NDE2002 predict. assure. improve.
National Seminar of ISNT
Chennai, 5. – 7. 12. 2002
www.nde2002.org

Prediction of Stresses in the Vicinity of Holes in the Launch Pedestal

P. Srinivas, P. Selvaraj, S. Sankaran

SHAR Central Designs, Satish Dhawan Space Centre
SHAR, Sriharikota – 524 124, India.
srinivaspeddi@yahoo.com, selvaraj189@hotmail.com,
sankaran@shar.gov.in

Abstract

The Launch Pedestal, which was built for launching PSLV rocket is also being used for launching the GSLV rocket. The rocket is supported by the Launch Pedestal till lift-off. The Launch Pedestal is a steel structure of size 10 m x 10 m in plan and is 7.627 m high. It has got four legs of 2 m height, fixed to the foundation; it is made up of a box girder of 3 m depth and the working platform is at 5 m level; a central tapered annular ring of 2.8 m inner diameter and 3.2 m outer diameter and 2.267 m height is welded over the working platform. On the top of this ring, the rocket is supported.

The PSLV is supported at eight locations using support blocks on the top of Launch Pedestal. The GSLV is supported on twelve locations using support blocks. The locations of support blocks of GSLV are different from that of PSLV, and hence provision is made to fix altogether twenty support blocks on the top of the Launch Pedestal. Each support block has got at least six holes while some of the support blocks have more than six holes. In addition to this, there are holes drilled for fixing the heat resistant refractory cement etc. to protect the Launch Pedestal from the severe thermal conditions during launch. In all, more than 200 of holes of size 10 mm and above are drilled on the 30 mm thick top plate of the Launch Pedestal. On this plate, either the 2800 kN weight of PSLV or 4100 kN of GSLV is transferred depending on the launch.

The Launch Pedestal is analysed and designed for the above loads initially without considering holes. Since the number of holes drilled are more, there was a need to assess the stresses around the holes. To predict the stresses in the vicinity of holes Finite Element Analysis of the top plate was carried out with and without holes and the results are presented.

1. INTRODUCTION

The Launch Pedestal (LP), which was built for launching the Polar Satellite Launch Vehicle (PSLV), is also being used for the launch of Geo-synchronous Satellite Launch Vehicle (GSLV). The rocket is supported on the LP till lift-off.

1.1 Details of LP are as follows

The view of the Launch Pedestal during lift-off is shown in Fig. 1(a). The view showing the plan of LP is shown in Fig. 1(b) and Fig. 1(c) shows the view of support block. LP is a steel structure of size 10 m X 10 m in plan, and 7.267 m high. It has got four legs of 2 m height, fixed to the foundation and is made up of a box girder of 3 m depth and the working platform is at 5 m level; a central tapered *annular ring* of 2.8 m inner diameter and 3.2 m outer diameter with thickness of 30 mm is placed at a height of 2.267 m above the working platform. On the top of this ring, the Launch Vehicle is supported. For the support of the Polar Satellite Launch Vehicle (PSLV), holes had to be drilled at the 8 support block locations in the top plate. It was proposed that the same LP will be used for the launch of Geo-synchronous Satellite Launch Vehicle (GSLV), which is supported at 12 locations using support blocks. The locations of the support blocks of PSLV are different from that of the GSLV. Each support block has got minimum of 6 holes or more.

In addition to this, there are holes drilled for fixing the heat resistant refractory cement etc. to protect the LP from the severe thermal conditions during lift-off. In all, 224 holes are drilled in the top plate till now, out of which 104 holes are of diameter 10 mm, 96 holes are of diameter 11 mm, and 24 holes are of diameter 16 mm. Fig. 2 shows the plan of the top plate of the launch pedestal for GSLV & PSLV.

On the top plate of LP, either the 2800 KN weight of PSLV or 4100 KN weight of GSLV is transferred depending on the launch. The LP is analysed and designed for the above loads initially without considering the holes. Since the number of holes has now been increased to 224, there was a need to assess the stresses in the top plate particularly in the vicinity of holes. There are various Non-Destructive Evaluation (NDE) methods to obtain the stresses. Few among the methods are:

- Theoretical (classical) solution.

