

2D Boolean for Sketchup

What can it do ?

It can trim 2d components and groups with respect to a selected face.

It also has the ability to translate or pushpull faces just after the boolean operation, based on material name. All in 1 go.

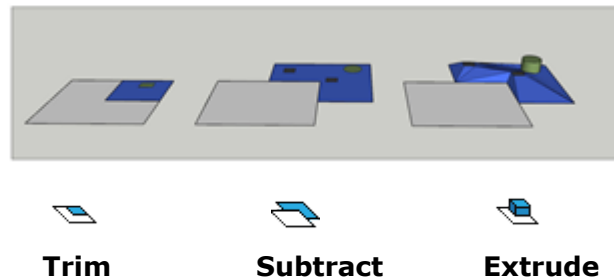
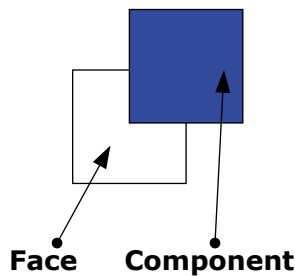
Default material could also be used, but then there is no way of distinguishing any individual face.

The benefit of using a 2d boolean this way is that no cutting geometry has to be created more than the face in context, and it is faster processing with less geometry created. Also non solids could be used.

The result can be exported as geometry which a material cannot.

Crisper results in Layout than using materials.

There are currently 3 modes available in this plugin. **Trim, Subtract and Extrude.**



As can be seen the component adapts to the shape of face. Face is not altered in any way. Neither is the component definition, which means this operation will not affect other instances in the model.

What it can't do (at the time of writing ver 1.3.1)

It can't do a boolean on 3D geometry.

It also has the restriction to only work on 1 face at a time.

Surfaces cannot be used, since they consist of multiple faces.

This then means that only planar operations can be made, without any draping capabilities.

Nested groups or components are not supported, due to lot's of errors digging for materials it is still under investigation.

There is no tiling mechanism. This plugin's functionality may be ported to **Hatchfaces** later on though:

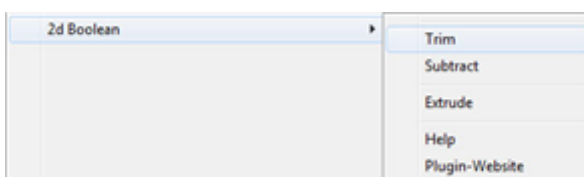
<http://sketchucation.com/forums/viewtopic.php?f=323&t=38637&hilit=hatchfaces>

Menu:

Since ver 1.3.1 the plugin is located in the **Plugins menu**.

Items available are: plugin operations and extras.

Currently extras are: a small help dialog and direct link to -> Sketchucation 2d boolean topic.



USAGE:

Just place a 2d component on a face and hit a button!

Now, if using component with glue behavior this is quite simple, since the plugin will detect which face is the target. However if using groups or components without glue behavior one also has to select the targetface otherwise the plugin won't run.

Multiple components or groups can be used and they will get merged into 1 component with glue behavior.

If using multiple components and tiling manually you might notice copies may not glue to the face, but that's ok. Only 1 component in the selection need to have glue behavior.

So the benefit of using components with glue behavior is great, since selecting a face while covered with components can be difficult in some occasions.

For ex a library of 2d components can be created and reused over and over, for decorating crisp line art for print.

Trim and **Subtract** operations are quite obvious what they do but the **Extrude** operation is a little more involved and can give unpredictable results if not used with care and some planning.

When running the extrude plugin you will get a dialog:

BooleanType: Appart from Trim and Subtract there is also **None**, which skips boolean operation.

