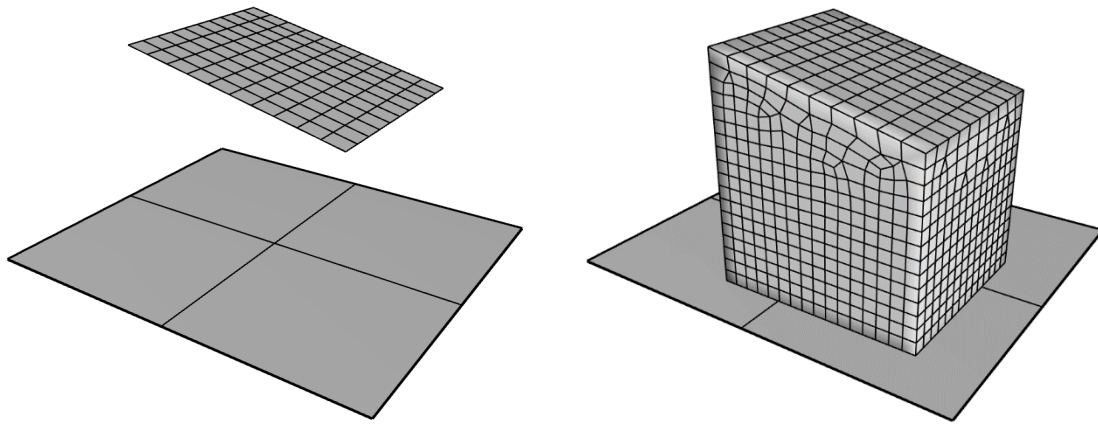


Figure 42: The initial mesh and bounding surface (left) and the results of mesh extrusion to form a watertight domain.

When *MeshMode* is set to *Structured*, an additional option *StructuredParams* appears:

**StructuredParams** [=FixNumberElements (default), FixEdgeLength]

If *FixNumberElements* is chosen, the number of elements along the extrusion direction is fixed and is defined by the option *NumElements*, which the user must specify.

If *FixEdgeLength* is chosen, the element length along the extrusion direction is fixed and is defined by the option *EdgeLength*, which the user must specify.

Figure 43 demonstrates the results of the extrusion when *MeshMode* = *Structured* and *StructuredParams* = *FixNumberElements* (left image) and when *StructuredParams* = *FixEdgeLength* (right image).

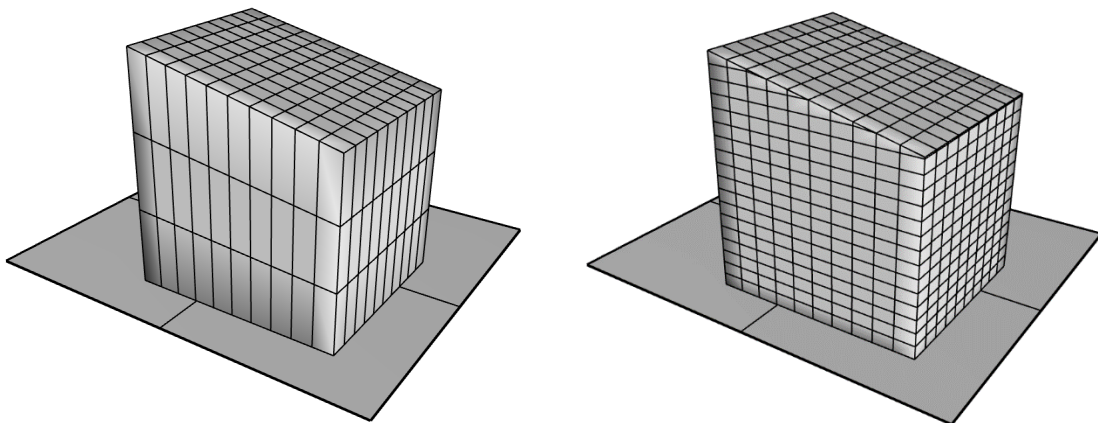


Figure 43: Results of mesh extrusion when using *Structured* mode and fixing the number of elements (left) or the edge length (right) along the extrusion.



## Joining Non-Manifold Surfaces

**\_NonManifoldMerge** is a *Rhino* command that joins several manifold or non-manifold surfaces and polysurfaces into a single non-manifold polysurface<sup>6</sup>. It is provided in *Griddle* for convenience as it can be used in the process of preparing surface meshes for volume meshing.

Starting with surfaces or polysurfaces that fully connect along one or more edges (as in Figure 44), a single polysurface can be created with the *NonManifoldMerge* tool. This polysurface can later be meshed using **\_Mesh** command resulting in a fully conformal non-manifold mesh with all parts of the mesh being properly attached, as opposed to the case of meshing multiple surfaces separately. Now the mesh intersection step (as in Figure 16) may be skipped and the user may continue with remeshing the initial mesh (with **GSurf**) and then creating a volume mesh (with **GVol**).

This procedure avoids problems related to using improper tolerance during the mesh intersection step (**GInt**) and thus obtains accurate results faster. However, the drawback of this approach is that the output surface mesh is now a single object (it can be split using **GExtract**). A simple example of such procedure is shown in Figure 44 and a detailed example is provided in *Griddle 2.0 Tutorial Examples*.

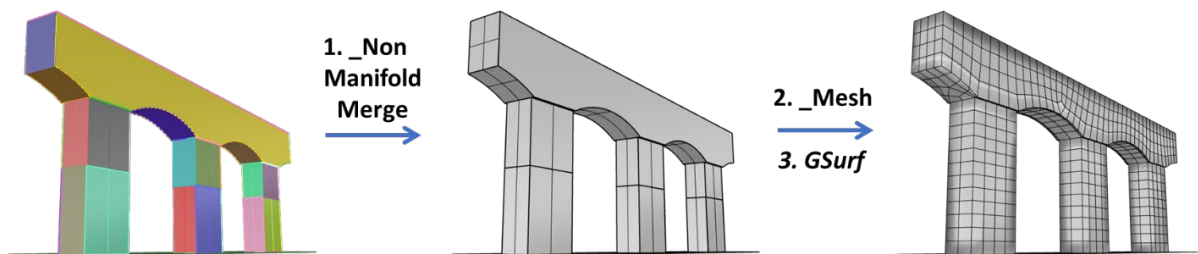


Figure 44: Three step process to create a single mesh out of multiple surfaces/polysurfaces.

Note that in certain cases *Rhino*'s **\_Mesh** command may not generate initially conformal mesh from a single complex polysurface created with **\_NonManifoldMerge** command. This may happen in cases when the **\_NonManifoldMerge** is applied to (poly-)surfaces that protrude through each other rather than connecting along the edges (as in Figure 44).

---

<sup>6</sup>In this context, manifold means that a surface edge is shared by at most two surfaces. In contrast, consider two intersecting planar surfaces and represented as a single polysurface object. Now the edge along the intersection line is connected to 4 sub-surfaces. Thus, the whole polysurface object will be non-manifold.



## Colorizing Objects

In general, all newly created objects in *Rhino* get the color of the active *Rhino* layer. To differentiate objects from each other by color, select several objects in *Rhino* and click on the **ColorizeObjects** icon to assign different (random) colors to each selected object (Figure 45). To revert to a colorization **by layer**, select objects, open the Properties tab (press **F3**), and in the item labelled *Display Color*, select *Display Color* → *By Layer*.

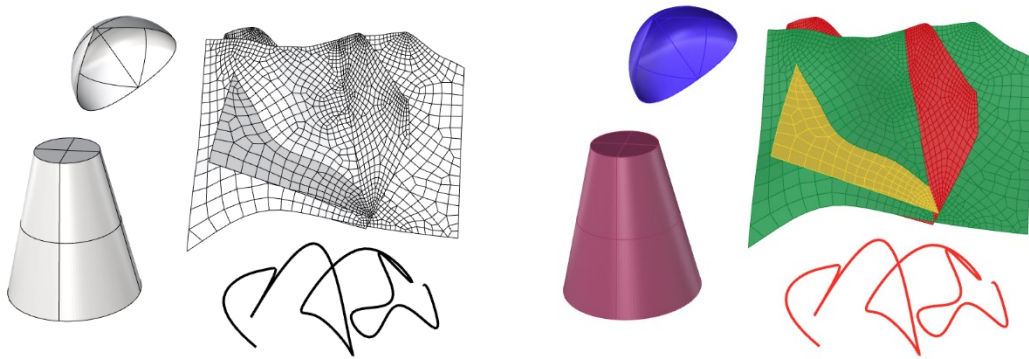


Figure 45: Various objects before and after *ColorizeObjects* is applied.

## Group Names Assignment

### Groups in *FLAC3D* and *3DEC*

*FLAC3D* and *3DEC* can associate names with volume elements (zones, blocks) and names or numbers with faces (face groups, joint ids). These named entities make it easier to reference areas of the model for material or boundary condition assignment.

The volume meshers ***GVol*** and ***BlockRanger*** deal with groups in a slightly different manner.

- ***GVol***, when generating a volume grid, requires a closed set of boundary surface meshes. The inside can be composed of “floating” surfaces, i.e., surfaces that are partially or completely disconnected from other surfaces. Because *Rhino* only operates with surface meshes and not volumes, it is not immediately clear if a set of input surface meshes encloses a watertight volume. For this reason, volumes cannot be directly named; only surfaces can be named.
- ***BlockRanger*** requires solids as input. There is no ambiguity about what is inside or outside a solid. For this reason, ***BlockRanger*** can name solids. ***BlockRanger*** does not allow “floating” surfaces and surfaces in ***BlockRanger*** do not use names.

### ***GVol*** Generated Groups – *FLAC3D*

A hypothetical section through a *Griddle* volume mesh generated for *FLAC3D* is shown in Figure 46. After filling the interior of the model domain with zones (elements), *Griddle* assigns an ID to each watertight volume. Figure 46 presents an example of such volume IDs as large white numbers. The IDs are used to generate group names for zones in *FLAC3D*.

#### **Zones**

*FLAC3D* zone groups are always assigned names automatically based on volume ID (as *Rhino* currently does not have capabilities to interact with volumes represented by surface meshes). To distinguish zone groups from other group names, *Griddle* assigns zone groups the prefix “ZG\_” and a suffix corresponding to the closed volume ID. For, example, area 1 in Figure 46 can be referenced as “ZG\_001” in *FLAC3D*. The “ZG\_” groups are stored in slot “SLOT 1”.

#### **Zone faces**

Zone faces (represented by the thick black and red lines below) are placed in face groups. Face groups can be assigned automatically based on volume IDs or, if faces belong to a named surface, surface name is used. All face groups are placed in slot “SLOT 1”.

*Griddle* creates external face groups with the prefix “EF\_” and internal face groups with the prefix “IF\_”. When surface name is not present (assigned automatically), names of external face groups are appended with the single volume ID to which the faces are attached. Similarly, internal face groups are appended with a combination of two volume IDs representing the two groups on either side of the faces (the larger number is always placed second in the name). If the *Rhino* surface mesh has a name, then this name is always used.



Some surfaces do not fully divide the volume into distinct pieces. Such “floating” surfaces are represented by the red lines in Figure 46. In automatic naming, their face group names contain a duplicate number. For example, faces group “IF\_002\_002” is designated from the “ZG\_002” zone groups on either side of it.

### Using Provided Surface Names

If the surface mesh has a name assigned to it in *Rhino* Properties, then this name will be used for *FLAC3D* face group (and prefixed with “EF\_” or “IF\_”). These names take precedence over the automatically assigned names shown in Figure 46.

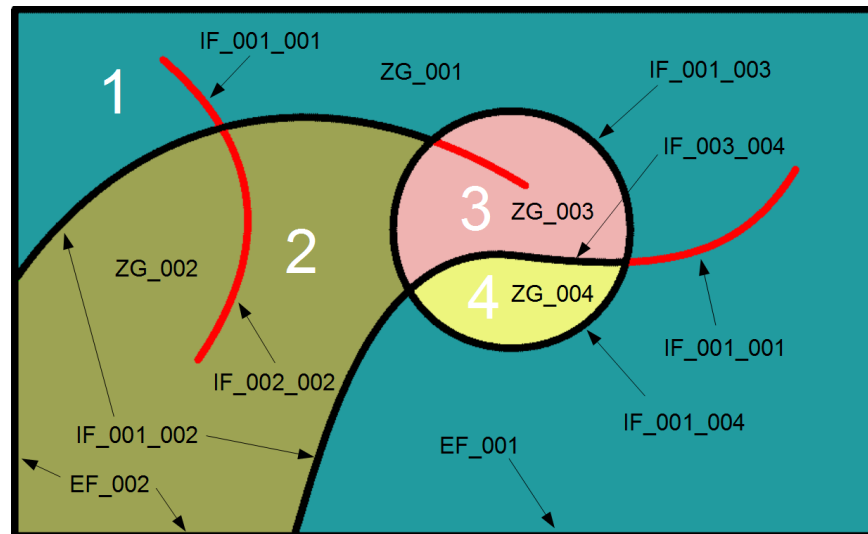


Figure 46: Zone and face groups from a *Griddle*-generated *FLAC3D* grid for automatic group name assignment.

## GVol Generated Groups – 3DEC

### Blocks

*3DEC* block groups are represented by strings containing numeric digits. Block groups start at “001” for *3DEC\_7.x* output and at “10001” for *3DEC 5.2* (due to *3DEC 5.2* specifics) and each new group name is increased by “1”. Block groups are assigned in random order. For example, block group names will be “001”, “002”, ... (similar to *FLAC3D* “ZG\_...” groups, see Figure 46). All groups are placed in “SLOT 1”.

### Joint group names

Joint groups are also represented by strings with integers. All external faces and joints within each group of rigid blocks are typically associated with IDs/groups “1” and “2”, and they are not output by *GVol*.

*GVol* outputs joint groups starting from names “3”, “4”, “5”, ... which are given to the surfaces separating various blocks (for *BlockType = Deformable*) or block groups as well as “floating” surfaces (for *BlockType = Rigid*). All groups are placed in “SLOT 1”.

Joint groups are arbitrarily assigned if corresponding surface mesh name is not provided (there is no relationship between joint group and the block group names surrounding it). Note that “floating” surface meshes (the red lines in Figure 46) will not appear in *3DEC* deformable output and therefore no names would be assigned to them.

3DEC also allows assigning a separate group name to joints. If a surface mesh has a name assigned to it in *Rhino* Properties, then this name will be output as the joint group name.

### BlockRanger-Generated Groups – FLAC3D

**BlockRanger** solids can be named in two ways: through the layer name that the solid belongs to and through the solid Properties Name field (Figure 47). The layer Name is used to generate zone group names for *FLAC3D* in “SLOT 1”. The Properties Name is used to generate zone group names for *FLAC3D* in “SLOT 2”.

The white text names in Figure 48 and Figure 49 are assumed to be *Rhino* layer names. In Figure 48, the black text prefixed with “ZG\_” indicates “SLOT 1” zone groups. The yellow text indicates “SLOT 2” zone groups. “SLOT 2” group names are automatically generated if the Properties Name field is left blank. A *Rhino* layer name cannot be blank. Example internal (“IF\_” prefix) and external (“EF\_” prefix) face group names are also shown in Figure 48. Note, a face group is not generated for the seam between the two bedrock solids or the seam between the two soil solids shown in Figure 48.

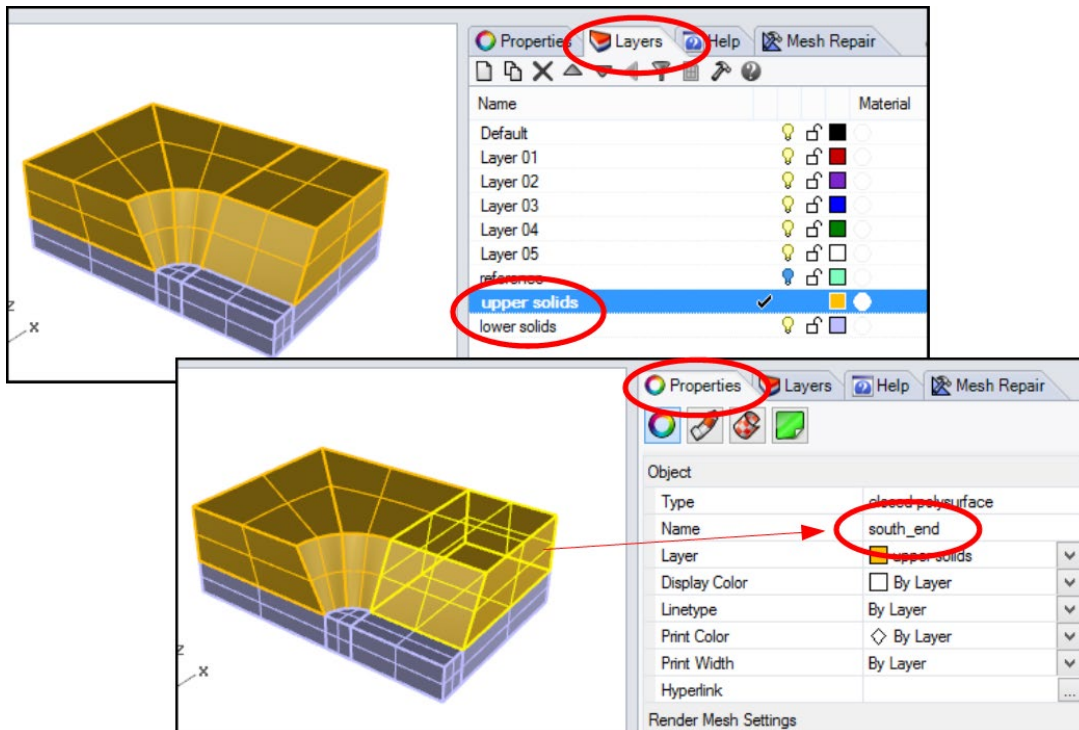


Figure 47: Naming solids by layer (top) and through the Properties Name field (bottom).

### BlockRanger Generated Groups – 3DEC

For 3DEC 5.2, **BlockRanger** uses the layer name as the 3DEC block group name (Figure 49). For example, the *Rhino* layer “soil” would result in a 3DEC block group called “soil”. 3DEC 7.x output uses block numbering similar to *FLAC3D* and does not use layer names. For either version of 3DEC, all block groups are placed in “SLOT 1”. Internal faces which separate different named layers are assigned joint set IDs starting from 2, as in Figure 49.

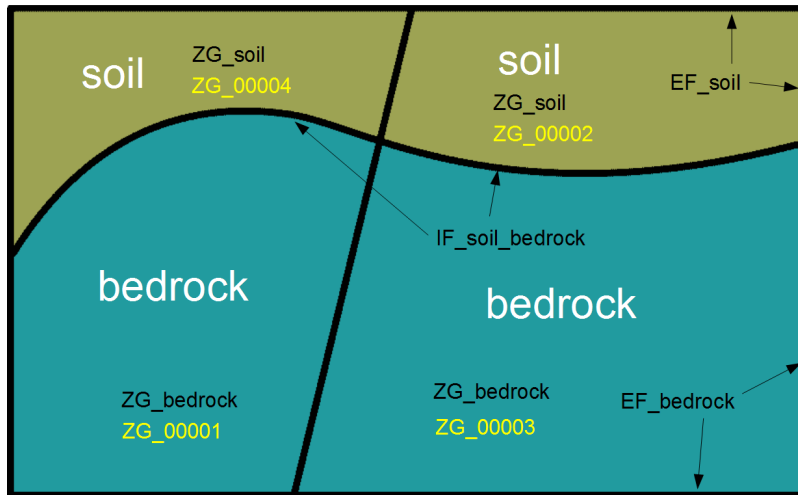


Figure 48: Zone and face groups from a *BlockRanger*-generated *FLAC3D* grid.

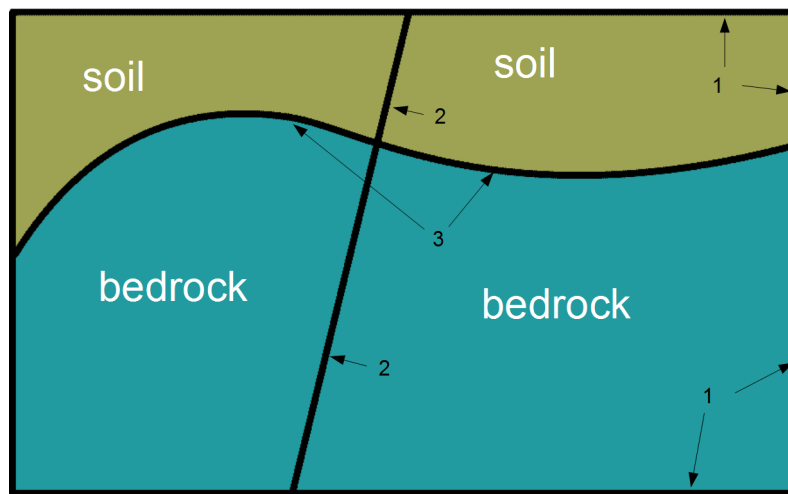


Figure 49: Block groups and joint set IDs from a *BlockRanger* generated *3DEC* model.

Note: **GVol** and **BlockRanger** replace all occurrences of whitespace characters (blanks, tabs, etc.) in name fields with an underscore character “\_” to avoid string interpretation problems in *FLAC3D* and *3DEC*. Names are case insensitive, e.g. “Upper\_Rock” is treated as the same name as “upper\_rock”.

## Group Names in Other Formats

When outputting in other formats (e.g. *ANSYS*, *NASTRAN*, etc.) no groups are assigned except for *ABAQUS* output in which:

- **GVol** assigns automatically generated names for element groups in watertight volumes in the format: “G10001”, “G10002”, etc.
- **BlockRanger** assigns names to element groups according to *Rhino* layer names.

## Importing Griddle Volume Meshes

Griddle's volume meshers **BlockRanger** and **GVol** output volume meshes into a text or binary file in the format corresponding to the selection of a numerical simulation software, such as *FLAC3D*, *3DEC*, *ABAQUS*, etc. The list below provides general tips on importing mesh files into corresponding software (note that workflow in the users' version of the software may differ from what is present below):

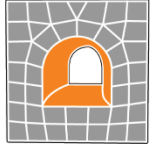
- *FLAC3D*: Navigate to menu *File* → *Grid* → *Import from FLAC3D...* and open the mesh file (\*.f3grid) or use the "zone import" command.
- *3DEC 7.0* or newer: Navigate to menu *File* → *Grid* → *Import from 3DEC...*  
or *File* → *Grid* → *Import from Griddle...* and open the mesh file (\*.3dgrid)  
or use the "block import" command.
- *3DEC 5.2*: Navigate to menu *File* → *Open Item...* and select *Data File: Call*. Then locate and open the mesh file (\*.3ddat) containing commands to generate the grid.
- *ABAQUS*: Workflow is provided for *Abaqus/CAE*. Navigate to menu *File* → *Import* → *Model* → *Abaqus Input File (\*.inp,...)* and open the mesh file (\*.inp)
- *ANSYS*: Workflow is provided for *ANSYS Workbench*. Importing depends on how the volume mesh is generated:
  - For **GVol** generated meshes, navigate to menu *File* → *Import* and select the mesh file (\*.cdb).  
Within the *Project Schematic* window, double click on *Setup* to open the *External Model* window. Within the *Properties of File* pane, uncheck the checkbox "Check Valid Blocked CDB File".  
Navigate back to the *Project Schematic* window and right-click on *Setup* and select *Update*, which will import the mesh.
  - For a **BlockRanger** generated mesh, first run script from *ANSYS Workbench* on the output file by navigating to menu *File* → *Scripting* → *Run Script File*. Within the *Scripts* folder, select the *ConvertAnsysFileToCdb.py* script and open the mesh file (\*.cdb) to convert it from the old-style format to a blocked CDB file.  
Once the file is converted, navigate to menu *File* → *Import* to import the converted CDB file. Within the *Project Schematic* window, right-click on *Setup* then *Update*, which will import the mesh.
- *LS\_DYNA*: Workflow is provided for *LS-PrePost*. Navigate to menu *File* → *Import* → *LS-DYNA Keyword File*, select *All Files* in the file filter, and open the mesh file (\*.lsdyna970).
- *NASTRAN*: Workflow is provided for *MSC-Patran*. Navigate to menu *File* → *Import* → *Source: MSC.Nastran Input* and open the mesh file (\*.bdf).

## References

Itasca Consulting Group Inc. (2019) *FLAC3D — Fast Lagrangian Analysis of Continua in Three Dimensions*, Ver. 7.0. Minneapolis: Itasca.

Itasca Consulting Group Inc. (2020) *3DEC — Three-Dimensional Distinct Element Code*, Ver. 7.0. Minneapolis: Itasca.

*Understanding Rhino Tolerances*: <https://wiki.mcneel.com/Rhino/faqtolerances>



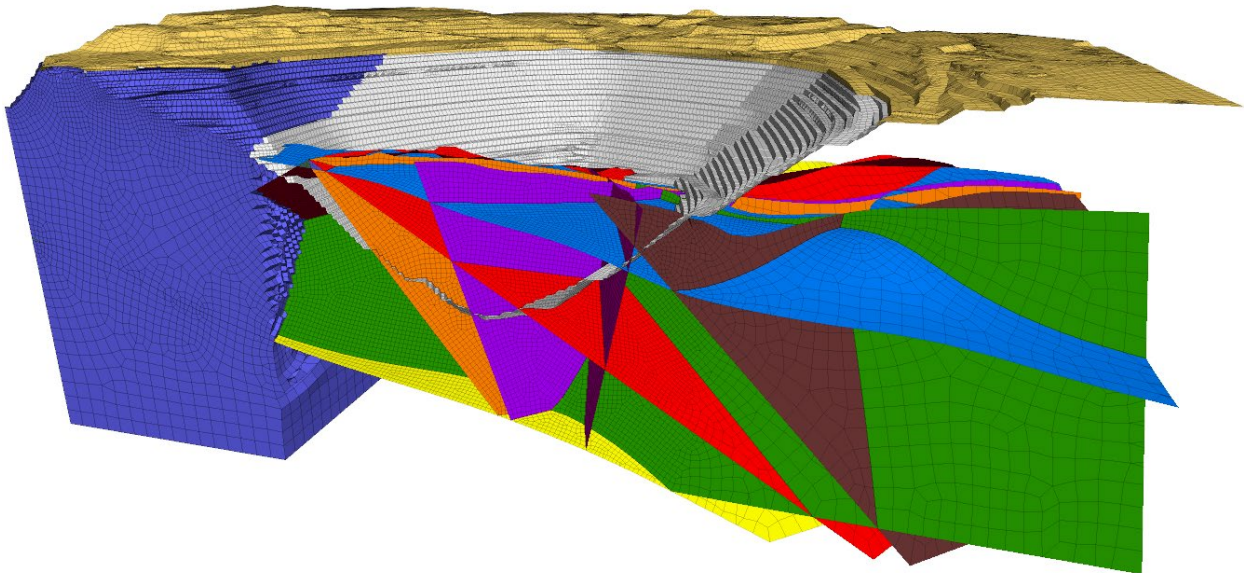
**Griddle™**

# Advanced Grid Generation for Engineers and Scientists

---

## *Griddle 2.0* Tutorial Examples

---



©2021

Itasca Consulting Group Inc.  
111 Third Avenue South, Suite 450  
Minneapolis, Minnesota 55401 USA

phone: (1) 612-371-4711  
fax: (1) 612-371-4717  
e-mail: [griddle@itascacg.com](mailto:griddle@itascacg.com)  
web: [www.itascacg.com](http://www.itascacg.com)

## Contents

<b><i>Introduction.....</i></b>	<b><i>3</i></b>
<b><i>Tutorial 1: A Single Cylinder .....</i></b>	<b><i>4</i></b>
<b><i>Tutorial 2: Vertical Shaft Excavation in a Stratified Soil.....</i></b>	<b><i>11</i></b>
<b><i>Tutorial 3: Meshing a Model with Analytical Surfaces.....</i></b>	<b><i>16</i></b>
<b><i>Tutorial 4: Creating a Structured Mesh with BlockRanger .....</i></b>	<b><i>25</i></b>
<b><i>Tutorial 5: Creation of a Hybrid Structured-Unstructured Mesh.....</i></b>	<b><i>36</i></b>
<b><i>Tutorial 6: Mesh Clean-up and Rebuilding.....</i></b>	<b><i>45</i></b>
<b><i>Tutorial 7: Large Open Pit Model with Multiple Intersecting Faults.....</i></b>	<b><i>52</i></b>



## Introduction

The tutorial examples provided in this document are designed to familiarize *Griddle* users with the principles of *Griddle* operation, workflows, and typical *Rhino* commands that allow the creation of structured and unstructured volume meshes suitable for numerical analysis.

The examples presented in the tutorial are not designed to illustrate exact (or even realistic) engineering models and conditions. However, they provide the basis on which *Griddle* users can build their knowledge for working with complex engineering models.

All the examples were developed in *Rhino 6*, but identical (or very similar) operations, commands, and workflows can be done in later versions of *Rhino*.

When working on the examples from the tutorial, periodically save the model in order to return to it if needed. *Rhino* provides a useful file saving feature, called *Incremental Save* (accessible from the top menu: **File** → **Incremental Save**).

Refer to the *Griddle* User Manual to find more information about *Griddle* commands and options, and the *Rhino* help system (top menu **Help** or **F1**) for *Rhino* operations.

The files corresponding to the tutorial examples can be accessed from the *Windows Start Menu* → *Itasca Griddle 2.0* → *Griddle 2.0 Tutorial Files* link. Clicking on the link opens user writable directory `ProgramData\Itasca\Griddle200` (typically on drive "C:") which contains the documentation and the tutorial material. Users can directly work in this directory. A reserve copy of the same material can be found in the `Documentation` subfolder of *Griddle* installation location (typically in `C:\Program Files\Itasca\Griddle200`).

## Tutorial 1: A Single Cylinder

Expected work time: 0.5 – 1 hour

This tutorial provides information on how to create unstructured and structured meshes for a simple cylinder using *Griddle's* **GVol** and **BlockRanger** volume meshers. Before proceeding with meshing, a cylindrical geometry is created, and this process differs depending on which volume mesher will be used.

### Generating unstructured mesh with *GVol*

1. Open *Rhino*, and when the **Templates** splash screen appears, select **Small Objects, Meters**. The same can be done by navigating to **File** → **New** from the *Rhino* top menu.
2. Save the *Rhino* project at a desired location.
3. Type the **\_Cylinder** command or select the **Solid** → **Cylinder** menu item. Type **0** and press **Enter** to set the location of the center of the cylinder base at the origin. Next, set the radius of the base to **2**. Enter **10** for the height of the cylinder. Press **Alt+Ctrl+E** to zoom out all viewports to the extent of all objects (or navigate to **View** → **Zoom** → **Zoom Extents All**). This completes construction of the geometry for a simple vertical cylinder (Figure 1).

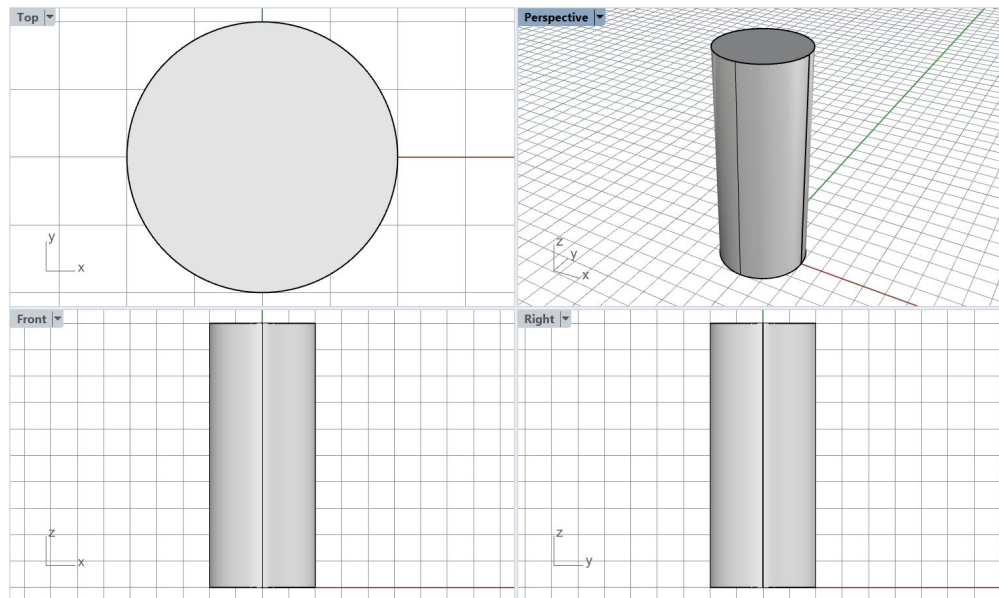


Figure 1: Solid representing a cylinder

The watertight geometry created in the previous steps can be referred to as a cylindrical solid. A solid is a closed surface that defines a volume with a clear interior and exterior. The next step is to triangulate the cylinder surface to create the initial triangular mesh.

4. Select the cylinder and type **\_Mesh** in the *Rhino* command prompt. If the **Polygon Mesh Detailed Options** dialog window opens, click on the **Simple Controls...** button in the lower-right corner of the window to open a simplified dialog.
5. In the simplified dialog, move the slider all the way to the right towards **More polygons** and click **OK** to create the initial surface mesh (Figure 2).

