************ MSC/NASTRAN SUPERELEMENT ANALYSIS ************
***** OF THE NASA/AMES PRESSURIZED WIND TUNNEL *****

by
DARA SABAHI and TED ROSE

ABSTRACT:

Under contract to NASA, Norman Engineering has performed a complete re-design of the 12 Foot Pressurized Wind Tunnel. This effort was done to fulfill the requirements of the ASME Pressure Vessel code, which required analyses for loadings varying from vacuum to being filled with approximately 55,000,000 pounds of water and pressurized. This effort required the creation of several finite element models and utilization of advanced features in MSC/NASTRAN and CADAM, including superelements and global-local analysis. This paper summarizes the effort involved, with special attention to obstacles encountered and how they were overcome.

INTRODUCTION:

The Twelve Foot Pressurized Wind Tunnel, located at the NASA/AMES Research Center in Moffet Field, California, was originally built in the early 1940s. After forty years of operation, metal fatigue and the discovery of cracks along the weld lines have made this tunnel inoperable. The tunnel has been modernized and re-designed by the Norman Engineering Company. The new design will utilize the existing foundations and internal geometry but will benefit from both a better grade of steel and improved welding techniques.

This wind tunnel has a 12 foot diameter test section and occupies an area approximately the size of a football field. The tunnel has a cylindrical cross-section and the inner diameter varies from 12 feet to 68 feet. Figure 1 shows the layout of the tunnel. The stiffened column lines and the remaining stiffener rings are identified as stations 1 to 34 along the wind flow direction. There are also a number of large access penetrations concentrated in the spherical settling chamber (between stations 1 and 3) and the plenum for the test section (between stations 5 and 6). The tunnel is designed according to ASME Section-8, Division-2 for an operating pressure of 73.5 PSI and hydro-tested up to 101 PSI. The weight of the water required for the hydro-test alone is approximately 55,000,000 pounds. In addition, the tunnel had to be designed for earthquake loadings and the "Loss-of-Blade" accident, where the fan looses half of its blades and imparts an imbalance load of approximately 700,000 pounds on the tunnel at operating speed.

Due to the complexity and size of the structure, use of the finite element method was required. A comprehensive series of static, dynamic and buckling analysis were performed on the tunnel using MSC/NASTRAN (version-64) with the CADAM (release 20.12) processor on a 4381-P02 IBM/VM System. Extensive use of superelements and the global-local method of analysis was required in order to perform the analyses and meet schedule requirements.

In addition, an internal program to perform a check of the ASME code requirements using the results from MSC/NASTRAN was developed. A convenient feature of this program was its ability to write the results in a format which could be read back into MSC/NASTRAN. Contour plots of these results could then be plotted, indicating areas which exceeded the code allowables if there were any.

DISCUSSION:

The finite element analysis of the tunnel involved three stages. The first stage was a low density mesh model referred to as the Coarse-Mesh model. The results from the Coarse-Mesh analysis were used in developing a high density mesh model, referred to as the Fine-Mesh model. The Coarse-Mesh model contained approximately 1927 elements and 7500 degrees of freedom (DOF). All dynamic analyses were performed on the Coarse-Mesh model.

The second stage involved an in-depth study in order to create the Fine-Mesh model. The requirement that an analysis be performed to determine the buckling load factors determined the refinement needed. Several models were created of a typical cylindrical section of the tunnel and buckling analyses were performed on them. The results of these analyses showed that a minimum mesh containing 60 grid points around the circumference was required to obtain adequate comparison with the theoretical buckling loads under pressure. The number of elements required along the length of the tunnel was determined by the decision to maintain an aspect ratio of less than 4:1 for the plate elements.

Once the mesh requirements were determined, a superelement approach was adopted for the Fine-Mesh version to minimize cost and facilitate updating the tunnel model for design improvements. The major advantages of superelements for this problem were the ability to make changes and study their effects without having to perform a complete re-solution of the analysis.

In the third stage of the analysis global-local techniques were used for localized stress analysis. The global-local method uses superelements to allow additional information to be obtained without solving the problem again. The method allows the boundary displacements or forces from the global solution to be applied to a revised version of selected areas(superelements) of the model for purposes of additional local analysis. Examples of this are detailed stress analysis around openings and performing local buckling analysis.

In order to verify that the stresses retrieved from the MSC/NASTRAN analysis conformed to ASME requirements, an ASME code-check program was developed to automate an otherwise arduous task.

STAGE 1: THE COARSE MESH MODEL

The first stage of the analysis or the Coarse-Mesh was a 7500 degree of freedom model shown in figure-2. This model was based on the existing geometry and rough hand calculations from ASME and AISC codes for the initial sizing of the supports and the tunnel shell. The Coarse-Mesh was first used for the initial design of the support columns and a preliminary check of the stiffener and plate sizes. This single level model was then partitioned into a multilevel superelement model and used to test the superelement and data-base partitioning techniques used in analyzing the Fine-Mesh. The Coarse-Mesh model was later updated after initial optimization runs were performed on the Fine-Mesh model. This updated coarse mesh model was then analyzed for seismic (using response spectrum analysis) and other required dynamic loads. The size of this model made it convenient to use to perform dynamic response analysis which required numerous iterations.

The lateral loads calculated using this seismic analysis were used to determine the most critical seismic loading and calculate an equivalent static load which was then applied to the Fine-Mesh model and combined with other required static loading conditions.

STAGE 2: THE FINE MESH MODEL

The Fine-Mesh model was a 220,000 DOF model. The mesh density of this model was set by various parameters such as buckling analysis requirements, local grid point densities, and structural discontinuities. Three major factors contributed to the use of superelement techniques on this model. First was the size of the model, which exceeds the 65,000 DOF limit for a single level model in MSC/NASTRAN (version 64). The second factor was the necessity for continuous updating of the model for design changes and optimization (A superelement analysis allows the analyst to perform an analysis for a model change by analyzing only the sections of the model which have changed). Thirdly a superelement analysis for a large model was deemed to be a more economical approach. Figure-3 shows the complete Fine-Mesh model while figure-4 is a plot of the plenum area of the Fine-Mesh. Recovered stress contour plots from the fine mesh for the hydro- test loading condition are on figures-5 through 7.

PROBLEMS ENCOUNTERED:

Analyzing a model of this size on the 4381 IBM/VM system presented Two major problems:

1. The maximum scratch disk space that could be allocated on a this system is one full disk volume. Individually, the IBM 3375 disk drives used in this analysis have only a fraction of the required disk space for a 220,000 DOF model. The maximum individual superelement successfully run on this system was a 15,000 DOF model with low connectivity. For collector superelements and the residual structure which have very high connectivity, determining a maximum model size was based on trial

and error.

