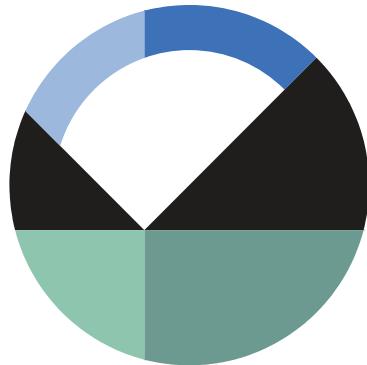


# 3D Model Building Basics



GEO-SLOPE International Ltd. | [www.geo-slope.com](http://www.geo-slope.com)

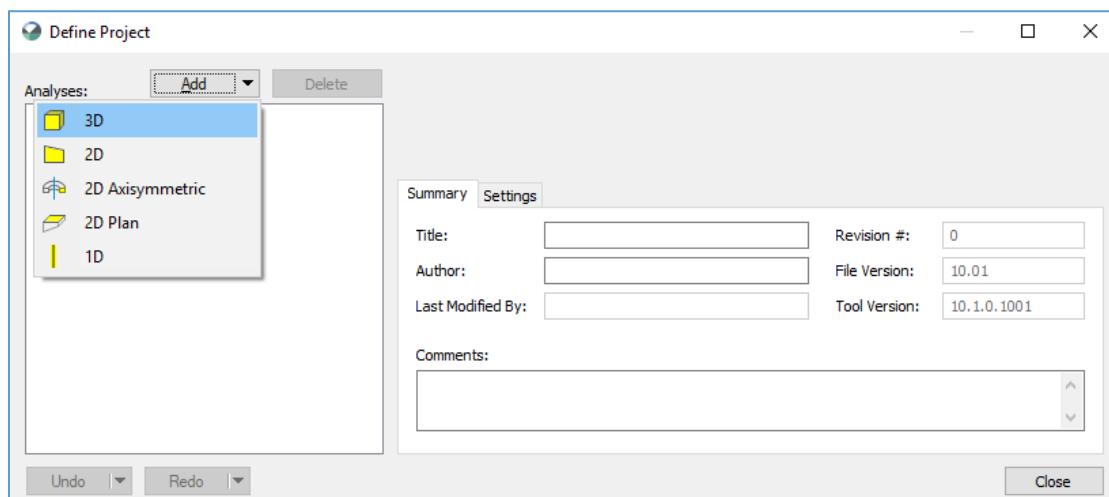
1200, 700 - 6th Ave SW, Calgary, AB, Canada T2P 0T8  
Main: +1 403 269 2002 | Fax: +1 888 463 2239

## Introduction

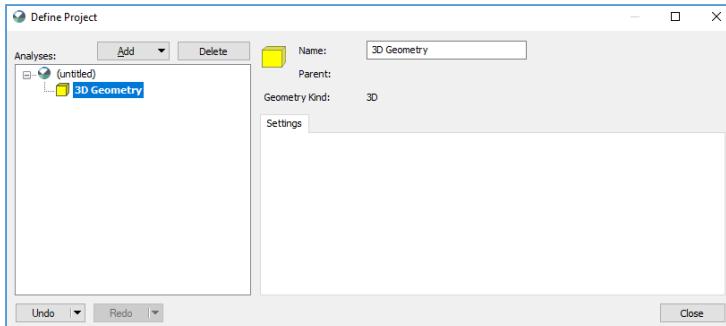
This is the first of four examples demonstrating the basics of setting up three-dimensional geometries in BUILD3D. This example summarizes how to create a 3D domain consisting of two solids, assign materials and boundary conditions, and mesh the model in BUILD3D.

## Setting up an Analysis

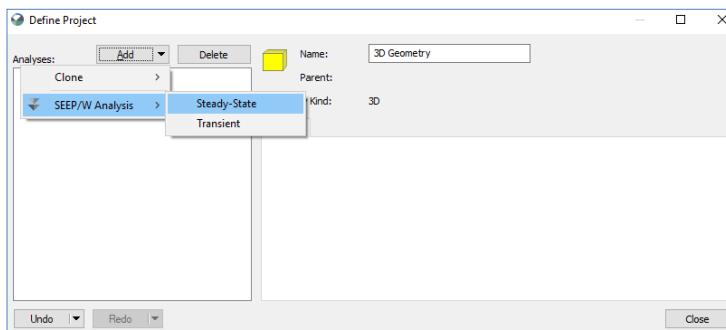
In GeoStudio, we will begin by specifying a 3D geometry and associating a seepage analysis to this geometry. To set up a new GeoStudio analysis, we will go to **File→New**, select the Metric template, and click the Create button. The Define Project dialog appears. This dialog allows you to add geometry and analyses to your GeoStudio project. Click on the Add button and select 3D from the list. This will indicate to GeoStudio that you wish to create 3-dimensional geometry. We then must specify which solver, or product, we wish to use on this geometry. Currently 3D seepage analysis is offered for 3D geometry. The other geometry options can be solved using any of the GeoStudio products.



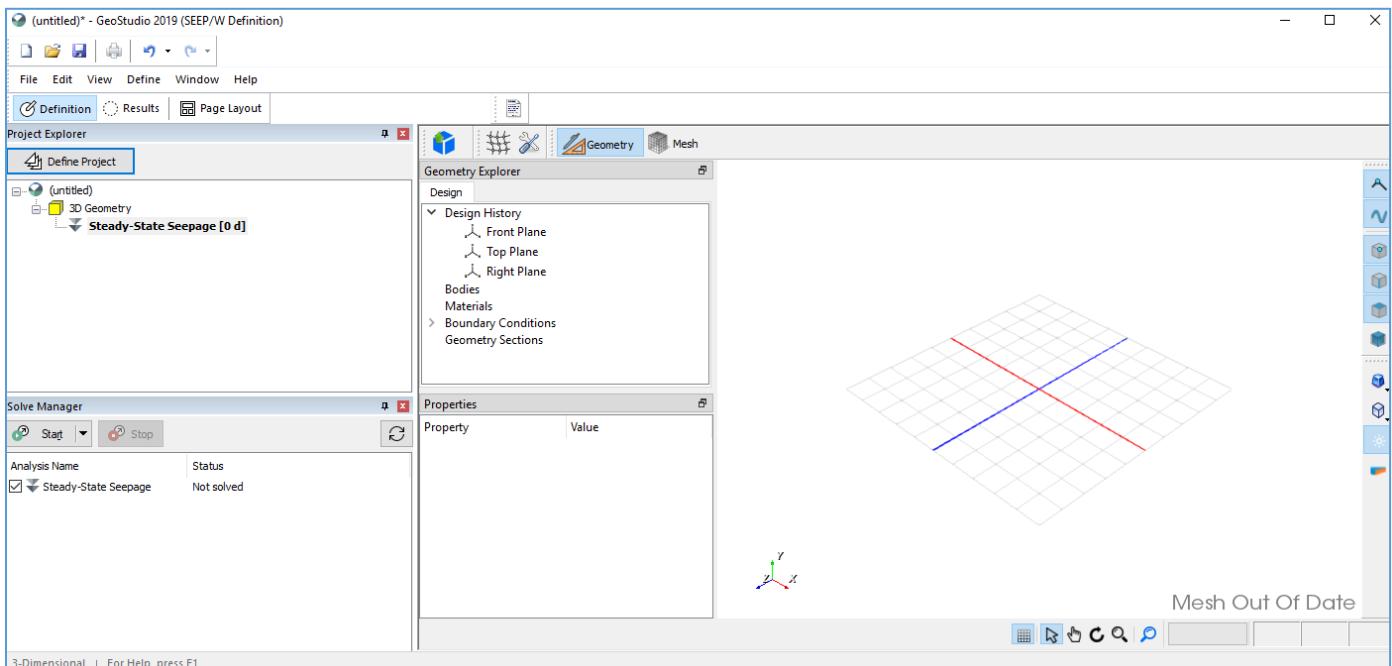
To add a seepage analysis to the 3D geometry, we will click on the Add button in the Define Project dialog, while the 3D Geometry icon is selected.



