

# Glasshopper User Manual

**Table of Contents**

1 INTRODUCTION ..... 3

    1.1 SETUP ..... 3

    1.2 GEOMETRY ..... 3

    1.3 BUILT-UP ..... 4

        1.3.1 MATERIALS ..... 4

        1.3.2 STACKING LAYERS ..... 4

    1.4 LOADS AND MESH ..... 5

    1.5 RESTRAINTS ..... 6

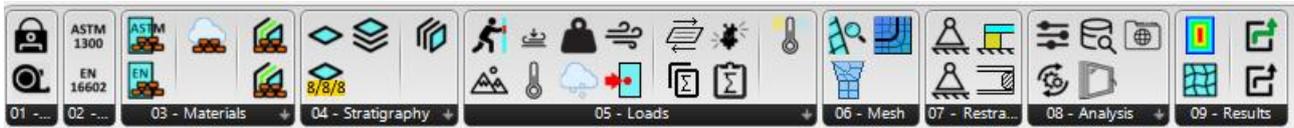
    1.6 LOAD COMBINATIONS ..... 6

    1.7 USE THE REFERENCE TEMPERATURES PANEL TO APPLY SEASONAL TEMPERATURE VARIATIONS AND  
        CONNECT ALL OUTPUTS TO THE ADD SHEAR COMPONENT TO AUTOMATICALLY SET THE SHEAR  
        MODULUS OF THE INTERLAYERS. .... 8

    1.8 CALCULATION ..... 8

    1.9 RESULTS ..... 9

# 1 Introduction



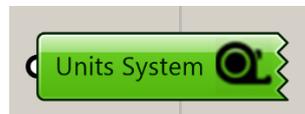
## 1.1 Setup

- 1.1.1.1 To get started, first log in to the service using the AuthComponent. Right-click on the "Provider" text and choose your login method: Google, Maffeis domain, or Maffeis username and password.



### 1.1.1.2

Next, use the **UnitsComponent** to select your preferred unit system. Right-click on the "Units System" text and select either metric or imperial units.



## 1.2 Geometry

To link geometry, you can either pass a surface to a **Brep** component or select a surface in Rhino, then right-click on the **Brep** component and choose "Set one Brep."



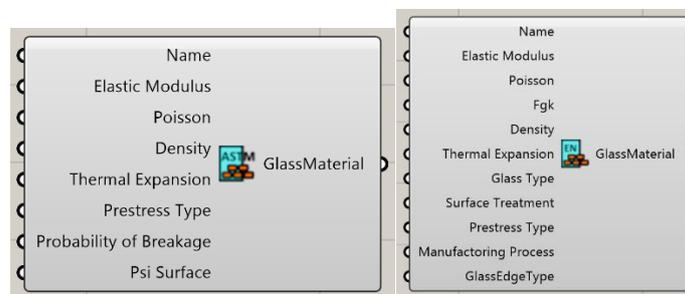
## 1.3 Built-up

### 1.3.1 Materials

1.3.1.1 **The materials available for your built-up include glass, interlayer, and air materials.**

#### 1.3.1.2 Glass materials

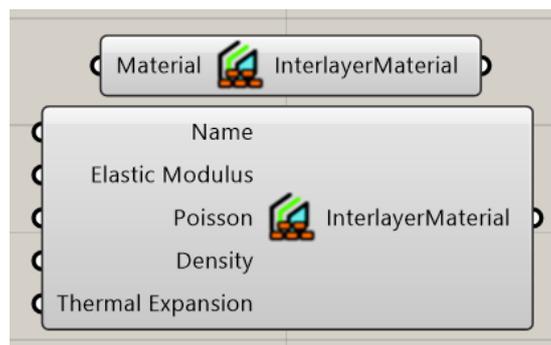
There are two types of glass materials available: those that follow the **ASTM** standard and those that follow the **EN16612** standard.



#### 1.3.1.3 Interlayer materials

You can select interlayer materials in two ways:

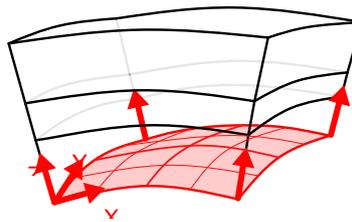
- **Database:** Right-click on the "Material" option in the first component to choose from our database of interlayer materials.
- **Custom:** Use the second component to input custom specifications.