2. The overall disk space available on the system for the storage of the model database was approximately 10% of the required disk space. Therefore the bulk of the database had to be stored on tape.

SOLUTIONS:

The following procedures were used to overcome these limitations:

SUPERELEMENTS

In order to overcome the first limitation, a multilevel superelement analysis was used . Each station of the tunnel was modeled as one superelement spanning from mid-station to mid-station. Station 5 which included the plenum was subdivided into 13 superelements and the plenum supporting skirt was modeled as a separate superelement.

The resulting 47 superelements were referred to as tip superelements and consisted primarily of plate elements. Reducing all of the tip superelements directly into the residual structure would have resulted in the residual structure being too large to run on the system. Therefore, the tip superelements were reduced into 8 new superelements, referred to as collectors. Each collector contained the external grid points of the tip elements and also the grid points containing the stiffener rings. By including the stiffeners in the collectors, it was possible to re-design the stiffeners without performing data recovery on the tip elements.

The twelve superelements of the plenum were originally reduced into a single collector but because the size of this collector was too large for the available disk space, an intermediate collector was used for eight of the tip superelements making this a three level superelement analysis.

Finally, the residual structure contained only the boundaries of the collectors, the column supports, and other support points. This placed all the constraints (SPCs) in the residual structure and allowed for quick and efficient iterations on the support system.

PARTITIONED DATA BASES

To alleviate the disk storage problem, database partitioning was used. For each superelement and collector three separate databases were used.

The first database contained the geometry, internal loading, and stiffness properties of the superelement. This database could be written onto tape and removed from the system as soon as the superelement had been reduced. It should be copied back onto the system when required for data recovery.

The second database contained the external matrices or downstream loading and stiffness. This database tended to be small and many of these could be on the system at a time.

The third database was used to store recovered data and results for post-processing.

The residual structure being the final level, only had an internal and a data-recovery database.

Performing the reduction involved assigning a read-only input database and two output databases to each superelement. For the tip superelements the input database consisted of the bulkdata only while the collectors had the bulkdata and the downstream databases of their tip elements assigned to them.

During data-recovery each collector and tip element had its internal database, along with the internal database and the data-recovery database of it's downstream superelements attached. The data-recovery step then yielded a new database (the third one) for each superelement which contained the post-processing information.

This technique effectively eliminated the disk space problem. The model was partitioned into small enough superelements so that the scratch disk space was adequate for running each of the superelements individually. Partitioning the databases also made it possible to keep the bulk of the databases on tape.

The collector and residual runs required access to the downstream data bases of the superelements which reduced into them. The downstream databases of the superelements were relatively small compared to the internal databases. Therefore, by backing up the large internal databases on tape and keeping their downstream databases available on disk, it was possible to reduce the complete model efficiently without overflowing the available disk space.

There were other advantages to using the above approach. Due to the size of the model and the number of changes made during the course of this analysis there were many opportunities to accidentally damage the database. Therefore, by keeping the internal data, external data, and recovered data of each superelement separately, damage to any of the databases could easily be corrected by either re-loading the database from tape or in the case of damage to any of the databases stored on tape only one superelement would need to be rerun . Another advantage of partitioning databases was in global-local analysis. Database partitioning allowed for easily utilizing the different global-local analysis methods.

STAGE 3: GLOBAL-LOCAL ANALYSIS

Due to the overall size and complexity of the structure, many structural discontinuities and transition details were not included in the Fine-Mesh model. Global-Local analysis was used to examine these areas of the tunnel which could not be properly modeled in the Fine-Mesh model (These details had only been approximately modeled in the Fine-Mesh model). This method allowed the use of a much finer mesh to examine these areas while taking into account the global forces and displacements imposed on these superelements by the rest of the model. During the course of this analysis different Global-Local techniques were used for a number of different applications.

The first method used was the displacement compatibility method. Since the databases of the Superelements had been partitioned, this method was extremely easy to apply. The local (refined) model of any superelement in question was generated and reduced with the same superelement number and tree as the original superelement. Afterwards data recovery was performed on this model by inserting its internal data base into a standard data-recovery run rather than that of the original superelement. This way the boundary (global) displacements were directly applied to the local model during data recovery.

For two cases such as a man access in the plenum section (figure-6) and a shaft penetration at station-16 (figure-7), the Local-Meshes were incorporated into the Fine-Mesh bulkdata and reduced all the way to the residual structure prior to data-recovery.

The stiffener rings at the supports were originally modeled using beam elements with offset neutral axes in the fine mesh model. The transition from the stiffeners to the support columns could not be modeled properly in the Fine-Mesh, which caused areas of stress concentration at the column attachment locations.

A typical support stiffener ring was chosen for a local study and was later used to predict the accuracy of all the remaining rings by comparison. The mesh for this model used QUAD8 elements with a mesh which was four times finer than the Fine-Mesh model in the regions which had high stress variation in the shell. The stiffener was also modeled with QUAD8 elements with an even finer mesh than the shell. The mesh transitions in this model were accomplished by using short interpolation elements (RSPLINEs) in complete rings around the shell. Figure-8 shows a plot of the high mesh density region of this model.

This model could not be analyzed using the displacement compatibility method used in the other smaller local models since the model was too large to be reduced with the available scratch disk space. Therefore the model had to be broken down into two superelements with the shell section in one superelement and the stiffener in the other. The residual structure of this model constituted of the ring of grids attaching the stiffener to the shell. The force method was used to retrieve the global forces acting on the model from the Fine-Mesh.

In the force method, the boundary forces on a superelement are calculated in the same manner as cutting a free-body diagram for that superelement. These boundary forces are then applied to the boundary of a refined (local) model and a detailed analysis is performed.

Using the force method on the stiffener model represented a constraint problem. This section of the tunnel was only constrained vertically at the columns and was allowed to move laterally and expand. Due to a slight imbalance between internal loads on the detailed model and the external forces from the fine mesh version on the boundary, using solution 61 (Superelement Statics) with soft spring elements for constraints yielded inaccurate results. The force imbalance was again caused by the incompatibility of the boundary elements in the refined model. Solution 91 (Superelement inertia relief) also posed a problem, since the residual structure did not have any mass for the inertia relief solution. This problem was overcome by placing small concentrated masses on the grid points of the residual structure and then running solution 91.

The results of the analysis for this local model are shown for the hydro-test load in figure-9.

