UNDERGRADUATE REPORT

SYSTEMS RESEARCH CENTER

Supported by the National Science Foundation Engineering Research Center Program (NSFD CD 8803012), the University of Maryland, Harvard University, and Industry

A Mathematical Modeling of a Machining Operation System Using Finite Element Method

by D. St. Rose
Advisor: G. Zhang

U.G. 91-5

TITLE

A MATHEMATICAL MODELING OF A MACHINING OPERATION SYSTEM USING FINITE ELEMENT METHOD

BY

DESMOND ST.ROSE

FACULTY ADVISOR: DR GUANGMING ZHANG

ABSTRACT

A mathematical modeling of a Machining Operation Systems to evaluate stresses generated by the cutting force during machining, was modeled using finite element method. The geometric model was assembled using patran commands. The model was subdivided into elements of quadrilateral surface, where each element has four nodes. Each element is meshed independently and assembled into an elemental stiffness matrix. Each stiffness matrix is then assembled into a global matrix to determine the stress distribution in the model due to the cutting force. For this model, there were 272 elements and 305 nodes. Equal distribution of cutting forces of 1000 Newtons was applied to the model at the area of the cutting tool in the negative Y direction. After applying the forces, the model was analyzed using patran finite element method software P/FEA. The highest stress concentration is at the area where the force is applied. The highest stress value is $4.12X10^{3}$ N/m^{2} from the Von Mises stress plot at the area where the cutting tool make contact, and from the displacement stress plot the highest stress value is $6.42X10^{-8} N/m^{2}$.

TABLE OF CONTENTS

List of Illustrations

INTRODUCTION

Finite element method is used for solving structural problems occurring in engineering and science by numerical solution. A mathematical geometric model of a machining operation systems was design on an electronic digital computer using Patran finite element method. Patran is an open ended, general purpose 3-d mechanical computer-aided engineering (MACE) software system that uses interactive graphics to link engineering design through finite element method. This software have the ability to develop and test products, analysis and give results of the stress distribution due to a cutting force[1].

This geometric model design is a finite element model of a transducer. A transducer is called a dynamometer typically used in the study of metal cutting, and to study the machining process, which is the removal of material from a workpiece.

Designing a dynamometer requires three basic requirements: Rigidity, Sensitivity, and Accuracy. The dynamometer should be sensitive to small changes in the cutting force, yet rigid enough to prevent the occurrence of chatter vibration, and at the same time have to be rigid to ensure that the cutting force does not deflect the cutting tool. By analyzing the intensity of the stress within the geometric model, the cutting force on the cutting tool can be controlled.

Finite element method analyze the model by creating nodes and quadrilateral (quad) elements within the boundaries of the mathematical model. The nodes are the four points of an element , see figure 1, page 2. Figure 2 on page 3 show the quad elements . The quads are meshed so there can be total connectivity within the model, see figure 3, page 6. The material properties of