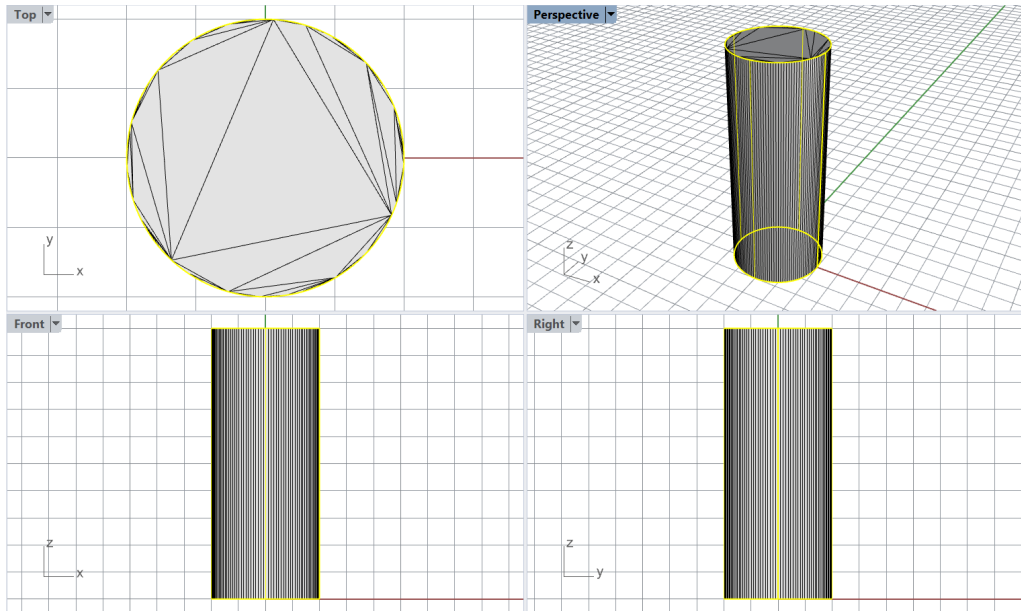



Figure 2: Original highlighted cylindrical surface and the newly created surface mesh superimposed on it.

6. While the original cylindrical surface is still highlighted (seen in light yellow in Figure 2), type the **\_Hide** command to hide it, making only the surface mesh visible. This triangular surface mesh serves as input to *Griddle* tools for unstructured meshing.
7. Select the surface mesh and type **\_GSurf** or click on the  icon in the *Griddle* toolbar to remesh the initial triangular mesh. Select **Mode = QuadDom**, **MinEdgeLength = 0.5**, **MaxEdgeLength = 0.5** and keep all other parameters at the default values. The resulting mesh is shown in Figure 3.

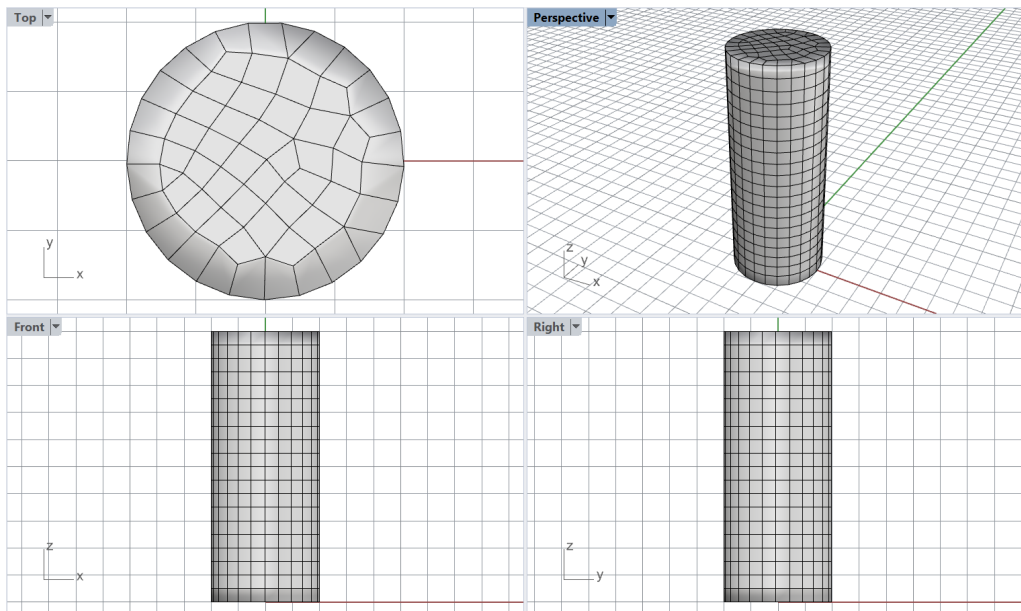



Figure 3: Quad-dominant surface mesh generated by **\_GSurf**.

8. Select the surface mesh generated in the previous step and type **\_GVol** or click on the  icon in the *Griddle* toolbar. Select **MeshSettings → Mode=HexDom**, leave all other parameters at the

default settings and press **Enter** twice. After the **Save As** dialog appears, type a desired name for the output file. **GVol** will generate a conformal hex-dominant volume mesh and output it using the *FLAC3D* binary format (the default format) to the selected file. This file can be imported into *FLAC3D* using *FLAC3D*'s **File** → **Grid** → **Import from FLAC3D ...** option (Figure 4).

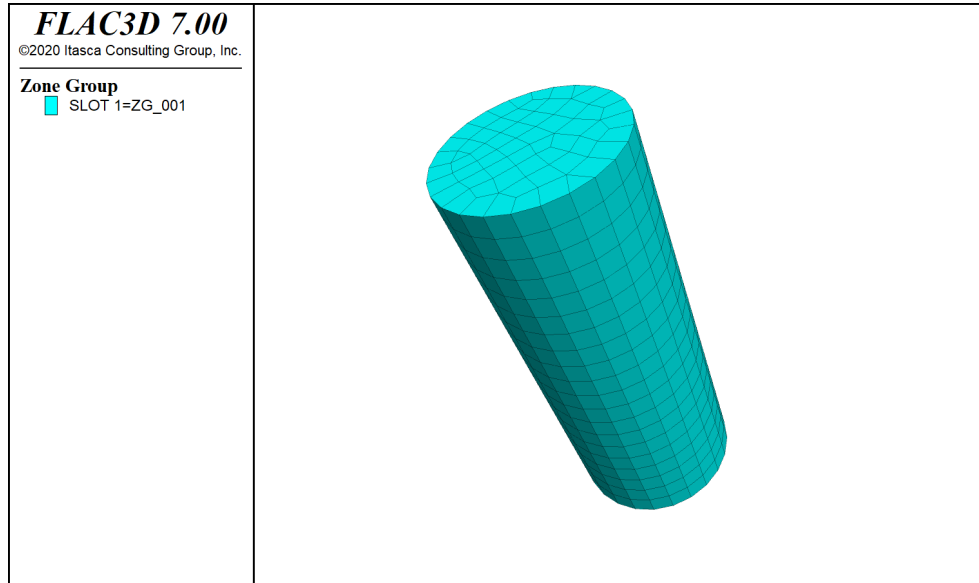


Figure 4: Volume mesh of cylinder imported into *FLAC3D*.

Although the mesh in Figure 3 and Figure 4 may resemble a structured mesh or a pure hexahedral mesh, the **GVol** output log file reports that the output mesh contains:

```
number of elements:
total: 1433
hexahedra: 1072 (74.81% of total, 94.04% of volume)
prisms: 67 (4.68% of total, 2.63% of volume)
pyramids: 187 (13.05% of total, 2.62% of volume)
tetrahedra: 107 (7.47% of total, 0.71% of volume)
```

This report confirms that the output mesh is hex-dominant as it was selected in **GVol** options. Note that users may get slightly different results with regard to the number of elements of each type (this also depends on the particular version of *Griddle*).

## Generating structured mesh with *BlockRanger*

**BlockRanger** is *Griddle*'s all-hex structured volume mesher, and it operates using different principles than the unstructured mesher *GVol*. As such, **BlockRanger** operates only on four-, five-, or six-sided solids (not surface meshes). The cylinder created in the previous section is not such a solid as it contains only three distinct sides. Therefore, to create a structured mesh for the cylinder, the initial solid must be subdivided into several pieces suitable for **BlockRanger**. There are different ways of accomplishing this task; the most commonly used is described below.

1. Open *Rhino*, and when the **Templates** splash screen appears, select **Small Objects, Meters**. The same can be done by navigating to **File** → **New** from the *Rhino* top menu.
2. Save the *Rhino* project at a desired location.
3. Double-click on *Top* viewport to maximize it.
4. In the *Rhino* command prompt, type the following commands to create the initial curves:
  - **\_Line** → *Start of line* = 1,1 → *End of line* = -1,1
  - **\_Line** → *Start of line* = 1,1 → *End of line* = 2,2
  - **\_Line** → *Start of line* = -1,1 → *End of line* = -2,2
  - **\_Arc** → *Center of arc* = 0 → *Start of arc* = 2,2 → *End point or angle* = -2,2

When creating an arc, make sure that the mouse cursor stays above the previously created lines, so the arc goes through the top portion of a circle (Figure 5).

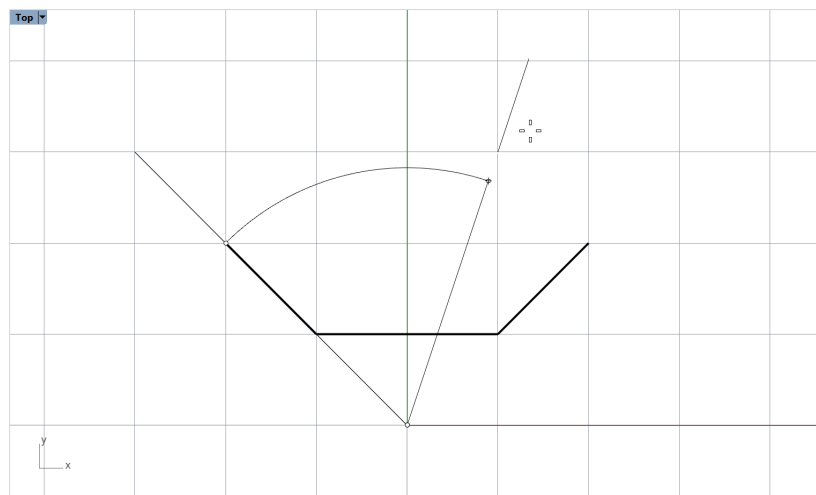


Figure 5: Creating an arc between two points.

5. Select all lines with **Ctrl+A** and type the **\_Join** command to create a single polyline as in Figure 6.

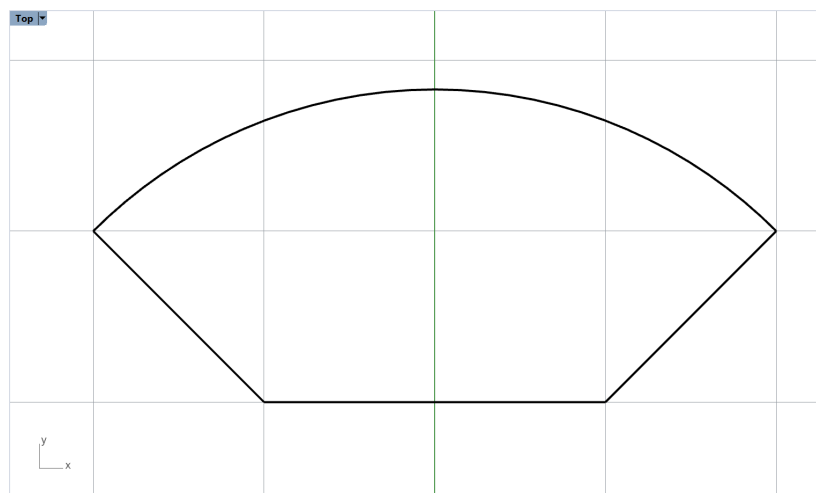



Figure 6: Single polyline consisting of three lines and an arc.

6. While the polyline is selected, type the **\_Rotate** command and click on *Copy=Yes*
  - *Center of rotation* = 0

- Angle of first reference point = 90 (Enter)
  - Angle of first reference point = -90 (Enter)
  - Angle of first reference point = 180 (Enter)
  - Press **Enter**
7. Type `_Polyline` and proceed with:
- Start of polyline = 1,1
  - Next point of polyline = -1,1
  - Next point of polyline = -1,-1
  - Next point of polyline = 1,-1
  - Next point of polyline = 1,1
8. Select all 5 objects and type `_ColorizeObjects` in the command prompt (or click on the  icon in the *Griddle* toolbar). The command will assign a random color to each object (Figure 7).

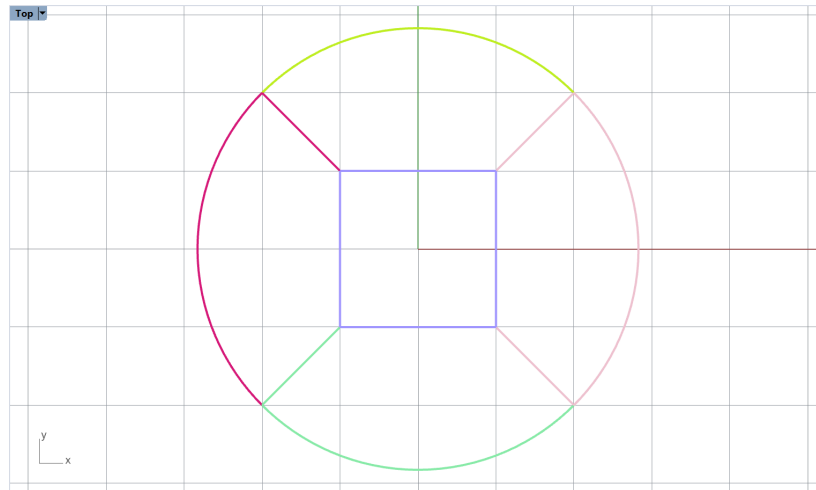



Figure 7: Five colored polylines (some edges overlap).

9. Switch to *Perspective* viewport by clicking on the icon under the viewport.
10. Select all objects and type the `_ExtrudeCrv` command and set *Extrusion distance* = 10.
11. Delete the original polylines while they are selected.
12. Select all objects (should be 5 polysurfaces) and type the `_Cap` command to close them and create watertight solids.
13. Select all and type the `_ColorizeObjects` command.

The resulting five solids represent a cylinder, but each solid is topologically equivalent to a rectangular prism. All these solids are suitable for **BlockRanger**.

14. Select all objects (or type the `_SelPolysrf` command to select only polysurfaces).
15. Type `_BR` or click on the  icon in the *Griddle* toolbar and specify only *GenerateSurfaceMesh* = *BySolid* to visualize the faces on the external and internal boundaries of the solids. Set *MeshSettings* → *MaxEdgeLength* = 0.85. The output volume mesh will be outputted by default in FLAC3D binary format.



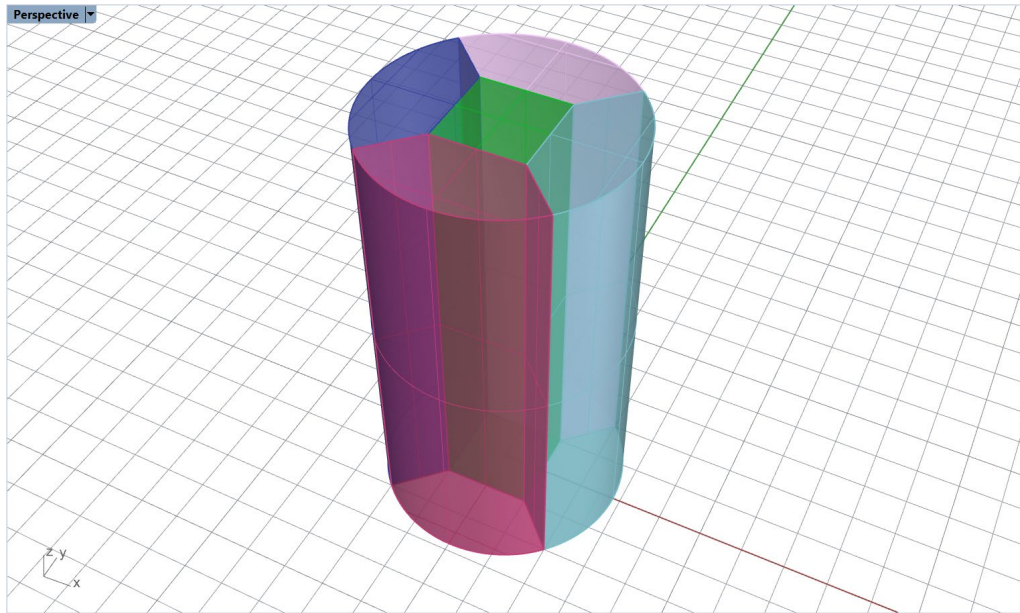


Figure 8: A cylinder subdivided into 5 solids (Ghosted view).

16. Type the **\_SelPolysrf** command to select only polysurfaces and delete them.

The resulting surface mesh is provided in Figure 9 and it shows high quality quadrilateral faces on the boundaries. Note that **BlockRanger** produces a single mesh; for better representation, the mesh in Figure 9 was split into several submeshes (using **GExtract** tool) which were colorized (with **ColorizeObjects** tool).

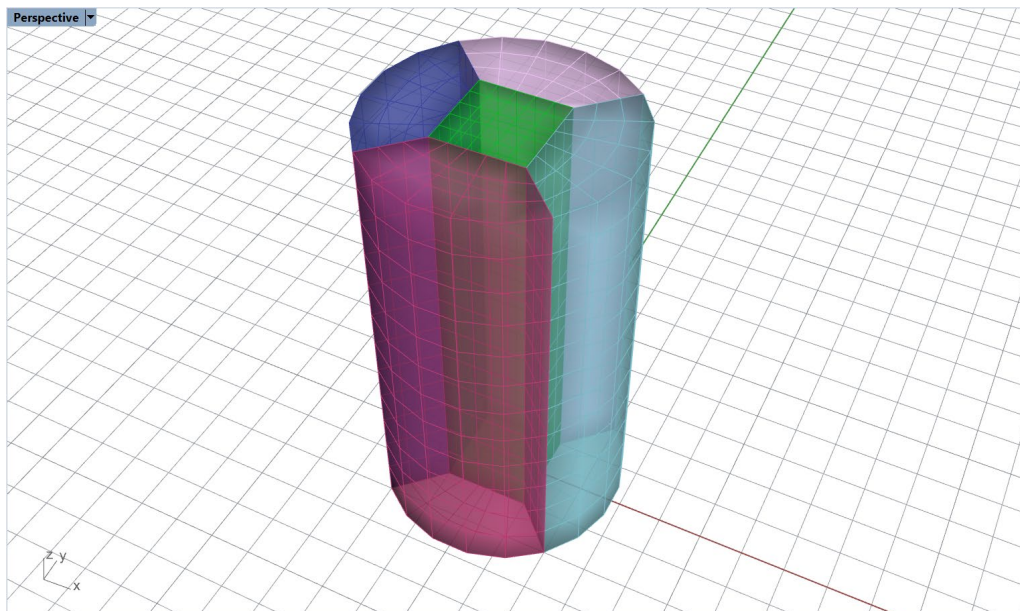


Figure 9: Surface mesh corresponding to the generated volume mesh (Ghosted view).

The output file with a volume mesh can be imported in **FLAC3D**. The zone plot will show five grouped regions composed of a structured pure hexahedral mesh (Figure 10). Note that for this simple extruded



geometry, users can use the *FLAC3D Extruder* to produce similar results. However, **BlockRanger** may operate on rather complex 3D solids, which will be shown in further tutorials.

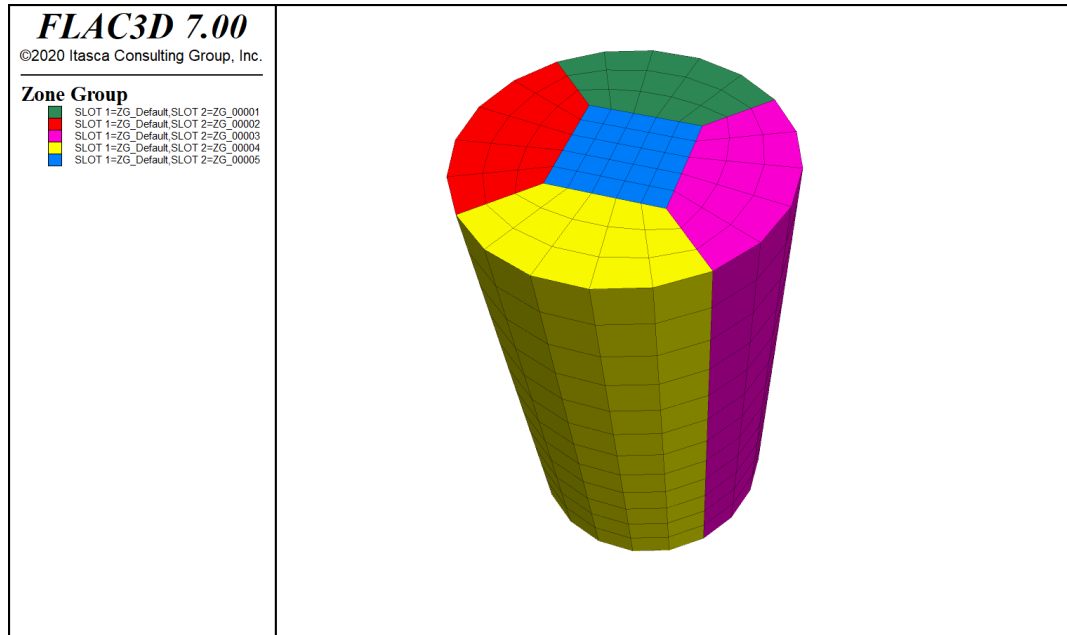


Figure 10: A cylinder subdivided into five solids (Ghosted view).

## Tutorial 2: Vertical Shaft Excavation in a Stratified Soil

*Expected work time: 0.5 – 1 hour*

This tutorial shows an example of constructing the geometry and mesh for a vertical shaft (150 ft deep, diameter of 30 ft) that is to be excavated in stages. The shaft is constructed in two-layered soil with the surface separating the soil layers located at a depth of 50 ft (Figure 11). The model domain is represented as a cube 200 ft × 200 ft × 235 ft. Each excavation stage is 10 ft deep; therefore, there are 15 stages as shown by the red circular pattern in Figure 11.

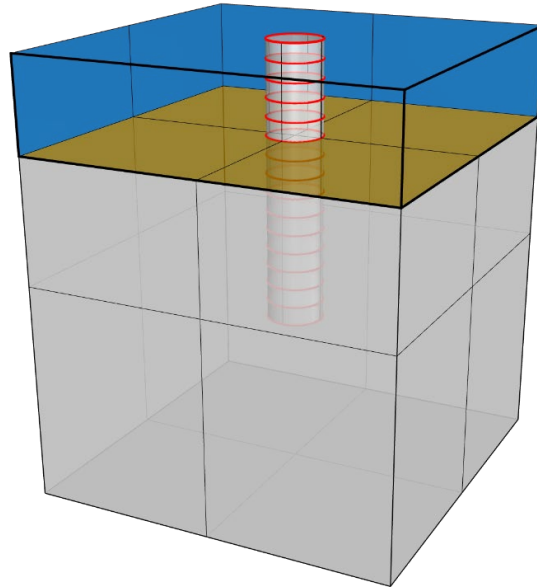


Figure 11: A general view of the model of a vertical shaft in a stratified soil.

### Generation of analytical geometry

1. Start *Rhino*, select **Large Objects, Feet** (similar method as step 1 in Tutorial 1) and save the *Rhino* project at a desired location.
2. Select or maximize *Perspective* viewport and switch to Shaded view.
3. Create a cylinder by typing the **\_Cylinder** command and specify *Base of cylinder* = 0, *Radius* = 15, *End of cylinder* = -10.
4. Create a line by using the **\_Line** command from 0,0,-10 to 0,0,-150.
5. Select the cylinder and type the **\_ArrayCrv** command. Then select the line and, in the pop-up dialog, set the *Number of items* to 15 and keep *Orientation* as *Freeform*; press **OK**. The command will copy the initial 10 ft deep cylinder multiple times to create the desired 15 stages of excavations.
6. Type the **\_SelCrv** command to select the curve and delete it.
7. Verify that 15 stages were created by selecting everything in the viewport (**Ctrl+A**); the command area should display the message:  
15 extrusions added to selection.
8. Create three points that will define the plane separating two layers of soil by using the **\_Point** command. For the coordinates, specify:
  - -100,-100,-50

- 100,-100,-50
  - 100,100,-50
9. Connect all points by a horizontal plane by typing the **\_Plane** command and start dragging a plane from one of the “side” points (the first or third in the list above). The objects should look as shown in Figure 12.

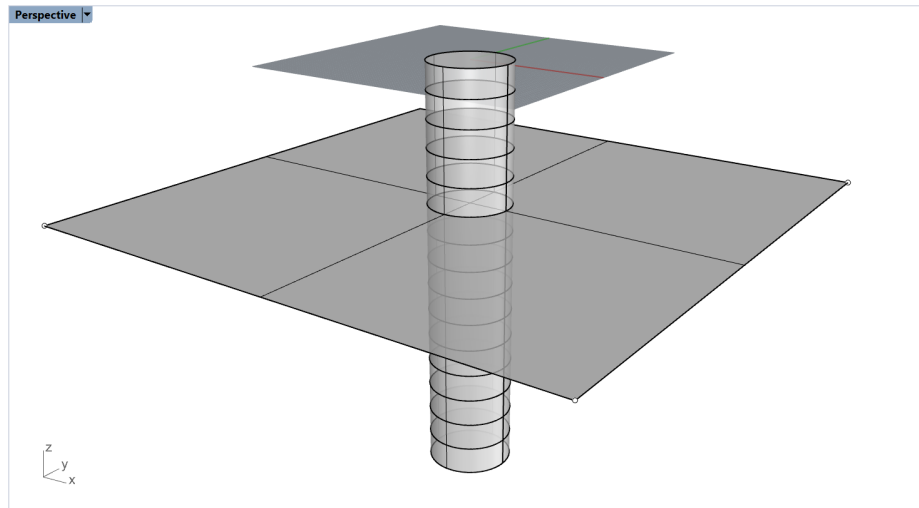



Figure 12: Construction of a shaft and a plane separating two layers of soil

10. Select and delete points using the **\_SelPt** command.
11. Create a domain box by typing the **\_Box** command (or select in the menu **Solid**→**Box**→**Diagonal**) and click on the *Diagonal* option to define the box by 2 points. Enter **-100,-100,235** for the coordinates of the first point and **100,100,0** for the second point.
12. Switch to Wireframe view and zoom to all extents if needed (**Alt+Ctrl+E**).
13. Select all objects and type the command **\_NonManifoldMerge** (or click on the  icon in the *Griddle* toolbar) to create a single non-manifold polysurface. This complex polysurface will serve as the initial geometry for meshing.

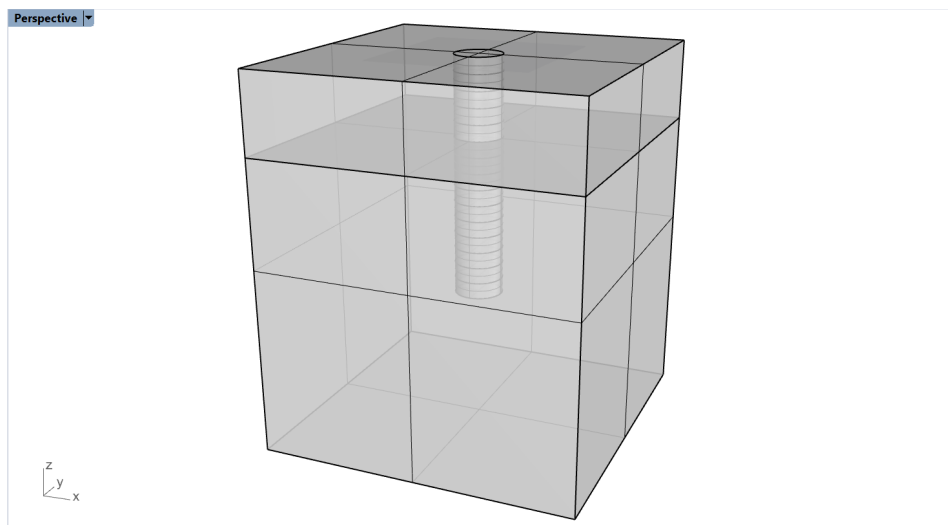


Figure 13: Complete initial geometry of the model of a shaft and two soil layers.

## Meshing with unstructured mesh

The geometry created in the previous step is ready for meshing.

14. Select all objects and type the command **\_Mesh**. If a simplified dialog appears, click on the *Detailed Controls...* button and enter the values as shown in Figure 14.

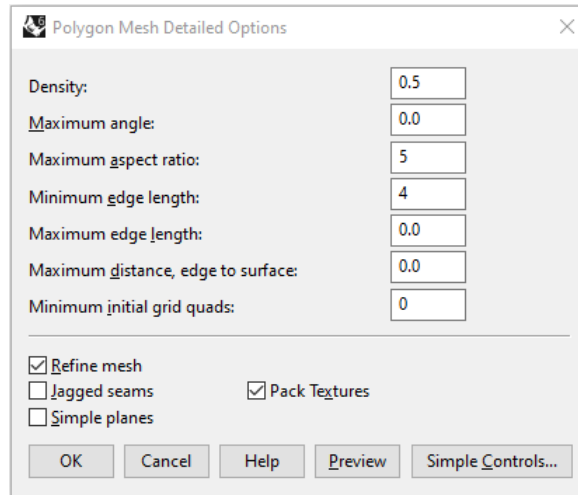



Figure 14: Rhino meshing dialog to create an initial mesh.

15. While the polysurface is still selected, hide it with the **\_Hide** command. If it is not selected, select it with the **\_SelPolysrf** command. Only the mesh should be visible at this point.
16. Select the surface mesh and type **\_GSurf** or click on the  icon in the *Griddle* toolbar to remesh the initial triangular mesh. Select *Mode = QuadDom*, *MinEdgeLength = 4*, *MaxEdgeLength = 20* and keep all other parameters at the default values. The resulting mesh is shown in Figure 15.

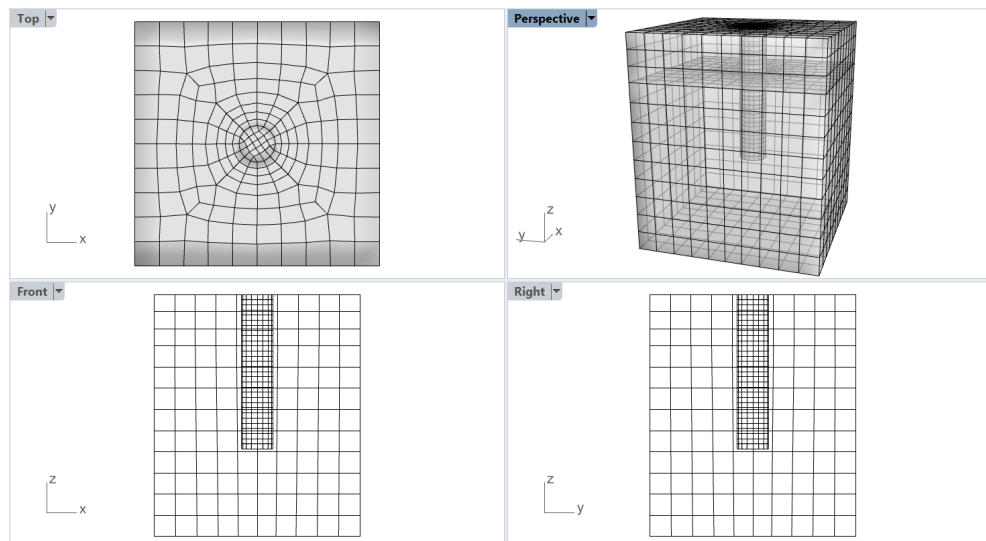



Figure 15: Rhino meshing dialog to create the initial mesh.

The surface mesh in Figure 15 looks good and rather symmetric despite the fact that this is an unstructured mesh. **GSurf** was able to create such a mesh due to simple symmetric geometry and the choice of meshing parameters. In general, however, unstructured meshes are rarely symmetric.

17. Now the model is ready for volume meshing. To show how the volume meshing parameters influence the final mesh, two cases are considered here.

- Select the surface mesh and type **\_GVol** or click on the  icon in the *Griddle* toolbar. Select *MeshSettings* → *Mode=HexDom*, leave all other parameters at the default settings and press **Enter** twice. Keep the output format as *FLAC3D Binary*.

The resulting *FLAC3D* grid is shown in Figure 16. The grid is of good quality, but the soil layers have a noticeably sharp change in zone size closer to the shaft. This can be improved by varying the *GVol* parameters.

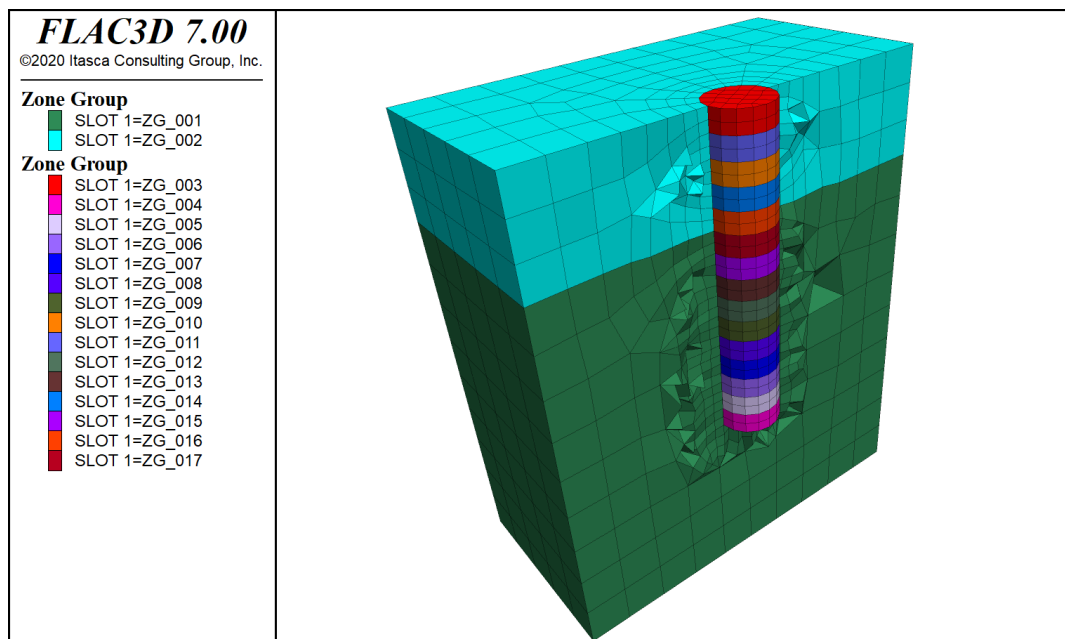


Figure 16: *FLAC3D* grid of the shaft model when using default *GVol* parameters.

