PROBE Utilization At The McDonnell Aircraft Company

by

Donald R. Ladwig
McDonnell Aircraft Company
McDonnell Douglas Corporation
St. Louis, Missouri

Export Authority: 22 CFR125.4(b)(13)

Abstract

The PROBE finite element analysis program has been widely used at the McDonnell Aircraft Company since 1985. Element formulations within PROBE utilize shape functions of varying polynomial "p-level" order. By increasing the p-level the accuracy of the solution can be improved without mesh refinement. A single analysis utilizing multiple p-levels can also provide unique checks for measuring solution accuracy. McDonnell Aircraft has used PROBE to obtain highly accurate solutions in structures containing stress concentrations and singularities. A sampling of various 2D and 3D McDonnell Aircraft structural models is presented. The problems encountered with using MSC/NASTRAN internal loads for input to a PROBE analysis are also discussed.

1. Introduction

The McDonnell Aircraft Company (MCAIR) makes widespread use of the PROBE finite element analysis program to perform detailed stress analysis. PROBE uses increasingly complex element shape functions or polynomial functions (p-level) to increase the solution accuracy. This approach enables PROBE to analyze models with high stress/strain gradients. MCAIR has successfully used PROBE to analyze a variety of structures of very complex shapes. We have analyzed stress/strain behavior near adhesive bond lines, stresses in reentrant corners and stress concentrations around holes and cutouts. PROBE is also used to analyze models where very precise answers are required. An advantage to PROBE elements is that they can be large and irregularly shaped, making representation of the part geometry an easy task. With the acquisition of St. Louis based Noetic Technologies by MacNeal-Schwendler in late 1989, PROBE became a new MSC computer-aided engineering software product.

2. PROBE Features and Software Interfaces at MCAIR

2.1 PROBE Features

Increasing the p-level increases the number of degrees-of-freedom in a PROBE solution. This differs from h-version codes such as MSC/NASTRAN which requires remeshing to increase the number of degrees-of-freedom. PROBE elements are capable of representing p-levels 1 through 8. A PROBE control card specifies the range of p-levels to be used for an analysis. A PROBE 2D quadrilateral element at p-level 1 is analogous to the MSC/NASTRAN QUAD4 membrane element. A PROBE 2D quadrilateral element at p-level 2 is analogous to the MSC/NASTRAN QUAD8 membrane element. The ability to increase the polynomial order results in elements that are relatively insensitive to shape. There are two versions of PROBE available. PROBE 2D includes:

- membrane analysis
- elasticity, heat transfer and axisymmetric solids analysis
- quadrilateral and triangular elements
- linear, quadratic and curved edges (mapping to exact curve definitions, circular, elliptical, etc.)

PROBE 3D includes:

- solid element analysis
- elasticity and modal analysis
- hexa, penta and tetra elements (linear, quadratic and circular edges)

By performing a PROBE analysis with multiple p-levels the user can plot the convergence of strain energy and other solution variables to determine the accuracy of the results.

2.2 PROBE Pre-Processing Interfaces At MCAIR

MCAIR uses the internally developed Computer Graphics Structural Analysis (CGSA) program for finite element pre- and post-processing. CGSA runs on a VAX host computer and can be accessed from both 3D color workstations or low cost terminals that emulate the Tektronix 4014. CGSA pre-processing for PROBE 2D models provides:

- model geometry (QUAD4/TRIA3 and QUAD8/TRIA6 connectivity for linear and quadratic elements respectively)
- assignment of isotropic and anisotropic material properties

CGSA pre-processing for PROBE 3D models provides:

- model geometry and constraints (hexa, penta and tetra elements for linear and quadratic elements)
- assignment of isotropic material properties
- model constraints (nodal)

CGSA assembles the model data and creates an ASCII input file for PROBE. The user enters the interactive PROBE software and loads the input file into the PROBE database. While in PROBE the loads and constraints are added. If element edges are to be mapped to conic sections, this is done in PROBE. PROBE is unique in the application of loads and constraints. PROBE does not allow point loads, all loads applied to an element are distributed over a finite region. CGSA does not model PROBE 2D loads and constraints and 3D loads because these must be associated with element edges and faces. Although PROBE 3D constraints are modeled at nodes, they are interpreted as edge and face constraints. When the finite element model is fully defined the analysis can be run on either the VAX or the CRAY computer.

2.3 PROBE Analysis Verification

Verification of the quality of a PROBE solution should include the following three checks:

- o Strain energy convergence.
  The percent relative error is calculated by extrapolating the strain energy based on three consecutive p-level solutions. This serves as an indication of the overall quality of the solution.

- o Load equilibrium checks.
  The equilibrium of elements and the continuity of interelement forces are examined. This helps to determine the local quality of the solution.

- o Continuity and convergence of point functionals such as stress or strain. This can be verified by comparing these values at a given node from each neighboring element. This helps to determine the local quality of the solution.

For the load equilibrium and point functional checks MCAIR has developed programs which read the ASCII results file from a 2D analysis and output the percent differences for each node or element in the model. PROBE 3D has some of these "quality assurance" checks automated.

PROBE can generate various types of output for results interpretation. Printed results include the standard node and element displacement, stress, and strain values. Error estimation based on strain energies from p-extension is provided. Element freebody data is calculated by integrating stresses on each element face or edge. PROBE can automatically calculate and plot any result along a user specified line in the model. Displacements, stresses and strains can be calculated at any point in the model, as opposed to just nodal positions. Stress intensity factors can be calculated for fracture mechanics analysis in 2D solutions.

Contour plot data for displacements, stresses, and strains can be generated in PROBE. Displacement plots are generated with PROBE's version of MOVIE-BYU. Contour plots of stress and strain are generated with a MCAIR developed color, solid fill contour plotting program and with PDA/PATRAN.

3. PROBE 2D Stress Analysis At MCAIR

PROBE 2D has been used at MCAIR since 1985. Each of our aircraft programs uses PROBE for detail stress analysis of complex parts, especially where high stress or strain gradients are expected.

In many cases the applied loads are extracted from an MSC/NASTRAN analysis and applied to the PROBE model. Typically, one PROBE element encompasses many MSC/NASTRAN elements, which results in loads from many nodes being applied to one PROBE element. The application of MSC/NASTRAN loads to a PROBE model can be very time consuming, often requiring more manhours than any other aspect of the analysis.

Several of the more interesting MCAIR applications of the 2D program are described in detail in the following paragraphs.

3.1 Fatigue test specimen for repair analysis

This application involved the analysis of a hole bonded repair on a vertical stabilizer. Standard repair techniques could not be used on the hole repair because of the damage size and location. Two problems needed to be solved to determine the integrity of the proposed repair. First, a peel and shear stress fatigue allowable had to be established, and second, an analysis of the repair configuration to determine the maximum peel and shear stresses had to be done. To establish a peel and shear stress fatigue allowable, data from a laboratory specimen was used. Figure 1 shows the test specimen which was a 0.05 inch thick titanium strip joined by a 0.026 inch thick carbon/expoxy adherend in a double lap configuration with a 0.5 inch lap. The adhesive was 0.011 inch thick staged FM300. Due to the symmetry of the problem, a 1/4 model was used. The PROBE finite element model is shown in Figure 2.