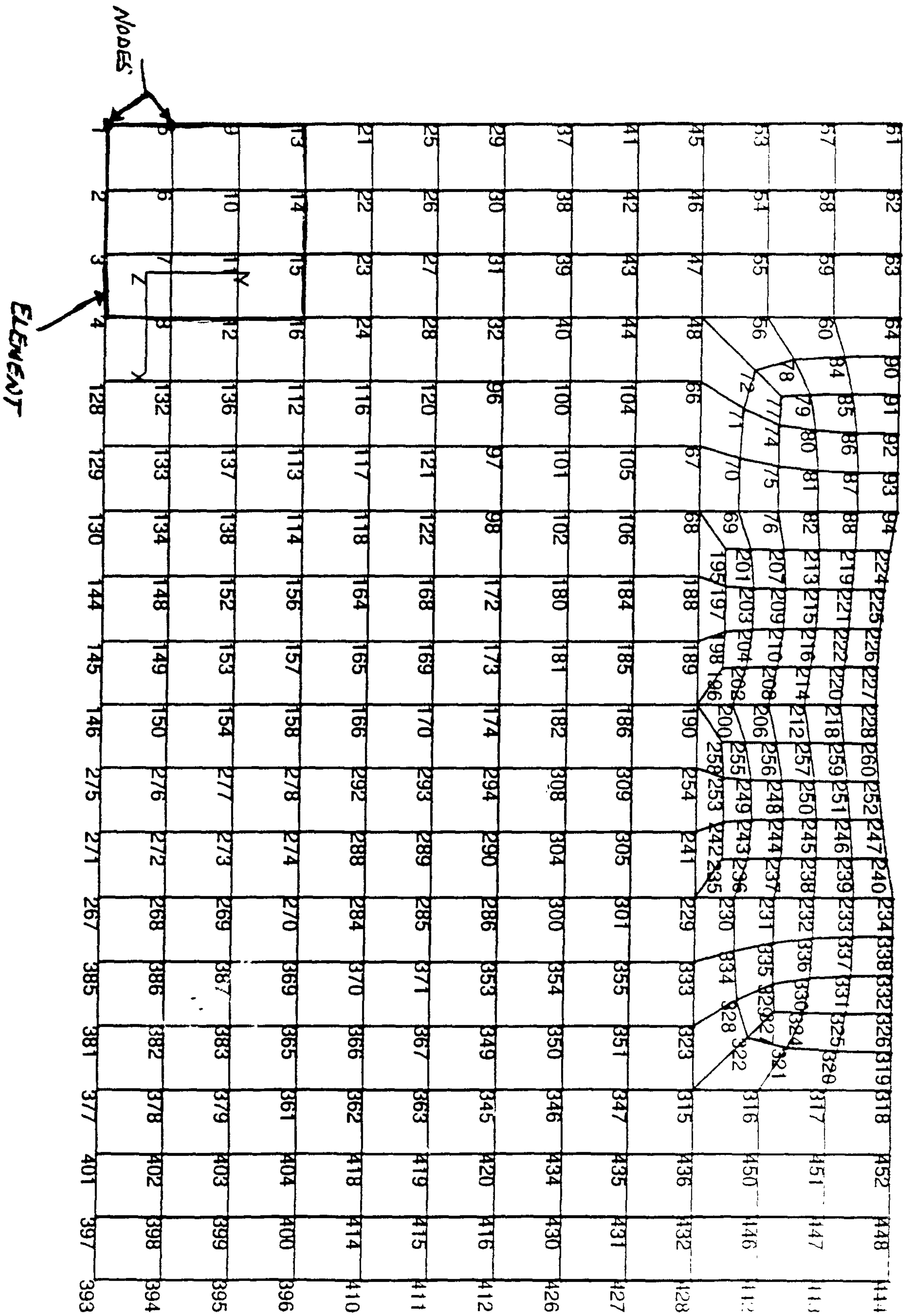

Figure 1. Nodes

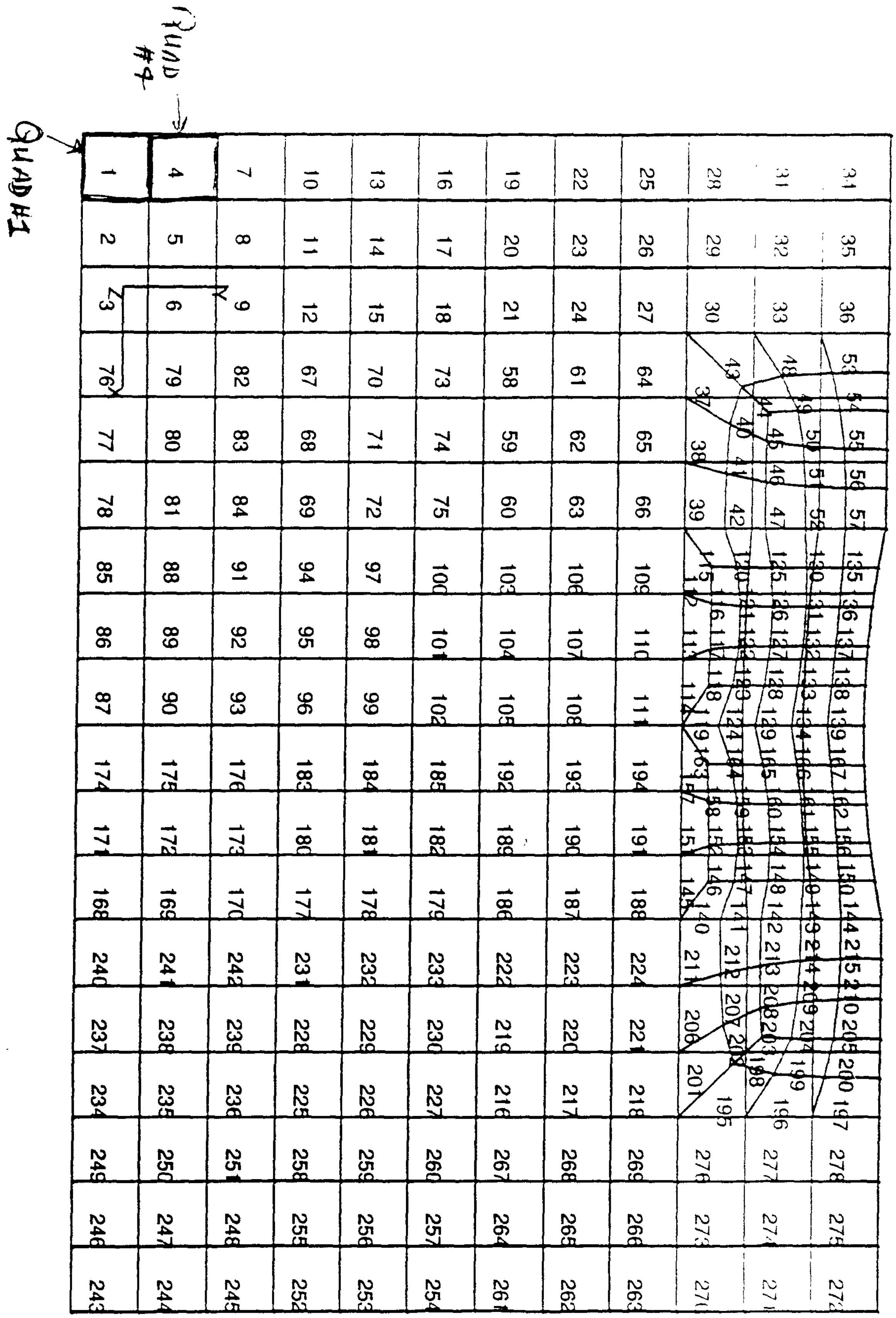

Figure 2. Quads

the model are steel with a modulus of elasticity, E, of $200X10^9$ N/m^2 and the Poisson's ratio, υ, of 0.3. After inputting this information into the finite element method analysis phase with the applied force, the model is ready to be post-processing. Post-processing assembles all the element matrices into one global matrix so the stresses within the model can be calculated . The highest stress distribution in the model can be seen from the intensity of the color within the model, see figure 8, page 13.

