

Web App User Manual

Table of Contents

1 INTRODUCTION	3
2 GEOMETRY	4
2.1 VIEW	4
2.2 DEFINE THE SHAPE OF YOUR GEOMETRY BY ENTERING THE POLYGON POINTS ONE BY ONE IN A COUNTERCLOCKWISE DIRECTION. TO CHANGE THE NUMBER OF POINTS, CLICK THE "CHANGE SHAPE" BUTTON. YOU CAN ALSO DEFINE THE RADIUS OF CURVATURE FOR A CYLINDRICAL SHAPE. THE GLOBAL AXES ARE DISPLAYED FOR REFERENCE.	4
2.3 BUILT-UP	4
2.3.1 ORIENTATION	5
2.3.2 ADDING LAYERS	5
2.3.3 ADJUSTING THICKNESS	5
2.3.4 ADDING NEW LAYERS	5
2.3.5 INTERLAYER SPECIFICATIONS	6
3 SUPPORTS	7
3.1.1 STRUCTURAL SILICONE PARAMETERS	8
3.1.2 SPACER SPECIFICATIONS	8
4 LOADS	9
4.1 LOAD TYPES	9
5 OPTIONS	11
6 ANALYSIS	12
6.1.1 VIEWING RESULTS	15
6.1.2 UNDERSTANDING DISPLACEMENT AND STRESS RESULTS	16
7 CHECK	17
8 REPORT	18

1 Introduction

This is a user manual for the **GLASSSS** web application, a tool created by **Maffeis Engineering SpA** for the analysis and design of glass structures. This document provides a comprehensive guide to using the app, from defining the **geometry** and **built-up** of your glass panel to applying **loads**, running an **analysis**, and generating a **report**.

What is GLASSSS?

GLASSSS is an advanced web application designed to simplify the complex process of designing and analyzing glass structures. By using this tool, engineers and designers can accurately define the properties of a glass panel, apply various load conditions, and perform a detailed **Finite Element Method (FEM)** analysis. This ensures that the glass structure meets safety standards and performs as expected under different environmental and mechanical stresses.

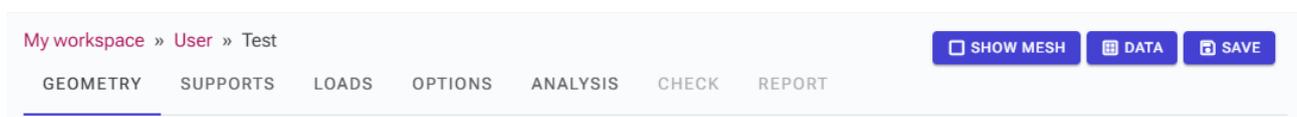
Alongside the FEM analysis, a novel Generative AI is presented for creating a fast preview.

Main tabs

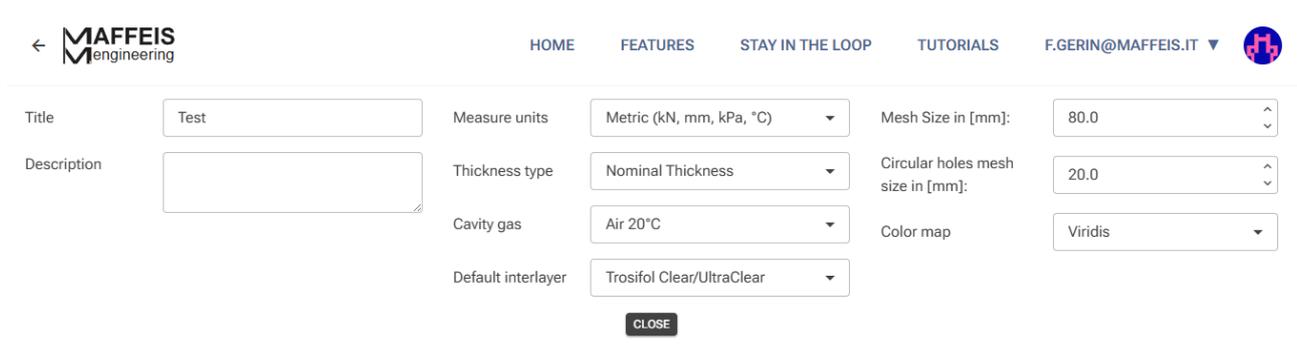
The GLASSSS interface is structured around a workflow composed of nine main sections:

- **Geometry:** How to define the shape and dimensions of your glass panel.
- **Supports:** How to add and configure edge and point supports.
- **Loads:** How to define and apply various load types.
- **Options:** How to set analysis parameters and safety standards.
- **Analysis:** How to run the analysis and view results.
- **Check:** How to interpret the working ratios and stress checks.
- **Report:** How to generate a final report of your project.

By following the sections in order, you can complete a full analysis of your glass structure from start to finish.