The model was validated by comparing of the shear stress distribution to a Hart-Smith composite bonded joint analysis technique [1]. The results of this comparison are shown in Figure 3. The solutions are very close except for the shear stress at the end of the bond. This difference is a result of the influence of the singularity at the bond termination which is not included in the Hart-Smith method (and which most other finite analysis codes would find hard to adequately analyze). From the PROBE solution of the test specimen, maximum allowable peel and shear fatigue stresses were determined and applied to the analysis of the repair configuration.

3.2 Lap weld

The longitudinal seam weld in a reaction control system duct was analyzed. The weld is made by lapping the two sides of the metal and melting them together as shown in Figure 4. Figure 5 shows the PROBE finite element mesh of the offset weld with physical and metallurgical notches. The diameter of the duct is approximately 4 inches, with the wall thickness being 0.024 inches. The internal pressure is approximately 255 psi. The mean hoop stress value calculated by PROBE agreed well with previous analysis and test data. Figure 6 is a plot of the PROBE maximum principal stresses. The stress concentration values that were calculated at each of the three notch locations is also shown.

3.3 Composite skin reentrant corner

A section of an upper wing skin, shown in Figure 7, was analyzed to gain a better understanding of the strain distribution in an area of high strain gradients. Figure 8 shows the original MSC/NASTRAN model of this area, Figure 9 shows the PROBE model. A new element was modeled for every five lamina in the composite skin, resulting in 179 grids and 131 elements. Figure 10 shows the balanced freebody loads for this model from the MSC/NASTRAN analysis. Note that these applied loads were numerous due to the various ribs, spars and bulkheads that introduced loads into the model. The magnitudes of these loads often vary significantly from grid to grid. Many assumptions were made when these loads were "smeared" out and applied to the PROBE model. Loads at a joint where two or more load introducing members are present require that these loads be split into components along each member. The large number of load vectors as well as the above assumptions resulted in the application of these loads onto the PROBE model taking over 50% of the model preparation time. The run time for p-levels 6 through 8 was 40 minutes, 49 seconds on a VAX 8650.

3.4 Wing skin and spar with holes, bosses

Two different models are shown where stress concentrations were analyzed around fastener and access holes. Figure 11 is a lower wing skin with various size holes and bosses. Figure 12 is a spar with various size holes, bosses and flanges throughout the model. These models were interesting because of the complex geometry, which included many thickness changes throughout the model in addition to the holes.

4. PROBE 3D Stress Analysis At MCAIR

PROBE 3D been used at MCAIR since early 1988. The original release had limited capabilities. Only with the recent addition of the tetrahedron element to the hexahedron and pentahedron elements has the program had any appreciable use. The most significant use of the program has been in the analysis of a landing gear.

4.1 Landing gear model

PROBE 3D was used to analyze the complex landing gear model in Figure 13. The model contains 1020 nodes and 319 elements. The model was run on a CRAY X-MP/18 computer running COS 1.17. Total run time was 1865 seconds for p-levels 1 through 6. Following is a chart showing the number of degrees-of-freedom and percent relative error in strain energy for each p-level.

p	Global DOF	Total Energy	Extrapolated Energy	Convergence Rate	Percent Rel. Error Est'd
1	1444	0.169602D+05	0.000000D+00	0.00	54.75
2	5281	0.234230D+05	0.000000D+00	1.70	18.14
3	9694	0.240328D+05	0.244226D+05	2.38	8.79
4	17510	0.241548D+05	0.241875D+05	1.78	5.19
5	29158	0.241940D+05	0.242206D+05	1.80	3.28
6	45595	0.242084D+05	0.242200D+05	1.80	2.19

A plot of maximum principal stress is shown in Figure 14. The results from this analysis agree well with results from other analyses.

5. Conclusions and Recommendations

MCAIR has found PROBE to be a very useful tool for detail stress analysis. Often this program is the only tool that can adequately capture the stress or strain behavior of a model. It is useful for analyzing bond lines because elements with high aspect ratio are typically modeled, and for analyzing singular points and other areas with high stress/strain gradients. PROBE allows simple meshes to be constructed, with the user increasing the p-level instead of remeshing for better answers. Quality assurance checks and other output provide excellent user feedback of the quality of the solution.

PROBE is not a tool for creating models to analyze load paths. Large internal loads models at MCAIR are modeled using MSC/NASTRAN. The extensive element library in MSC/NASTRAN allows versatile representation of aircraft structures. PROBE is used where a detailed understanding of stress or strain behavior is necessary.

Because of the small number of elements that may be required for a PROBE analysis, a graphics pre-processor may not be required to generate the model. Typical aircraft models for PROBE analysis will require graphics pre-processors to minimize model preparation time. Graphics pre-processors that would model the PROBE unique features, such as elements with circular or elliptical sides and boundary conditions (loads and constraints) that are applied to element edges and faces instead of nodes, would further reduce model preparation time.

It would be helpful if MSC could facilitate the transfer of loads and other boundary conditions from MSC/NASTRAN to PROBE. PROBE requires a stress state to be defined along a boundary, while MSC/NASTRAN can provide discrete nodal loads. The boundary conditions can include complex applied loads or enforced displacements taken from numerous MSC/NASTRAN nodal positions and are smeared out along a given PROBE element. The application of boundary conditions taken from a MSC/NASTRAN solution onto a PROBE model can be very tedious and time consuming.

PROBE 3D needs the addition of a quadrilateral and triangular shell elements to simplify model definition. PROBE 3D does offer a "thin" solid, where the p-level can be varied through the thickness independent of the rest of the model. These elements are still solid elements and must be modeled as such.

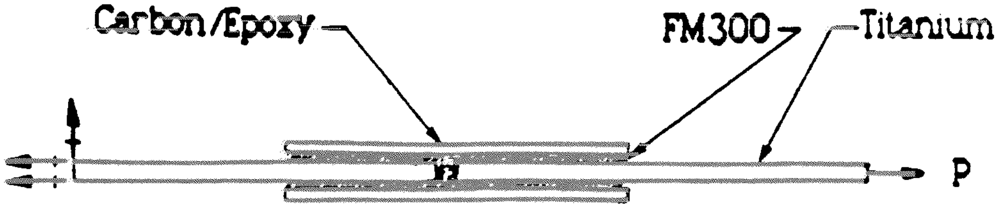
Even though approximations would be made when mating shell and solid elements, this could be handled by using multipoint constraints.

Geometric and material nonlinear analysis is becoming more widely used throughout MCAIR. The addition of nonlinear analysis and shell elements to PROBE 3D would allow even greater use of the program.

