Commonwealth Edison
One First National Plaza, Chicago, Illinois
Address Reply to: Post Office Box 767
Chicago, Illinois 60690 - 0767

August 1, 1986

Mr. Harold R. Denton
U.S. Nuclear Regulatory Commission
Office of Nuclear Reactor Regulation
Washington, DC. 20555

Subject: Braidwood Station Unit 2
Reactor Vessel Nozzle Analysis
NRC Docket No. 50-457

Reference: April 2, 1986 A.D. Miosi letter to H.R. Denton

Dear Mr. Denton:

Enclosed is supplemental information covering three items which you require for review of the Braidwood Unit 2 Reactor Vessel Inlet Nozzle "F" indication as discussed with members of your staff.

The first item contained in Attachment 1 concerns a substantiation of the fracture toughness requirements found in Appendix G of ASME Section III in light of the proposed grindout. General Electric compared the hoop stresses in the nozzle section with and without the proposed grindout. The stress intensity factor, K_I, was calculated for both cases (one inch grindout vs. no grindout). Both of these comparisons yielded negligible differences from the original Appendix G analysis as performed by Babcock and Wilcox. Therefore, the original Appendix G analysis is still valid.

The second item, contained in Attachment 2, concerns itself with the validations performed by General Electric for the two compter programs utilized in the analysis addressed in the referenced letter. Three examples are included for the stress linearization program, STRDIS. Three sample problems and an information/program status sheet are included for the finite element program, ANSYS (approved for design use).

The final item, contained in Attachment 3, concerns itself with clarification of the location of the indication, shown in Figure 2, and the classification of the stress in the section analyzed. The governing section with the highest primary stress is the shell thickness and is classified as a local primary membrane stress (P_L). It was shown in the General Electric report that even with the 1 inch grindout, the area of reinforcement requirements are satisfied.

B001
1/1

Should you have any questions concerning this matter please contact this office.

One signed original and fifteen copies of this letter and attachments are provided for your review.

Very truly yours,

Anthony Miosi

A. D. Miosi
Nuclear Licensing Administrator

/klj
cc: J. Stevens

1941K

BRAIDWOOD UNIT 2 REACTOR INLET NOZZLE
APPENDIX G ANALYSIS

Background

The fracture toughness requirements for the Braidwood Unit 2 reactor inlet nozzle have been satisfied by demonstrating compliance with Appendix G of the ASME Code. Specifically, a nozzle corner flaw of 1 inch depth was postulated and pressure temperature curves were established to assure the safety margin requirements of Appendix G (Reference 1). However, because of a UT indication in the reactor vessel to nozzle weld, a local grindout repair of up to 1 inch depth has been proposed to remove the indications. The purpose of the analysis described here is to demonstrate that even with the grindout the nozzle still meets the original Appendix G requirements evaluated in [1].

Technical Approach

For the beltline region the heatup/cooldown events are limiting from the viewpoint of Appendix G analysis. However, the hydrotest is the governing case for the reactor nozzle and is therefore selected for analysis. The original Appendix G analysis for the nozzle postulated a one inch nozzle corner flaw. The minimum temperature for the hydrotest was determined corresponding to a safety margin of 1.5. The approach used here is to compare the hoop stresses in the nozzle section for the one inch grindout case with the corresponding stress distribution without the grindout. Stress intensity factors are calculated as a function of crack depth for both cases. Based on a comparison of the calculated stress intensity factor it is shown that the presence of the grindout has a negligible effect so that the original Appendix G analysis is still valid.

Results

Figure 1 shows the axisymmetric finite element model of the nozzle. The section of the element through the nozzle corner with the highest stress was considered in the evaluation. The axisymmetric model considered the commonly used assumption of a spherical shell with 1.5 times the radius of the vessel.

Figure 2 shows the variation of hoop stress across the nozzle thickness for a pressure of 3125 psi (corresponding to the design hydrotest) [Reference 2]. It is seen that with the grindout, the nozzle section stress is slightly higher but the percent change is very small. Stress intensity factors for the nozzle corner flaw were determined using the stress distribution for the two cases. The stresses were magnified such that the surface stress at the nozzle corresponds to the ASME Code stress index of 3.1 (NB-3338.2) on the inside surface. Figure 3 shows the calculated stress intensity factor as a function of crack depth for a nozzle corner flaw in the longitudinal plane. The differences in the K value are negligible for the 1 inch postulated flaw.

Conclusion

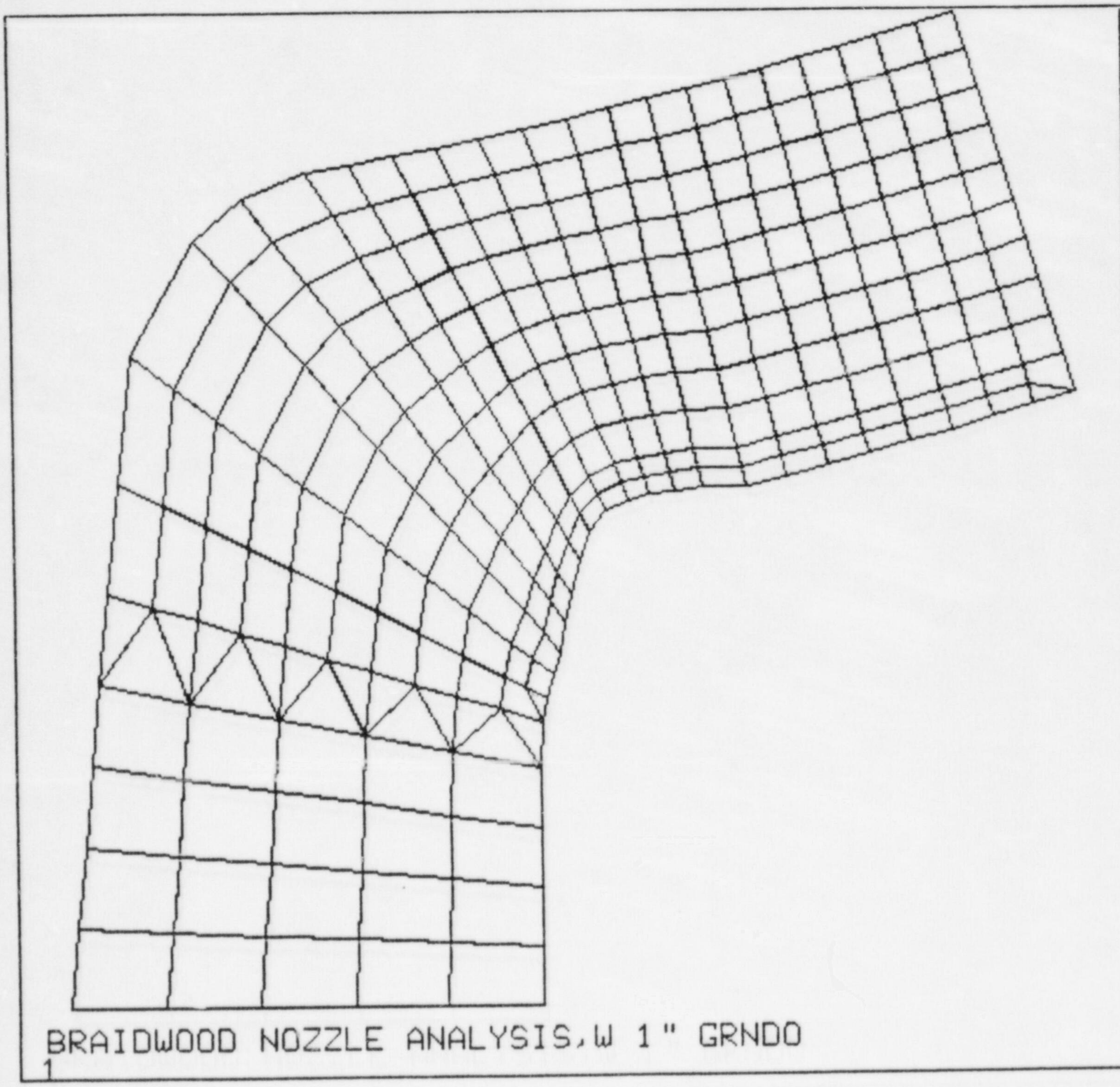
Comparison of the calculated stress intensity factor for the case with the 1 inch grindout and no grindout confirm that the change in the calculated K value is negligible. Thus the original Appendix G analysis in [1] remains valid.