With the value of the stress, the cutting force can be determined during the machining process. There are other methods to detect stresses during machining such as strain gages sensing elements.

PROCEDURE AND METHOD

The model was assembled on the computer using patran commands. There are three phases of the finite element method to analyze the model, the geometric, analysis and the post-processing. In the geometric phase, the dimension of the model was input using the grid command. The grid command defines points in space for the construction of the geometric model[1]. 36 grid points were input to construct the model, see figure 4, page 7. The line commands were used to connect the grid points, so the model can be patched. A patched command creates a parametric cubic geometry surface of the model between two lines into quadrilateral (quad) surfaces[1]. There were 24 patches each consists of nine quad elements, see figure 4. The next phase is the analysis phase where the model was then meshed so there can be total connectivity between each nodes of separate elements. Meshing takes place at the boundaries of the patches and this can be seen as the bold numbers, see figure 3. After meshing, there were 305 nodes and 272 elements. Displacement forces were placed on the bottom surface of the model along patches 1, 8, 9,13, 20 and 21 to prevent the model from moving when the cutting forces are applied. Distributed forces of 1000 Newtons was applied to the model using the force command. The distributed forces represent the forces that the cutting tool exert on the workpiece during the machining process, see figure 5, page 10. The material properties of the model were input using the PMAT command. The finite element method assigns the material properties to each element through the PFEG command.

When the above design procedures are completed, the finite element method moves into its third phase, the post-processing phase. The post-processing phase assembles all the stiffness matrices into a global matrix. The stiffness matrix is the matrix assembled for each quad elements with X and Y values at each node of an element. Since the model is two dimensional, there are two degrees of freedom, hence two deflections at each nodal point , X and Y. For this model, there were 610 degrees of freedom with 610 equations and 272 stiffness matrices. The global matrix is used to calculate the deflections at each nodes, so the global matrix was assembled into a 610 X 610 matrix. The calculated deflections at each nodes are used to determine the stresses in the model due to the distributed forces of the cutting tool when applied to the workpiece during the machining process.