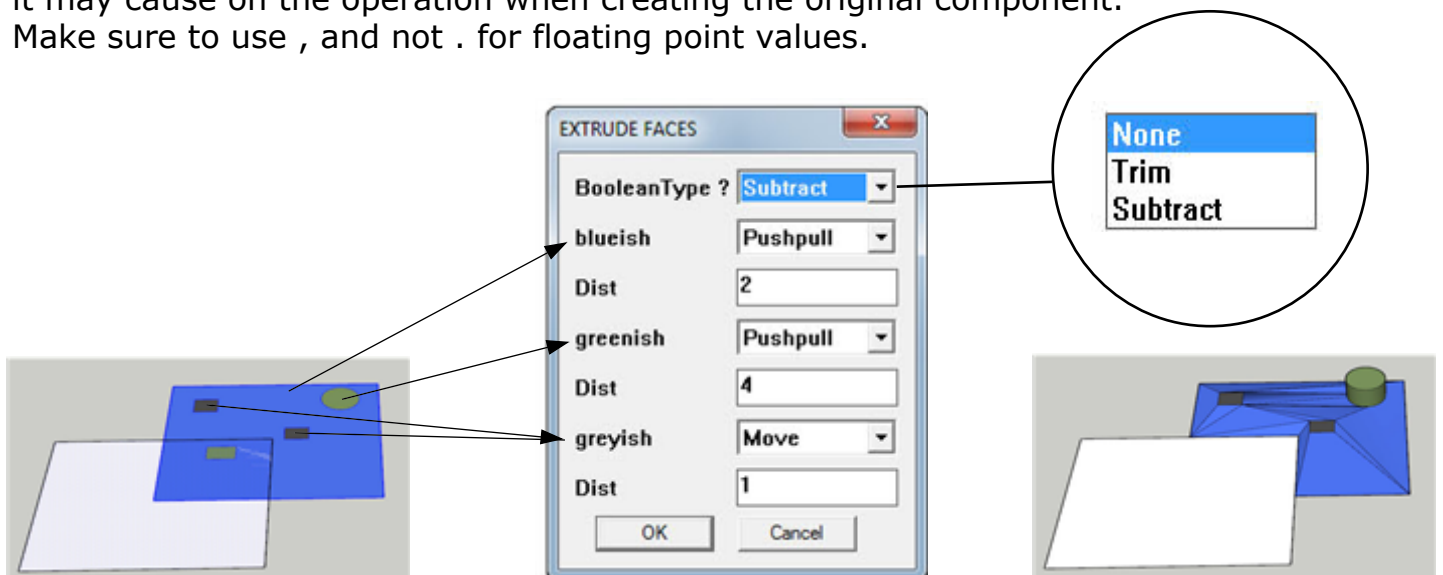
This way you can use any group even 3D geometry (if you select any face in the model as well). It will pick up the components face materials and send them to dialog.

Each material then has **Pushpull** or **Move** method available to be selected in **dropdown menu**. **Move** is a translation in Face normal direction and it will always be performed first since **Pushpull** is a destructive method removing face Id's. Different combinations can be made by running the plugin several times. However Pushpull is the safest way to get predictable results.

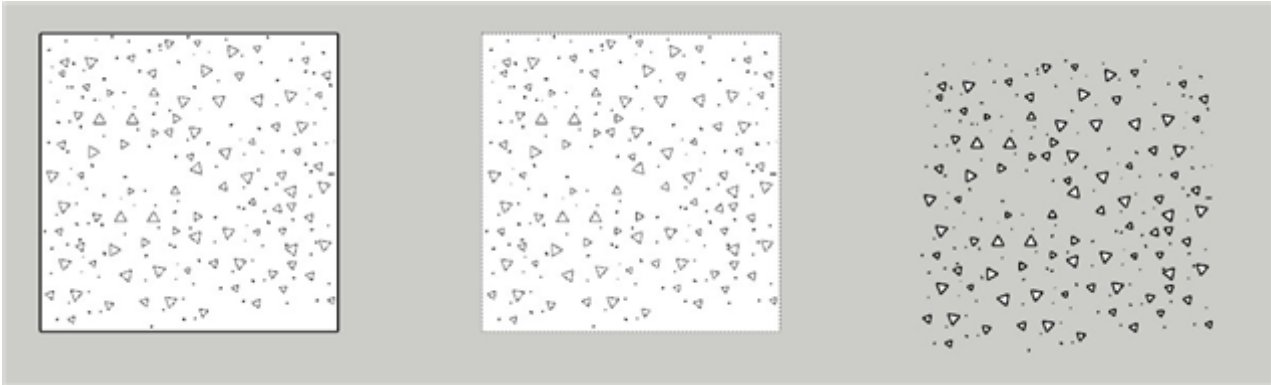
Dist: Is the distance amount (in current unit) you want to extrude.

