Finite Element Analysis of EC Insert Plug

Robert Wands

Purpose

The proposed EC calorimeter insert plug was modeled with ANSYS to verify that the shell thickness calculated with beam formulas are adequate.

Finite Element Model

The finite element model and dimensions is shown in Fig. 1. The geometry and shell thicknesses used were the best numbers available as of 3/28/86. The model includes only the inner and outer shells and intermediate structural discs. The total weight of the plug is calculated to be 75000 lbs. The plug is supported against this weight at the four nodes indicated in Fig. 1. A vertical constraint was used. The calorimeter plates are not explicitly modeled. Their weight is placed on the inner shell by giving the shell material an appropriate density and applying a global acceleration.

In addition to the weight loading, there will also be a pressure loading applied to both end plates as a result of preloading the calorimeter plates compressively. This pressure is estimated to be 20 psi, and was represented in the model as a uniform pressure applied across each end plate. The large axial force produced by this pressure precludes the possibility of attaching the inner shell to both end plates. Such attachments would be under unreasonably high stress as the plates were preloaded, and the inner shell would be under a state of tension in trying to resist the axial force. In the real structure, the inner shell will be attached to at most one of the end plates. The axial force is then developed solely in the outer shell, which has a considerable area of attachment. To emulate this in the finite model, nodal coupling was used to couple the shell laterally to both end plates and all intermediate discs to ensure weight transfer, but axially the shell was only coupled to one of the end plates.

The materials used were assumed to be SS 304 with a Young's modulus of 28.3×10^6 psi. Stresses were evaluated according to the limits and classifications of the ASME Boiler and Pressure Vessel Code, Section VIII, Division 2, Appendix 4 assuming a maximum allowable stress intensity of 20000 psi for primary membrane stress.

Stress and Deflection Results

The deflected geometry is shown in Fig. 2. Comparison with Fig. 1. shows that the nodal coupling used to allow axial movement of the inner shell relative to all but one end plate has been effective. The total relative movement of the inner shell and free plate is 0.80 in. The inner shell behaves

very much as a beam of circular section under this loading. The highest primary membrane stress is 17000 psi. There is not a significant bending component through the thickness of the shell; the shell resists bending by the development of membrane stresses. The maximum vertical deflection of the inner shell is -0.041. This deflection includes all structural deflection relative to the support nodes.

Figure 3 is a plot of all portions of the structure which exceed 20000 psi maximum stress intensity. The eight large rectangular elements are connected to the two support nodes. The high stress intensities in this region are developed because this support bears only on the outer shell, and not directly against an end plate, as the support at the other end does. In practice, a heavy reinforcement will be used in the vicinity of this support to distribute the reaction more favorably.

The inner radii of the end plates represent the other highly stressed region. This is a more fundamental effect and can be compared to a simple hand calculation for verification.

A look at the component stresses in these elements shows a maximum bending stress of ~40000 psi. This is representative of bending about a radial line. From Formulas for Stress and Strain, by Roark and Young, Table 24, case 2a, the maximum moment for an annular plate with a uniform pressure P and simply supported at the outer edge is

$$M = K_m P a^2$$

where

M = moment about a radial line at the inner radius

P = pressure = 20 psi

a = outer radius = 34

K_m = factor related to b/a, where b = inner radius

K_m can be found for a b/a ratio of $2.5/34 = 0.07$ by extrapolation of the tabulated values for this case. The result is $K_m = 0.407$. Then

$$M = 0.407 (20)(34)^2$$

$$M = 9410 \text{ in} \cdot \text{lbs/in}$$

Then the maximum bending stress is

$$\sigma = \frac{6M}{t^2} = \frac{6(9410)}{1.25^2}$$

$$\sigma = 36100 \text{ psi}$$

This compares favorably with the ANSYS value.

This stress is a primary bending stress and by Division 2 rules cannot exceed $1.5 S_m$, or 30000 psi. It is therefore necessary to reinforce the 2.5" radius opening.

The required reinforcement is most easily calculated by the rules of AD-530 of Division 2.

The total cross sectional area of reinforcement required is

$$A = 0.5 dt_r$$

d = diameter of finished opening

t_r = minimum thickness which meets requirements of AD-702 in the absence of the opening

The thickness t_r is found from

$$t_r = d \left(\frac{CP}{S} \right)^{1/2}$$

where d = diameter of head = 68 in

P = pressure = 20 psi

S = membrane stress intensity limit = 20000 psi

C = factor dependent on method of attachment to shell; use 0.3

Then

$$t_r = 68 \left(\frac{0.3(20)}{20000} \right)^{1/2}$$

$$t_r = 1.18 \text{ in}$$

Then, the required area of reinforcement is

$$A = 0.5 (5.0)(1.18)$$

$$A = 2.95 \text{ in}^2$$

This reinforcement can be distributed radially to a distance of 5 inches from the centerline of the hole as shown in the figure below. If the distribution is uniform, the additional head thickness t_a necessary to achieve the reinforcement is

$$A = 2.95 \text{ in}^2 = 5 t_a$$

$$t_a = 0.59 \text{ in}$$

The total head thickness should be $1.18 + 0.59 = 1.77$ inches around the hole.

Conclusion

The proposed insert plug inner shell is adequately sized for the required loading. The end plates meet Division 2 requirements for 20 psi flat heads, but must be reinforced to a total thickness of at least 1.77 inches in the vicinity of the 5 inch diameter holes. The plug support which does not fall at the axial location of a disc must be reinforced to reduce the membrane and bending stress.