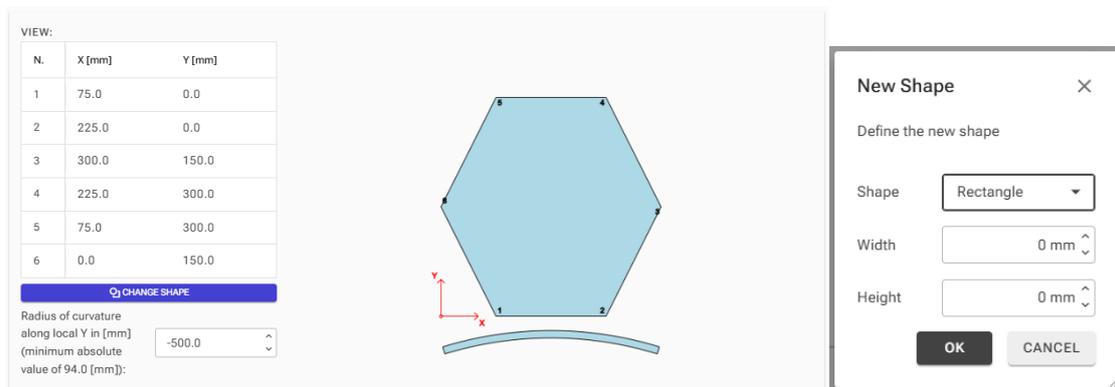
To enable mesh visualization, click **Show Mesh**. Clicking the **Data** panel will open a dropdown menu with various options. Remember to click **Save** frequently, as auto-save is not available.



2 Geometry

This section allows you to define the geometric properties of your glass panel.

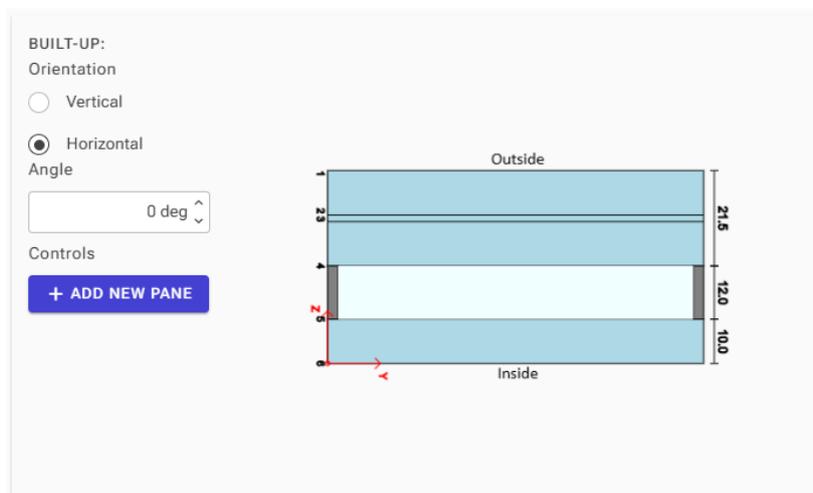
2.1 View



2.2 Define the shape of your geometry by entering the polygon points one by one in a counterclockwise direction. To change the number of points, click the "Change Shape" button. You can also define the radius of curvature for a cylindrical shape. The global axes are displayed for reference.

2.3 Built-up

This section shows the layers that make up your glass package. The surfaces are numbered **from outside to inside**, and the **thickness of each package** is displayed on the right side of the image.

