STRUCTURAL ANALYSIS OF COMPOSITE TEST PANELS

Sean Berhan
LARSS student

Marshall Rouse
LARSS mentor

Structures Directorate
Structural Mechanics Division
Aircraft Structures Branch

ABSTRACT

An analytical study was conducted for a 1/4 scale model skin-stringer compression panel using STAGS (STructural Analysis of General Shells) code for finite element analysis. This was a study to investigate scaling methods for composite structures using reduced ply thicknesses for 1/2 and 1/4 scale models. A computer model for a curved panel in a pressure-box was also constructed in order to simulate test conditions in the pressure box test machine. The pressure box is a test facility at NASA Langley research center used to simulate flight conditions in fuselage panels. This model will be used to investigate modifications to the pressure-box for curved panels with different radii.

INTRODUCTION/BACKGROUND INFORMATION

Composites are materials in which non-woven fibers are oriented in a specific matrix for the purpose of increasing structural efficiency, i.e. producing stronger, lighter structures. The composite materials are made up of several plies or laminae. Each lamina consists of one row of parallel fibers or filaments. The lamina are stacked with various orientations of the filament directions between leach layer, resulting in a laminate matrix which has the desired strength or stiffness.

The use of polymeric composite materials in aircraft structural design can be a major factor in cost and weight minimization. However, the testing of composite structures using scale models have failed in the past to accurately predict the strength of full-scale structures. In response to this, a new method of composite structure scaling has been implemented using reduced laminate ply thicknesses corresponding to 1/2 and 1/4 scale models. Reduction of the ply thicknesses allows for scaling the original structure without changing the ply layups as was the case with previous scaling methods such as the ply-level method. It is believed that this approach will allow for a more accurate modeling of actual components.

Five test panels will be loaded to failure in the testing of this method ; one full scale, two 1/2 scale and two 1/4 scale models. A finite element computer model of a 1/4 scale panel was developed and analyzed in order to predict the structural response which could be expected to be seen in the testing of the actual model.

The curved panel to be tested in the pressure-box test machine has a hybrid composite skin consisting of a Hercules AS4 graphite fiber and Fibrite 938 epoxy material system with intraplied S2-938 fiberglass-epoxy material straps. The pressure-box test machine is designed to apply axial loads up to 7000 lb/in and internal pressure loads up to 20 psig. Axial loads are applied to the test panel by two 225-kip hydraulic actuators connected to a curved steel plate known as the axial load plate. Circumferential or hoop loads which develop in the skin of the test panel are provided by an annular steel plate known as a hoop load plate and two steel rods connected to the sides of the panel. The steel rods include turnbuckles which are used to regulate the proper hoop loads in the panel for any given load condition. Pressure is applied to the concave side of the test panel using a 100 psi air supply source and a pneumatic control system.

Finite element analysis of the test panel is needed as an analytical tool to determine the turnbuckle forces needed to create an appropriate stress state in the panel. It is also used to compare analytical and experimental results when using different loading conditions.

SUMMARY

The panel design is a five-stringer composite stiffened compression panel. They are made from Hercules AS4/3502 CFRP graphite epoxy material. They include aluminum ribs with shear-ties to simulate how the fuselage panels would be fastened to the aluminum frame in an actual aircraft. The skin of the panel has a quasi-isotropic laminate with 0^{o} ,+/- 45^{o} , 90^{o} plies.

A linear finite element analysis of the 1/4 scale model was conducted using STAGS (STructural Analysis of General Shells) computer code developed at Lockheed Palo Alto Research Laboratory. STAGS is a finite element code for general-purpose analysis of shell structures. In order to perform the analysis the panel was broken down into rectangular shell units which were individually specified as to their geometry, position, material composition, etc. Each shell unit was designated with a specific number of rows and columns, according to their size and shape, to create enough nodes for an accurate finite element analysis.. The shell units were combined to form a shell structure of the panel. A 10,000 lb load was applied to simulate the conditions that the panel will be exposed to during testing.

The finite element model was post-processed using PATRAN 3 computer code. First, a model file was created from the STAGS code, along with a nodal displacement file and an integrated element stress file. These files were imported into PATRAN 3 for the post-processing. PATRAN 3 was used to generate a graphical representation of the model and computer-enhanced images of the stresses and displacements caused by the applied load. Figure shows the finite element model.