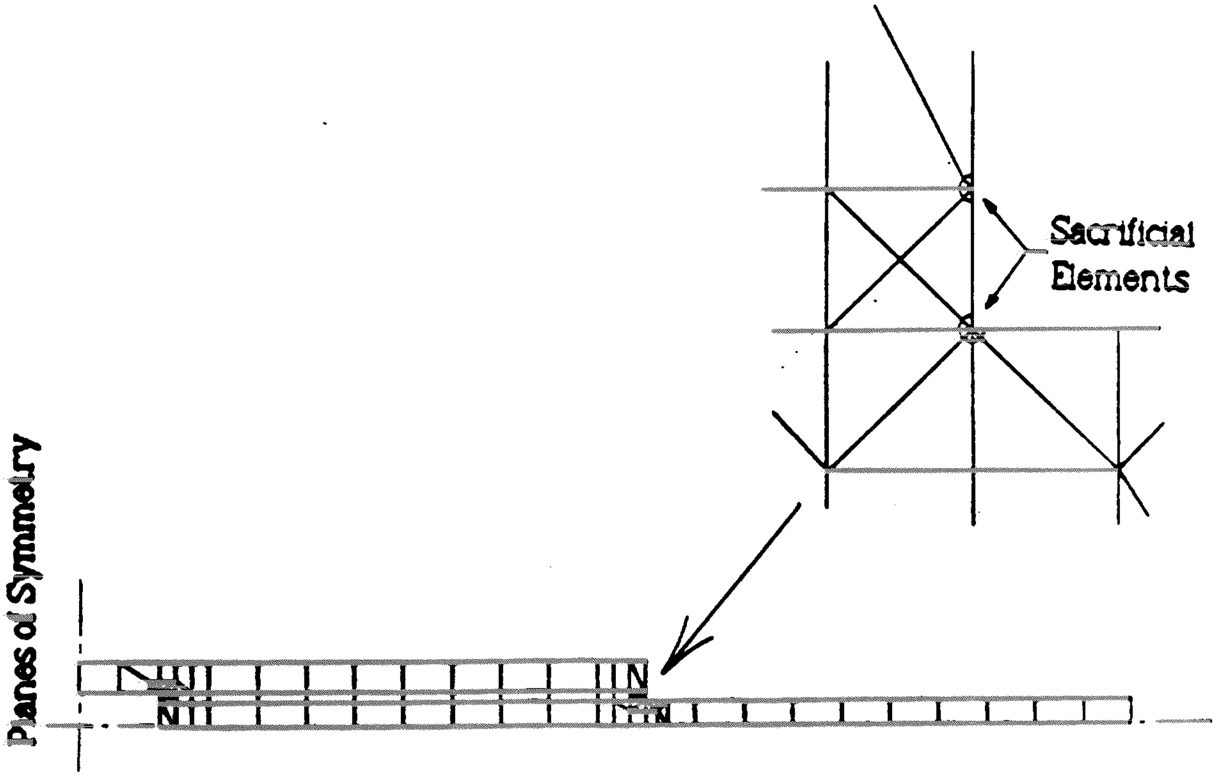
Acknowledgements - The author is grateful to D. Duewer, P. G. Mueller, R. Peterson, and J. Strain of McDonnell Aircraft and to M. J. Heskitt and J. C. Owen of MSC/NOETIC for their assistance in assembling the information for each of the models presented.