- Create a new volume mesh using the following *GVol* parameters:  
*MeshSettings* → *Mode=HexDom*, *MaxGradation=1*, *TargetSize=5*, *Optimization=10*,  
*ShapeQuality=1*. For the explanation of each parameter, refer to the *Griddle 2.0 User Manual*.  
The resulting mesh is shown in Figure 17.

The grid surrounding the shaft in Figure 17 has many more zones, and the transition from coarse to finer zones is more gradual. Typically, such results are achieved when *MaxGradation* is set to a small value ( $< 0.5$ ), but in this case, as the domain boundaries are relatively close to the shaft, a larger value of *MaxGradation* has to be used to force the mesher to quickly change from a large zone size on the boundary of the domain to a finer size specified by the *TargetSize*. Use of the highest possible values for *Optimization* and *ShapeQuality* allows for improving zone quality.

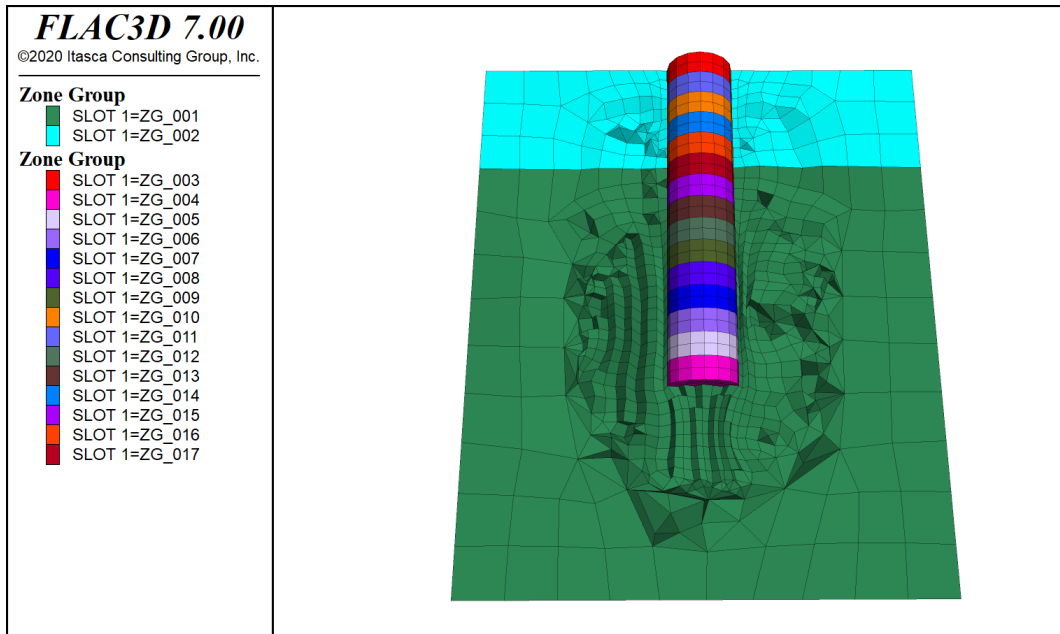


Figure 17: *FLAC3D* grid of the shaft model when using custom *GVol* parameters.

Excavation of the shaft can be modeled by gradual removal of the stages inside the shaft (i.e., zone groups ZG\_003 – ZG\_017 as in Figure 16 and Figure 17).

## Tutorial 3: Meshing a Model with Analytical Surfaces

Expected work time: 1 hour

This tutorial describes two meshing approaches for a model containing analytical objects, such as *Rhino* (poly)surfaces and solids defined by NURBS (non-uniform rational basis spline), SubD (high-precision Catmull Clark subdivision surfaces; only in *Rhino 7* and later) or BRep (boundary representation).

1. Open project “T3\_BridgeSupport.3dm” located in folder “TutorialExamples\3\_BridgeSupport”. Create a copy of this initial project with a different name at a desired location (use **File** → **Save As**).
2. The model represents a part of a complex bridge support structure as shown in Figure 18. The model consists of 13 surfaces, seven polysurfaces, and two extrusions represented by BRep objects. Note how the objects are split by each other (e.g., the top light blue surface is split in three pieces); this is done for better meshing results.

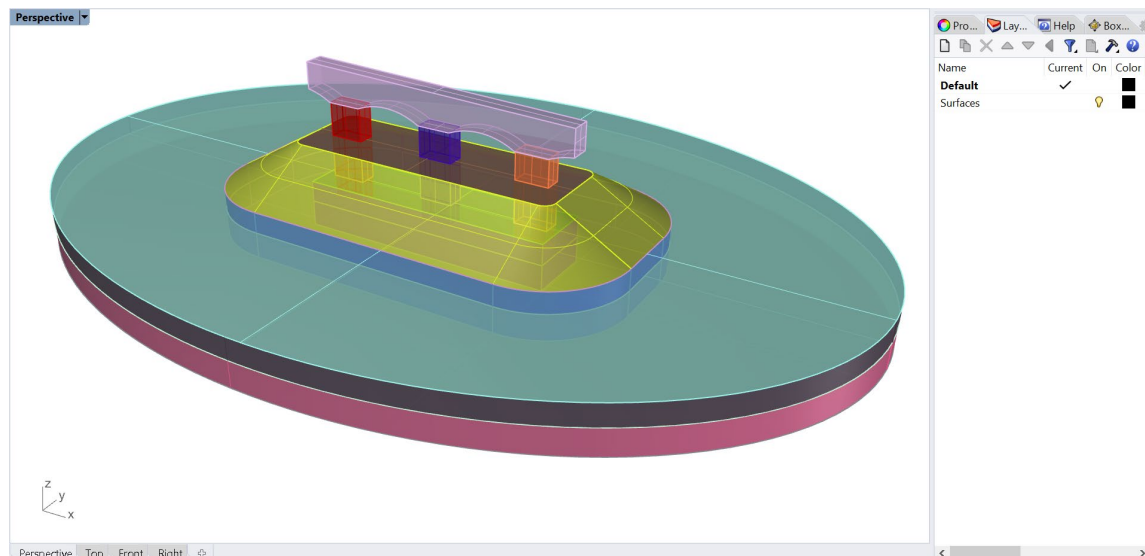


Figure 18: Model of a bridge support structure.

The typical approach to construct meshes when starting with initial (poly)surfaces such as in Figure 18 consists of the following:<sup>1</sup>

- Mesh (triangulate) all the (poly)surfaces to create initial rough meshes; if initial meshes are provided, this step is skipped.
- Intersect the initial meshes with *Griddle*’s surface mesh intersector **GInt**.
- Remesh all meshes with surface remesher **GSurf**.
- Create volume mesh with unstructured volume mesher **GVol**.

Users should follow this general approach in the majority of cases. However, in certain situations, a slight variation of this approach may be more efficient in preparing the model for volume meshing. The standard approach is described in the first sub-section of this tutorial, and potential issues are outlined.

---

<sup>1</sup> See also *Griddle 2.0 User Manual*, Section “Using Griddle Tools for Unstructured Meshing”.



The second sub-section presents a variation of the approach, which can be applied to models similar to the one described in this Tutorial.

### Mesh generation using *GInt* and *GSurf* (approach 1)

3. Maximize *Perspective* viewport and switch to Shaded view for better visibility.
4. Navigate to the *Layers* pane and create a new layer “*Meshes\_Approach1*”. If the pane is not visible, it can be opened from **Edit** → **Layers** → **Edit Layers...**.
5. Select all objects (**Ctrl+A**) and type the **\_Mesh** command to create the initial mesh of all objects. Use detailed controls with the following settings to construct a fine initial mesh (Figure 19).

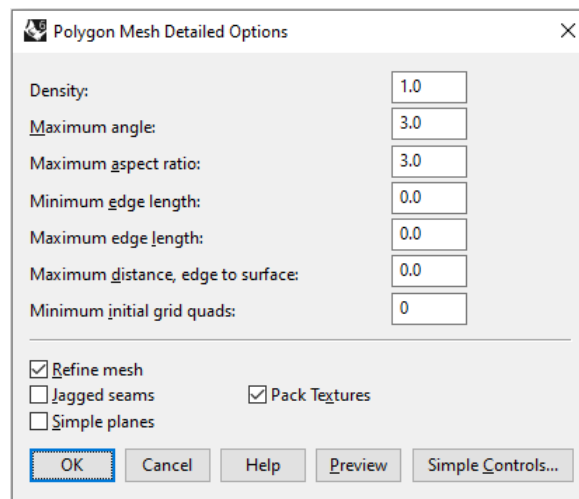


Figure 19: *Rhino* meshing dialog to create fine initial mesh.

6. While all the (poly)surfaces are selected, invert the selection by typing the command **\_Invert**, which will select meshes only. Navigate to the *Properties* pane (or press **F3**) and change the mesh layer to “*Meshes\_Approach1*”. Open the *Layers* pane and turn off the “*Surfaces*” layer (click on the lightbulb).

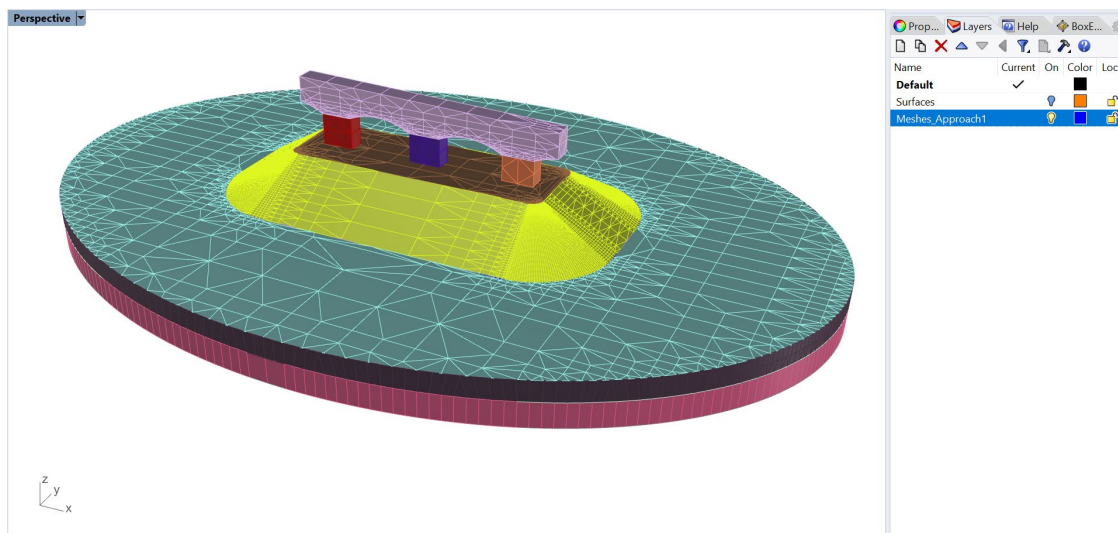


Figure 20: The initial *Rhino* mesh.

The mesh in Figure 20 is not conformal and contains gaps along the curved boundaries (Figure 21). This is due to fact that surface meshes are a discrete representation of the initial analytical (in this case, BRep) surfaces, and the mesh for each object is created independently from any other objects. The gaps and non-conformal faces can be fixed by intersecting meshes using the **Glnt** surface mesh intersector with a properly selected tolerance. Having a watertight surface mesh domain is a required condition for volume meshing. At the same time, making all model meshes fully conformal is usually needed but is not always required for volume meshing (e.g., separate watertight domains must contain conformal surface meshes but the domains themselves may be non-conformal in the areas of contact).

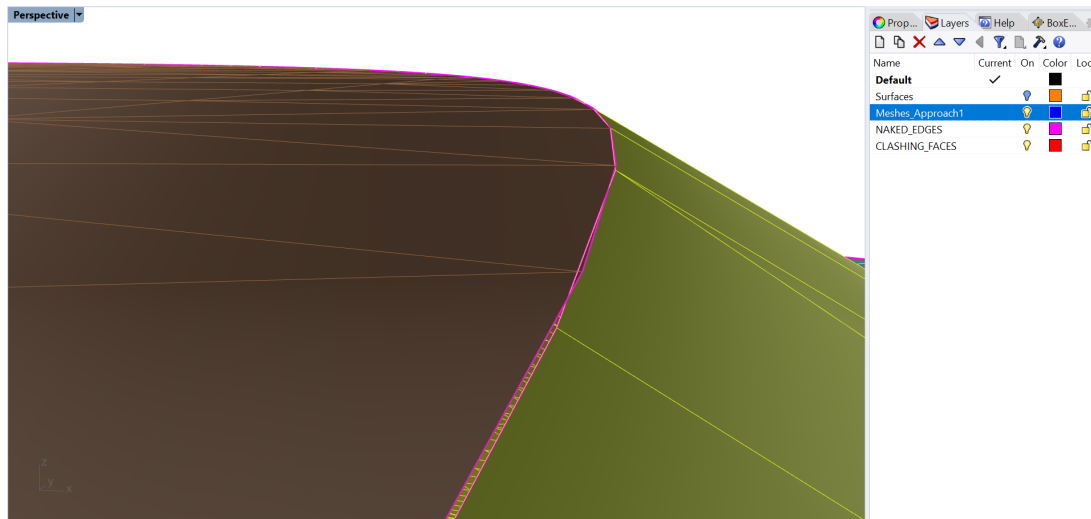




Figure 21: Mismatching mesh boundaries in the initial *Rhino* mesh.

In certain cases, it may be challenging to determine a proper tolerance value when intersecting many meshes. This is especially true for big models with large variations in face (element) sizes and with gaps/mismatches between neighboring faces that are of similar size as some of the mesh faces. The reason for this challenge is that a small **Glnt** tolerance may not be sufficient to close the gaps and intersect all the faces to make them conformal, while a large tolerance may create undesired intersections and changes of smaller faces that fall within the tolerance. Therefore, it is recommended to do mesh intersections with minimal acceptable tolerance to avoid creating incorrect intersections. Users may need to do several trials starting with 0 tolerance and gradually increase it until all faces are properly intersected. This process is outlined below.

7. Select all surface meshes and type **\_Glnt** or click on the  icon in the *Griddle* toolbar. For the first trial, keep *Tolerance* = 0, and click on *AdvancedParameters*. By default, **Glnt** preserves initial mesh shapes and other parameters, which is set by the option *OutputMesh* = *Separated*. In this case, the output mesh needs to be merged to clearly identify if there are any gaps or non-conformal faces between the meshes. Click on *OutputMesh* to set it to *Merged*<sup>2</sup>. Press **Enter** twice to execute **Glnt**.

<sup>2</sup> When setting *OutputMesh* = *Merged*, **Glnt** intersects all selected surface meshes and creates a single merged mesh assigned to the default (current) layer. Merged meshes have a smaller number of boundary edges and allow for easy identification of holes, gaps, or non-conformal faces within the mesh.

8. To check if there are any gaps or non-conformal edges/faces left, select the output mesh and type **\_GHeal** or click on the  icon in the *Griddle* toolbar. Select the *ShowErrors* option, which will create two new layers, “NAKED\_EDGES” and “CLASHING\_FACES”, containing outlines of open mesh boundaries (e.g., internal holes) and non-conformal faces. The results show that there is a large number of gaps and some non-conformal faces (Figure 22 and Figure 23). This is because **GInt** *Tolerance* was set to 0 to intersect only those mesh faces that are in full contact. Naked edges can also be visualized with *Rhino*’s **\_ShowEdges** command.

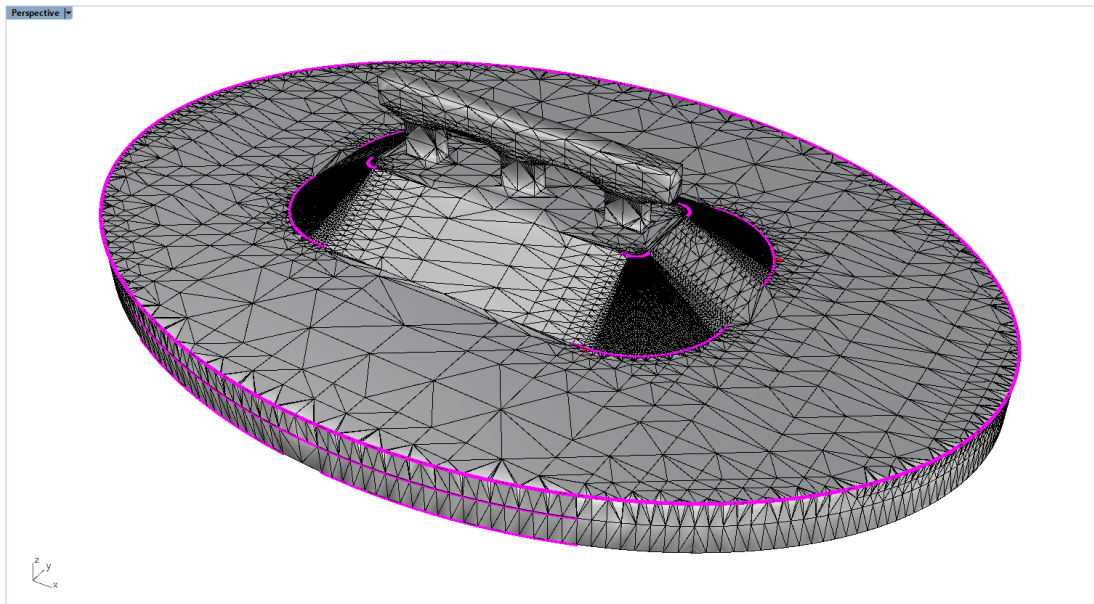


Figure 22: Naked edges (in pink) and clashing faces (red) after using *GInt* with *Tolerance* = 0.

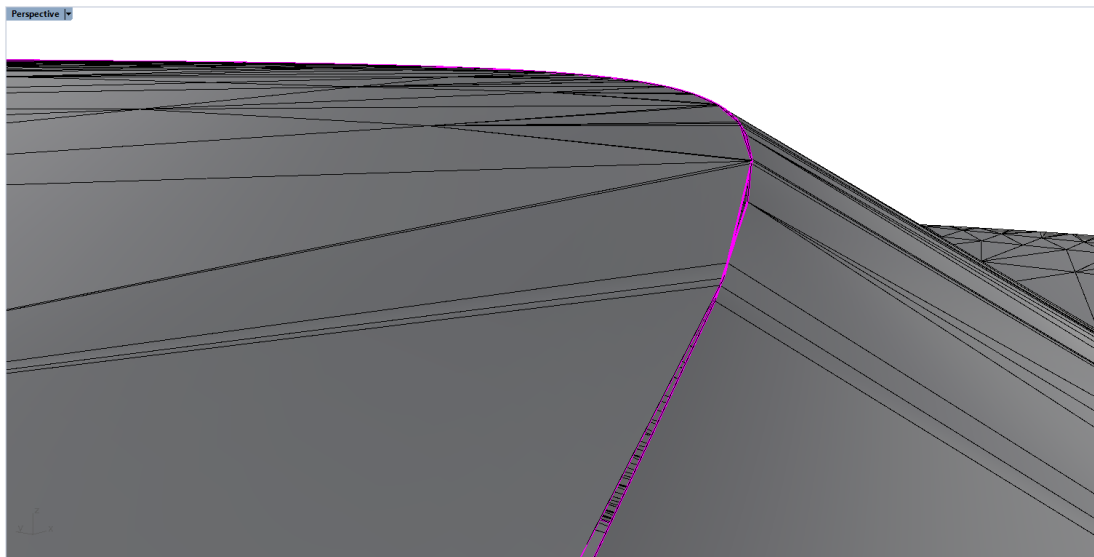


Figure 23: Zoomed naked edges (in pink) after using *GInt* with *Tolerance* = 0. Compare with Figure 21.

9. Undo the last two commands (**\_GHeal** and **\_GInt**) by pressing **Ctrl+Z** twice. Ensure the initial **\_Mesh** command is not undone.

10. Select all initial surface meshes and run **GInt** again. This time specify *Tolerance* = 0.005 and keep all other parameters the same as in the previous run. After **GInt** executes, visualize naked edges and clashing faces as outlined previously in step 8. The results show that there are still some naked edges and clashing faces (Figure 24), which indicates that **GInt** *Tolerance* is still not sufficient.

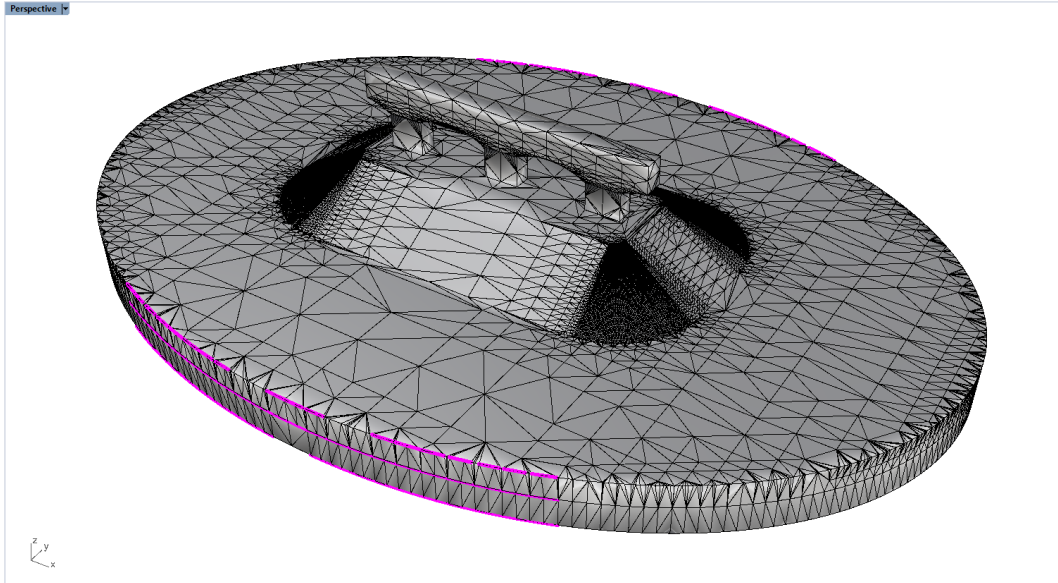




Figure 24: Naked edges (in pink) and clashing faces (red) after using **GInt** with *Tolerance* = 0.005. Compare with Figure 22.

11. Repeat steps 9 and 10 using *Tolerance* = 0.015. The output mesh does not have naked edges or clashing faces, which means that all faces from distinct meshes were properly intersected. To see this, **\_Hide** the mesh and select all objects (**Ctrl+A**). *Rhino* should report that no objects are selected.
12. Undo the last operations (**\_Hide/\_Show**, **\_GHeal**, **\_GInt**) by pressing **Ctrl+Z** until the initial *Rhino* mesh is shown as in Figure 20.
13. Run **GInt** again with *Tolerance* = 0.015 and *OutputMesh* = *Separated* (in *AdvancedParameters*). Note that this time all distinct meshes are preserved, and it would be hard to check for naked edges as each mesh has its own boundary. That is why a merged mesh was used in the previous steps.
14. Select all surface meshes and remesh with **GSurf** (  ) using the following parameters: *Mode* = *QuadDom*, *MinEdgeLength* = 0.25, *MaxEdgeLength* = 5, *RidgeAngle* = 5, *AdvancedParameters* → *MaxGradation* = 0.06, and keep all other parameters at the default values. The resulting meshes are shown in Figure 25. These meshes look good overall, but zoomed views of the curved bottom / top embankment sections reveal (yellow-green mesh) that these areas contain small elements (faces) that do not fully conform to the initial shape (Figure 26). This is due to independent meshing of the initial BRep surfaces and due to using a large tolerance in **GInt**<sup>3</sup>.

<sup>3</sup>Another potential issue can be revealed by completing step 14 with *OutputMesh* = *Merged*. The single output mesh may contain a few overlapping (clashing) faces that were introduced by **GSurf**. However, these clashing faces do not affect the ability to create a volume mesh, as they are between separate watertight sub-domains. In general, clashing faces within a single mesh can be fixed using **GHeal**'s *FixClashingFaces* function, which locally intersects them.

15. Create a volume mesh using **GVol** (  ) and any desired (or default) parameters. **GVol** automatically checks input surface meshes for the presence of naked edges (e.g., holes, gaps) and clashing (non-conformal) faces to assure that input meshes create a watertight domain; if any such issues are found, an information message is shown. The message does not prevent **GVol** from running (click **Yes** to proceed); it is only a reminder for users to verify that input meshes are valid. In this case, **GVol** input consists of multiple surface meshes with boundaries treated as naked edges.

The initial error check and the message can be turned off by setting *IniErrorCheck* to *No* (in **GVol**'s *MeshSettings* options).

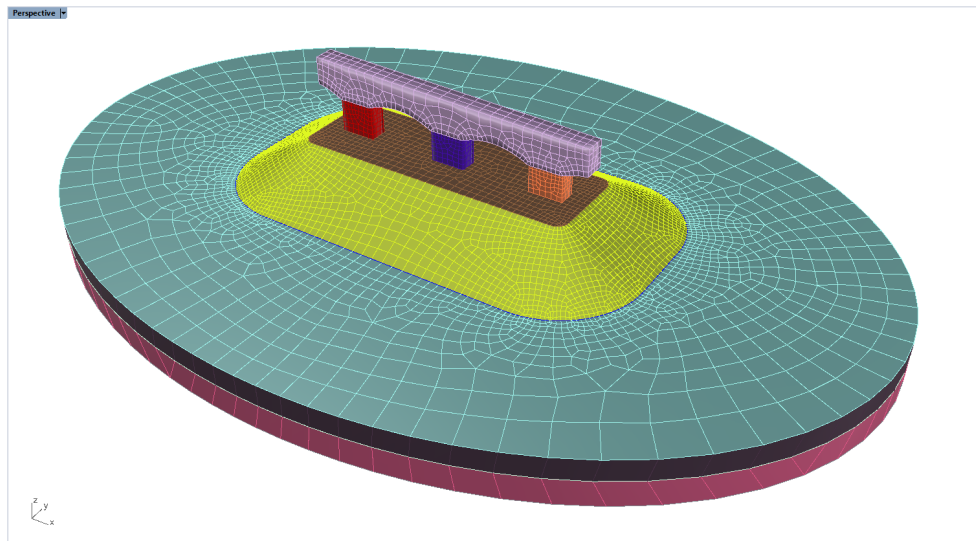


Figure 25: Remeshed surface meshes using standard approach.

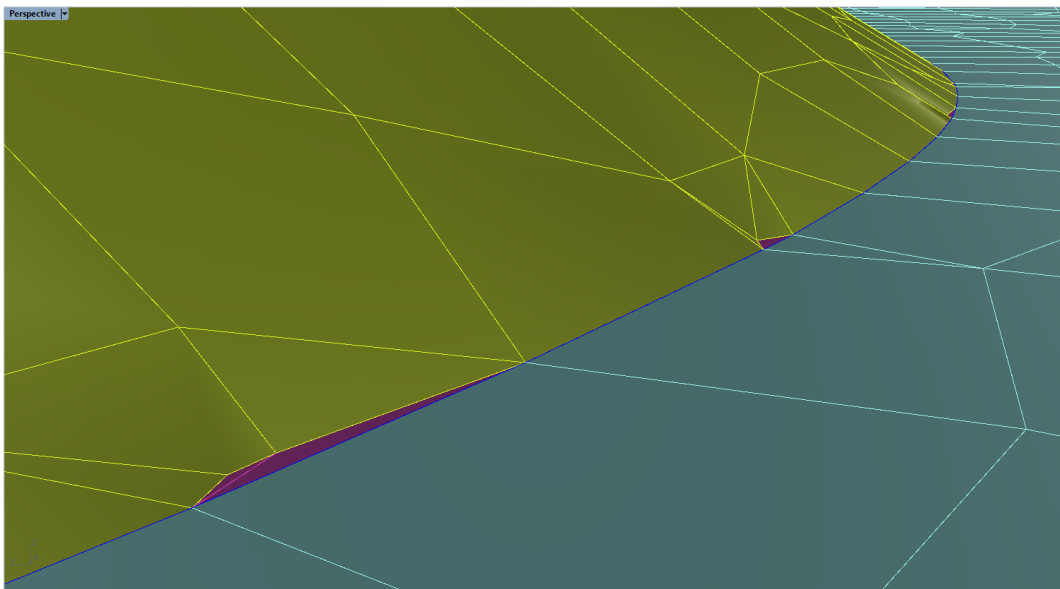



Figure 26: Slightly deformed surface meshes when using standard approach.



## Mesh generation using *NonManifoldMerge* and *GSurf* (approach 2)

The issues in the previous approach can be avoided if all initial (poly)surfaces are merged into a single non-manifold polysurface before meshing. The initial meshing (triangulation) of all connecting surfaces within a single object happens in a conformal manner, and no gaps are created between mesh faces. Therefore, with this approach, surface mesh intersection is often not needed. The drawback of this approach is that when the initial objects are merged into a single non-manifold polysurface, user-specified object information is lost (e.g., surface names, colors, etc.). However, if these parameters are not important, the approach outlined below may be much easier, more efficient, and yield better quality meshes.

16. Navigate to the *Layers* pane and create a new layer named “Meshes\_Approach2”. Turn on layer “Surfaces” to show the initial surfaces.  
Delete layers (if any) “NAKED\_EDGES”, “CLASHING\_FACES”, and “MESHING\_ERRORS”.
17. Select all objects in layer “Surfaces” (as in Figure 18) and use command **\_NonManifoldMerge** (  ) to create a single non-manifold polysurface (Figure 27). The new polysurface is placed into the current layer (“Default”); move it back to the “Surface” layer by changing the object layer assignment in the *Properties* pane (**F3**).

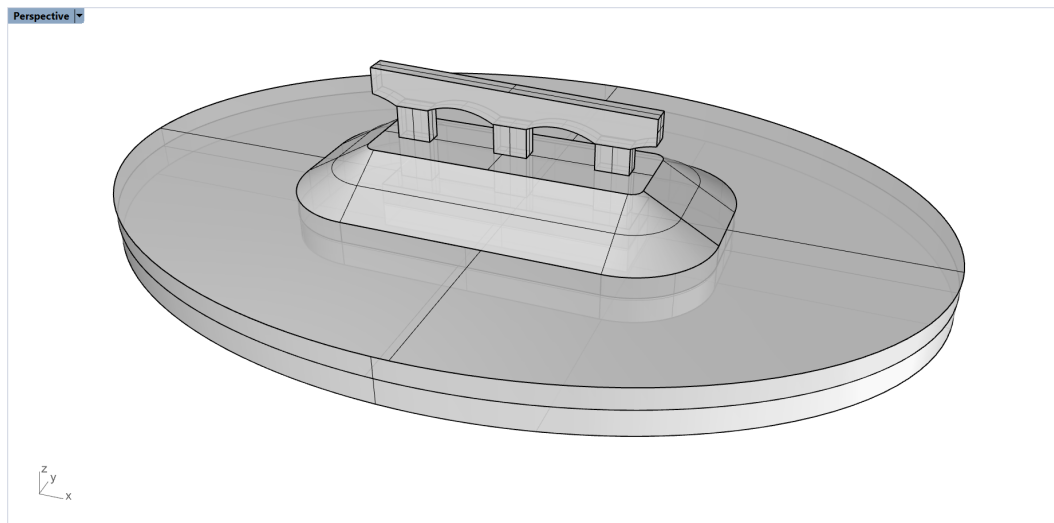




Figure 27: Single polysurface constructed by the *NonManifoldMerge* command.

18. Mesh the polysurface using the **\_Mesh** command with the same settings as in Figure 19. Note that the default mesh settings and *Simple Controls* can be used in this case, as there will be no issues with gaps or non-conformal faces (there is no need to have a highly detailed initial mesh).
19. Assign layer “Meshes\_Approach2” to the newly created mesh and turn off the “Surface” layer so only the mesh is present in the viewports.
20. Remesh the initial mesh with **GSurf** (  ) using the same settings as in Step 14. The quality of elements (faces) around the curved parts of the embankment is much higher if compared with the mesh in Figure 25 (or with the meshes in layer “Meshes\_Approach1”).

21. Verify that there are no naked edges (gaps, holes) or clashing faces in the final surface mesh using **GHeal** (  ) → *ShowErrors*. There should be no curves outlining any problems in layers “NAKED\_EDGES” and “CLASHING\_FACES”.

