

Master's Dissertation at the Div. of Structural Mechanics



DESIGN TOOL FOR STRENGTHENED GLASS - DESIGN AND IMPLEMENTATION

Jens Malmberg

Presentation

May 2006

Report

will be published as
report TVSM-5144

Supervisors

Kent Persson, *PhD*
Div. of Structural Mechanics

Jonas Lindemann, *Ph.D.*
Div. of Structural Mechanics

The work is performed at

Div. of Structural Mechanics,
Lund Institute of Technology,
Lund University

In cooperation with

Fasadglas Bäcklin AB

Background

During the last few years architects have become more interested in using glass in supporting parts of building structures, and in these structures minimizing the amount of other materials. The fixings of glass structures raise interesting problems, since this is where other types of materials usually need to be employed. An example of this is when glass is attached to railings, where the glass may be fixed through cylindrical or countersunk holes in the glass. Through these holes the glass may be connected onto the supporting structure by metal bolts, where the fixings need is to carry itself and the loads.

Laminated glass usually consists of two layers of very brittle hardened glass with a thin intermediate foil of PVB. PVB is a highly elastic material that keeps the glass in place in the event of failure, thereby preventing people from getting hurt.

Objective

Fasadglas is in need of a dimensioning tool where different design parameters concerning laminated glass can be determined. These parameters include distance between holes and edges, glass thickness, etc. The objective of this master's thesis is to develop such a tool, making it possible to perform

analyses on laminated glass in a simple manner. Material relationships established in an earlier master's thesis, as well as existing building codes, will be implemented. The tool will be based on the finite element method.

Project Tasks

- Implement a finite element solver, using shell elements with composite material formulation
- Implement a mesh generator for triangular or quadrilateral elements
- Implement a graphical interface capable of handling different geometries, supports and load types
- Implement relations between calculated stresses/strains and design parameters
- Calibrate and verify FE-analyses. The results will be verified by comparison to experimental results yielded in an earlier master thesis. The computational part will be programmed in FORTRAN, whereas the graphical interface will be developed in Python.



LUND
UNIVERSITY