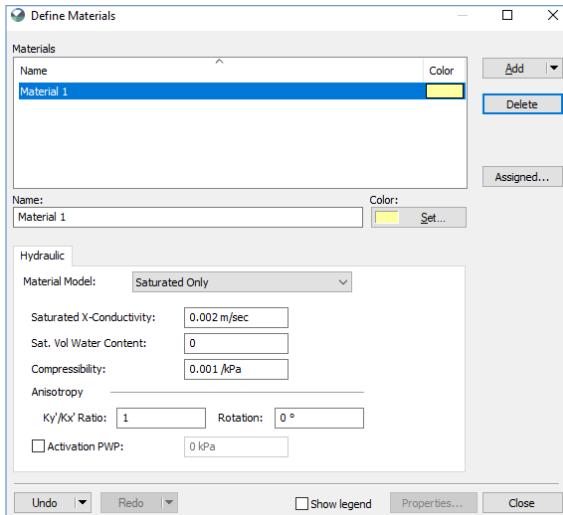
We will create a Steady-State SEEP/W analysis. Click on the Close button to save your selection.



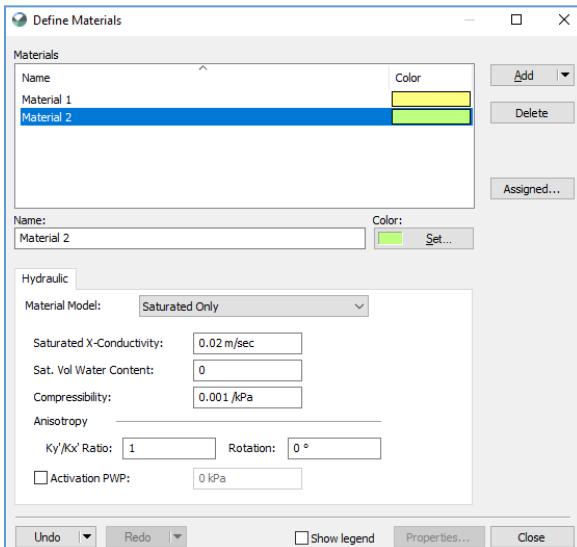
Your GeoStudio windows should have a similar configuration as shown here. The Define Project pane indicates that there is a 3D seepage analysis associated with a 3D geometry.



We will now create two materials and two boundary conditions to use in the 3D seepage analysis. To define the materials, go to Define→Materials, select Add→New, change the name of the material, and select the Saturated Only material model. We will set the saturated hydraulic conductivity in the x dimension to be 0.002 m/s, and the compressibility to 0.001 /kPa. We do not need to specify a volumetric water content as this is a steady-state analysis and thus, the water stored in the domain does not change over time.

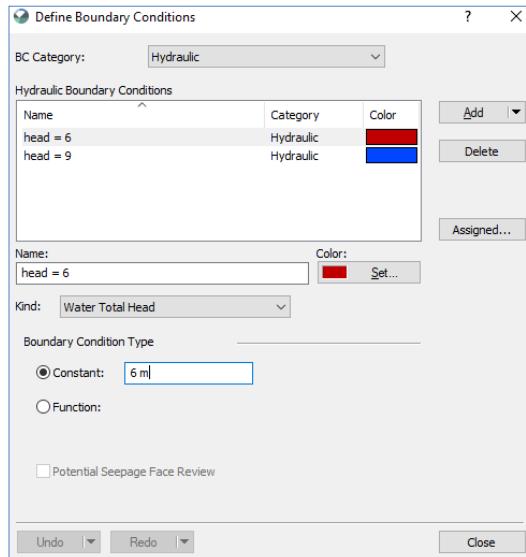


Now let us add a second material by pressing the Add button and selecting the Clone option. After modifying the material name, change the hydraulic conductivity value to 0.02 m/s and then click the Close button.



To establish the boundary conditions, go to Define→Boundary Conditions, add a new hydraulic boundary condition, and change its name to 'head = 6m'. Select the Water Total Head option from the dropdown menu and set it to a constant value of 6 m. Let us change the color associated with this

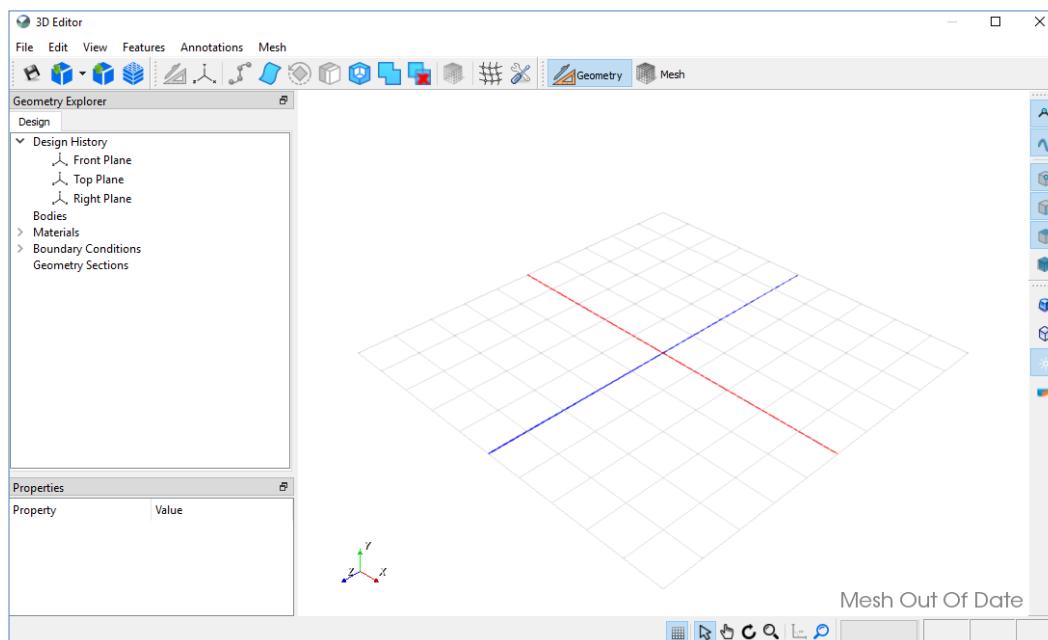
boundary condition so it is different from the default options. We will clone this boundary condition, modify its name to ‘head = 9m’, change the total head value to 9 m and select a different color. We can now close this window.



In this example, we will use the defined materials and boundary conditions in our 3D model construction to demonstrate features, but not to demonstrate the analysis of a 3D problem. Before continuing, let us save the model by going to **File→Save As** and naming our file ‘3D Model Building Basics.gsz’.