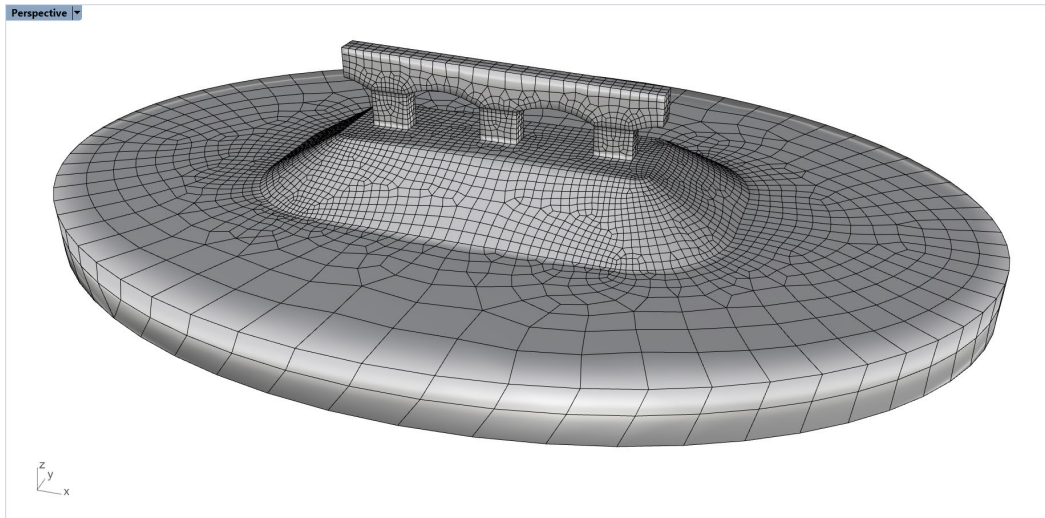





Figure 28: Remeshed surface mesh in the second approach.

22. The mesh is ready for volume meshing. Run **GVol** (  ) with any desired (or default) parameters. Due to the higher quality of input surface mesh(es), the total number of elements in the output volume mesh is smaller in this approach than in the previous one (e.g., for Hex-dominant mesh, this approach produces in total ~47,000 elements and the first approach results in ~69,000 elements).

As mentioned previously, the issue of this approach is that all initial (poly)surfaces must be merged into a single nonmanifold polysurface before meshing. If there is a need to assign a name, color, or any other property to a specific surface mesh before volume meshing, the merged surface mesh can be easily split into sub-meshes using *Griddle*’s tool **GExtract**.

23. Select the merged surface mesh (as in Figure 28) and type the **\_GExtract** command or click on the  icon in the *Griddle* toolbar. Select option *AllSurfaces* and use *MaxBreakAngle* = 180 to separate meshes only along non-manifold edges (the break angle will not have an effect as it is set to 180°). Alternatively, if break angle = 89° is used, all meshes that connect at an angle > 89° will also be separated (thus, meshes that are perpendicular to each other will be separated).
24. Select all (36) meshes and use *Griddle*’s command **\_ColorizeObjects** or click on the  icon in the *Griddle* toolbar. This will assign a unique color to each *Rhino* object (Figure 29).
25. Now a name can be assigned to each surface mesh if there is a need to transfer surface mesh names into a volume mesh (available only for *FLAC3D* and *3DEC* output formats in **GVol**).



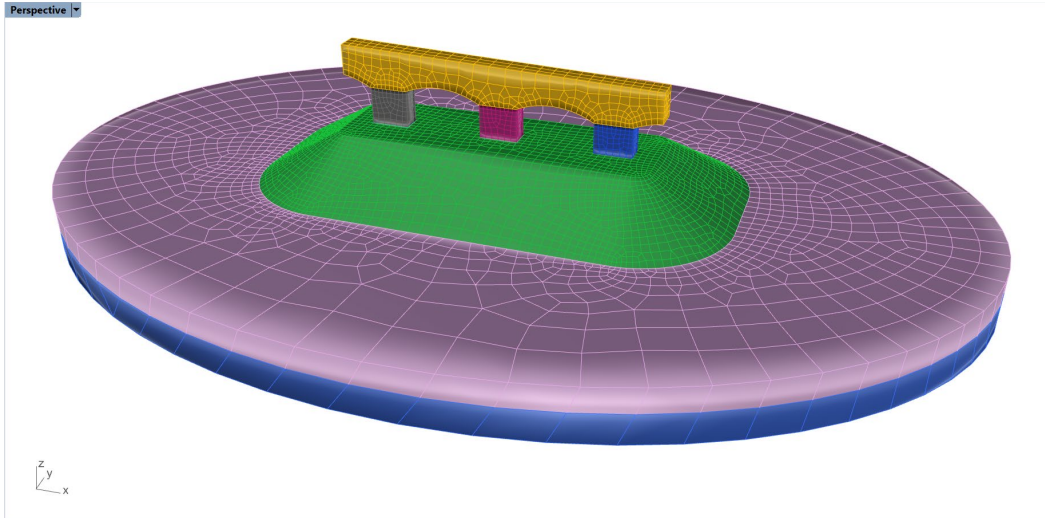


Figure 29: Mesh from Figure 28 “exploded” into sub-meshes, which are subsequently colored.

## Tutorial 4: Creating a Structured Mesh with *BlockRanger*

*Expected work time: 1–2 hours*

This tutorial provides detailed information on how to build a block-structured hexahedral mesh from an initial CAD model provided by a DXF file. To create a mesh using **BlockRanger** (**\_BR** command), an assembly of six-, five-, or four-sided *Rhino* solids that conform to the reference model must be created. The creation of such an assembly is the objective of this example. The reference model is shown in the left side of Figure 30. The final hexahedral mesh is depicted to the right. This 3D slope model has a shape similar to a bathtub; hence, it is sometimes referred to as the “bathtub” model.

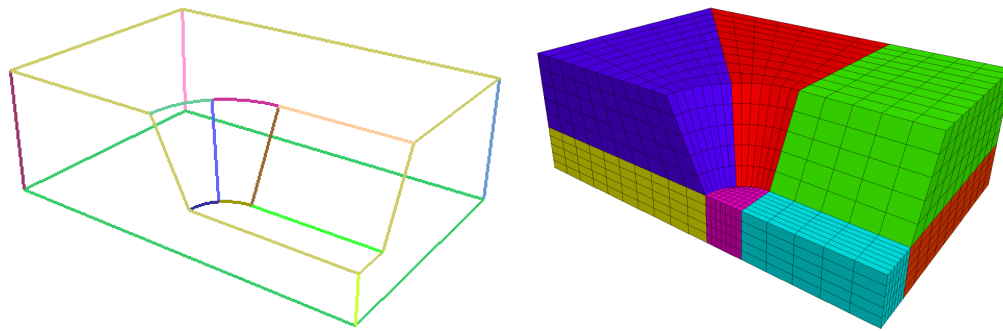


Figure 30: Reference model (left) and *BlockRanger* generated mesh (right).

### Importing the geometry

1. Start *Rhino*, select **Large Objects, Meters** (similar method as step 1 in Tutorial 1) and save the *Rhino* project at a desired location.
2. Import the initial geometry from a DXF file by navigating to **File** → **Import** and select “T4\_3DSlope.dxf” located in folder “TutorialExamples\4\_StructuredMesh\_SlopeModel”. In the **Import** dialog, select **Model** → **Meters** and **Layout Units** → **Millimeters**, and keep other parameters at the default values.
3. Maximize *Perspective* viewport and make sure Shaded view is selected.
4. In the top menu, navigate to **View** → **Display Options...** This will open the **Rhino Options** dialog; select **View** → **Display Modes** → **Shaded** → **Objects** → **Curves** and set **Curve Width** to 4. This will display all curves thicker in the Shaded view. The initial settings can be always restored by clicking on **Restore Defaults**.
5. Navigate to the *Layers* pane as shown in Figure 31. If the pane is not visible, open it by going to **Edit** → **Layers** → **Edit Layers...** In the *Layers* pane, rename the automatically created layer “lines” to “Reference” and delete all layers except “Default” and “Reference”. Ensure that all objects are now in layer “Reference” by selecting all objects and navigating to the *Properties* pane (or press **F3**).
6. Select all objects and type the command **\_Move**. For the *Point to move from*, select the bottom-left corner of the model and for the *Point to move to*, type 0. This will move all objects and align the bottom-left corner of the model with zero coordinates. Compare the results with Figure 31.

It is always recommended to move/center objects imported from a DXF (or other formats) closer to zero coordinates as it improves the accuracy of *Rhino* operations.

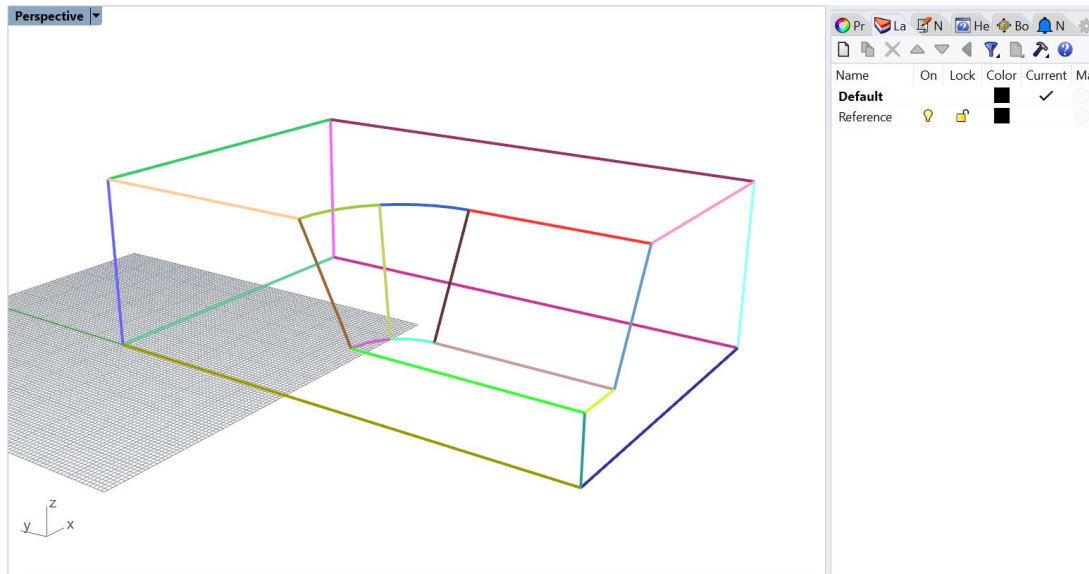


Figure 31: The reference model.

## Choosing the right mesh block layout for the model

**BlockRanger** operates on solids and meshes each solid block individually. The present model may be decomposed into solid blocks in many ways. Some of the possible options are shown in Figure 32 through Figure 35. The arrangement of blocks depicted in Figure 34 will be used in this tutorial.

Each of the blocks shown in Figure 32 through Figure 35 satisfy **BlockRanger** requirements for solids, which includes the following:

- Permissible types of solids:
  - four-sided solids made of three-sided faces (a topological tetrahedron);
  - five-sided solids made of two three-sided faces connected through three four-sided faces (a topological triangular prism); and
  - six-sided solids made of four-sided faces (a topological hexahedron).
- Faces must be simple faces that cannot be further “exploded” into simpler faces.
- Face edges must only be simple curves that cannot be further “exploded” into simpler curves.

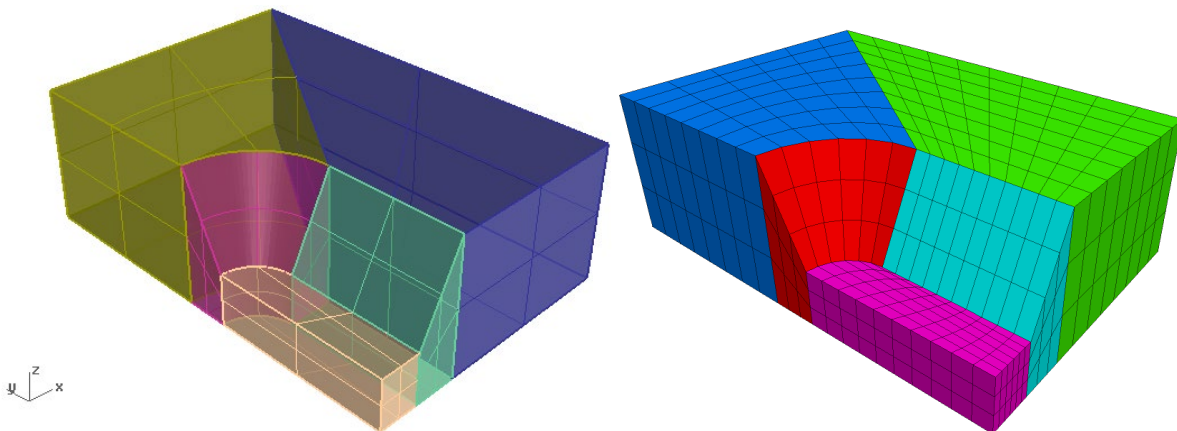


Figure 32: Solid layout resulting in five mesh blocks.

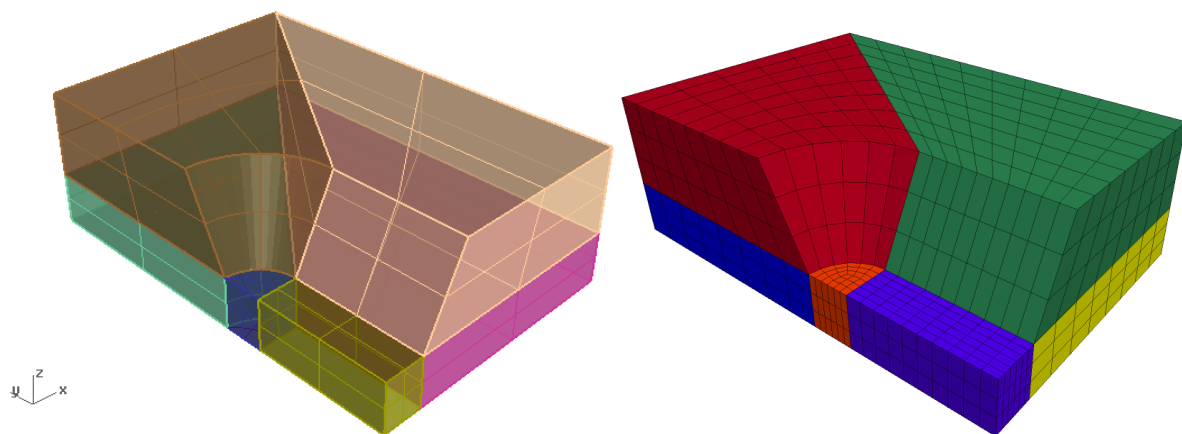


Figure 33: Solid layout resulting in six mesh blocks.

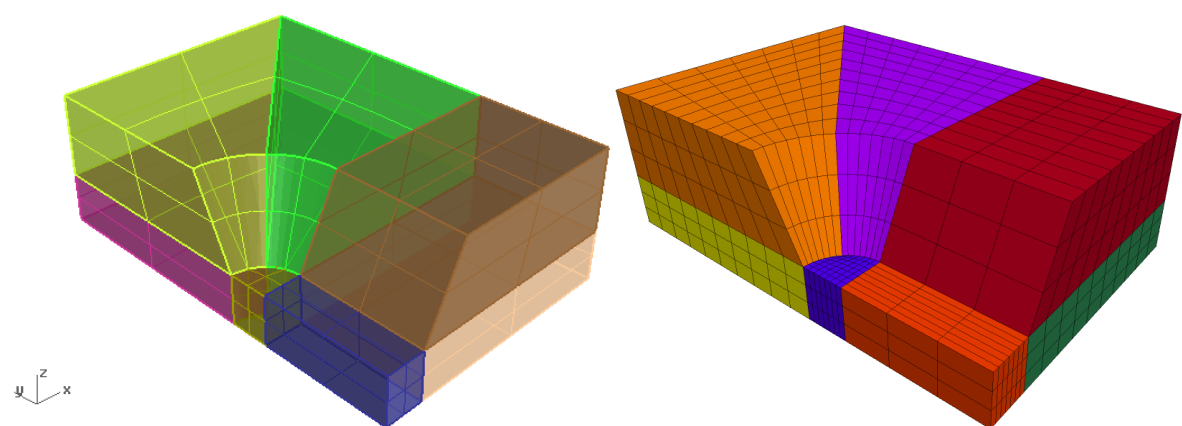


Figure 34: Solid layout resulting in eight mesh blocks.

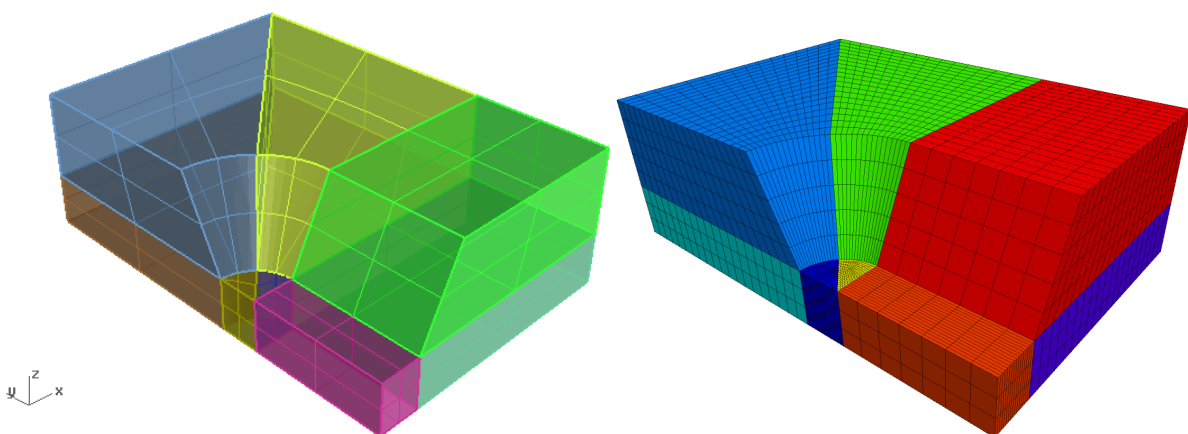


Figure 35: Alternate solid layout resulting in nine mesh blocks.

## Building solids from curves

- Click on **Osnap** (Object Snap) at the bottom of the screen (Figure 36) to highlight it. Make sure that all checkboxes designating what to snap to are checked, as shown in Figure 36. **Grid Snap** may be turned off.

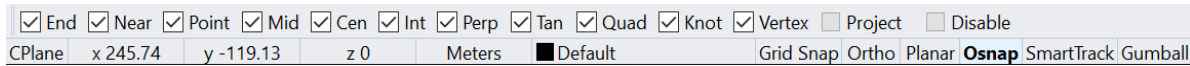


Figure 36: Rhino status bar.

- Starting with the reference model presented in Figure 31, create new construction lines as shown in Figure 38. Use the **\_Line** command and start by connecting nodes from the middle of the model (where the curved segments are) to the sides of the model. Pay attention to when the mouse tooltip shows “Perp” or “Perp, Int” (or other types of intersections), which means that the line under construction is perpendicular to (and intersects) another line.

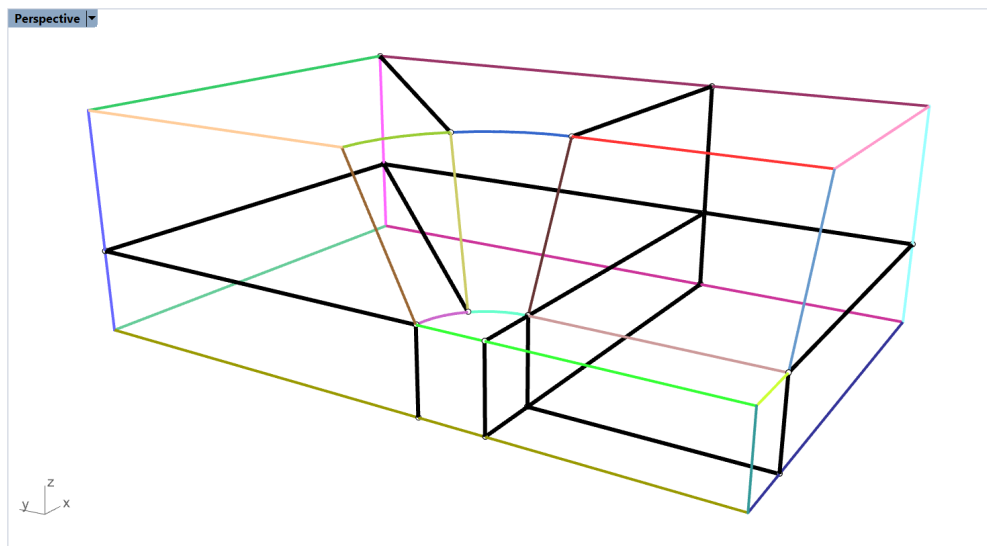


Figure 37: Creating new construction lines (in black).

The curves created in the previous step (Figure 31) are polylines, and some of them are composed of multiple linear segments that cannot be used directly to build surfaces. They must be retraced with arcs (**Curve → Arc**) or higher order curves (**Curve → Free-Form → Interpolate Points**) to create simpler two-noded curves.

- Zoom in around the curved segments in the model and select in the top menu **Curve → Arc → Start, Point, End**. Create separate arcs at each curved segment of the model as shown in Figure 38 and Figure 39. In total, four arcs should be created. Pay attention to when the mouse tooltip shows “End, Int, Knot” when connecting to the points in the middle of the curved segments. Such a tooltip means the mouse is at the end of a line, hit an intersection with another line, and is at a knot.
- After the arcs are created, the initial polylines that have been retraced can be deleted.

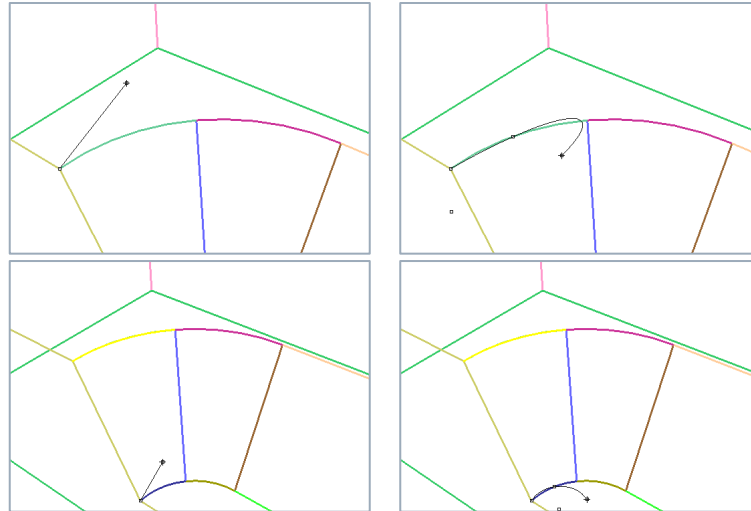


Figure 38: Building the upper (top) and lower (bottom) arcs.

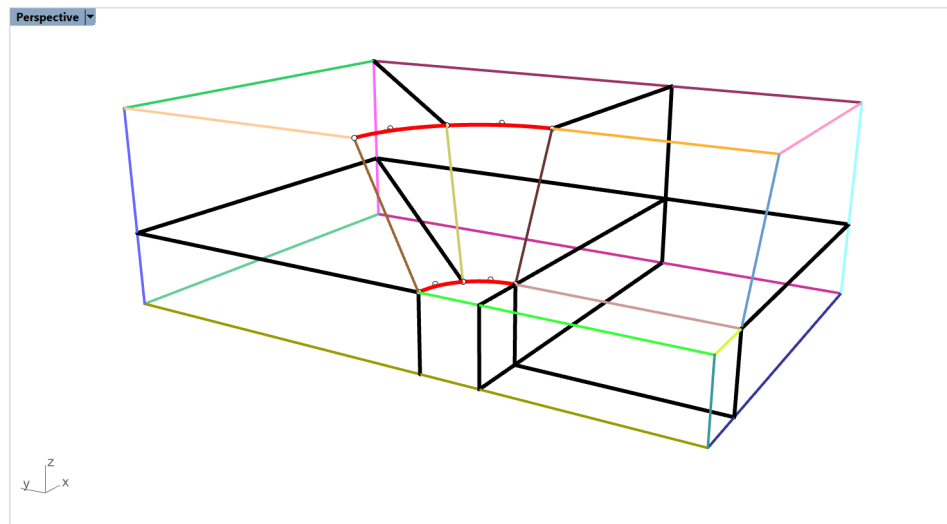


Figure 39: Red arcs retrace the original polylines at the curved section of the model.

11. Select the bottom two arcs and copy them using the **\_Copy** command. For *Point to copy from*, select the left corner of the arcs as shown in Figure 40, then, within command options, click on *Vertical* to set it to *Yes*, and drag the corner down until it hits the bottom line and the mouse tooltip is as in Figure 40.

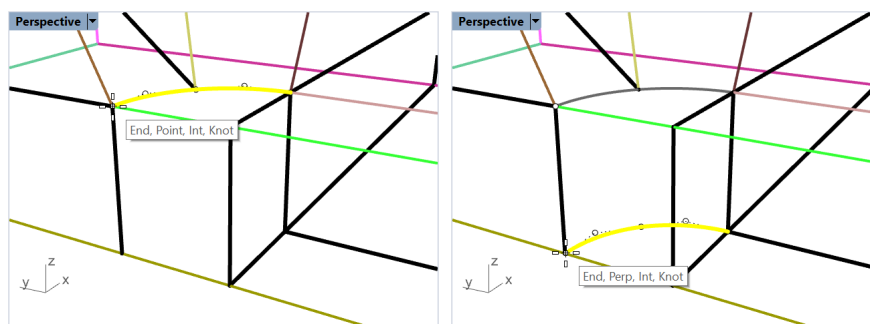


Figure 40: Copying the arcs.

12. Connect the middle nodes as shown in Figure 41 with the dark blue line. Connect the bottom end of the dark blue line to the opposite corner as shown in Figure 41 with the light blue line.

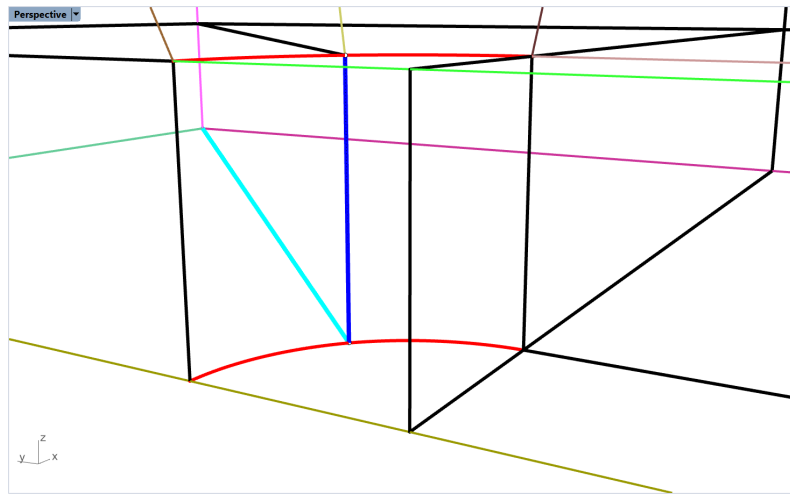


Figure 41: Connecting the middle ends of the arcs (blue lines).

13. Split all intersecting lines (mostly those from the reference model) by using the **\_Split** command. For this, first select the line to split and then select a line that intersects or connects to it. For example, the tan line at the bottom of Figure 41 can be split in three pieces by the connecting black lines. The resulting wireframe should contain 56 curves (**Ctrl+A** and check the number of selected objects) and is the basis for creating solid blocks (Figure 42).

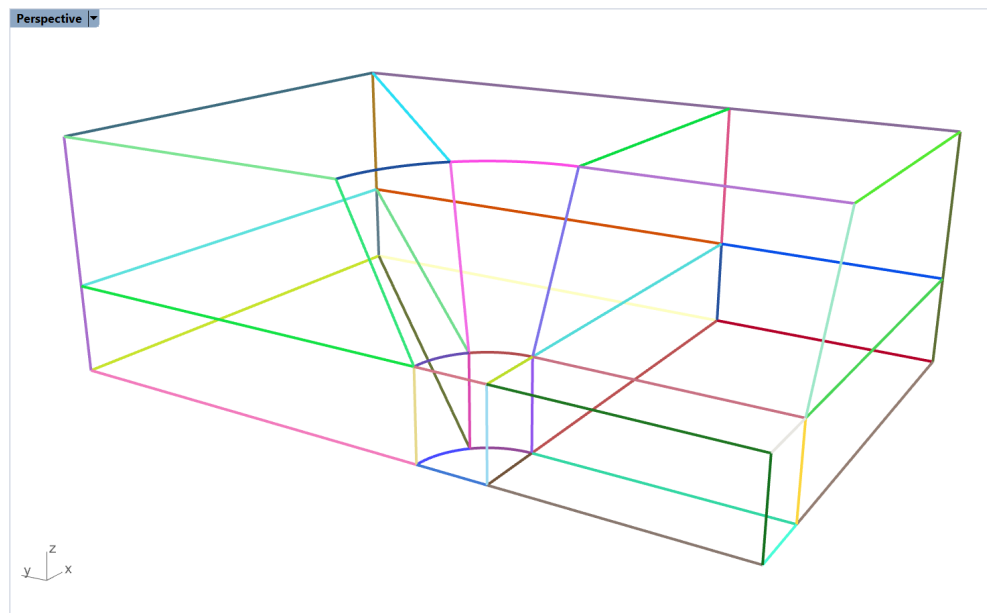


Figure 42: Model wireframe ready to be used for block creation.

14. Start creating a block at the top-left side of the model. Select the two arcs and two lines as shown in Figure 43 (selected lines are in yellow), type the **\_Loft** command and use the settings shown in the right side of Figure 43. Click **OK** to build a lofted polysurface between the several curves. Alternatively, a single surface can be built between each pair of curves/lines.



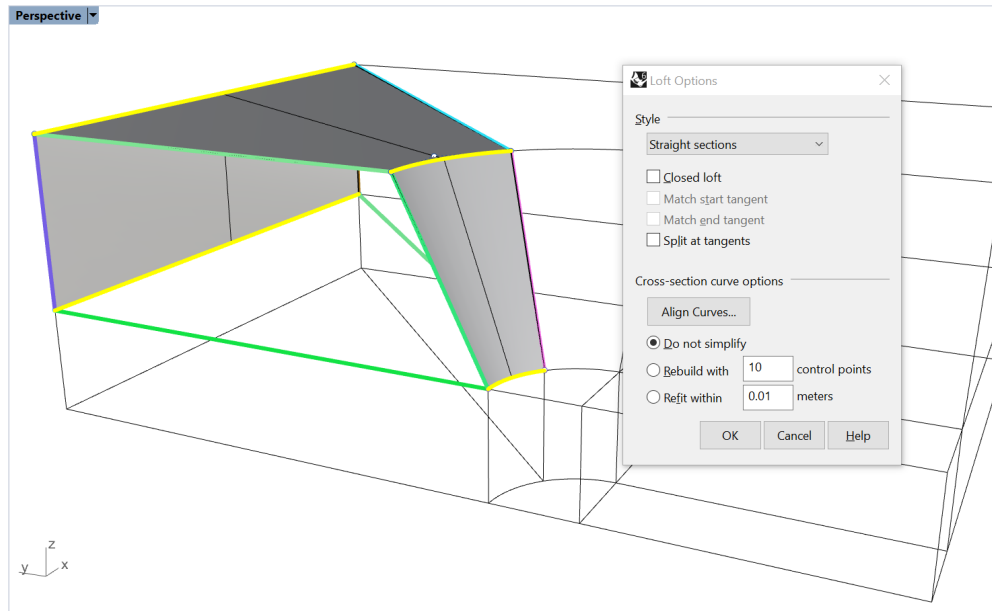


Figure 43: Building the surfaces for the first solid. Note that the grid has been hidden on this image.

15. Now, select only pairs of lines/curves composing the remaining surfaces of the first solid and **\_Loft** them. In total, six sides must be created as in Figure 44.

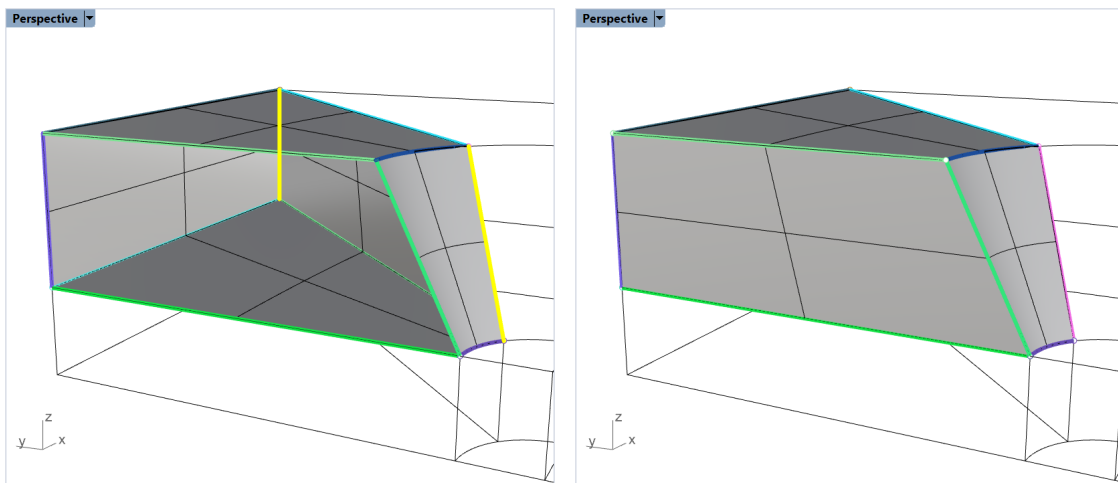


Figure 44: Building surfaces of the first solid.

16. Select all newly created surfaces and a polysurface and join them with the **\_Join** command. This is the first solid block. Hide it with the **\_Hide** command, as it will be easier to create other solids when nothing obstructs the view.
17. In the same manner, build all other solid blocks except for the small central block, as it requires an additional operation (described below). The stages are outlined in Figure 45.