References

1. "Appendix G Analysis Report #12" for Westinghouse Nuclear Energy Systems, Rev. 2. Performed by Babcock & Wilcox Company, February 1983.

2. "ASME Code Evaluation of the Braidwood Unit 2 Nozzle F to Include Effects of Proposed Grindouts". General Electric Report MDE#41-0386 Rev. 0, DRF A00-02669, February 1986.

FIGURE 1 - Braidwood Inlet Nozzle Finite Element Model



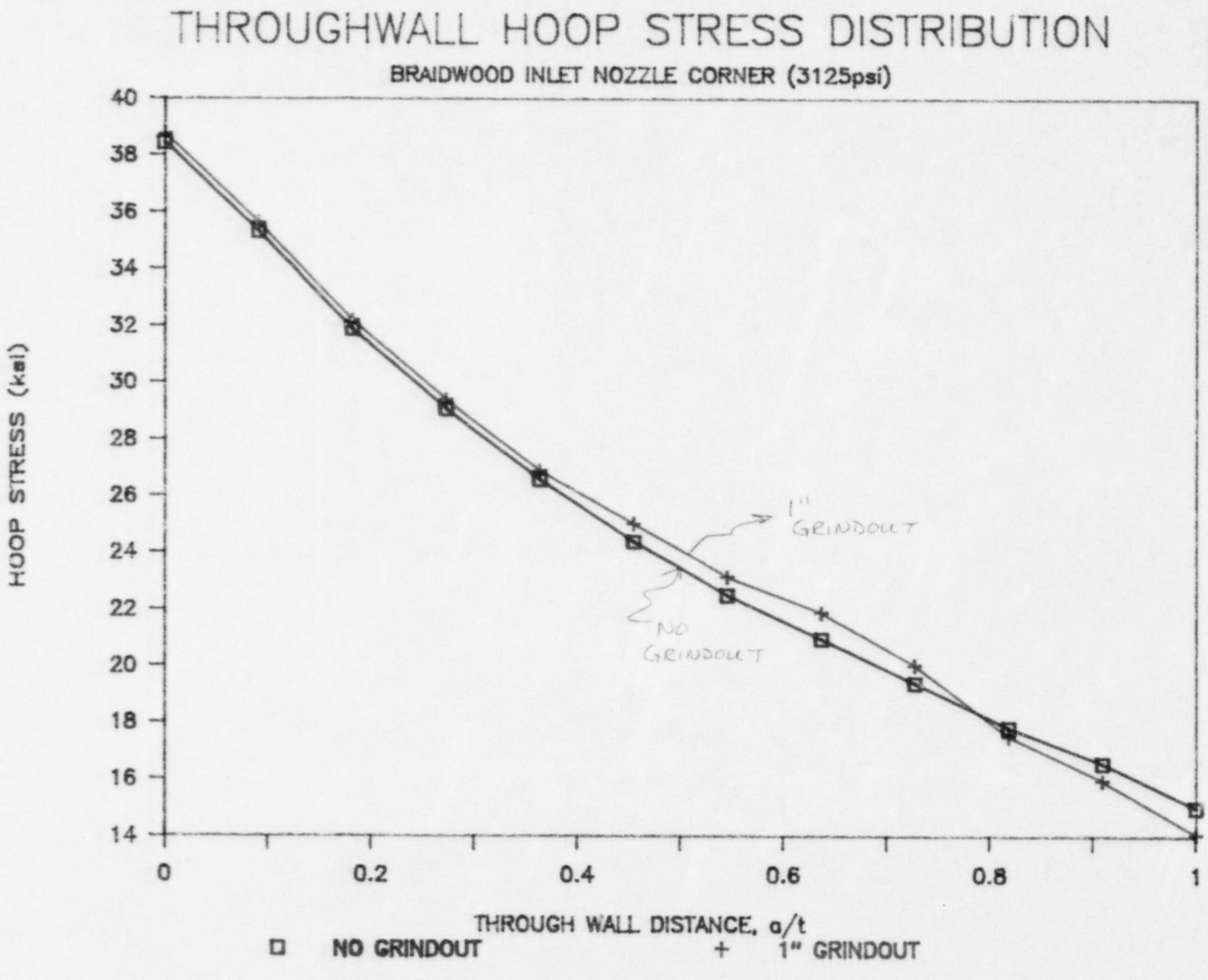


FIGURE 2 - Hoop Stress Comparison

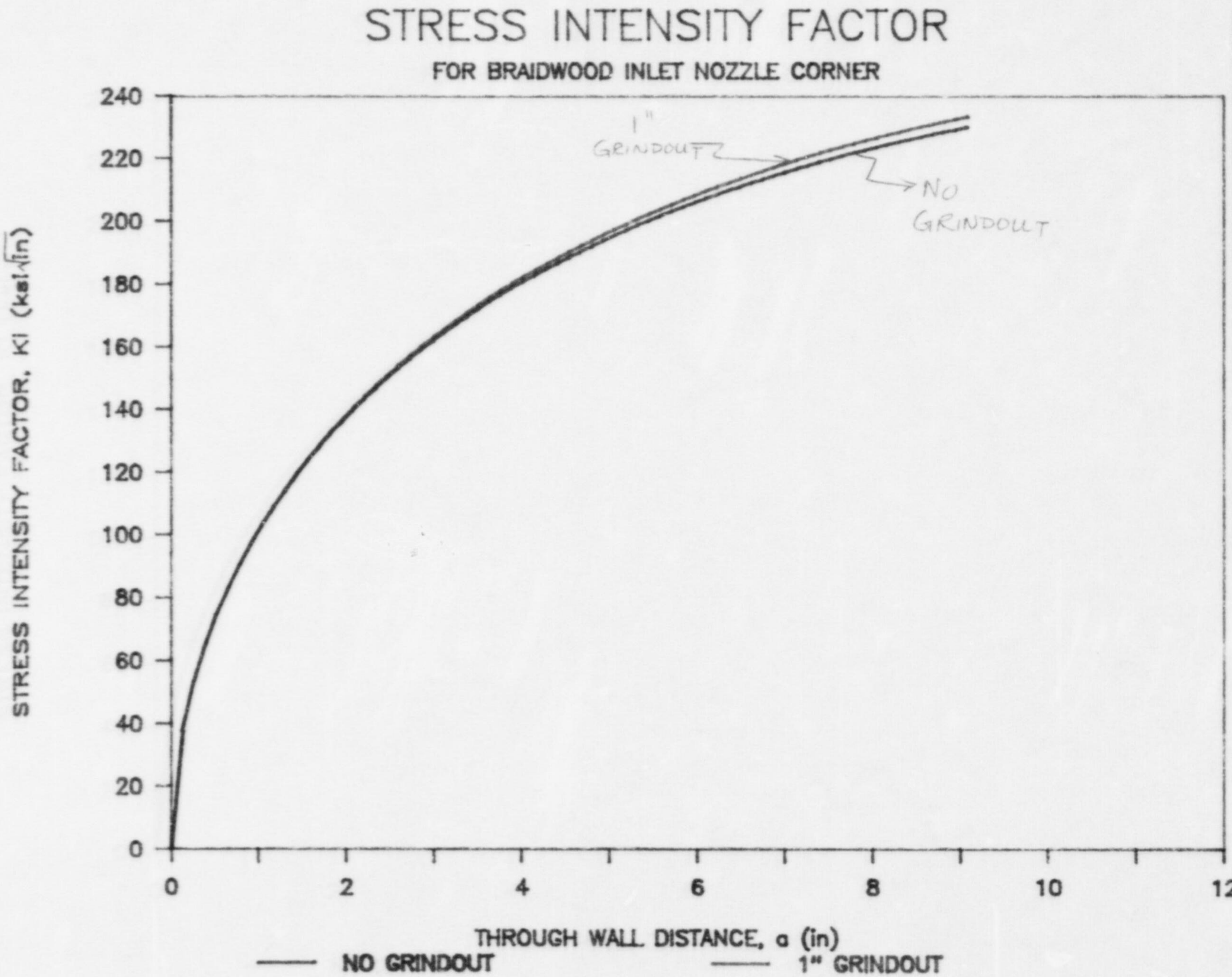
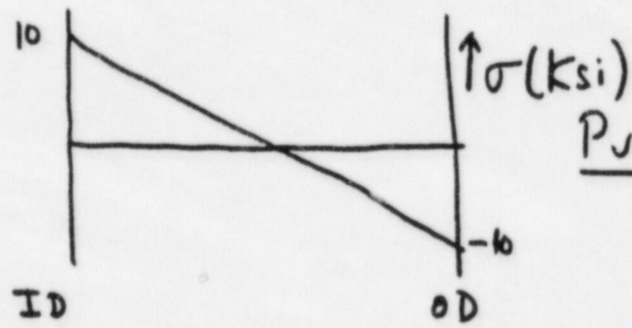


FIGURE 3 - Stress Intensity Factor Comparison

STRDIS - STRESS LINEARIZATION PROGRAM

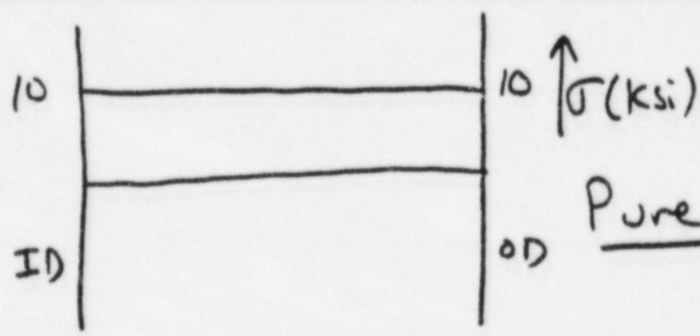
3 cases given.

1)



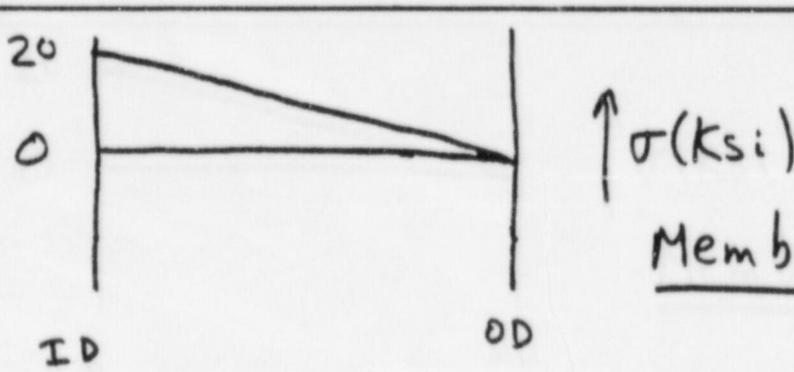
Pure bending

2)



Pure membrane

3)