## Creating the 3D Geometry

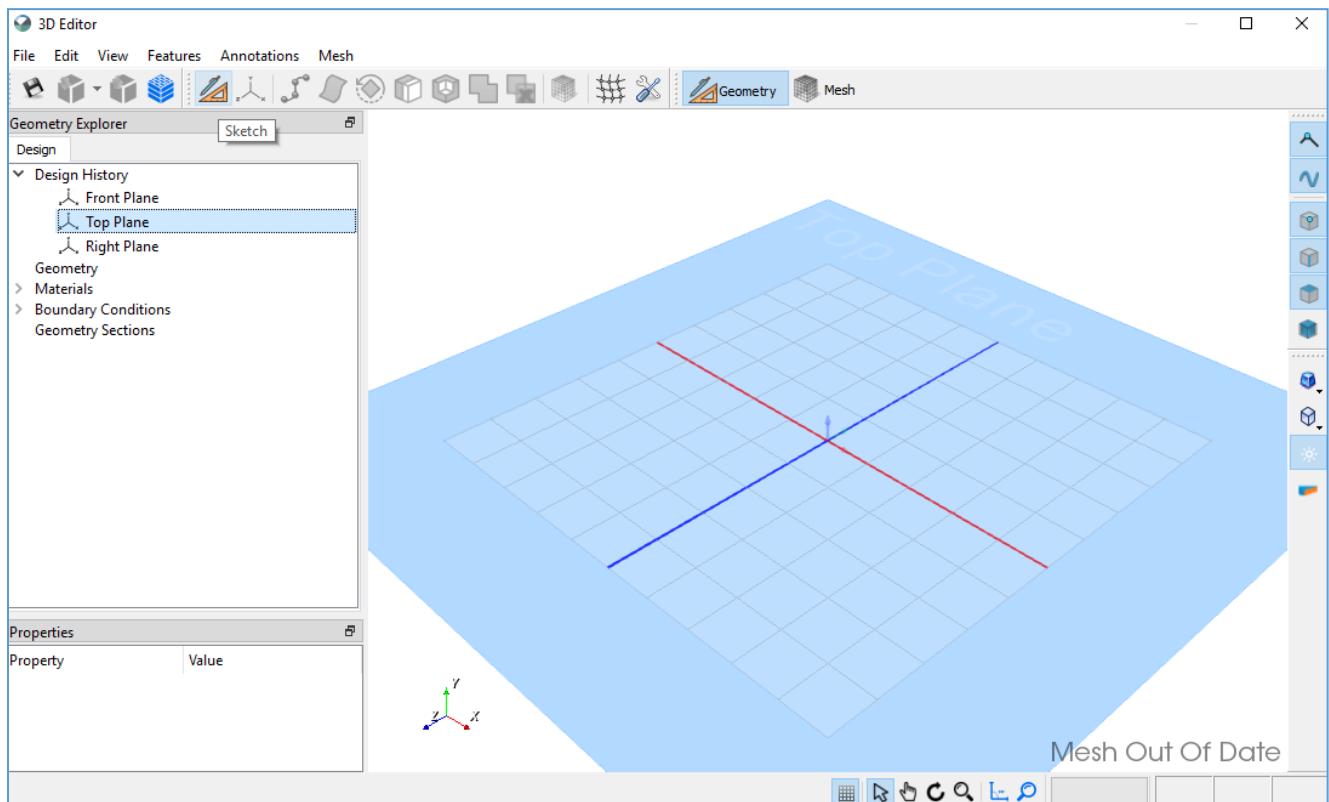
Let us now create the 3D geometry using the features available in GeoStudio BUILD3D. To open the 3D editor, select **Define→Launch BUILD3D**. Ensure the Geometry icon is selected.



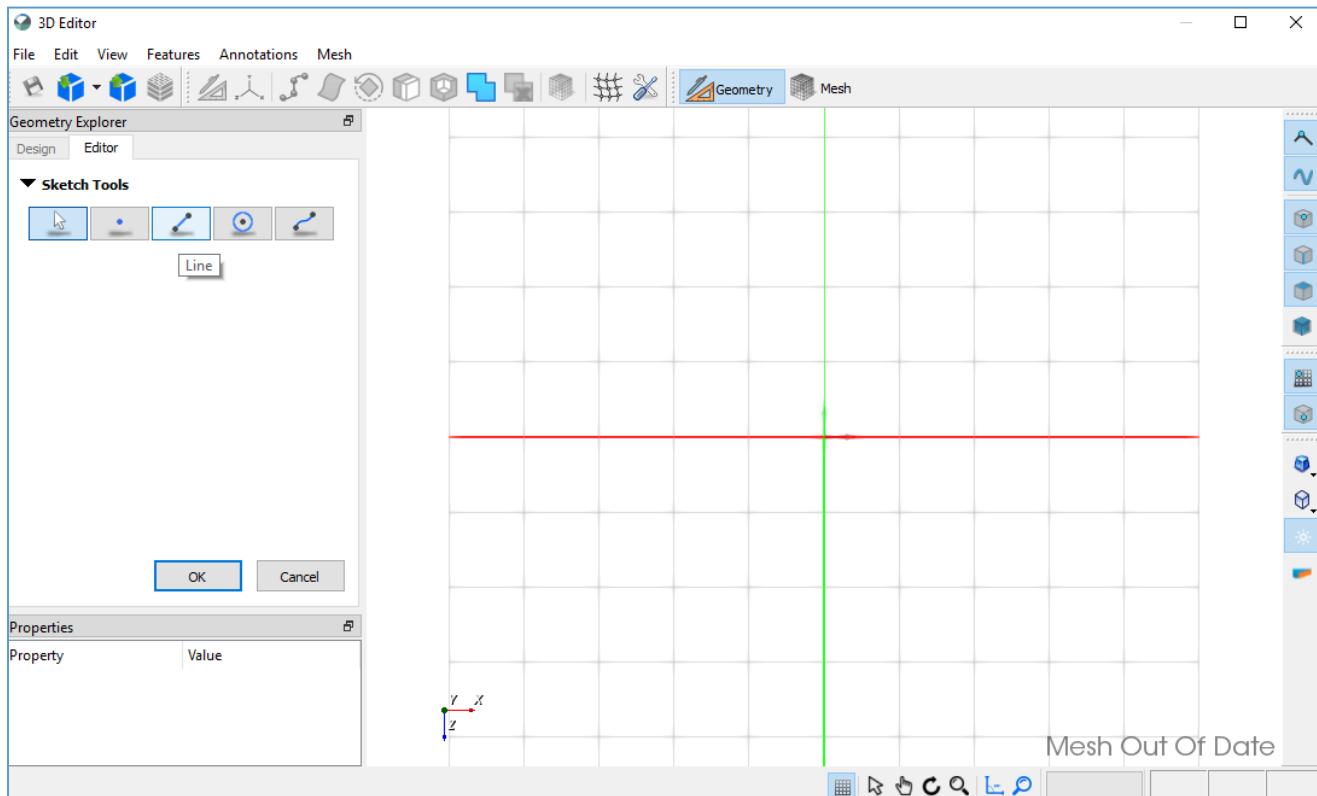
GeoStudio BUILD3D offers the ability to import geometry from other sources but this feature will be explored in the ‘Imported Surfaces and Swept Sketches’ example. The 3D editor includes a Geometry Explorer window and a Properties window. The Geometry Explorer contains items used to define your model, such as geometry, materials, and boundary conditions. The Properties Window reflects the properties of selected items.

In the Geometry Explorer you will see three construction planes: Front Plane, Top Plane, and Right Plane, which assist in drawing sketches in the 3D model space. The Front Plane is the  $z=0$  plane, Top Plane is the  $y=0$  plane, and the Right Plane is the  $x=0$  plane. These sketch planes represent local  $xy$  coordinate systems. It is important to note that GeoStudio assumes gravity acts in the negative  $y$  direction in both 3D and 2D analyses. This direction is fixed so models must be constructed accordingly.

We will sketch a simple shape on the Top Plane and extrude it perpendicular to the construction plane to create our first solid. To do so, select the Top Plane and click on the Sketch Tool icon.