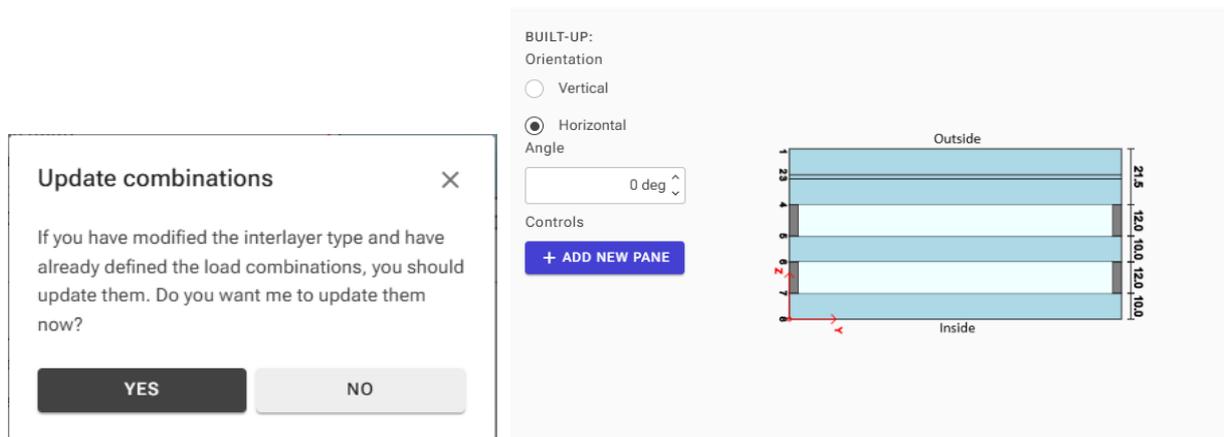


2.3.1 Orientation

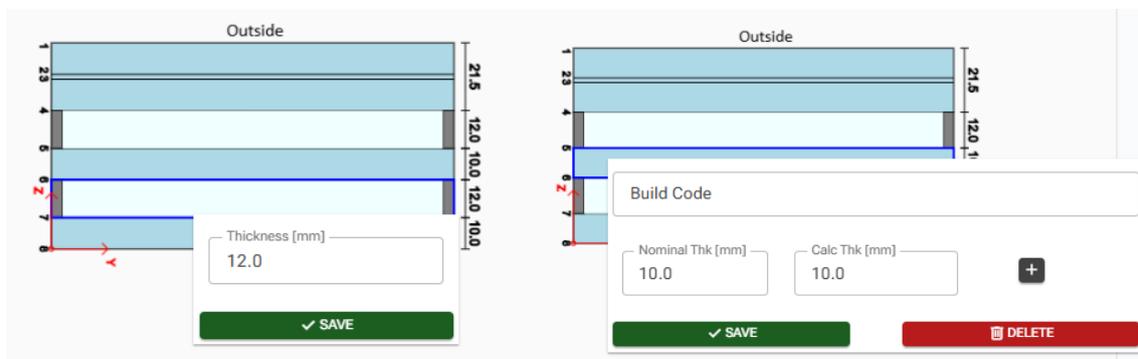
Select a **vertical** or **horizontal** orientation or set a **custom rotation angle** in degrees. The rotation is performed around the Y-axis, following the right-hand rule, with respect to the orientation chosen.

2.3.2 Adding Layers

To add a new package and create a gas camera, click on **"Add New Pane"**. A new pane will always be placed on the inside of the existing glass package.



If your glass package includes interlayers, it's crucial to **update the parameters of your load combinations to reflect the secant modulus of these interlayers**. This ensures accurate calculations for their behavior under load.



2.3.3 Adjusting Thickness

- To adjust the camera's thickness, click on the **camera icon** and update its thickness in millimeters.
- To modify a glass pane's thickness, click on the **glass pane** itself to update both the nominal and calculated thickness.

2.3.4 Adding New Layers

You can add a new layer in two ways:

- Click the **"+" button**.

- Enter the **build-up code**.

The screenshot shows a software interface for configuring a glass build-up. At the top is a text input field labeled 'Build Code'. Below it are two glass panes, 'Pane 1' and 'Pane 2'. Each pane has two input fields: 'Nominal Thk (mm)' and 'Calc Thk (mm)', both containing the value '10.0'. To the right of each pane is a red square button with a white trash icon. Between the panes is an 'Interlayer 1' section with a 'Qty' input field containing '1' and a 'Material' dropdown menu showing 'Trosifol Clear/UltraClear'. At the bottom are two buttons: a green 'SAVE' button and a red 'DELETE' button.

- Set the nominal and calculated thickness for each glass layer. Then, select the number of interlayers so that their total thickness equals 0.36 times the chosen quantity. Choose the interlayer's material type from the dropdown menu.

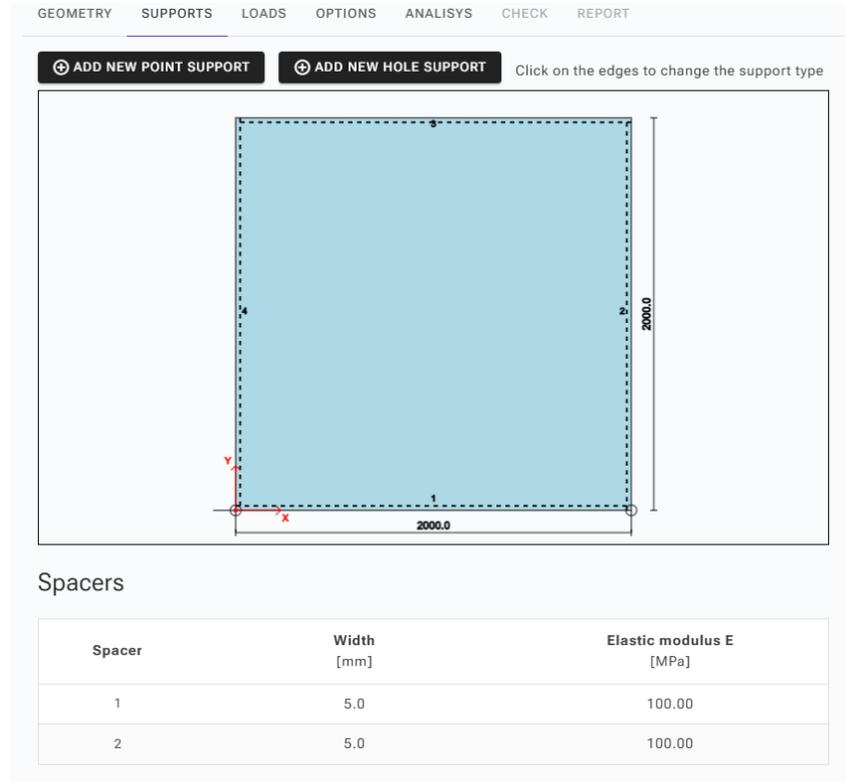
2.3.5 Interlayer Specifications

You can adjust the interlayer specifications by:

