Burst Pressure Prediction of Cylindrical Shell Intersection

Liping Xue, G. E. O. Widera
Marquette University
Center for Joining and Manufacturing Assembly
Milwaukee, Wisconsin 53201

Zhifu Sang
Nanjing University of Technology
College of Mechanical Engineering
Nanjing, Jiangsu 210009
P. R. China

ABSTRACT

The objective of this paper is to estimate the burst pressure and failure location of a cylindrical shell intersection by use of finite element analysis (FEA). Three dimensional 20 node structural solid elements are employed to perform a static, nonlinear (both geometry of deformation and material behavior) FEA via ANSYS. To verify the accuracy of the numerical prediction, an on-site burst test was carried out by hydraulically pressurizing the test vessel to burst. Based on the reasonable agreement between the numerical simulation and the experimental results, it is concluded that the finite element method can be employed with sufficient accuracy to predict the burst pressure and failure location of cylindrical shell intersections.

INTRODUCTION

Cylindrical shell intersections are structural elements which occur often in many industries. The action of various loads leads to high local stresses in the intersection region resulting in stress concentrations there. Due to the difficulties arising during and defects created by welding, this region will become the weakest point and the failure source of the whole structure. The failure of a cylindrical shell intersection can cause extensive property damage, personal injury, environmental pollution, and even loss of life. It is, therefore, very important to be able to predict the magnitude of the burst pressure and the location of the failure of this kind of structure.

Cottam and Gill[1] carried out eleven tests, to rupture, on mild steel cylindrical pressure vessels with flush nozzles. Two cylindrical vessels without nozzles were also tested to establish datum curves for vessels with nozzles. E. C. Rodabaugh[2] summarized 31 available burst test data and failure locations on basic configuration pipe connections. The basic configuration was a pipe connection consisting of the vessel with a uniform-wall branch pipe; there was no pad or any other type of reinforcement other than that provided by a fillet weld on the outside surface of the intersection. The burst tests were conducted on two cylindrical shell intersections (90 degree intersection and 30 degree lateral) by Sang, Xue, Lin and Widera[3]. Inspection of the metallographic structure and microstructure of the fracture in the initiation area showed that the cylinder wall produced obvious plastic deformation in the area of the fracture and was representative of a typical ductile fracture. Although there exists data on burst pressures of cylindrical shell intersections determined from experiments, data from this approach is limited due to the difficulties and cost associated with fabricating a precise shell intersection, when compared with the large number of possible configurations in real world applications. Therefore, the finite element analysis is used to estimate the burst pressure. An elastic-plastic failure analysis of thin toroidal shells using large strain-large displacement elastic plastic finite element analysis (LS-LD-EP-FEA) was performed by Jones et al.[4]. A comparison between finite element analysis results and test date from the PVRC burst disk program was provided by Jones and Holliday[5]. The results showed that LS-LD-EP-FEA can provide a best-estimate analysis of the burst of a disk, but that the accuracy depended on the material stress-strain curve. The burst pressure and its location in DOT-39 refrigerant cylinders were determined using both experimental burst tests and finite element analysis modeling[6]. Having studied the effect of external corrosion defects through a nonlinear numerical model based on the finite element method, Loureiro et al.[7] developed a simple procedure for estimating the burst pressure of corroded pipes. The burst pressures of three pipes, each containing a single internal corrosion pit, were predicted via finite element analyses, using a very fine finite element mesh, by Chouchaoui et al.[8]. The use of non-linear FEA to predict the failure pressure of real corrosion defects was investigated by Cronin[9] using the results from 25 burst tests on pipe sections removed from service due to the presence of corrosion defects. The author concluded that elastic-plastic FEA provided an accurate prediction of the burst pressure and the failure location of complex-shaped corrosion defects. Nevertheless, no similar body of knowledge appears to be available for predicting the burst pressure and the failure location of cylindrical shell intersection by use of a static, nonlinear finite element analysis.
EXPERIMENTAL STUDY

Description of Test Vessel

The test vessel consisted of a cylinder, nozzle, standard elliptical head, filling and venting connections and supports. Details of the structure and dimensions for the test vessel are given in Table 1.