The force method was used to perform buckling runs on large sections of the tunnel (A complete buckling analysis of the tunnel was not within the budget of this analysis), since the calendar time required to perform these analyses on the IBM/VM system was prohibitive. To satisfy the requirements for buckling analysis for the tunnel under full vacuum loading, a Cray computer at the AMES/NASA center was used for the buckling analysis of the critical sections of the tunnel. These sections included the settling chamber sphere and the cones attached to it (all of which were contained in one collector), the test chamber plenum sphere (which was again contained in one collector), and finally the south leg of the tunnel from stations 21 through 28 (which was contained in two collectors).

The procedure used to perform the buckling analyses for the critical sections of the tunnel on the Cray required creating isolated models of the critical sections to be run on the Cray.

The first step required isolating the bulkdata for the area of the Fine-Mesh model that required the buckling analysis. The superelement designations were then removed from the grid points, which turned the isolated section into a residual structure only model. The Cray computer could easily handle the largest buckling problem, which was on the order of 45,000 DOF. The force method was then used to retrieve the global forces exerted by the structure on the boundaries of these models. For the south leg buckling model the DMIG output deck of the force method for both of the collectors were edited and combined into one input DMIG deck.

Since the superelement boundaries were usually chosen in the unstiffened area between the stiffeners, the boundaries of these models required stiffening to account for the stiffness of the attached structure. If this weren't done, the analysis would retrieve inaccurate and small eigenvalues for these boundaries. The boundaries were usually stiffened by using elements with relatively high moments of inertia. These boundaries were located far from regions of interest for critical buckling.

ASME CODE CHECKING

Post-processing the results also presented a problem. ASME section-8, division-2 does not have a simple, over-the-board allowable stress category. The basic allowable stress can be increased by 50% for bending stresses and local membrane stress concentrations and 200% for secondary stresses such as thermal stresses and stresses through the thickness of the shell. In addition, for wind and seismic loading a factor of 1.2 is applied to the allowable stresses.

The ASME Code is based on the Maximum Shear Stress Theory. The stresses that signify a failure criteria as defined by the

ASME Code are not readily available from MSC/NASTRAN output. An ASME check program was developed by Mr. Peter Hill of Norman Engineering Co. to evaluate Stress Intensities as defined by ASME.

As part of this program a set of DMAP Alters were applied to the MSC/NASTRAN data-recovery runs using OUTPUT2 requests to produce a "FORTRAN readable file" containing the EQEXINS and the EGPSTR data blocks. The EQEXINS data block contains a map showing the relation between the external and internal gridpoint IDs, while the EGPSTR data block contains the gridpoint stresses derived from element stresses as defined by a SURFACE or VOLUME output request. The EGPSTR data block was then modified to contain the Stress Intensity values as defined by the ASME code with the Maximum Shear values. The ratios of actual to allowable stress were represented in the Shear-XY column.

In addition to the above, the "Triaxial Stress Test" (the sum of the three principal stresses should be less than or equal to four times the allowable stress) was performed at every grid point.

After the EGPSTR data block had been updated by the ASME Codecheck Program, a series of DMAP alters were used to place the modified EGPSTR data block from the FORTRAN readable file and the OUGV1 data block (containing deformations from the Data-Recovery database) along with all the other datablocks of the corresponding superelement required for GRASP/DBC database conversion for post-processing in a single database.

The stress contour plots of the Fine-Mesh (Figures-5 through 12) demonstrate the results of this post-processing procedure. Please note that this procedure also allowed for plotting contours of stress ratios calculated by the ASME Codecheck Program.

SUMMARY AND CONCLUSIONS:

The analysis of the Twelve Foot Pressurized Wind Tunnel was performed on an IBM-4381 Computer. This system does not have the speed and the capacity of a supercomputer such as Cray. Approximately 3000 CPU seconds (two hours real time during normal operations) were needed just to read and store the 180,000 lines of bulkdata for the Fine-Mesh model. The use of superelements, database partitioning and global-local methods enabled the IBM computer to efficiently perform the bulk of the required finite element analyses. Furthermore this analysis did not require a team of engineers and analysts. One engineer was able to model, analyze, and document the 220,000 DOF model used for the Fine-Mesh analysis and all the subsequent local models using advanced superelement techniques in. All the static, dynamic, buckling, and local analysis required for the tunnel were completed within 1-1/2 years during which the design of the tunnel proceeded in parallel to the finite element analysis.