Membrane + bending

CASE 1 - Pure bending

```
INPUT STRESSES ARE:
      10.0       7.5       2.5      -2.5      -7.5     -10.0
INPUT COORD ARE :
      0.       1.250     3.750     6.250     8.750    10.000

     MEMBRANE STRESS =        0.     BENDING STRESSES =(+ OR-)      10.00
     PEAKS1 =      -0.00    PEAKS2 =      0.00

     MEMBRANE PLUS BENDING STRESS =      10.00
```

CASE 2 - Pure membrane

```
INPUT STRESSES ARE:
      10.0      10.0      10.0      10.0      10.0      10.0
INPUT COORD ARE :
      0.       1.250     3.750     6.250     8.750    10.000

     MEMBRANE STRESS =      10.00    BENDING STRESSES =(+ OR-)      0.
     PEAKS1 =      0.       PEAKS2 =      0.

     MEMBRANE PLUS BENDING STRESS =      10.00
```

CASE 3 - Membrane + bending

```
INPUT STRESSES ARE:
      20.0      17.5      12.5       7.5       2.5       0.
INPUT COORD ARE :
      0.       1.250     3.750     6.250     8.750    10.000

     MEMBRANE STRESS =      10.00    BENDING STRESSES =(+ OR-)      10.00
     PEAKS1 =      -0.00    PEAKS2 =      0.00

     MEMBRANE PLUS BENDING STRESS =      20.00
```

Attachment 2

GENERAL ELECTRIC

ENGINEERING COMPUTER PROGRAM STATUS

(Ref. EOP 40-3.00)

ENGINEERING COMPUTER PROGRAM NAME: ANSYS04V

RESPONSIBLE ENGINEER

RECOVERY CLASSIFICATION

DESIGN RECORD FILE NUMBER: 313-01272

EWA NUMBER: EAT20-75

NAME: G. C. Mok | COMP: 523

LEVEL 1 PLANNED LEVEL 2 DESIGN REVIEW DATE 8439

APPROVING MANAGER

TITLE: J. E. [illegible] MANAGER, CORE & FUEL [illegible]

DATE: JUL 26 84

LEVEL 2R COMPUTATIONS SECTION LIBRARY IMPLEMENTATION

SELECT NAME: ANSYS04V

DATE: 11-07-84

EXPIRATION DATE: 8552

APPROVING MANAGER

TITLE: Mgr. CFT

DATE: 12-27-84

LEVEL 2 COMPUTATIONS SECTION LIBRARY IMPLEMENTATION

SELECT NAME

DATE

APPROVING MANAGER

TITLE

DATE

LEVEL 3 COMPUTATIONS SECTION LIBRARY IMPLEMENTATION

SELECT NAME

DATE

APPROVING MANAGER

TITLE

DATE

LEVEL 4

APPROVING MANAGER

TITLE

DATE

GENERAL ELECTRIC

ENGINEERING COMPUTER PROGRAM ABSTRACT

(Ref. EOP 40-3.00)

RESPONSIBLE ENGINEER	RECOVERY CLASSIFICATION	ENGINEERING COMPUTER PROGRAM NAME
		ANSYS04V
	DESIGN RECORD FILE NO.	ABSTRACT
G. C. Mok (NAME) — 523 (COMP)	B13-01272	12/19/84 (DATE) — 1 (REV. NO.)

APPLICATION STATEMENT

ANSYS04 is a large scale, general purpose finite element computer program with interactive capabilities. The program is an expanded version of the ANSYS03 computer program. The additional capabilities in ANSYS04 are the interactive capabilities, several new finite elements, and analysis options. ANSYS04 operates on the VAX computer while ANSYS03 runs on the Honeywell computer.

The ANSYS04 options shown in Table 1 are acceptable for design use and have been verified. Use of all other options for design is also acceptable provided the results are independently verified in accordance with EOP 42-6.00.

The analysis method is based on standard displacement formulation of the finite element method. Users of this program should have some educational background and work experience with the finite element method. Previous experience is recommended for correct use of the nonlinear analysis option. The user is responsible for determining whether the models are appropriate for the application and that correct results are produced.

PROGRAM DESCRIPTION

INPUTS

Finite element model geometry, material properties (mechanical or thermal), structural loadings or thermal conditions.

OUTPUTS

Finite element geometry plots. Analysis result plots, mechanical deformations, forces and stresses, temperature distribution (for heat transfer analysis only).

DOCUMENTATION

See attachment.

COMPUTER REQUIREMENTS

The ANSYS04 program is available on the VAX computer and has interactive capabilities.

G.C. Mok [signature] — 12/19/84

PREPARED BY: (PRINT NAME AND SIGN) — DATE

Table 1

Analysis	Geometry	Elements
Heat Transfer (Key = -1)	1-30 Frames	STIF 31,32,33,34,35,56,66
	2D or Axisym. Solids	STIF 31,32,34,55,67,71,75,77
	3D Solids	STIF 31,33,34,57,68,69,70,71
Static Analysis (Key = 0)		
Elastic	1-3D Frames	STIF 1,3,4,8,9,10,12,23,27,29,58
	Shells	STIF 11,41,43,63
	2D or Axisym. Solids	STIF 25,42,54,61,82
	3D Solids	STIF 44,45,52
Static Analysis (Key = 0)		
Elastic-Plastic*	1-3D Frames	STIF 1,20,23
	Shells	STIF 48
	2D or Axisym. Solids	STIF 42
	3D Solids	STIF 45
Mode-Frequency Response Spectrum (Key = 2)		
Elastic	1-3D Frame	STIF 1,3,4,9,10,14,21,40
	Shells	STIF 11,43,63
	2D or Axisym. Solids	STIF 25

*Iterations required with the same or new stiffness matrix.

Table 1 (Continued)

Analysis	Geometry	Elements
Non-Linear Transient-Dynamic (Key = 4)		
Elastic	1-3D Frame	STIF 8
	2-3D Frames or	STIF 21,40
	2D Axisymetric	STIF 40
Elastic-Plastic*	1-3D Frame	STIF 1,21
Linear Transient-Dynamic (Key =5)		
Elastic	1-3D Frame	STIF 1,3,14,21,40
Reduced Harmonic (Key = 6)		
Elastic	2-3D Frame	STIF 1,14,21,40

*Iterations required with the same or new stiffness matrix.

APPLICABLE DOCUMENTS

1. User's Manual: "ANSYS--Engineering Analysis System User's Manual for ANSYS Revision 4.0," G. J. DeSalvo and J. A. Swanson, Swanson Analysis Systems, Inc., 1982.

2. Example Manual: "ANSYS-Engineering Analysis System Example Manual," by G. J. DeSalvo and J. A. Swanson, Swanson Analysis Systems, Inc., April 1975.