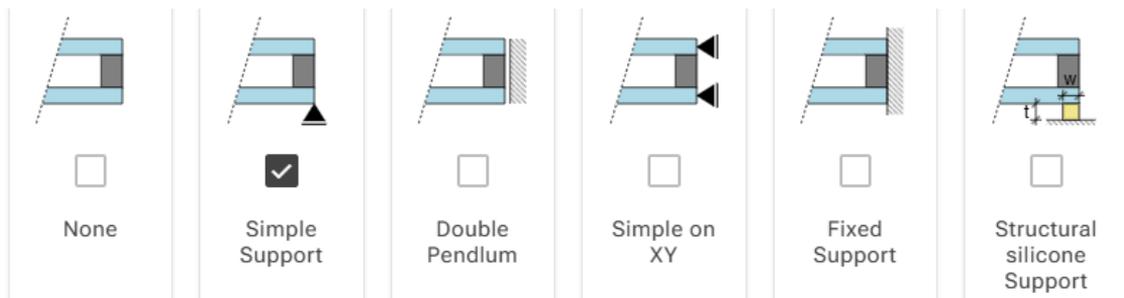
1. **Setting the Quantity (Qty) of Minimum Base Thickness:** The total interlayer thickness will be calculated as $0.38 \times \text{Qty}$.
2. **Choosing the Material:** Select from a list of commonly used materials.

Note: If your desired material is not available in the list, you can manually input its **secant modulus** directly into the load cases table, found within the "Loads" section.

3 Supports



Adjust edge supports, or add point supports and hole fixings in this section. To modify an edge support, click on the desired edge.



You have several support types to choose from:

- **None:** No support is applied.
- **Simple Support:** Fixed *out of plane* direction in *bottom surfaces* of each package.
- **Double Pendulum:** Fixed the *in plane* directions in *all surfaces*.
- **Simple on XY:** Fixed *in plane* directions in *bottom surfaces* of each package.
- **Fixed Support:** Fixed *all directions* in *all surfaces*.
- **Structural Silicone Support:** A silicone material is modelled to support the *inside* surface.

3.1.1 Structural Silicone Parameters

Structural silicones

Edge	Width [mm]	Thickness [mm]	Elastic modulus E [MPa]	
2	5.0	5.0	DOWSIL 121	1.33

If "Structural Silicone Support" is selected, a figure will appear, allowing you to define the following parameters for the silicone seal:

- **Width**
- **Thickness**
- **Elastic Modulus:** select from listed materials or put a custom value.

The silicone is modelled as point springs at each node of the edge's mesh, with stiffness values calculated based on the data you provide.

3.1.2 Spacer Specifications

Spacers

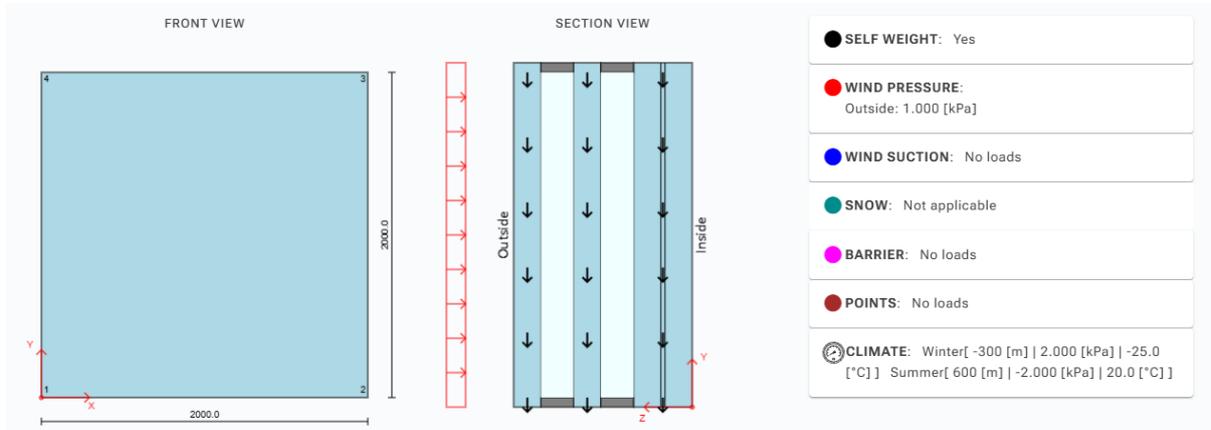
Spacer	Width [mm]	Elastic modulus E [MPa]
1	5.0	100.00
2	5.0	100.00

If cameras are present in your design, a figure will appear, allowing you to select the spacer specifications for each camera. This includes the **width** and **elastic modulus** for each.