References

1. L. J. Hart-Smith, "Analysis and Design of Advanced Composite Bonded Joints", NASA CR-2218, January 1973.



FATIGUE TEST SPECIMEN
FIGURE 1



FATIGUE TEST SPECIMEN
PROBE FINITE ELEMENT MODEL
FIGURE 2

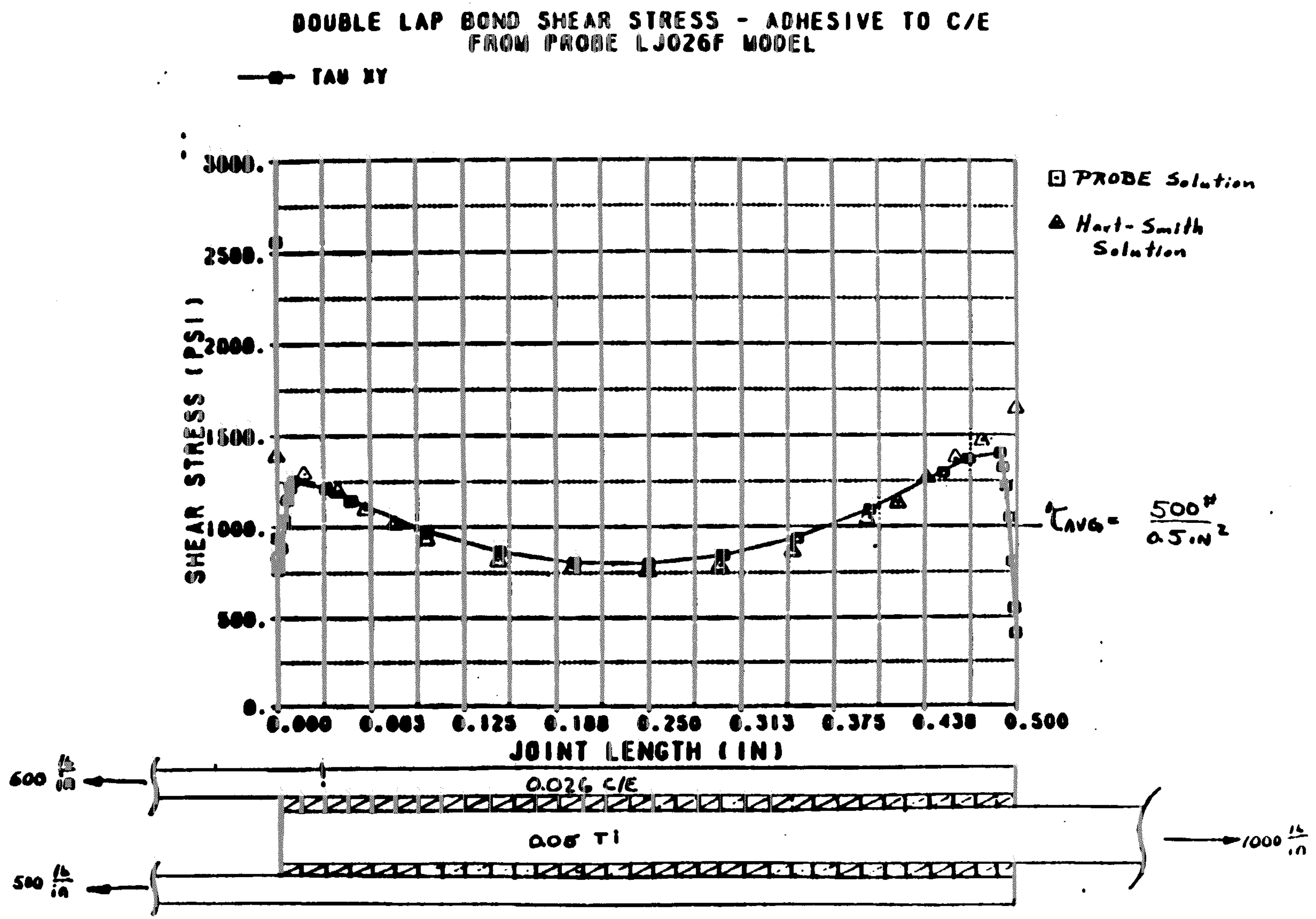
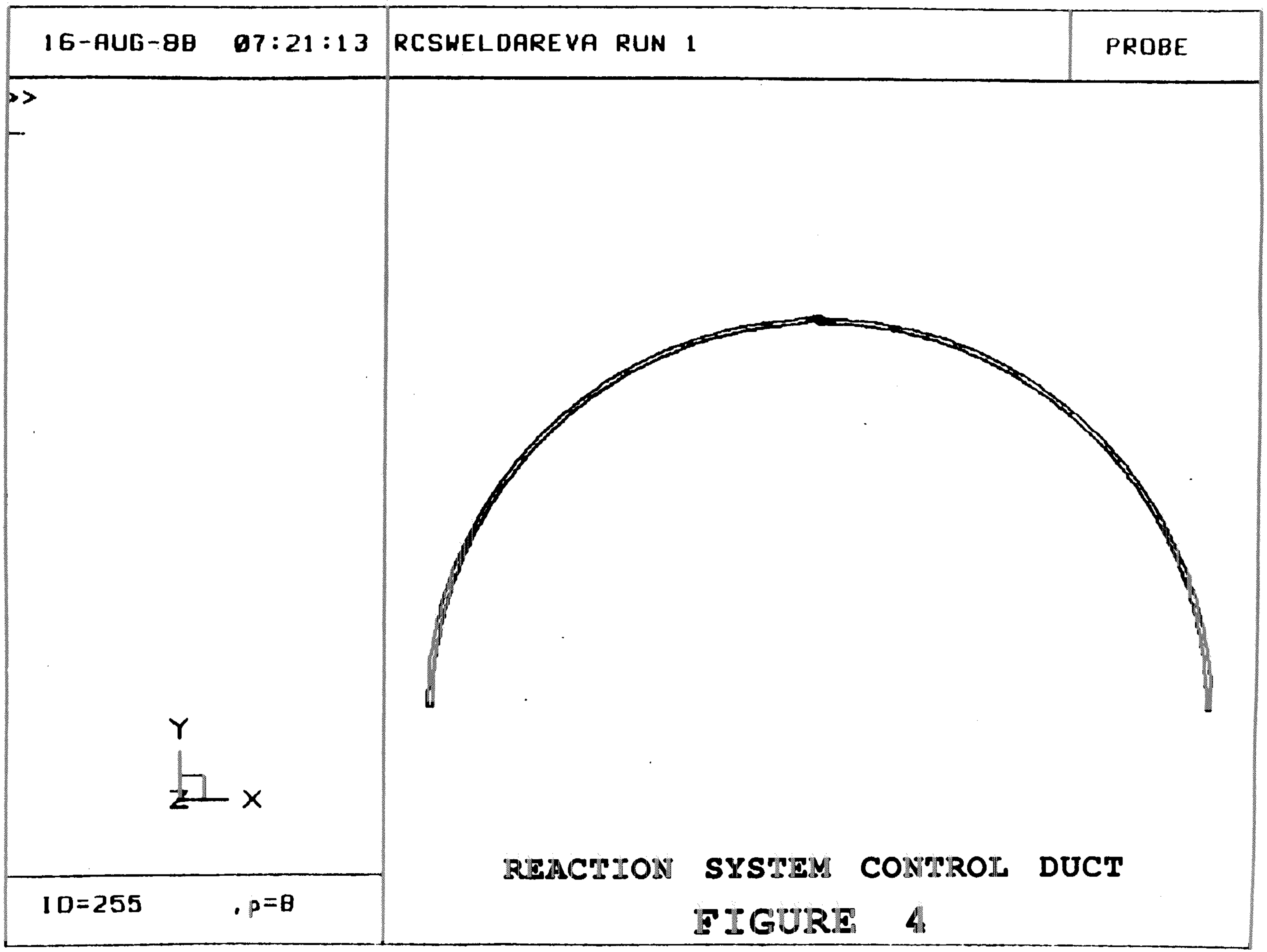
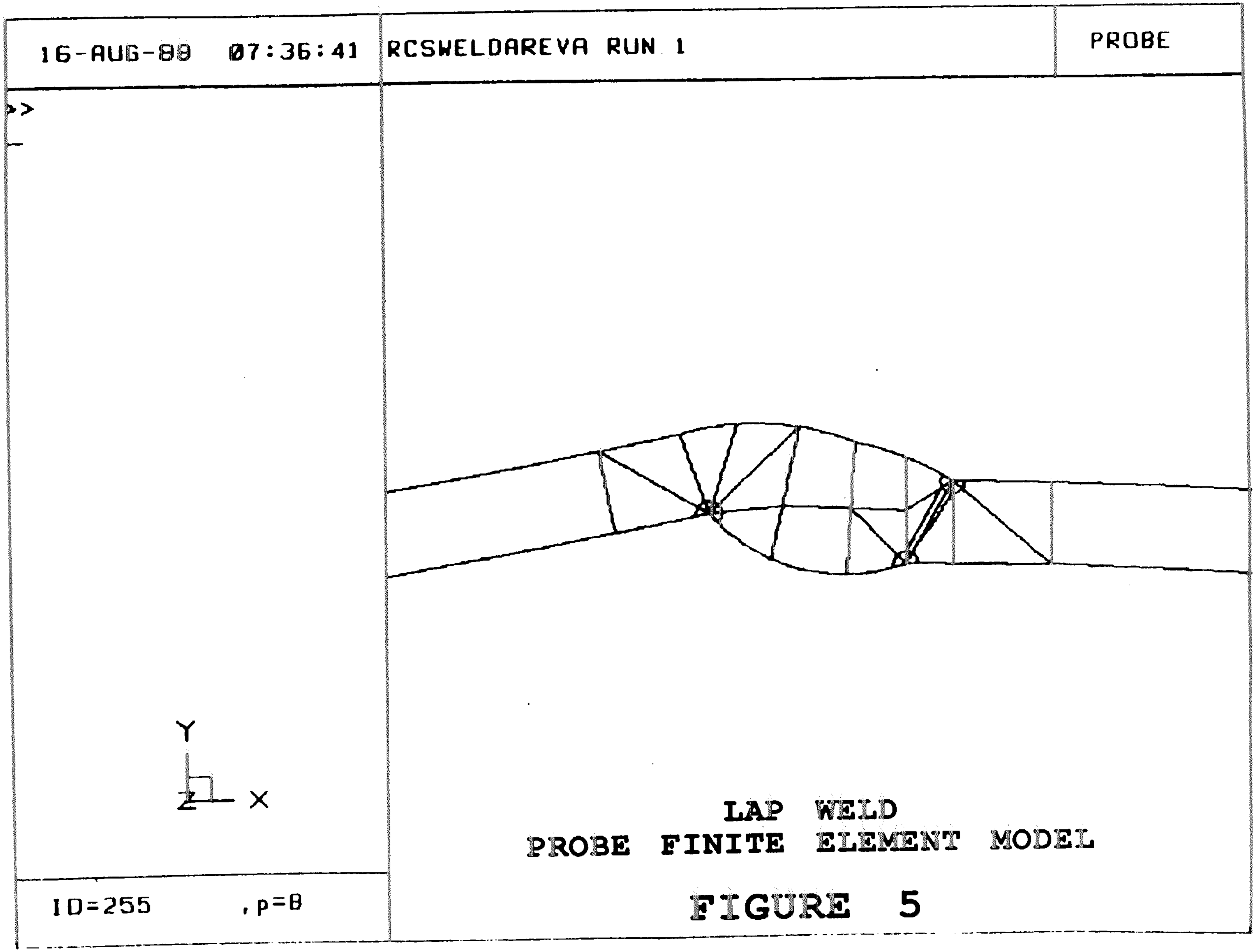