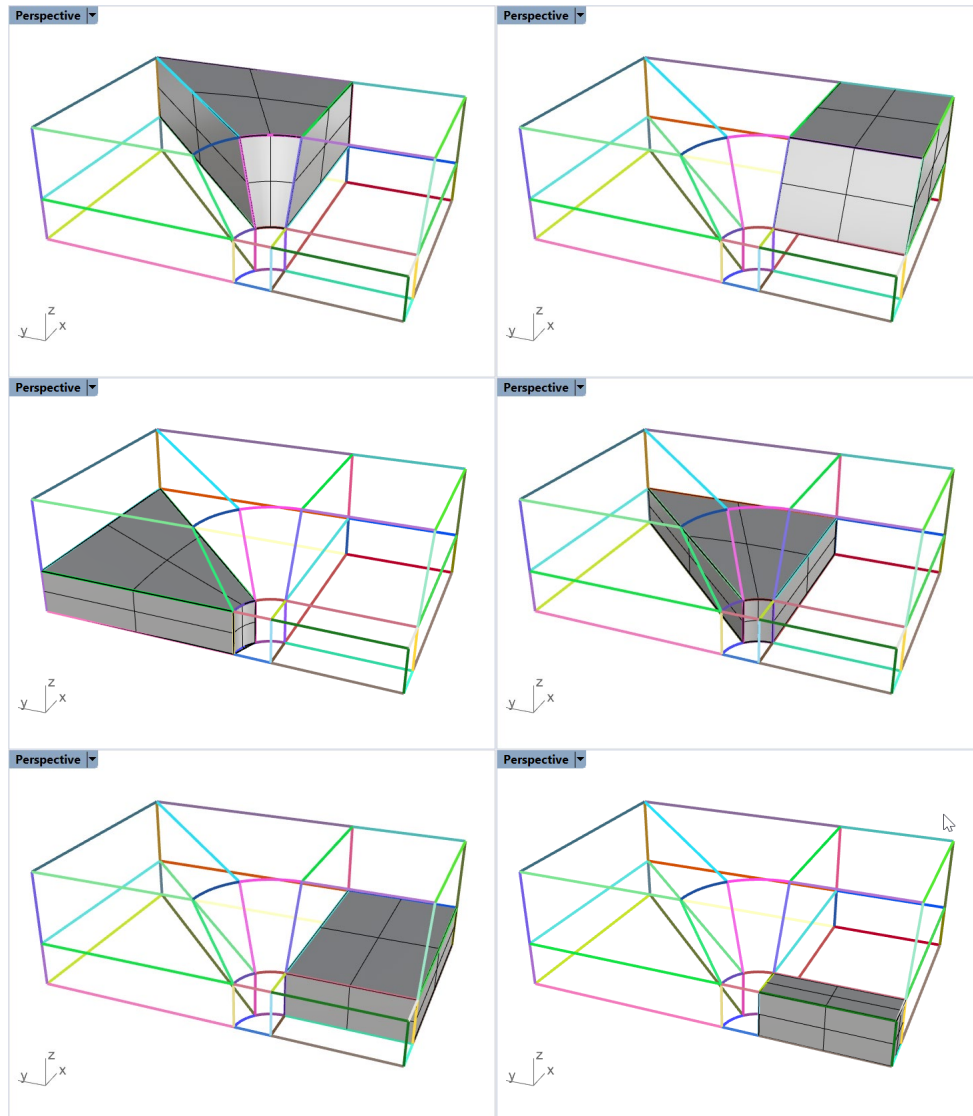


Figure 45: Building solid blocks for the model.

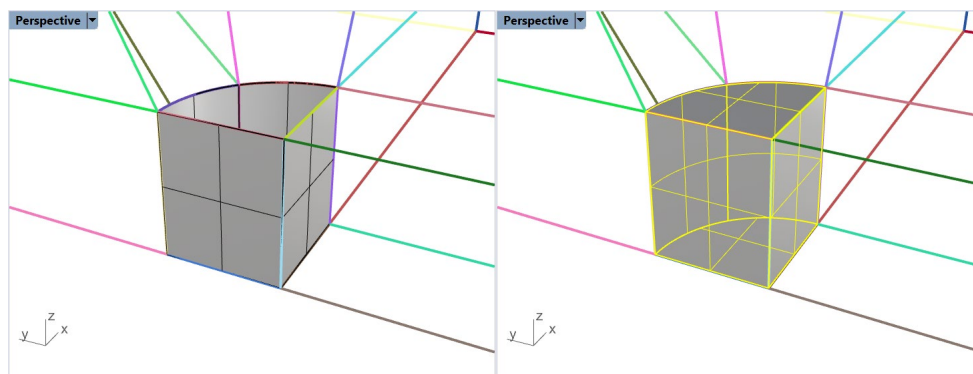


Figure 46: Building the last solid block in the model.

18. For the remaining central block, build only vertical surfaces using the **\_Loft** command and join all four of them with the **\_Join** command (Figure 46, left).
19. Select a single polysurface as in Figure 46, left and cap the holes with the **\_Cap** command (Figure 46, right). The resulting solid is the last block needed before meshing.
20. Show all blocks that were previously hidden with the **\_Show** command. There should be eight solid blocks as in Figure 47.

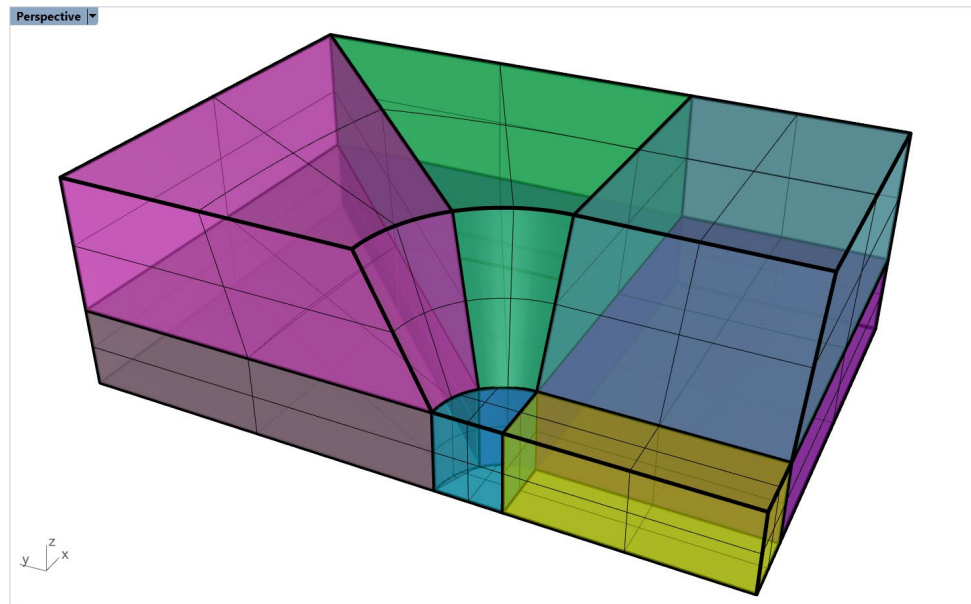


Figure 47: Solid blocks of the model (Ghosted view).

21. Select all curves by typing the **\_SelCrv** command and assign layer “Reference” to them (navigate to *Properties* pane or press **F3**). Hide this layer in the *Layers* pane (press on the lightbulb to turn it off).
22. Select all remaining objects with **Ctrl+A** and verify that *Rhino* reports only eight polysurfaces have been selected. If any surfaces are reported, this is an indication that they were not joined to a solid block. Find and join them so that only eight closed polysurfaces (solids) are present. While the solids are selected, navigate to the *Properties* pane (or press **F3**) and ensure that object *Type* indicates that they are closed polysurfaces (Figure 48). If the *Type* field says “varies”, most likely it means that some solids are not closed (probably incorrect surfaces were created).
23. Create new layers “Upper solids” and “Lower solids” in the *Layers* pane. Assign distinct colors to these layers. Select only upper solids and assign them to the layer “Upper solids” (through the *Properties* pane). Complete the same for the lower solids (Figure 49).

Now all solids are ready for meshing.

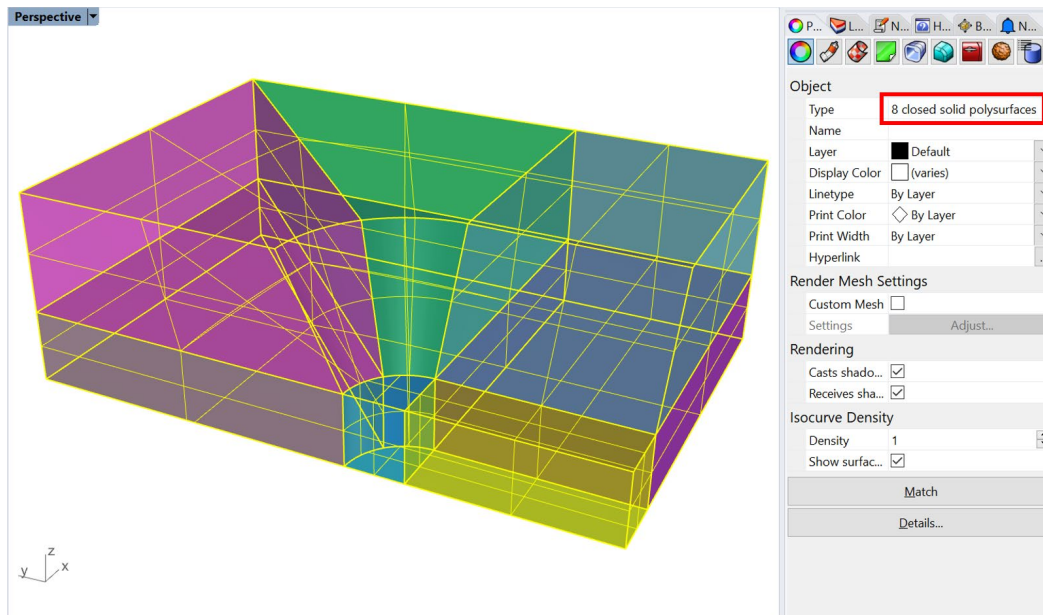


Figure 48: Solid blocks of the model (Ghosted view).

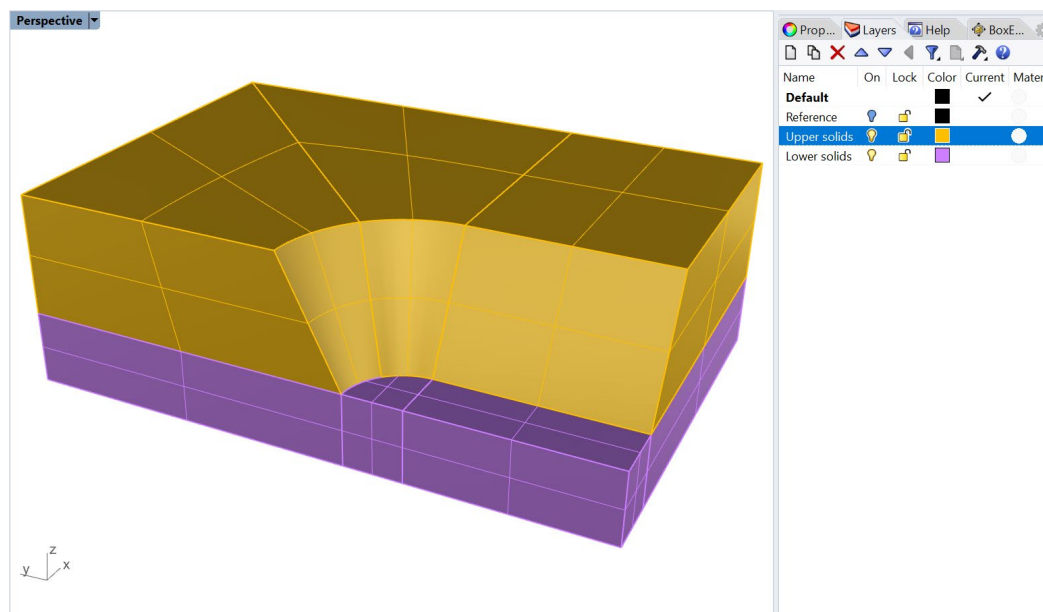



Figure 49: The model after the “Upper solids” and “Lower solids” layers have been specified.

## Meshing with *BlockRanger*

24. Select all solids and type the `_BR` command or click on the  icon in the *Griddle* toolbar. Use the default **BlockRanger** parameters and save the mesh in *FLAC3D* binary format.

In the *Rhino* command area, **BlockRanger** should report: “8 solids processed, 8 solids meshed, 0 errors.” If any of the solids do not satisfy the **BlockRanger** requirements outlined previously, the corresponding solids will remain highlighted in *Rhino* and **BlockRanger** will report that it processed less than 8 solids.

Figure 50 shows a structured *FLAC3D* grid created by **BlockRanger**, as well as Zone groups from SLOT1. SLOT2 groups correspond to the original solid blocks and are shown in Figure 51.

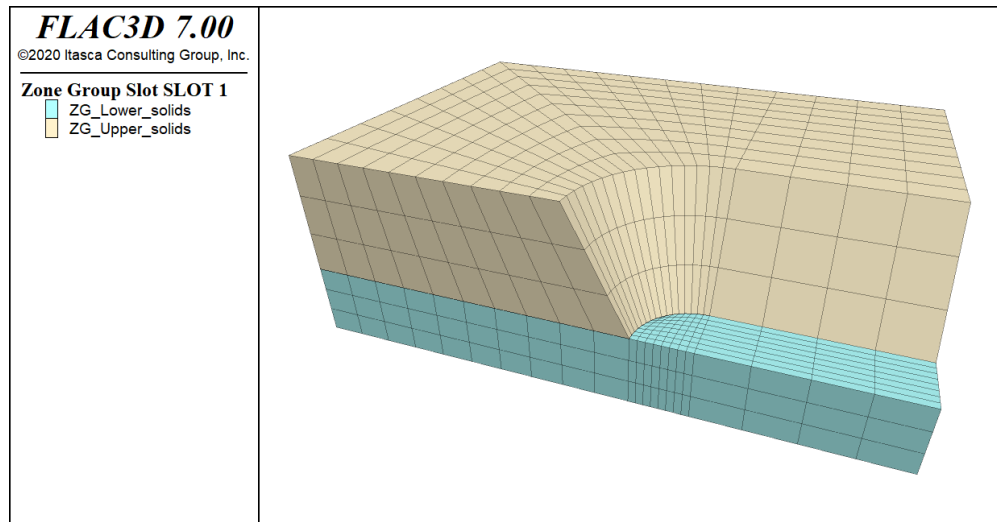


Figure 50: In *Rhino*, solids organized in layers are processed into *FLAC3D* groups in SLOT1.

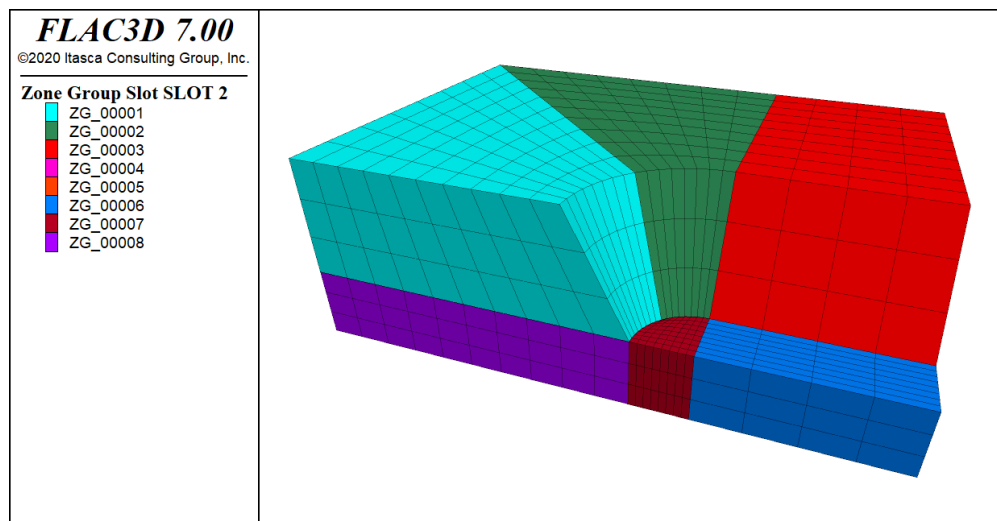


Figure 51: *FLAC3D* SLOT2 groups correspond to the individual *Rhino* solids.

As an additional exercise, change the **BlockRanger** meshing parameters and observe how the output mesh changes. For example, try setting *MaxEdgeLength* = 3.0 or *MaxEdgeLength* = 100.0 and *MinEdgeResolution* = 5. The simplest way to check the results is by generating bounding surface mesh with *GenerateSurfaceMesh* option. Use different choices in the option and observe how it changes the bounding surface mesh. The *ByModel* choice results in surface mesh on the boundary of the whole model only, selecting *ByLayer* adds faces the boundary between layers, and selecting *BySolid* adds faces on the boundaries between each solid.

## Tutorial 5: Creation of a Hybrid Structured-Unstructured Mesh

*Expected work time: 1–3 hours*

This tutorial describes a methodology that can be used to create hybrid volume meshes consisting of structured and unstructured meshes. This methodology ensures that both types of meshes are perfectly connected and fully conformal.

Structured meshes often provide high-quality elements and conform better to desired mesh parameters (such as element size, type). These meshes, however, can typically be created only for regular shape objects, for example, man-made structures. On the other hand, unstructured meshes can fill objects of any shape and are well suited for meshing irregular geologic features. In this example, a tunnel excavation is filled with a structured mesh produced by **BlockRanger**, while the surrounding rock domain containing irregular topographic and fault surfaces is filled with an unstructured mesh.

### Importing the geometry

1. Start *Rhino*, select **Large Objects, Meters** (similar method as step 1 in Tutorial 1) and save the *Rhino* project at a desired location.
2. Import the initial geometry from a DXF file by navigating to **File** → **Import** and select “T5\_geometry.dxf” from folder “TutorialExamples\5\_HybridMesh\_Tunnel”. In the **Import** dialog, select **Model** → **Meters** and **Layout Units** → **Millimeters**, and keep other parameters at the default values.
3. Maximize *Perspective* viewport, select Shaded view, and zoom out to the extent of the model by pressing **Ctrl+Alt+E**.
4. In the top menu, navigate to **View** → **Display Options...** This will open the **Rhino Options** dialog; select **View** → **Display Modes** → **Shaded** → **Objects** → **Curves** and set **Curve Width** to 4. This will display all curves thicker in the Shaded view. The initial settings can always be restored by clicking on **Restore Defaults**.

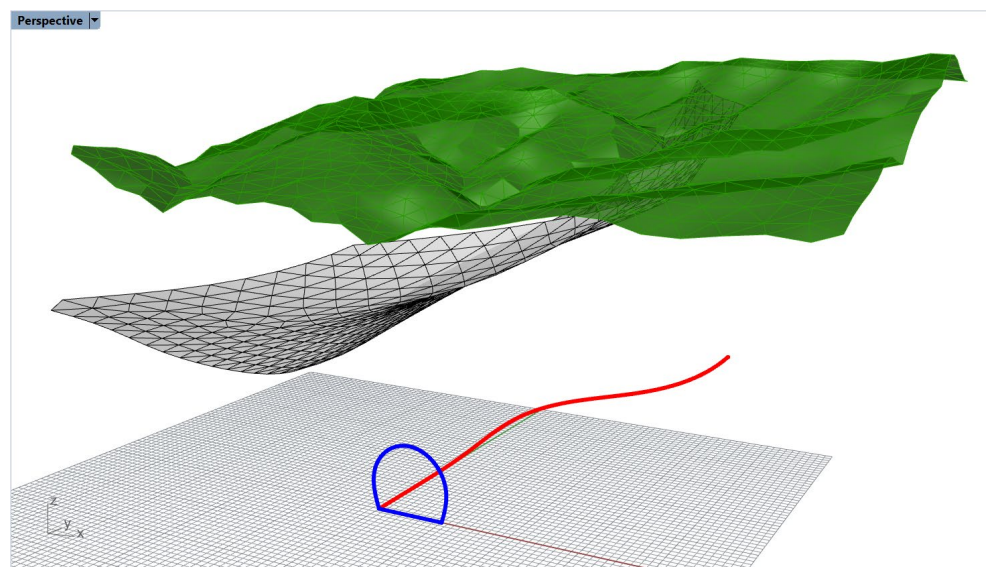


Figure 52: DXF geometry imported to *Rhino* corresponding to the tunnel profile and excavation direction.



5. Navigate to the *Layers* pane and delete all layers except for “Default”, “Direction”, “Topo”, “Fault”, and “TunnelProfile”.

The imported geometry contains two meshes, one corresponding to the topographic surface and one corresponding to a large fault. Note that the fault almost reaches to the topo surface but does not intersect it. This will need to be fixed, as the fault is supposed to intersect the surface. The geometry also contains a horseshoe-shaped curve (in blue) corresponding to the tunnel profile and a red curve corresponding to the tunnel excavation direction. The tunnel must be constructed along the red curve in such a way that its cross-section is always perpendicular to the curve. Note that the front end of the red curve is already aligned with the bottom-left corner of the tunnel profile.

### Construction and meshing of the tunnel with structured mesh

6. Turn off layers “Topo” and “Fault” to leave only the profile and direction curves visible.
7. Tunnel profile consists of several curves (in blue). Select all of them and type **\_Join**.
8. Select the red curve (excavation direction) and type the command **\_Sweep1**. For cross-section curves, select the single blue curve (tunnel profile). When offered to *Drag seam point to adjust*, drag it from the bottom-left corner of the tunnel profile to the topmost point of the profile and press **Enter** (maximize *Front* viewport for ease of dragging the seam point; this can be done while the command is active; Figure 53). It is important to move the *seam point* from the corner to create the proper geometry suitable for **BlockRanger**. For other options in the **\_Sweep1** command, use those shown in Figure 54.

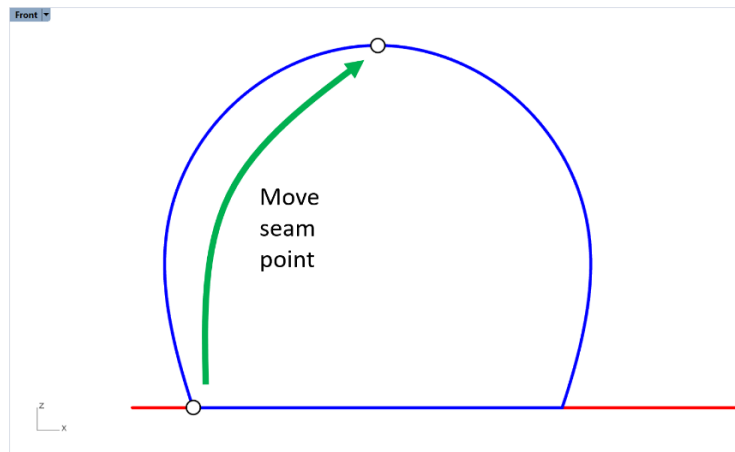


Figure 53: Drag seam point to the top of the tunnel profile.

9. Create new layer “Tunnel” and assign a newly created tunnel surface to it.
10. Select the tunnel profile curve (blue) and create the tunnel cross-section surface out of it by using the command **\_PlanarSrf**.
11. Select the initial cross-section surface and use command **\_ArrayCrv** to duplicate it along the direction path (red curve) with a spacing of roughly 5 m; this will create excavation stages that can be gradually removed during numerical simulations (Figure 55). In the command options pop-up window, set *Distance between items* = 5, *Orientation* = *Freeform*.
12. Turn off layers “TunnelProfile” and “Direction” as they will no longer be needed.



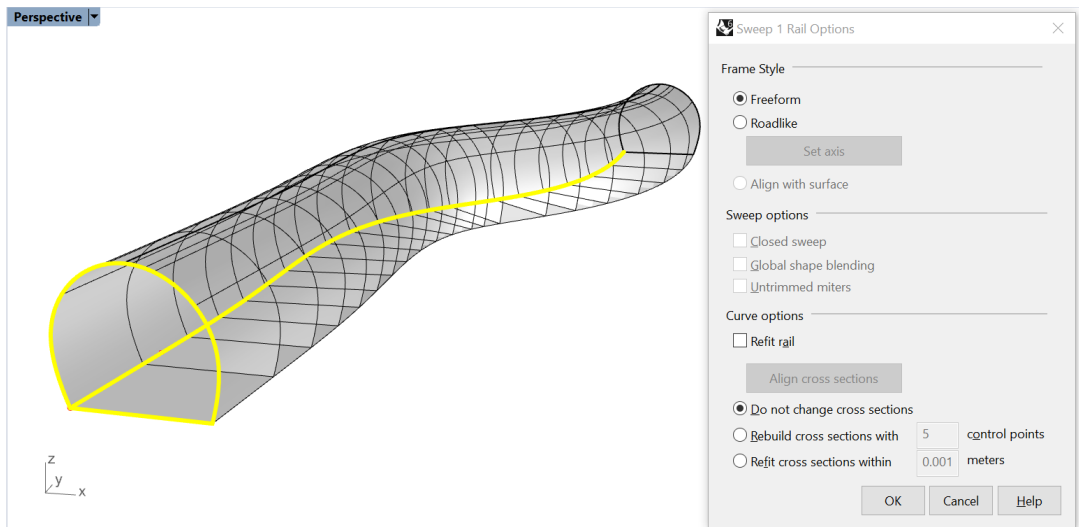


Figure 54: Creation of tunnel surface by sweeping the profile curve along the direction path.

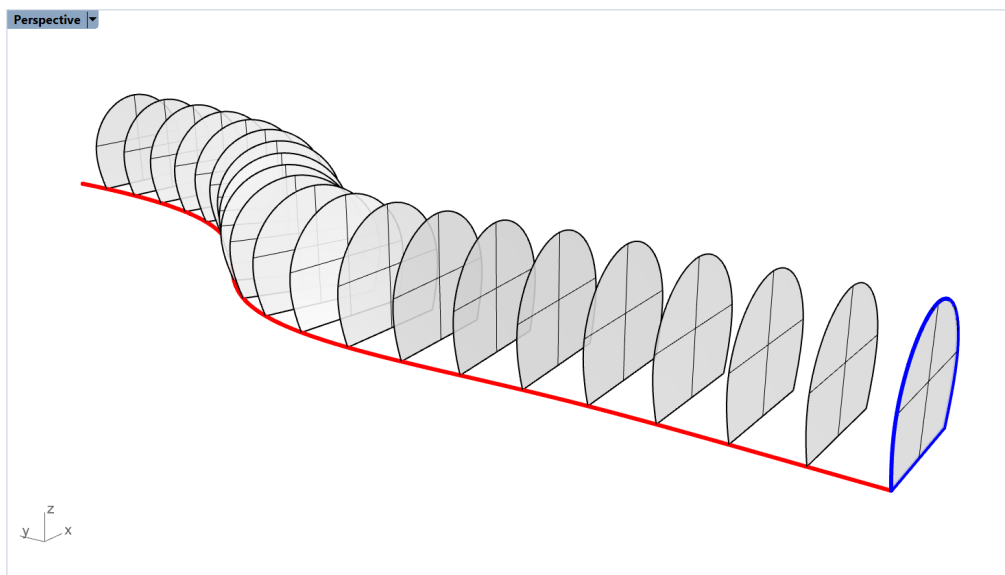


Figure 55: Creation of excavation stages by duplicating initial cross-section surface. Tunnel surface is not shown.

13. Split the tunnel surface with all the cross-section surfaces by selecting the tunnel surface only, then using the **\_Split** command and selecting all other surfaces (**Ctrl+A**). A message stating that "One polysurface split into 21 pieces." should appear in *Rhino*'s command area.
14. Turn off layer "Tunnel". Only the tunnel cross-sections should be visible. Delete all of them (21 objects).
15. Turn the "Tunnel" layer back on. Select all 21 polysurfaces and use the **\_Cap** command to close the front and back of each excavation stage. A *Rhino* notification should appear that 42 caps have been created. Now each excavation stage is represented by a separate watertight polysurface (or by a solid), which can be easily meshed using **BlockRanger**.
16. Colorize all stages with *Griddle*'s **\_ColorizeObjects** command (Figure 56).

Before proceeding with meshing the tunnel, some additional work is needed to trim parts of the tunnel that extend past the boundaries of the model.

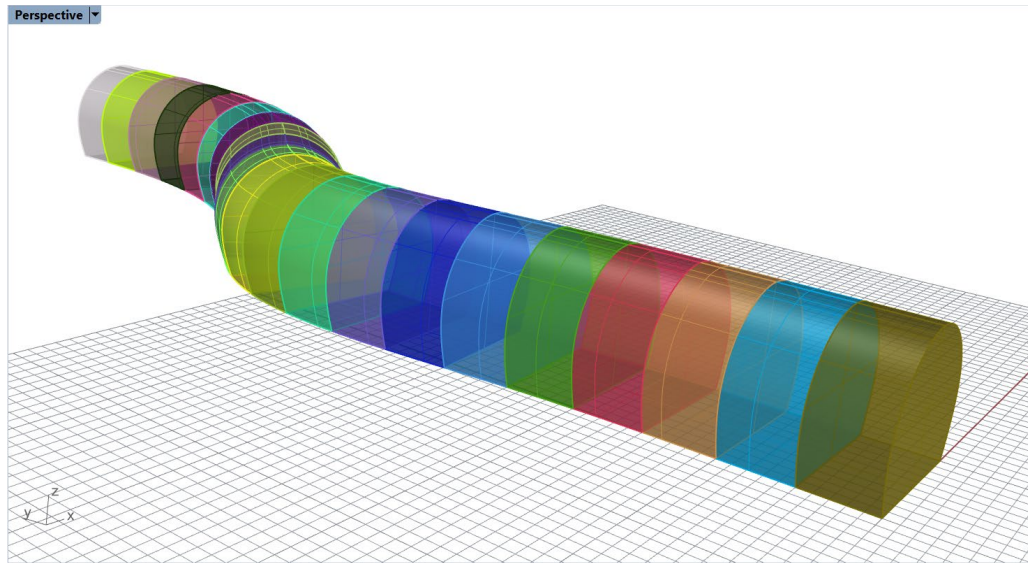


Figure 56: Watertight polysurfaces (solids) representing each excavation stage of the tunnel.

17. Turn off the “Tunnel” layer and turn on the “Topo” layer to show the topographic surface mesh.
18. Create a planar surface underneath the topo mesh by using the **\_Plane** command and selecting the *Center* option to specify the center of the plane and its size.
  - *Center of plane*: 0,50,-25
  - *Other corner or length*: 150
  - *Width*: press **Enter**
19. Select the topo mesh and duplicate its border using the **\_DupBorder** command.
20. Select the border curve and extrude it downward using the **\_ExtrudeCrv** command in such a way that all parts of the extrusion go below the plane (Figure 57). This will create a polysurface that intersects the horizontal plane. Delete the duplicated border.

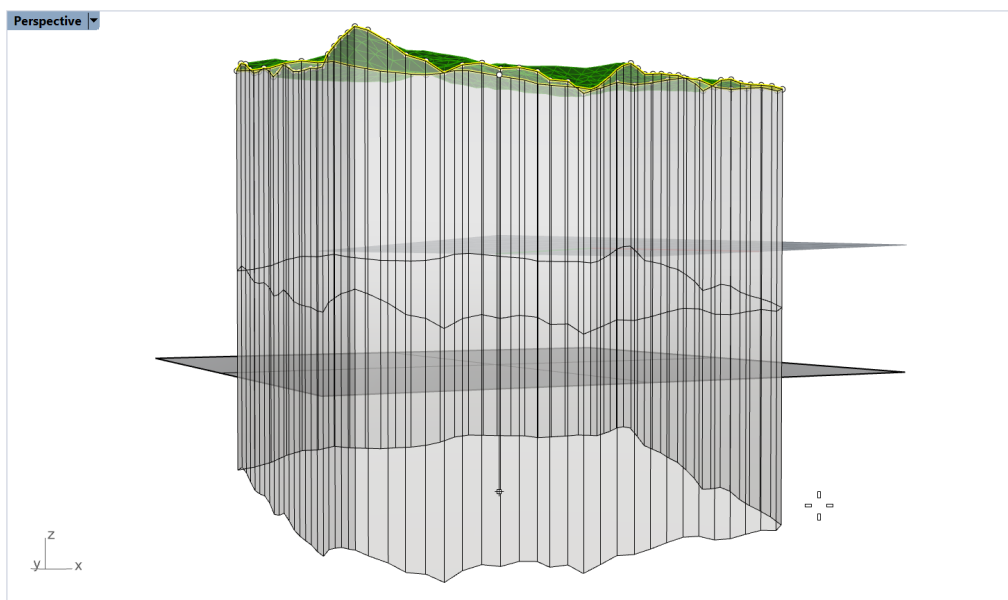


Figure 57: Extrusion of topographic surface border.

21. Select the extruded polysurface and split it by the horizontal plane using the **\_Split** command. Delete the bottom piece of the polysurface.
22. Select the planar surface and split it by the remainder of the extruded polysurface. Delete the outer piece of the planar surface.
23. Join the bottom surface with the extruded polysurface (**\_Join** command) to create a closed domain (together with the topo mesh), which will serve as the modeling domain after meshing.
24. Create new layer “Domain” and place the newly created polysurface in it (by changing the layer assignment in the polysurface properties).
25. Turn on the “Tunnel” layer and note that the front and back sections of the tunnel stick out from the domain (Figure 58). Split them by the domain and delete excessive parts.