3. Verification Manual: "ANSYS--Engineering Analysis System Verification Manual," by G. J. DeSalvo, Swanson Analysis System, Inc., June 1976.

4. B. J. Branlund, "ANSYS04 Software Management Plan, Revision 0," February 1984. (DRF #B13 01272)

5. B. J. Branlund, "ANSYS04 Hardware/Software System Specification," I.D. No. 5230022, Revision 0, February 1984. (DRF #B13 01272)

6. B. J. Branlund, "ANSYS04 Software Test Report," Revision 0, June 1984 (DRF #B13 01272)

7. G.C. Mok and B. J. Branlund, "Summary of Results of ANSYS04V Verification," Letter report, SASR, (Structural Analysis Service Report) 84-57, December 1984.

8. B.J. Branlund, "ANSYS04V Engineering Computer Program," Revision 0, December 1984. (DRF #B13 01272)

9. B.J. Branlund and G.C. Mok, "ANSYS04V User's Manual," Letter to ANSYS Users, December 20, 1984, (Letter Number BJB-84-05).

Attachment 2

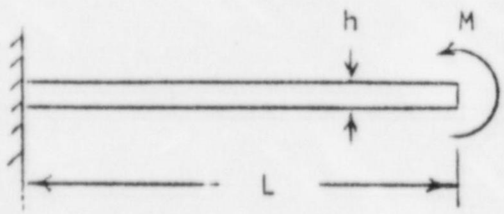
VERIFICATION PROBLEM NO. 16

TITLE: Bending of a Solid Beam.

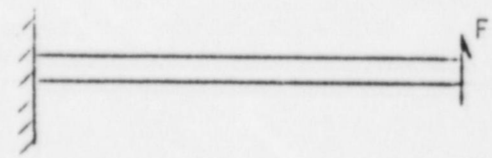
TYPE: Static analysis (K20=0), plane stress elements (STIF42).

REFERENCE: Roark (Ref. 6), Pages 104, 106.

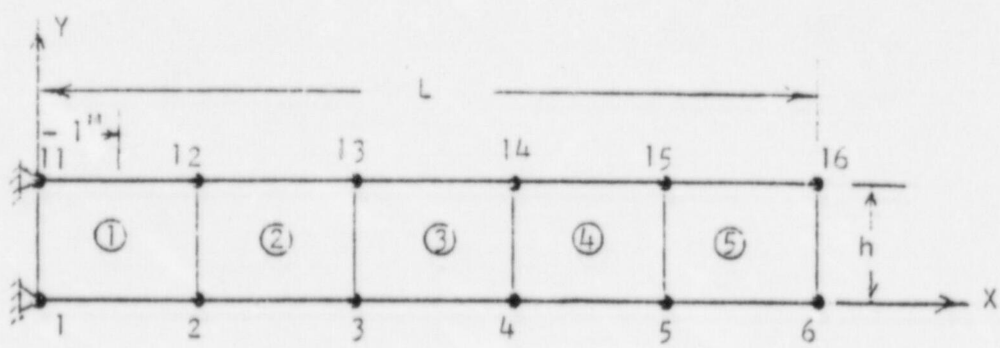
PROBLEM: A beam of length L and height h is built-in at one end and loaded at the free end with 1) a moment M, and 2) a shear force F. Determine the deflection δ at the free end and the bending stress σ_{Bend} 1 in. from the wall.



Case 1



Case 2



Finite Element Model

GIVEN: L = 10 in, h = 2 in, M = 2,000 in-lb, F = 300 lb, $E = 30 \times 10^6$ psi.

MODELING HINTS: The stiffness matrix formed in the first load step is also used in the second load step. The end moment is represented by equal and opposite forces separated by a distance h.

VERIFICATION PROBLEM NO. 16 (Continued)

SOLUTION COMPARISON:

	Case 1		Case 2	
	δ, in	σ_{Bend}, psi	δ, in	σ_{Bend}, psi
Theory	0.005	3000.	0.005	4050.
ANSYS	0.005	3000.	0.00505	4050.
Difference	None	None	1.0%	None

RUN TIME: 5 Central Processing Seconds

GE run duplicates ANSYS answers given here

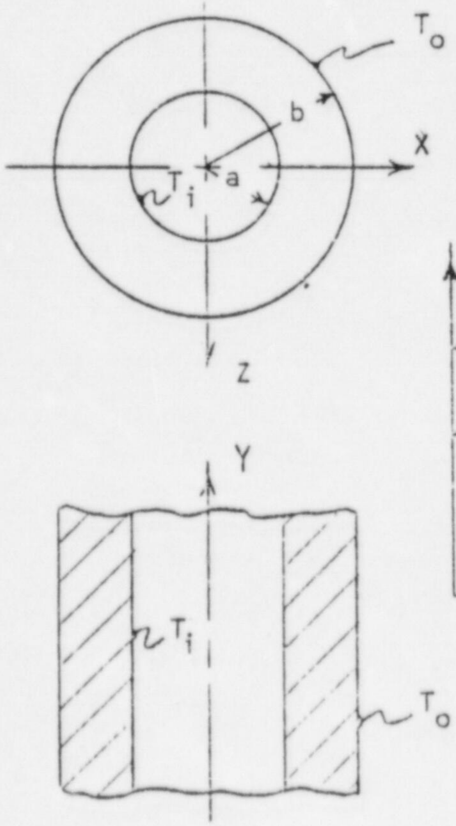
VERIFICATION PROBLEM NO. 33

TITLE: Thermal Stresses in a Long Cylinder.

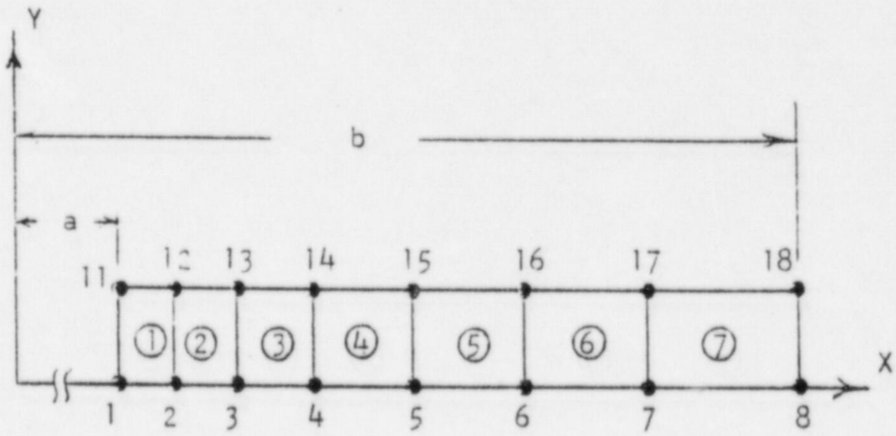
TYPE: Static, thermal stress analysis (K20=0), axisymmetric plane elements (STIF42).