- Experimental Methods[1] like Photo elasticity, Brittle coating methods etc.
- Numerical Methods

The closed form solution is available for the stresses around a hole in the plate subjected to pure tension[2]. Curves are generated with respect to the ratio of diameter of hole to the width of the plate versus the Stress concentration factor (k). The maximum value[3] of k is found to be 3. The closed form solution for a plate with a hole subjected to pure bending[4] gives a stress concentration factor of 1.8 around the hole, as per the relation given below,

$k = (5 + 3\nu) / (3 + \nu)$,

where k is the stress concentration factor and ν is poissons ratio (for steel it is 1/3). But if the number of holes present is more and they are closely spaced as in this case, there is no closed form solution found in the literature. Because of the limitations inherent in the experimental methods, these methods were also not adopted. Thus, in the absence of any theoretical or proper experimental methods, it was decided to go in for a Numerical method to find out the stresses. Finite Element Analysis (FEA) is one of the widely used methods to obtain the stresses. FEA of the top plate was carried out with and without holes and the results are presented.

2. ANALYTICAL MODEL

Since the structure, along with the loading and boundary conditions are symmetric, for critical load cases, only one quarter of the plate is modelled, and symmetric boundary conditions are applied. Fig. 3 shows the plan of the quarter plate. The modelling of the small holes posed the biggest challenge during modelling stage. Various softwares like NISA[5], ANSYS, STAAD etc., are available for FEA purposes. To begin with, NISA software was chosen.

2.1 Modelling Difficulties

Initially, the model was developed in AutoCAD 2002. For analysis purposes, the model was to be exported to the NISA software (version 6/ Display III) in the form of IGES format. However, this attempt was unsuccessful as problems were encountered during the conversion process. So, the whole model was recreated in NISA package, which was a tedious process. All the coordinate locations (57 in number) of holes had to be noted from the AutoCAD and fed to the NISA package. Then segregation of holes according to their diameters was done and the circles were created with respective diameters. For defining a surface in a 3-D space, patches are to be made. But, in NISA it was found to be difficult to patch a small hole in a large plate. Also, it was observed during modelling, that if the dimensions of the plate are large compared to the diameter of the hole, like in this case, the software is not capable of auto meshing, leaving the choice of only manual meshing to the user. Manual meshing of approximately 57 holes by creating patches is a very tedious and time-consuming process.

Then the model was tried for analysis in ANSYS[6]. The available Workstation configuration is as below for problem solving:

Processor: Pentium III; 512 MB RAM; Hard Disk Drive: 40 GB capacity.

But, attempt of IGES format conversion from AutoCAD 2002 to ANSYS Linear Plus (Version 5.3) package, was also found unsuccessful. The model was then remade in ANSYS package. Initially, only a quarter model was made in ANSYS. Then all the 57 circles with corresponding areas equal to holes were modelled. With the help of Boolean operations, the holes were subtracted from the pedestal plate. Then the model was tried for auto meshing by giving the required parameters of material properties etc. However, the meshing was getting aborted with the default memory allocations.

At this stage, it was thought that changing the default memory allocations may help in getting the solution. This was done by allocating a memory of 150 MB for total workspace instead of 32 MB and 48 MB for database instead of 16 MB. Though this attempt helped to overcome the meshing problem, the solution could still not be obtained.

Then it was thought of creating model again by dividing the plate into 3 sectors and meshing it individually. With the ANSYS command "GLUE", the 3 area sectors were first glued to obtain continuity in mesh and the meshing was done individually, followed by merging of nodes. Only then the solution was possible.

Element Chosen: Shell 93. This element has got 6 degrees of freedom at each node. The SMART SIZING, which specifies meshing parameters for automatic (smart) element sizing of the element was adopted.

Boundary Conditions: Pinned on both the edges of the plate; that is on the inner and outer edges.