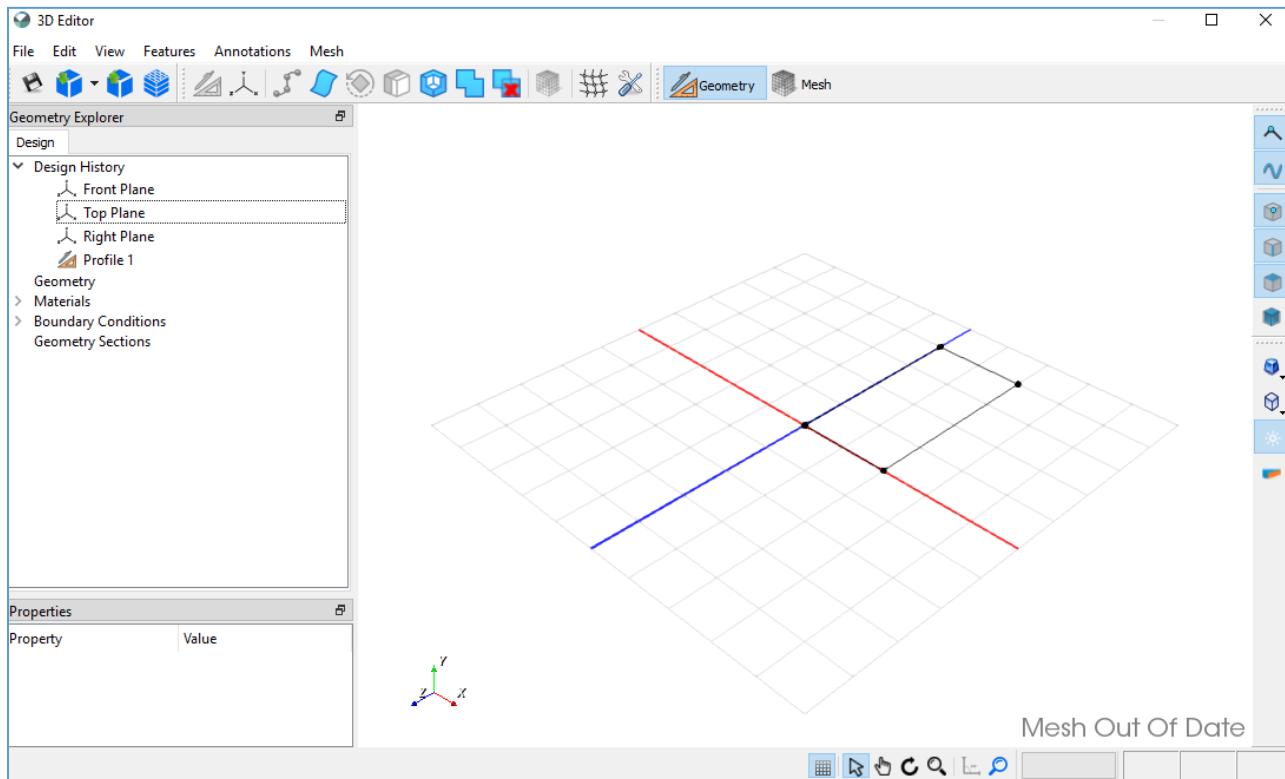


A 2D window will open which allows you to draw a sketch on the  $y=0$  plane. Select the Line icon and move your mouse cursor to the sketch plane where the grid appears. As you move your mouse, you will see 2D coordinates of your mouse position on the plane. As your cursor approaches a grid line the tooltip also displays 'GSnapped' which stands for grid-snapped, indicating that your mouse position aligns with a grid line.

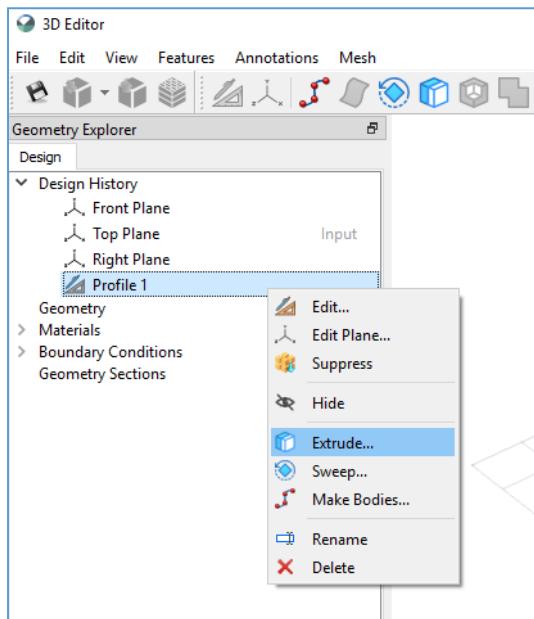


Create a rectangular shape, by clicking your mouse at  $(0,0)$ ,  $(2,0)$ ,  $(2,4)$ ,  $(0,4)$  and then pressing the right mouse button to close the rectangle. Alternatively we could have clicked our mouse on position  $(0,0)$  again to close the rectangle. Click the OK button in the Sketch Tools window to accept your changes.

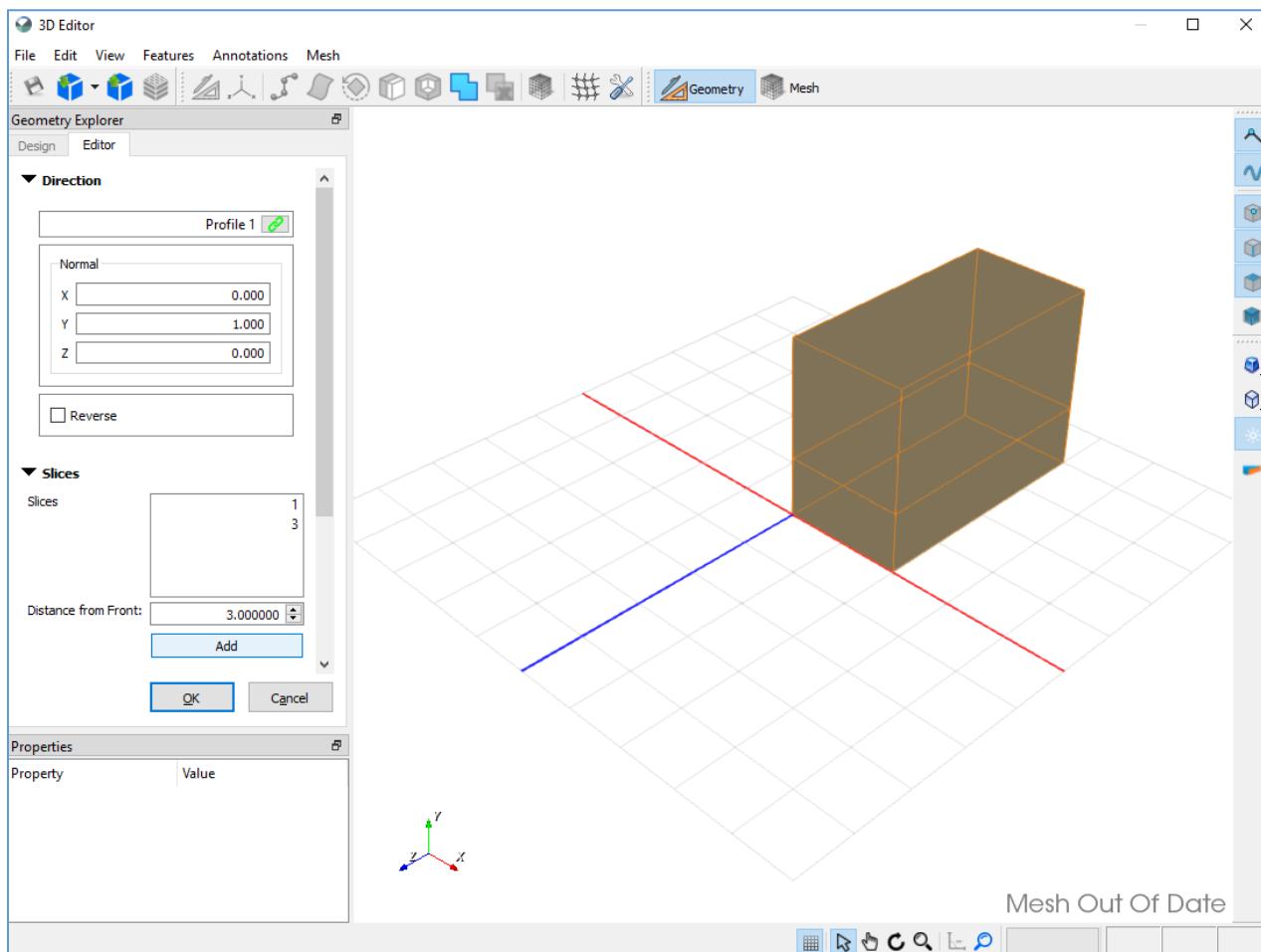
The rectangle is visible on the  $y=0$  plane, and a Profile 1 item appears in the Design History.