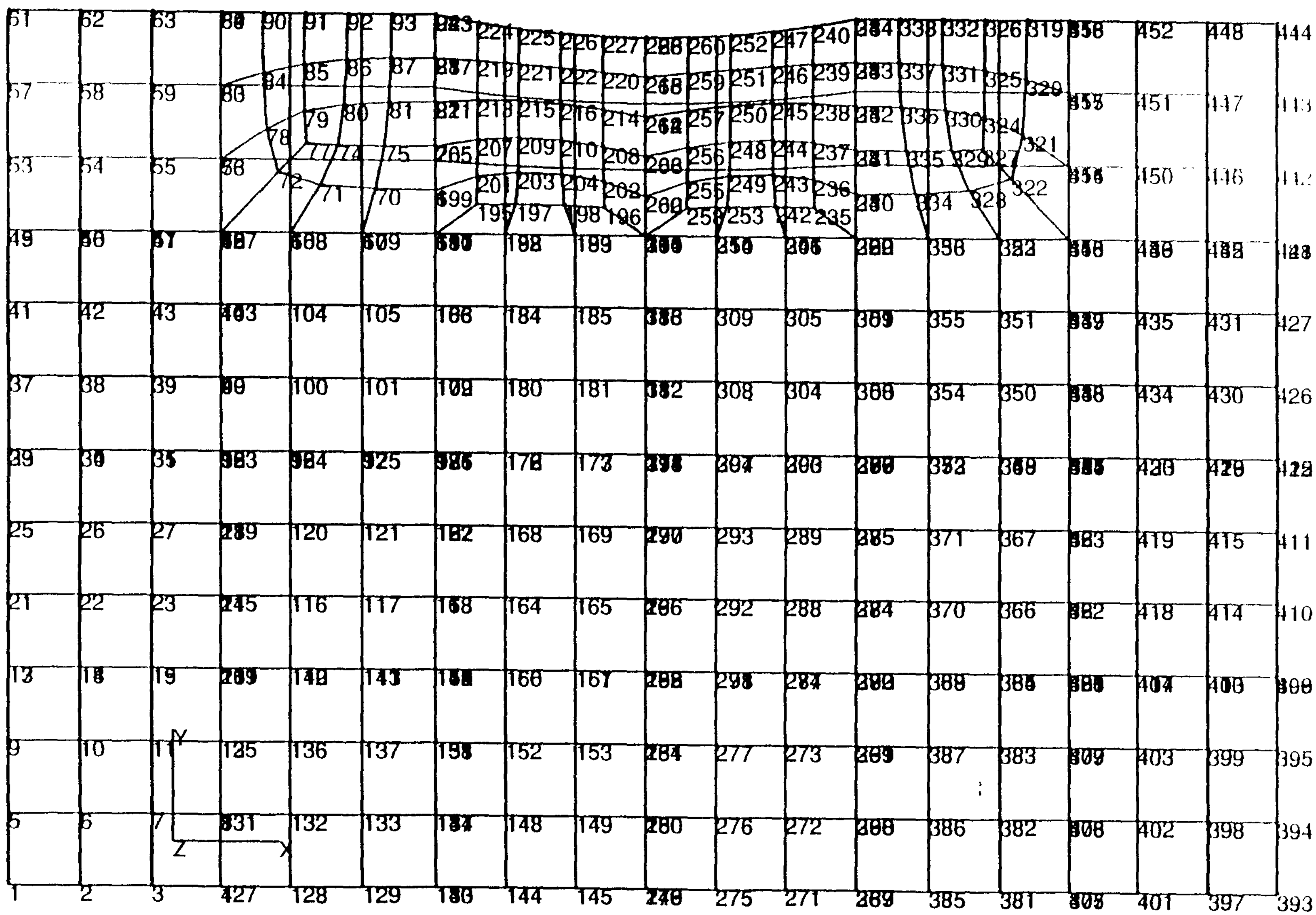

Figure 3. Nodes Meshing

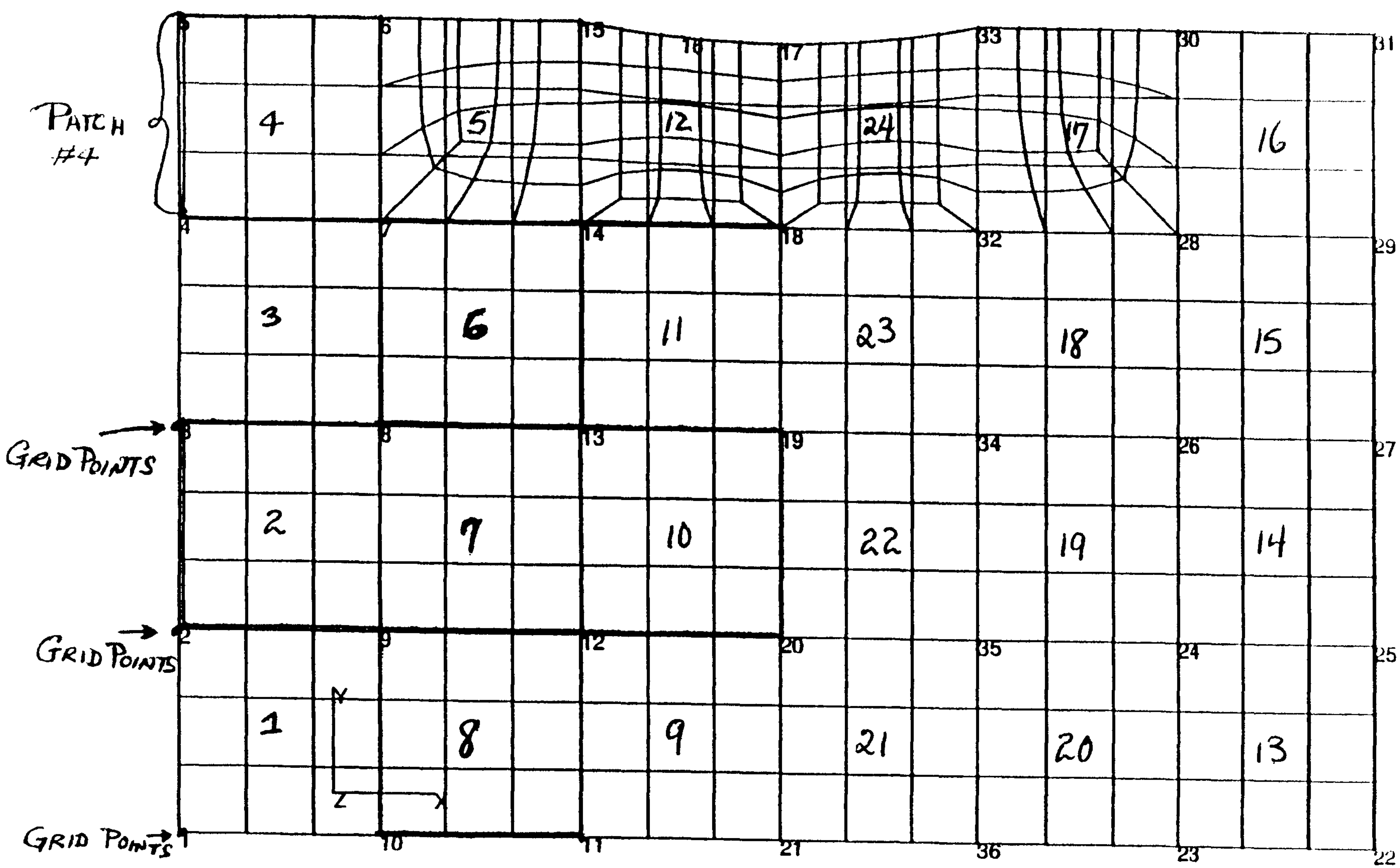

Figure 4. Grids Points and Patches

RESULTS

The forces, which have to be normal to the plane, were applied to the model on patch number 12 at nodes 74, 156, 157, 158, 159, 160 and patch number 24 at nodes 234, 287, 293, 298, 305. This give a total of 11 points loads. Figure 5 show the distributed forces and the displacement forces. The nodal thickness was set at 1.0 inches. There was 610 equations to calculate 610 defections. The deflections consists of movement in both the X and Y direction. The nodes at the displacements forces are zero because the nodes was constraint from deflecting. Looking at figure 6, page 11, the deformed and undeformed shape of the model is seen in window number 1, where it can be seen how much the workpiece deformed during the machining process. The stress plot were generated using total displacement stress result, and the highest stress value is at the point of contact of the cutting tool close to the cutting edge. The highest stress value is $6.42X10^{-8}$ N/m^2 and this can be seen in window 2. The largest unconstructed diagonal occur at equation 352 which is $5.878X10^{11}$ N/m^2, and the smallest unconstructed diagonal occur at equation 25 which is $9.157X10^{10}$ N/m^2. The highest stresses are seen around the area where the cutting tool is applied. By increasing or decreasing the force of the cutting tool, the intensity of the stresses in the model increased or decreased respectively. At node 5, the deflection in the X direction is $-3.75X10^{-9}$ N/m^2 and in the Y direction $-4.018X10^{-9}$ N/m^2, at node 305 the displacement in the X direction is $-4.89X10^{-10}$ N/m^2 and in the Y direction $-6.389X10^{-8}$ N/m^2. The sum of forces in the X direction is $2.913X10^{-13}$ N/m^2 and in the Y direction is $1.095X10^{-4}$ N/m^2. Stress distribution plots are seen in figure 7, page 12, figure 8, page 13 and figure 9, page 14. In figure 7, a contour stress plot result is seen in window 2. The contour stress plot is similar to the stress distribution seen in the Mean Stress plot result in figure 9. Figure 8 show a Von Mises stress plot result.