Total number of nodes formed: 12129; Total number of elements formed: 3914

The pressures applied are as follows:

As the GSLV load of 4100 KN is greater than the PSLV load of 2800 KN, only GSLV load is considered for analysis. There are 12 support blocks and hence the load on each support block is 4100 / 12 = 341.67 KN. Out of 12 support blocks, 8 nos. are having the area of (0.2 * 0.2) m^2. Hence the pressure applied is 8541666.67 N /m^2 (8.54 MPa). Remaining 4 support blocks are having the area of (0.4 * 0.2) m^2 and hence the pressure applied in these locations is 4270833.33 N / m^2 (4.27 MPa).

The analytical model of the plate without holes is shown in Fig. 4 and with holes is shown in Fig. 5. The problem was analysed and the results were thus obtained.

3. RESULTS AND DISCUSSIONS

Von-Mises Stress plots, which give the combined effect of normal and shear stresses were plotted. The Von - Mises Stress plots for the plate without holes is shown in Fig. 6 and for the plate with holes is shown in Fig. 7. It may be noticed from the stress plots that the maximum values of stresses from the plate without holes is in the band of 0.148 E 09 N / m^2 (148 MPa) to 0.167 E 09 N / m^2 (167 MPa). However, the stresses in the model with holes are considerably high near the vicinity of holes in the band of 0.245 E 09 N/m^2 (245 MPa) to 0.367 E 09 N / m^2 (367 MPa). This shows that the values of stresses increased by a factor of 1.7 to 2.2. (However, the maximum stress is 0.11 E 10 N/m^2 (1100 MPa), found at only one element, which is highly distorted, due to bad aspect ratio. Hence, this value is ignored). The structural energy error norm is 9.6%, which is acceptable for the present work.

Also it is observed that the stresses in the band mentioned above are found to be only near the vicinity of the hole. The affected length is approximately equal to the diameter of the hole. From Fig.8, it can be seen that the stress concentration is more around the holes only and the analysis confirmed that the region near the boundary is not affected as per the Sant Venants Theory[7].

4. CONCLUSIONS

- The maximum stresses in the plate without holes are found to be in the band of 148 MPa to 167 MPa.
- The maximum stresses in the plate with holes, near the vicinity of holes are found to be from 245 MPa to 367 MPa.
- The stress concentration factor is found to be in the range of 1.7 to 2.2.
- The stress concentration is highly localized.
- Though the loading is not distributed through out and holes are at many locations, the analysis revealed that the stress concentration factors are found to be lying approximately within the range given in the literature.

5. ACKNOWLEDGEMENTS

This paper is presented with the kind permission of Shri K. Narayana, Director, Satish Dhawan Space Centre SHAR (SDSC), Sriharikota. The authors are grateful to Shri P. VenkateswaraRao, Associate Director, SDSC, and Shri V. R. Subramanyam, Deputy General Manager, SHAR Central Designs, SDSC, for their help and encouragements during the course of this work.

6. REFERENCES

1. "Experimental Stress Analysis" by Dally, James W Riley & F William, Mc Graw Hill, New York, 1965.
2. "Theory of Elasticity" by S. P. Timoshenko & J. N Goodier, McGraw Hill Publications, New York, 1970.
3. "Design of Machine Elements" by V. B. Bhandari, Tata McGraw Hill Publications, New Delhi, 1996.
4. "Theory of plates and shells" by S. P. Timoshenko & Wonowsky-Krieger, McGraw Hill Publications, Auckland, 1970.
5. "NISA Manual", Version 6, 1995.
6. "ANSYS Manual", Version 5.3, 1994.
7. "Structural Analysis" by R.C. Koates, M.G. Coutie & F.K. Kong, ELBS Publications, 1980.