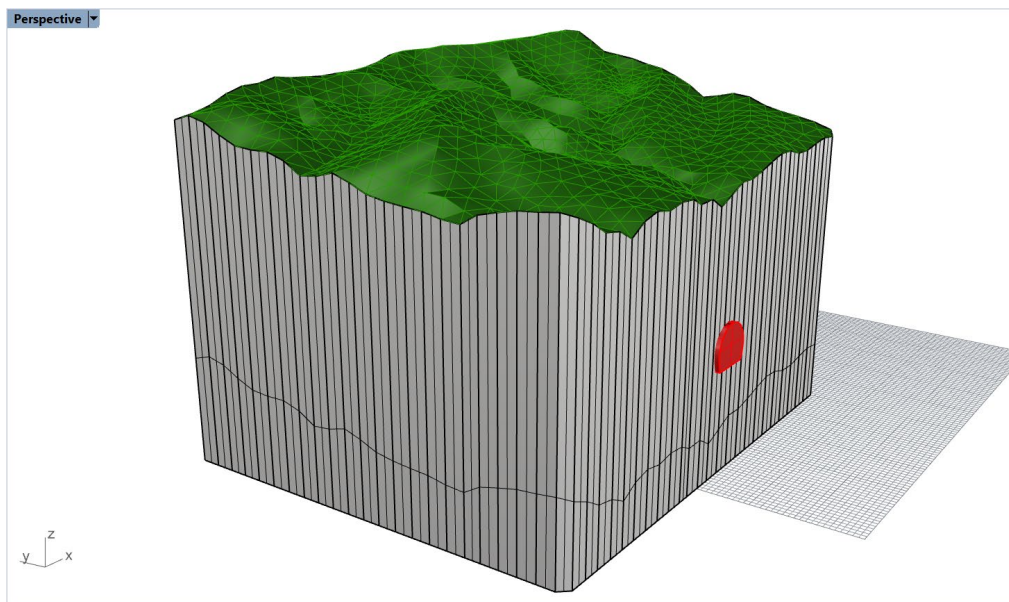



Figure 58: The front part of the tunnel extends through the modeling domain.

26. Hide layers “Topo” and “Domain” so only the tunnel stages are present.
27. Now the front and the back of the tunnel contain openings. Select the front and back tunnel stages and cap them using the **\_Cap** command to create watertight solids. At this point, the tunnel is ready for meshing.
28. Select all 21 solids corresponding to the tunnel excavation stages and type the **\_BR** command or click on the  icon in the *Griddle* toolbar. Use all default **BlockRanger** parameters and set *GenerateSurfaceMesh* = *ByModel*. **BlockRanger** will create a structured volume mesh for the tunnel and will output it at the user-specified location (keep *FLAC3D* binary format as output format). **BlockRanger** will also create surface mesh corresponding to the tunnel’s external surface and will output it into *Rhino* (Figure 59). This surface mesh will be used to create an unstructured volume mesh outside the tunnel.
29. Create new layer “TunnelMesh”. Change the layer of newly created surface mesh from “Default” to “TunnelMesh”. Now turn off layer “Tunnel”, leaving only one layer active as shown in Figure 59.

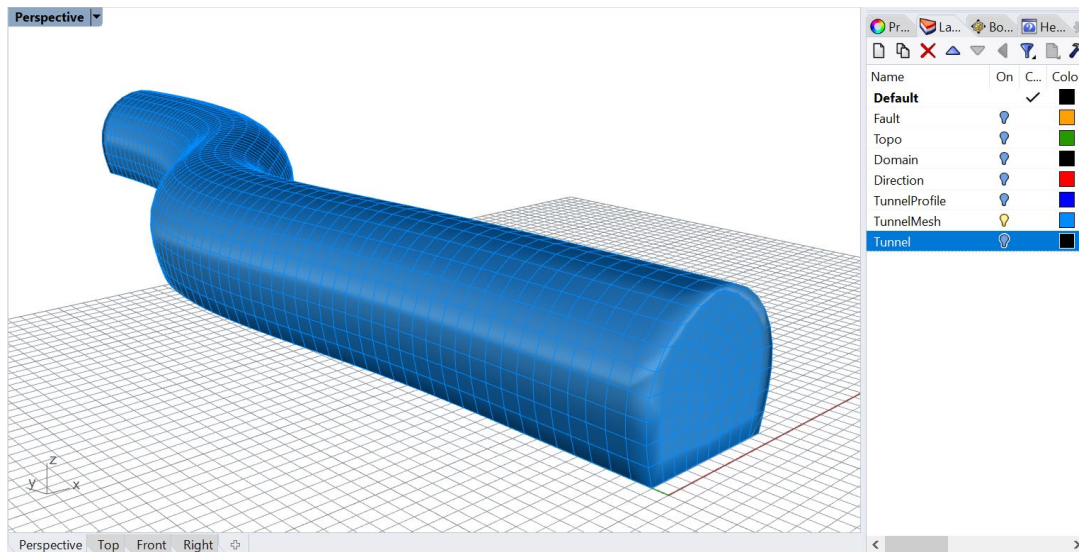




Figure 59: Surface mesh corresponding to the exterior of the tunnel's structured volume mesh.

## Meshing the tunnel exterior with unstructured mesh

To create an unstructured domain mesh that is conformal with the tunnel structured mesh, bounding surface mesh presented in Figure 59 will be used.

30. First, the front and back caps of the bounding surface mesh should be removed to leave only the "shell" of the tunnel. Type the **\_GExtract** command or click on the  icon in the *Griddle* toolbar. Select option *AllSurfaces* and use *MaxBreakAngle* = 85° to separate the caps. Select and delete them afterwards. The tunnel mesh should be hollow inside.
31. In the *Layers* pane, right-click on layer "TunnelMesh" and select *Duplicate Layer and Objects*. The next few steps will cause changes in the tunnel mesh, but volume meshing (the final step) requires the original mesh. Therefore, the duplicated tunnel mesh is used in the next steps. Turn off layer "TunnelMesh" and keep the duplicated layer on.
32. Select the tunnel mesh and use the command **\_DupBorder** to duplicate its open boundary.
33. Turn on layer "Domain". Select the domain polysurface and split it with the duplicated tunnel boundaries by typing the **\_Split** command and selecting the curves only.
34. Delete parts of the domain corresponding to the tunnel front and back to create openings for the tunnel (Figure 60).
35. Select the domain and create the initial mesh for it using the **\_Mesh** command. Use *Simple Controls* and select the middle position with the slider. While the polysurface is still selected, delete it. The initial mesh is not very good, but it will be remeshed in future steps.
36. Turn on the "Fault" layer. Note that the fault does not fully intersect the domain and the topo meshes (turn "Topo" layer on and off to see that). To extend the fault mesh to the domain boundaries, select it and type the **\_GExtend** command or click on the  icon in the *Griddle* toolbar. Select option *ExtendAllBoundaries*, specify *ExtendLength* = 10, and keep *MeshType* = *Merged*. The command will extend the mesh by adding new faces, and the extended mesh will fully intersect the domain and topo meshes. Turn on the "Topo" layer. The result should look similar to Figure 61.
37. Select all objects with **Ctrl+A** and use **GInt** to intersect all meshes. Use a tolerance of 0.001 and keep all other parameters at defaults.



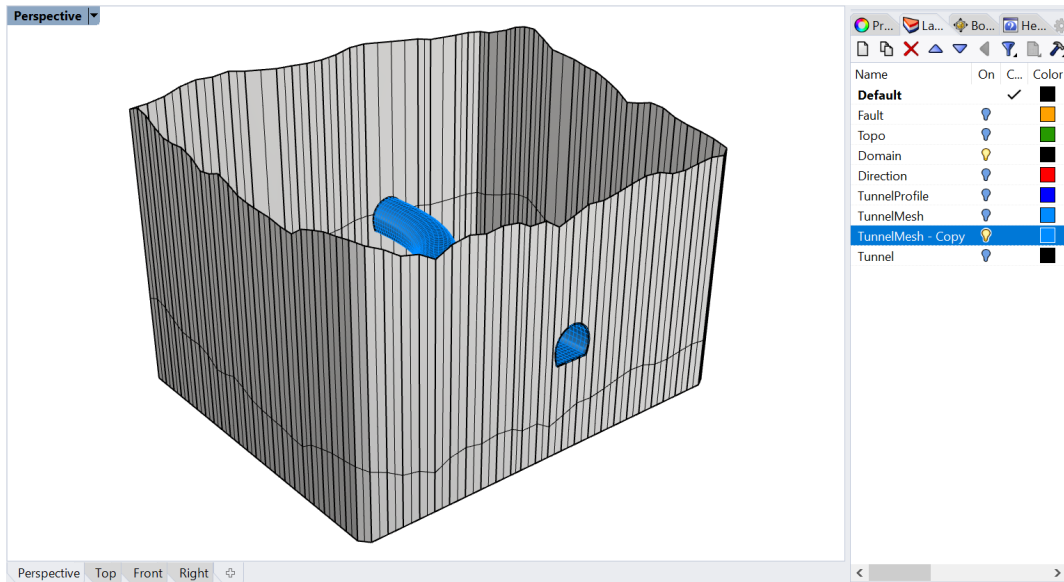


Figure 60: Removing parts of the domain at tunnel front and back.

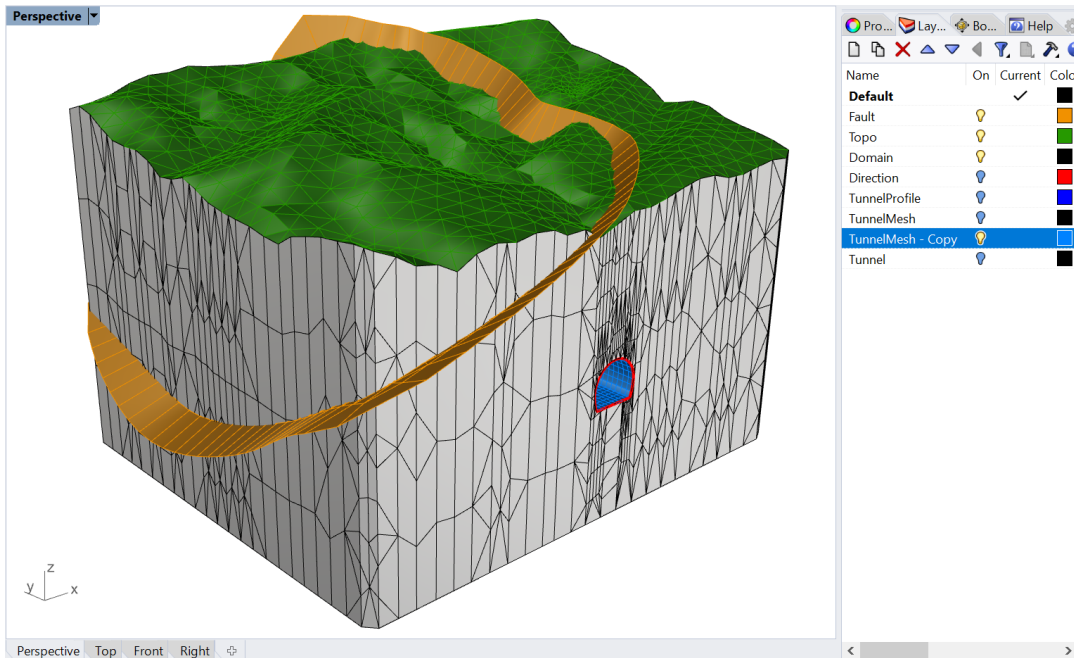


Figure 61: Extended fault mesh intersects domain and topo meshes. Note that the colors of topo and tunnel meshes and duplicated tunnel borders (curves) are changed from default.

38. Delete the copy of the tunnel mesh (blue mesh in Figure 61) or delete the whole layer containing it.
39. Select all objects again with **Ctrl+A** and use **GSurf** to remesh the meshes. Because the duplicated tunnel borders are selected as well (red curves in Figure 61), they will serve as hard edges to preserve conformity with the tunnel mesh. Use the following parameters for **GSurf**: *Mode* = *QuadDom*, *MinEdgeLength* = 1, *MaxEdgeLength* = 10, *RidgeAngle* = 40, and keep all other parameters at default values.  
After **GSurf** completes remeshing, the duplicated tunnel borders stay selected. Delete them, as they are no longer needed (Figure 62).

Turn on layer “TunnelMesh” and zoom in to the tunnel front or back (Figure 63). It can be easily seen that both tunnel and domain meshes are conformal (nodes and edges match along the border). This is due to using **GInt** to create proper initial intersections between the domain and tunnel meshes and due to using the border of the original tunnel mesh as a hard edge during remeshing.

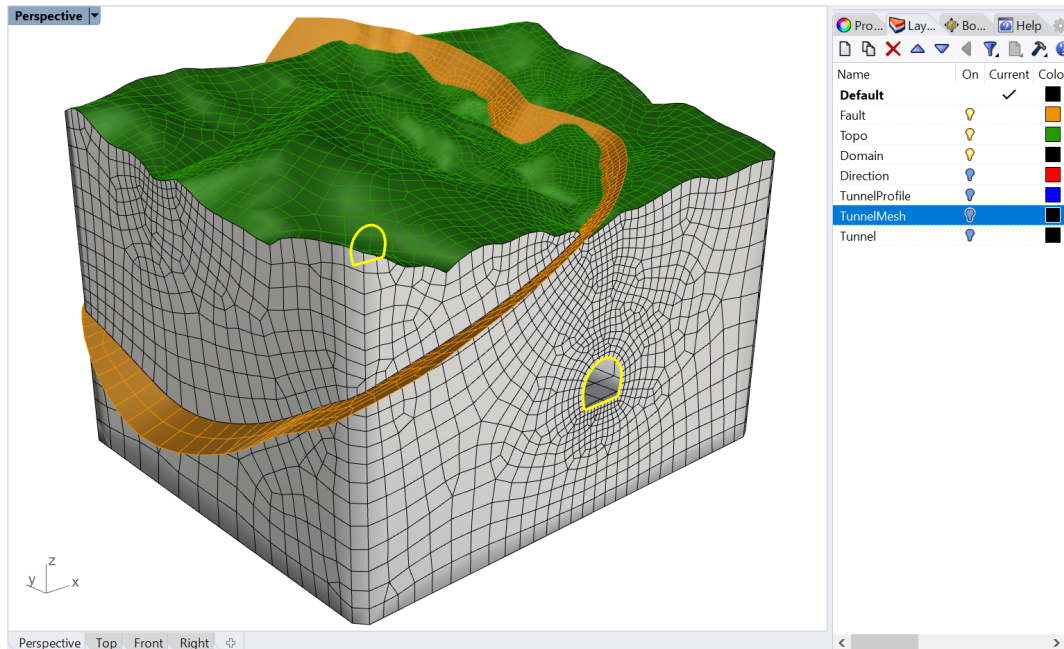


Figure 62: Result of remeshing of all meshes (except for the tunnel mesh) while preserving mesh edges and nodes along the tunnel borders (hard edges).

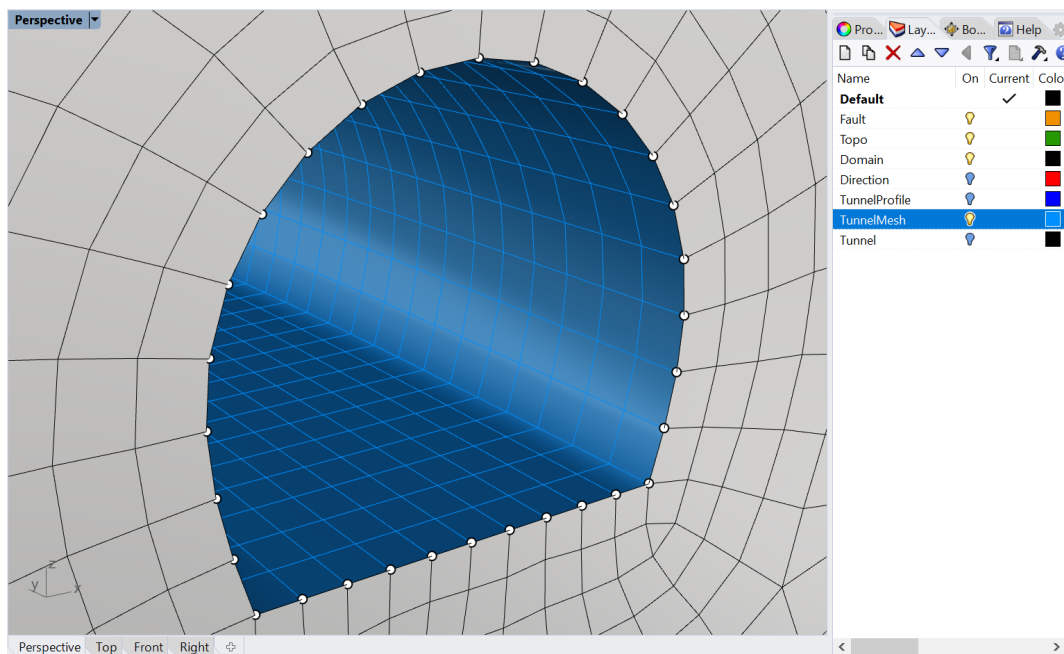


Figure 63: Close view of the tunnel front showing that tunnel and domain meshes are conformal.

40. Assign names to each of the meshes in the object *Properties* pane (click on a mesh and press **F3**), e.g., name fault “Fault”, tunnel mesh - “Tunnel”, etc. These names will be passed as surface names within the volume mesh.

Now the model is ready for volume meshing with **GVol**. Even though parts of the fault mesh extend outside the watertight domain, volume meshing can be carried out. **GVol** typically marks parts of surface meshes causing meshing errors with red outlines; in this case, however, red outlines will simply indicate that the “external” mesh faces were ignored during the meshing process.

41. Select all meshes and use **GVol** to create an unstructured volume mesh for the exterior of the tunnel. For parameters, use *HexDom* mesh and output in *FLAC3D* binary format; keep all other parameters at defaults. Save the output file at the same location as the tunnel structured mesh (do **NOT** overwrite it). Note that the volume mesh will not be created inside the tunnel this time, as the interior part of the tunnel is not a watertight domain; only the exterior of the tunnel constitutes a watertight domain.

Two meshes (grids) have been created in this example: a structured volume mesh with stages corresponding to the interior of the tunnel and an unstructured volume mesh corresponding to the exterior of the tunnel (domain). Both meshes can be loaded in *FLAC3D* (or any other code, if another format was used) and used for numerical modeling. Because the meshes are conformal and faces, edges, and nodes are duplicated along the boundary of the tunnel, *FLAC3D* can merge the meshes to avoid using attach conditions. After importing both meshes, use the merge command:

```
flac3d>zone gp merge
--- 2610 gridpoints merged and 5160 surface faces removed.
```

The result is shown in Figure 64.

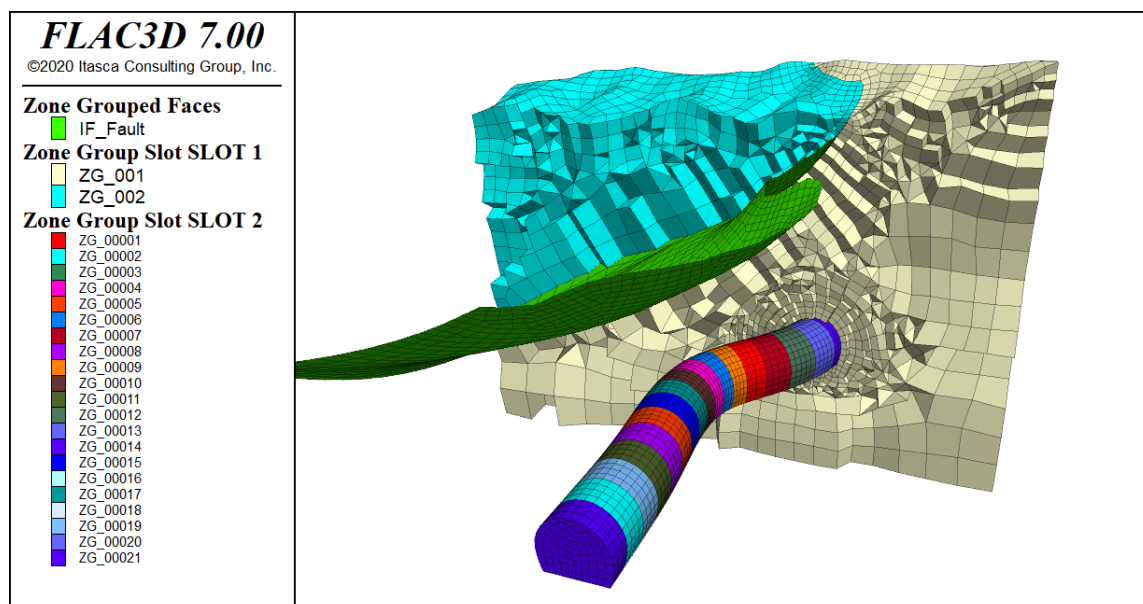


Figure 64: Structured and unstructured meshes loaded in *FLAC3D*.



## Tutorial 6: Mesh Clean-up and Rebuilding

*Expected work time: 1 hour*

This tutorial provides an example of a model that requires initial mesh cleaning and fixing before further model operations can be done. The model provided as a DXF file contains an existing triangular surface mesh defining drifts in a mine and a topographical surface (Figure 65). Such surfaces often come from minimally processed field measurements by various surveillance systems, and they often consist of triangular meshes, which generally are not clean (often non-conformal, contain overlapping and duplicate faces, missing faces, etc.) Thus, they typically need to be cleaned and remeshed to produce surface meshes suitable for volume meshing.

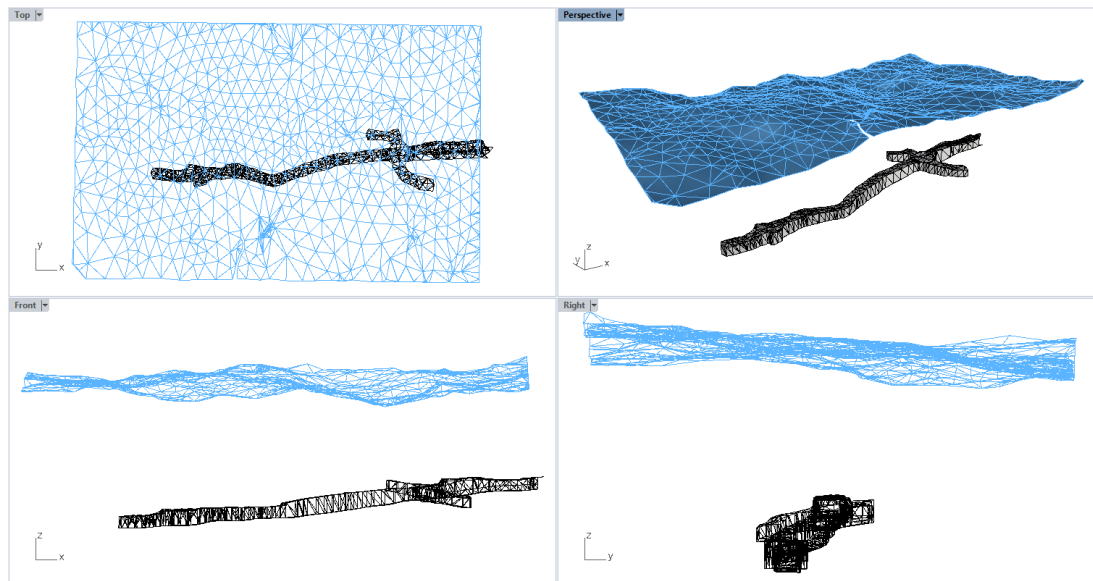


Figure 65: Drift geometry from a mine.

### Importing the geometry

1. Start *Rhino*, select **Large Objects, Meters** (similar method as step 1 in Tutorial 1) and save the *Rhino* project at a desired location.
2. Import the initial geometry from a DXF file by navigating to **File** → **Import** and select “T6\_drifts.dxf” located in folder “TutorialExamples\6\_MeshCleanup\_Drifts”. In the **Import** dialog, select **Model** → **Meters** and **Layout Units** → **Millimeters**, keep other parameters at default values.
3. If any of the viewport is maximized, double-click on the viewport icon to restore all four views as in Figure 65.
4. Imported objects are located far from the origin and may not be visible. Zoom out to the extent of the model by pressing **Ctrl+Alt+E**. Click on the topographic and drift excavation meshes to ensure that they are in separate layers. Delete all layers except for “Default”, “EXCAVATION”, and “TOPO”.
5. Hover the mouse around the model and pay attention to the coordinates in the lower-left corner of the information pane: the displayed coordinates are far from the origin (zero coordinates). This is problematic as it limits the number of significant digits available for geometrical manipulations and for numerical analysis. It is good practice to translate the model to locate it around the origin.

6. Select all objects (**Ctrl + A**) and type the **\_Move** command. In the *Top* viewport select the top-left corner node as the point to move from and type 0 for the point to move to. Press **Ctrl+Alt+E** to zoom to the new model extents.
7. Maximize *Perspective* viewport and select Shaded view (Figure 66).
8. In the top menu, navigate to **View → Display Options...** This will open the **Rhino Options** dialog; select **View → Display Modes → Shaded → Objects → Curves** and set **Curve Width** to 4. This will display all curves thicker in the Shaded view. The initial settings can always be restored by clicking on **Restore Defaults**.

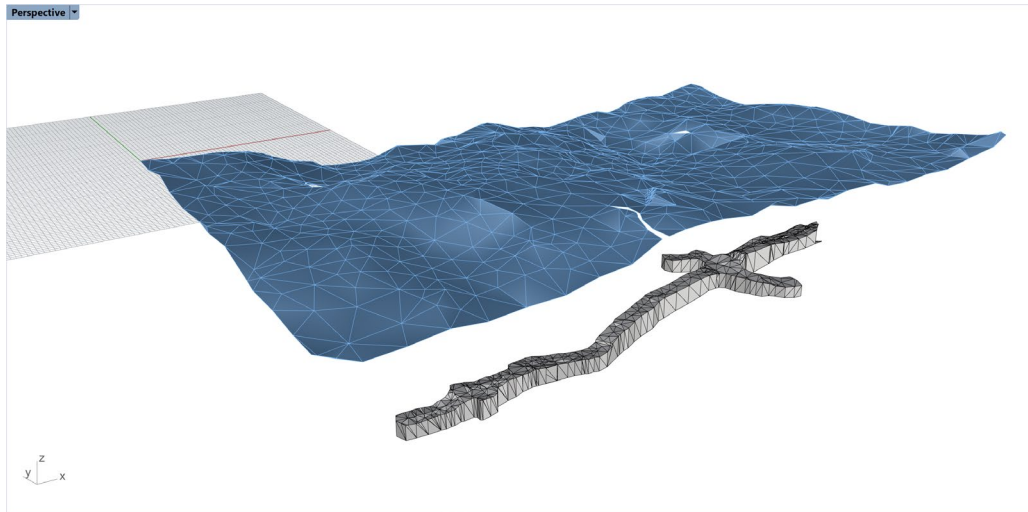


Figure 66: Drift model located near zero coordinates.

## Mesh clean-up

Two meshes imported from the DXF file contain a number of issues, such as holes, nonconformal faces, duplicate faces, naked edges, etc. Such surface meshes are not adequate for creating a valid watertight mesh domain suitable for volume meshing. The meshes need to be cleaned up first.

*Rhino* has some built-in utilities that allow for checking and summarizing surface mesh issues. One such utility can be called by typing the **\_Check** command (try this by selecting all meshes; a pop-up window will report a summary of various issues). Another useful command is **\_MeshRepair**. However, using that utility for numerous issues may be tedious and cumbersome. *Griddle* offers the command **\_GHeal**, which identifies and displays major surface mesh problems (that otherwise would prevent volume meshing) and attempts to fix numerous issues at once.

9. To check the meshes for naked edges (holes, cracks, disjointed faces) and clashing faces (non-conformal faces within a single mesh), type **\_GHeal** and select *ShowErrors* (Figure 67). Two new layers will appear: “NAKED\_EDGES” (pink curves) and “CLASHING\_FACES” (red curves), which contain curves outlining corresponding issues. In this case, the layers contain many curves. After mesh repair operations, the number of issues (and corresponding curves) will decrease, which can be checked by calling **\_GHeal → ShowErrors** again (the outlines of the issues will be updated). To

clearly visualize only problematic parts, layers “EXCAVATION” and “TOPO” can be turned off in the *Layers* pane (Figure 68).

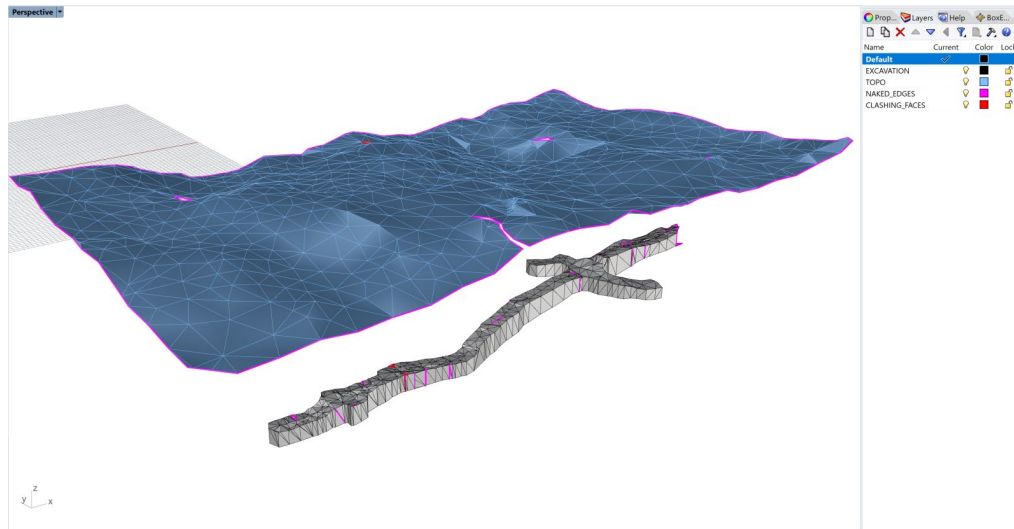


Figure 67: Displaying naked edges and clashing faces.

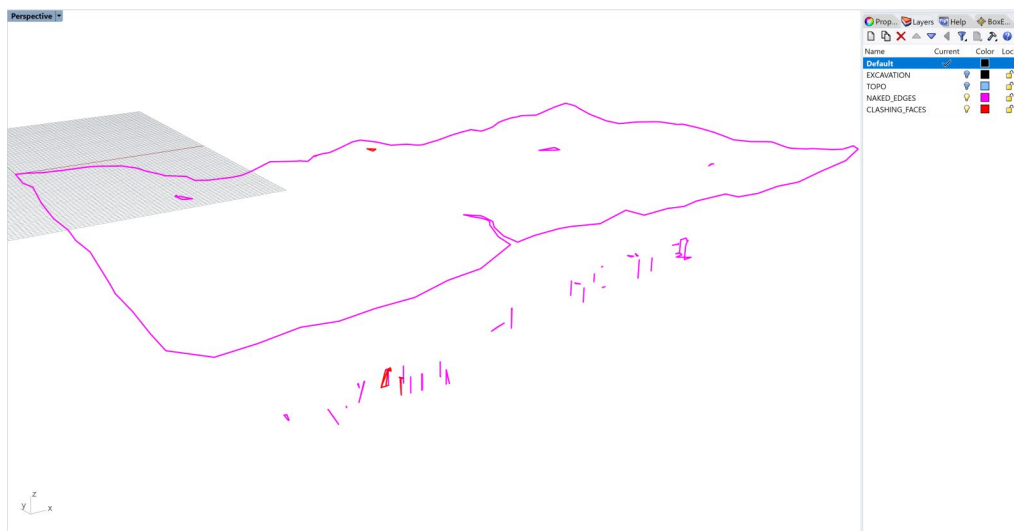


Figure 68: Displaying naked edges and clashing faces with meshes hidden.

10. To start fixing the meshes, first intersect all non-conformal faces within the meshes using **GInt**. Set tolerance to 0.002 and keep all other parameters at the defaults. After mesh intersections, update the outlines of the issues by calling **\_GHeal** → *ShowErrors*. Note that there are no more clashing faces, layer “CLASHING\_FACES” is empty, and fewer naked edges remain.

The remaining issues can be fixed using **GHeal** in automatic mode with the *AutomaticHeal* option (the manual mode option simply calls *Rhino's* **\_MeshRepair** command). *AutomaticHeal* allows for fixing numerous issues at once, but it is important to note that **GHeal** should be applied to problems discretely: only select to fix issues that are expected to be present. In certain cases, it is even better to fix issues one by one to get the best results. For example, first *FillHoles* and *FixClashingFaces* only, then *MendCracks* and then *AlignNormals*. In this model, there are holes and a “crack” in the topographic

mesh, and there are holes and protrusions (single faces with non-manifold edges) in the drift excavation mesh. There are also misaligned normals in both meshes. These issues are fixed in a few steps as outlined below.

11. Select all meshes and use **\_GHeal** → *AutomaticHeal* → *IssuesToFix* and set only *FillHoles*, *FixClashingFaces*, and *AlignNormals* to *Yes*, and all others are set to *No*. Press **Enter** and keep *Parameters* = *Automatic*. Press **Enter** again to run the command. All the holes in the topo and drift meshes will be filled, and outlines of the remaining problems will be updated automatically.
12. There is a big “crack” in the topo mesh, and it cannot be readily fixed by the **GHeal** command. (If **GHeal** is run with *MendCracks* = *Yes*, only half of the crack is “stitched”.) The topo mesh must be remeshed first, which will make it easier for **GHeal** to identify and fix the problems as the mesh becomes more uniform and regular (the same applies to the drift excavation mesh). Select the topo and drift meshes only (ensure no curves are selected), and remesh them using **GSurf** with *Mode* = *Tri*, *MinEdgeLength* = 0.5, *MaxEdgeLength* = 5, while all other parameters remain at defaults. A rather fine mesh is used to preserve the original mesh details.
13. Select the topo mesh only and fix the crack using **\_GHeal** → *AutomaticHeal* → *IssuesToFix*, set *MendCracks* = *Yes* and all others to *No*; press **Enter** and keep *Parameters* = *Automatic*. Now the topo mesh contains no issues and has a single naked poly-edge outlining its boundary (Figure 69).