REFERENCE: Timoshenko (Ref. 4), Page 234, Problem 1.

PROBLEM: Determine the axial stress σ_a and the tangential (hoop) stress σ_t at the inner and outer surfaces of the long thick-walled cylinder described in Verification Problem No. 32.



Problem Sketch



Finite Element Model

GIVEN: $a = 0.1875$ in, $b = 0.625$ in, $E = 30 \times 10^6$ psi, $\alpha = 1.435 \times 10^{-5}$ in/in-°F, $\nu = 0.3$.

MODELING HINTS: Use the same model as developed for the thermal analysis. Surface stresses are requested on elements 1 and 7. The extra displacement shapes are suppressed. Nodal coupling is used to insure symmetry.

VERIFICATION PROBLEM NO. 33 (Continued)

SOLUTION COMPARISON:

	X = 0.1875 in		X = 0.625 in	
	σ_a, psi	σ_t, psi	σ_a, psi	σ_t, psi
Theory	420.	420.	-194.	-194.
ANSYS	417.	410.	-196.	-197.
Difference	0.8%	2.5%	1.0%	1.5%

RUN TIME: 3 Central Processing Seconds

GE answers duplicate results given here.

Attachement 2

VERIFICATION PROBLEM NO. 38

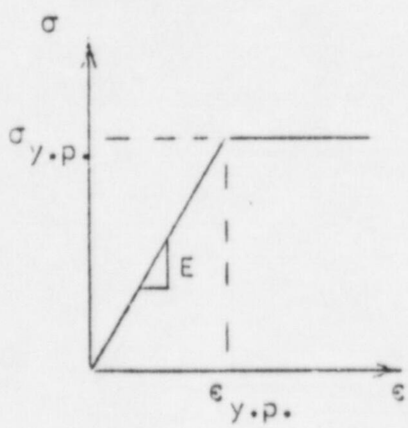
TITLE: Plastic Loading of a Thick-Walled Cylinder Under Pressure.

TYPE: Static, plastic analysis (K20=0), axisymmetric plane elements (STIF42).

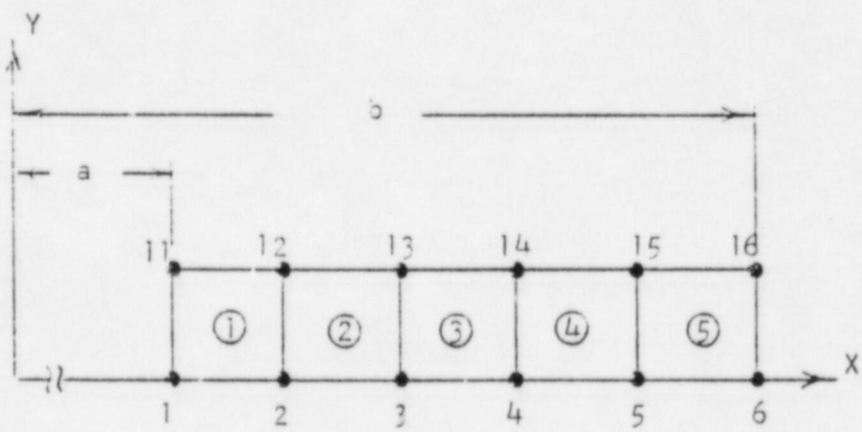
REFERENCE: Timoshenko (Ref. 4), Page 388, Article 70.

PROBLEM: A long thick-walled cylinder is subjected to an internal pressure p. For $p = p_{el}$, the maximum pressure at which the wall remains elastic, determine the radial stress σ_r and the tangential (hoop) stress σ_t at locations near the inner and outer surfaces. For $p = p_{ult}$, the pressure required to just bring the entire wall into a state of plastic flow, determine the effective stress σ_{eff} at the same locations.

PROBLEM SKETCH: See Verification Problem No. 25.



Stress-Strain Curve



Finite Element Model

GIVEN: $E = 30 \times 10^6$ psi, $\sigma_{yp} = 30{,}000$ psi, $\nu = 0.3$, a = 4 in, b = 8 in.

CALCULATED INPUT: $p_{el} = 12{,}990.381$ psi, $p_{ult} = 24{,}011.32$ psi. Note, the theory available for this problem is based on the Tresca (maximum shear) yield criterion. ANSYS uses the Von Mises yield criterion. The pressures are calculated from the theory by using $\tau_{y.p.} = \sigma_{y.p.}/\sqrt{3}$. This procedure is sufficient to calculate approximate loads but the resulting stress components should not be compared.

MODELING HINTS: Three intermediate loadings are input between p_{el} and p_{ult}. The extra displacement shapes are suppressed, although it is not necessary to do so. Nodal coupling is used to insure symmetry.

VERIFICATION PROBLEM NO. 38 (Continued)

DATA INPUT LISTING (Continued):

```
L   -2   0-10 10
M        0
N           END
O           END
P        1         1 16664.028
         0
L   -2   0-10 10
M        0
N           END
O           END
P        1         1 20337.675
         0
L   -2   0-20  1
M        0
N           END
O           END
P        1         1 24012.
         0
S  FINISH

   1         12         24         36         48         60         72      80
```

SOLUTION COMPARISON:

Fully elastic:

	X = 4.4 in.		X = 7.6 in.	
	σ_r, psi	σ_t, psi	σ_r, psi	σ_t, psi
Theory	-9,984.	18,644.	-467.	9,128.
ANSYS	-9,960.	18,682.	-464.	9,100.
Difference	0.24%	0.20%	0.827	0.31%

GF answers duplicate results in table

VERIFICATION PROBLEM NO. 38 (Continued)

SOLUTION COMPARISON (Continued)

Fully plastic:

	X = 4.4 in.		X = 7.6 in.	
	σ_{eff}, psi	Status	σ_{eff}, psi	Status
Theory	30,000	Plastic	30,000	Plastic
ANSYS	29,860.	Plastic	29,953	Plastic
Difference	0.47%	None	0.15%	None

RUN TIME: 60 Central Processing Seconds

BRAIDWOOD NOZZLE ANALYSIS

Figure 1 shows the location of the proposed grindout. The grindout is located at the nozzle to shell weld. Figure 1 illustrates the section through which the ASME Code Section III analysis was performed (Reference 1). This section considers a cut through the shell wall. This section was considered since it was the limiting section as shown in Reference 2. In Reference 2, the membrane stress for a section through the nozzle wall was 9.4 ksi.

