CURRENT DEVELOPMENT IN SELECTED STRESS AND THERMAL ANALYSIS SOFTWARE INTERFACES WITH PRO-ENGINEER

James Schulze, SMTS
Sandia National Laboratories
Albuquerque, New Mexico

Ever since PRO-ENGINEER has become a dominating CAD package available to the public, some of us have been saying, "Gee, if only I could export my geometry to a stress analysis program without having to recreate any of the details already created, wouldn't that be spectacular?" Well, much to the credit of the major stress and thermal analysis software vendors, some of them have been listening to design engineers like me badger them to furnish a seamless interface between PRO and their stress analysis programs. The down side of this problem is the fact that a lot of problems still exist with most of the vendors and their interfaces.

I want to discuss the interfaces that I feel are currently "State of the Art", and how they are developing and the future for finally arriving at a transparent procedure that an engineer at a workstation can utilize in his or her design process. In years past, engineers would develop a design and changes would evolve based on intuition, or somebody else's critical evaluation. Then the design would be forwarded to the production group, or the stress analysis group for further evaluation and analysis. Maybe data from a preliminary prototype would be collected and an evaluation report made. All of this took time and increased the cost of the item to be manufactured.

Today, the engineer must assume responsibility for design and functional capability early on in the design process, if for no other reason than costs associated with diverse channels of critiquing. For that reason, one place to enhance the design process is to have the ability to do preliminary stress and thermal analysis during the initial design phase. This is both cost and time effective. But, as I am sure you are aware, this has been easier said than done.

Four programs have been selected which address the problems found in the interface area, and how well they handle geometry from PRO-E. These selected vendors have been involved with Sandia Labs to further develop the interfaces between themselves and PRO-E and hopefully reach the goal of a truly transparent process, where all one would do is export the geometry, add the boundary and load parameters, pick an icon or push a button. Surprising, perhaps, is the fact that two of the vendors are old time PC advocates for stress and thermal software who see the potential for workstation applications. All of the vendors are committed to a product that will be of tremendous benefit to the engineers who want to assume more design responsibility. There are undoubtedly other vendors for this type of software whom I have overlooked, but time

constraints dictated that I concentrate on a few rather than a cursory glance at the majority.

In order to evaluate the different companies with different approaches to this transfer, a simple model was developed that was not too complicated in structure, was easy to mesh by hand, if need be, and had a straight forward look for applying loads and boundary conditions. Built into the model were several obvious flaws. As you can see in the bracket shown in fig. 1, there are no fillets, and where the gusset meets the plate, there is a sharp corner which should provide a high stress area. Normally, you would load this bracket in compression against the support member, but for clarity and to show potential problems, I am going to load it backwards to show stresses you would not normally see. Four vendor software products were evaluated and the results are now discussed.

The first software evaluated was the COSMOS/M Engineer program from Structural Research and Analysis Corporation. They are currently working on the final Beta version due to be released in late summer 1993. This vendor is well known in the Personal Computer arena for a very sophisticated product covering most aspects of stress and thermal analysis. Their approach is to offer a seamless interface to PRO ENGINEER. This interface allows the direct translation of the PRO-E model geometry with an identical command menu structure. Note in fig. 2, the correct geometry has been transferred through the use of the neutral file which is part of the PRO-E interface. With the release of Pro-E 11, there will be a COSMOS/M geometry file option for a more direct transfer. The option also exists to export the PRO-Mesh model along with the loads and boundary conditions. After transfer of the model to COSMOS/M, you start at the top of the menu and follow the command sequence, although it is not always crystal clear as to what the next step should be. Although I am not a stress analyst, I was able to figure out how to run the program with little guidance. COSMOS/M has one of the best meshers I have seen and when you issue the mesh command, you can watch the mesher at work on your part. I did not need to define surfaces or curves before meshing as the program seemed to know what the part was. Notice in fig. 3 that the mesh has a good aspect ratio and that it is an evenly distributed Tetrahedral mesh. You have the option of either a 4 noded tetrahedral, a 10 noded tetrahedral or a more sophisticated 4 noded tetrahedral with rotational degrees of freedom. Not many stress analysts like the tetrahedral elements, but in the current library of auto meshers, nothing else is currently available. There is some work being done to develop 3-D quad pavers, but nothing available at this time. I recommend the use of the 10 noded tet for the result accuracy even though the solution time is much longer in comparison to the 4 noded tet. Meshing this part on a IRIS Indigo R4000, takes about 90 seconds. Released versions of the software will have the ability to specify a variable mesh density at different locations rather than having a fixed overall mesh size. This will optimize the number of the total elements in the model, so the most accurate results in the high stress area can be achieved with minimum solution time. Applying loads and boundary conditions requires you locate the correct regions, curves, or surfaces, but you need to change colors manually (fig. 4) on your different loads and constraints so that locating the correct areas after applying these

conditions can easily be seen. This is one minor detail that still needs some refining. Much of the needed work revolves around the matching of the GEOSTAR command structure to the Interface structure, and these changes should show up in the released version.

COSMOS/M Engineer has a large variety of post processing features for showing results (figs. 5,6) and I have opted to show a few of them. Creating multiple windows is not hard to do and enhances the display allowing comparisons in a snap. There is the capability to create Postscript and HPGL files for most of the printers, although these slides are created with the SILICON GRAPHICS 'Image Snap' feature. The total time from exportation of the PRO-E file to viewable results in windows for this model was about 30 minutes.

The next software package that I evaluated was the APPLIED STRUCTURE from the RASNA Corporation. RASNA was started several years ago with the goal of creating optimization and performance analysis tools geared specifically for the mechanical design engineer. They focused on ease of use, speed, and the ability to do optimization. Their products use a very tight code and takes less than 40 megabytes of disc space without sacrificing power.

The interface RASNA has developed for PRO-E is very good. When you are ready to transfer your part, all that is required of you is to click on an icon, and the part is transferred to the APPLIED STRUCTURE program via a neutral file specific to RASNA. The PRO screen disappears, and the APPLIED STRUCTURE screen appears with the part following in seconds (fig. 7).

Their product is easy to use once you master the idea of the 'P' element. APPLIED STRUCTURE is similar to other FEA type products in that the geometry must be split up into a mesh before analysis. However, their use of the P-method ,which uses higher order mathematics, allows the designer to use far fewer elements that map directly to the geometry. This speeds up model meshing dramatically. I have always contended that manually having to mesh a part probably keeps more engineers from delving into stress analysis than any other single item. My bracket (fig. 8), after meshing, contained 21 elements and all of the rounds and radii were accurately accounted for. Loads and boundary conditions (fig. 9) are easily applied to either the elements or the geometry. Notice the use of clean, simple symbols for depicting loads and boundary conditions. One nice feature is the ability to apply an accurate bearing load to a curved surface. Many programs require you to apply a pressure rather than a force, and if you follow that method, you might not get the right answer. For instance, applying a bearing load in the hole shown on the bracket was easy. However, to keep the tests consistent, I did not apply it here, as some of the candidates did not have the ability to handle sinusoidal loads correctly. After finishing the loads and boundary conditions, I used the RASNA material library to find the material I wanted and after applying , I sent this problem to the solver. The solution accuracy is controlled by the user who specifies the fidelity of the answer before the solution is carried out, a useful feature is you only want approximate answers

with out a long wait. In this case, I opted for a normal answer knowing the part is not that complicated. After only 6 minutes, I had results. Not bad! Less than one half hour from exporting from PRO-E to plotted results on the screen.

As an engineering tool the software seems to be very accurate and give answers in a reasonable amount of time. The accuracy issue was already touched upon by mentioning that P-element map directly to the geometry. There is no geometry approximation. The results (fig. 10) from solutions for the bracket show very good answers when compared against a baseline. The higher order math makes analysis relatively foolproof by allowing the software to go out and find problem areas regardless of the mesh density as can be seen in the stress plot. Without putting a radius at the joining of the gusset to the bottom plate, an abnormally high stress occurs.

Once the part performance is understood you can play around with the part and optimize it, a feature unique to RASNA'S products. These sensitivity and optimization studies are easily set up if you want the program to give you the ideal part design.

One thing you must be careful about is the different methods of applying forces, pressures, etc., since units can be different from program to program. For instance, when applying forces in RASNA, you are given the option of total versus incidental forces to an edge or surface. This feature is not always obvious in some programs and may cause you to apply the wrong loads if you are not careful.

The third software candidate evaluated is from ALGOR, a very well established PC -based stress and thermal analysis company that manufactures mechanical design optimization software in which a full range of engineering analysis is tightly integrated. ALGOR software is used around the world and is, by far, the largest distributor of analysis software. ALGOR is making a major thrust into the workstation market with a just released version similar to their popular PC software, except it has a more closely controlled interface to PRO-E. ALGOR'S MENU-PAK for PRO-E (just released) enables engineers to design parts in PRO, and without leaving, they can utilize ALGOR'S design optimization and FEA software. These programs interface at five basic design engineering levels: NURBS surface, CAD wireframe, surface mesh, stereolithography, and tetrahedral mesh. Again, all that is required to export the model from PRO-E is to create an IGES 128 file, and this is done automatically when you click on an ALGOR icon (fig.11). Algor has created a very exciting new interface and guide program called the "ROADMAP", for following the needed sequence to obtain solutions. The integrated ROADMAP introduces a new method in human interfacing, allowing casual users an on screen recipe prompting them through the analysis. From what I have seen of it so far, it is indeed very much oriented toward the casual user and details the exact procedure to follow depending on the solution requested. People more familiar with ALGOR will immediately feel at home with the X-window motif (fig. 12) that is used in the Unix environment. ALGOR has a program called HYPER-GEN which is used to create the tetrahedral mesh on the part you have imported to it. This is a very fast automesher and created the mesh in seconds on the my test part. All of this activity takes place within the SUPERDRAW II program, so

attaching loads and boundary conditions (fig. 13) is very easy. It is exciting to watch the part come over, the correct Xwindow being displayed, and the mesh created without you doing a thing other than locate the windows where you want them. Of course, if you do not want the tetrahedral mesh, the option exists, while you are in SUPERDRAW, to manually mesh the part with normal elements from ALGOR'S regular programs.

After submitting the problem to the solver, one of the more useful features ALGOR presents you with is a window to monitor solution progress, a feature that is carried over from the PC environment. This is one feature I like very much as it allows you to monitor progress through the solution cycle and if you tend to watch the progress of the solver like I do, it is easy to tell when the solution process dies, where otherwise you may not know if something went wrong for quite a while. With complex models often taking hours of time for results to appear, this is an important feature. In this case though, the solution time is just under 6 minutes and total time from importing the file to viewing results is less than 20 minutes overall. The results are brought up in SUPERVIEW, just like you do in the PC version and then request the proper results. Notice in fig. 14&15, that you have a multitude of options for plotting varied solutions. In this case I have asked for the Von Mises stress output (fig. 14), the displacement plot (fig. 15), and the displaced vs. non-displaced overlay (fig. 16).

One of the newest options in SUPERVIEW, is a feature called 'Slice'. If you want to see the inside of a can, for instance, use the 'slice' feature, and you have the part showing what is not sliced away. About as easy as cutting a apple in two and just about as quick. Another great option, which is available through out ALGOR, is the 'Jetview' feature, which selects a dynamically rotated view based on the direction of a pointed arrow that you manipulate on the screen.