Figures

Figure 1a

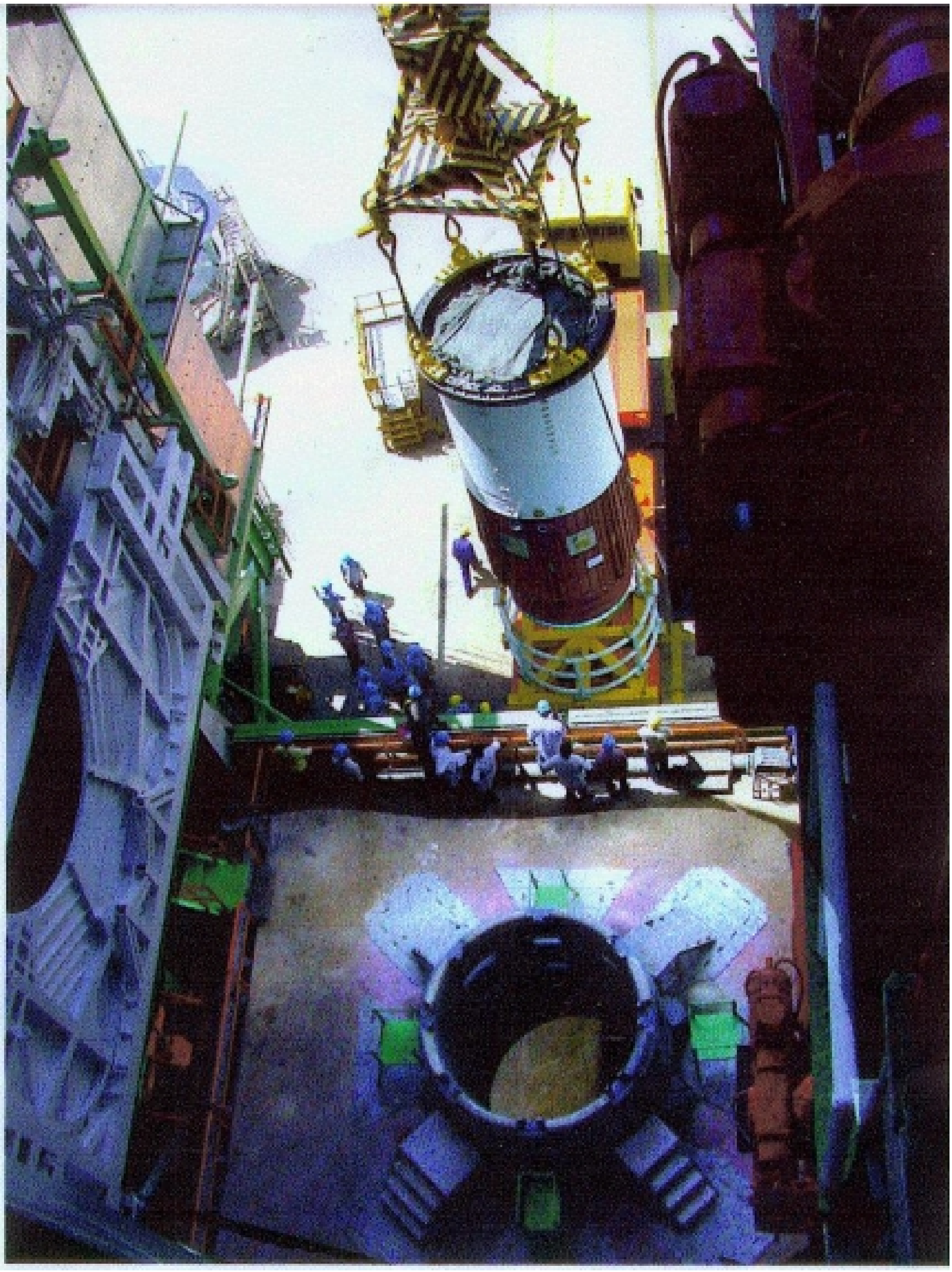

Figure 1b

Figure 1 c: View showing support block of LP