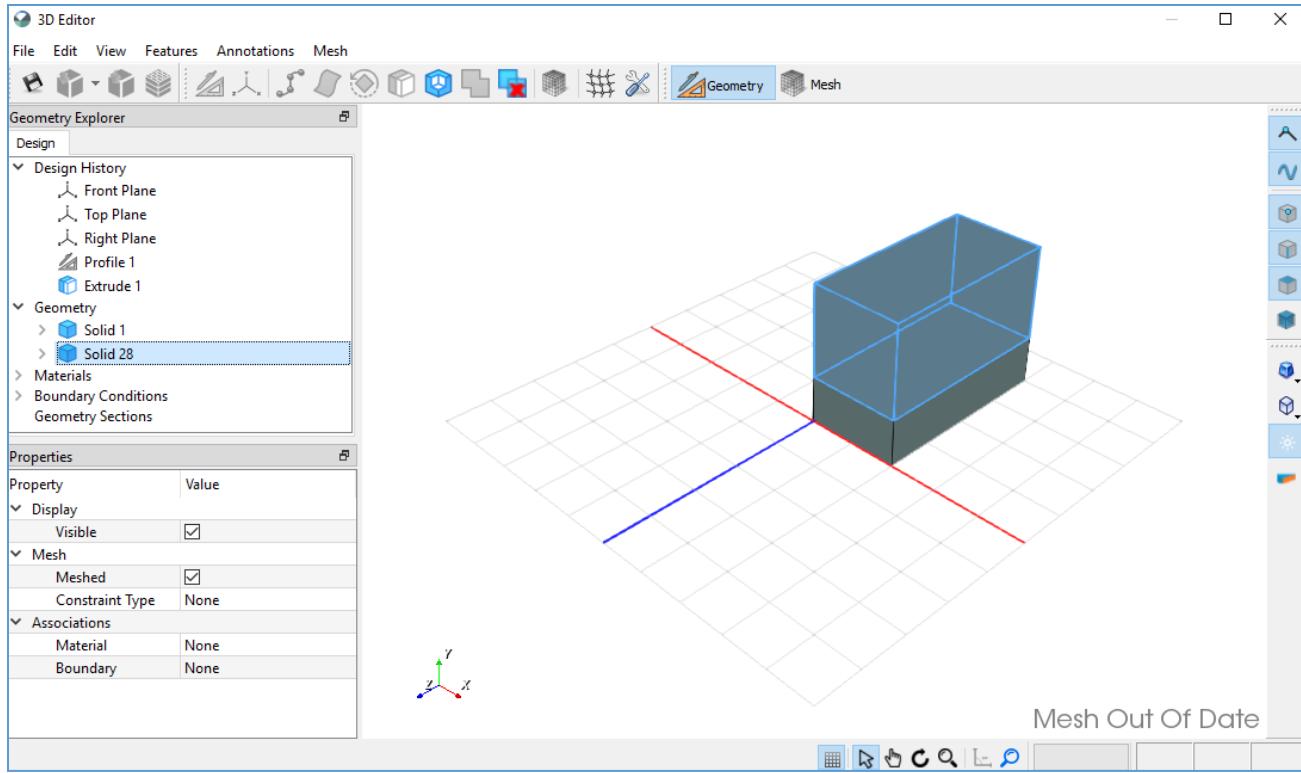
We can extrude the sketched rectangle by right-clicking on Profile 1 and selecting Extrude in the dropdown list.



The extrusion dialog appears with options for direction of extrusion and number of slices. An extrusion is always along a straight line given the user-defined direction. The default direction is perpendicular to the sketch plane, in the y dimension. We will leave the extrusion direction as the default. The Slices portion of the window controls the number and spacing of solids you wish to create from your sketch. A default of 1 appears. This indicates that your rectangle will be extruded a distance of 1 metre from the sketch plane. We will add another slice at 3 m by entering 3 in the ‘Distance from Front’ edit box and clicking the Add button. You will see a 1 m thick slice below a 2 m thick slice in the 3D window. The 1 m slice is offset 1 m from the sketch plane. The 3 m slice is offset 3 m from the sketch plane (2 m from where the 1 m slice ends). Click the OK button to accept your changes.

