21st International Research/Expert Conference
"Trends in the Development of Machinery and Associated Technology"
TMT 2018, Karlovy Vary, Czech Republic, 18th – 22nd September, 2018

POINT FIXED LAMINATED GLASS STRESS DISTRIBUTION UNDER OUT OF PLANE LOAD

Eren Kalay
Faculty of Mechanical Engineering
Istanbul Technical University
Turkey

C. Erdem Imrak
Faculty of Mechanical Engineering
Istanbul Technical University,
Turkey

ABSTRACT
Glass has been used for a long while in the cases where high load bearing capacity is not necessary. The innovational developments in glass and steel industry have made possible designers to create visually transparent facades that can satisfy high endurance limit. In order to enhance the transparency of structures minimizing the structural system is necessary. Spider glazing facade systems are highly in use among different glazing systems cause of visual clarity. This system has four point fixed holes. Stress concentration factor is one of the most critical thing to take into account in these holes. The aim of this paper is understanding the behavior of laminated glass especially on the mechanical connection between glass and point fix connector under out of plane load. It is presented analytically and experimentally that drilled hole shape on the glass structures has a great importance on the stress distribution.
Keywords: Point fixed, laminated glass, FEM Analysis

1. INTRODUCTION
Laminated glass that also called safety glass has two or more glass layer with and interlayer material like EVA or PVB that mostly used in practice. In loading conditions, special attention needs to investigate stress or strain values both experimentally and numerically. The holes on the glass panel creates high stress concentrations and these stresses can cause catastrophic failure because of glass brittleness [1]. For simple geometries it is possible to calculate the stresses or strains on components but due to the complication of the calculation of bolted assemblies, analysis is usually carried out by finite element method. In this paper, it is aimed to understand the behavior of glass panels under out of plane load and after that some constructive enhancements are applied to the holes to reduce stress value.

2. FEM ANALYSIS OF THE GLASS PANEL
Finite element analysis of this project is performed in Ansys 17.2. It is really important that modeling and experiment are in compliance due to its stress or strain value. Glass shows linear properties cause of its brittleness that causes catastrophic failures just after linear region. Material properties of glass PVB and steel is shown in Fig 1 [2]. Glass has 6 mm thickness and 0,76 mm thickness of PVB that layered between these two glasses under high pressure. The diameter of the holes that drilled from corners is 36 mm.

Table 1. Material properties of glass, PVB and steel

Material	Material behaviour	E-Young Modulus [N/mm2]	Poisson Ratio
Glass Panel	Linear	70.000	0,23
PVB	Nonlinear	1.5	0,49
Steel	Bilinear	210000	0,3

As is known, nonlinear properties should be added manually from the sources that contains stress strain curve values in tabular. For linear situation, it is enough to add poisson ratio and young modulus to identify the material.

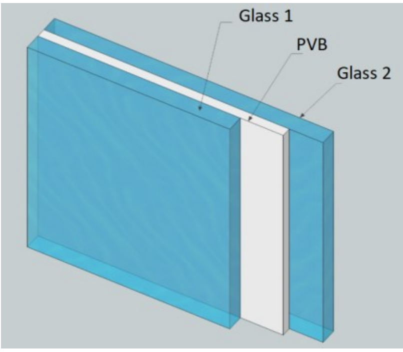

Fig 1. Laminated glass and bolt

After including material properties meshing process took place and Solid186 element type was used for glass panel. This element has 20 nodes and high displacement ability. Mesh quality is crucial to obtain good results for both stress and strain [3]. The glass around the hole and bolt is seen in Figure 1. Bush material is used not to contact glass and bolt to prevent crack failure and also uniform force distribution between glass and bolt should be provided in assembly.