Table 1 Dimensions of Test Vessel and Material Properties

<table>
<thead>
<tr>
<th>Scheme of Test Vessel</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>Dimension</th>
<th>$D_o$ mm</th>
<th>$T$ mm</th>
<th>$L$ mm</th>
<th>$L_v$ mm</th>
<th>$d_o$ mm</th>
<th>$t$ mm</th>
<th>$L_n$ mm</th>
<th>$d/D$</th>
<th>$t/T$</th>
<th>$D/T$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>408</td>
<td>9</td>
<td>1200</td>
<td>600</td>
<td>133</td>
<td>6</td>
<td>300</td>
<td>0.318</td>
<td>0.667</td>
<td>44.3</td>
</tr>
<tr>
<td>Material Properties</td>
<td>Yield strength (MPa)</td>
<td>Ultimate tensile strength (MPa)</td>
<td>Elongation coefficient (%)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Vessel (A672)</td>
<td>295</td>
<td>425</td>
<td>33.9</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Nozzle (A106-80 Gr.A)</td>
<td>316</td>
<td>472</td>
<td>30.48</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Material Properties

The material of the cylinder and nozzle were A672 and A106-80 Gr.A, respectively. The main purpose of this study was to determine the burst pressure of the test vessel and therefore, the yield and ultimate stresses of the material become important parameters.

Typical stress-strain curves of A672 and A106-80 Gr.A up to fracture were shown in Fig. 1. Average values of the yield strength $\sigma_y$ and ultimate strength $\sigma_u$ for material A672 and A106-80 Gr.A are 295 and 425 MPa, 316 and 472 MPa, respectively. These two plots indicated the elastic tensile modulus of $1.91 \times 10^5$ MPa for material A672 and $2.12 \times 10^5$ MPa for material A672-80 Gr.A, respectively, which were used throughout the analysis.
Fabrication of Test Vessel

Single V-groove butt joints, Fig. 2 (a), were used for the longitudinal and circumferential welds. A single bevel groove fillet weld, Fig. 2 (b), was employed for the vessel-nozzle corner joint. Tungsten-inert gas arc welding (root of welds) and manual electric-arc welding (deposit of weld rod material) were used for all welds.

Experimental Procedure and Results

Burst test of the cylindrical shell intersection was performed by hydraulically pressurizing the vessel from 0 MPa to burst, at room temperature. Unfortunately, the test vessel leaked at the weld before rupture because of the existence of a porosity in the weld. The internal pressure was 16 MPa at leakage. The old weld was removed and the test vessel was re-welded. The ratios of fillet legs to nozzle thickness and vessel thickness were 8/6 and 8/9 (see Fig. 2), respectively. After re-welding, the vessel was pressurized to burst. The burst pressure was 21.43 MPa. The failure occurred off-axis. The fracture surface, shown in Fig. 3, indicated that failure was by ductile rupture.
FINTIE ELEMENT ANALYSIS

Finite Element Model
A finite element model is developed, in parallel with the experimental program, to simulate the failure of the test cylindrical shell intersection. A static, nonlinear (both geometry of deformation and material behavior) finite element analysis of the cylindrical shell is performed by use of ANSYS. Three-dimensional 20 node structural solid elements are used to generate the finite element model. Due to the symmetry of the geometry and loading, only a half of the structure is analyzed. The guidelines for FEA modeling of cylinder-to-cylinder intersections developed by Widera and Xue\cite{10} are employed to mesh the structure. This leads to a finite element model having twenty-four elements in the circumferential direction of the nozzle, two elements through the thickness, twenty elements in the axial direction of the vessel and nozzle prior to reaching the decay distances of vessel and nozzle (3.0\sqrt{RT} and 3.0\sqrt{rt}, respectively). The maximum aspect ratio at the intersection area is less than 5. Fig. 4 illustrates the finite element mesh of the analysis model including the vessel, flush nozzle, and fillet weld at the juncture.
Material Properties

Analogous material properties are employed to correlate the results of the finite element analysis with those of the experimental test results. The true stress-true strain data of material A672 and A106-GrA shown in Table 2 and Table 3, respectively, were employed for the FEA material properties input.

The computer program utilizes the multi-linear kinematic hardening behavior. The material is assumed to be isotropic. Poisson’s ratio is taken as 0.3 for the elastic strains. The plasticity ratio governing convergence of this analysis is set to 15% rather than the default value of 30%.

Table 2 Multi-linear Material Models for Vessel Material A672

<table>
<thead>
<tr>
<th>Points</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
<th>10</th>
<th>11</th>
</tr>
</thead>
<tbody>
<tr>
<td>Strain (µε)</td>
<td>1544</td>
<td>5983</td>
<td>13700</td>
<td>21400</td>
<td>49000</td>
<td>81800</td>
<td>113900</td>
<td>167600</td>
<td>196800</td>
<td>220200</td>
<td>245000</td>
</tr>
<tr>
<td>Stress (MPa)</td>
<td>295</td>
<td>296.83</td>
<td>317.18</td>
<td>333.05</td>
<td>389.37</td>
<td>434.33</td>
<td>465.33</td>
<td>503</td>
<td>519.11</td>
<td>529.75</td>
<td>537.88</td>
</tr>
</tbody>
</table>