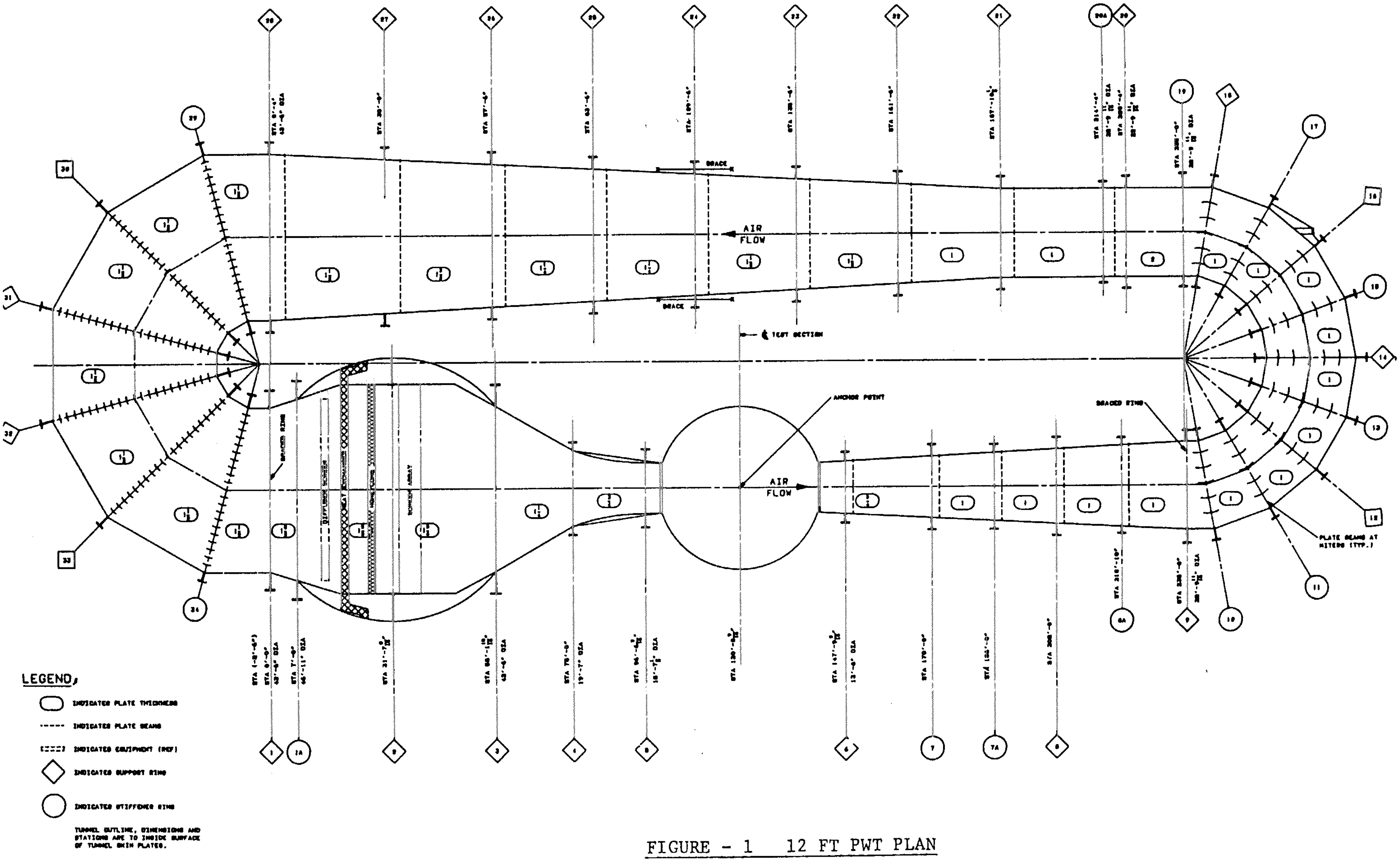


FIGURE - 1 12 FT PWT PLAN

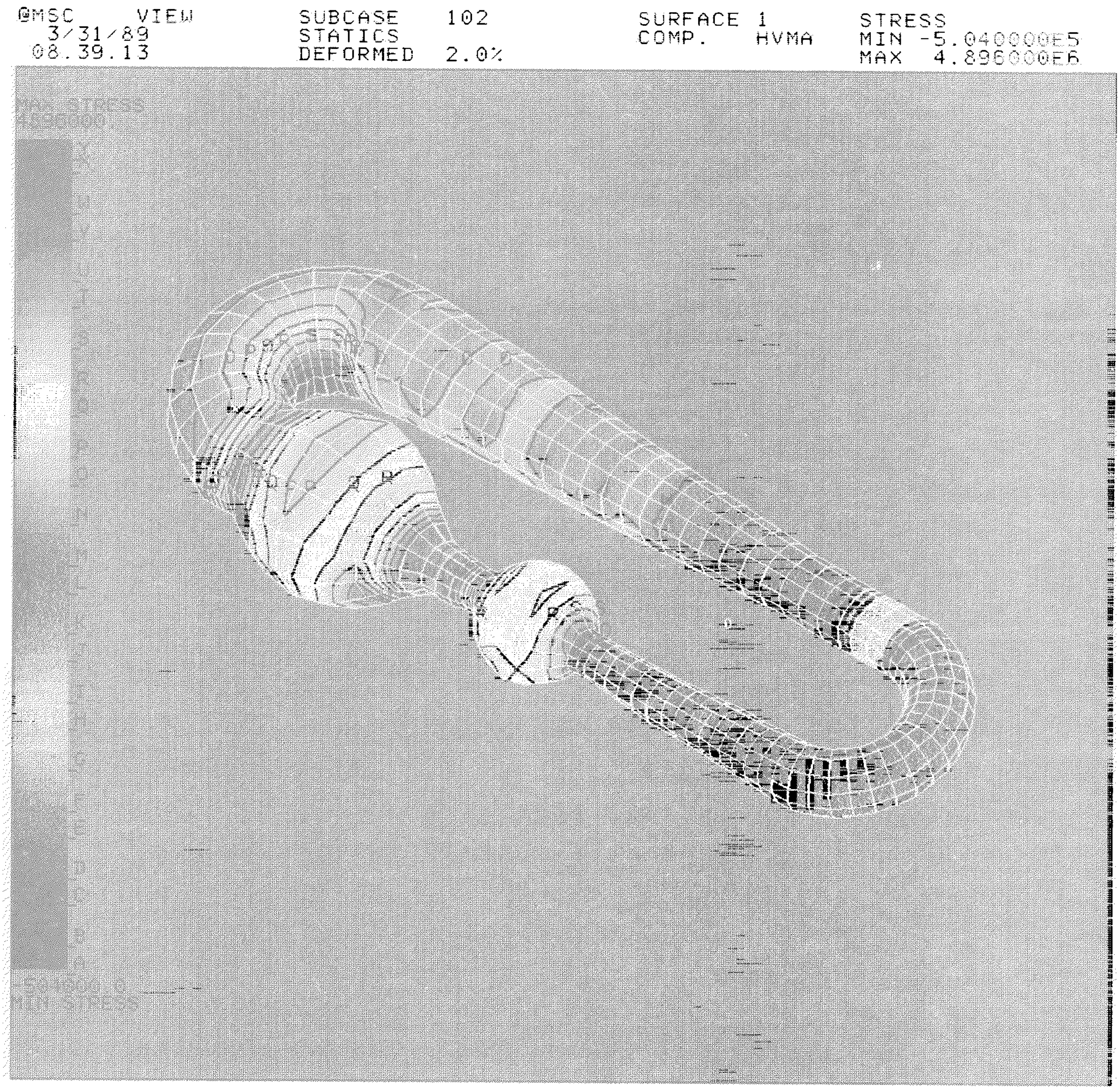


FIGURE - 2 COARSE-MESH STRESS PLOT FOR HYDROTEST LOADS