In cases with two cameras, one refers to the **outside camera** and the second to the **inside camera**.

The spacer is modelled using truss elements that connect the mesh nodes at the edge across the camera.

4 Loads



Define your loads and load combinations here. All loads have a specified **duration**, which is used to calculate the **secant modulus** of interlayers for load combinations.

4.1 Load Types

4.1.1.1 Self-Weight

Enable this to apply gravity along the negative global Z-axis.

Self-Weight This switch lets you choose whether to include the self-weight of the glass in the calculation.

- **Include Glass Self-Weight:** Considers the glass weight, affecting results, crucial for structural safety.
- **Exclude Glass Self-Weight:** Ignores the glass weight, useful if it's negligible or already considered elsewhere.

Select the appropriate option based on project needs, as it significantly impacts calculation results and the final design.

Is it enabled?

Load duration:

4.1.1.2 Wind Load

A uniform load applied normal to the glass surface. Specify the intensity in kPa for **pressure** and **suction** on both **inside** and **outside** surfaces.

Wind pressure

Represents the force of the wind pushing the glass inward. Enter the value based on local meteorological data, in kPa, to ensure the safety and structural integrity of the glass.

Wind pressure, outside [kPa] Wind pressure, inside [kPa] Load duration

4.1.1.3 Snow Load

A uniform load in kPa, projected onto the horizontal footprint. It's always applied to the top side with respect to the Z-axis.

Snow load

Is assumed to act vertically and refer to a horizontal projection of the glass area. The load value to be entered always has a positive sign, and the load is always oriented downward (toward the negative direction of the global Z axis). The load on the horizontal projection of the glass surface must be provided; the reduction due to inclination is considered automatically.

Snow load [kPa] 1.000	Load duration 3 weeks
--------------------------	--------------------------

4.1.1.4 Barrier Load

A horizontal force defined by:

- **Intensity:** kN/m. A positive value is from the inside surface; a negative value is from the outside.
- **Height:** The height along the global Z-axis where the load is applied.
- **Load Width:** The spacing in mm for the load's width.
- **Orientation:** Choose to apply the load **orthogonal to the surface** or **horizontally**.

Barrier Loads

Enter the barrier load value, from internal surface to external, in kN/m. This value is a force per unit length representing the crowd force on the glass, applied at a certain height, with direction from inside to outside. To change the direction use the negative sign.

#	Load [kN/m]	Ortho	Y [mm]	Height [mm]	Load duration
1	1.000	<input checked="" type="radio"/>	800.0	50.0	30 seconds
2	-2.000	<input type="radio"/>	1000.0	50.0	30 seconds

+ ADD BARRIER LOAD

DELETED BARRIER LOAD

5 Options

You can set the **maximum allowable deflection** and define the verification standard:

- **ASTM 1300** (USA)
- **EN 16612** (European Union)

If you choose the **EN 16612** standard, you can set the **gamma coefficients** (γ_M) for material safety factors. These coefficients are used to calculate the design resistance (f_{gd}) based on the characteristic glass resistance (f_{gk}). You can view the values for load duration, f_{gd} at the surface, and f_{gd} at the edge.

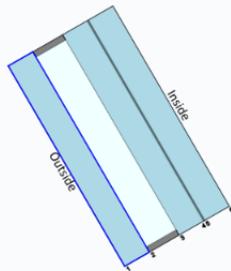
Verification options

$\gamma_{M,A}$ $\gamma_{M,V}$

Package	Pane	f_{gk} [MPa]	Glass type \odot	Surface treatment \odot	Prestress type \odot	Manufacturing process \odot	Glass edge type \odot
1	1	45.0	FloatGlass	AsProduced	Annealed	HorizontalTou...	AsCut
2	1	45.0	FloatGlass	AsProduced	Annealed	HorizontalTou...	AsCut
2	2	45.0	FloatGlass	AsProduced	Annealed	HorizontalTou...	AsCut