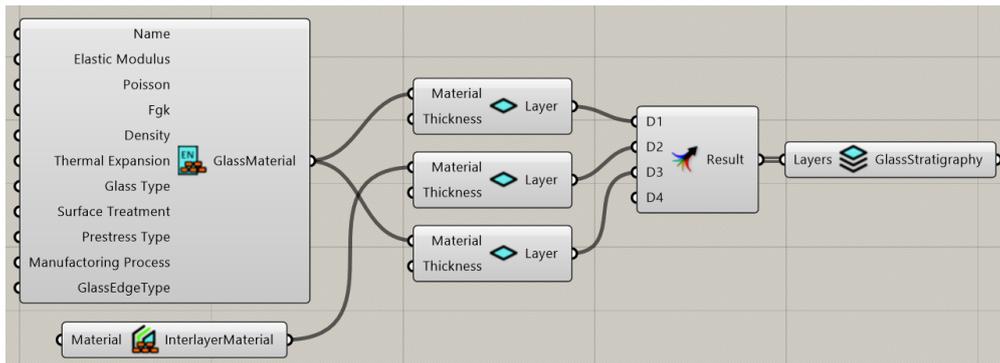
### 1.3.2 Stacking layers

To define the thickness of a material layer, link the material to the **LayerComponent** and assign the desired thickness.

Once all individual layers are configured, merge all **LayerComponents** into the **StratigraphyComponent** to create the complete IGU (Insulating Glass Unit) built-up.



The 3D structure is created by extruding **from inside** to outside along the mean of the normals in each point.

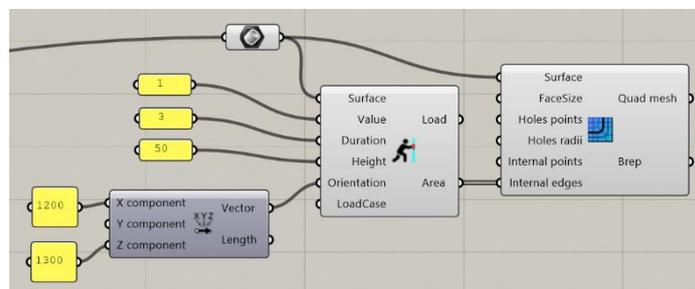


## 1.4 Loads and Mesh



### 1.4.1.1 Example: barrier load

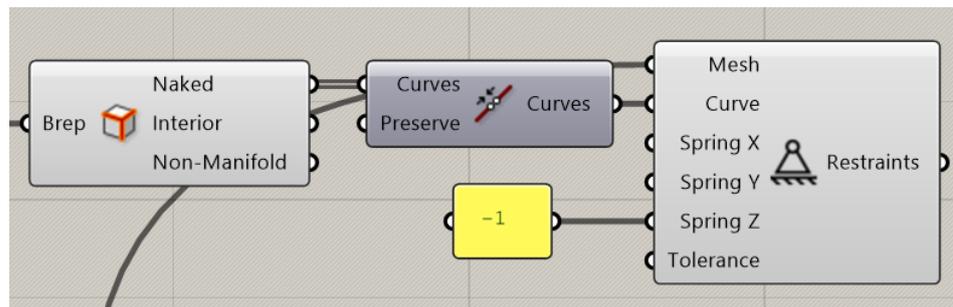
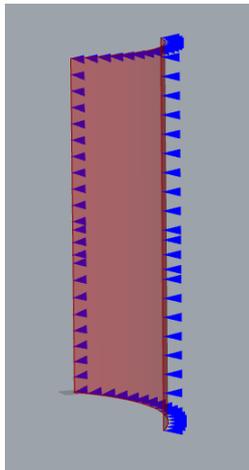
The **Barrier Load** component works by intersecting the glass surface with a horizontal plane at a specified height. The resulting area is then used as input for the Rhino **Quad Mesh** component as its **Internal Edges**.



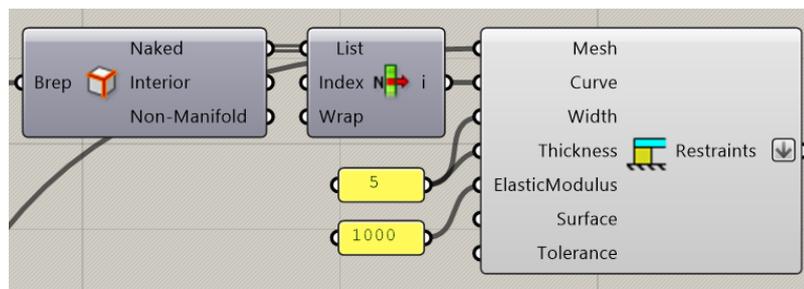
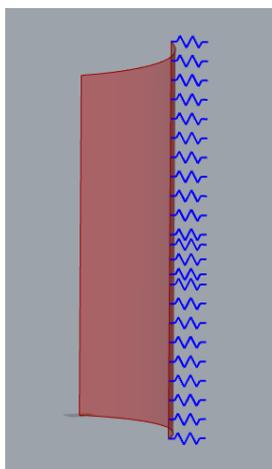
## 1.5 Restraints