DOUBLE LAP BOND SHEAR STRESS - ADHESIVE TO C/E
FROM PROBE LJ026F MODEL
TAU XY
SHEAR STRESS (PSI)
3000.
2500.
2000.
1500.
1000.
500.
0.
0.000
0.063
0.125
0.188
0.250
0.313
0.375
0.438
0.500
JOINT LENGTH (IN)
⊡ PROBE Solution
▲ Hart-Smith Solution
$\tau_{AVG} = \frac{500^{\#}}{0.5\,in^2}$
0.026 C/E
0.05 Ti
600 lb/in
500 lb/in
1000 lb/in

FIGURE 3



REACTION SYSTEM CONTROL DUCT

FIGURE 4



LAP WELD
PROBE FINITE ELEMENT MODEL

FIGURE 5

16-AUG-88 13:00:07 RCSWELDAREVA RUN 1 PROBE

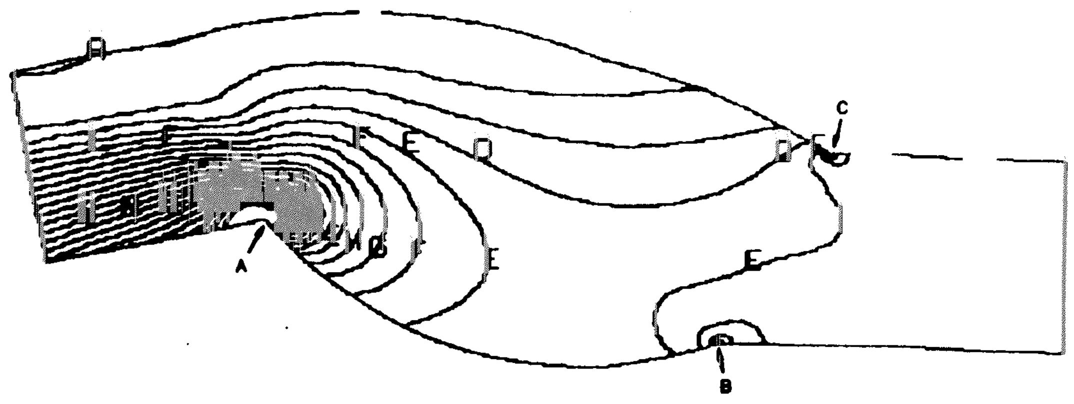
>>

CONTOUR LEGEND

A = 0.000E+00
B = 0.500E+04
C = 0.100E+05
D = 0.150E+05
E = 0.200E+05
F = 0.250E+05
G = 0.300E+05
H = 0.350E+05
I = 0.400E+05
J = 0.450E+05
K = 0.500E+05
L = 0.550E+05
M = 0.600E+05
N = 0.650E+05
O = 0.700E+05
P = 0.750E+05
Q = 0.800E+05
R = 0.850E+05
S = 0.900E+05
T = 0.950E+05
U = 0.100E+06

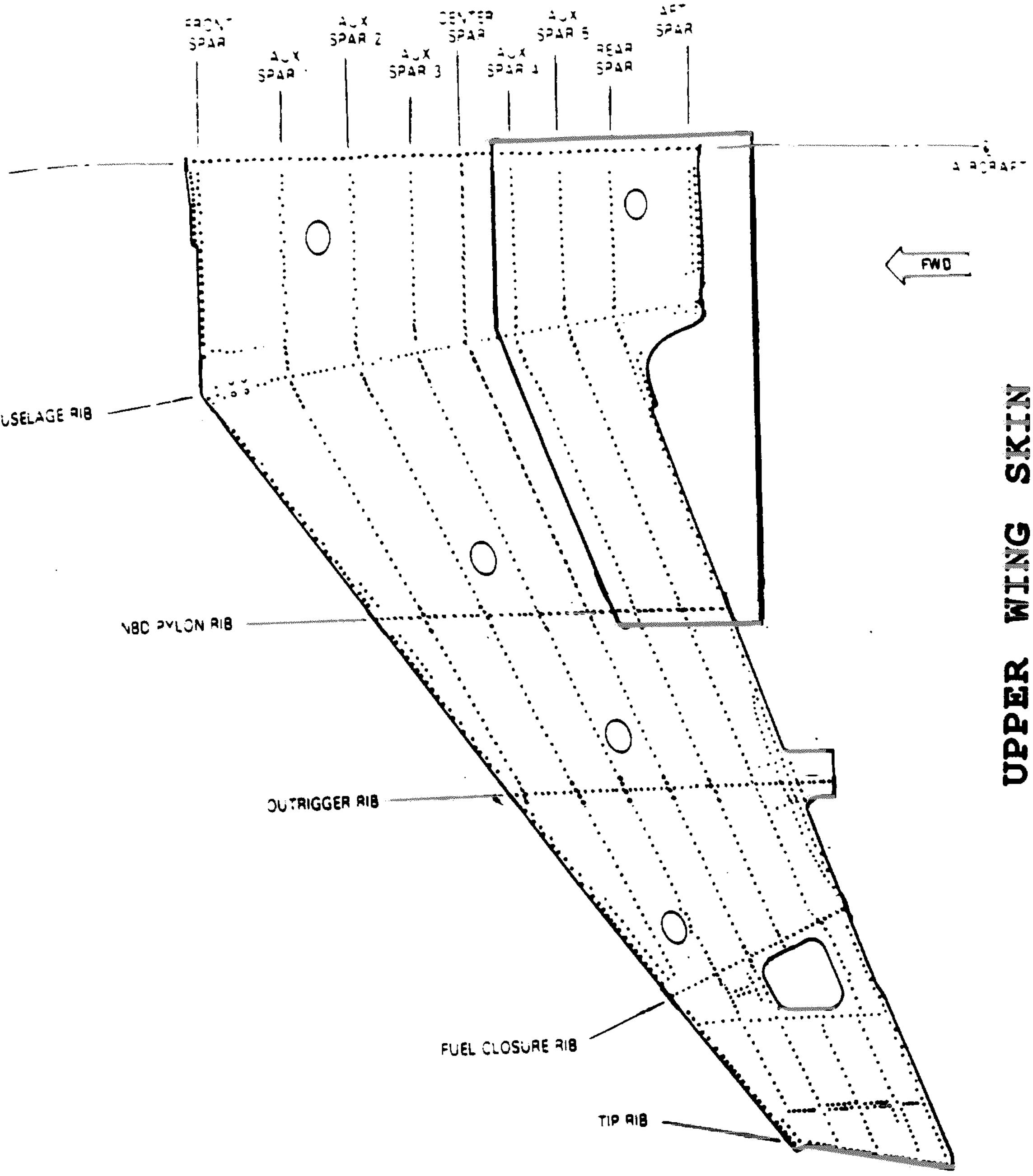
ID=255 ,p=8
SIGMA-1

$\sigma_{A\max}$ = 300 ksi; K_{tA} = 15
$\sigma_{B\max}$ = 50 ksi; K_{tB} = 2.5
$\sigma_{C\max}$ = 60 ksi; K_{tC} = 3