The fourth software package evaluated was PATRAN. PATRAN has been around longer than most stress analysis programs and is one of the most powerful programs in terms of various solvers, i.e., linear, non-linear, thermal, vibration, etc., etc. It is also one of the largest users of disc space. Do not attempt to use this program without a lot of storage and a fast machine, unless time is not a factor. Its size approaches 150 megabytes of assorted modules. On the other hand, there is not much the program won't handle, and there lies the problem if you can call it that. You should be very knowledgeable in the field of stress and thermal analysis before you attempt to use this program. In the past, its use was devoted to analysts and only the mechanical or design engineers skilled in analysis attempted its use. Recently, attempts have been made to make the program more friendly with an interface to the PRO-E program. The interface seems to require more effort on the part of the engineer to invoke when compared to other vendors products, but once there, you can import the part via several means (fig. 17) from PRO-E to PATRAN, and then recreate the solid geometry, or you can import the PRO-Meshed model and attach loads and boundary conditions to your part in PATRAN, since it has the capability of solving either brick, shell, or tetrahedral elements. The PATRAN people have an agreement with PARAMETRIC TECHNOLOGY to use the prt. file, but in this

case the IGES file worked as well. In fact, once in PATRAN, you can pick the prt. file from a menu pick and have it displayed immediately.

The quickest way, by far, is to use PRO-Mesh, assuming of course that you have it on your computer, and I would highly recommend this method rather than creating elements yourself. A colleague of mine offered to run the bracket part on his SILICON GRAPHICS IRIS R-4000 since he had the PATRAN license. After importing the IGES file, he opted to do the meshing manually and this took some time. In fact, he had to cut the model in half to decrease the meshing and solve time. Meshing this part with a fine mesh took 1 1/2 hours and then the solving time was also about 1 1/2 hours. Had he used the PRO-Mesh feature, that would have decreased the meshing time. When observing the results, (fig. 18) you can see the increase in definition for stresses is due to the increase in mesh density, but the increase in definition does not come for free.

As I have mentioned, my colleagues are using this program and enjoy it but find the meshing to be very time consuming. Again, I stress that neither I nor some of my colleagues are stress analysis experts and are not as familiar with Patran as we could be.

In conclusion, if you want to start using your workstation computer's vast capabilities for problem solving, and you can master the needed minimal skills required to run these new stress and thermal analysis programs, you will find your productivity increase along with the time you save, to become major components of your work day. Even with the longest time measured in my tests, that is still less than if I had sent the part to the Company's stress and analysis group, and much less expense. There is no reason to bog down the regular stress analysts with simple problems, and if you do enough of these simple problems, the cost of the software will soon pay for itself.