Design values

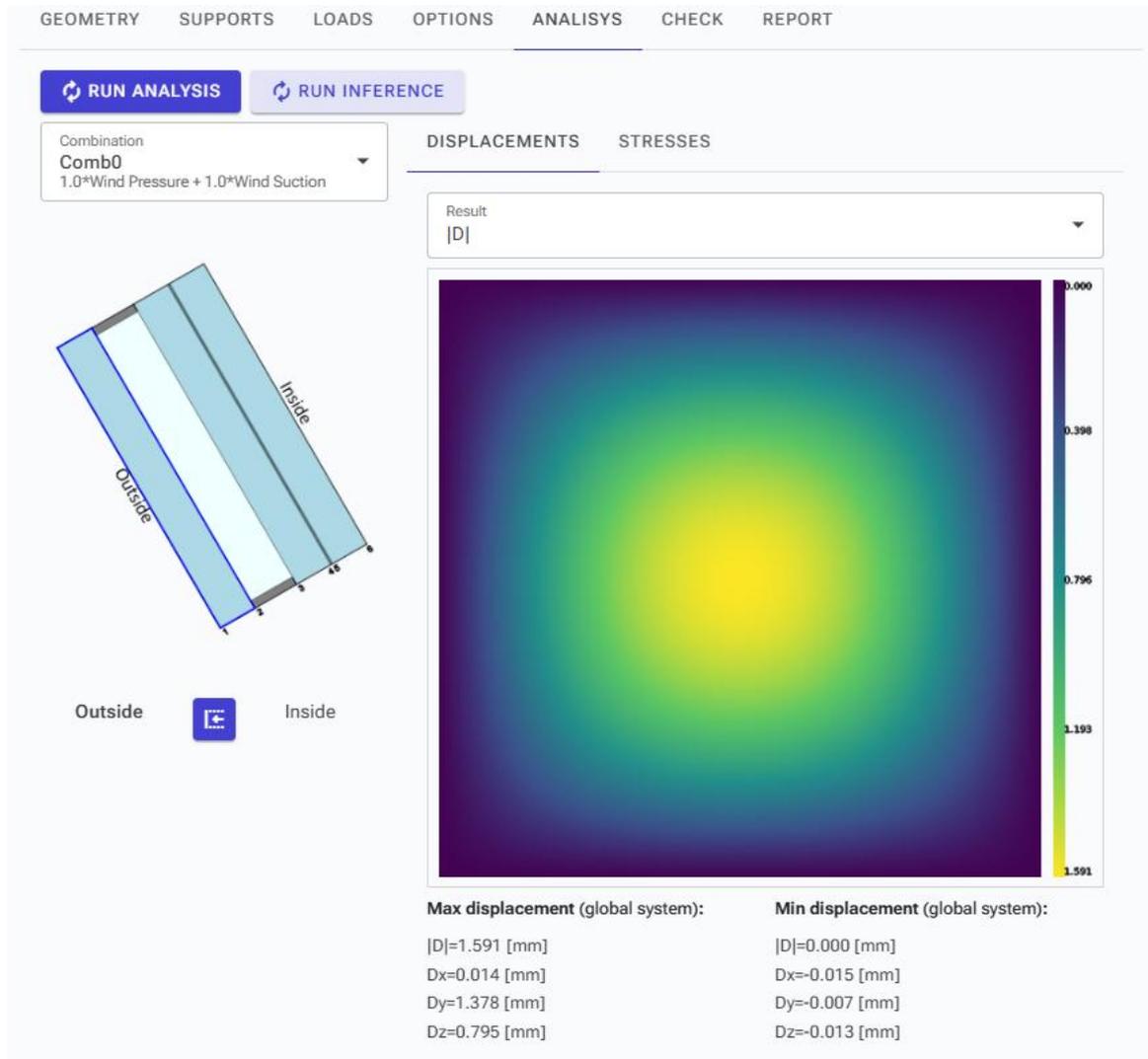
Select a pane to view resistances



Package 1, Pane 1

Name	Duration	f_{gd} surface [MPa]	f_{gd} edge [MPa]
Wind Pressure	3 seconds	25.0	20.0
Wind Suction	3 seconds	25.0	20.0
Barrier 1	30 secon...	22.4	17.9
Barrier 2	30 secon...	22.4	17.9
Pressure Summer	8 hours	14.6	11.6
Pressure Winter	8 hours	14.6	11.6
Temperature Sum...	8 hours	14.6	11.6
Temperature Winter	8 hours	14.6	11.6
Altitude Summer	10 years	8.1	6.5
Altitude Winter	10 years	8.1	6.5

6 Analysis



You have two options for analysis:

- **Run Analysis:** This performs a full Finite Element Method (FEM) analysis.
- **Run Inference:** This provides a quick preview of results using AI-driven calculations.

"Check and Report" options are only available after a successful FEM analysis.

If your analysis is successful, you'll see results for **displacements** and **stresses**.

6.1.1 Run Analysis

L'analisi è fatta in cloud ...

6.1.2 Run inference

GLASSS AI is based on a neural network trained and validated on a very large dataset of glasses. It thus exploits the state-of-the-art techniques of deep learning to prepare a fast and efficient solver handling the inputs of the user and predicting the outcome of typical FEM calculations over stresses and displacements along the three orientation axes. This tool's sole purpose is to provide the reader with an ultra-fast estimation of the order of magnitude of the stresses and deformations results to-be-output by the FEM.

The AI system does not allow the user to obtain a certified document. The AI aims to be a support for the user to accelerate workloads, obtain indications during the design process and soften the testing stage of pre-designed glass structures. The use of FEM is mandatory to obtain a certification of the calculation.