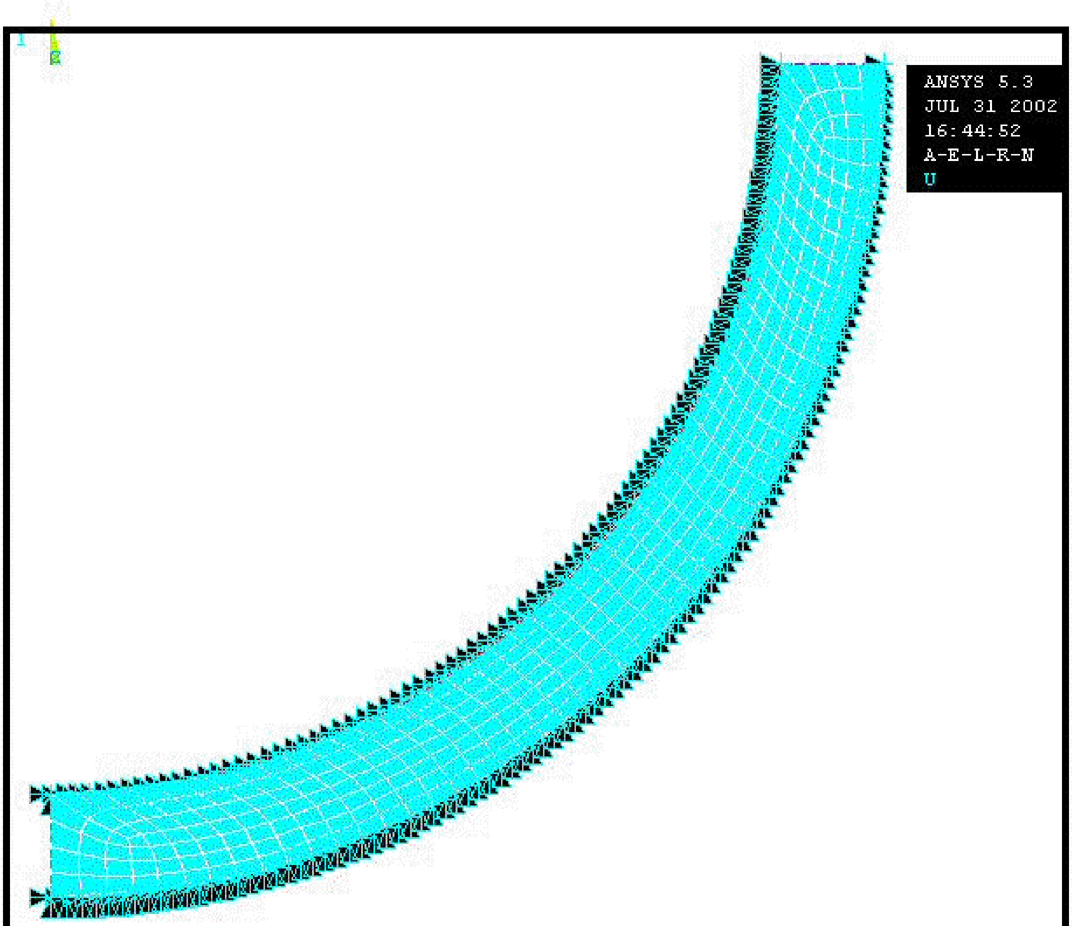

FIG 4

ANSYS MODEL OF LAUNCH PEDESTAL
(WITHOUT HOLES)