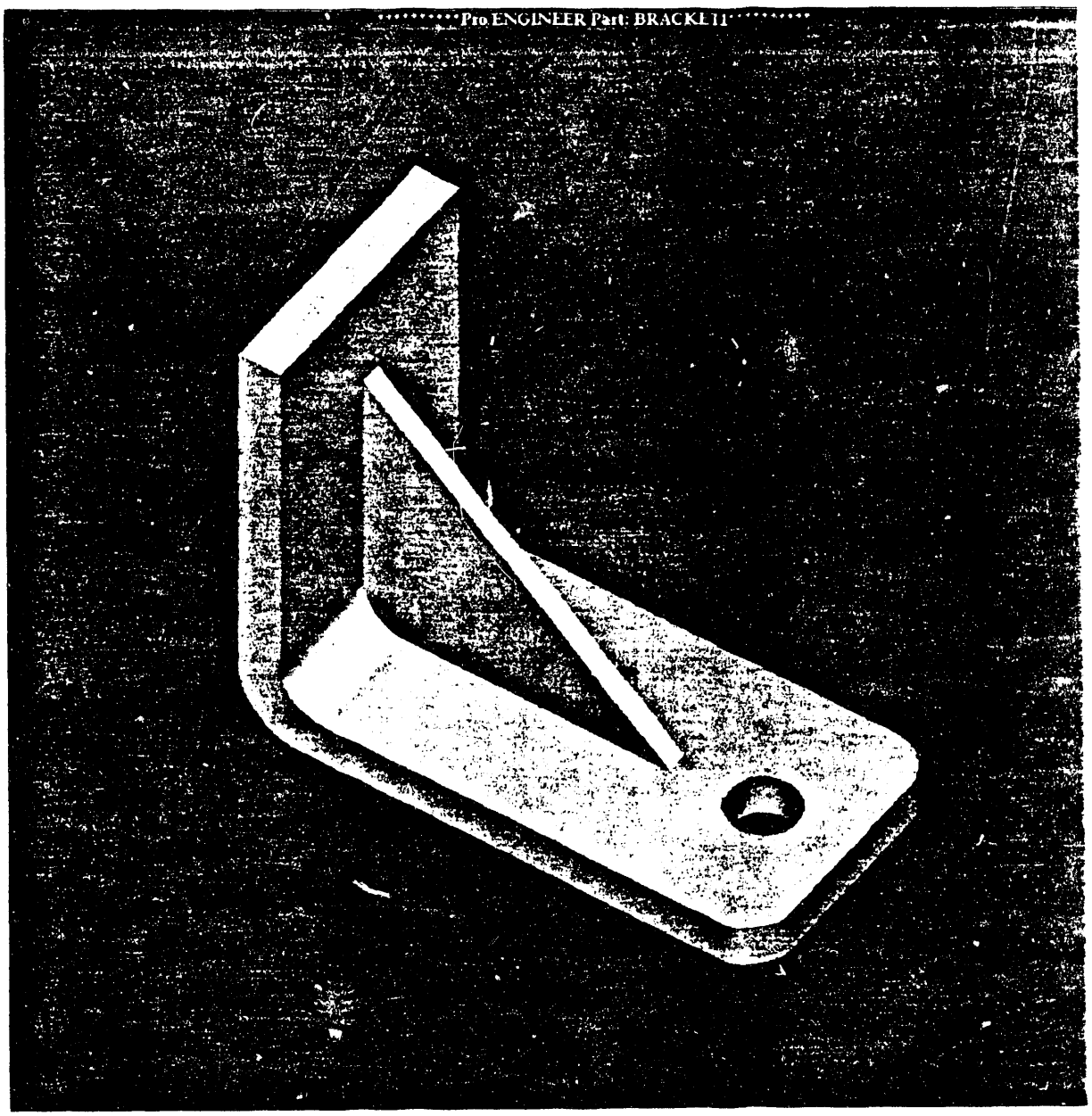


FIGURE 1

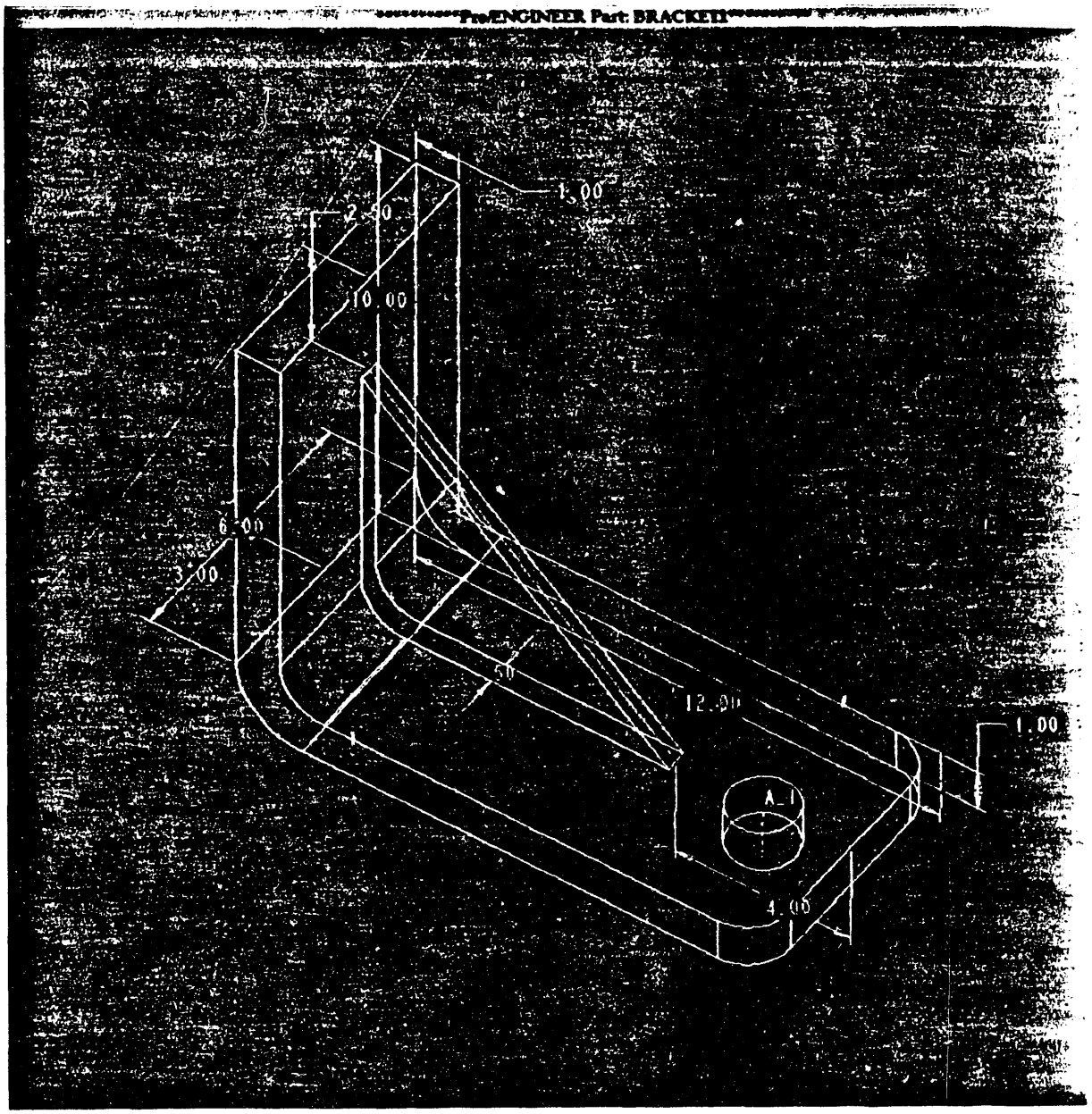

ProENGINEER Part: BRACKET1
1.00
2.00
10.00
6.00
3.00
.50
12.00
1.00
A_1
4.00

FIGURE 1A

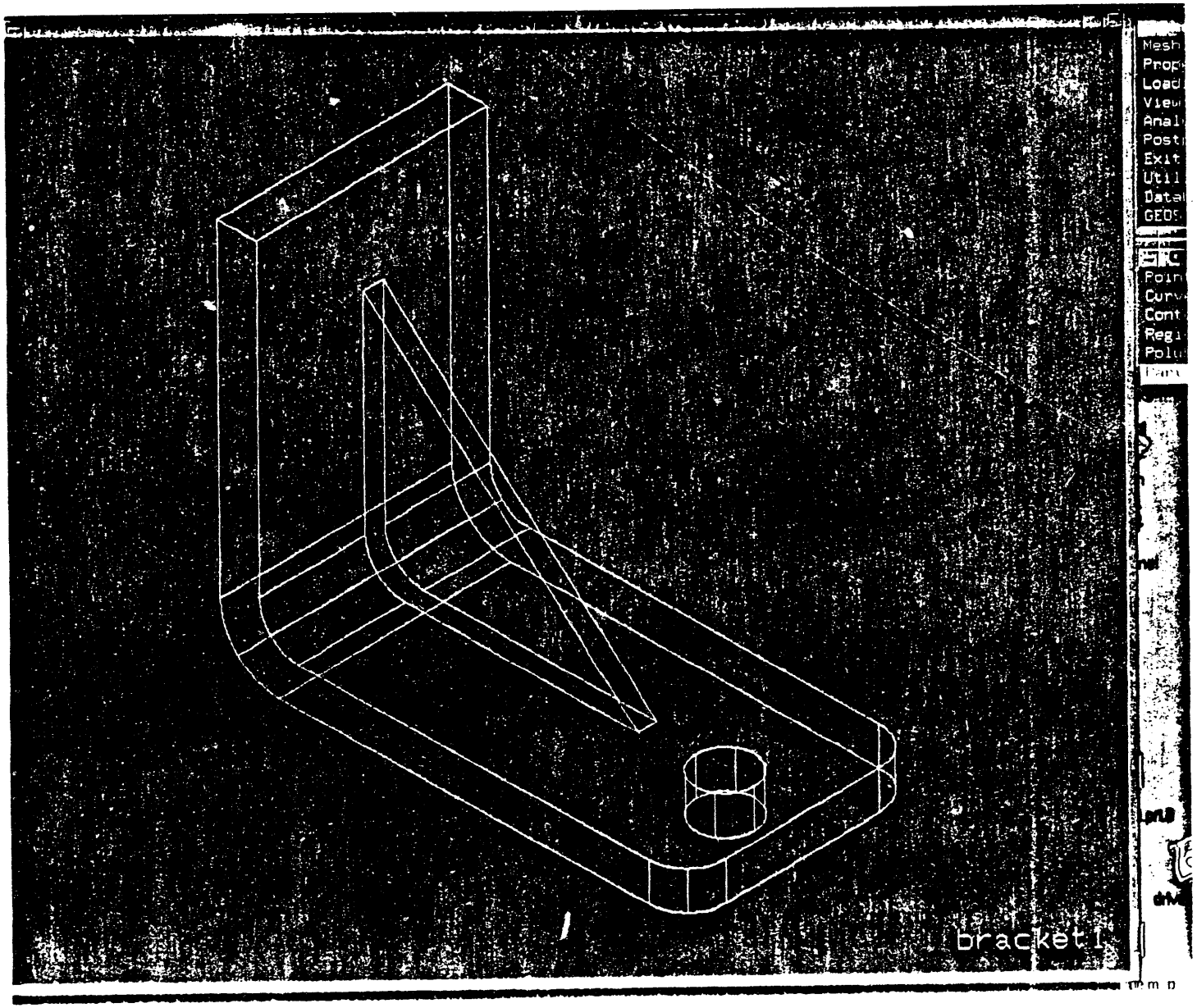


FIGURE 2

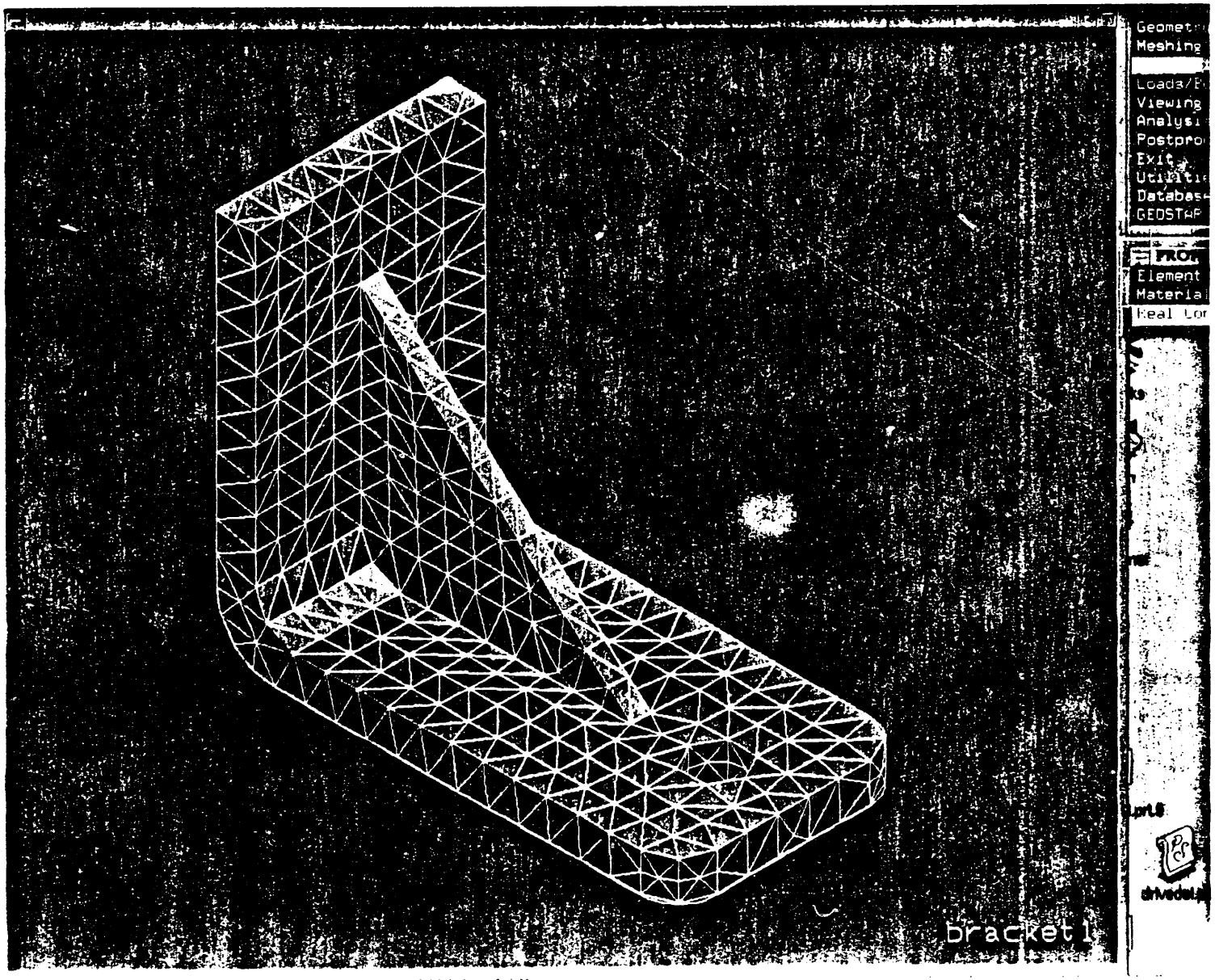

Geomet
Meshing
Loads/B
Viewing
Analysi
Postpro
Exit
Utilit
Databas
GEDSTAR
PRO/
Element
Materia
Real Con
bracket

FIGURE 3

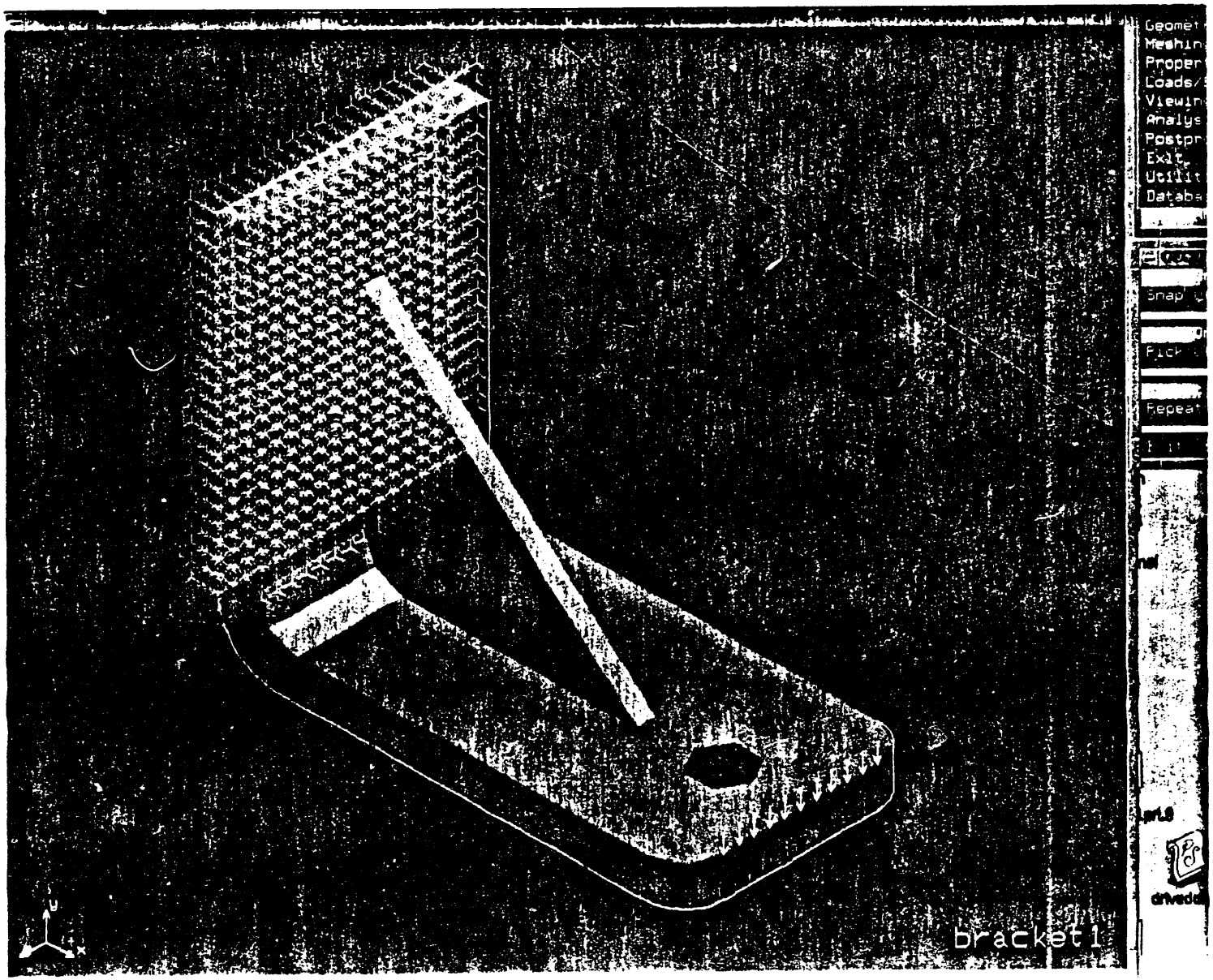

Geomet
Meshin
Proper
Loads/
Viewin
Analys
Postpr
Exit
Utilit
Databa
Snap
Pick
Repeat
bracket1