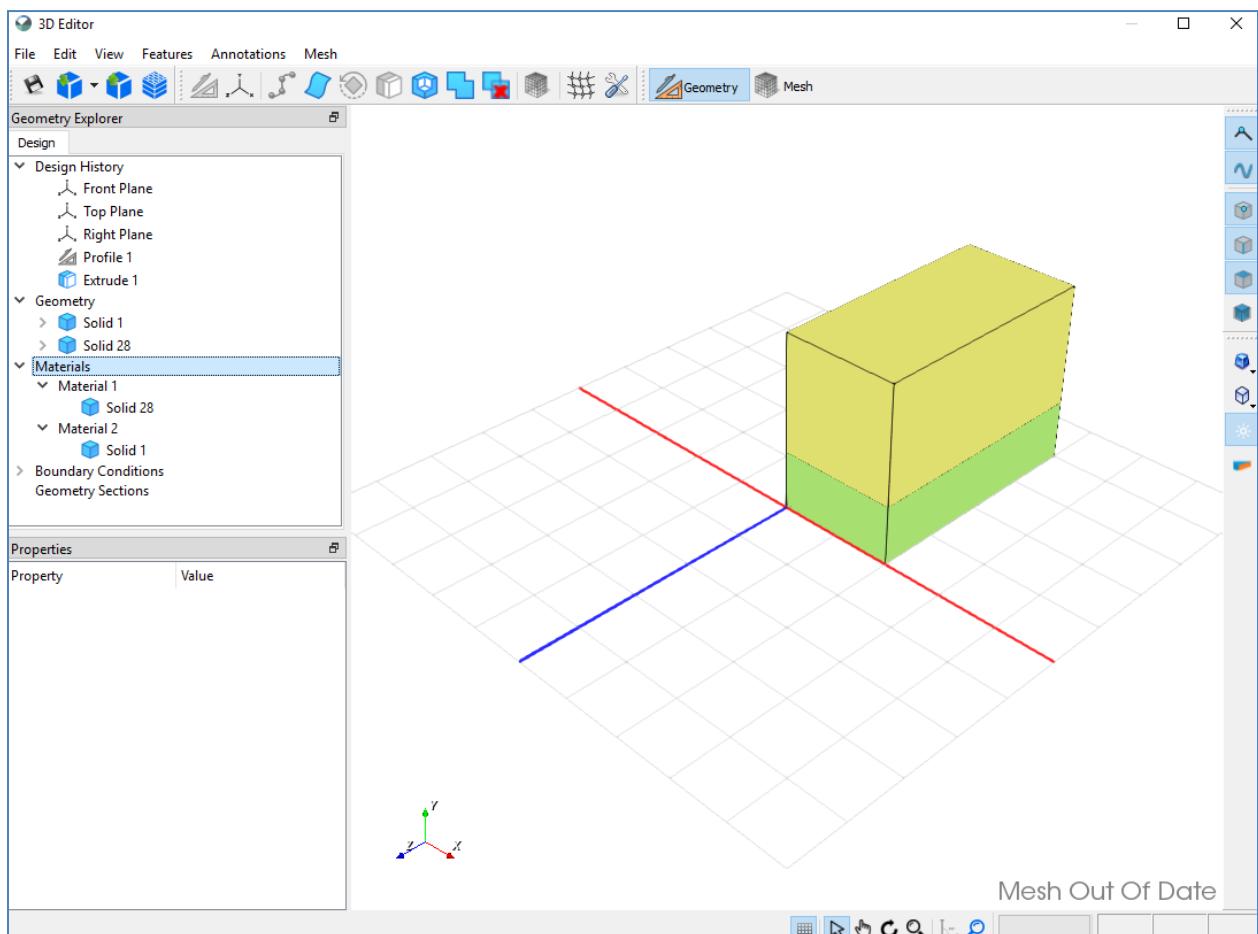
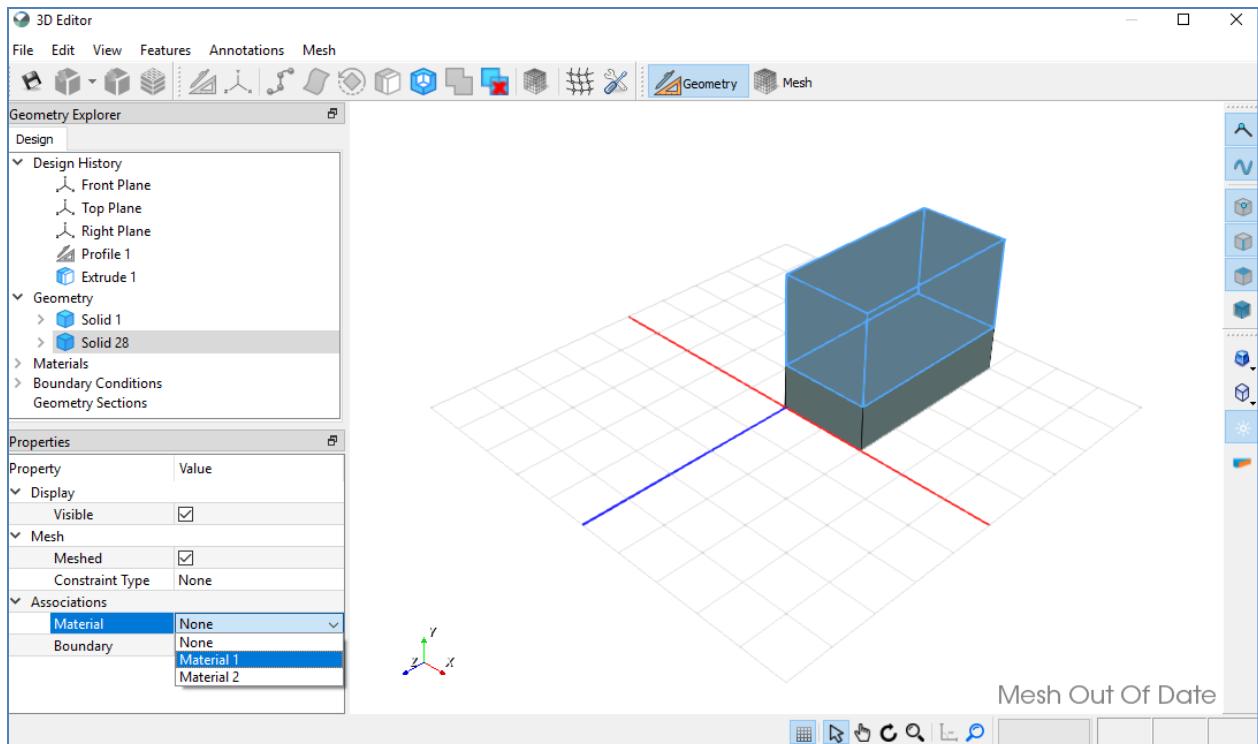


Two solids appear under Bodies in the Geometry Explorer. When selected, the solid is highlighted in the 3D view, and its properties are reflected in the Properties window. If you click the expand arrow beside a solid in the tree view, you will see a solid is composed of Faces. Similarly, Faces are composed of Edges, and Edges are composed of Vertices. Any of these geometric entities can be assigned materials, boundary conditions, and meshing constraints as appropriate to your model.



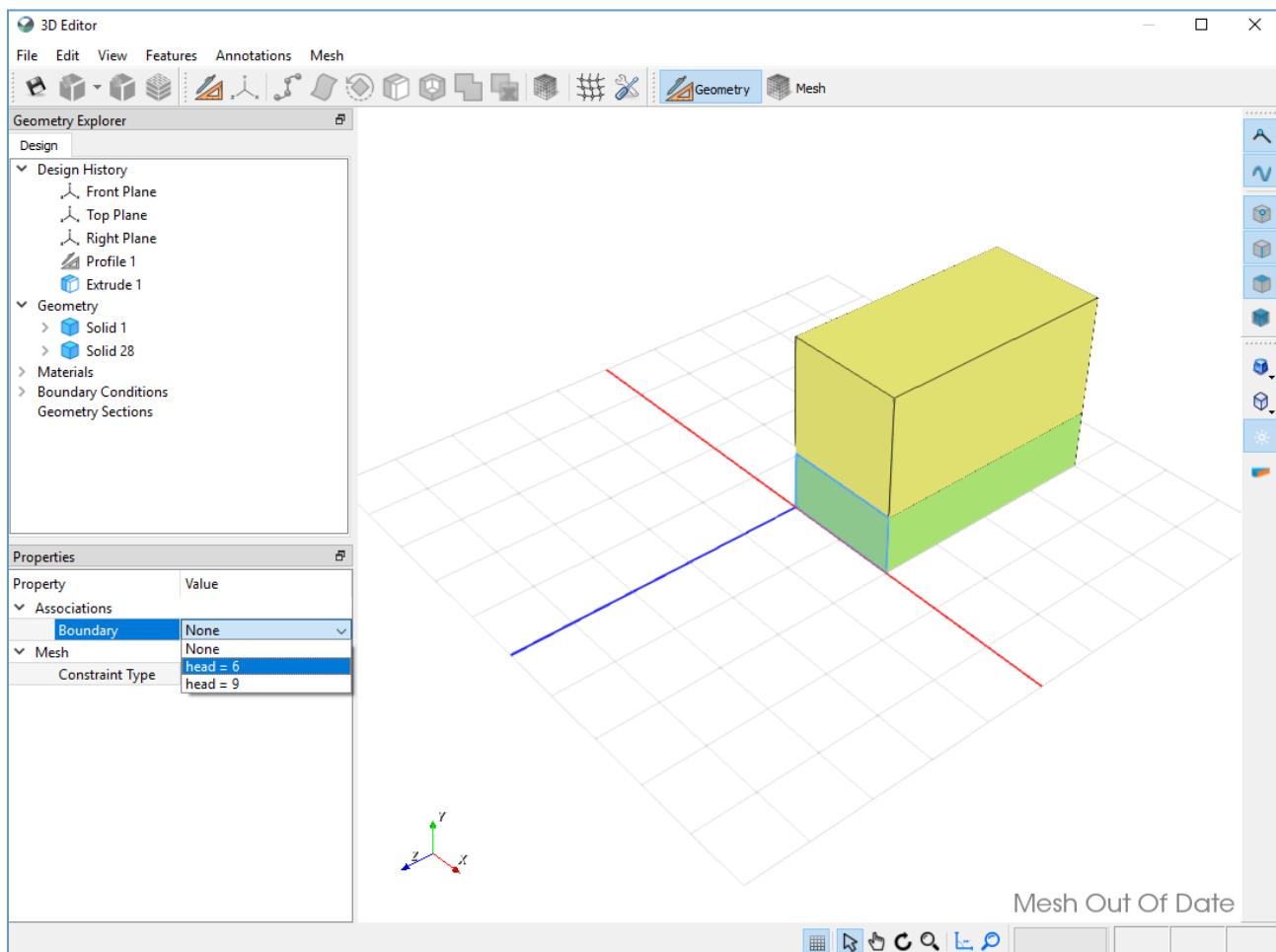
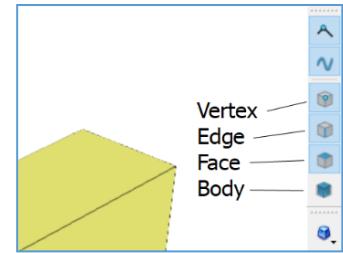
There are several controls that allow for interaction with the 3D view. **Zoom** in or out using the mouse wheel. **Zoom** to the selected object by pressing the blue spyglass in the bottom toolbar. If nothing is selected, this will fit everything to the window. **Pan** by holding down the middle mouse button and dragging your mouse across the 3D view. Alternatively, you may select the hand symbol on the bottom toolbar to pan. **Rotate** by pressing Shift and holding down the middle mouse button, dragging across the screen. Alternatively, select the circular arrow in the bottom toolbar to rotate. This rotation mode uses the centroid of whatever is visible on your screen as the rotation point. To rotate about a more specific centre, hover your mouse over a face for example, until it highlights, then press the Alt and Shift keys along with middle mouse button to rotate. **Multi-select** objects by holding down the Ctrl key while clicking.