HENCKY VON MISES CONTOURS UNITS - PSF

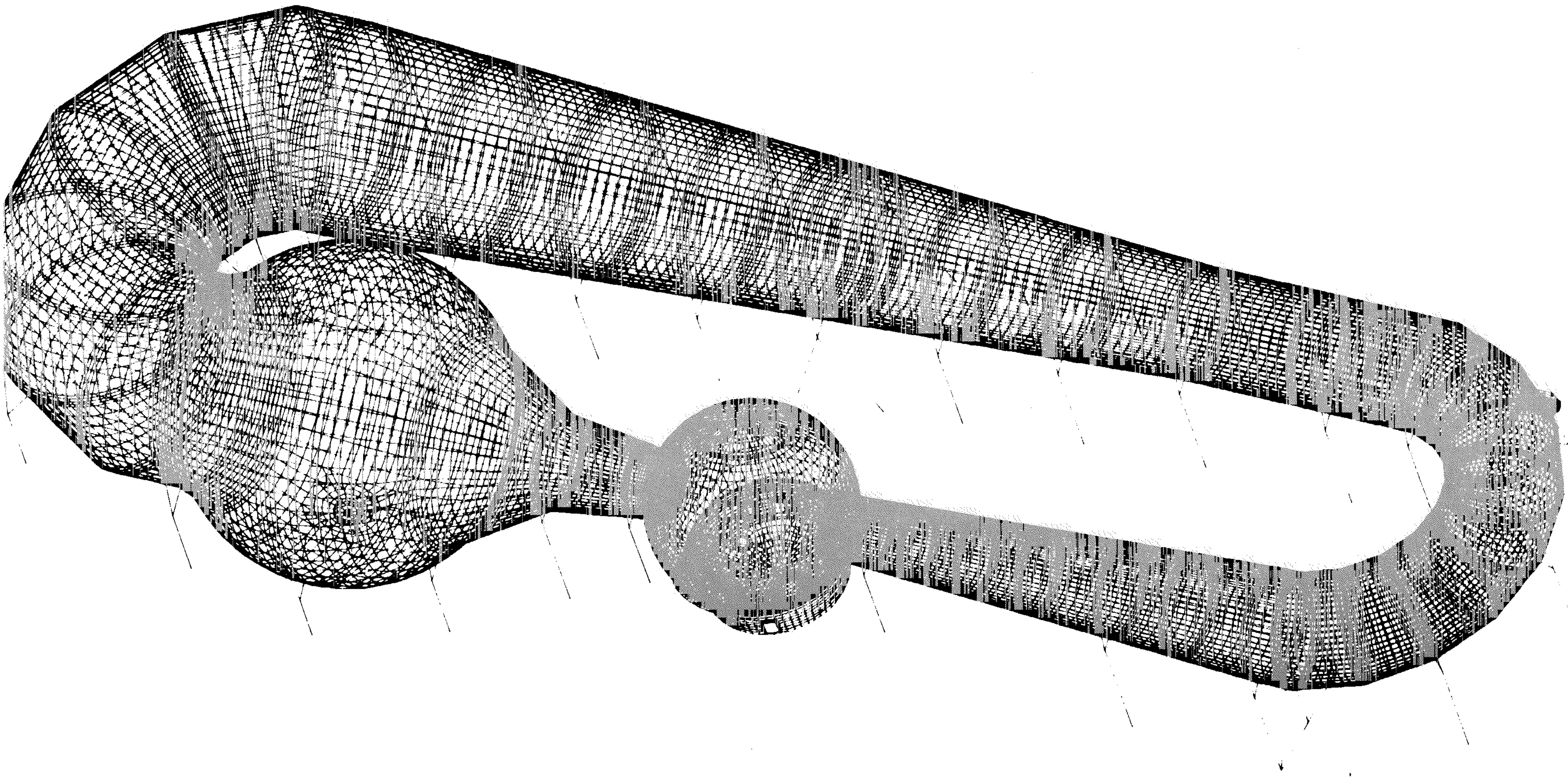


FIGURE - 3 FINE-MESH MODEL

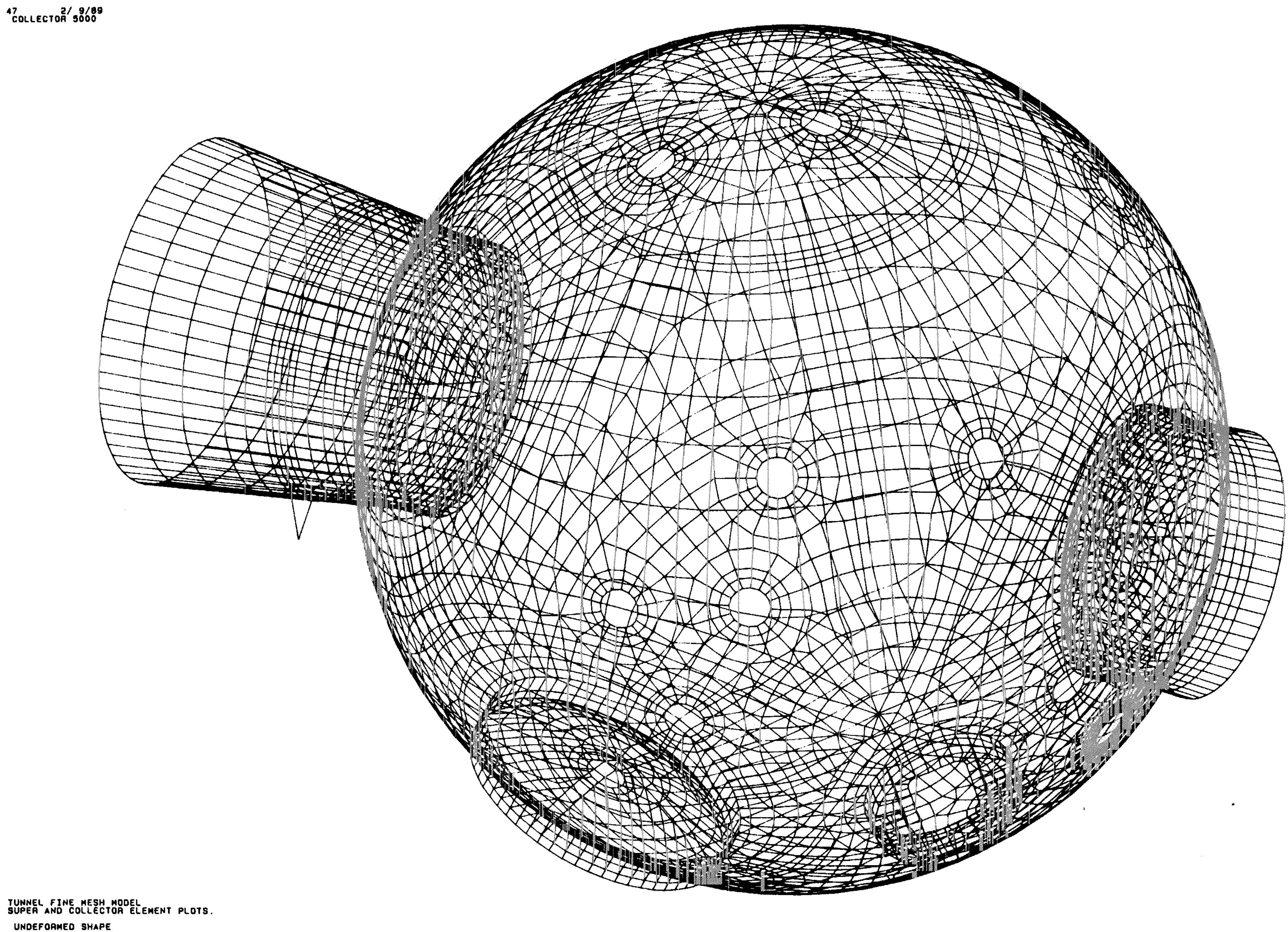


FIGURE - 4 FINE-MESH MODEL OF PLENUM