FIGURE 4

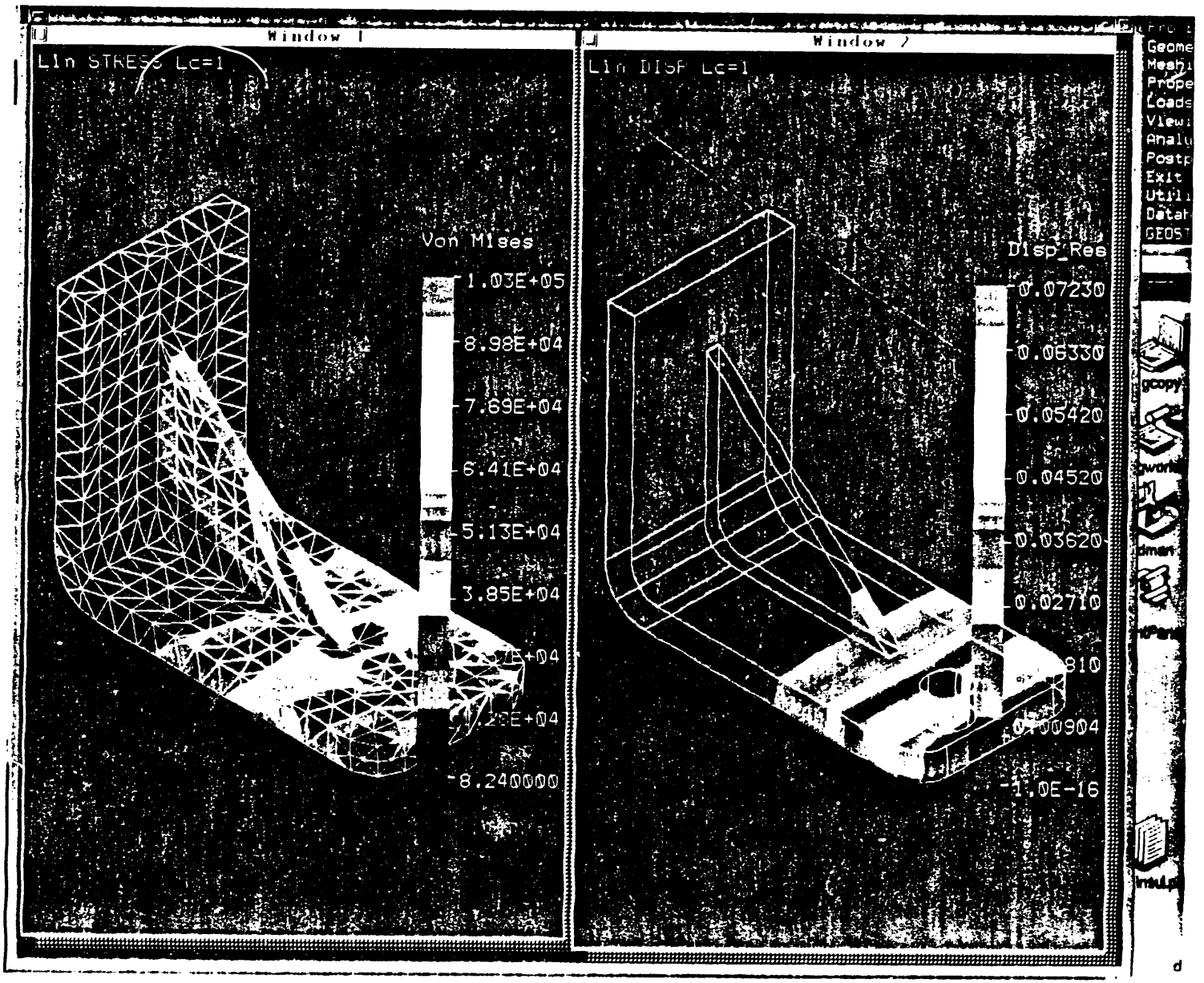

Window 1
Lin STRESS Lc=1
Von Mises
1.03E+05
8.98E+04
7.69E+04
6.41E+04
5.13E+04
3.85E+04
8.240000
Window 2
Lin DISP Lc=1
Disp_Res
0.07230
0.06330
0.05420
0.04520
0.03620
0.02710
1.0E-16

FIGURE 5

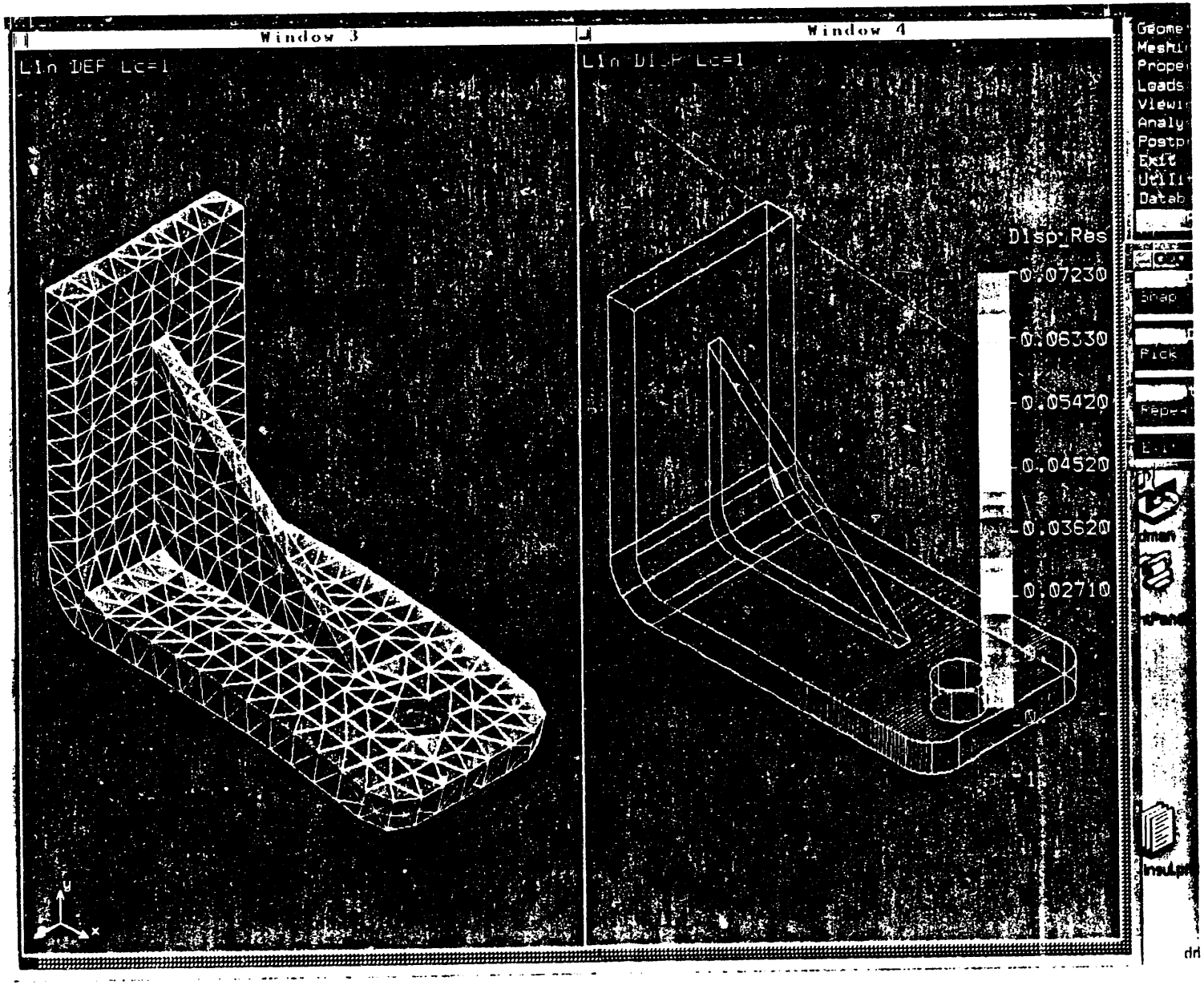

Window 3
Lin DEF Lc=1
Window 4
Lin DISP Lc=1
Disp_Res
0.07230
0.06330
0.05420
0.04520
0.03620
0.02710
Geome
Meshi
Prope
Loads
Viewi
Analy
Postp
Exit
Utili
Datab
Snap
Pick