LAP WELD
PROBE PRINCIPAL STRESSES

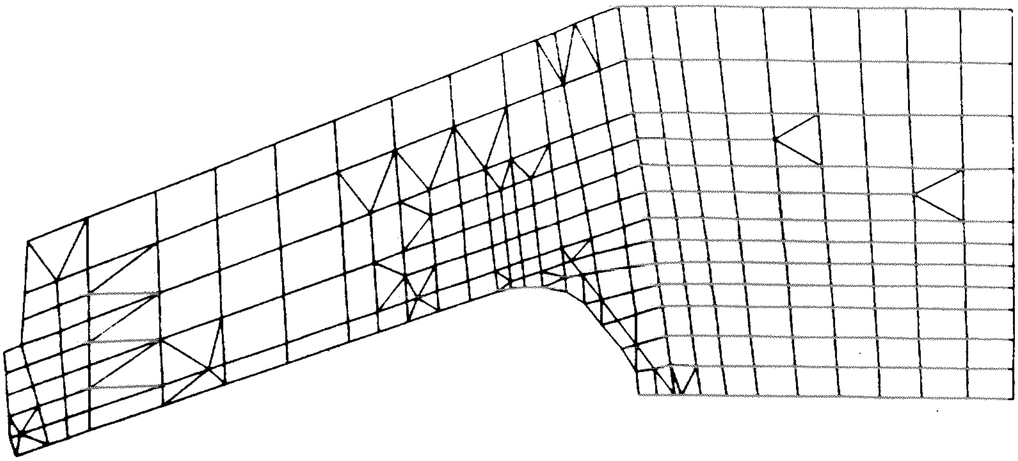
FIGURE 6



UPPER WING SKIN

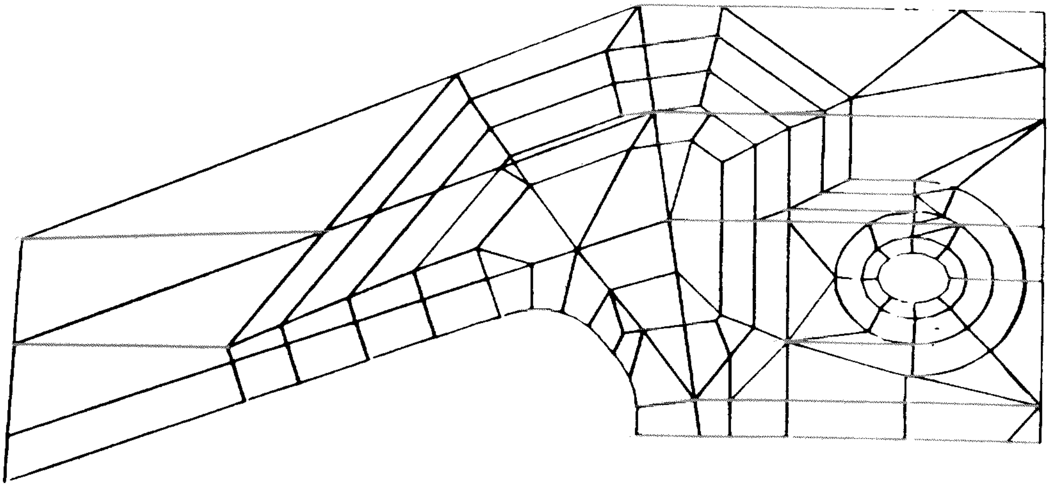
FIGURE 7

WING SKIN REENTRANT CORNER



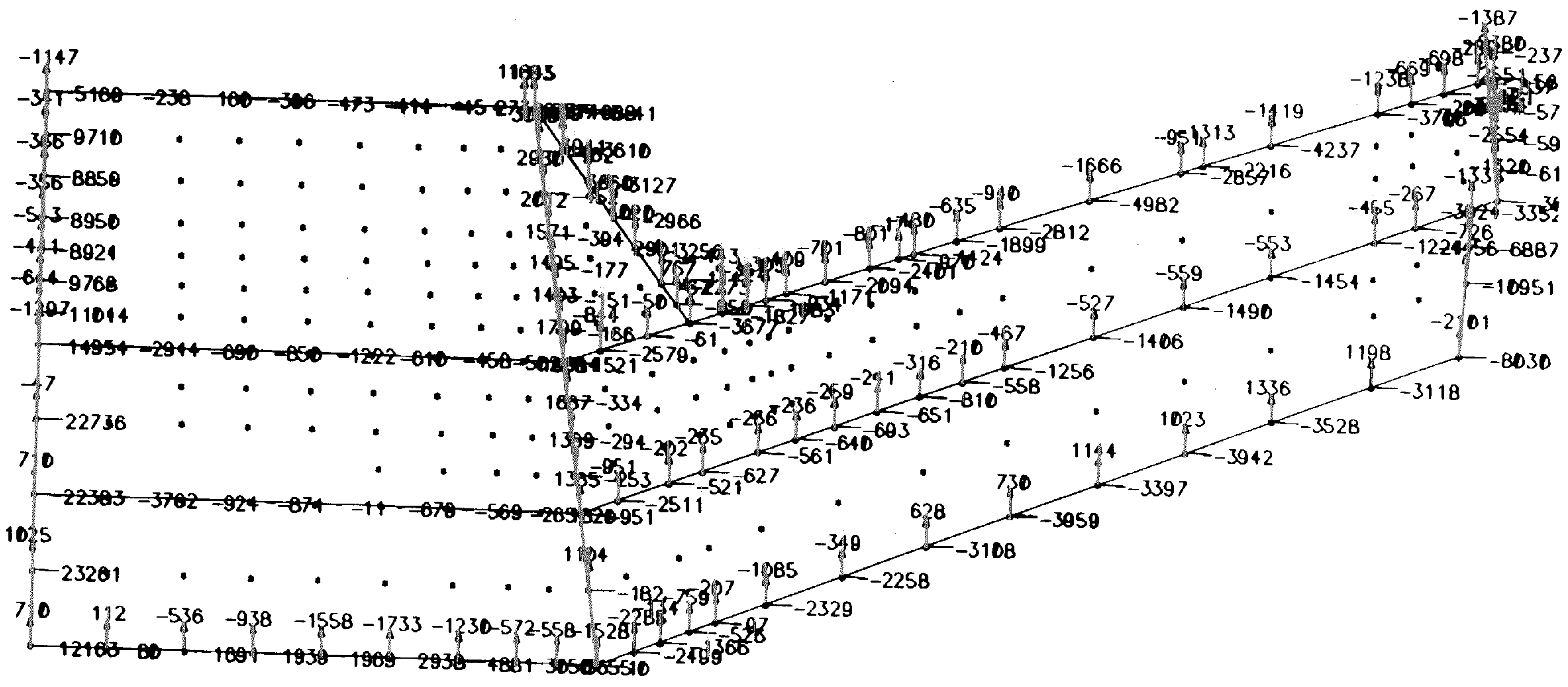
NASTRAN FINITE ELEMENT MODEL

FIGURE 8



PROBE FINITE ELEMENT MODEL

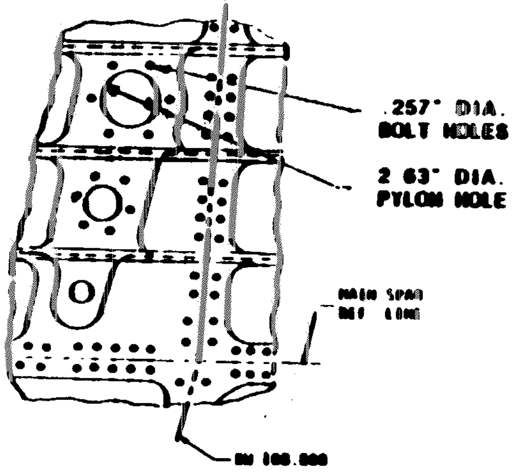
FIGURE 9



WING SKIN REENTRANT CORNER NASTRAN FREEBODY LOADS