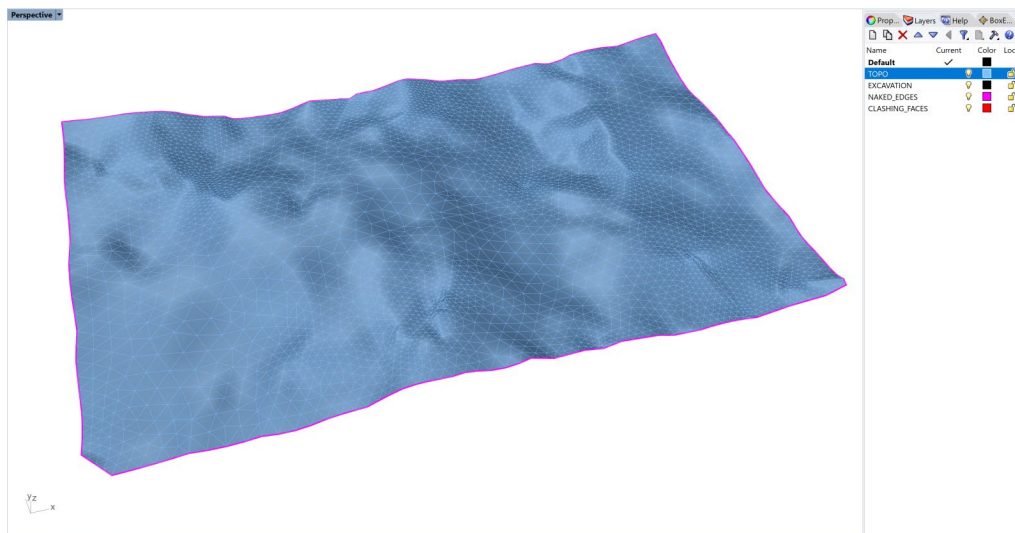


Figure 69: Remeshed and fixed topographic mesh. (The grid is not shown.)

14. Select the drift mesh only and fix the remaining issues by calling **\_GHeal** → *AutomaticHeal* → *IssuesToFix*, set *RemoveProtrusions* = *Yes* and all others to *No*, press **Enter**, and keep *Parameters* = *Automatic*. After fixing the drift mesh, the only remaining problems are a few faces that stick out from the front part of the drift as shown by the pink outlines in Figure 70. They will be resolved later when finalizing model construction.

At this point, all outstanding problems are fixed and the meshes are ready to be used to construct the full model. Note that, in general, **GHeal**'s ability to fix various mesh problems significantly depends on

the input mesh itself. Thus, results may vary depending on how the initial meshes are intersected and/or remeshed.

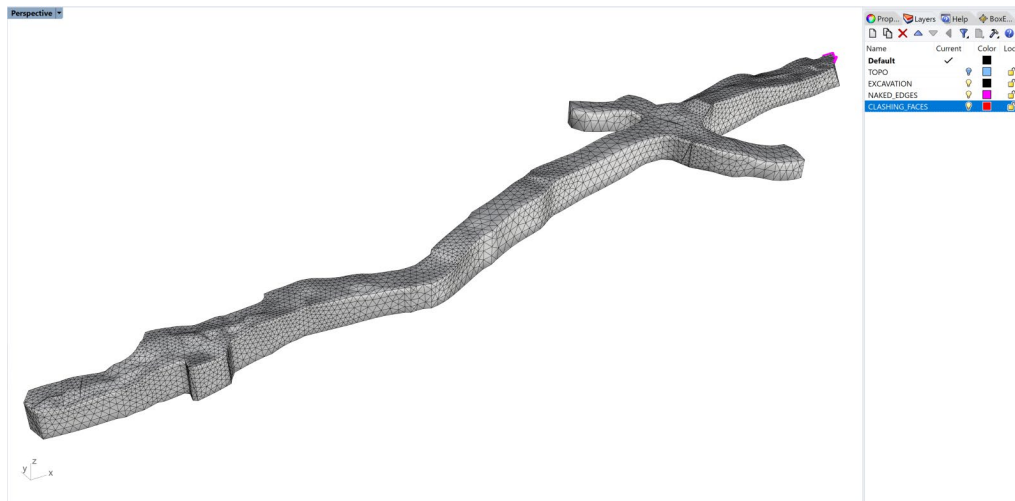


Figure 70: Remeshed and fixed drift mesh. (The grid and topographic mesh are not shown.)

## Building model domain and creating volume mesh

Before final remeshing and volume meshing, a watertight modeling domain must be created. This can be done using *Griddle's GExtrude* command.

15. Create a plane using the **\_Plane** command and *Center* option. Then specify:

- Center of plane ( Deformable ): 91,-55,-100
- Other corner or length ( 3Point ): 250
- Width. Press Enter to use length: 200

This will create a plane under the drift mesh as shown in Figure 71.

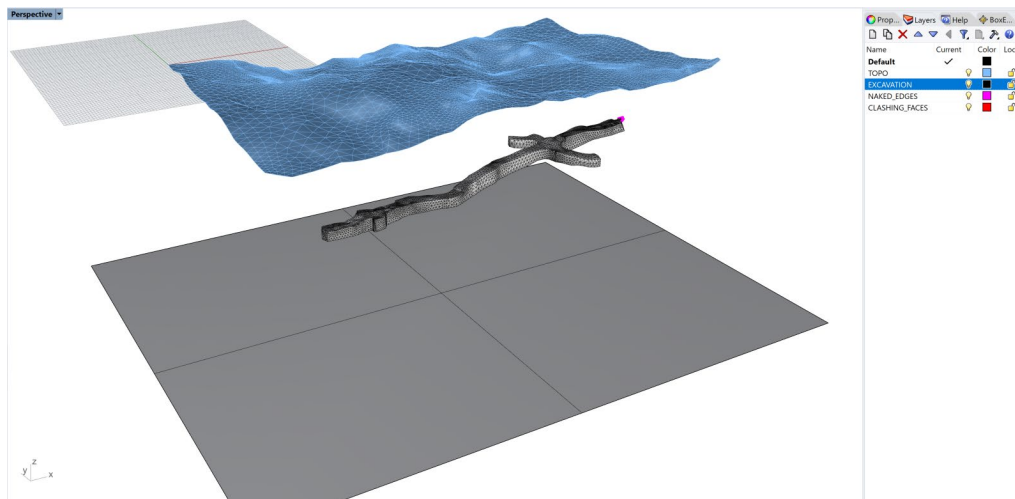



Figure 71: Creating a plane under the drift mesh.



16. Select the topo mesh and the plane and type **\_GExtrude** or click on the  icon in the *Griddle* toolbar to extrude the mesh to the plane and create a watertight domain. Use the following parameters: *ExtrMeshType* = *Tri*, *MeshOutput* = *Merged*, *MeshMode* = *Unstructured*, *MinEdgeLength* = 0.5, and *MaxEdgeLength* = 10.

The command will extrude the topographic mesh along its boundary until the intersection with the plane and will create side and bottom meshes. The result should look as shown in Figure 72.

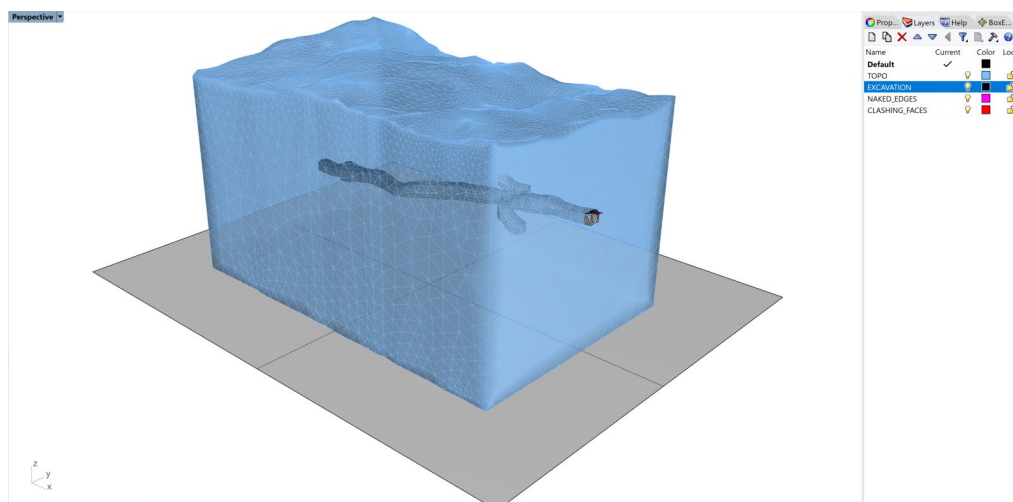




Figure 72: Creating watertight mesh domain with *GExtrude* command (Ghosted view).

17. Select and delete the plane. Turn off layers “NAKED\_EDGES” and “CLASHING\_FACES”.

Note that a piece of the drift mesh protrudes from the domain mesh. It can be easily separated and removed after intersection of meshes and remeshing.

18. Select all meshes (**Ctrl+A**) and intersect them using **GInt** with *MeshType* = *Tri*, *Tolerance* = 0, and *AdvancedParameters* → *OutputMesh* = *Merged*. This will create a single conformal mesh.
19. Remesh the mesh with **GSurf** using *Mode* = *QuadDom*, *MinEdgeLength* = 1, *MaxEdgeLength* = 10, *RidgeAngle*=40, and all other parameters at defaults. A large *RidgeAngle* is used to smooth the mesh. Note that the piece of the drift mesh still protrudes from the domain mesh.
20. To separate the drift mesh from the domain mesh and remove the protruding part of the drift, select the whole mesh and type the **\_GExtract** command or click on the  icon in the *Griddle* toolbar. Select option *AllSurfaces* and use *MaxBreakAngle* = 180 to separate meshes only along non-manifold edges (the break angle will not have an effect as it is set to 180°).
21. Select all meshes and use *Griddle*'s command **\_ColorizeObjects** or click on the  icon in the *Griddle* toolbar. This will assign a unique color to each mesh piece (Figure 73).
22. Delete all parts of the drift mesh that protrude from the domain mesh.
23. If desired, assign specific names to the drift excavation mesh and its front portion (entrance).
24. The surface meshes are ready for volume meshing. Create a *HexDominant* mesh using **GVol** with *MaxGradation* = 1, *TargetSize* = 7, and set the output format as desired (*FLAC3D* is used here; note, *Tet* mesh should be used for *3DEC*). If a warning about the presence of naked edges appears, press

**Yes** to continue with volume meshing (naked edges are present in the final model due to mesh extraction done earlier). Meshing may take some time due to the fine resolution of the topographic and drift meshes. The results of the volume mesh imported in *FLAC3D* are shown in Figure 74.

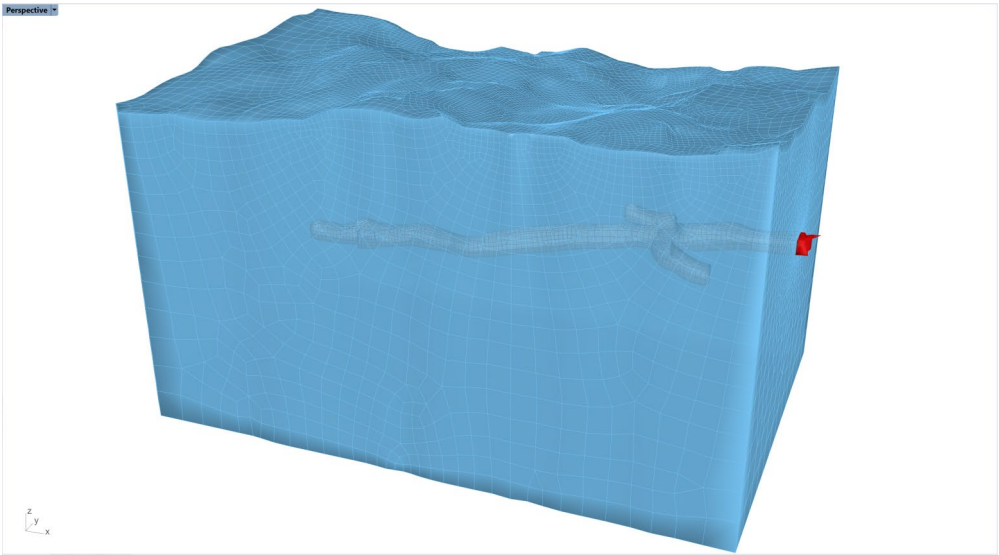


Figure 73: Meshes separated with the `_GExtract` command.

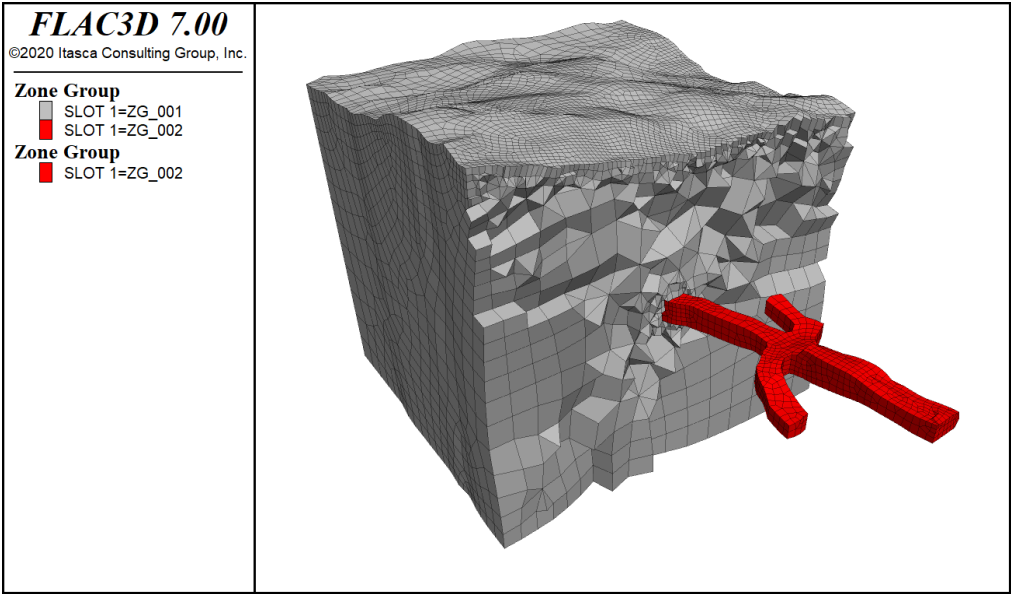


Figure 74: Final volume mesh obtained from the cleaned-up DXF meshes.



## Tutorial 7: Large Open Pit Model with Multiple Intersecting Faults

Expected work time: 1–3 hours

*Griddle* provides powerful tools for cleaning, preparing, and meshing very large models. This tutorial describes a workflow that can be used to prepare a large open pit mine model for volume meshing. The model contains multiple intersecting faults and stratigraphic layers. Even though the model is artificial, the initial surface meshes resemble those that are often obtained from minimally processed field data. Note that due to the complex workflow, the user may get slightly different results than those presented in the tutorial; however, this should not affect the workflow as a whole. Additionally, it is assumed that advanced **GInt** and **GSurf** parameters are initially configured to the defaults; this can be achieved by restarting *Rhino* or resetting the parameters through the option **GInt, GSurf** → *AdvancedParameters* → *Reset*.

Navigate to folder “TutorialExamples\7\_LargeOpenPitMine”, open file “T7\_LargeOpenPitModel.3dm”, and examine the model (Figure 75, Figure 76).

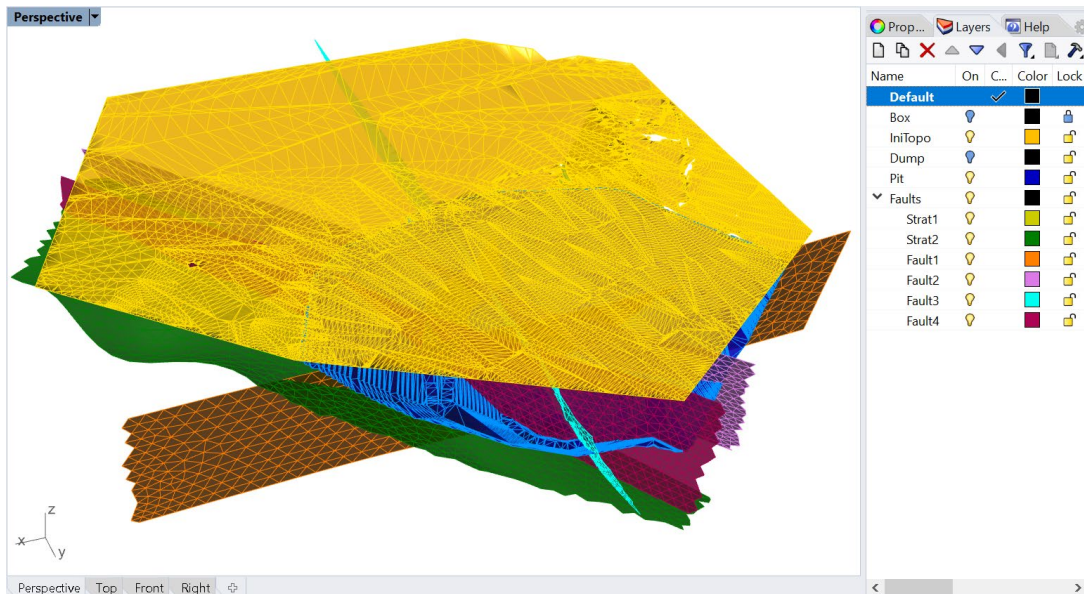


Figure 75: Initial model with topographic surface shown. Ghosted view.

Only a part of the open pit mine is provided in the model. It contains the area of interest (yellow dotted line in Figure 76) and includes the pit wall that may have the potential for instability due to several faults and a stratigraphic boundary crossing it and due to a large pit dump above the wall.

Upon examination of the initial model, following observations can be made:

- There is a single mesh in each layer. Meshes do not have specific names (see *Properties* pane / **F3**).
- Layer “Box” is locked, and it contains a polysurface representing the desired modeling domain. The layer is locked to avoid accidental modification of objects within the layer. To enable selection and modification of the objects, unlock the layer.

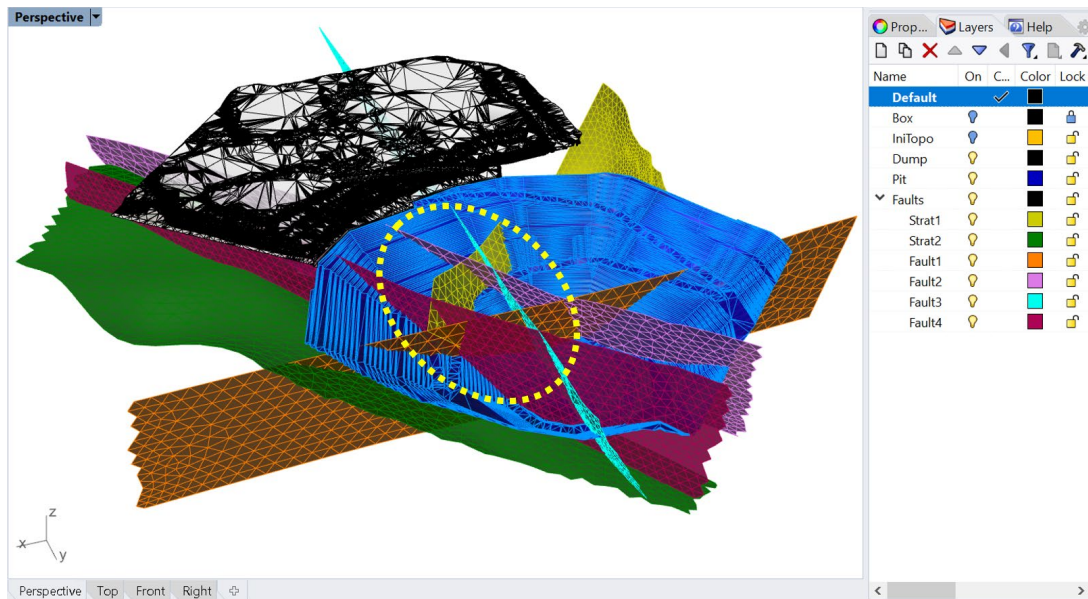


Figure 76: Initial model with large pit dump shown. Ghosted view.

- Surface meshes in layers “Pit”, “Dump”, and “IniTopo” seem to be minimally processed and contain a very large number of faces; face reduction may be needed. All other meshes appear to be remeshed and trimmed to roughly match the size of the modeling domain.
- The surface mesh corresponding to the initial topographic surface (layer “IniTopo”) visibly contains multiple holes and possibly other issues.
- None of the meshes appear to be conformal at the areas of contact or intersection.
- Some of the faults and stratigraphic boundaries seem to terminate on one another. However, it is not clear if meshes are in full contact or if there are small gaps between the meshes.
- Neither “PitDump” nor “Pit” meshes fully connect/intersect to the initial topographic surface. To verify if two meshes intersect each other, use the command **\_MeshIntersect**. The command will create polylines tracing the intersecting mesh parts (do not forget to delete these polylines afterwards).
- All meshes are located away from zero coordinates but not so far as to have any effect on the accuracy of calculations. Thus, there is no need to move the model closer to the origin.

### Preparation of the pit, pit dump, and topo meshes

1. Turn off all layers except for “IniTopo”. Select the topographic mesh. To make the mesh more uniform, remesh it with **\_GSurf**: *Mode = Tri, MinEdgeLength = 20, MaxEdgeLength = 100, RidgeAngle = 20°*, and keep other parameters at the defaults.
2. After remeshing, some of the initial mesh problems are fixed (e.g., clashing faces). However, the mesh still contains several large holes. Fill them using **\_GHeal** → *AutomaticHeal* → *IssuesToFix*: *FillHoles = Yes* and all other functions set to *No*. Keep *Parameters = Automatic*. After this, ensure that there are no more holes (turn on/off “IniTopo” layer) and then delete layers “NAKED\_EDGES” and “CLASHING\_FACES”.
3. Remesh the topo mesh again with the same parameters used earlier.

4. Turn off layer “IniTopo” and turn on layer “Dump”. The mesh representing the pit dump consists of a large number of non-uniform triangular faces and working with it may be difficult (in particular, remeshing). To check the number of faces in a surface mesh, use the command **\_PolygonCount**; the command reports that there are 204,360 triangular polygons in the mesh.
5. There is no need for such a highly accurate representation of the pit dump, so the number of faces in the mesh can be safely reduced without impacting the model. Select the pit dump mesh and type **\_ReduceMesh**. In the pop-up dialog, specify to reduce the polygon count by 85%. Keep all other parameters at the defaults.
6. Turn on layers “IniTopo” and “Pit”. Zoom in at any region where meshes visually intersect (e.g., Figure 77). One can find that in certain regions, the meshes overlap and protrude through one another or conversely, do not fully connect. This is typically due to obtaining meshes from different sources and initial mesh processing (including mesh separation, intersection, and remeshing).

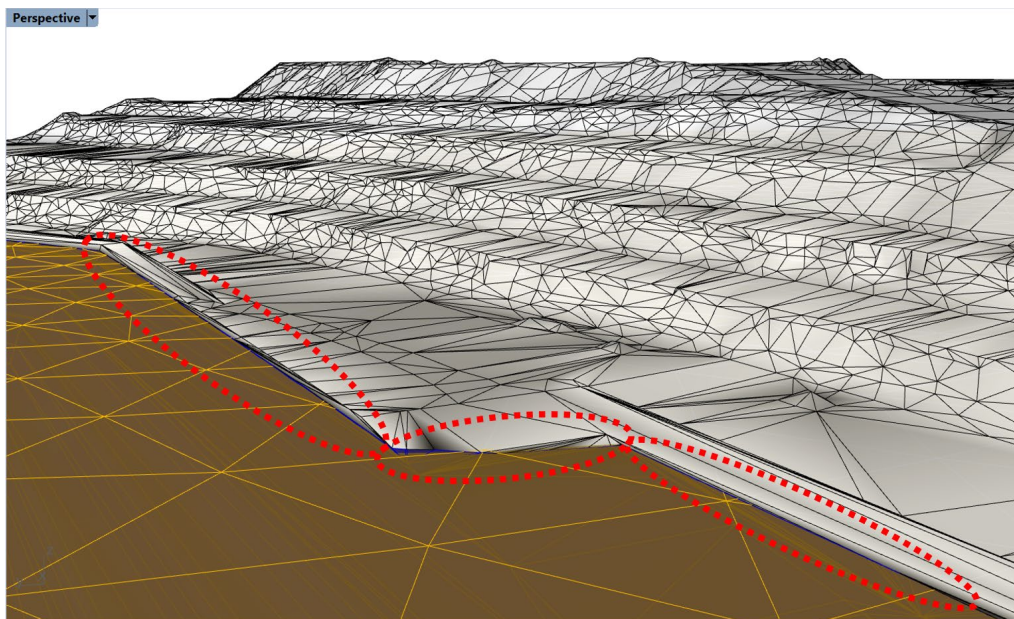


Figure 77: Zoomed view of the intersection between the pit, pit dump, and topo meshes. Marked regions denote some of the areas where intersections are not perfect.

Having several meshes (with rather different face sizes) overlap and/or terminate in close proximity to one another may present difficulties when using mesh intersector **GInt**. In this case, it can be challenging to find such a **GInt** tolerance that would be large enough to close all gaps and small enough that undesirable mesh deformations do not occur. Also, these nearly overlapping faces present similar challenges when remeshing and proceeding with volume meshing.

The simplest way to approach this challenge is to spatially separate meshes, so there are only two meshes intersecting along the same line, making it much easier to select a **GInt** tolerance to prepare meshes for final intersection and remeshing.

In order to separate the meshes for this model, a thin boundary layer will be extracted from the pit dump mesh along a part of its boundary. After that, the pit dump mesh will be extended downward and intersected with the topo mesh.



7. Turn off layer “IniTopo” and “Pit” layers.
8. To extract a thin boundary layer, the pit dump mesh must be remeshed such that it contains small elements (faces) along the boundary. Select the pit dump mesh and remesh it with **\_GSurf: Mode = Tri, MinEdgeLength = 5, MaxEdgeLength = 5, and RidgeAngle = 60°**. Remeshing may take a minute. Large *RidgeAngle* value is used to smooth the mesh.
9. Select the pit mesh and use the **\_GExtract → BoundaryFaces** option. This operation will extract a single layer of faces along the boundary of the mesh. Repeat this operation one more time to extract the next boundary layer of faces such that the two outermost rings of boundary faces have been separated from the dump mesh (Figure 78).

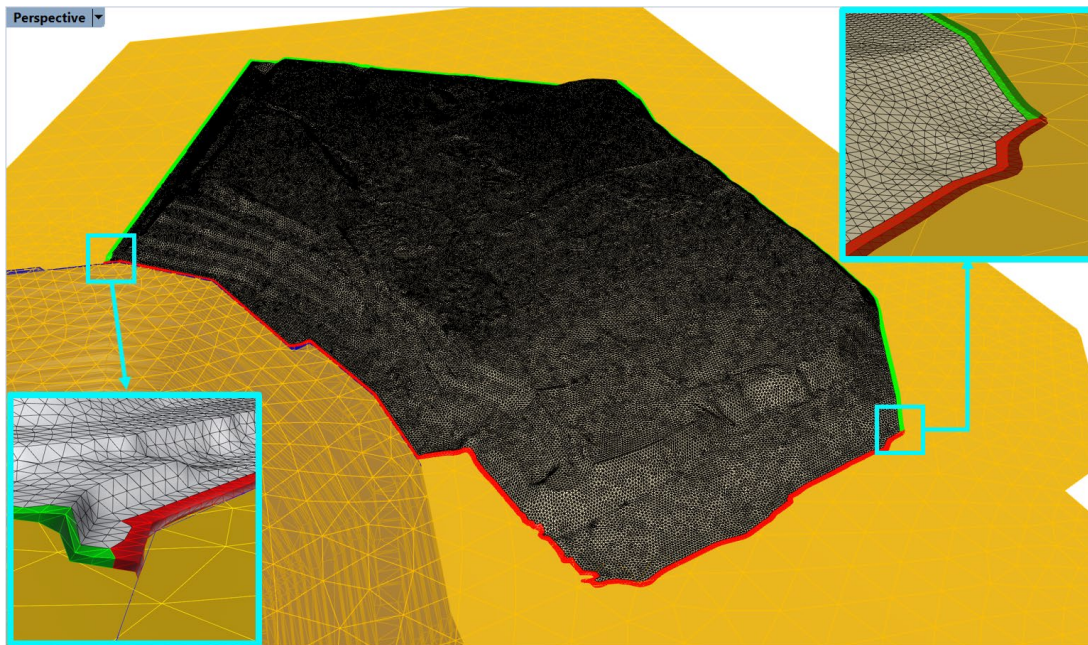


Figure 78: Boundary faces extracted from the pit dump mesh (note that colors are changed from default gray color). Faces in red were separated and will be deleted.

10. Hide the main pit dump mesh using the **\_Hide** command. The only objects that should be visible are the two thin boundary layers of the dump mesh.
11. Using the **\_ExtractMeshFaces** command, extract the faces from the boundary layers that are closest to the pit boundary and those that almost overlap the topo mesh (shown in Figure 78 in red). While the command is in face selection mode, other layers can be turned on and off and specific areas can be zoomed in/out to help with selection (use the **\_Zoom** command while in selection mode). Note that faces can be removed from the selection when pressing **Ctrl**.
12. After extracting faces from the layers, delete them (only faces in red in Figure 78). This will create a thin gap between the pit mesh and the dump mesh and will avoid having nearly overlapping faces between the pit dump and topo meshes.
13. Unhide the rest of the dump mesh with the **\_Show** command. First select the remaining thin layers of faces and add the main pit dump mesh to the selection. Type the **\_Join** command to join all meshes. Now a small gap between the pit and the dump meshes can be clearly visible as in Figure 79 (turn on “Pit” layer to check).

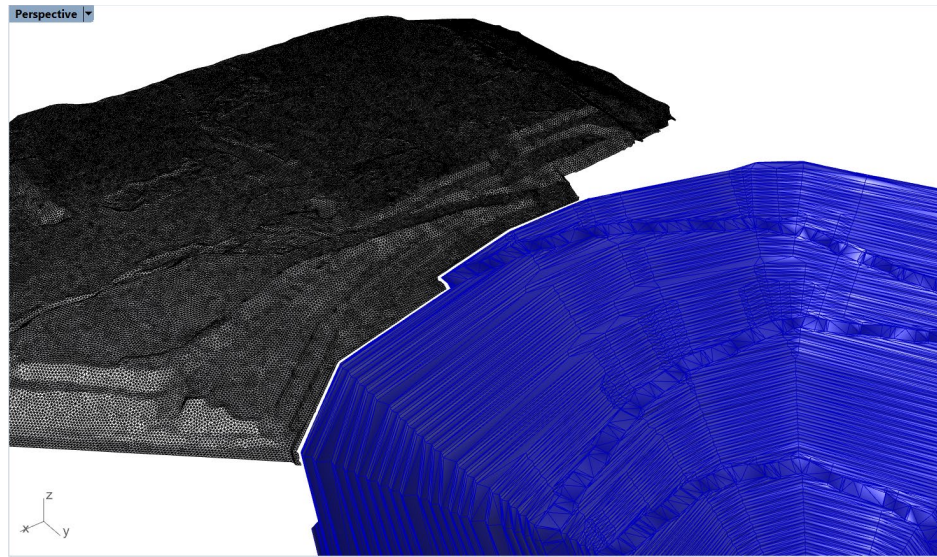


Figure 79: A gap between the pit and pit dump meshes.

14. Select the pit dump mesh and remesh it with **\_GSurf: Mode = Tri, MinEdgeLength = 20, MaxEdgeLength = 100, and RidgeAngle = 60°**. This operation will further reduce the number of mesh faces and smooth it, making it easier to work with the mesh. Remeshing may take a minute.
15. Turn off “Pit” layer and turn on “IniTopo” layer. Double-click on the viewport icon in the top-left corner of the model view (e.g., **Top**); four standard viewports will be shown.
16. Select the pit dump mesh and use **\_GExtend** → *FreeExtend* option. The mesh boundary will be highlighted in magenta; select the whole boundary (**Ctrl+A**) and press **Enter**. Now the boundary can be dragged to a desired location, and the mesh will be extended to the new boundary. While the cursor is in drag mode, move it to *Front* or *Right* viewports (Figure 80). Then press and hold **Shift** and drag the mouse to position it under the topo mesh as in Figure 80. When the whole boundary is below the topo mesh, release the mouse button. This will extend the dump mesh along its boundary strictly downwards (Figure 80, Figure 81).

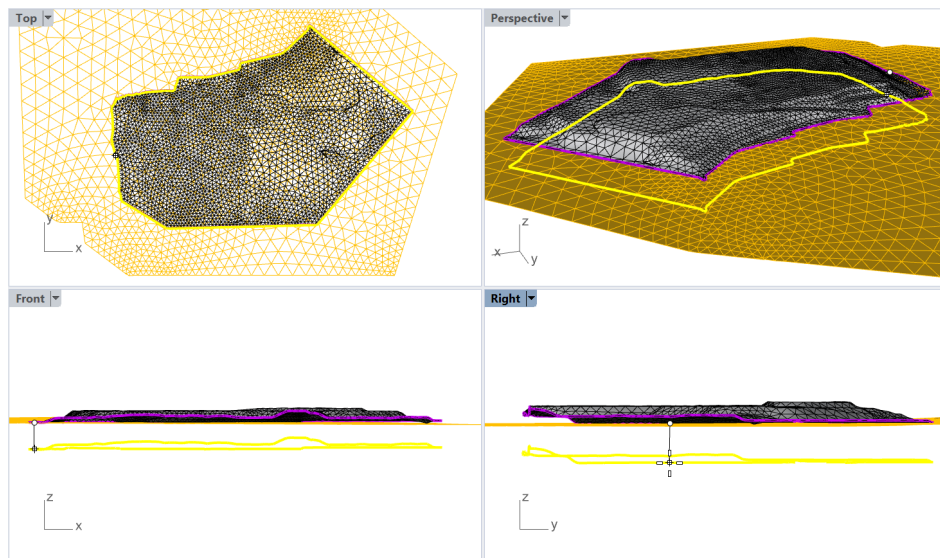


Figure 80: Extension of the pit dump mesh downward along its boundary (*FreeExtend* mode).