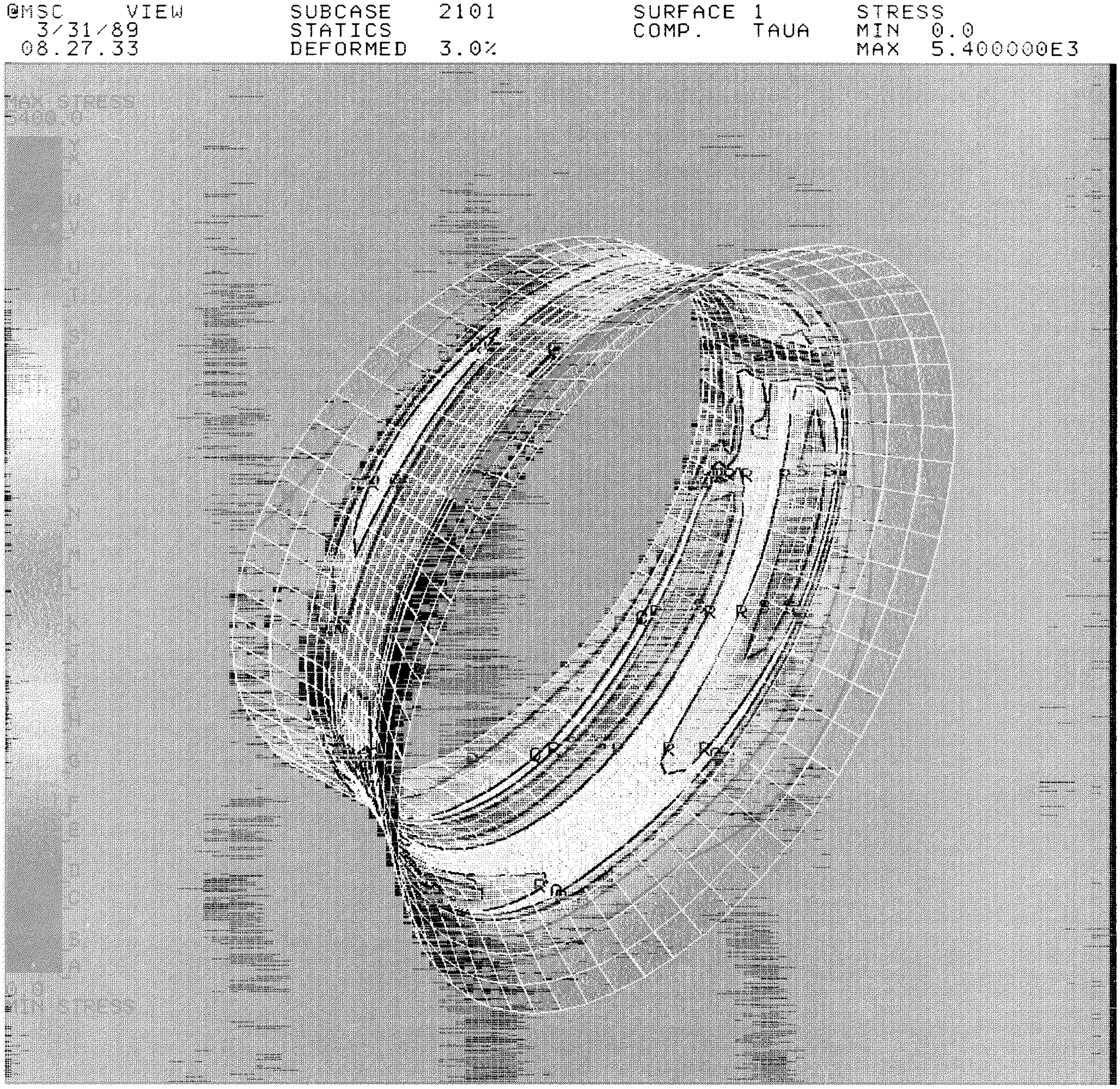


FIGURE - 5 STATION 1&1A FINE-MESH SUPERELEMENT STRESS PLOT FOR HYDROTEST LOADING

STRESS INTENSITY CONTOURS UNITS - KSF

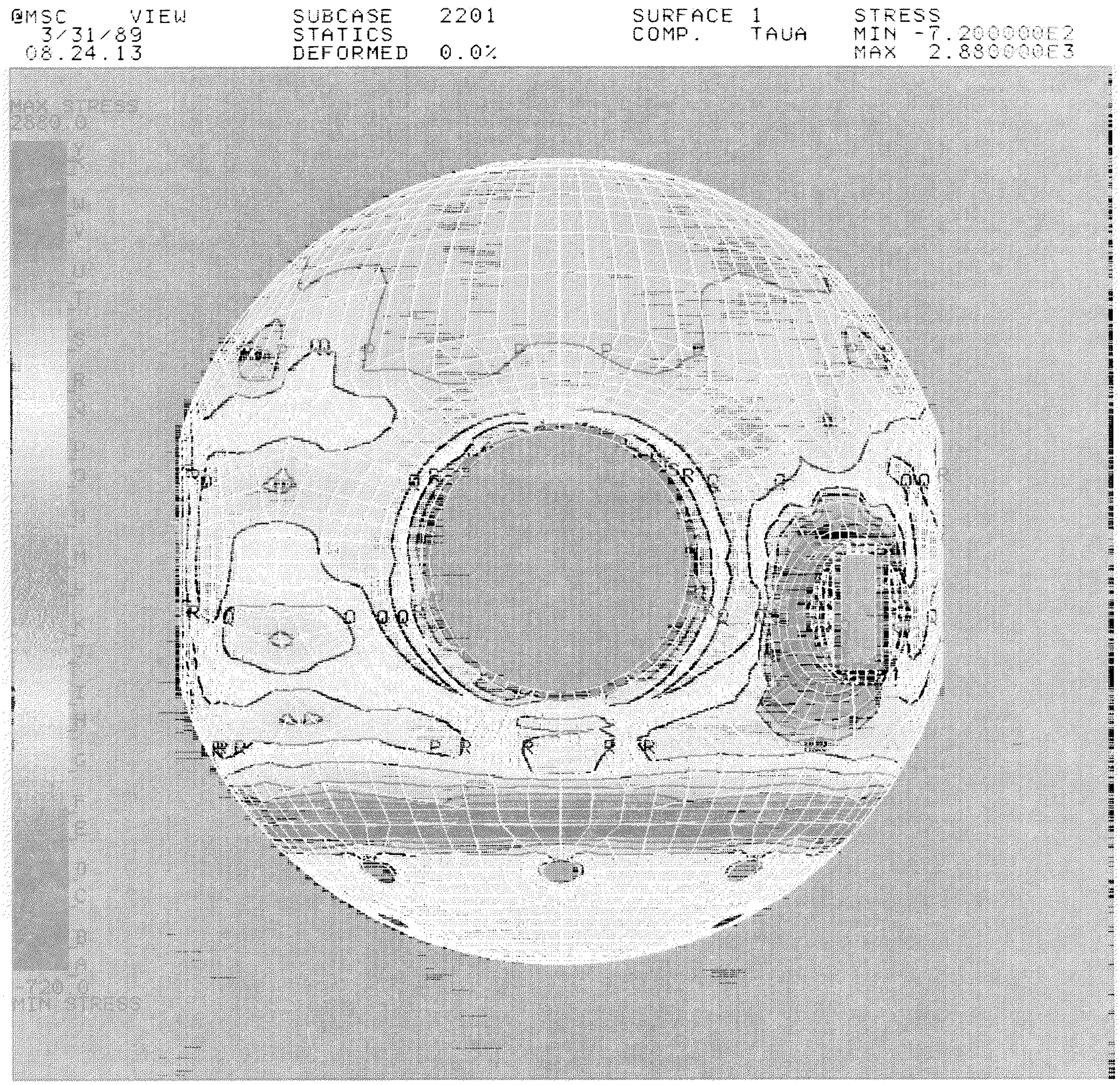