0 values or blank will be ignored, so that is a way to speed up the process instead of typing in a small value. Negative values can be used but pay attention what affect it may cause on the operation when creating the original component.

Make sure to use , and not . for floating point values.



CREATING LIBRARY COMPONENTS



There are many ways to create 2d components.

These 3 examples will almost bring the same result. If boundary edges is not wished for one can try hiding the edges(example 2).

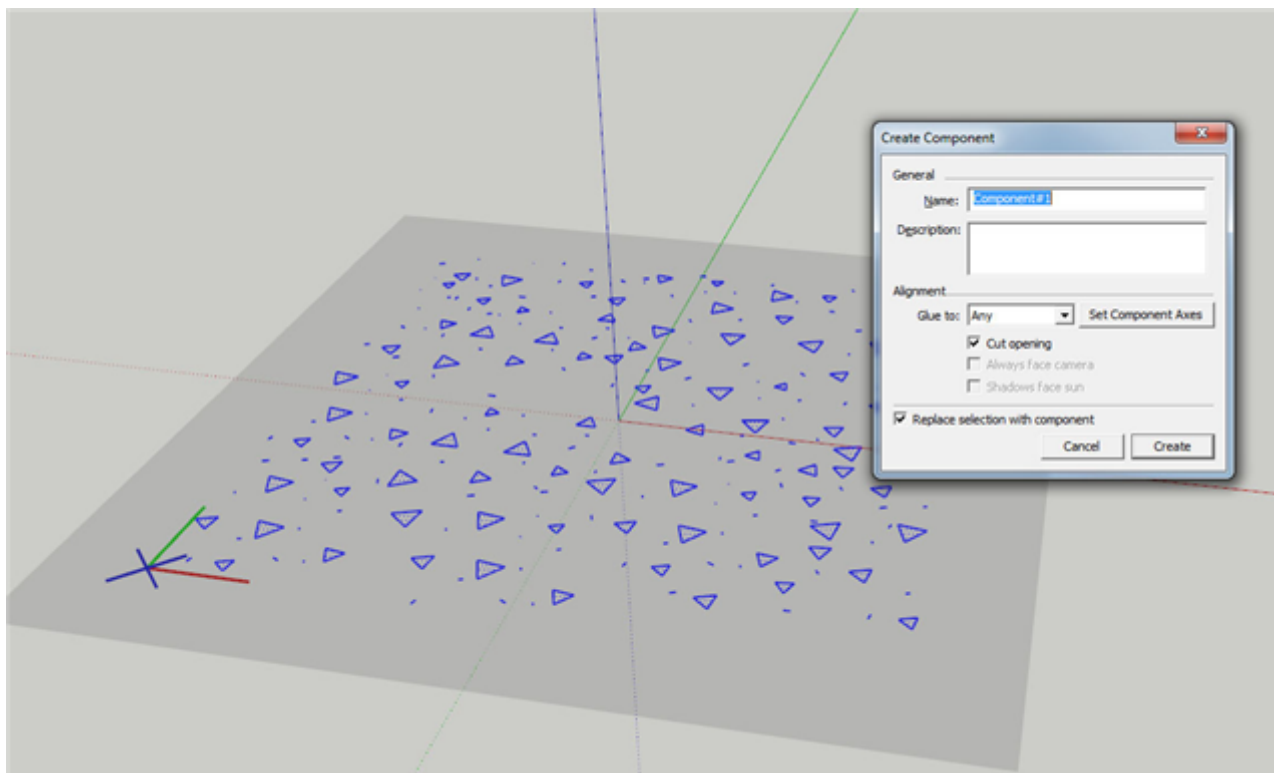
In the third example only the wanted geometry is created. Which may be more predictable during intersections, however it will be more difficult to align, especially while tiling.

This process of creating "right" kind of geometry is still under investigation as well as methods for creating better intersection results.

There are a few golden rules to follow. Try building the glue components at **Origin** with face "up" in **Z_AXIS** (blue axis). **Glue to:** any or some other parameter.