Calculations in Reference 1 show that even with the 1" deep grindout, the area of reinforcement requirements are satisfied. Therefore, the grindout is outside the area of reinforcement. Furthermore, since the governing section with the highest primary stress is in the shell thickness, the appropriate classification for the stress through the section is P_L (Table NB-3217-1) under "Any Shell or Head".

It should also be noted that according to Section NB-3331-b of Section III, ASME Code (1973), satisfaction of the area of reinforcement requirement assures compliance with NB-3221.1 (P_m), NB-3221.2 (P_L) and NB-3221.3 (P_L+P_b) in the vicinity of the openings and no specific analysis showing satisfaction of these stress limits is required.

REFERENCES

1. "ASME Code Evaluation of the Braidwood Unit 2 Nozzle F to Include Effects of Proposed Grindouts". General Electric Report MDE#41-0386 Rev. 0, DRF A00-02669, February 1986.

2. "Thermal/Mechanical Analysis of the Inlet Nozzle Report #5" for Westinghouse Nuclear Energy Systems, Rev. 1. Performed by Babcock & Wilcox Company, March 1983.

BRAIDWOOD NOZZLE CONFIGURATION

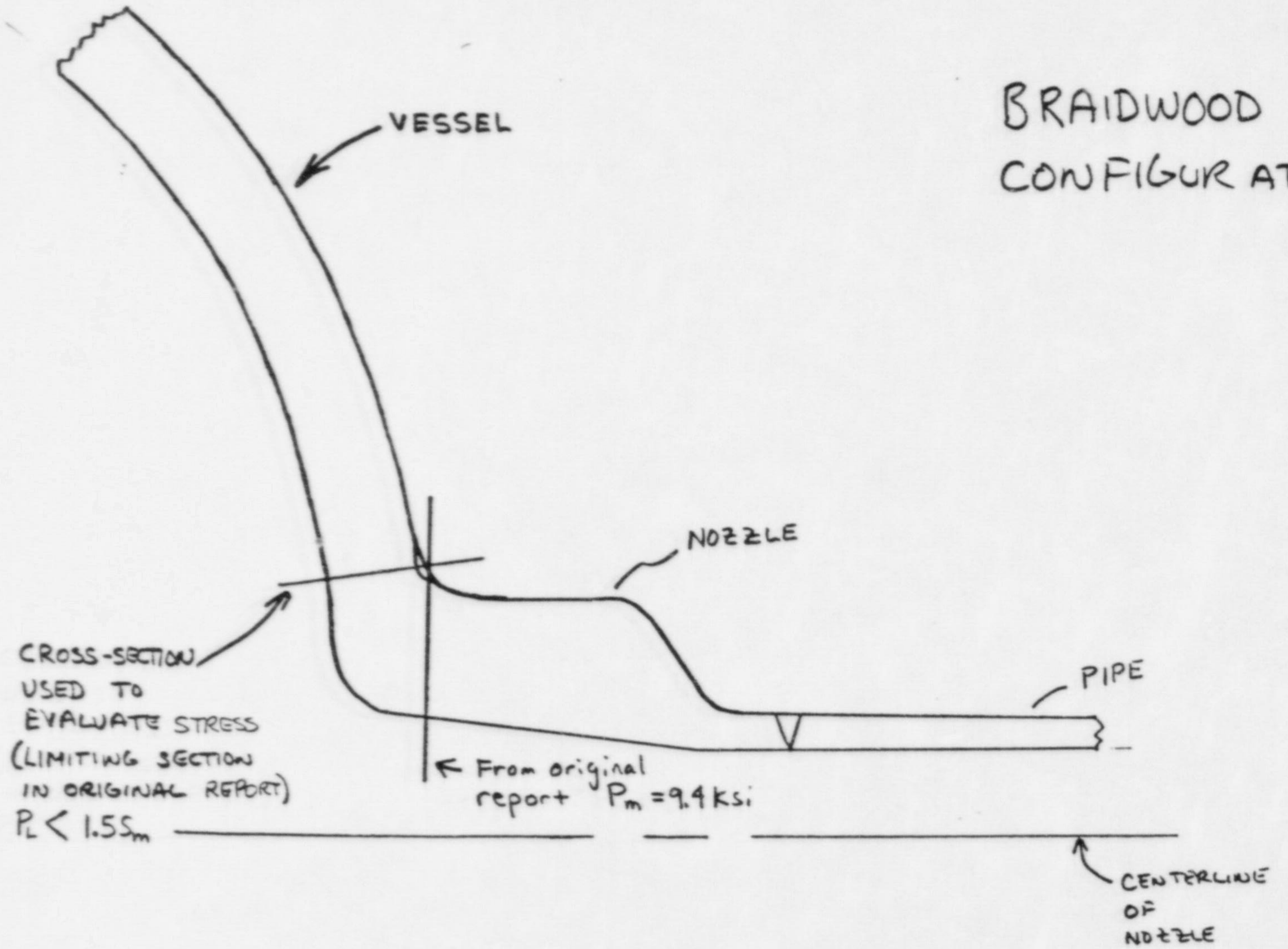
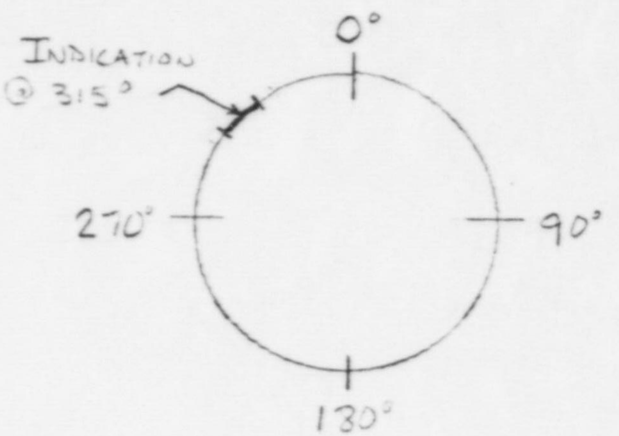