The newly created solids do not have a material assigned. In the Properties window you will see 'None' in the Material list for the selected solid. Click on the dropdown arrow next to None in the Material Associations and select Material 1 for the top solid. Similarly, assign Material 2 to the bottom solid. The solids adopt the colour assigned to each material. Expanding the Materials list in the Geometry Explorer provides information on the solids belonging to specific materials.

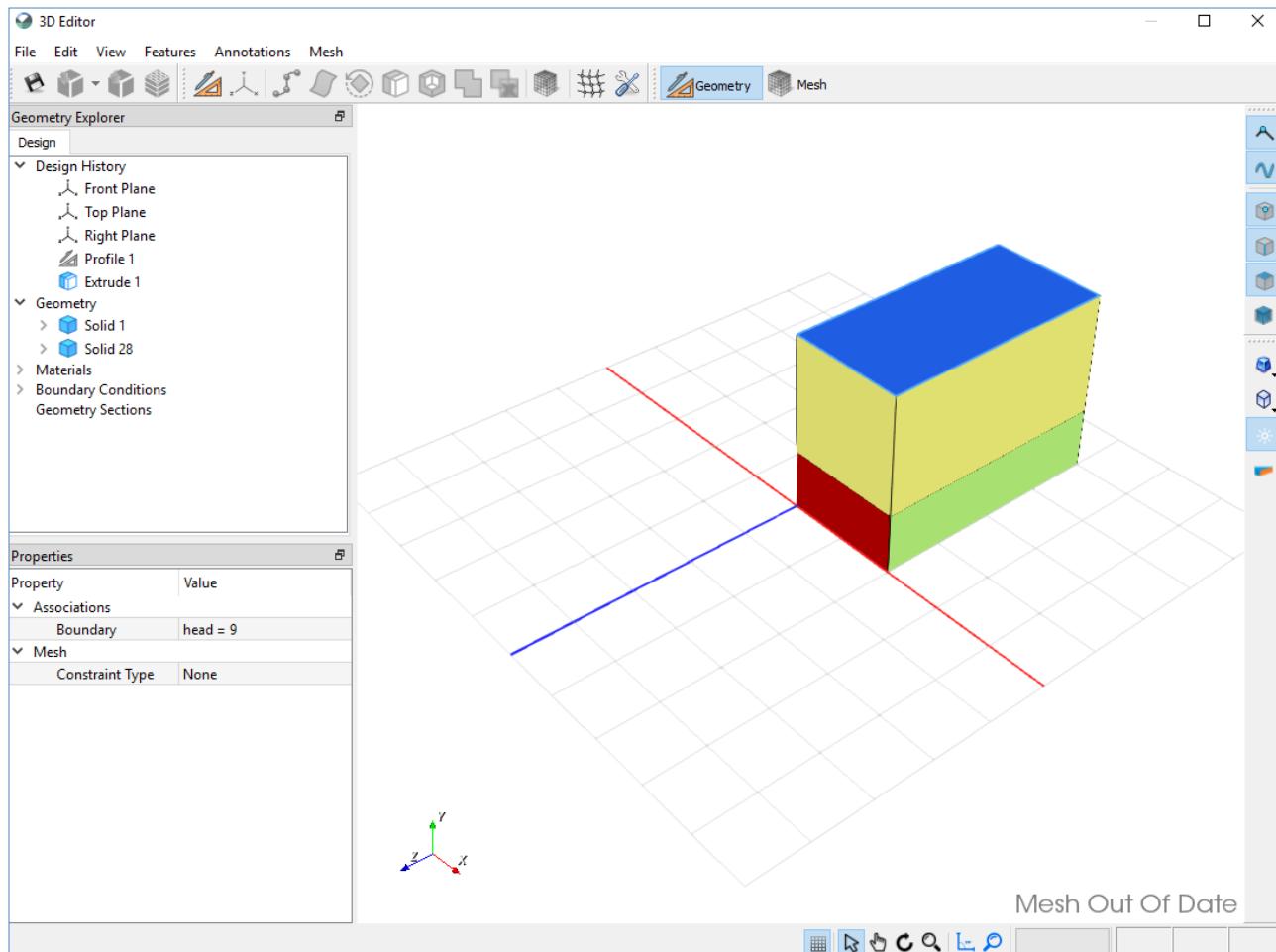


We will assign boundary conditions to two different faces on our model. We can select a face using the tree view or we can select a face directly in the 3D view. Four selection modes are available for Vertex, Edge, Face, and Body selection in the 3D View. Ensure that the Face selection icon is highlighted.

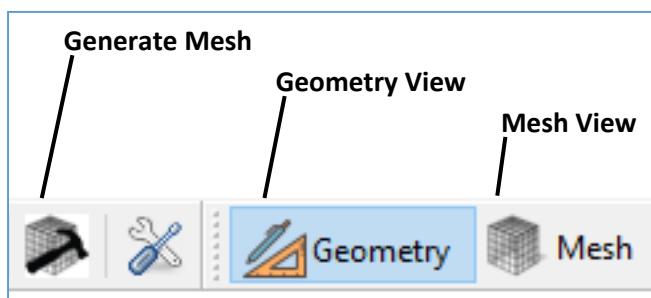
Select the bottom left face and in the Properties window for that face, assign the boundary condition ‘head = 6m’.