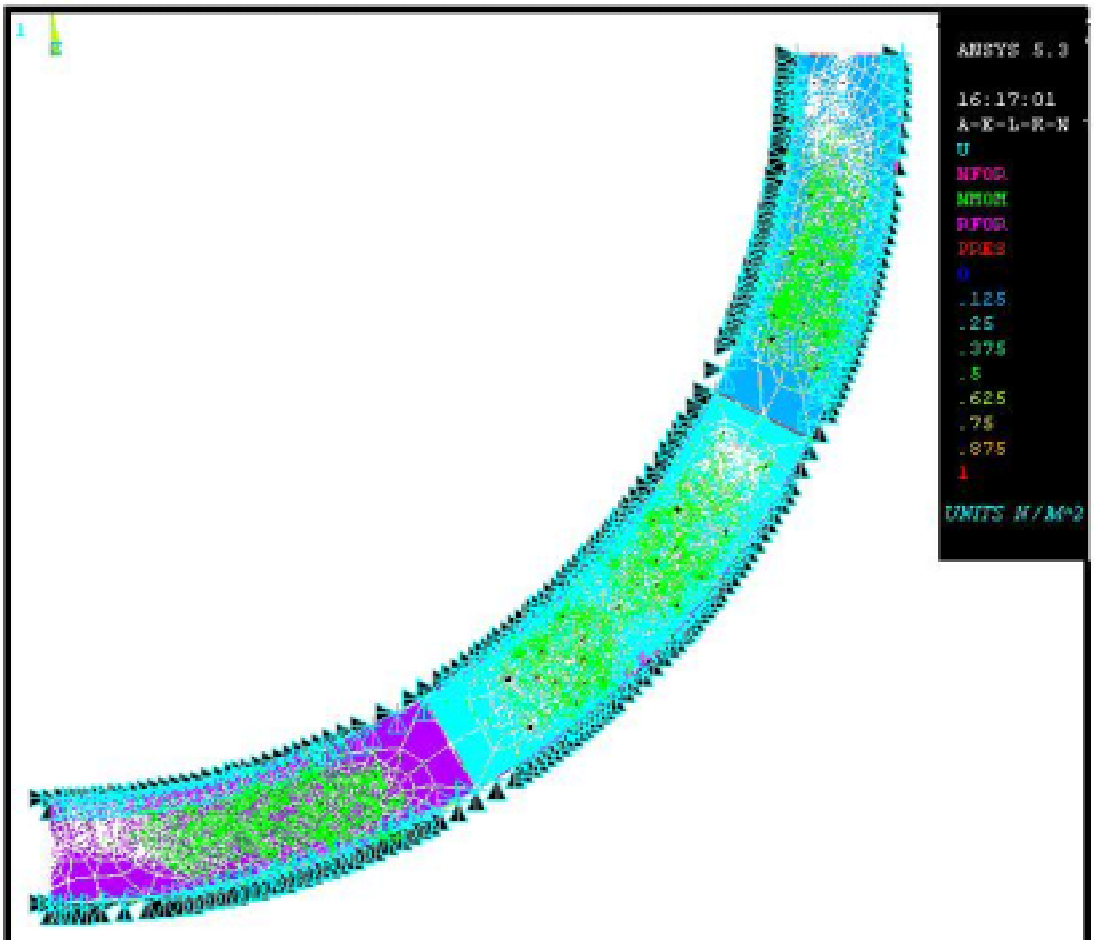

FIG. 5

ANSYS MODEL OF LAUNCH PEDESTAL (WITH HOLES)