FIGURE 6

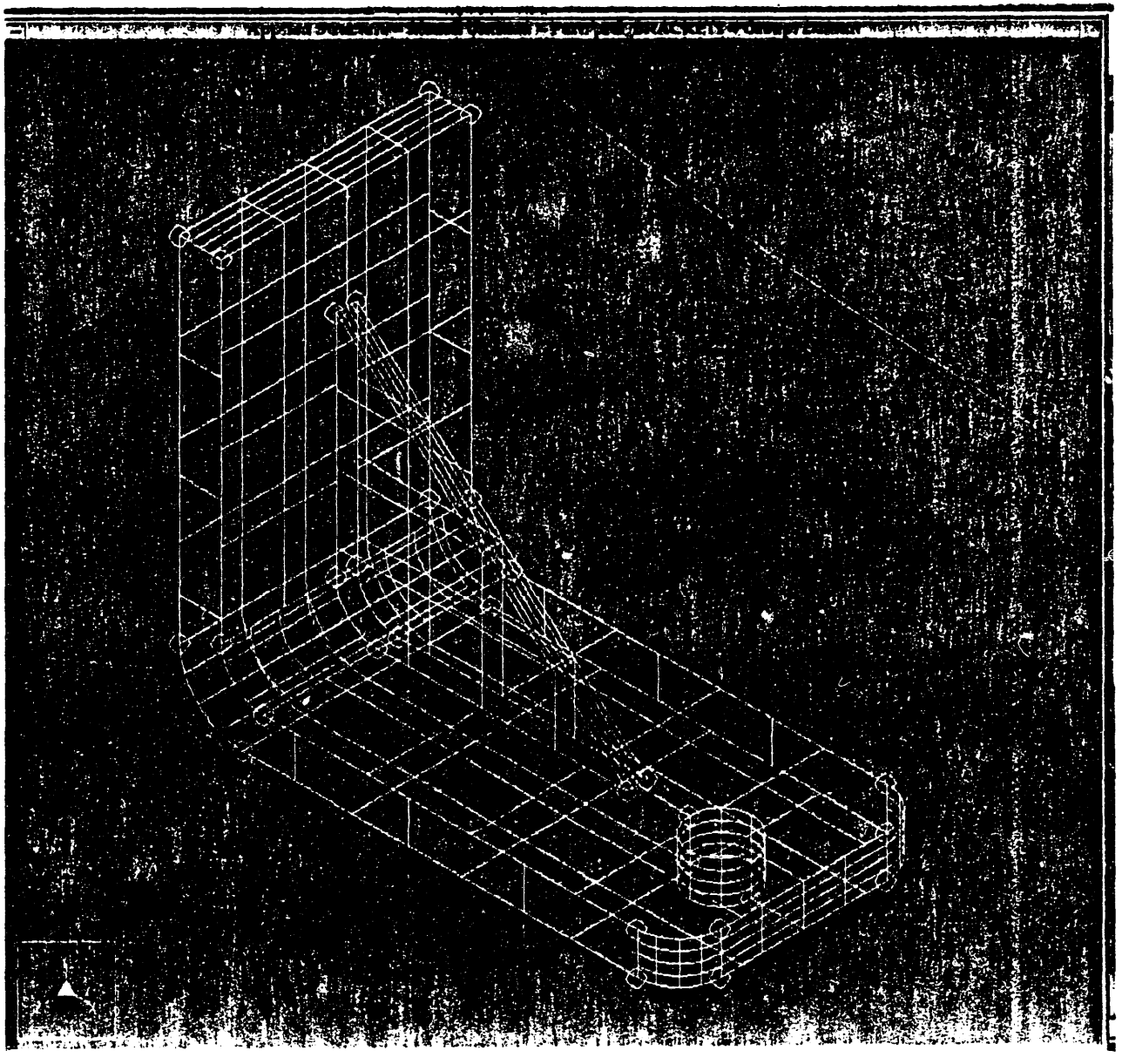


FIGURE 7

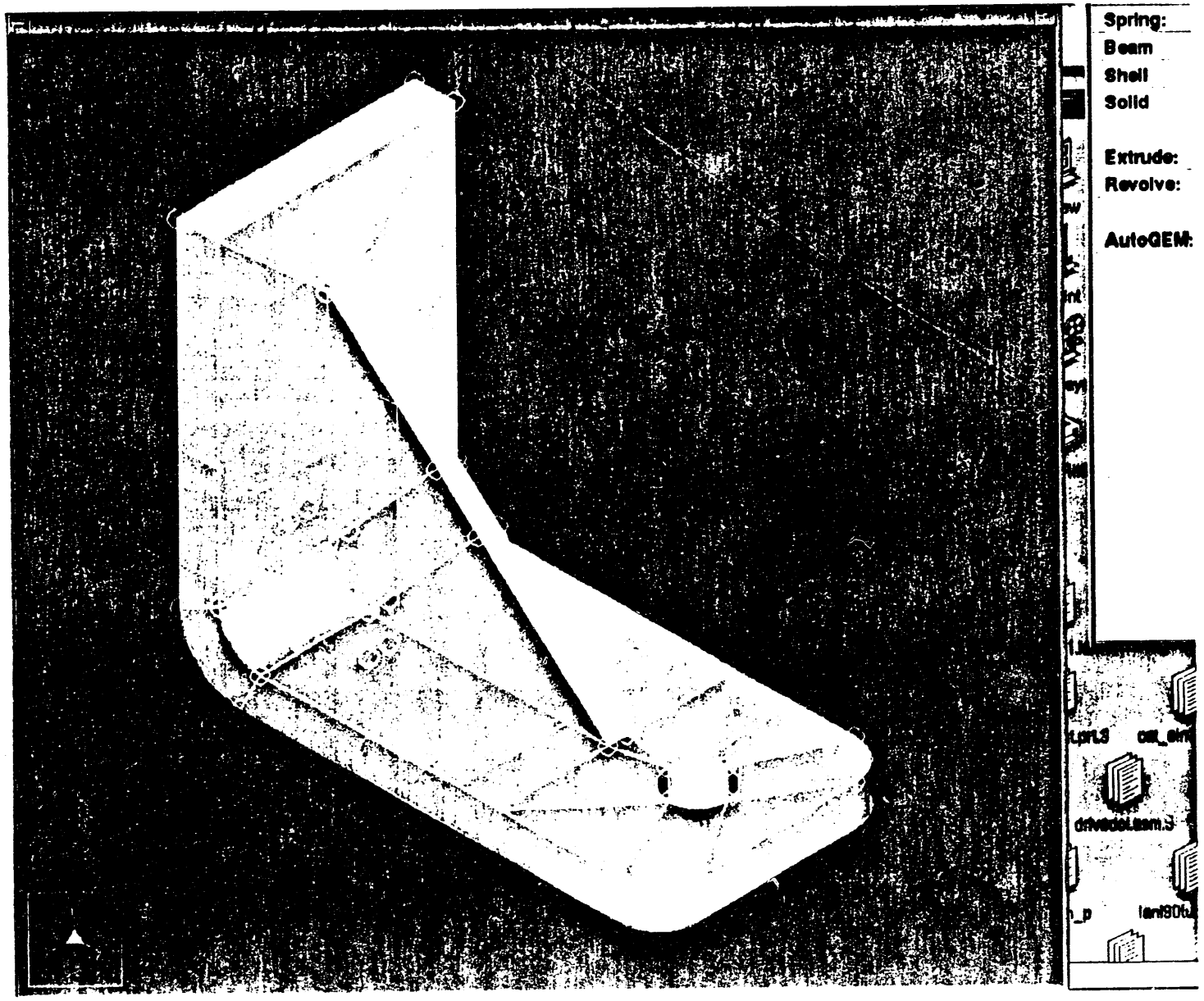


FIGURE 8

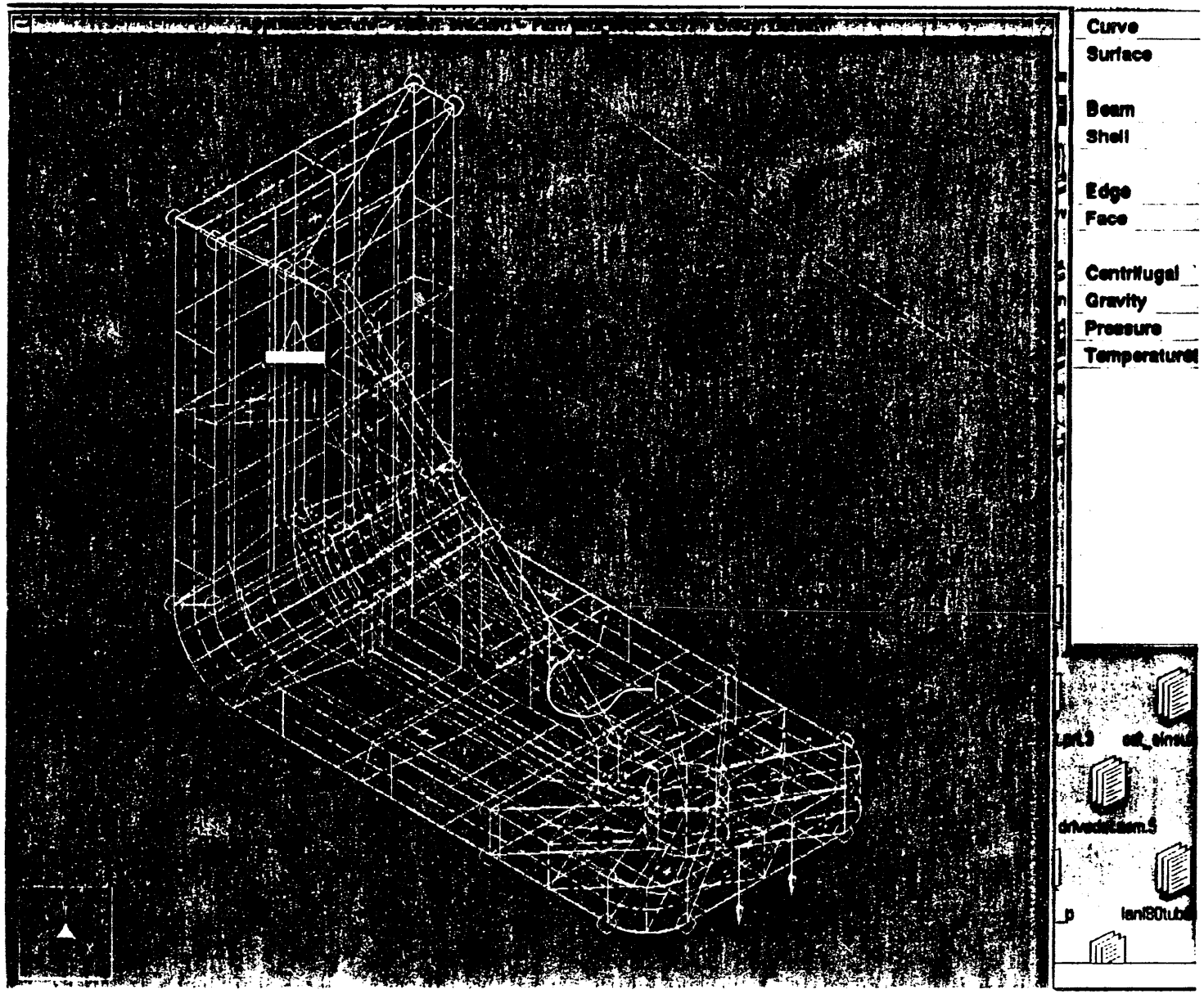

Curve
Surface
Beam
Shell
Edge
Face
Centrifugal
Gravity
Pressure
Temperature

FIGURE 9

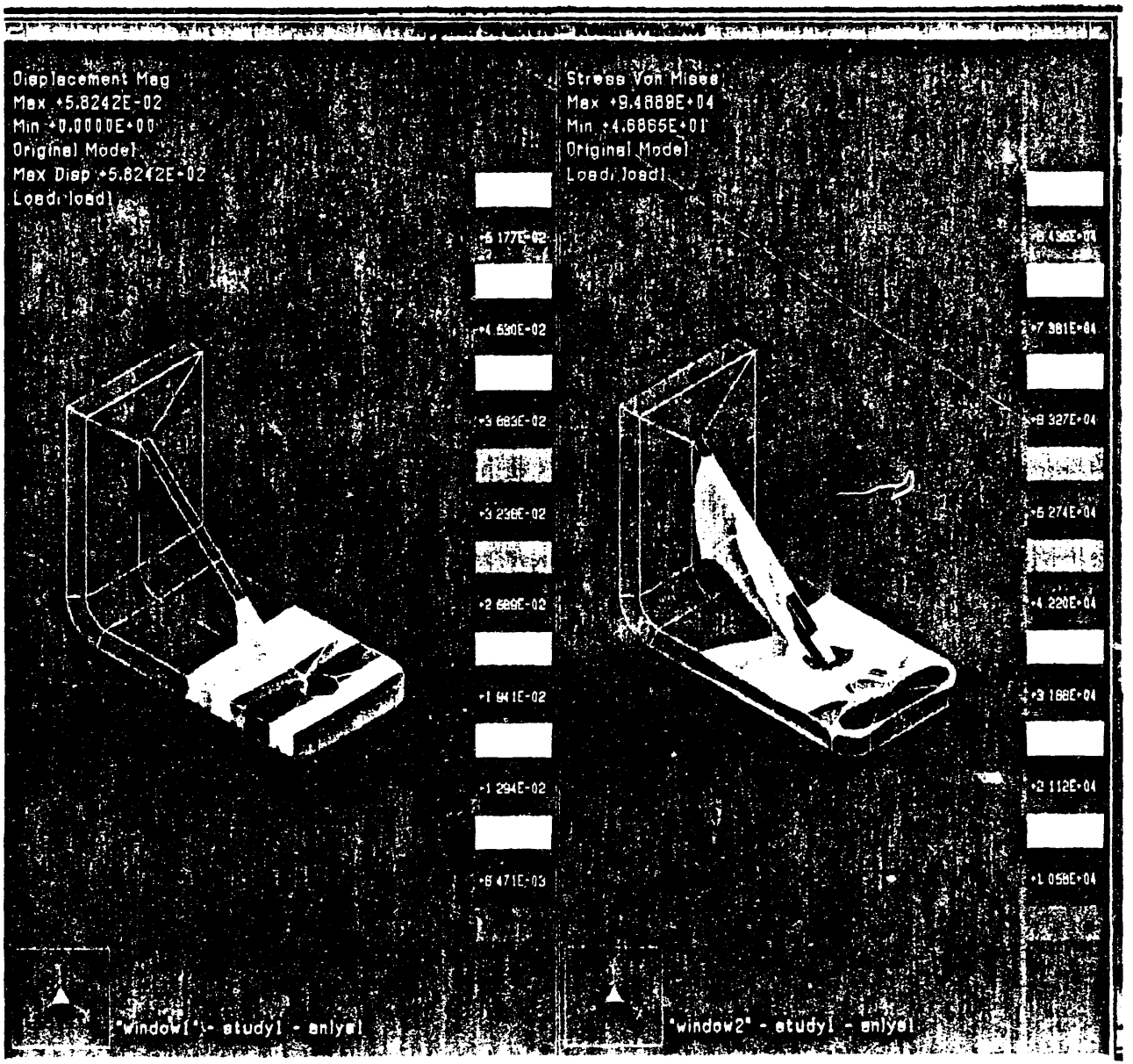

Displacement Mag
Max +5.8242E-02
Min +0.0000E+00
Original Model
Max Disp +5.8242E-02
Load:load1
+5.177E-02
+4.530E-02
+3.883E-02