FIGURE - 6 PLENUM FINE-MESH STRESS PLOT FOR HYDROTEST LOADING

STRESS INTENSITY CONTOUR UNITS - KSF

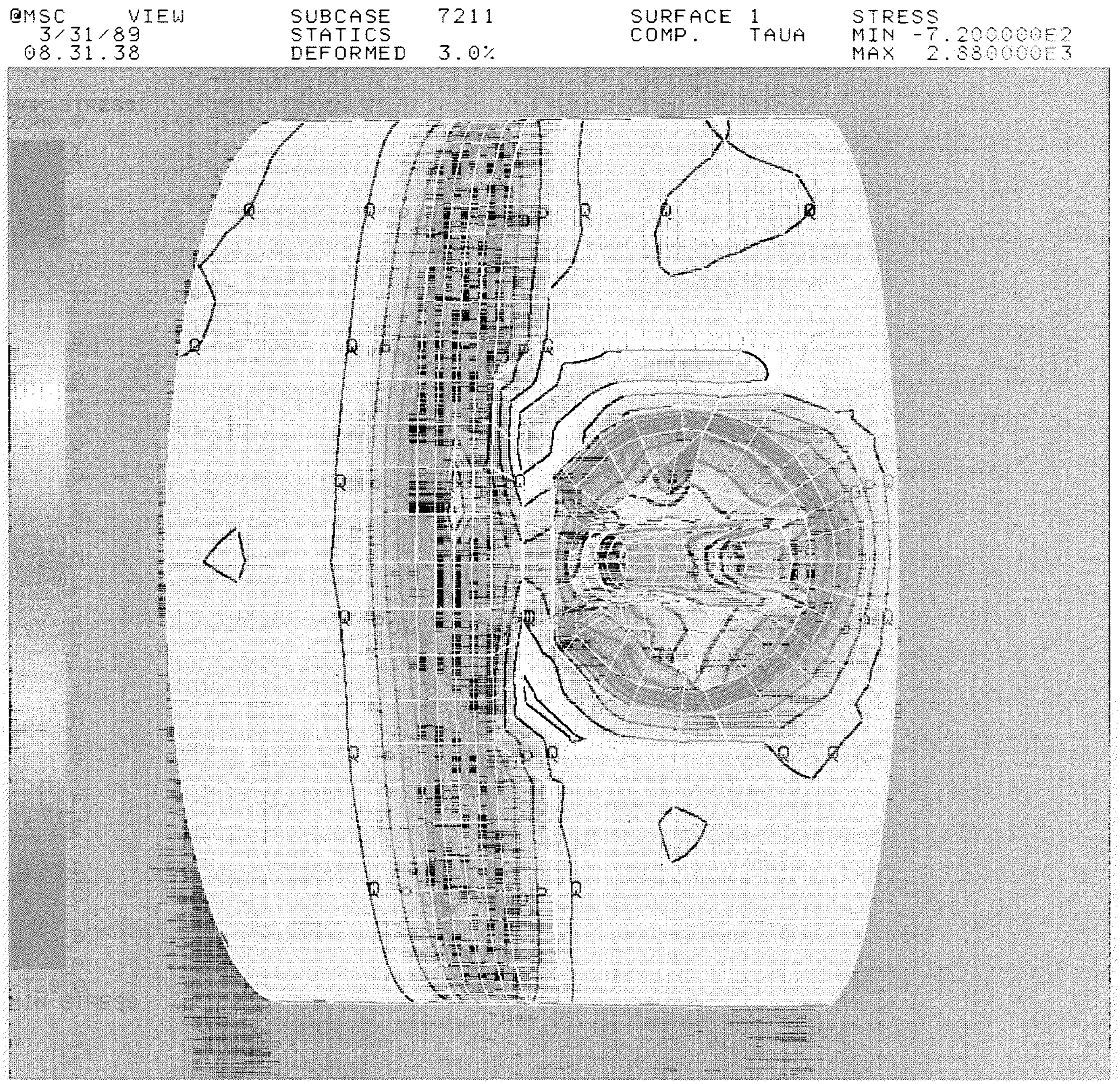
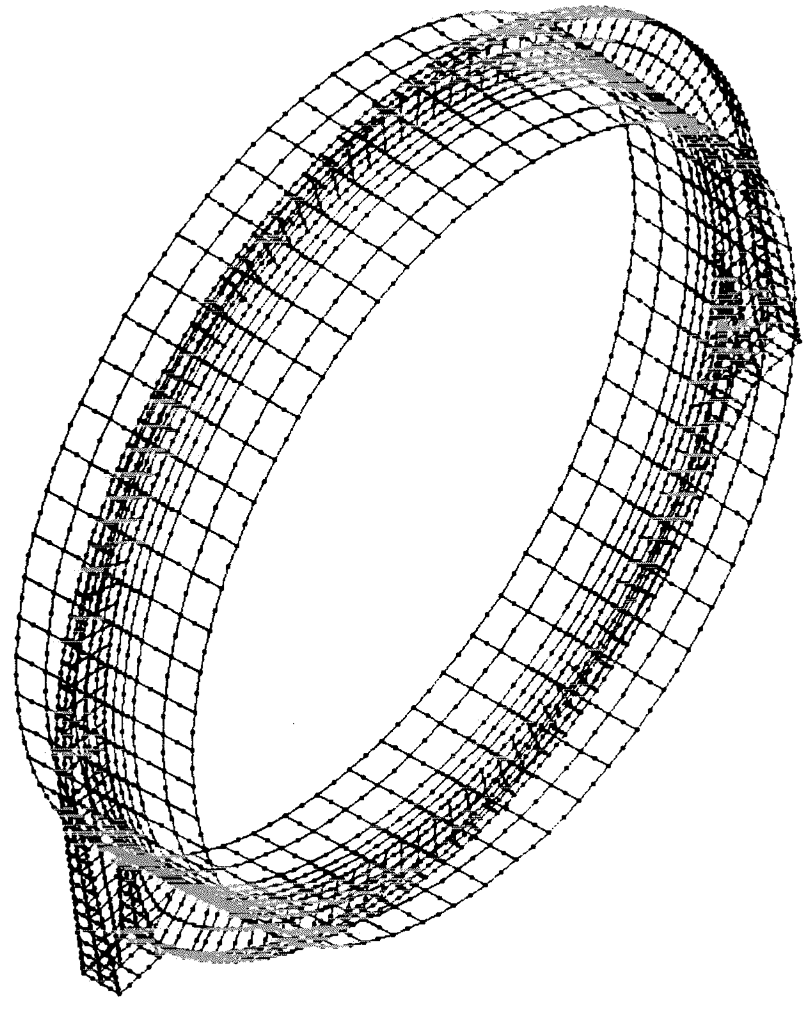