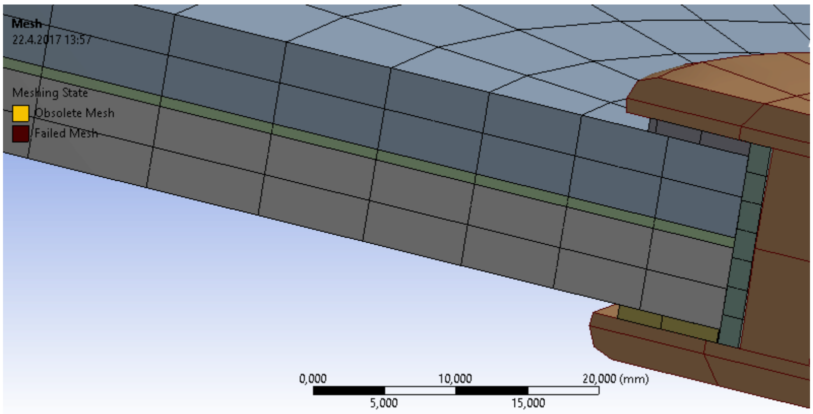

Fig 1. Meshing the assembly

Finite element analysis took place for three situations, bolted connection with no gap, with gap and with chamfered hole. Procedures for modelling are same for all these three conditions. The surface between glass and bolt is considered frictionless. Bolts are fixed and cylindrical support applied between bolt and hole of the glass. Cylindrical support should be specialized to move free radially. As it seen in the Fig 2, maximum stress value occurs towards center of the glass panel. The red section shows where maximum stress occurs and also where to stick the strain gages also.

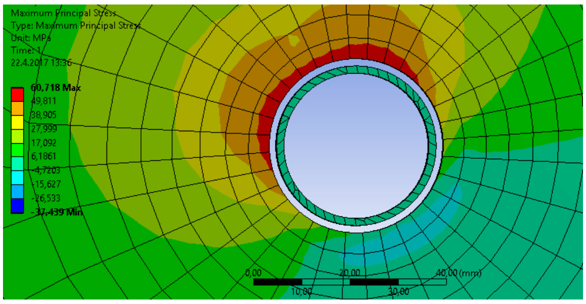

Fig 2. Stress distribution around the hole for the connection with gap

3. EXPERIMENTAL PROCEDURE

What we need before the test is a micrometer to measure the thickness of the glass, water gage to provide flatness of the glass panel, strain gages to measure strain mainly. In this paper Prosig marked device was used to collect the data from experiment.

Fig 3. Experimental setup

The thickness of the glass panel was checked after providing flatness with micrometer. Glass panel fastened to the frame by bolts from the corners. It should be taken into account that centering the bolts through the frame hole matters to obtain correct results and same results for all holes. This setup rearranged for other two conditions and processes are repeated.

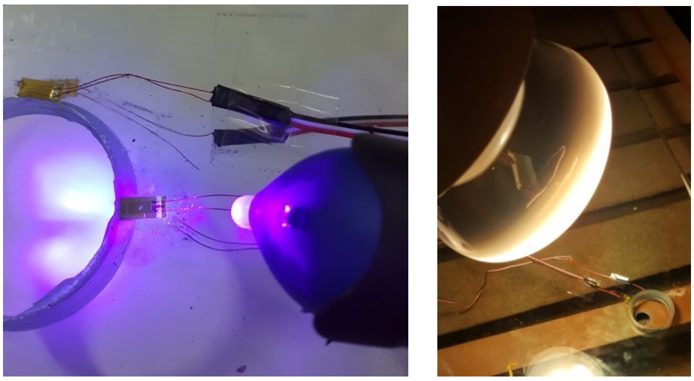

Fig 4. UV lights to stick gages

After specifying where maximum stress value emerges on the glass panel in finite element analysis gages were stuck there. First low powered UV light was used to stick and then 300 W powered UV light is used to stick quite solid after pouring special glue that is used for glasses between glass and gage surface. Next, 5000 N force was applied perpendicular to the glass surface. First the stress values that belong to the connection with no gap was obtained and these processes are repeated for other two situations.

4. RESULTS AND DISCUSSION

After gathering the experimental results, FEM results and test results are compared as shown in Table 2. It is obvious to claim that, bolted connection with no gap is the worst solution among these three conditions.

Table 2. Stress values for FEM and experiment are in comparison

Connection Type	FEM [MPa]	Experiment [MPa]
With no gap	124	105
With gap	60	57
Chamfered hole	53	46

Assembling parts with gap and chamfered drilled hole nearly have the same stress results but in order to avoid some eccentricity problems while mounting connection with gap could be the most proper choice. The difference between finite element analysis and experiment can be reduced with using two strain gages that perpendicular to each other. That provides to obtain the strain value that perpendicular to center direction which has considered in this paper.

5. REFERENCES

[1] Kalay E.: Spider Camın Düzleme Dik Yükleme Halinde Gerilme Analizi, Master Thesis, Faculty of Mechanical Engineering, Istanbul Technical University, Istanbul, 2017.

[2] Gergely M., László G. V., György S., László D.: Finite element analysis of laminated structural glass plates with polyvinyl butyral (PVB) interlayer, Civil Engineering, Periodica Polytechnica, 2012.

[3] Mocibob D.: Glass Panel under Shear Loading - Use of Glass Envelopes in Building Stabilization, Master Thesis, Lousanne Polytechnic, 2008.