+3.236E-02
+2.589E-02
+1.941E-02
+1.294E-02
+6.471E-03
"window1"- study1 - anlys1
Stress Von Mises
Max +9.4889E+04
Min +4.6865E+01
Original Model
Load:load1
+8.436E+04
+7.381E+04
+6.327E+04
+5.274E+04
+4.220E+04
+3.166E+04
+2.112E+04
+1.059E+04
"window2" - study1 - anlys1

FIGURE 10

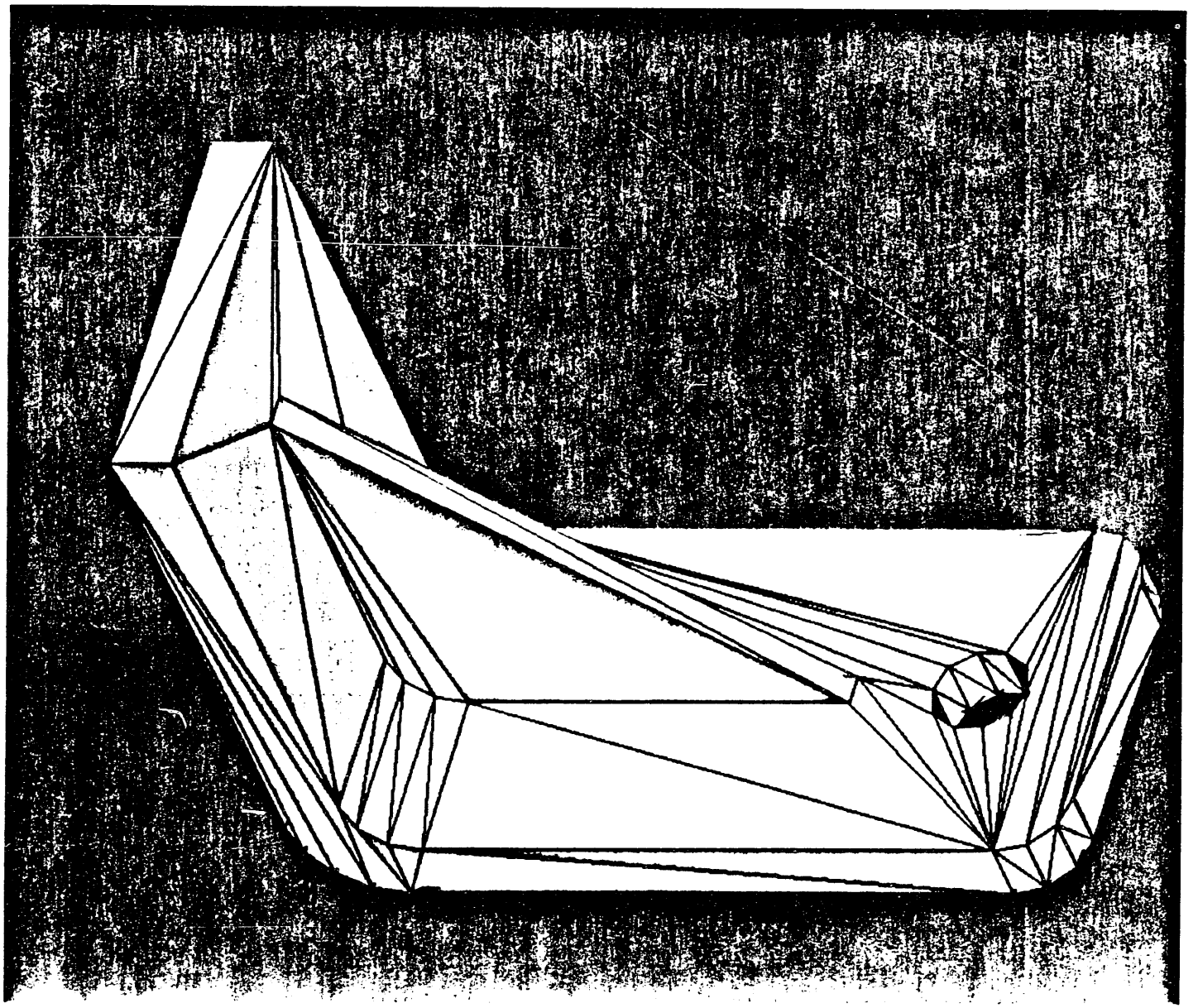


FIGURE 11

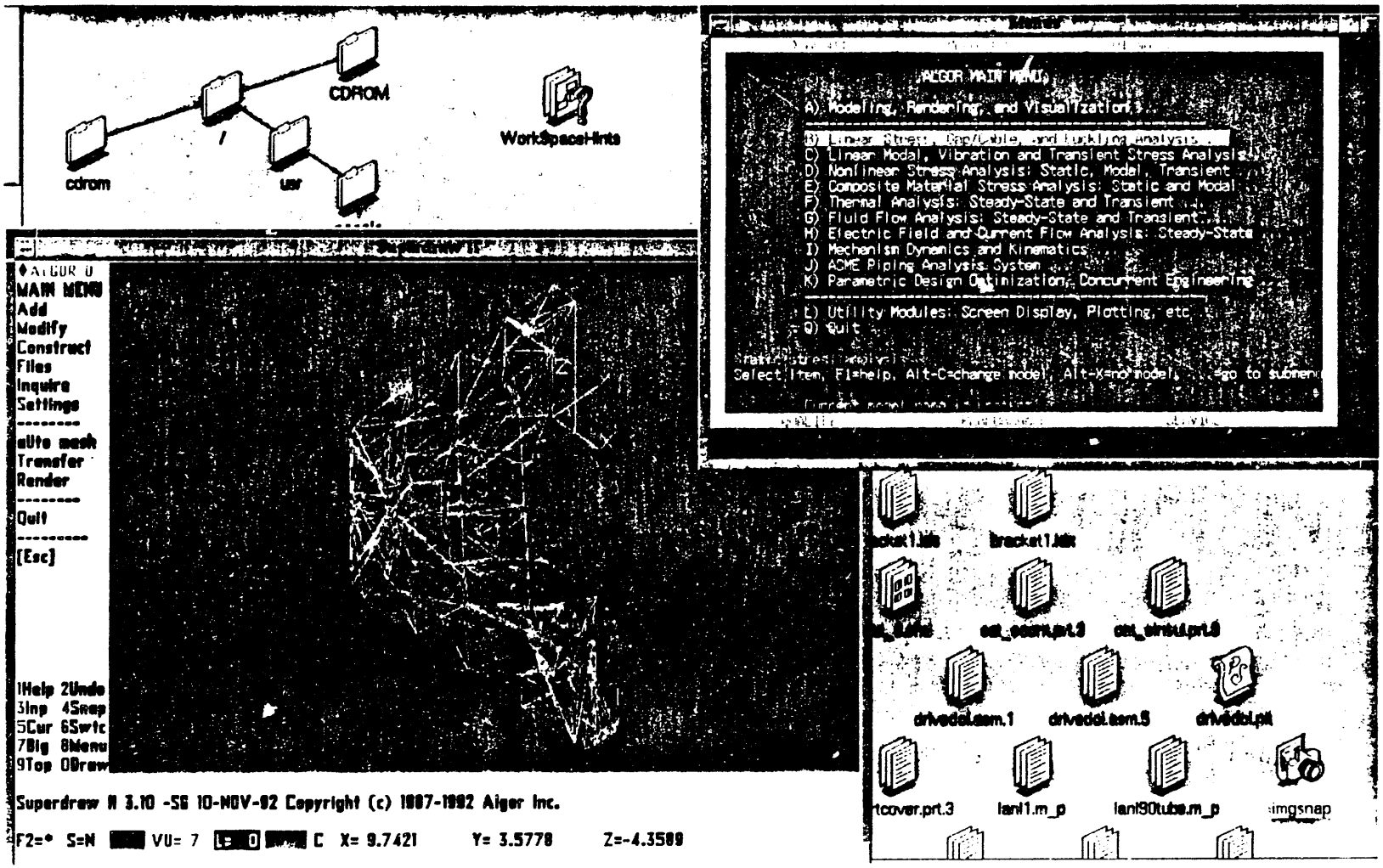
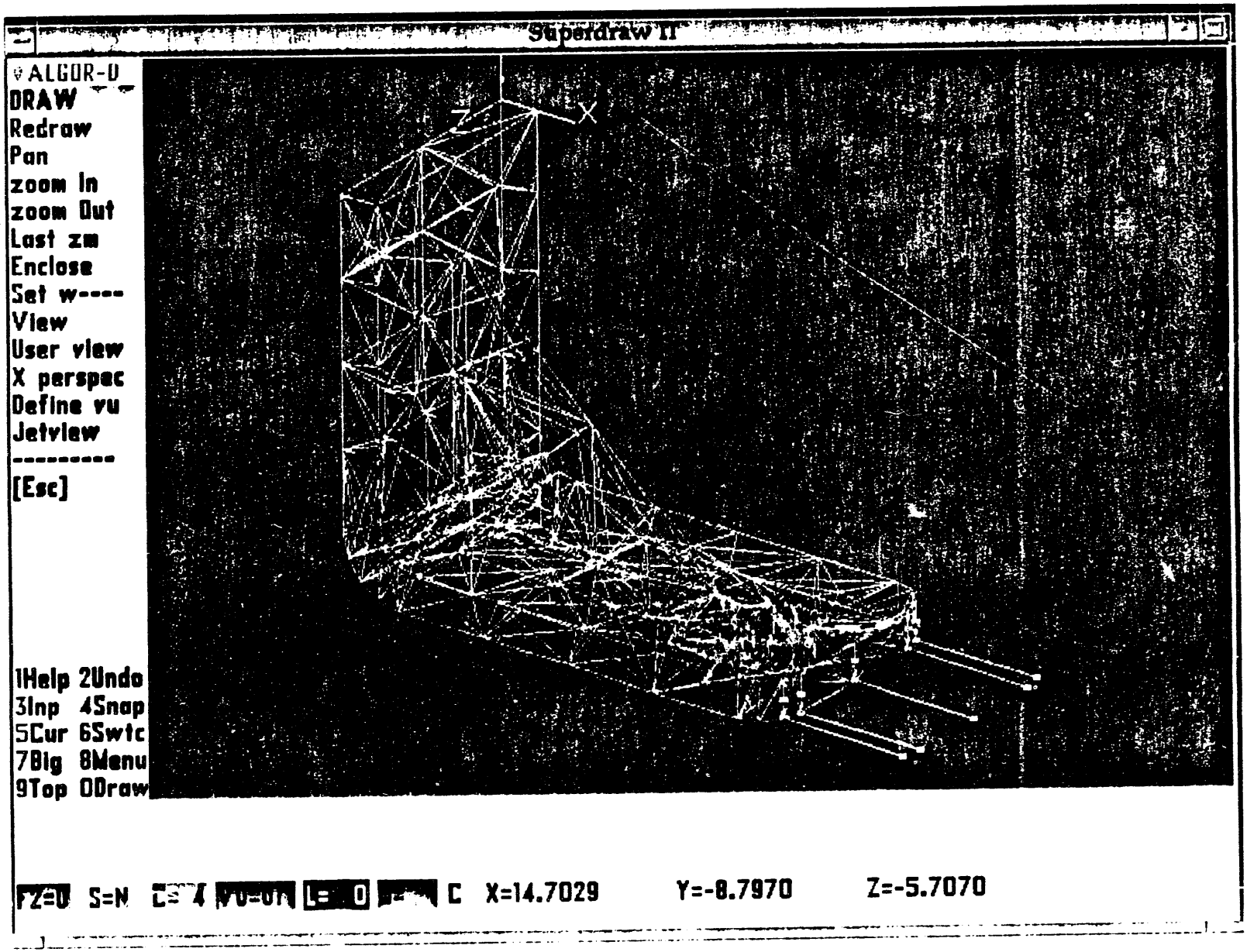