The use of the AI for those glasses whose features lay outside the training operating conditions is blocked, as any prediction would significantly deviate from the results obtained through FEM calculations. To do so, the platform automatically disables the **“Run inference”** button.

Even within supported conditions, predictions involving scenarios that may lead to **stresses exceeding 100 MPa** or **displacements greater than 10 cm** in any direction can show increased deviations from real-world behaviour. This is because such values lie beyond the network's current generalization capabilities. If the user foresees such extreme cases for his/her/their glass structure, the technical team encourages the use of the traditional FEM instead of the AI. Similarly, some discrepancies may occur in scenarios involving extremely low stresses or displacements.

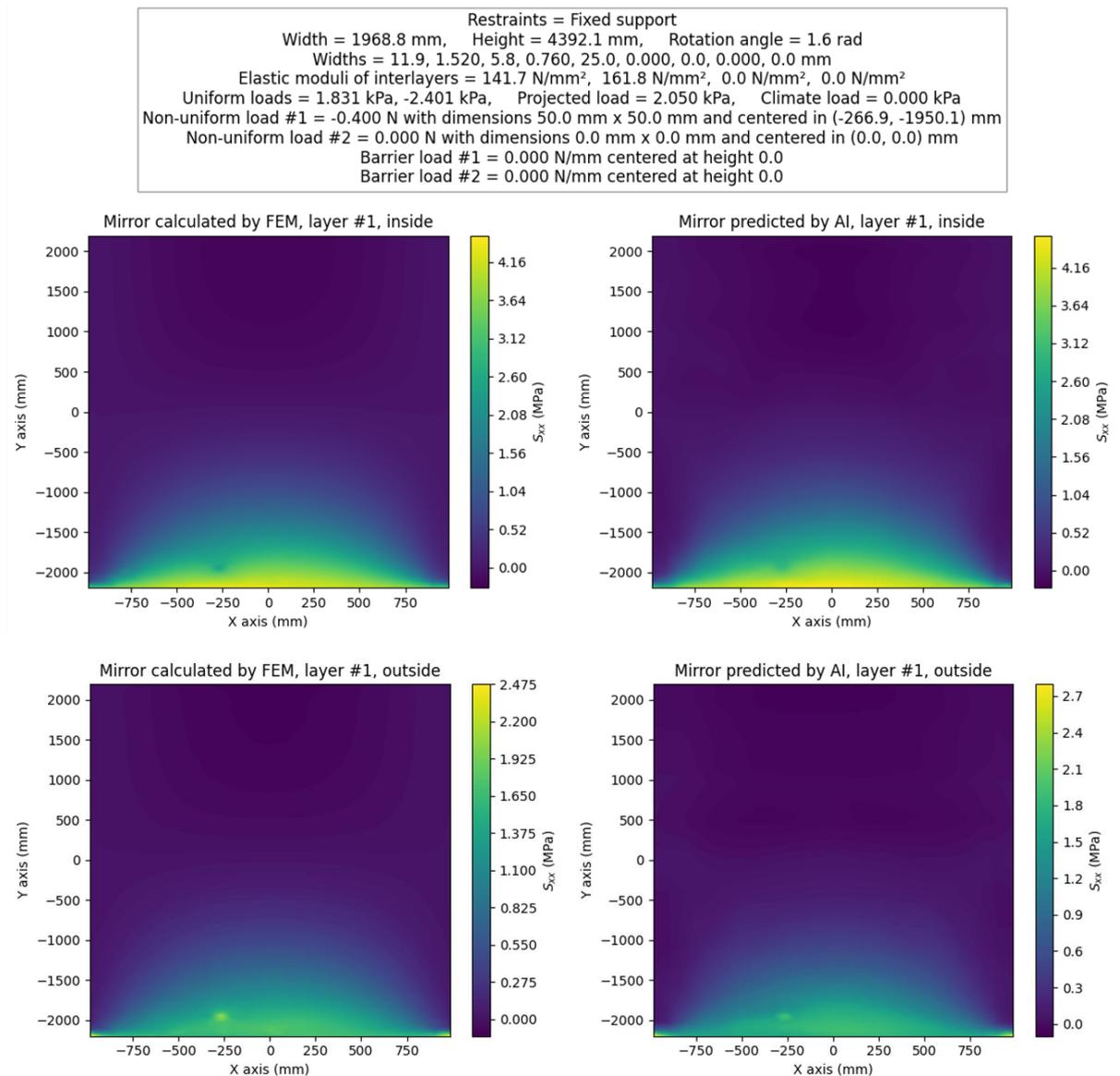
Do not despair though! We are continuously enhancing the AI models to improve speed, **accuracy** and **performance**! Future versions will deliver higher precision, broader applicability, and faster computations.