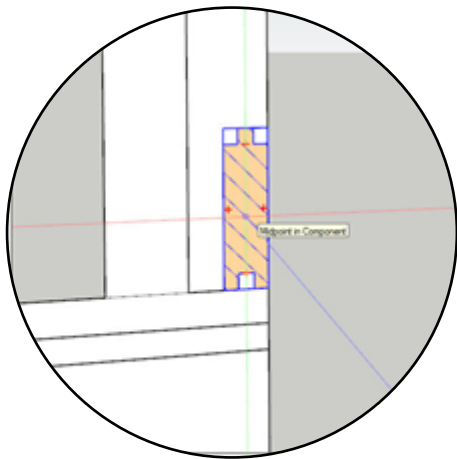
Cut opening is useful for holes.

If creating components for extrusion always try if the faces pushpulls/translates correctly. Sometimes if tiling, copying and pasting in faces their edgeloops can be rotating in undesired direction which will give inverse pushpull results.

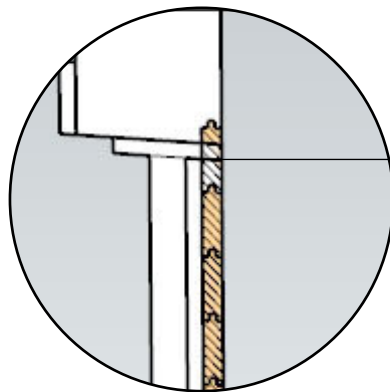
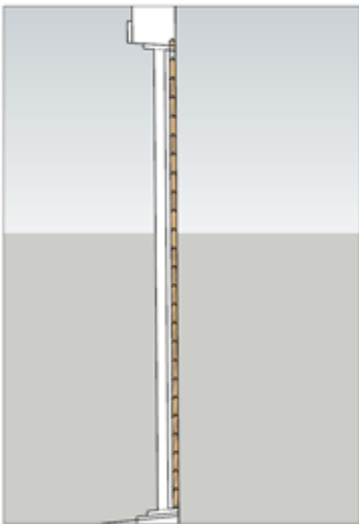


TUTORIAL

We are going to decorate some details of this garage.

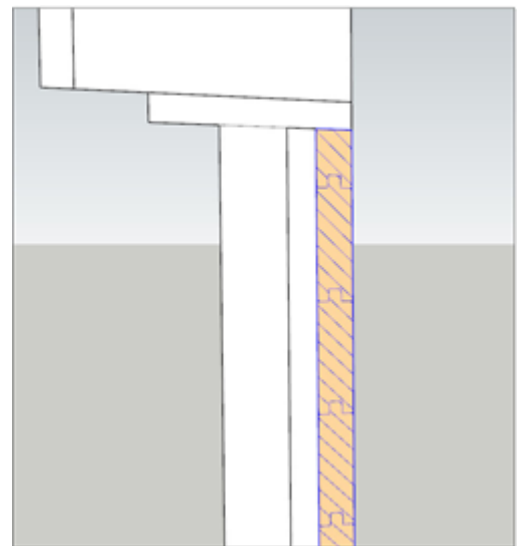
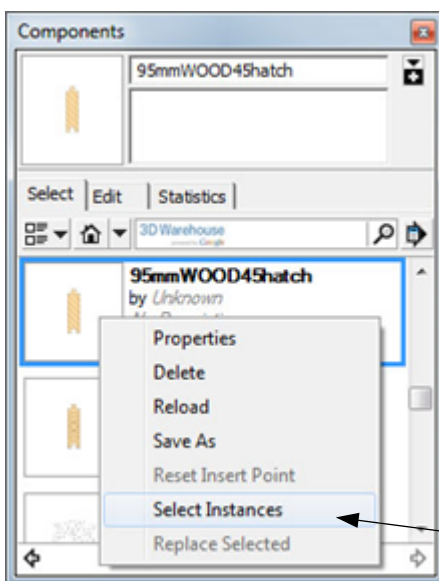


1) Start by adding in a component and place it at start with the desire to tile it.



Face limit

2) Tile manually until last copy exceeds the limit of the targetface.