CDROM
cdrom
usr
WorkSpace4-links
A) Modeling, Rendering, and Visualization
B) Linear Stress, Gap/Cable, and Buckling Analysis
C) Linear Modal, Vibration and Transient Stress Analysis
D) Nonlinear Stress Analysis: Static, Modal, Transient
E) Composite Material Stress Analysis: Static and Modal
F) Thermal Analysis: Steady-State and Transient
G) Fluid Flow Analysis: Steady-State and Transient
H) Electric Field and Current Flow Analysis: Steady-State
I) Mechanism Dynamics and Kinematics
J) ACME Piping Analysis System
K) Parametric Design Optimization, Concurrent Engineering
L) Utility Modules: Screen Display, Plotting, etc
Q) Quit
Select Item, F1=help, Alt-C=change model, Alt-X=no model, ...go to submenu
MAIN MENU
Add
Modify
Construct
Files
Inquire
Settings
Mesh
Transfer
Render
Quit
[Esc]
1Help 2Undo
3Inp 4Snap
5Cur 6Swtc
7Big 8Menu
9Top 0Draw
Superdraw N 3.10 -SG 10-NOV-92 Copyright (c) 1987-1992 Aigor Inc.
F2=* S=N VU= 7 C X= 9.7421 Y= 3.5778 Z=-4.3589
bracket1.idb
bracket1.stk
drivedol.asm.1
drivedol.asm.5
drivedol.prt
tcover.prt.3
lani1.m_p
lani90tube.m_p
imgsnap

FIGURE 12


Superdraw II
ALGOR-D
DRAW
Redraw
Pan
zoom In
zoom Out
Last zm
Enclose
Set w----
View
User view
X perspec
Define vu
Jetview

[Esc]
1Help 2Undo
3Inp 4Snap
5Cur 6Swtc
7Big 8Menu
9Top 0Draw
Z
X
C X=14.7029 Y=-8.7970 Z=-5.7070

FIGURE 13


SuperView
♦ ALGOR-V
STRESS-01
Hidden 1
Light
♦Post
Aux post
General
Ignd boX
♦Smoothed
Max obs
Bitmap
De dither
[Esc]
9860.6
9039.3
8218
7396.8
6575.3
5754
4932.7
4111.4
3290.1
2468.8
1647.5
826.22
4.916
Z
X
1Help 2Undo
3Inp 4Snap
5Cur 6Swtc
7Big 8Menu
9Top 0Draw
Dither method = Von Mises Stress
Min=4.91596, Max=9860.56
[Auto range] [No frills]
SVIEW 4.02 File:bracket1 04/26/93 08:55 LC 1/ 1 Vu=01 Lo= 117 La= 42 R= 108

FIGURE 14

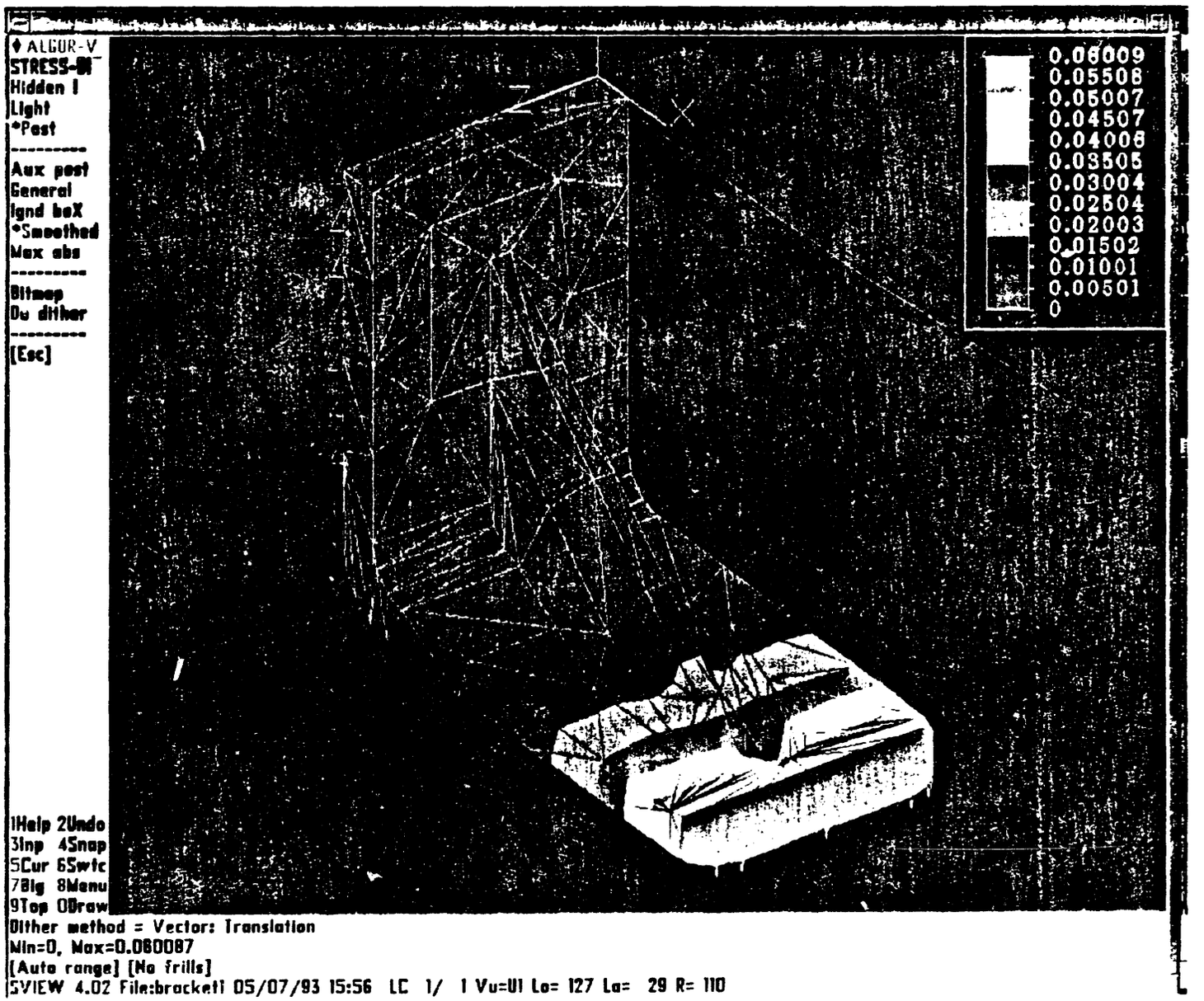
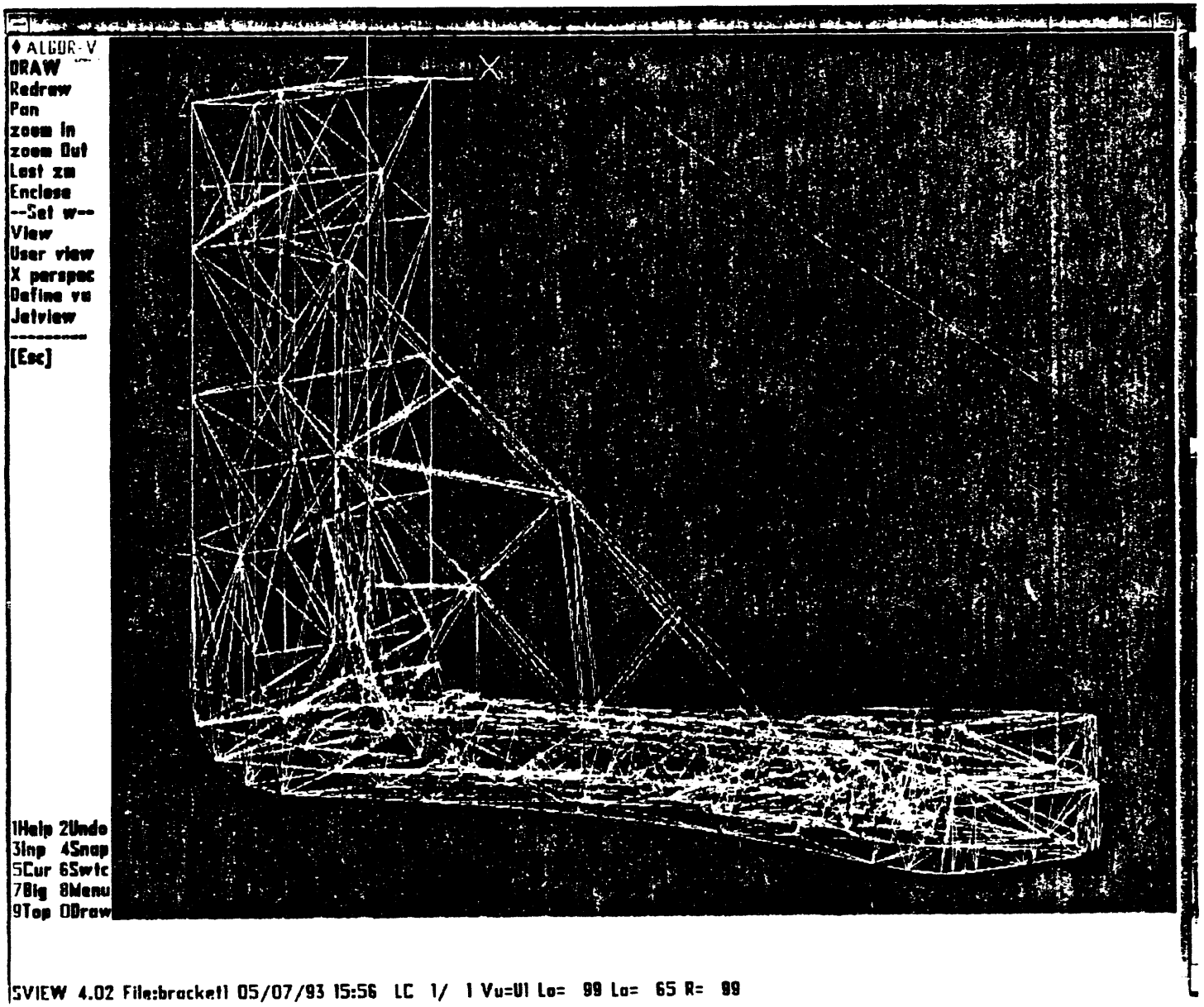