The same operation can be done by using **\_GExtend** → *ExtendSelectedBoundary* and specifying *ExtendLength* = 150 and *Direction* = *AlongVector*. Select the whole boundary and type 0,0,0 for the beginning of the extension vector and 0,0,-1 for the end. This will extend the mesh downwards.

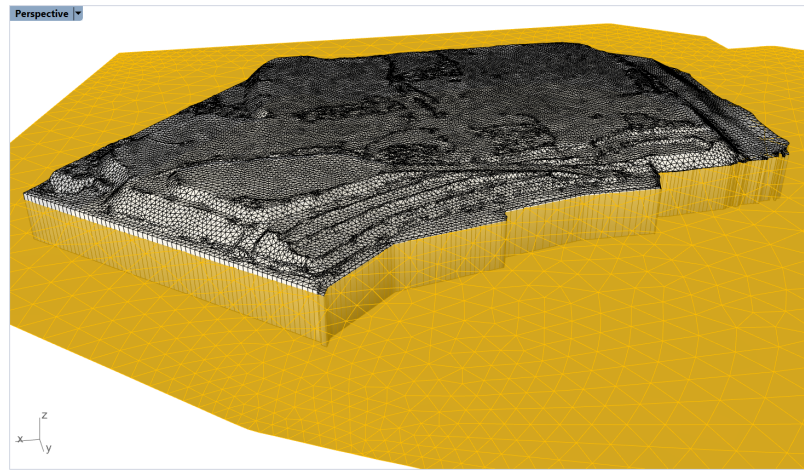


Figure 81: Extended pit dump mesh fully intersecting topo mesh.

Now the dump and topo meshes fully intersect each other. A similar process will be used to extend part of the pit mesh; however, the pit mesh must be remeshed first.

17. Turn on the “Pit” layer and turn off all others. Remesh the pit mesh with **\_GSurf**: *Mode* = *Tri*, *MinEdgeLength* = 10, *MaxEdgeLength* = 40, and *RidgeAngle* = 20°. Restore all four viewports by double-clicking on the current viewport icon (if any viewport is maximized). Turn on layer “Box” and switch *Top* viewport to Ghosted view. Select the pit dump mesh and use the **\_GExtend** → *FreeExtend* option. Select only the top part of the pit boundary in such a way that the selection goes a little beyond the box but not to the corner points of the top boundary<sup>4</sup>, as in Figure 82. Like before, move the cursor to *Front* or *Right* viewports and while holding **Shift**, drag the mouse upward, so the extended mesh is sure to intersect the topo mesh (the exact extension length does not matter). The resulting mesh should look similar to Figure 83.
18. Navigate to the *Layers* pane and unlock layer “Box” (click on the lock icon). Turn on the layer if it is turned off. Mesh the box polysurface with the **\_Mesh** command, using simple controls and the fewest number of polygons. After meshing, the polysurface is still selected. Delete it.

At this point, the pit, pit dump, topo, and box meshes are in full contact. Now they can be easily intersected and excessive parts removed. The approach described below intersects and merges meshes and then separates sub-meshes along nonmanifold edges<sup>5</sup>.

<sup>4</sup> Extending a part of the top pit boundary (not whole) allows for later extraction of sub-meshes along non-manifold intersection lines. If the whole top boundary of the pit mesh is extended, the faces around the corners may connect without forming non-manifold connections.

<sup>5</sup> An alternative approach is to use *Rhino*’s **\_MeshSplit** command. However, the command does not work properly in *Rhino* 6; it was fixed in *Rhino* 7. If using *Rhino* 7 or later, the user can split one mesh by another provided they are conformal (intersect with **GInt**, *OutputMesh* = *Separated*). This approach is simpler than the one outlined here.

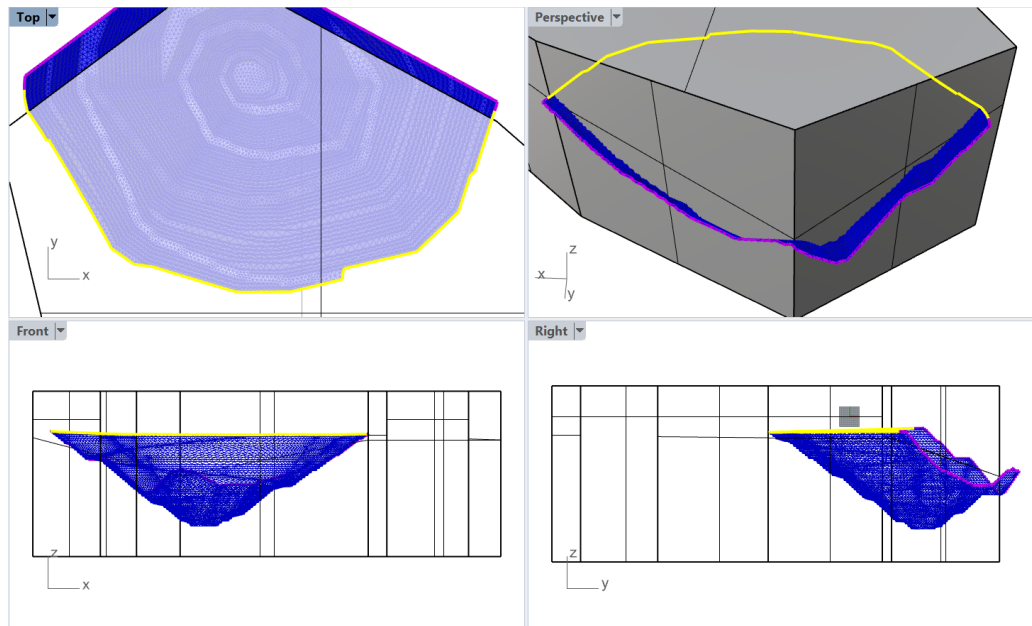


Figure 82: Selection of a piece of the pit boundary for mesh extension.

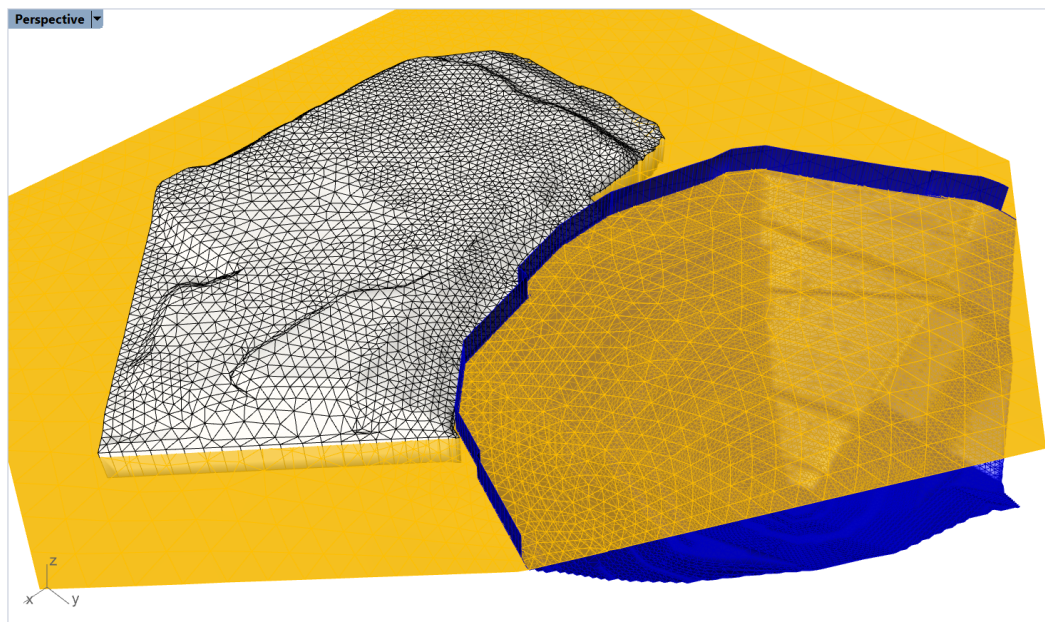


Figure 83: Extended pit mesh. Note that the mesh is not extended all the way to the corner points of its top boundary.

19. Select the pit, pit dump, topo, and box meshes and intersect them with **\_GInt** → *Tolerance* = 0, *AdvancedParameters: OutputMesh* = *Merged*, and keep other parameters at the defaults. This will create a single conformal nonmanifold mesh within the current ("Default") layer.
20. Select the merged mesh and use **\_GExtract** → *AllSurfaces: MaxBreakAngle* = 180 to extract meshes along the nonmanifold edges (the break angle will not have an effect as it is set to 180°).
21. Delete the top part of the extracted box mesh, excessive parts of the topo, pit, and dump meshes that extend through the topo and beyond box mesh. Merge all pieces of the box mesh and all pieces of the topo mesh (including the pieces underneath the dump and overtop of the pit) using the **\_Join**



command. In the *Properties* pane, reassign layer “Box” to the box mesh, layer “IniTopo” to the topo mesh, layer “Dump” to the dump mesh, and layer “Pit” to the pit mesh.

22. Turn off all layers except for the default layer (e.g., “Default”). Check if there are any small pieces of meshes left after the extraction; press **Ctrl+A** to select all of them. Usually, these would be narrow single faces<sup>6</sup> that may not be visible when larger objects are present. Ensure that no more than a few pieces are present (there will most likely be only one piece selected). Delete these pieces. If there are many small pieces, most likely the meshes were not intersected or extracted properly; undo and repeat the last several steps according to the instructions.
23. Turn layers “Box”, “IniTopo”, “Dump”, and “Pit” back on. Select all meshes and use **\_GHeal** → *ShowErrors* to check if there are any internal naked edges. Turn off layer “CLASHING\_FACES” (they will be fixed by remeshing) and type the command **\_SelCrv** to select the outlines of naked edges. The command should report that only four curves are selected. If this is the case, delete the curves while they are selected. If more than four curves are present, there are holes within some of the meshes, which is likely due to incorrect mesh intersection or extraction; undo and repeat the last several steps according to the instructions.
24. All four meshes should be remeshed to make them more uniform and to simplify further work. First, assign a specific element size to each mesh in the hyperlink field of the mesh properties pane (**F3**):
  - The box and topo meshes: *elemsize:75*
  - The pit dump mesh: *elemsize:25*
  - The pit mesh: *elemsize:15*

Then use **\_GSurf**: *Mode = Tri*, *MinEdgeLength = 10*, *MaxEdgeLength = 100*, and *RidgeAngle = 20°* to remesh all four meshes. The resulting meshes should look like that in Figure 84.

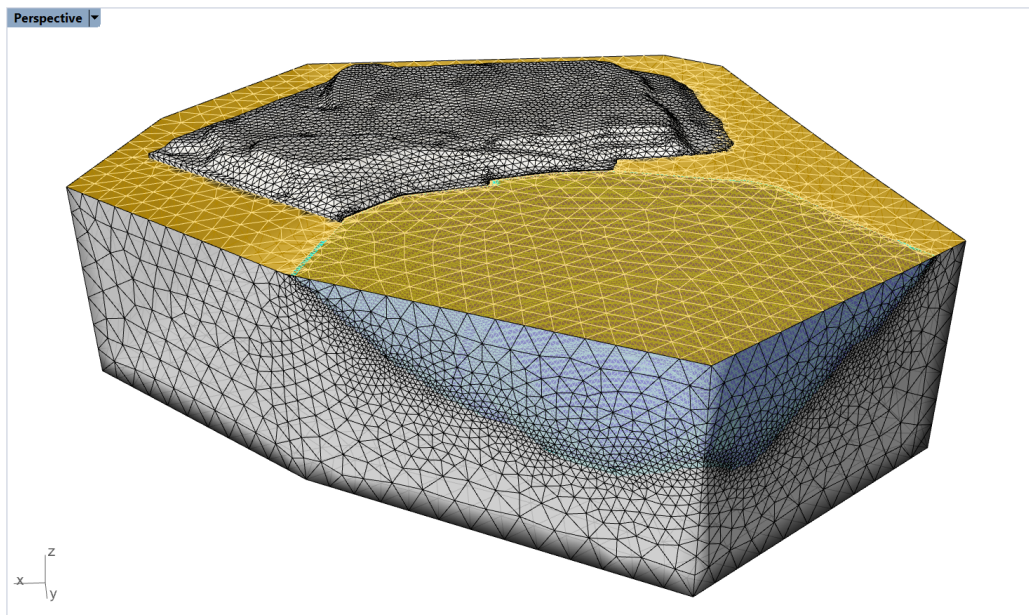


Figure 84: Intersected box, pit, pit dump, and topo meshes (Ghosted view).

<sup>6</sup> If there are extremely small / thin faces around mesh intersection areas (which sometimes happens after intersecting meshes), **GExtract** may extract such small pieces and create separate mesh objects from them.

At this point, a volume mesh could be created from the surface meshes shown Figure 84. However, the model does not yet include faults. The next section provides information about extending, intersecting, and trimming faults before including them into the model.

## Preparation of the fault meshes

Turn on “Faults” layers and turn off all others. Explore the areas where meshes visually terminate on one another and/or use the **\_MeshIntersect** command to see if there is full intersection between meshes. Mesh “Strat1” should terminate on “Fault1” and the rest of the fault meshes should terminate on “Strat2”. Unfortunately, none of these meshes fully intersect/terminate as they should (e.g., see Figure 85). This is a common situation when working with meshes generated from field data. As before, such meshes need to be extended, properly intersected, and then excessive pieces removed.

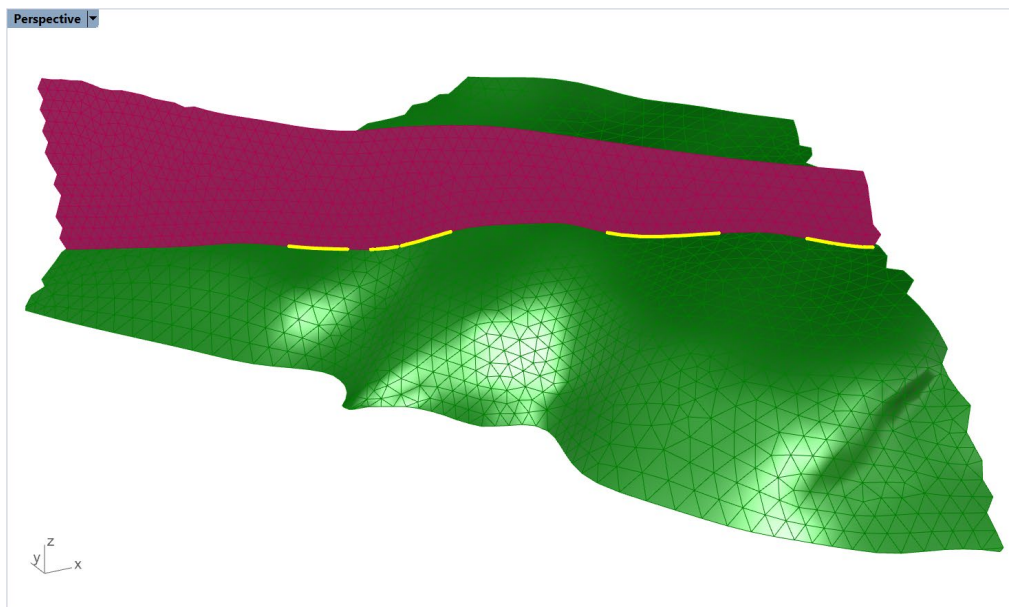


Figure 85: The **\_MeshIntersect** command shows actual intersections (yellow lines) between “Fault4” and “Strat2” meshes. Other parts of the “Fault4” mesh do not intersect the “Strat2” mesh, leaving minor gaps.

25. Turn on only “Strat1” and “Fault1” layers and extend the “Strat1” mesh only along the edge that visually connects to the “Fault1” mesh. Use **\_GExtend** → *ExtendSelectedBoundary: ExtendLength = 20, MeshType = Merged*. Additional segments of the boundary can be included in the selection by clicking on them and can be removed when holding **Ctrl** and clicking. Ensure that the extended portion fully intersects and is enclosed within the “Fault1” mesh (as in Figure 86).
26. Repeat this operation to extend meshes “Fault2”, “Fault3”, and “Fault4” so they extend through the “Strat2” mesh. Use the same **\_GExtend** parameters as in the previous step. Ensure that the extended parts fully intersect and are enclosed within the “Strat2” mesh (zoom in at intersection corners).
27. Turn on layers “Box”, “Strat1”, and “Fault1” and turn off all other layers. Intersect all three meshes with **\_GInt** → *AdvancedParameters: OutputMesh = Merged, Tolerance = 0*, and keep other parameters at the defaults. This will create a single conformal nonmanifold mesh within the current (“Default”) layer.

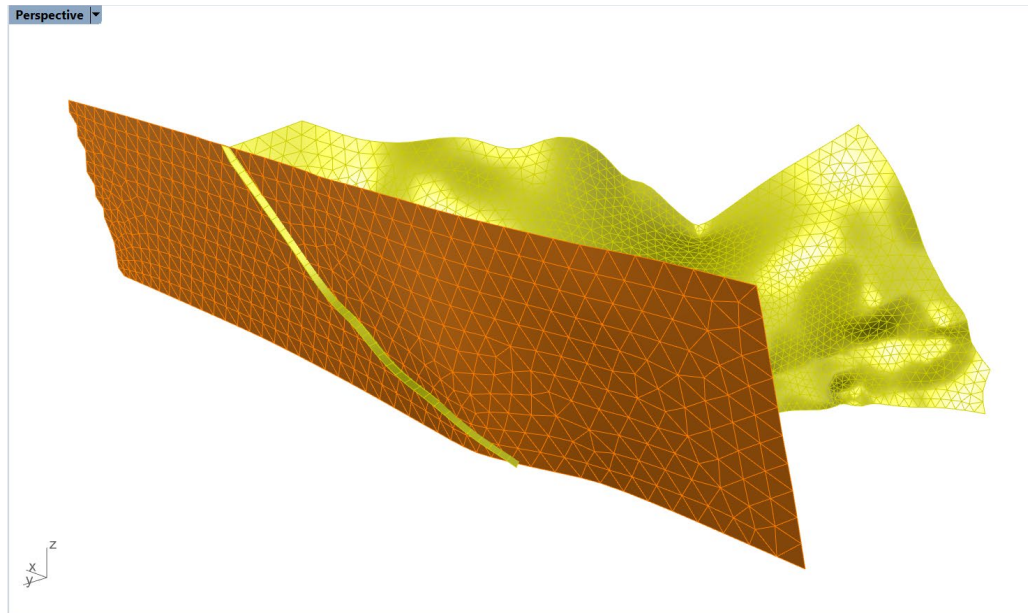


Figure 86: Extension of “Strat1” mesh along the part of the boundary closest to “Fault1”.

28. Use **\_GExtract** → *AllSurfaces: MaxBreakAngle* = 180 to extract meshes along nonmanifold edges (break angle will not have an effect as it is set to 180°).
29. Delete excessive parts of meshes located outside the box. Hide the box with the **\_Hide** command and delete the excessive part of the “Strat1” mesh extending through the “Fault1” mesh. In the *Properties* pane, reassign layers “Strat1” and “Fault1” to the corresponding meshes.
30. Show the box using the **\_Show** command. The box is still in the “Default” layer, no need to reassign it yet. Turn off layers “Strat1” and “Fault1” and turn on layers “Strat2” and “Fault2”. Repeat the operations from the previous three steps to intersect, separate, and remove excessive mesh parts. Note that the box will be split in two pieces during these operations. There is no need to join them and reassign layers “Box” and “Strat2” until all meshes are intersected and excessive parts removed.
31. Repeat the previous four steps for “Fault3” and “Fault4” meshes. Remember to include the box when intersecting (in case it was hidden earlier) and to delete thin mesh pieces underneath the “Strat2” mesh.
32. Join two pieces of the box mesh and place them into the “Box” layer (**\_Join**). Join (up to five) pieces of the “Strat2” mesh and place them in the “Strat2” layer. Turn on all “Faults” and “Box” layers only.

Now there should only be seven meshes, as in Figure 87 (check with **Ctrl+A**). All these meshes are perfectly intersected between one another and are ready to be intersected with the pit, pit dump, and topo meshes, followed by final remeshing and volume meshing.

### Final mesh intersection and remeshing

33. Turn on all the layers. Select all meshes and intersect them using **\_GInt** → *Tolerance* = 0.001, *AdvancedParameters: OutputMesh* = *Separated*.
34. Remesh all meshes using **\_GSurf**: *Mode* = *Tri*, *MinEdgeLength* = 10, *MaxEdgeLength* = 75, *RidgeAngle* = 10° and *AdvancedParameters: MaxGradation* = 0.2, *Optimization* = 10, *ShapeQuality* = 0.9, and *OutputMesh* = *Separated*. The results are shown in Figure 88.



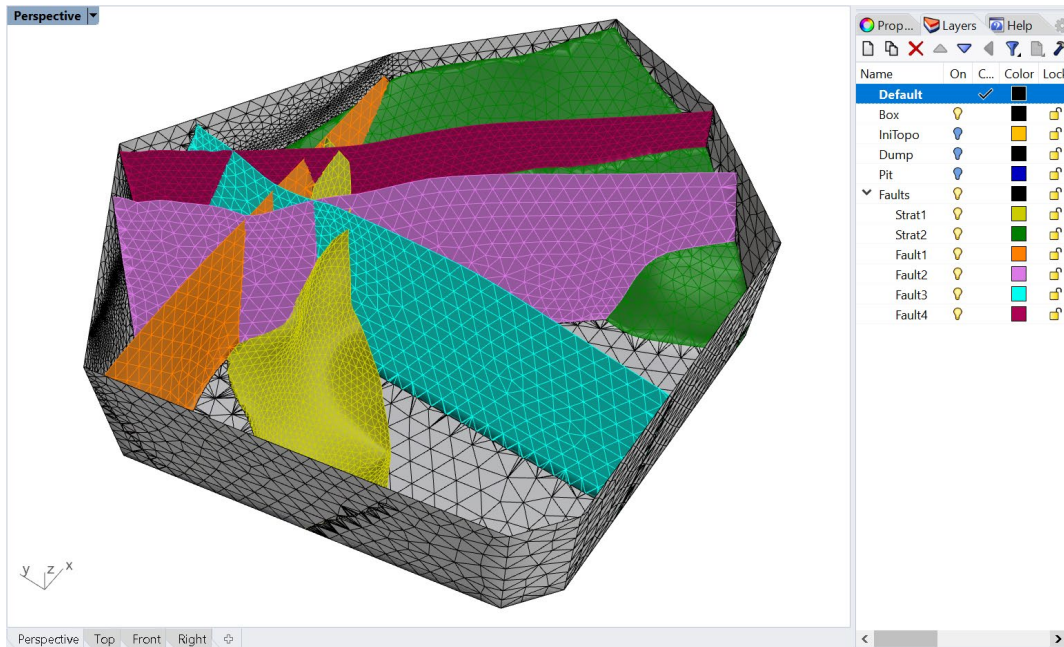


Figure 87: Intersected and trimmed “Faults” meshes.

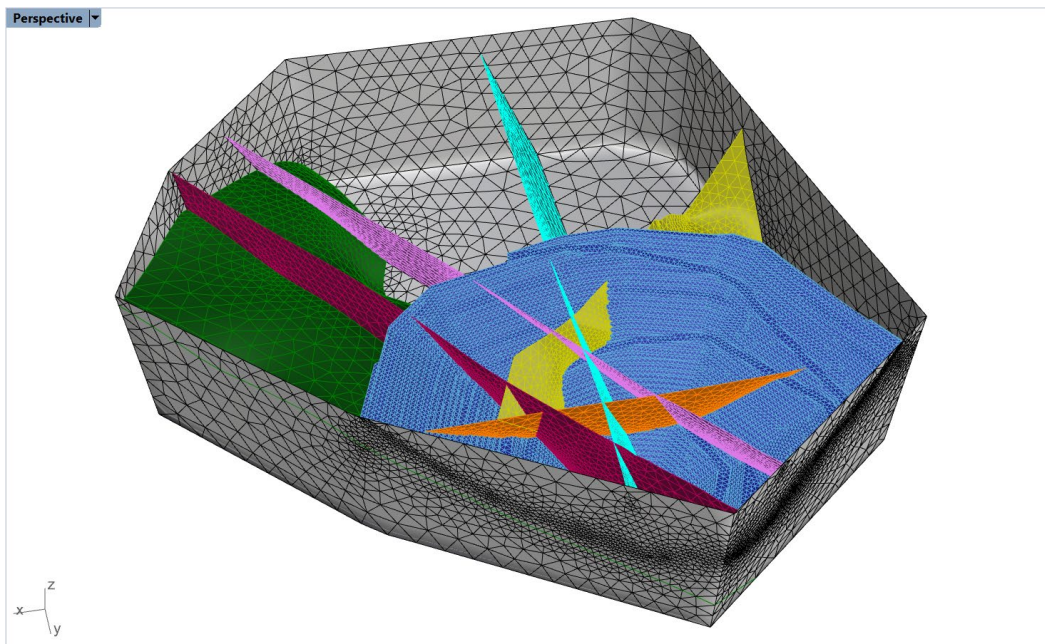


Figure 88: Final triangular surface meshes (the pit dump and topo meshes are not shown).

35. Verify that the final surface meshes do not contain any holes or clashing faces by calling **\_GHeal** → **ShowErrors** (select all meshes first). After that, type **\_SelCrv**. *Rhino* should report that only 10 curves are selected.

If more than 10 curves are selected, that indicates the presence of nonconformal faces, internal holes, and/or clashing faces. To locate issues, hide all layers except for “NAKED\_EDGES” and “CLASHING\_FACES” and start deleting large curves corresponding to mesh boundaries (there should be 10 curves). Any curves left indicate potential issues.

If any issues are present, investigate each problematic area. It may happen that some nearly overlapping faces from different meshes got merged into a single face during remeshing. In such a case, one of the meshes may contain a hole. An example of this is shown in Figure 89, which was obtained when conducting step 24 of this tutorial with *RidgeAngle* = 10°. In this case, nearly overlapping faces from the pit mesh and the topo mesh got merged after remeshing, and the resulting single face got assigned to the pit mesh only. This is not a critical issue as meshes still form a watertight domain and the volume mesh can be generated. The only negative consequence is that the face group designating mesh surface within the volume mesh may contain a hole or the wrong group would be assigned to the “overlapping” face<sup>7</sup>. Such situations are more often encountered when working with models containing a large number of faults intersecting at very small angles (so faces from one mesh are very close to the faces from another mesh near intersections; depending on which remeshing parameters are used, some faces may become merged between meshes).

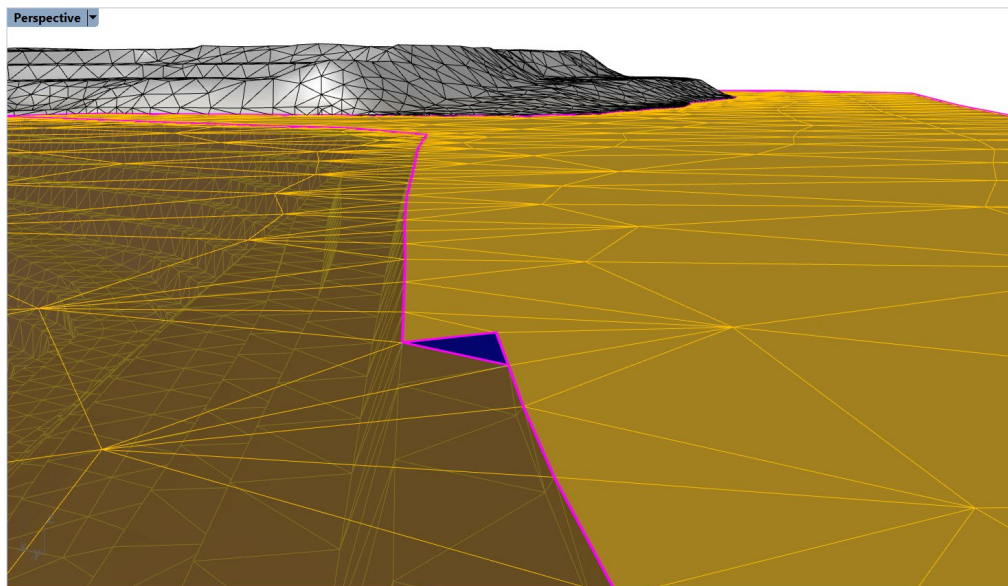


Figure 89: Overlapping faces (in blue) created a hole in the topo mesh.

36. Assign names in the meshes *Properties* pane as desired. These names will be transferred as named face groups (surfaces) to the volume mesh (currently, only for *FLAC3D* and *3DEC*). See details about group naming and surface naming in the *Griddle 2.0 User Manual*.
37. Create a tetrahedral volume mesh in *FLAC3D* binary format. Keep all other **GVol** parameters at default values. Tetrahedral volume mesh generation is usually a fast process<sup>8</sup>, and it is used here. In case *Griddle* issues a message about the presence of naked edges, read the message and press **Yes** to continue. The final mesh is shown in Figure 90.

<sup>7</sup> Fixing the hole created by overlapping faces will not resolve the situation, as no volume elements are created between the overlapping faces (there is zero volume between them).

<sup>8</sup> In general, it is much easier to generate pure tetrahedral meshes for very complex models, as filling any random volume with tetrahedrons is a simpler process than filling a volume with a combination of various shapes. Besides, the average quality of tetrahedral elements in such cases is typically higher than for hex-dominant meshes.

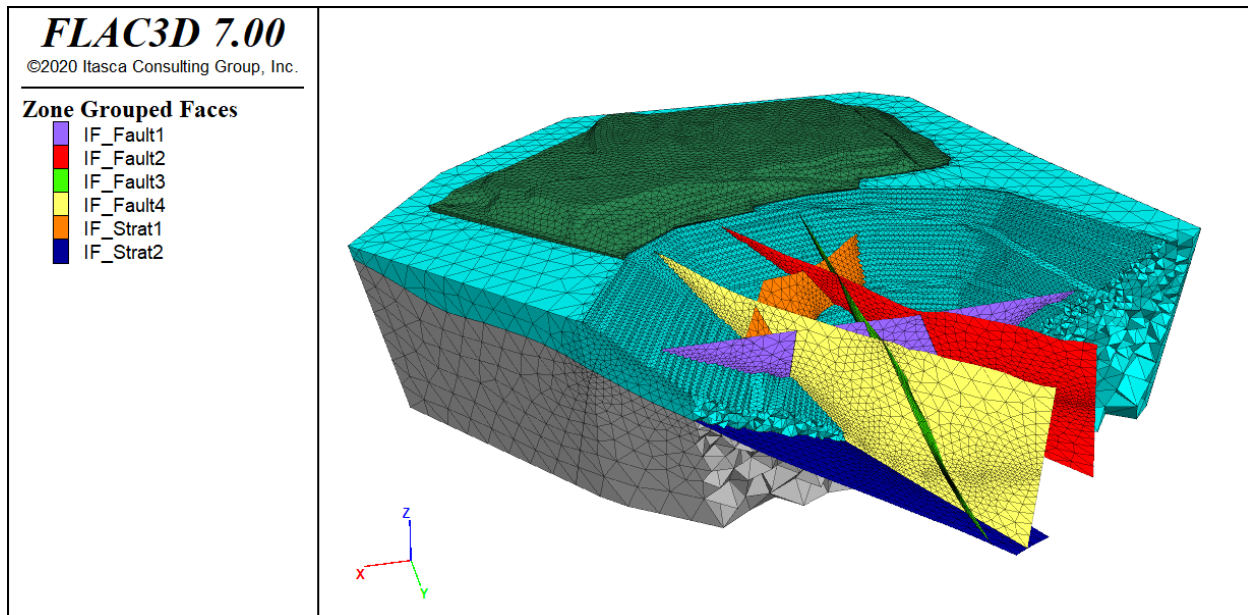


Figure 90: Tetrahedral volume mesh of open pit mine model.

As an optional exercise, remesh all meshes again using the same **GSurf** parameters as in step 34 but with *QuadDom* mesh type. Then generate a hex-dominant (HexDom) volume mesh in *FLAC3D* binary format (which may take several minutes for such a large model). Save the file under a different name than the previous file. Compare log outputs from **GVol** for both cases to see what kind and how many elements are generated.

When generating the hex-dominant volume mesh for the model, **GVol** will produce about half of the total number of elements compared to a tetrahedral mesh. However, a HexDom mesh contains a large number of undesirable pyramidal elements<sup>9</sup>. For such complex models, engineers often generate pure tetrahedral meshes and then split each tetrahedron into several hexahedrons within *FLAC3D* to obtain a pure hexahedral grid (pure hexahedral grids yield significantly more accurate numerical results in *FLAC3D*).

Note that this model is not suitable for outputting in *3DEC* deformable blocks format as faults do not connect to the topo mesh, and, thus, they do not cut blocks within the pit or the rest of the model. *3DEC* rigid block format could be used instead, but the surface meshes will need to be significantly coarsened first to produce a coarser volume mesh that can be loaded into *3DEC* (as *3DEC* will try to detect and create a contact between each rigid tetrahedral element/block, resulting in an enormous amount of block contacts).

<sup>9</sup> *FLAC3D* grid densification becomes rather challenging if pyramids are present.