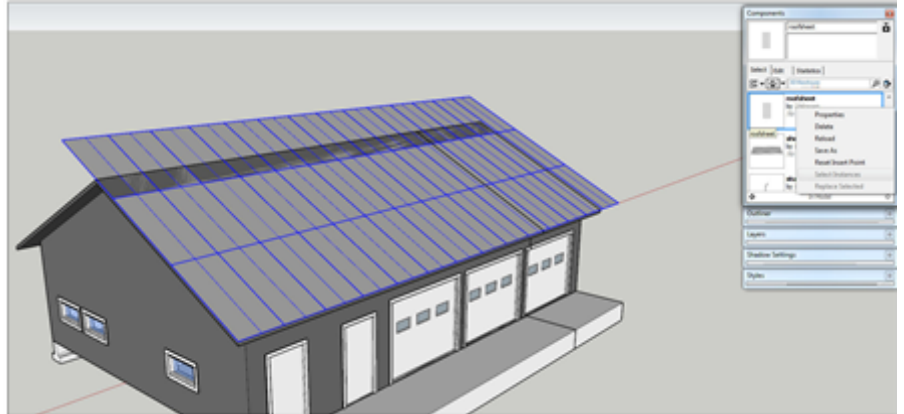
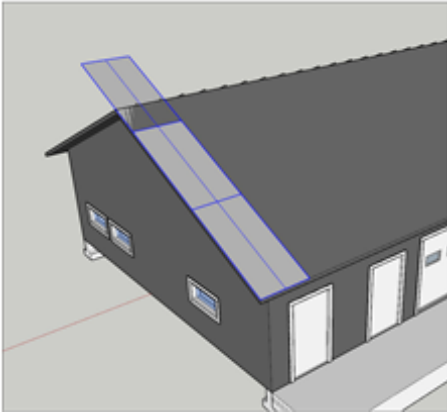


3) Now, if no instances of this component are in the model you can use **Select instances** to quickly grab all of these components. *It is yet another advantage of using components.*

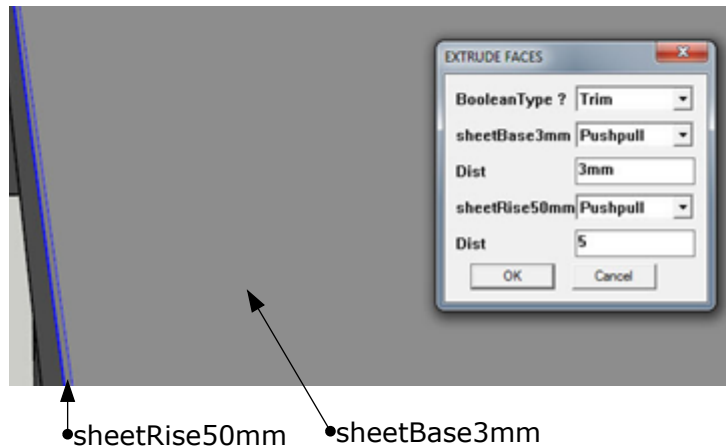
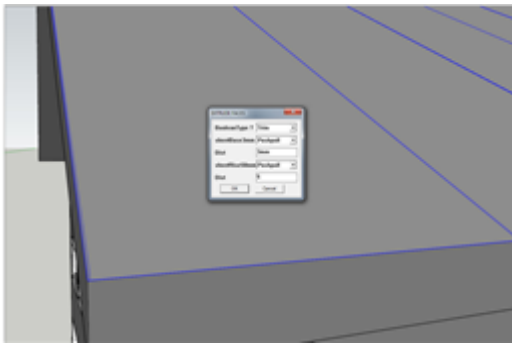
4) Push the 2d boolean **Trim-2D Geometry to face** and the result will be 1 component snug to the face with no Z-fighting.