Figures 1 and 2 show the panel with strain contours in the X and Y directions, respectively. Both plots clearly show the areas of greatest stress intensity surrounding the point on the panel where the compressive load was applied parallel to the X-axis. Figure 3 is a displacement plot, again showing the greatest deformation at the point of the applied load.

The same procedure utilizing STAGS and PATRAN 3 was used for a quarter model of a curved panel in the pressure-box test machine described above. This model included a hoop load plate, an axial load plate, test panel, turnbuckles, and an axial load actuator. The computer simulation included an axial load of 1000 lbs applied through the actuator and a 1000 lb load through the turnbuckle, as well as a uniform pressure of 18.2 psi applied to the concave side of the curved test panel. The boundary conditions around the straight edges of the model were fixed to permit radial displacements only. This was done to more accurately simulate a cylindrical shell subjected to internal pressure. Figures 4 and 5 represent strain contour plots of the curved panel model in the X and Y directions, respectively.

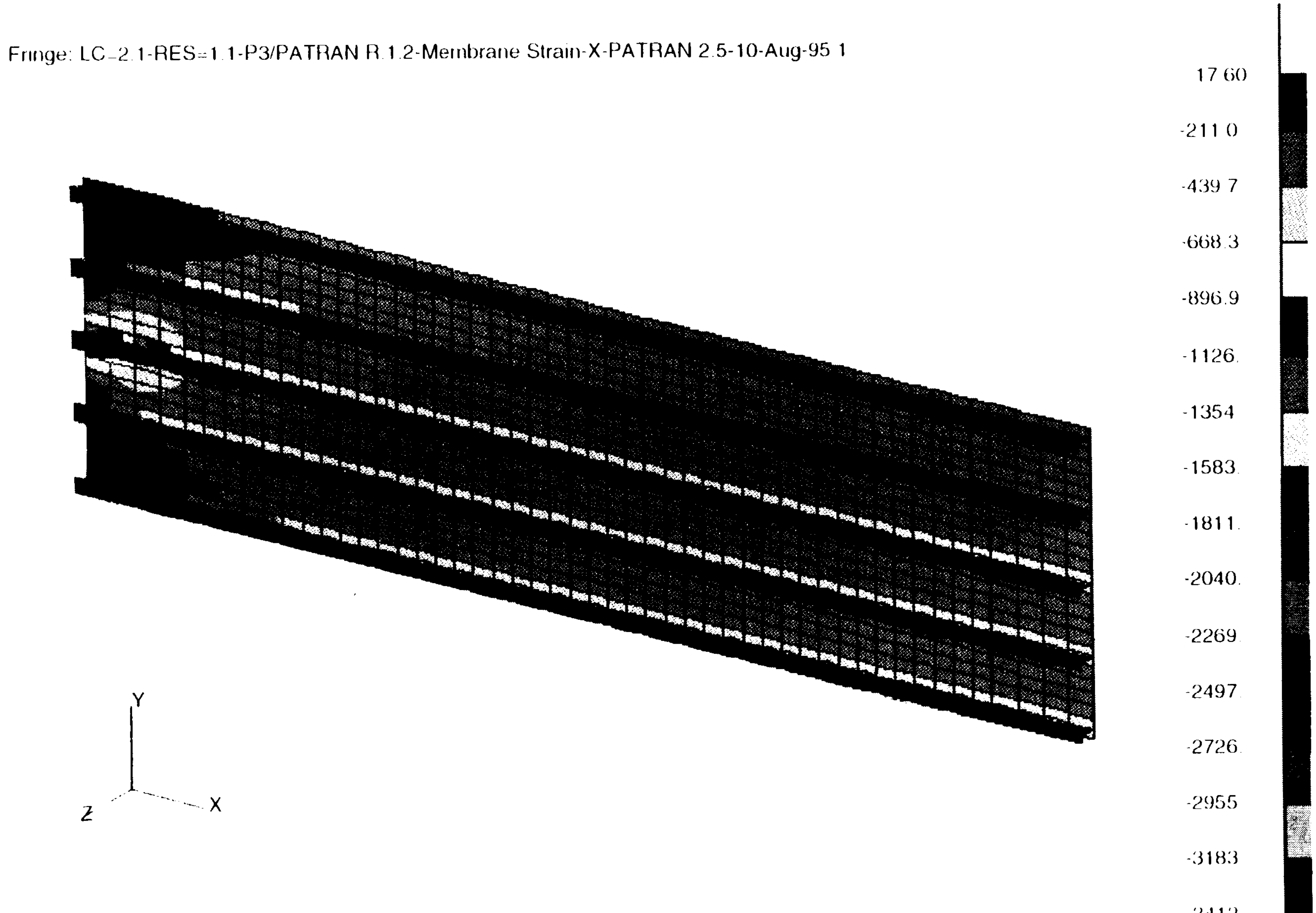


Figure 1. Strain contour plot of compression panel in the X-direction

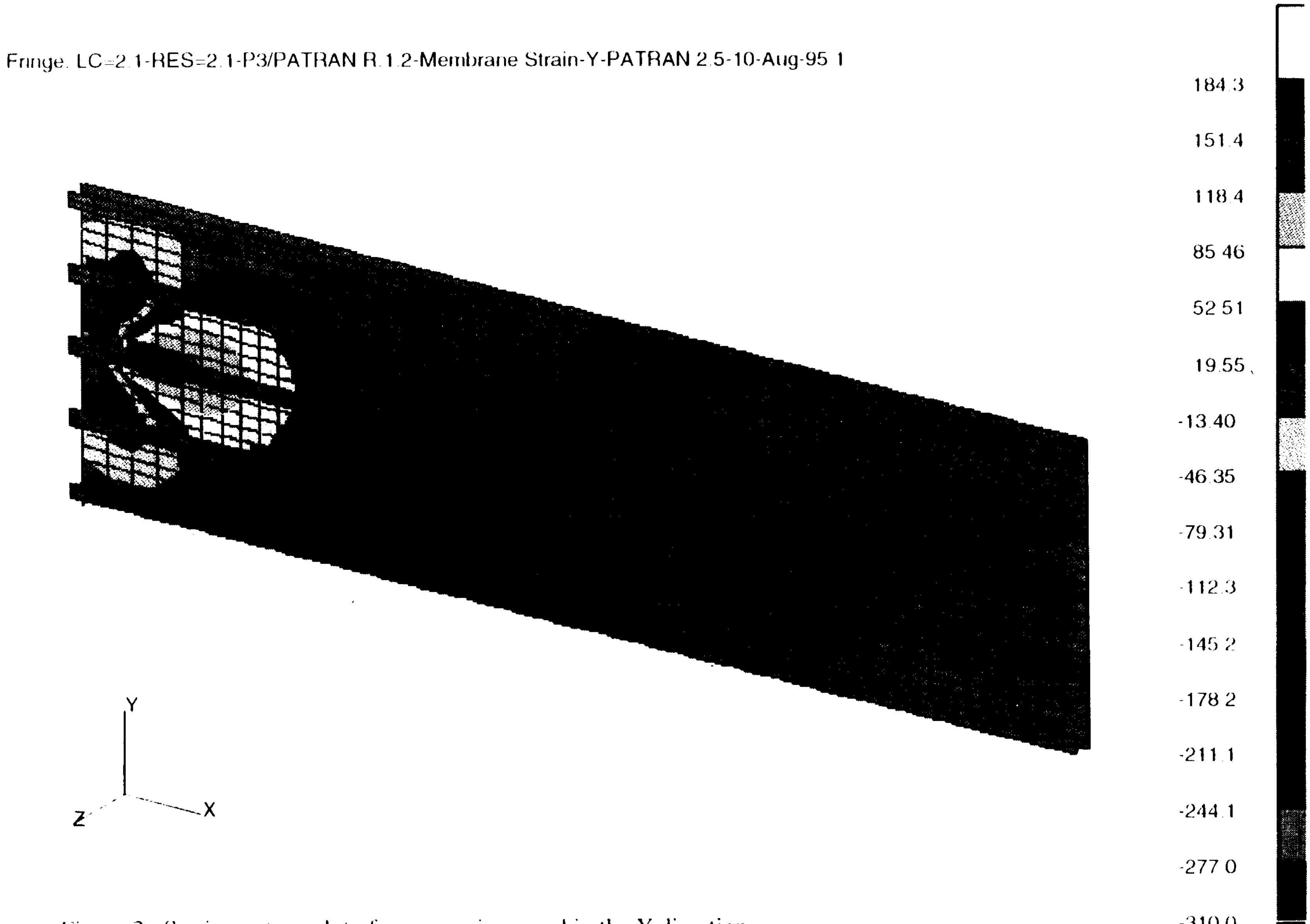


Figure 2. Strain contour plot of compression panel in the Y-direction

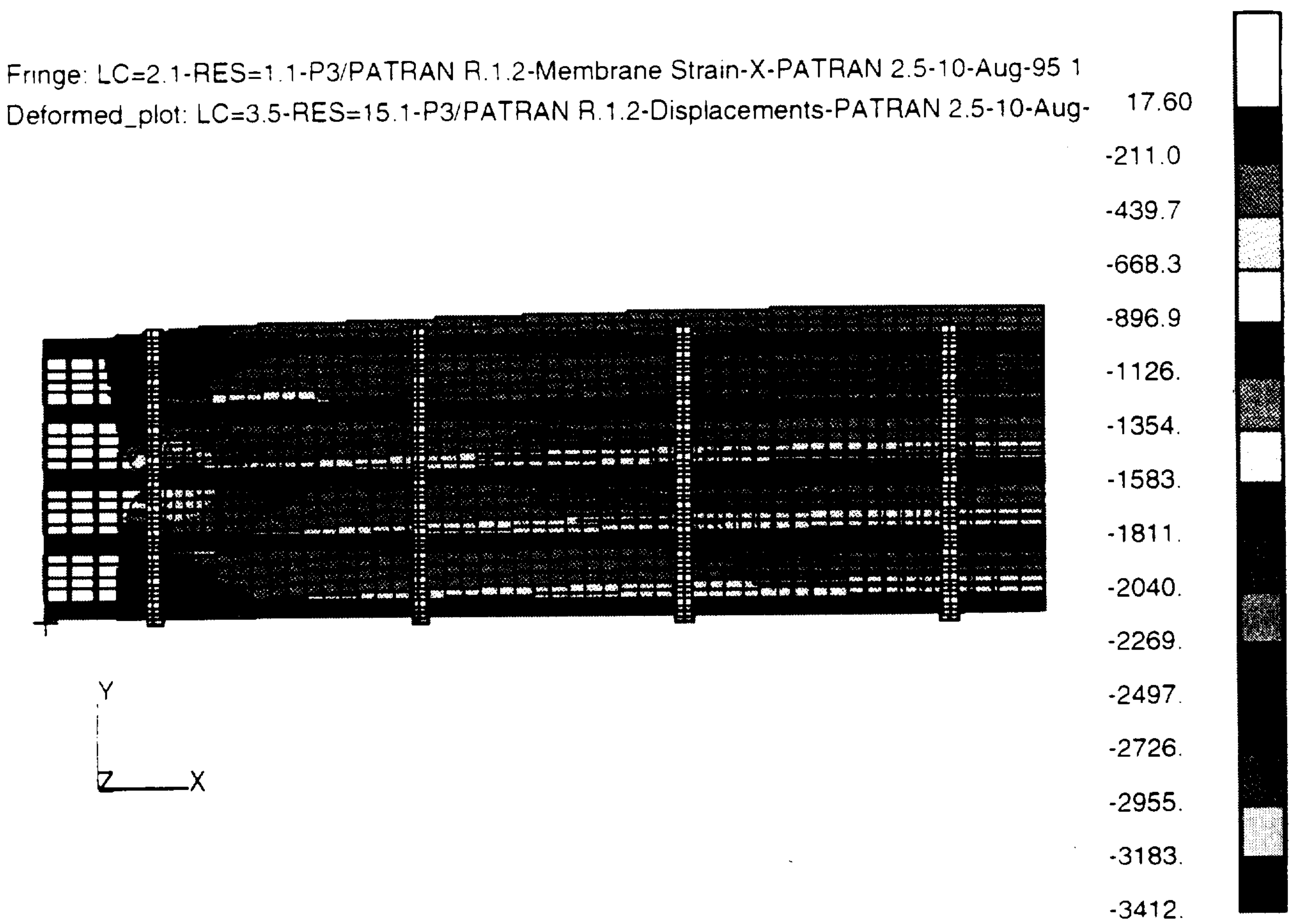


Figure 3. Deformation plot of compression panel

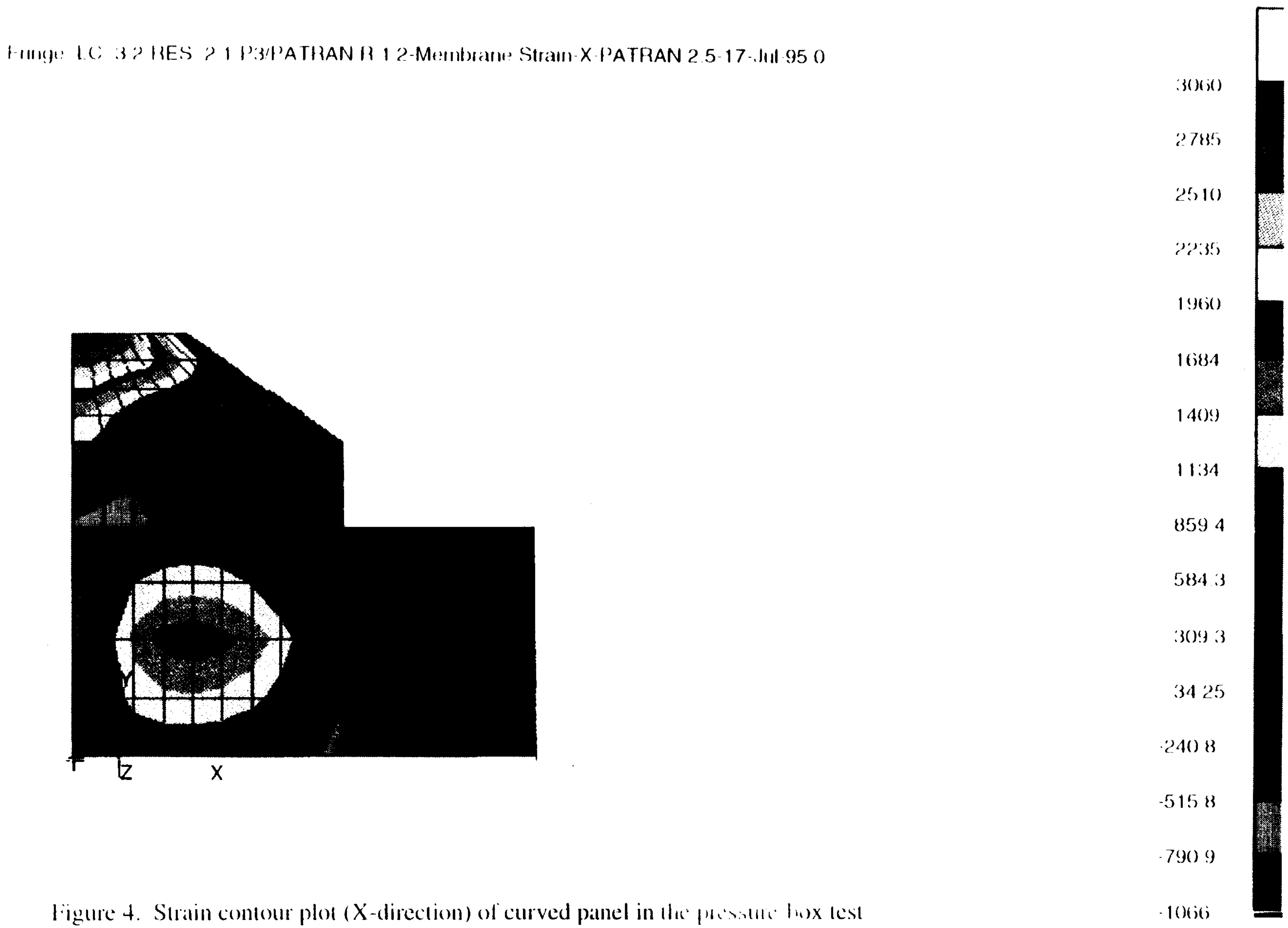
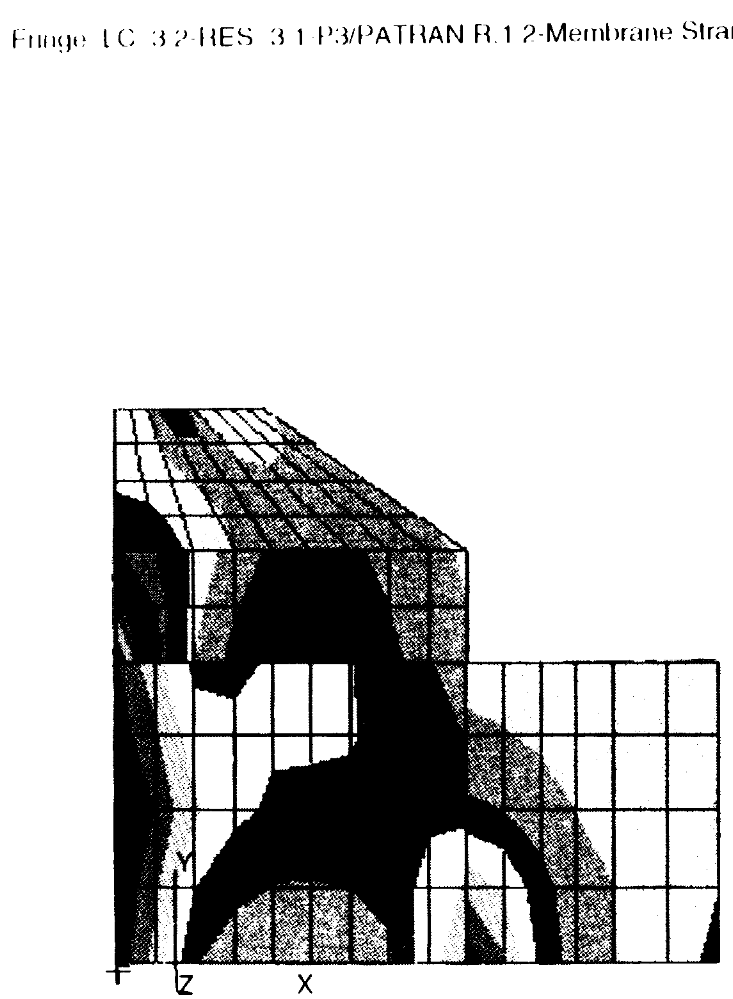