### 1.5.1.1 Example: Z support on all edges

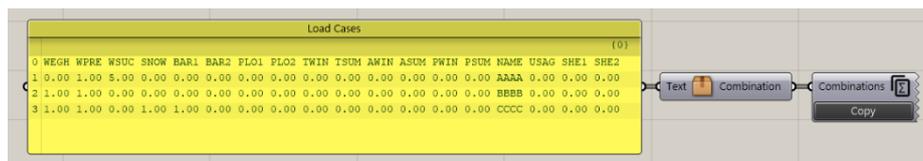
The following blocks are for fixing the z direction in all sides.



### 1.5.1.2 Example: silicone support on one edge



## 1.6 Load Combinations



Use the **yellow panel** in the template above to define the **coefficients** for your **load cases**. Each row represents a single load case, and the columns are labeled with the load types, identified by these **codenames**:

- WEGH: dead load (weight).
- WPRE: wind pressure.
- WSUC: wind suction.
- SNOW: snow load.
- BAR1: first barrier load.
- BAR2: second barrier load.
- PLO1: first point load.
- PLO2: second point load.
- TWIN/TSUM: temperature winter and summer.
- AWIN/ASUM: altitude winter and summer.
- PWIN/PSUM: pressure winter and summer.
- NAME: name of the loadcase.
- USAG: type of check: 0 if none, 1 for strength, 2 for deflections.
- SHE1/SHE2: shear modulus of the first/second interlayer.

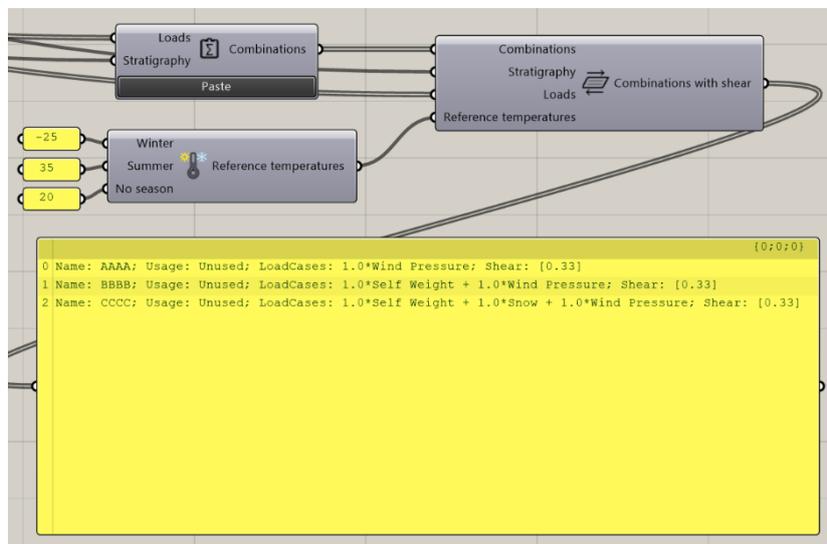
However, you can also click the **Copy** button and paste the data into an Excel file. The table will appear as follows:

A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P	
1	Name	Usage	Self Weight	Wind Pressure	Wind Suction	Snow	Barrier 1	Barrier 2	Point load 1	Point load 2	Temperature Winter	Temperature Summer	Altitude Winter	Altitude Summer	Pressure Winter	Pressure Summer
2	Comb_1	Strength	2	0	0	0	0	0	0	0	0	0	0	0	0	0
3	Comb_2	Strength	1	1	0	0	0	0	0	0	0	0	0	0	0	0
4	Comb_3	Strength	1	0	1	0	0	0	0	0	0	0	0	0	0	0
5	Comb_4	Strength	1	0	0	1	0	0	0	0	0	0	0	0	0	0
6	Comb_5	Strength	1	0	0	0	1	0	0	0	0	0	0	0	0	0
7	Comb_6	Strength	1	0	0	0	0	1	0	0	0	0	0	0	0	0
8	Comb_7	Strength	1	0	0	0	0	0	1	0	0	0	0	0	0	0
9	Comb_8	Strength	1	0	0	0	0	0	0	1	0	0	0	0	0	0
10	Comb_9	Strength	1	0	0	0	0	0	0	0	1	0	0	0	0	0
11	Comb_10	Strength	1	0	0	0	0	0	0	0	0	1	0	0	0	0
12	Comb_11	Strength	1	0	0	0	0	0	0	0	0	0	1	0	0	0
13	Comb_12	Deflection	1	0	0	0	0	0	0	0	0	0	0	1	0	0
14	Comb_13	Deflection	1	0	0	0	0	0	0	0	0	0	0	0	1	0
15	Comb_14	Unused	1	0	0	0	0	0	0	0	0	0	0	0	0	1