TUTORIAL

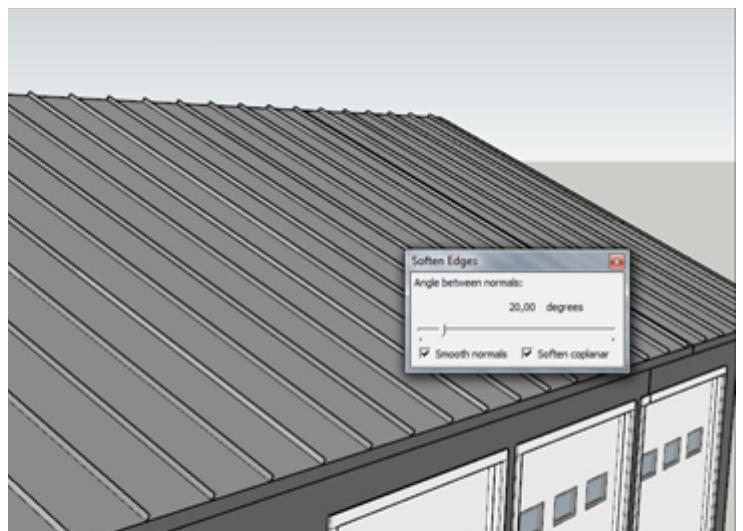
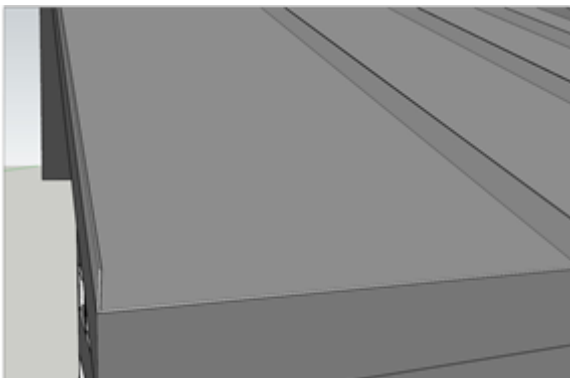
We are going to create some metal roofing



- 1) This simple roof consist of 1 face, so we can just tile the components until the face is covered. Just like the last tutorial **select instances** to grab all components.



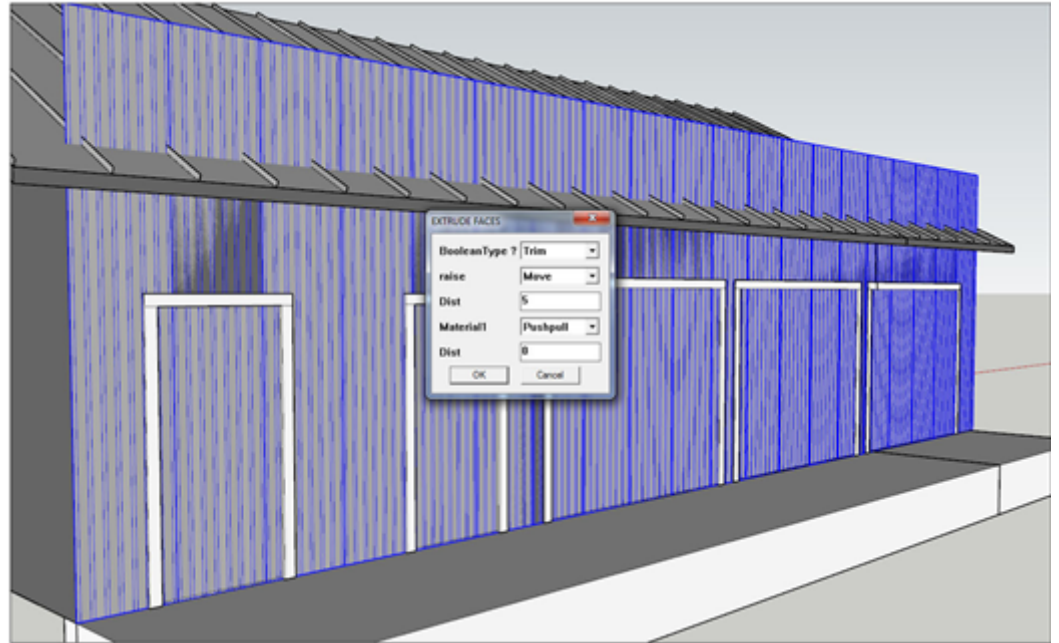
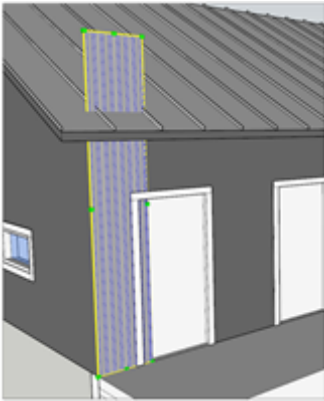
- 2) Hit the **2D-Boolean-extrusion** button and a dialog will pop up.
- 3) If we zoom in we can see that there is actually a border face. There are 2 separate materials. They have same color properties, but the name are different and that is what counts. So in this case to create the standing seams we pushpull sheetRise material 50 mm. Also we want to **Trim** components to face.