FIGURE - 7 STATION 16 (SHAFT PENETRATION) LOCAL-MESH SUPERELEMENT STRESS PLOT

FOR DESIGN LOADING

STRESS INTENSITY CONTOURS UNITS - KSF



LOCAL MODEL FOR SHELL TO STIFFENER ATTACHMENT REGION, STIFFENER, AND COLUMN TO STIFFENER TRANSITION FOR STATION 23.

FIGURE - 8

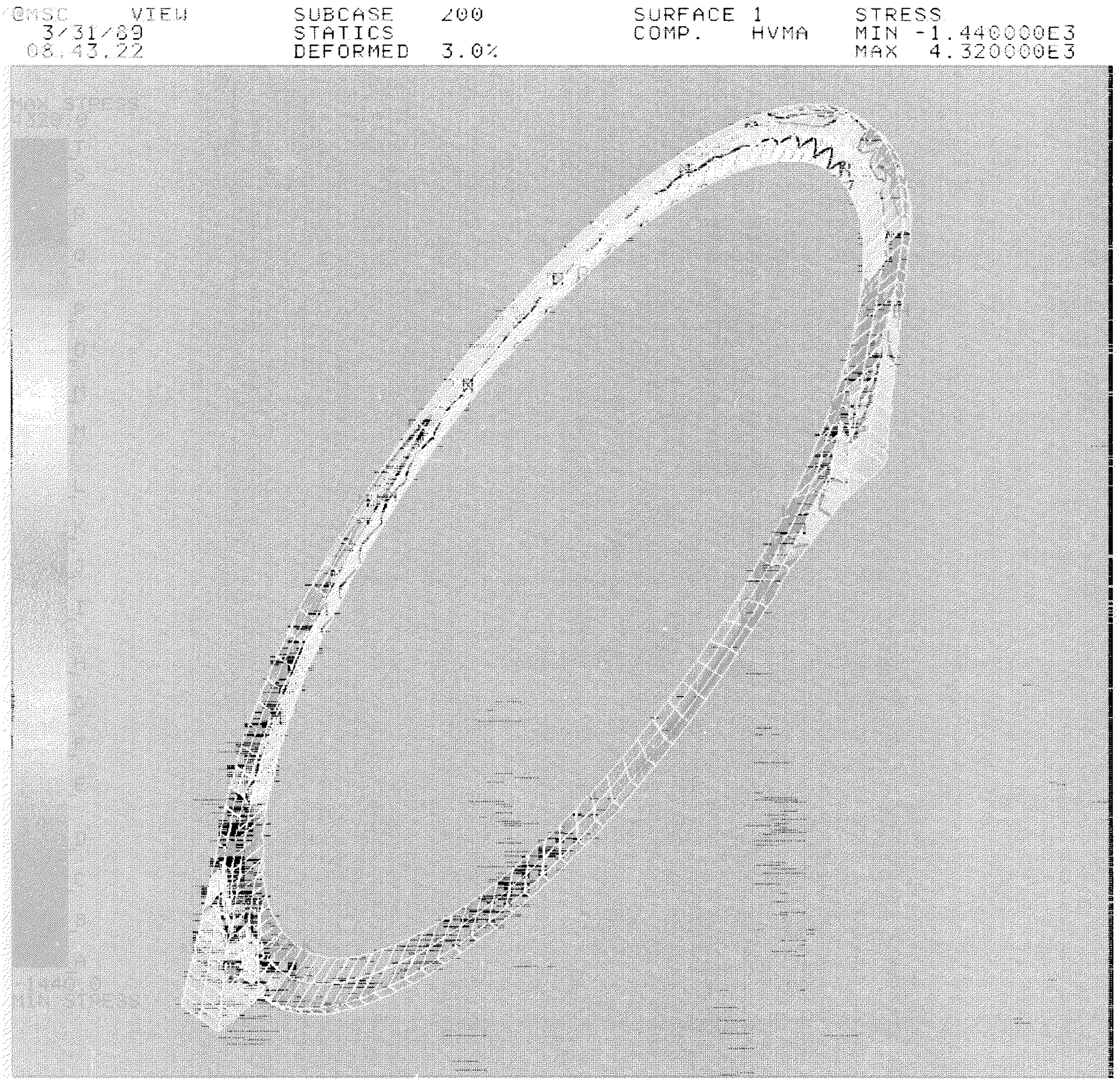


FIGURE - 9 STATION 23 STIFFENER LOCAL-MESH STRESS PLOT FOR HYDROTEST LOADING

HENCKY VON MISES CONTOURS UNITS - KSF