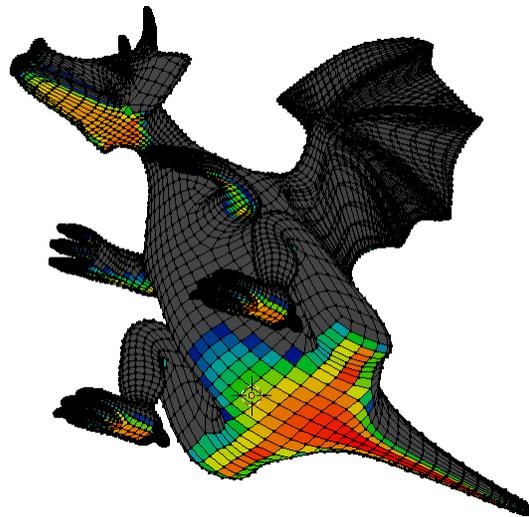


Making supports for your model

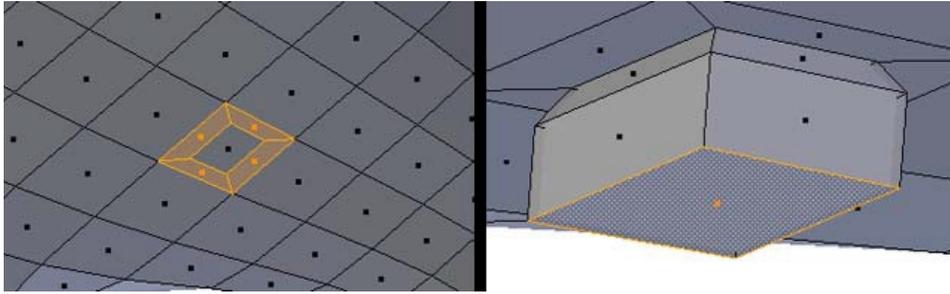
If you have no other choice, then you must make supports for your model. This is not an exact science and you need to know what the printer you are using can do. Your dragon does not need holes in its bottom to shake out extra powder if you are using a material extrusion printer. Nor do material extrusion printers support textures; everything will be the same color:

1. Open the file you saved for a material extrusion printer. Go into **Object Mode**. Select the outer shell.
2. Select the inner shell then select the outer shell. Press *Ctrl + J* to join them.
3. Change to **Solid View**. Go into **Edit Mode**. Check the **Mesh Analysis** checkbox in the **3D View** properties panel. Set the **Type** to **Overhang**. Set **limit the selection** to visible. The dragon will look similar to the following screenshot:



If you were making a support for the dragon, the **Mesh Analysis** would look like this. The colored areas have all been flagged for needing support. The tail, bottom, and feet will all be on the printer bed, so they don't need to be supported. The belly, paws, and head do need support:

1. To build a support, go into face select mode. Select a polygon that has been colored red by the **Mesh Analysis**, say under the dragon's chin. Press *I* to inset the face and then scale it down to about the smallest size you think the printer can print, as seen on the left-hand side of the following screenshot. This is to make the support easy to break off after you have printed it and minimize the sanding you will need to do.
2. Select the polygon in the center of the inset. Extrude it a little.
3. Set the pivot center to **Median Point**. Scale up the face to a size that you think can support the weight.
4. Extrude it again as seen on the right-hand side of the following screenshot. Press *Shift + S* and choose **Cursor to Center**. Set the pivot center to **3D Cursor**. Press *S Z 0 Enter* to make your support flat on the bottom and extend down to the printing bed.



5. Now you have a support that goes all the way to the printing bed, as seen in the following screenshot. Make as many as you need to hold up the unsupported part of the object.
6. Go into **Object Mode**. Select layer 1 as well as layer 12. Select the inner shell as well as the outer shell. Press *Ctrl + J* to join them.