You may edit the parameters in each column, but keep in mind:

- **Do not** remove any columns or change the column names, as this will cause errors.
- You are free to add or delete rows as needed.

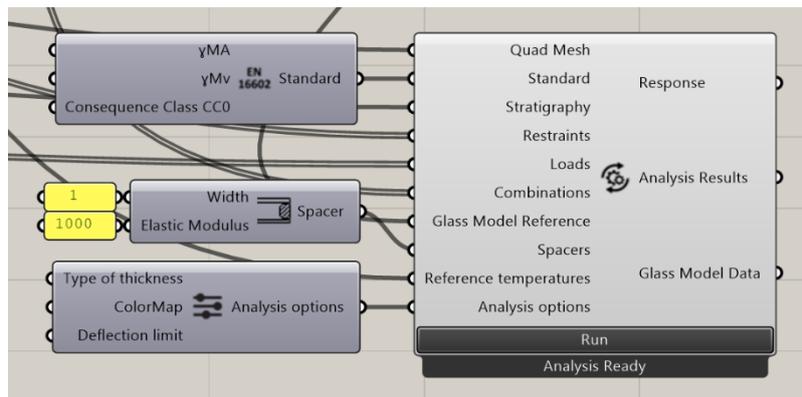
Once the table has been modified, select it in Excel and use the **Paste** button in the template to import the load combinations back into Grasshopper.



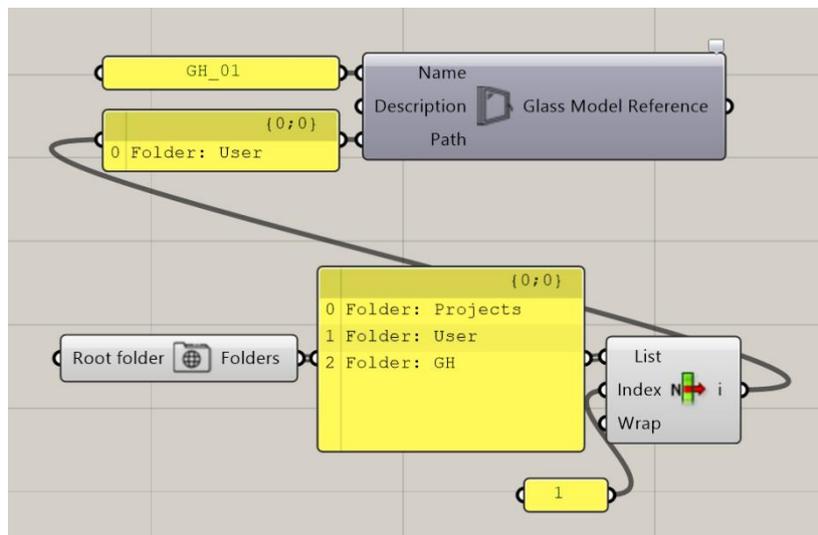
Connect all outputs to the Add Shear component and use the Reference Temperatures panel to apply seasonal temperature variations; the software will automatically determine the interlayer's shear modulus based on the temperature and the shortest load duration.

## 1.7 Calculation

The Glass Analysis component processes all the previously defined quantities, a selected standard, and the analysis options. The calculations are sent to our cloud servers for processing, which then returns the analysis results.