Table 3 Multi-linear Material Models for Vessel Material A106 Gr.A#

<table>
<thead>
<tr>
<th>Points</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
<th>10</th>
<th>11</th>
<th>12</th>
<th>13</th>
</tr>
</thead>
<tbody>
<tr>
<td>Strain (µε)</td>
<td>1491</td>
<td>1828</td>
<td>1854</td>
<td>5636</td>
<td>7817</td>
<td>16731</td>
<td>20513</td>
<td>53275</td>
<td>86941</td>
<td>145062</td>
<td>174655</td>
<td>195282</td>
<td>220251</td>
</tr>
<tr>
<td>Stress (MPa)</td>
<td>316</td>
<td>325.98</td>
<td>329.6</td>
<td>338.67</td>
<td>340.73</td>
<td>355.93</td>
<td>368.13</td>
<td>443.15</td>
<td>490.98</td>
<td>543.41</td>
<td>562.07</td>
<td>573.16</td>
<td>583</td>
</tr>
</tbody>
</table>

Boundary Conditions and Loading

Symmetry boundary conditions are employed on the two symmetry planes, the longitudinal plane (y = 0) and transverse plane (z = 0). The right end of the vessel is fixed for all directions except the axial (z) direction of the vessel. The finite element model is subjected to an internal pressure. This pressure is gradually increased for each step, with convergence achieved for each pressure increment. Equivalent axial stresses (see Eqs. (1) and (2)) are imposed as boundary conditions at the ends of the nozzle and vessel to simulate the contained pressure. Stresses are also gradually increased each load step in proportion to the internal pressure.
Finite Element Results

For a finite element analysis, a failure definition is required to estimate the burst pressure. As in Ref.[11] the burst pressure is taken as the one for which the structure approaches dimensional instability, i.e., the unbounded deflection for a small increment in pressure. The arc-length method is employed to solve this nonlinear problem. The maximum equivalent plastic strain occurs at the node 12229, which is 15 degree “off-axis” on the outside surface of the vessel. Figure 5 shows the location of Node 12229. Figure 6 depicts the contour plot of nodal equivalent plastic strain when internal pressure reaches 10.0 MPa. The failure location can be seen very clearly in this figure. Figure 6 shows the pressure versus equivalent plastic strain curves of node 12229.

\[
S_{st} = \frac{PD}{4t} \left( \frac{D}{D} \right)^2
\]

(1)

\[
S_{st} = \frac{Pd}{4t} \left( \frac{d}{d} \right)^2
\]

(2)

Fig. 5 Location of Node 12229

Fig. 6 Contour Plot of Nodal Equivalent Plastic Strain
Figure 7 demonstrates that the slope of the pressure-equivalent plastic strain curve becomes vanishingly small and the maximum nodal von Mises stress reaches the true ultimate tensile strength of material A106 Gr.A when the pressure reaches 18.3 MPa. Based on the failure criterion, this is the estimated burst pressure. The predicted burst pressure is 18.3 MPa, which is 14.5% lower than the experimental burst pressure of 21.43 MPa. The agreement is reasonable, considering the leakage and re-welding of the test vessel. For this model, the average maximum equivalent plastic strains through wall thickness occur off the longitudinal plane at node 12229. This coincides with the experimental result (see Fig. 3). Therefore, this point can be regarded as the failure location.

CONCLUSION

The burst pressures of a cylindrical shell intersection have been determined by use of static, nonlinear finite element analyses employing the arc-length method. The FEA predicted burst pressure of the intersection is checked against the experimental data. Based on the agreement, the following conclusions can be reached:

1. The finite element simulation can be employed to accurately estimate the burst pressure of a cylindrical shell intersection.
2. The failure location can be estimated based on the maximum of the average equivalent plastic strain across the thickness at various sections.
3. Both experimental and FEA results show that the failure of this cylindrical shell intersection occurs off longitudinal axis of the vessel.

NOMENCLATURE

\[ D \] \hspace{1cm} \text{Mean diameter of vessel} \\
\[ D_i \] \hspace{1cm} \text{Inside diameter of vessel} \\
\[ D_o \] \hspace{1cm} \text{Outside diameter of vessel} \\
\[ d \] \hspace{1cm} \text{Mean diameter of nozzle} \\
\[ d_i \] \hspace{1cm} \text{Inside diameter of nozzle} \\
\[ d_o \] \hspace{1cm} \text{Outside diameter of nozzle} \\
\[ L \] \hspace{1cm} \text{Length of cylinder}
\( L_v \)  Half length of cylinder \\
\( l_n \)  Length of nozzle \\
\( S_{\text{nn}} \)  Equivalent axial stress of nozzle \\
\( S_{\text{av}} \)  Equivalent axial stress of vessel \\
\( T \)  Wall thickness of cylinder \\
\( t \)  Wall thickness of nozzle \\
\( \sigma_y \)  Engineering yield strength \\
\( \sigma_u \)  Engineering ultimate strength \\

**REFERENCE**