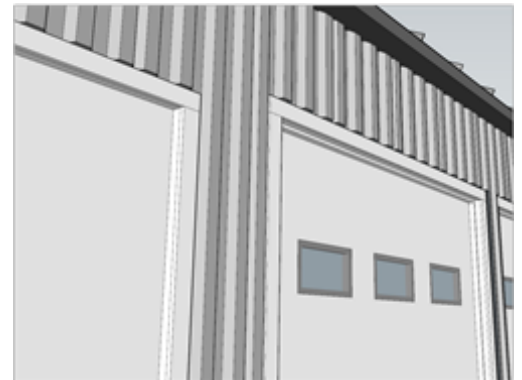
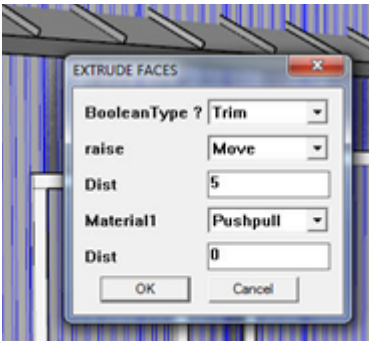
- 4) And there you go! Simple sheet roofing created. If desired one can adjust edge smoothing. This is just a simple example. With some planning one can add screws or other details as well.

TUTORIAL

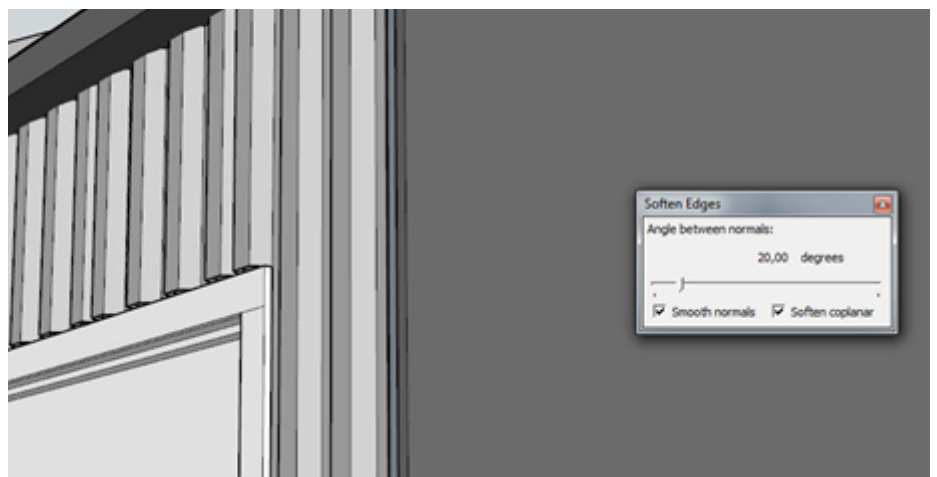
We are going to create some metal sheet walls



- 1) Like in the last tutorials we apply the components and tile them accordingly, and **select instances**.
- 2) Hit the **2D-Boolean-extrusion** button and a dialog will pop up. Just like before this component has 2 different materials. And we are going to create corrugated sheets by translating some of the faces.



- 3) After running the plugin one can see that a translation has taken place. But the thickness of the sheet is still the same.



- 4) However there are some errors due to translations around holes. Which in most cases can be fixed with some smoothing, or manual removal.