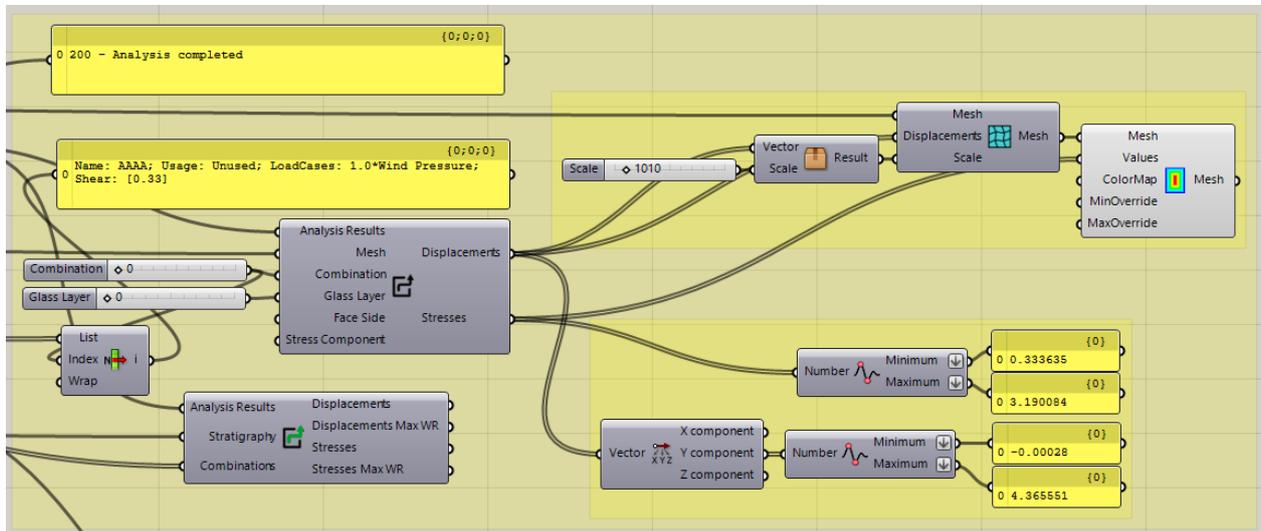
This example demonstrates the application of the **EN16602 standard** with a **spacer** inserted into the cavities. Use the available **Analysis options** to control the visualization and analysis parameters: select between **Nominal or Reduced thickness**, define the **color map**, and establish the **deflection limit**.



First, use the component circuit to **select the folder and file name** where the analysis results will be saved (this is accessible via the web app). Next, **connect the output of the Glass Model Reference** to the input of the **Run Analysis** component.

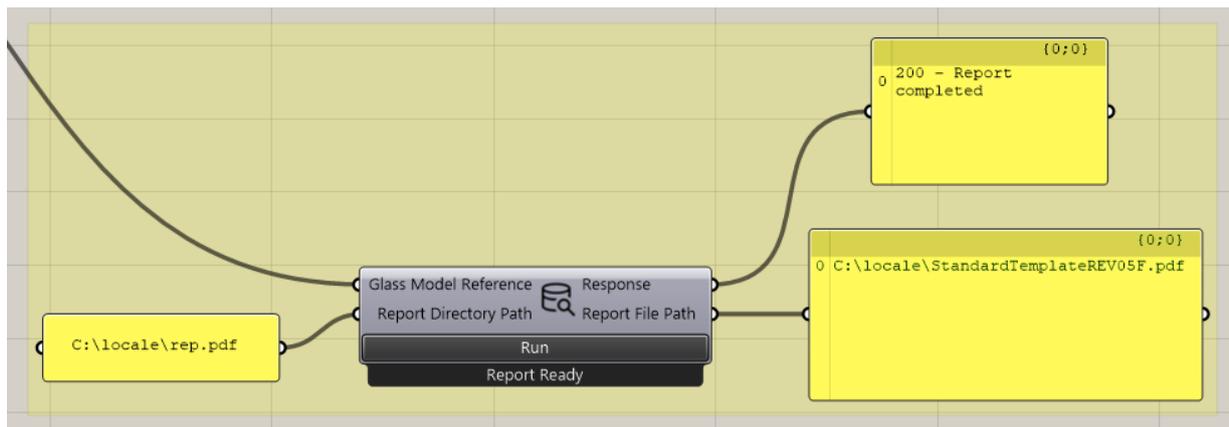
## 1.8 Results

Use the following circuit to see the output of the analysis.



First, choose the **combination number** and **glass layer**. Then, configure the viewing direction by right-clicking the face side and selecting **Inside** or **Outside**. Finally, right-click on the **Stress Component** menu to select the desired stress for visualization:

- **$\sigma_{id}$** : Von Mises stress
- **$\sigma_x/\sigma_y$** : Global x and y stresses
- **$\sigma_1/\sigma_2/\sigma_3$** : Principal stresses
- **$\tau_{xy}$** : Global xy shear stress



Use the component shown above to **create the PDF report** of the analysis. The generated report will be available at the **report file path** on your computer.