FIGURE 1

BRAIDWOOD UNIT 2 REACTOR VESSEL COLD LEG NOZZLE-TO-SHELL WELD INDICATION



INDICATION DIMENSIONS

LENGTH (ℓ) = 0.75 IN.
WIDTH (a) = 0.73 IN.
DEPTH = AT O.D. SURFACE

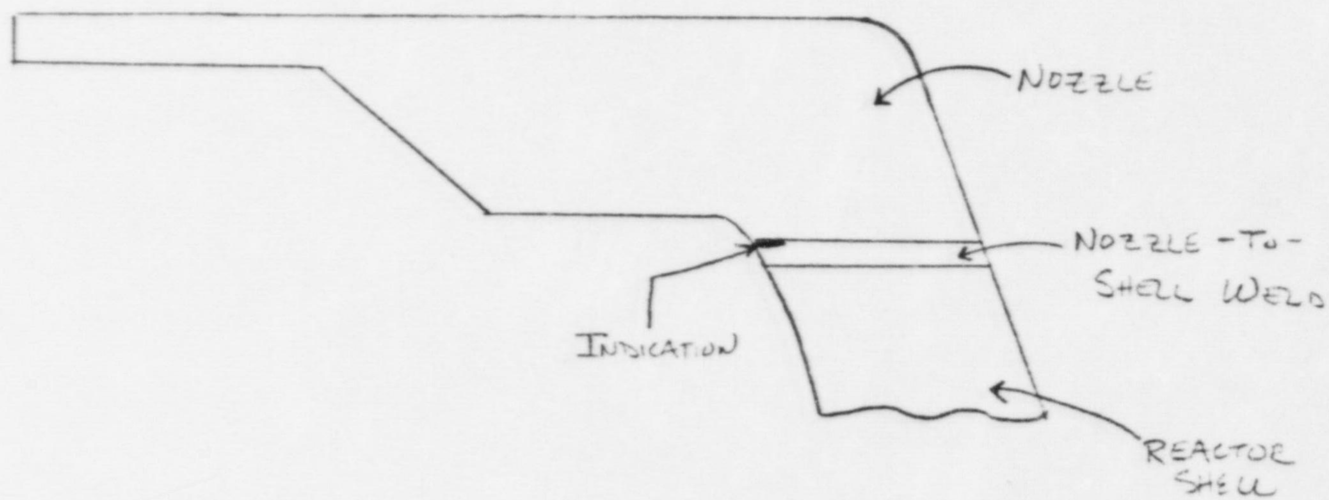
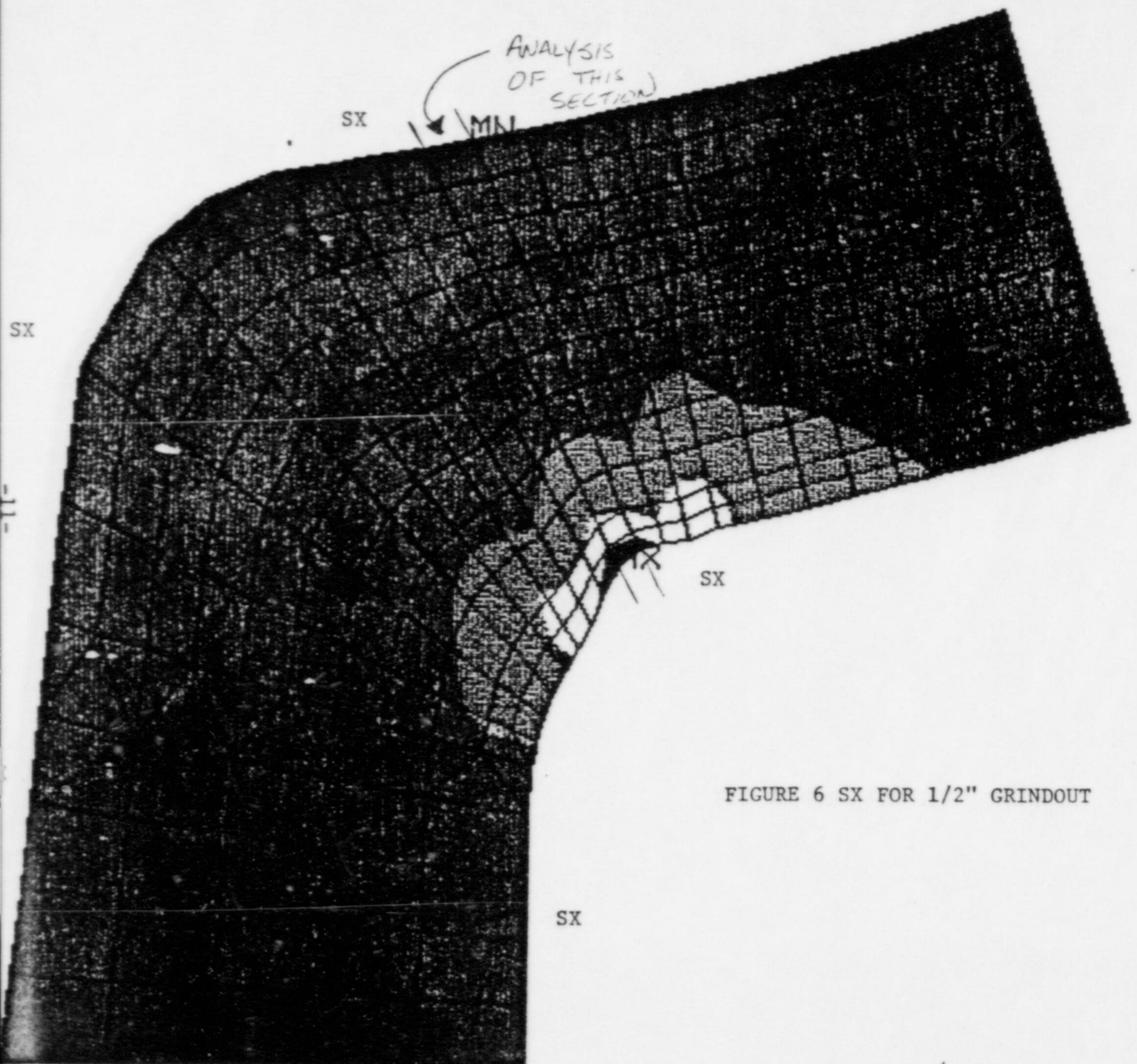


FIGURE 2



13.2246
POST1
STEP=1
ITER=1
STRESS PLOT
SX

AUTO SCALING
ZV=1
DIST=13.7
XF=28.4
YF=37.3
MX=17722
MN=-3009
INC=4000

FIGURE 6 SX FOR 1/2" GRINDOUT

RAIDWOOD - 1/2" GRINDOUT - 1000 PSI