Figure 4. Strain contour plot (X-direction) of curved panel in the pressure box test machine



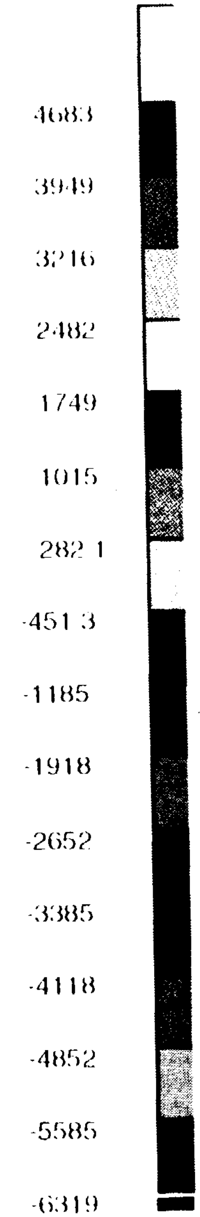


Figure 5. Strain contour plot (Y-direction) of curved panel in the pressure-box test machine

RESULTS/CONCLUSION

A finite element model has been developed to study the structural response and scaling effect of stiffened compression panels. This model will serve as a basis for further analysis to conclude these objectives. The finite element model of the curved panel in the pressure-box will be used to develop modifications to the pressure-box test machine for panels with different radii. Further refinement of the model will be carried out to serve this purpose.