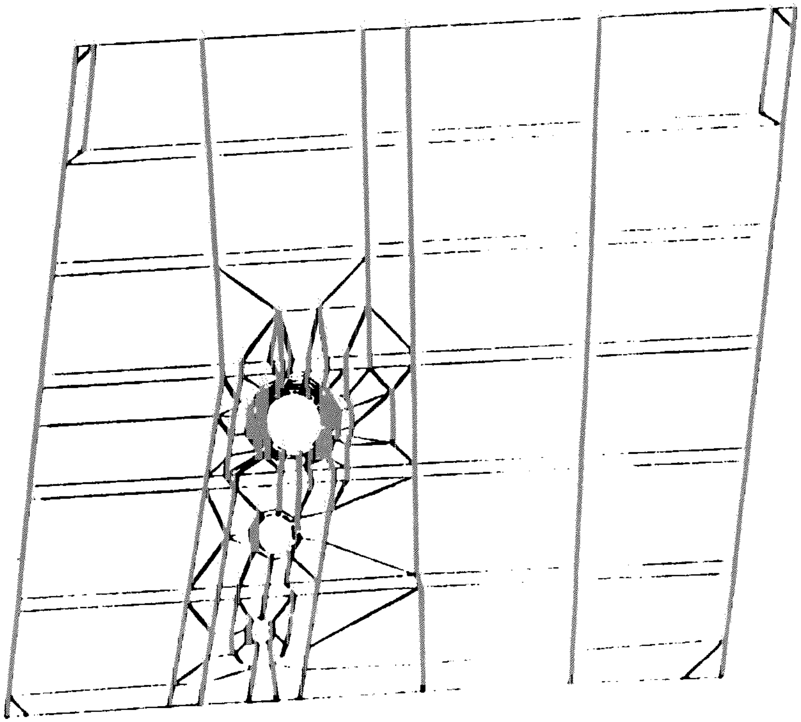
FIGURE 10

LOWER WING SKIN @ LARGE PYLON HOLE
PROBE FINITE ELEMENT MODEL



ACTUAL LOWER SKIN

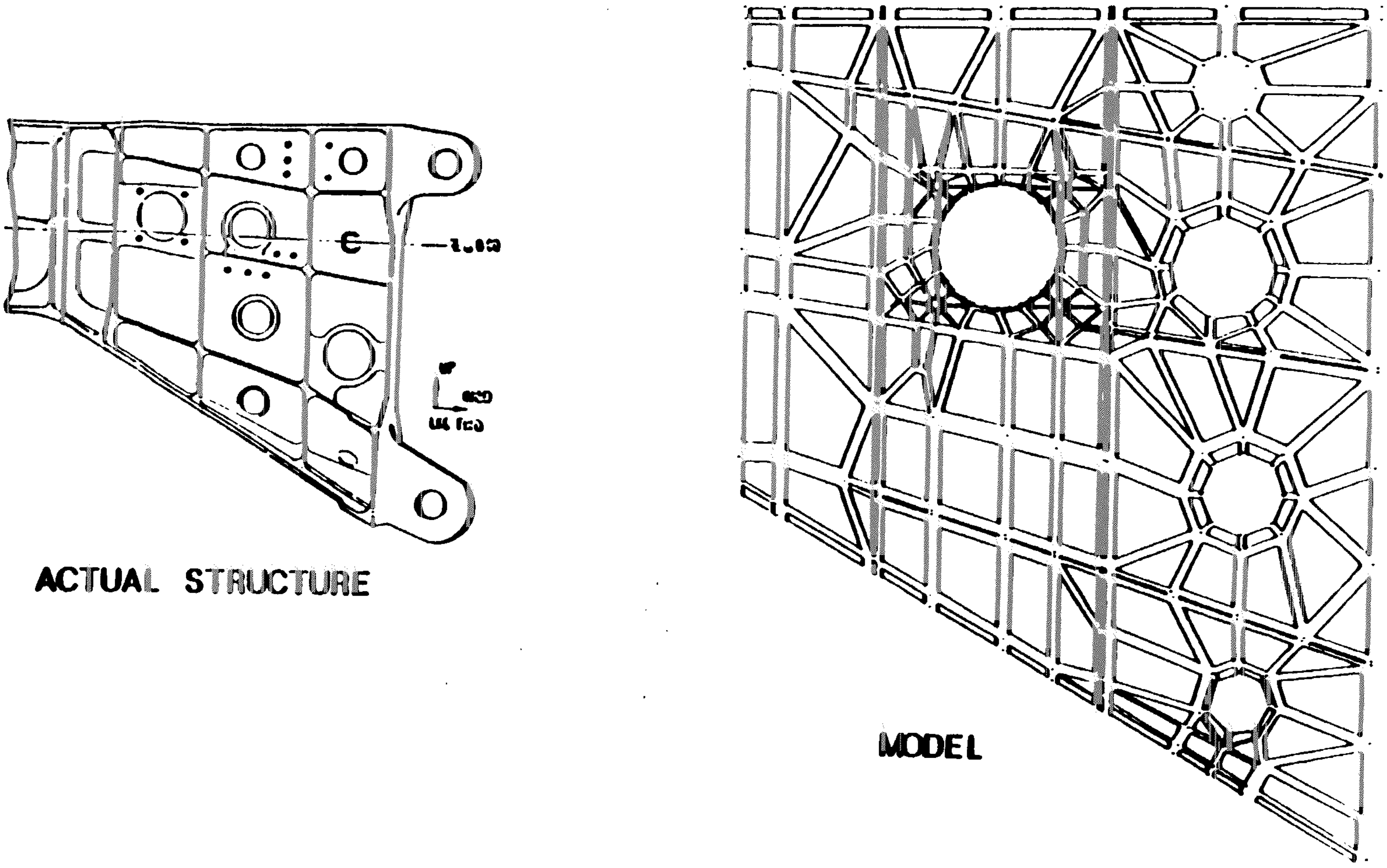
MODEL



NUMBER OF ELEMENTS = 281

FIGURE 11

MAIN SPAR FUEL TRANSFER HOLE PROBE FINITE ELEMENT MODEL

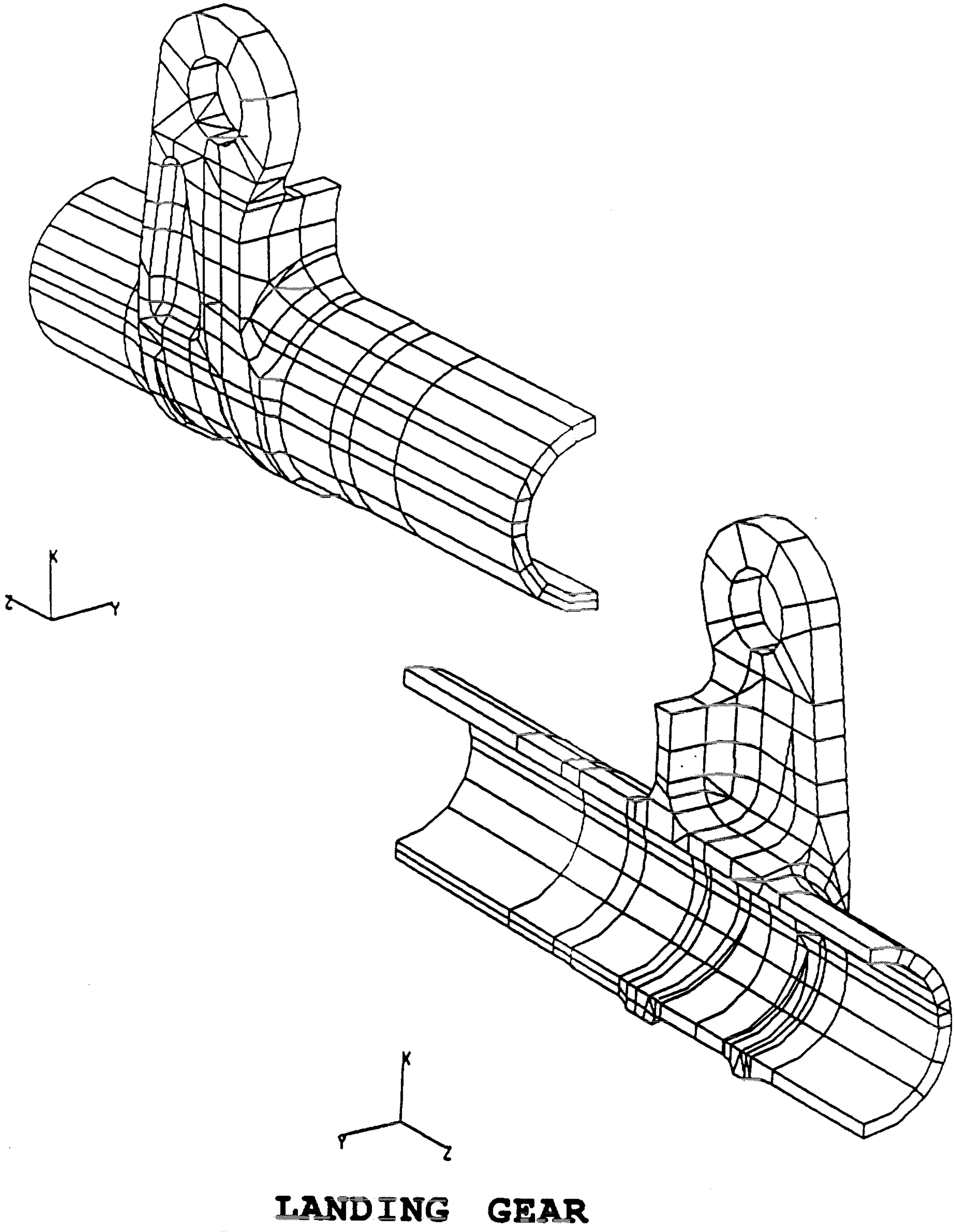


ACTUAL STRUCTURE

MODEL

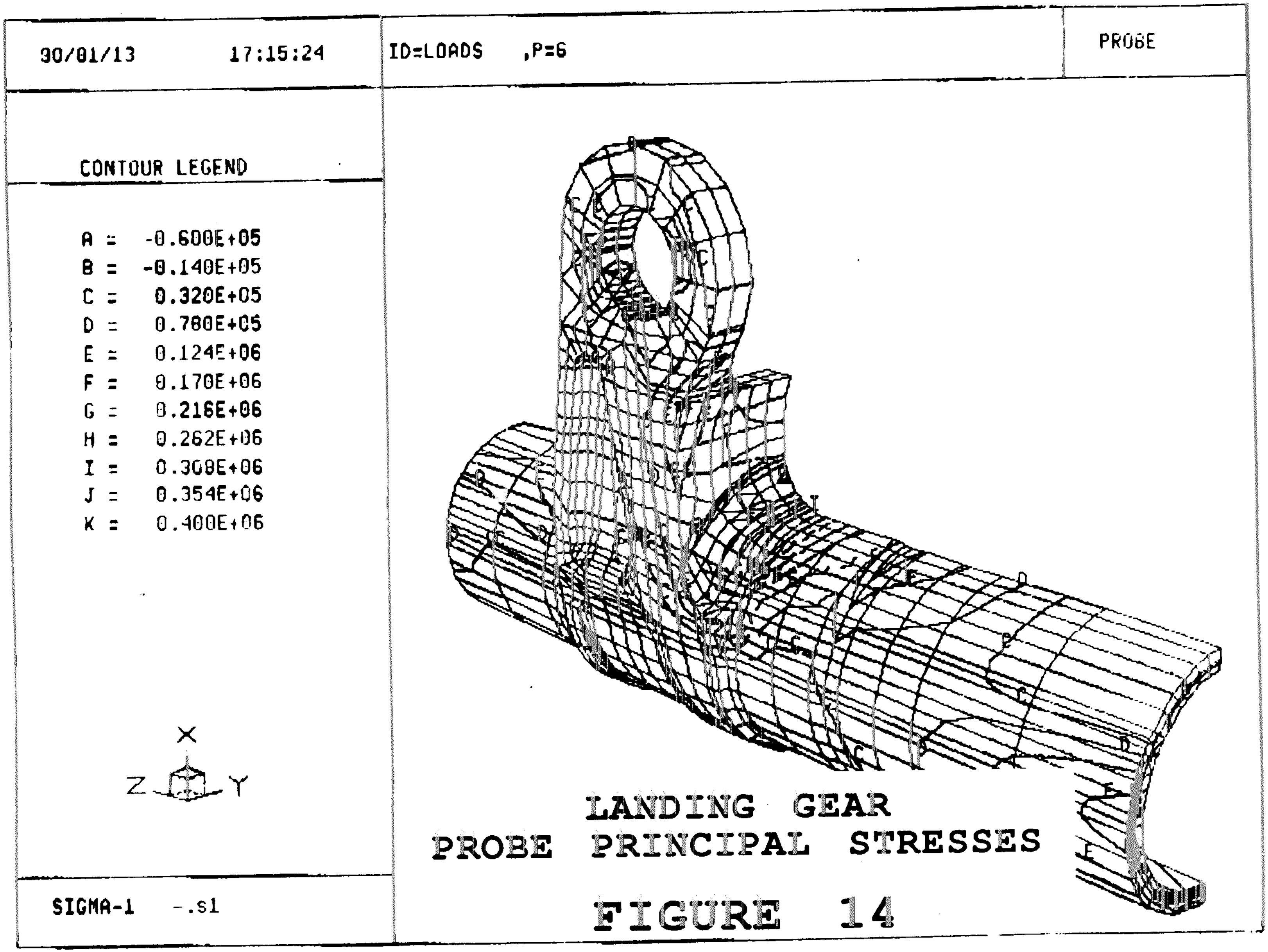
FIGURE 12

NUMBER
OF ELEMENTS = 296



LANDING GEAR
PROBE 3D FINITE ELEMENT MODEL

FIGURE 13



90/01/13 17:15:24

ID=LOADS ,P=6

PROBE

CONTOUR LEGEND

A = -0.600E+05
B = -0.140E+05
C = 0.320E+05
D = 0.780E+05
E = 0.124E+06
F = 0.170E+06
G = 0.216E+06
H = 0.262E+06
I = 0.308E+06
J = 0.354E+06
K = 0.400E+06

SIGMA-1 -.s1

LANDING GEAR
PROBE PRINCIPAL STRESSES

FIGURE 14