DISCUSSION

In this project, a finite element model for representation of a Machining Operation Systems was design. The main objective of this project was to gain experience through specific transducer design. The finite element model formulates the entire machining operation system as an assemblage of three major requirements; rigidity, sensitivity, and accuracy. By analyzing stress distribution, the cutting force can be controlled to such an extent that dynamic vibration can be minimized. By requiring that the tool's instantaneous position be compatible with the dynamic variations of the deformation of the tool holder, a stress distribution can be identified through the finite element analysis. The fundamental concept of the finite element method is that any region that is made up of elements, the behavior of a system can be determined by considering the behavior of its component.

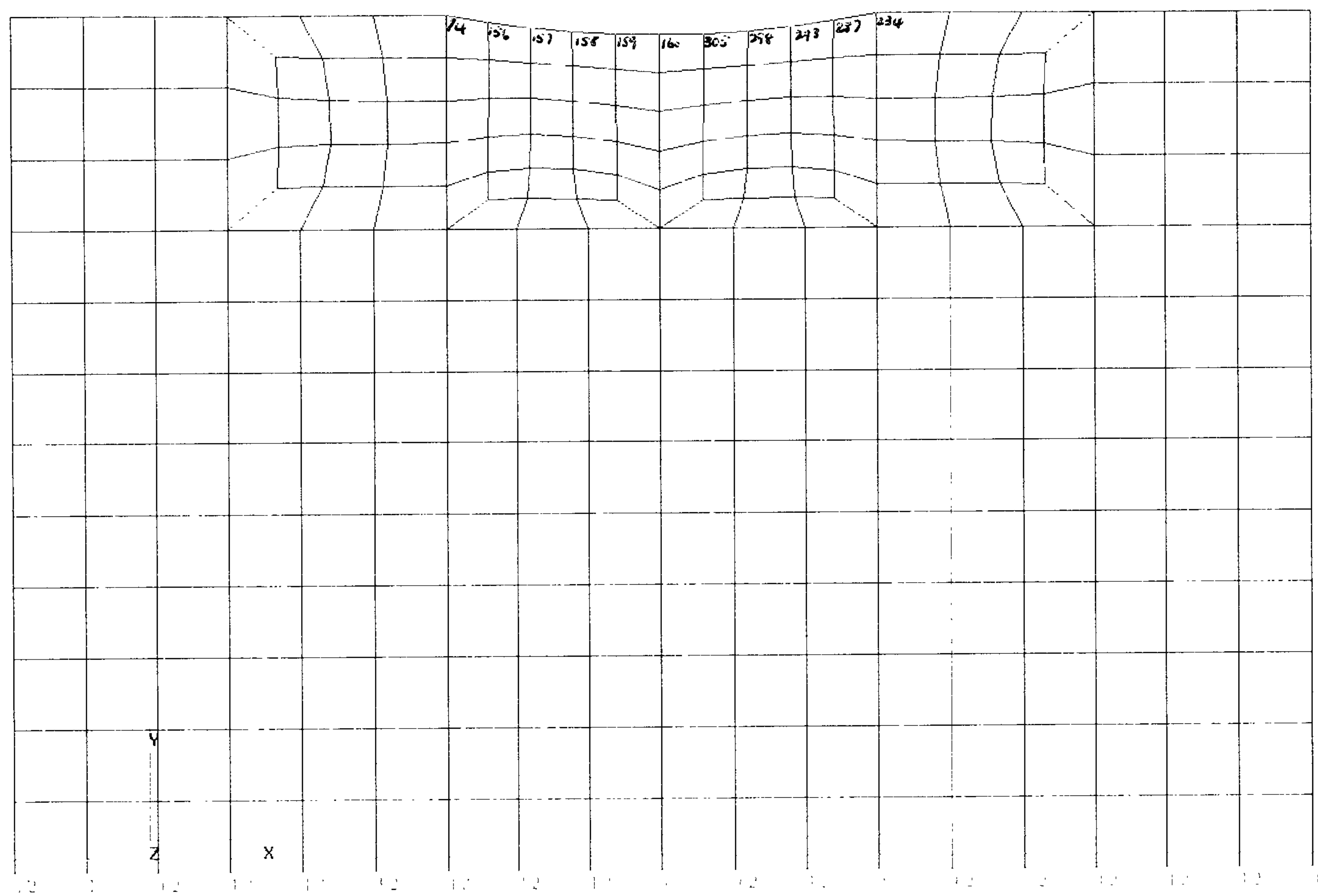

Figure 5. Displacement and Applied Forces

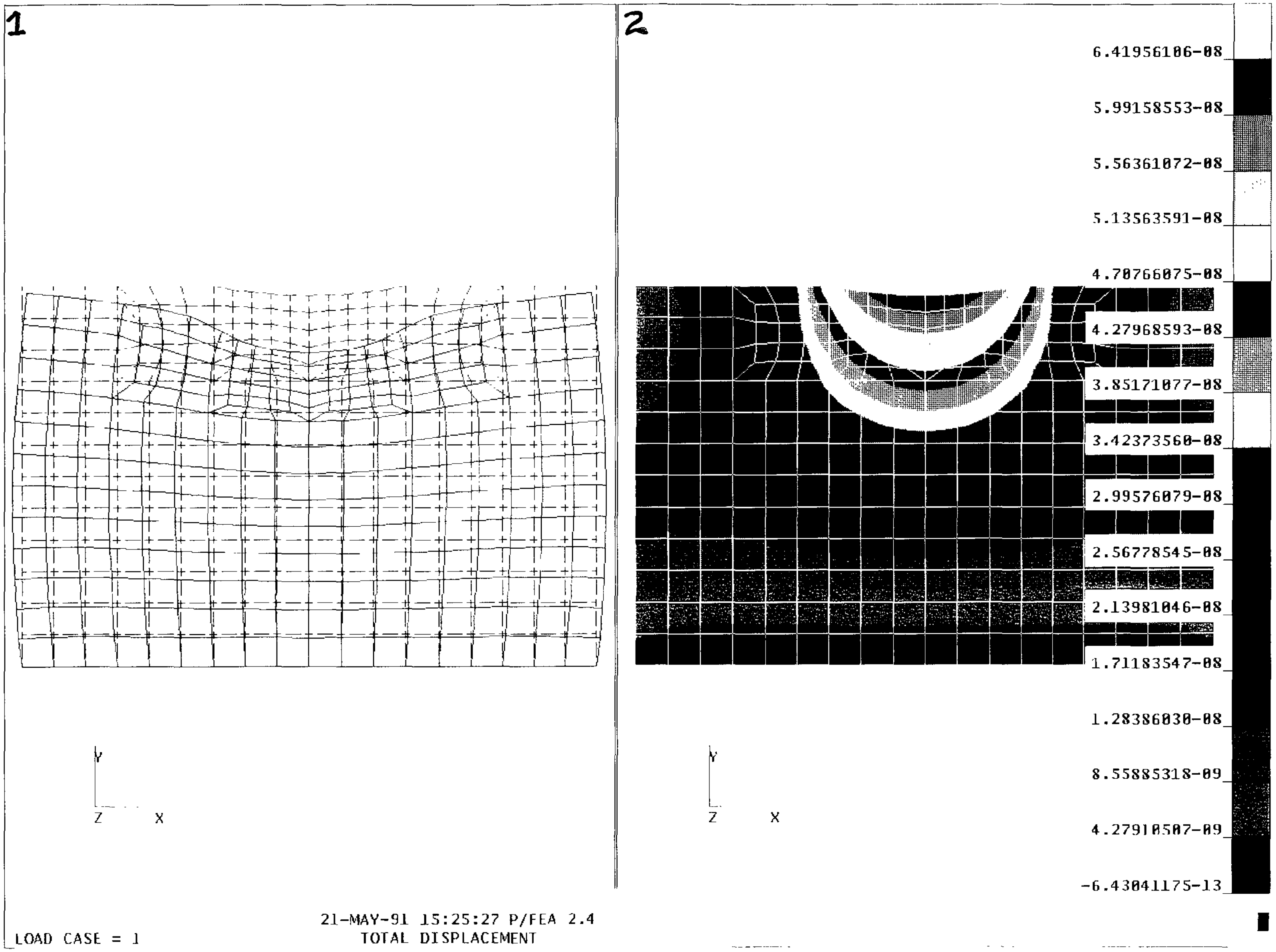

Figure 6. Total Displacement Stress Plot with Deformed and Undeformed Shape

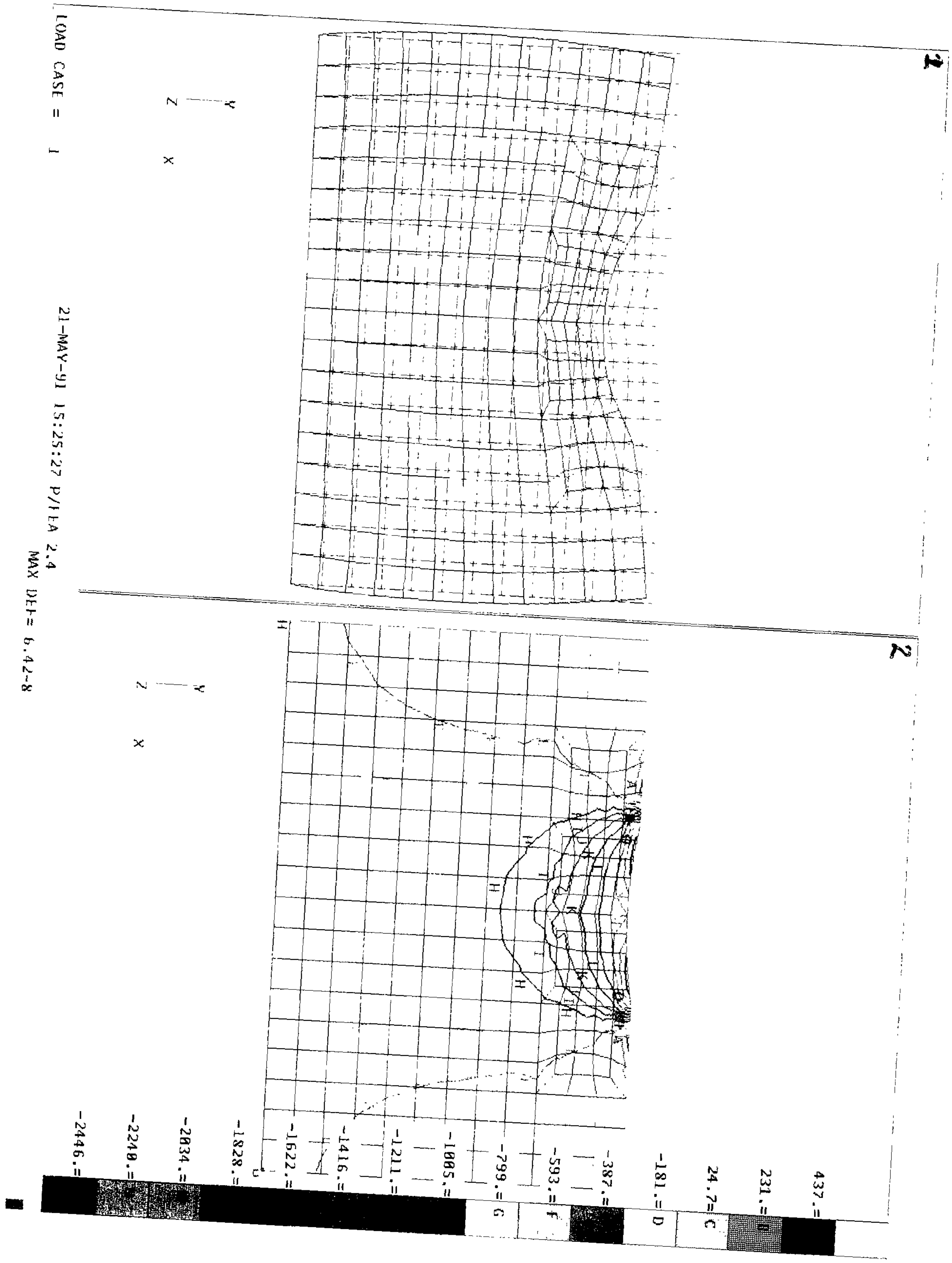

Figure 7. Contour Stress Plot Result