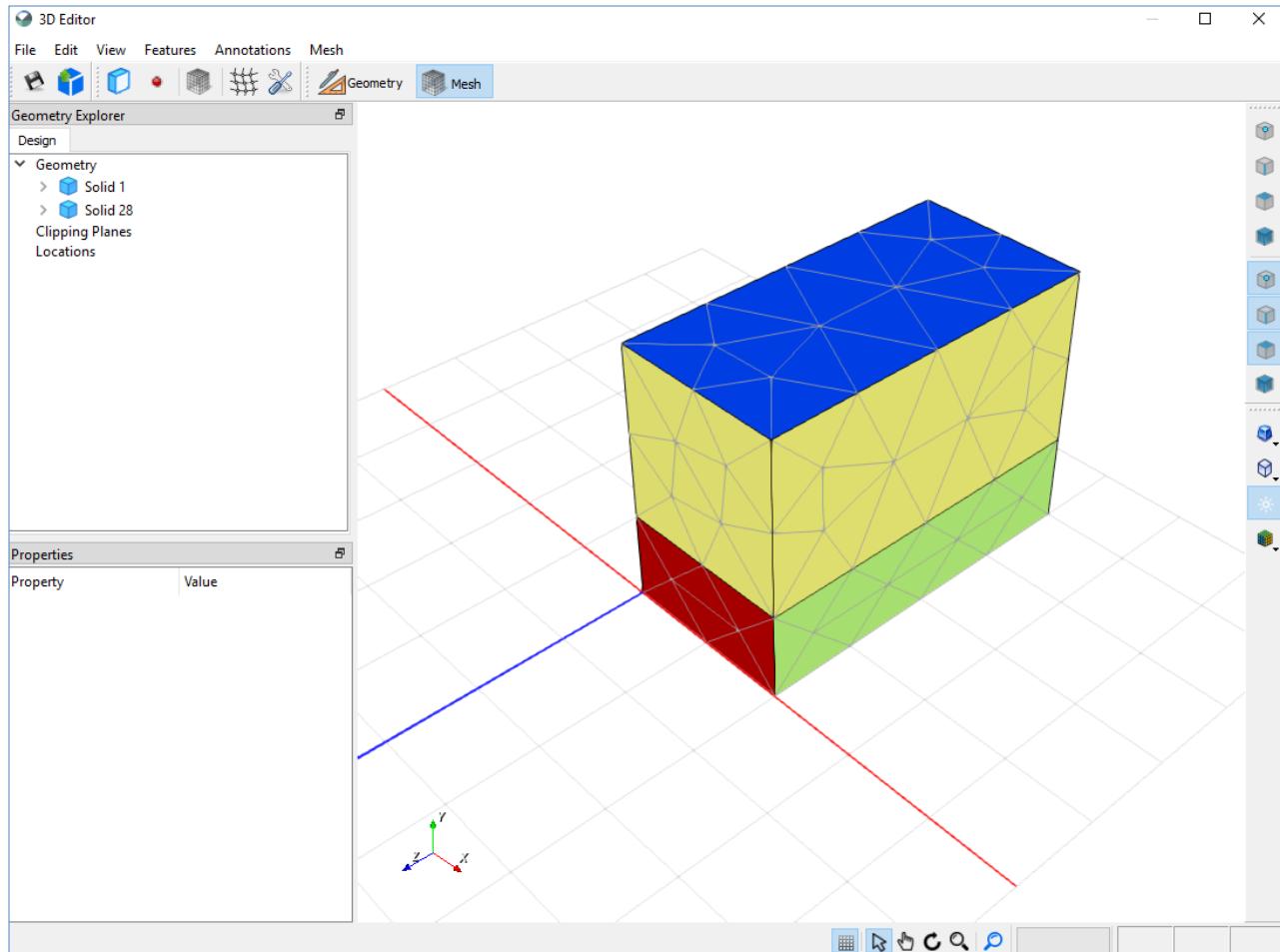
Similarly, select the top face and assign the ‘head=9m’ boundary condition in the Properties window. The model should now be coloured according to your material and boundary condition assignments.



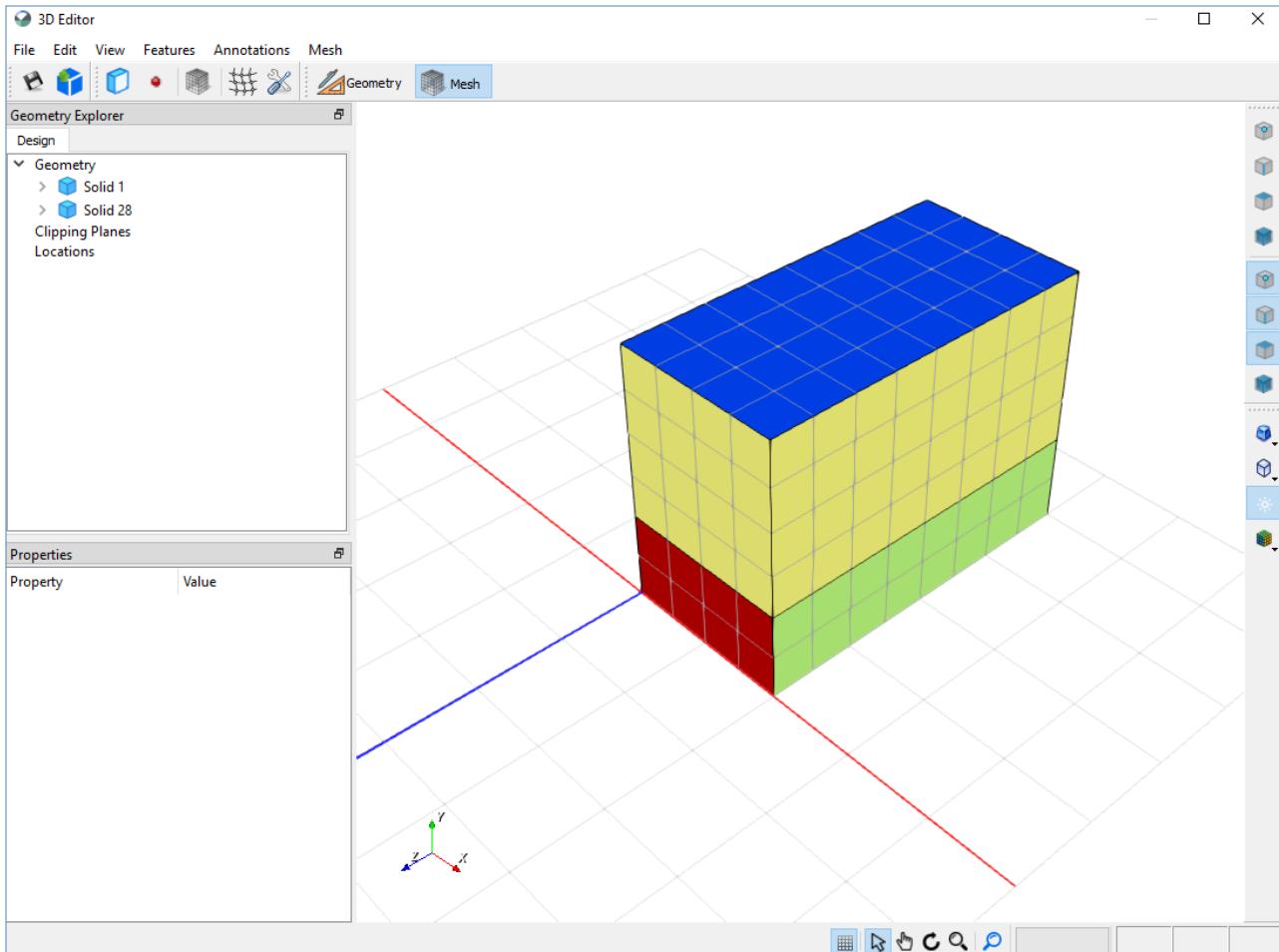
The 3D model development is almost complete. The final step is to generate the finite element mesh. The icons in the top toolbar include multiple options for establishing and viewing the mesh. The Mesh icon is used to generate a mesh and includes options for defining the mesh type and edge length. Beside the Geometry icon is the Mesh View icon which allows you to view the mesh on the domain.



Click the Generate Mesh icon to create a mesh given the default parameters for mesh type and edge size. When we do so, the view automatically switches to Mesh View.



To modify the mesh settings, switch back to Geometry view. Click the Mesh Settings icon, uncheck Use Default, change Edge Length to 0.5, the mesh type to Hexahedral, and click OK. This sets your preference for mesh type, but you must click the Generate Mesh icon again to create the mesh with these settings.



Now that we have generated our geometry, applied materials and boundary conditions, and discretized the domain into a finite element mesh, the model is ready for solving. When we close the BUILD3D window, the 3D model appears in the GeoStudio window. We can solve by pressing Start in the Solve Manager window. In this case, we will not solve the model as the applied boundary conditions are not logical and were only used to demonstrate the model building capabilities.