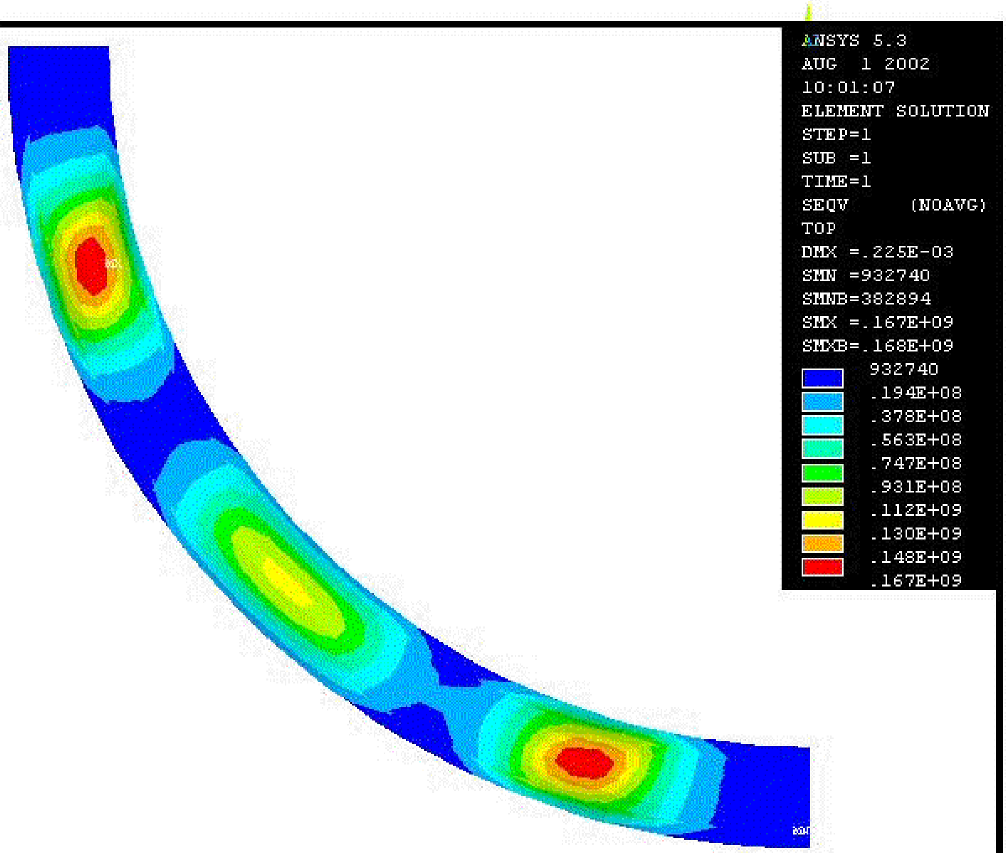

FIG 6

VON-MISES STRESS PLOTS

(LP WITHOUT HOLES)

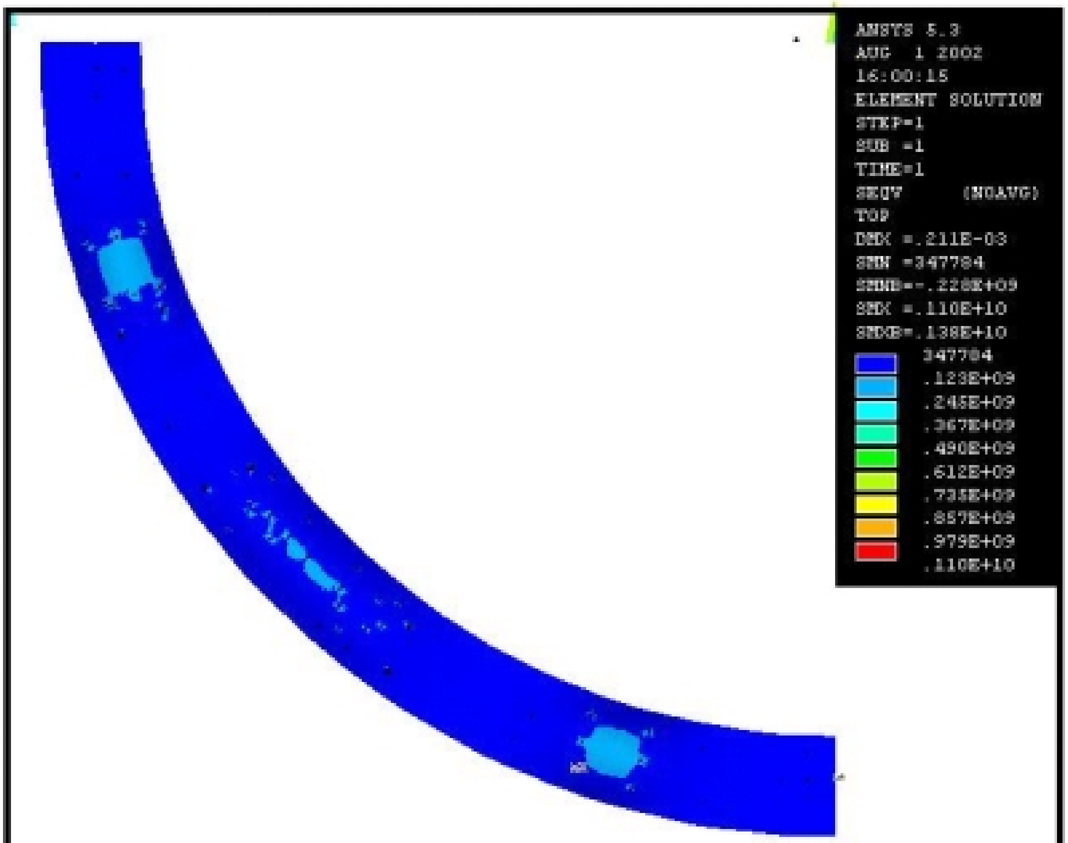

FIG. 7

VON-MISES STRESS PLOTS
(LP WITH HOLES)