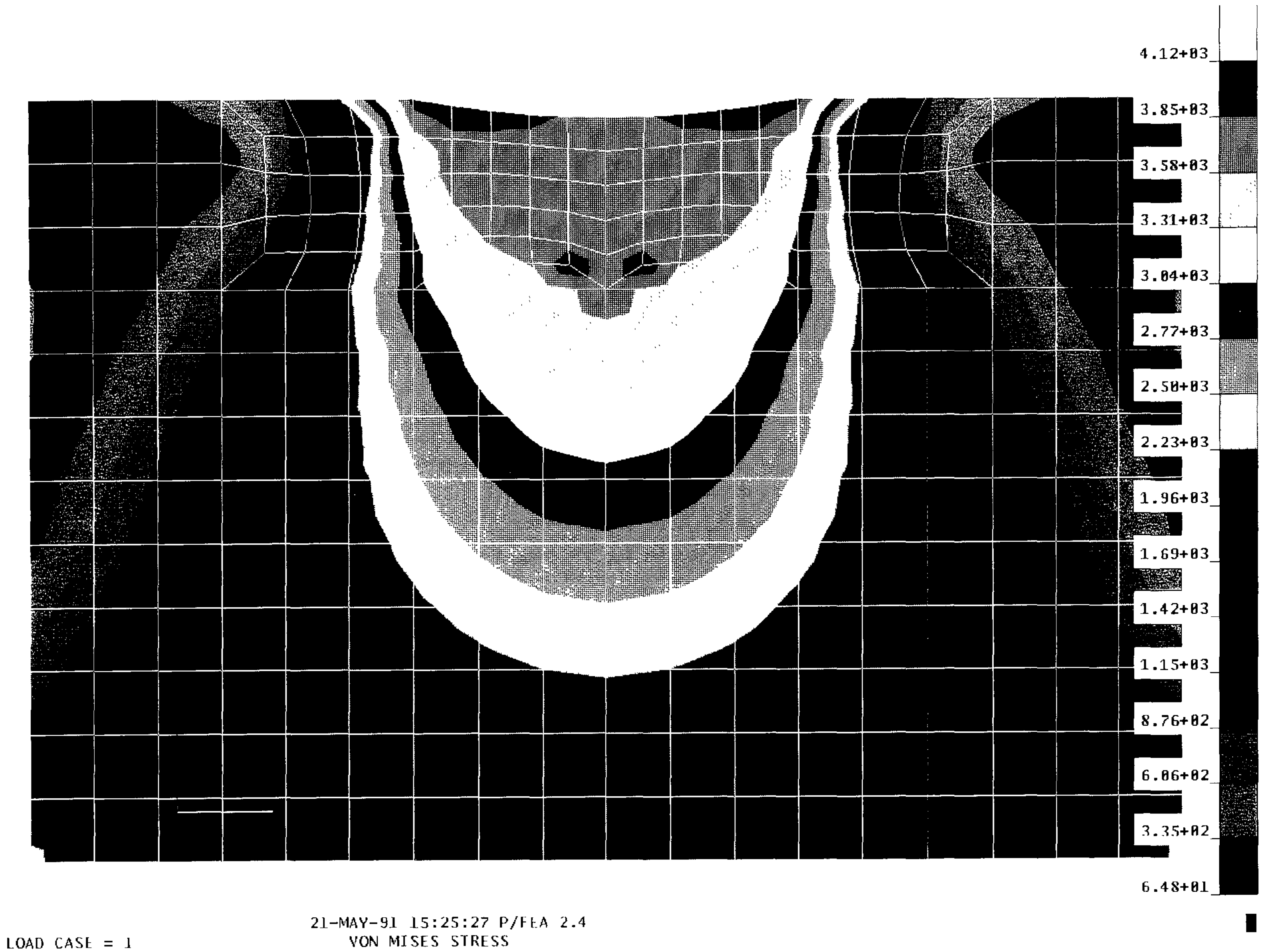

Figure 9. Mean Stress Plot

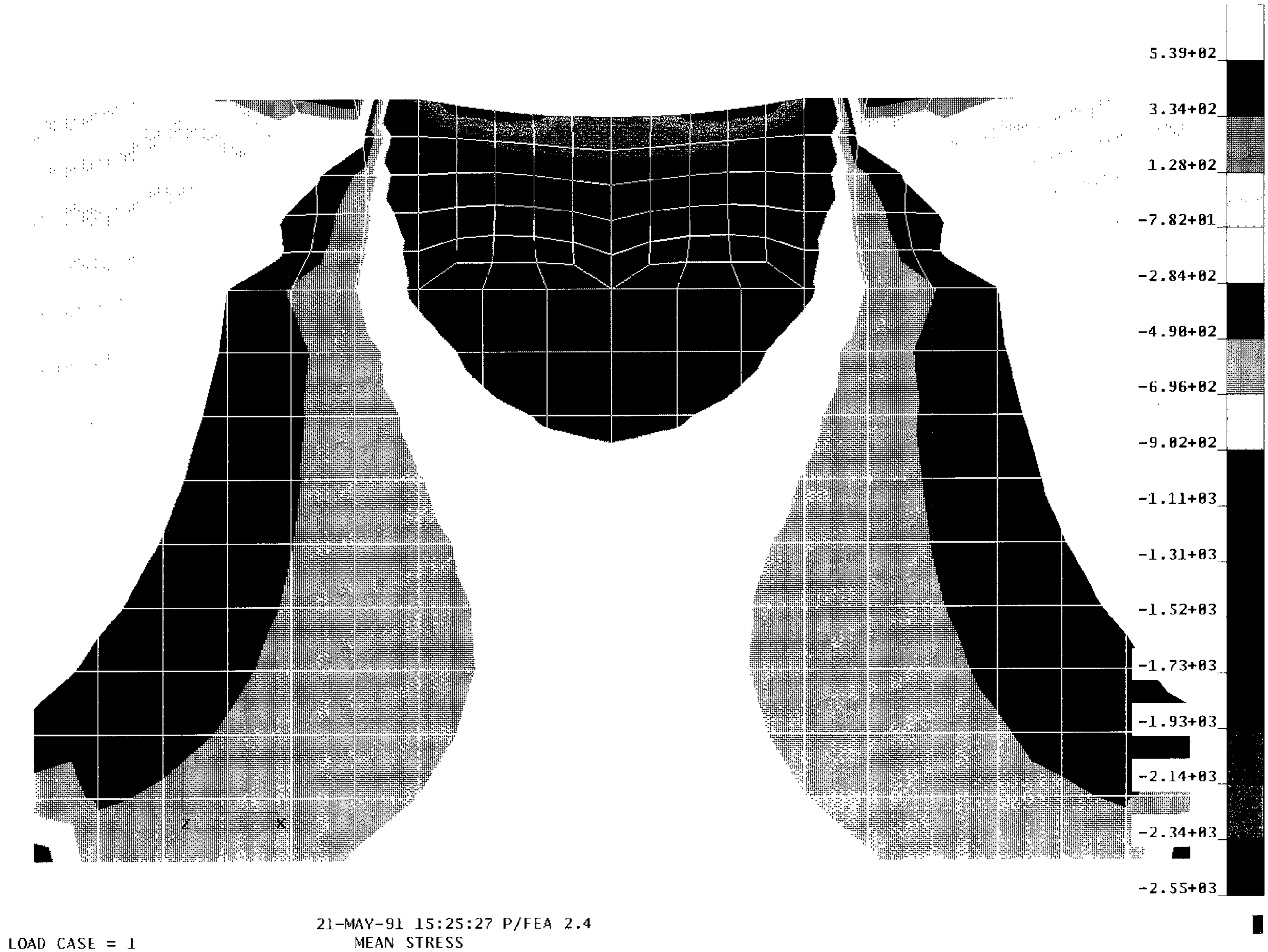

Figure 8. Von Mises Stress Plot Result

REFERENCES

Patran User Manual, Volume 1 and II

Akin, J. E., "Finite Element Analysis for Undergraduates," New York, Academic Press, 1987.

Akin, J. E., "Application and Implementation of Finite Element Methods," New York, Academic Press, 1982.

Cook, R. D., "Concepts and Applications of Finite Element Analysis," New York, John Wiley, 1974.

Zhang, G. M. and Hwang, T. W., "Analysis of the Cutting Dynamics in Microscale," Symposium on Fundamental Issues in Machining, 1990 ASME Winter Annual Meeting, PED-Vol. 43, pp. 25-37.

Zhang, G. M. and Kapoor, S. G., "Dynamic Generation of the Machined Surface, Part I: Mathematical Description of the Random Excitation System," Journal of Engineering for Industry, Transactions of ASME, May 1991, pp. 137-144.

Zhang, G. M., and Kapoor, S. G., "Dynamic Generation of the Machined Surface, Part II: Mathematical Description of the Tool Vibratory Motion and Construction of Surface Topography," Journal of Engineering for Industry, Transactions of the ASME, May 1991, pp. 145-153.