6.1.2.1 Available Cases

1. Rectangular base of the glass structure.
2. Self-weight included.
3. Edge constraints of the following classes:
 - Simple support on all edges.
 - Simple support on edges #1, #3 and not restrained on edges #2, #4.
 - Simple support on edges #2, #4 and not restrained on edges #1, #3.
 - Fixed support on edge #1 and not restrained on edges #2, #3, #4.
4. Horizontal or vertical glass.
5. No holes within the glass structure.
6. Any combo of dimensions between 50 cm x 50 cm and 3.2 m x 6 m (or vice versa) works.
7. Glass structure with maximum five layers of glass, each separated by any combination of PVB and/or air interlayers.
8. Thickness of each glass layer as by EU/US regulations.
9. Thickness of each PVB interlayer selected among [0.38 mm, 0.76 mm, 1.14 mm, 1.52 mm, 1.90 mm].

10. Elastic modulus of each PVB interlayer within [0.01 N/mm², 250 N/mm²].
11. Thickness of each air interlayer within [12 mm, 20 mm].
12. Total uniform load on each surface within [-5 kPa, +5 kPa].
13. Total projected load on each surface within [0 kPa, +5 kPa].
14. Total climate load within [-5 kPa, +5 kPa].
15. Any combination of non-uniform loads and barrier loads, provided that:
 - Up to two non-uniform loads with:
 - Side length to be chosen between 50 mm, 100 mm or 305 mm.
 - Load to be chosen between [1000 N, 1330 N, 1500 N, 2000 N] on either side of the glass structure (outside/inside).
 - Up to two barrier loads with:
 - Load within [0 N/mm, 3 N/mm].

6.1.2.2 Comparison between FEM and AI outputs



6.1.3 Viewing Results

If the analysis is successful, you can view displacements and stresses.

1. **Select the Load Combination:** Choose the desired load combination from the combination menu.
2. **Select the Layer:** Use your mouse pointer to click on the icon for the specific layer you wish to view.
3. **Choose Surface:** Select whether you want to see the **outside** or **inside** surface.

6.1.4 Understanding Displacement and Stress Results

For **displacement results**, you can view:

- The **absolute value of deformation**.
- The components of deformation along the **global axes**.

For **stress results**, you can see:

- **Maximum principal stresses (σ_1 , σ_2 , σ_3):** The first principal stress represents **maximum traction**, and the third represents **maximum compression**.
- **Sigma X (σ_x):** Stress along the global X-axis.
- **Sigma Y (σ_y):** Stress along the global Y-axis.
- **Tau XY (τ_{xy}):** Shear stress along the global XY plane.
- **Sigma ID (σ_{id}):** Von Mises stress.

7 Check

This section displays the **working ratios** for each load case. Results are **color-coded** to easily identify whether they pass or fail.

For each load condition, you can view the **maximum calculated values** alongside the **allowable limits**, specifying the:

- **Package** and **pane** where it occurs.
- **Location** of the stress (on the surface or at an edge).

GEOMETRY	SUPPORTS	LOADS	OPTIONS	ANALISYS	CHECK	REPORT	
Strength							
Maximum working ratio: 222.35 %							
Combination	Package	Pane	Location	Side	σ_{applied} [MPa]	$\sigma_{\text{allowable}}$ [MPa]	WR
ASD 2	1	1	Surface	Top	44.9	20.2	222.35 %
ASD 3	1	1	Edge	Top	9.4	18.3	51.36 %
ASD 4	1	1	Surface	Top	39.5	23.3	169.32 %
Deformation							
Maximum working ratio: 112.15 %							
Combination	Package	Pane		δ_{max} [mm]	δ_{lim} [mm]	WR	
ASDs 1	1	1		22.4	20.0	112.15 %	
ASDs 2	1	1		18.6	20.0	93.07 %	

8 Report

You can generate a detailed report of your calculations. This report includes all relevant data and analysis details.

To save or print the report, use the download or print buttons located in the top bar.

Please ensure you adhere to the **terms of use** as accepted upon accessing the service.

The screenshot shows the 'REPORT' tab selected in the software interface. The table of contents is as follows:

1 Introduction	1
1.1 Project information	1
1.2 Calculation methodology	1
2 Geometry	1
2.1 Glass build-up	2
2.2 Mesh	2
2.3 Supports	3
3 Material properties	3
3.1 Glass	3
3.1.1 Design values, package 1, pane 1	3
4 Load assumptions	4
4.1 Dead load	4
4.2 Wind load	4
4.3 Snow load	4
4.4 Barrier loads (line loads)	4
4.4.1 Barrier load 1	4
4.4.2 Barrier load 2	4
4.5 Point loads	4
4.5.1 Point load 1	4
4.5.2 Point load 2	5
4.6 Climate load	5
5 Load combinations	5
5.1 Allowable Strength Design (ASD)	5
5.2 Allowable Strength Design for Serviceability (ASDs)	6
6 Results	6
6.1 Critical combination check	6
6.1.1 Stress - ASD 2	6
6.1.1.1 Package 1	6
6.1.2 Deflection - ASDs 1	8
6.1.2.1 Package 1	8
6.2 Stress check (ASD)	9
6.3 Deflection check (ASDs)	9
7 Conclusion	10