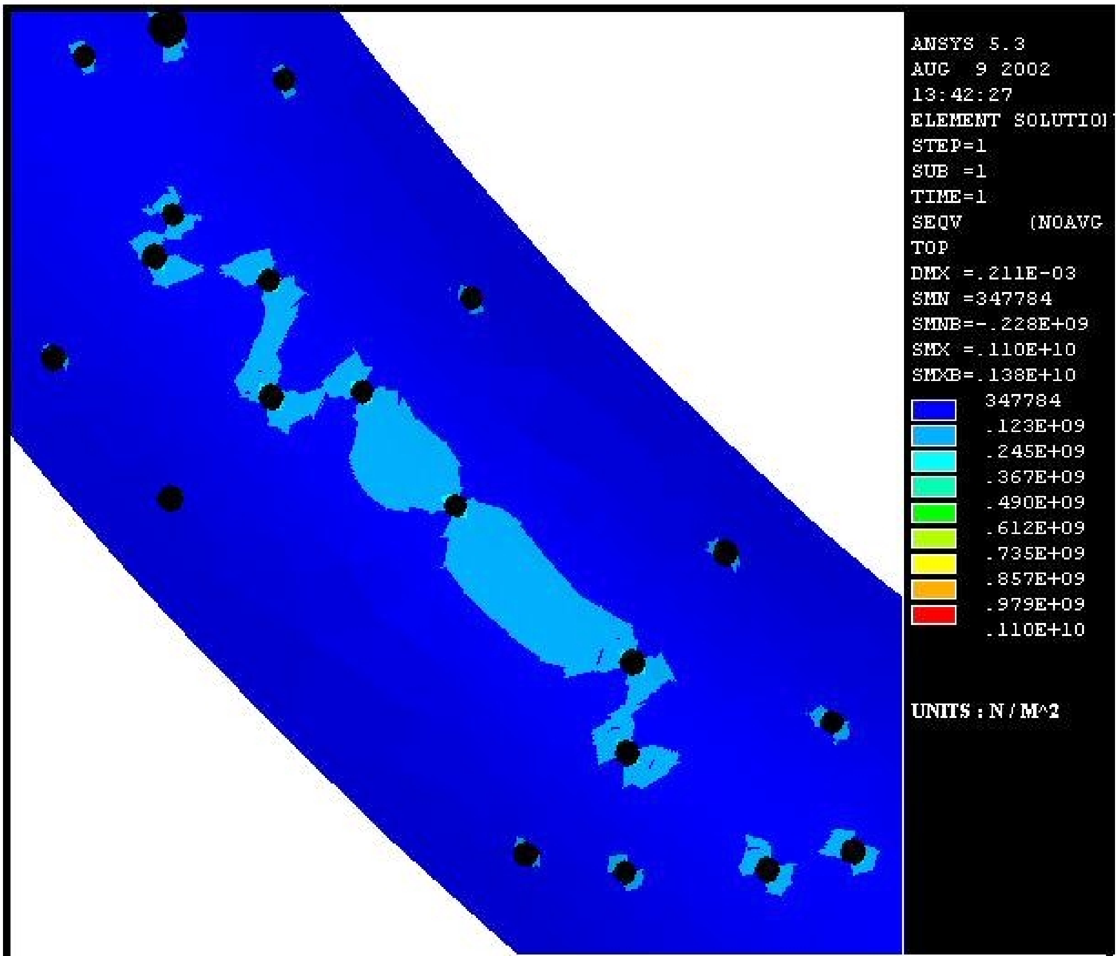

FIG. 8

VON- MISES STRESS PLOTS