♦ ALGOR-V
STRESS-BI
Hidden l
Light
•Post
Aux post
General
Ignd boX
•Smoothed
Max abs
Bitmap
Ou dither
[Esc]
0.06009
0.05508
0.05007
0.04507
0.04006
0.03505
0.03004
0.02504
0.02003
0.01502
0.01001
0.00501
0
1Help 2Undo
3Inp 4Snap
5Cur 6Swtc
7Big 8Menu
9Top 0Draw
Dither method = Vector: Translation
Min=0, Max=0.060087
[Auto range] [No frills]
SVIEW 4.02 File:bracket1 05/07/93 15:56 LC 1/ 1 Vu=01 La= 127 La= 29 R= 110

FIGURE 15


♦ ALGOR-V
DRAW
Redraw
Pan
zoom in
zoom Out
Last zm
Enclose
--Set w--
View
User view
X perspec
Define vu
Jetview

[Esc]
Z
X
1Help 2Undo
3Inp 4Snap
5Cur 6Swtc
7Big 8Menu
9Top 0Draw
SVIEW 4.02 File:bracket1 05/07/93 15:56 LC 1/ 1 Vu=01 Lo= 99 Lo= 65 R= 99

FIGURE 16

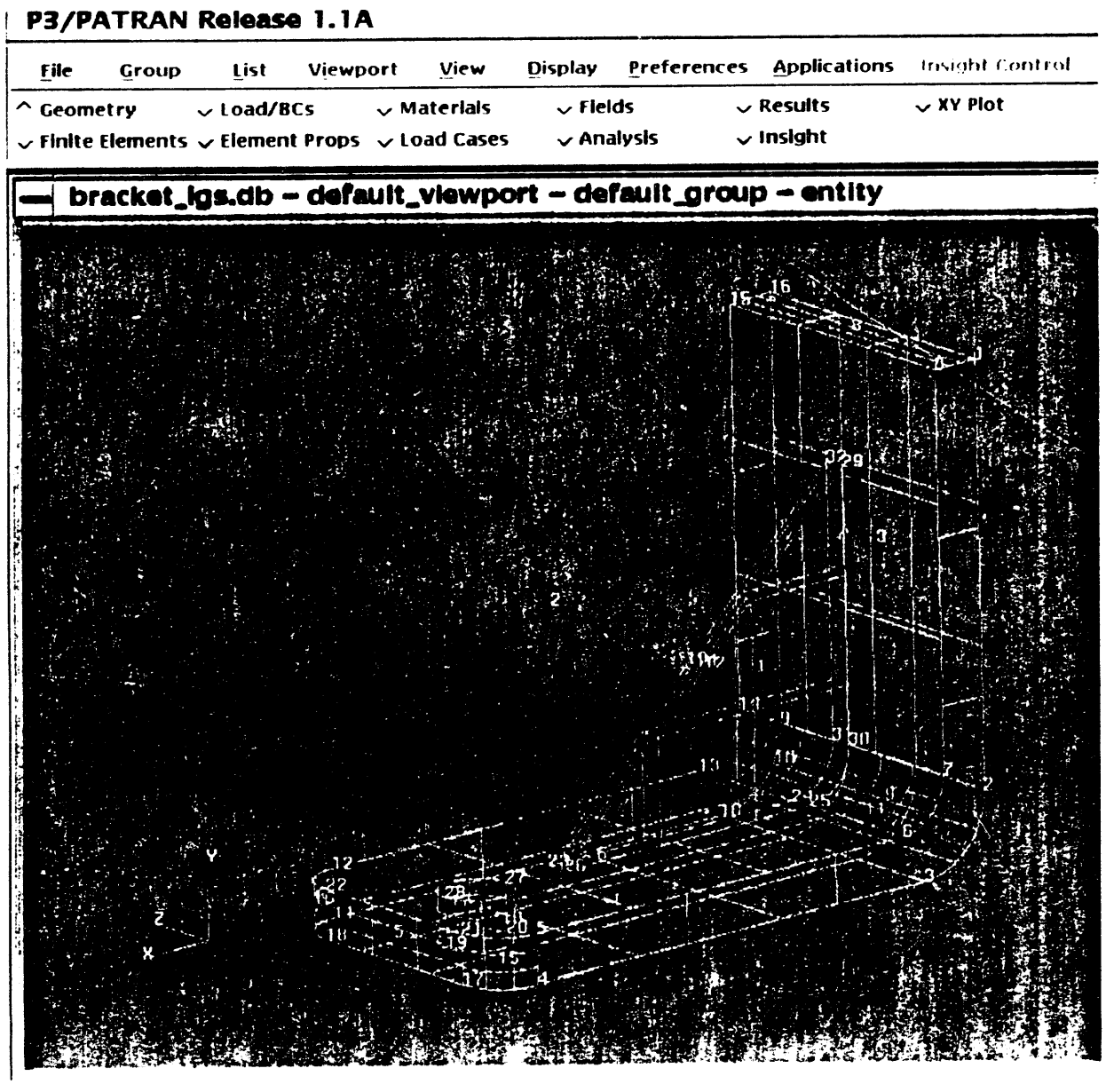

P3/PATRAN Release 1.1A
File
Group
List
Viewport
View
Display
Preferences
Applications
Insight Control
Geometry
Load/BCs
Materials
Fields
Results
XY Plot
Finite Elements
Element Props
Load Cases
Analysis
Insight
bracket_igs.db – default_viewport – default_group – entity
Y
Z
X

FIGURE 17

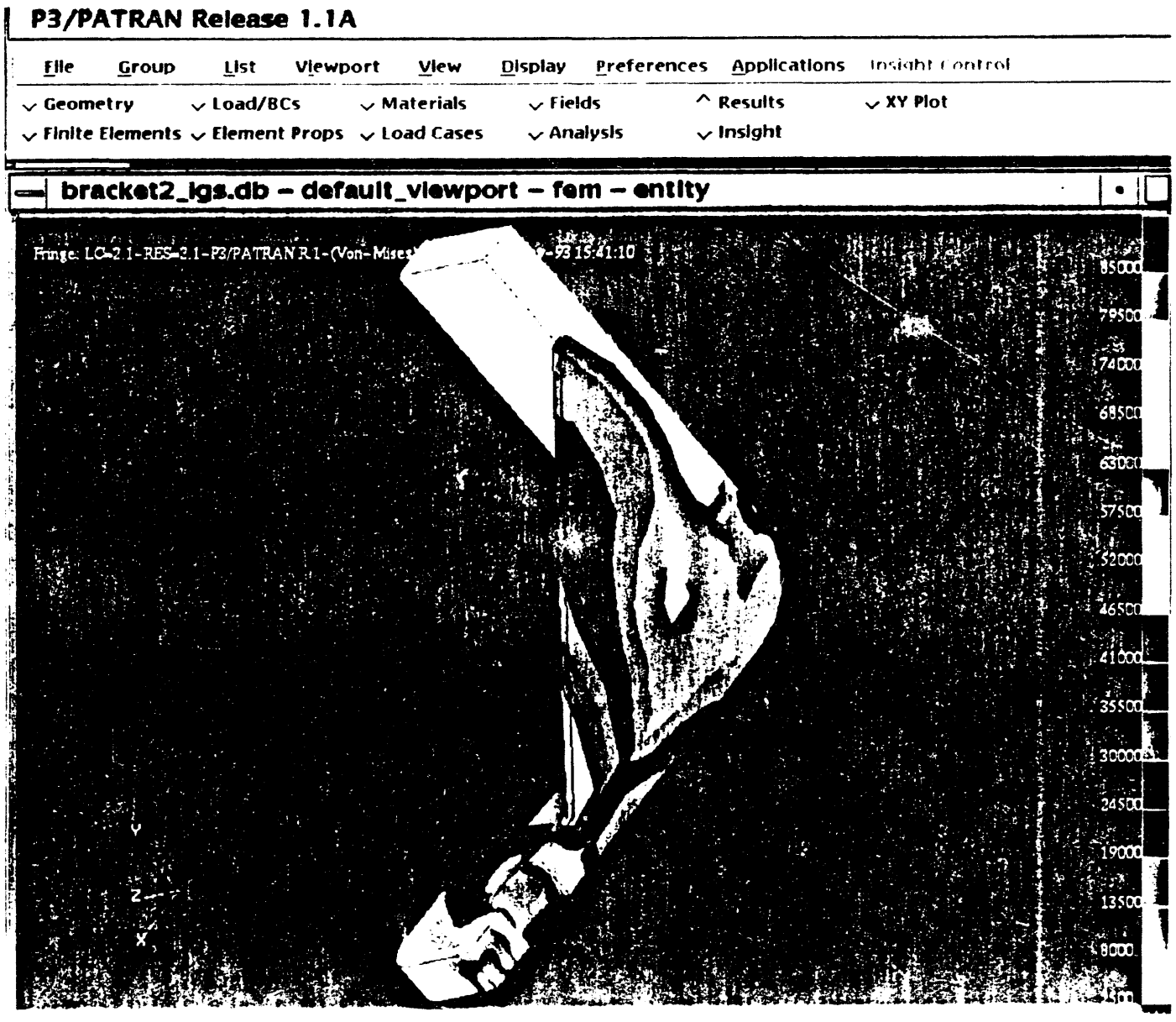

P3/PATRAN Release 1.1A
File
Group
List
Viewport
View
Display
Preferences
Applications
Insight Control
Geometry
Load/BCs
Materials
Fields
Results
XY Plot
Finite Elements
Element Props
Load Cases
Analysis
Insight
bracket2_igs.db – default_viewport – fem – entity
85000
79500
74000
68500
63000
57500
52000
46500
41000
35500
30000
24500
19000
13500
8000
Y
Z
X

FIGURE 18

END

DATE
FILMED
9/13/93