ANSYS
86/ 3/28
9.7875
PLOT NO. 1
POST1
ELEMENTS

ORIG SCALING
XV=1
YV=1
ZV=1
DIST=46.9
XF=-11.5
YF=-1.04
ZF=50.1
HIDDEN

Fig 1 Finite Element Model
Geometry and Dimensions

ANSYS
 86/ 3/28
 9.7881
 PLOT NO. 2
 POST1
 STEP=1
 ITER=1
 DISPLACEMENT

ORIG SCALING
 XV=1
 YV=1
 ZV=1
 DIST=46.9
 XF=-11.5
 YF=-1.04
 ZF=50.1
 HIDDEN
 DMAX=.418
 DSCA=11.2

Fig2. Deflected Shape

INSERT PLUG SUPER ELEMENT

ANSYS
86/ 3/28
9.7944
PLOT NO. 6
POST1
ELEMENTS

ORIG SCALING
XV=1
YV=1
ZV=1
DIST=38.5
XF=-6.11
YF=-6.21
ZF=55.5
HIDDEN

Fig3. Elements with a maximum stress intensity greater than 20000 psi

Appendix A Finite Element Analysis of EC Insert Plug

Robert Wands

Purpose

The previous EC insert plug analysis assumed that the inner shell was connected to one of the end plates only. The purpose of this analysis was to connect the inner shell to both end plates and evaluate the resulting shell stresses and connection forces.

Finite Element Model

The model used is identical to that used in the previous analysis with the exception that the nodes at the previously free end of the inner shell were coupled axially with the adjacent end plate. The coupling was in translation only. The inner shell could exert no edge moment on the end plates at the connection.

Stress and Deflection Results

Figure A1 shows the deflected shape. The attachment at the end plates can be clearly seen. Post-processing indicates that an axial tensile force of 26500 lbs is imposed on the tube by the end plates. This force results in a general membrane stress of

$$\sigma = \frac{P}{A} = \frac{26500}{2\pi (2.5)(0.125)} = 13500 \text{ psi}$$

This membrane stress is apparently sufficient to increase the stress intensity to greater than 20000 psi in several elements of the inner shell. This can be seen in Fig. A2. Since this is a general primary membrane stress; and cannot exceed the Code-allowable of 20000 psi, a thicker inner shell wall should be used. An increase in thickness to 3/16" would reduce the stress due to axial load to 9000 psi, and decrease the bending stresses in the tube due to calorimeter plate loading.

The weld used to attach the inner shell to the end plates can be sized using the criteria that it must resist a total axial force of 26500 lbs. In the real structure, some edge bending moment and hence secondary bending stress will be developed in the weld and inner shell. This moment can be estimated by assuming that the 3/16" shell will simply follow the rotation of the inner radius of the end plate. This rotation can be calculated by using formulas from Formulas for Stress and Strain by Roark and Young, Table 24, case 2c.

$$a = 34$$

$$b = 2.5$$

$$t = 1.25$$

$$D = \frac{Et^3}{12(1-\nu^2)} = 5.06 (10^6)$$

$$b/a = 0.07$$

$$K_{\theta b} = -0.028$$

$$q = 20$$

Then

$$\alpha_b = K_{\theta b} \frac{qa^3}{D} = 4.35 (10^{-3}) \text{ radians}$$

This rotation can now be imposed on the end of the inner shell and a resulting edge moment calculated using Table 30, case 10.

$$R = 2.5 \text{ in}$$

$$t = 3/16 = 0.1875 \text{ in}$$

$$\lambda = \frac{3(1-\nu^2)}{2.5^2(0.1875)^2}^{1/4} = 1.88 \text{ in}$$

$$D = \frac{Et^3}{12(1-\nu^2)} = 1.7 (10^4) \text{ in}^4$$

$$\text{Then } \psi_A = 4.35 (10^{-3}) = \frac{M_O}{1.7(10^4)(1.88)}$$

$$M_O = 139.0 \text{ in} \cdot \text{lbs/in}$$

$$\text{Then } \sigma = \frac{6 M_O}{t^2} = \frac{6(139.0)}{(0.1875)^2}$$

$$\sigma = 23700 \text{ psi}$$

This is well below the Code allowable of $3 S_m = 60000 \text{ psi}$ for secondary stresses.

ANSYS
86/ 4/ 9
12.1769
PLOT NO. 3
POST1
STEP=1
ITER=1
DISPLACEMENT

ORIG SCALING
XV=1
YV=1
ZV=1
DIST=46.9
XF=-11.5
YF=-1.04
ZF=50.1
HIDDEN
DMAX=.0713
DSCA=65.7

Fig A1 Deflected
Shape

INSERT PLUG SUPER ELEMENT - INNER SHELL TIED AT BOTH ENOPLATES

ANSYS

86/ 4/ 9

12.1872

PLOT NO. 10

POST1

ELEMENTS

ORIG SCALING

XV=1

YV=1

ZV=1

DIST=38.5

XF=-6.11

YF=-6.21

ZF=55.5

HIDDEN

Fig A2 Element with
Stress intensity greater
than Code allowable
of 20000 psi.

ELEMENTS WITH SI GREATER THAN 20000 ~~XXXXXX~~