INVESTIGATION AND MODELING ON DYNAMIC FAILURE MODE-I TYPE CRACK TRANSITIONS IN PRESSURE VESSEL GEOMETRY

A.M. Senthil Anbazhagan
Lead Mechanical Engineer, Bilfinger Tebodin & Partner LLC, Muscat, Sultanate of Oman

M. Dev Anand
Professor and Director Research, Faculty of Mechanical Engineering, Noorul Islam University, Kumaracoil, Kanyakumari District, Tamilnadu, India

ABSTRACT

The objective of this work is to find out the deflection and deformation of axially cracked ASME pressure vessel geometry during internal pressure loading condition so that the location of deformation and its shapes could be identified for knowing the exact criticality of the deflection around the cracked surfaces. The background of this study is based on the failures often occurring in the oil and gas development offshore and onshore industries. Normally process equipments are having its own design life according to its material properties. Some of the equipments need frequent monitoring as well as proper maintenance during operations. As a code practices that normally process equipments need to be designed for maximum life span of 20 to 30 years of operation. However failures during operation are unpredictable as this purely depends on the operating as well as environmental conditions. If the operation went wrong or the environment load exceed than design loads there is a chance of failure. There kind of failures are not avertable all the time but it is diagnosable. With respect to this, if any cracks generate or propagate on the surfaces then that will be the major motive for equipments failures. In this paper we have investigated the seriousness of deflection and deformation of the cracked pressure vessel. The reason for this study is as we understood that the effect of deformation around the cracked surface is one of the major reasons for faster crack propagation. Also the chances of failure on the usual deflecting area are more. So if any crack already generated, how the vessel body would behave during operation is investigated using FEM simulation. And the behavior of vessel body and the obtained deflections and deformations results are presented in the below theory.

Keywords: ASME, Crack Propagation, Finite Element Analysis, Load Confrontation, Stress Intensity Factor, Failures Vessels. Linear Elastic Fracture Mechanics, Elastic Plastic Regions, Yielding Fracture Mechanics, Crack Tip Opening Displacement.
1. INTRODUCTION

This paper explains the method of estimating the deformation and deflections using FEM for the internally cracked pressure vessel due to internal pressure loading condition. The direction of the crack is in axial. The solid model of the vessel body has been generated along with crack using the software called SOLIDWORKS. Then the model has been imported in to the finite element simulation software COSMOS. The axial MODE-I type [4] crack has been considered for this study. For knowing the entire deformation and deflections around the cracked surfaces, we have taken full 3-D model for simulation. As far as pressure equipments are concerned, if any crack is generated, that will propagate endlessly with respect to the loading conditions and finally the system gets fail. We have taken 300mm thickness vessel with 150mm internal axial crack for analysis. The considered crack angle and dimension are as follows in Figure number [3]. The reason for the thicker vessel geometry chosen is to estimate deflection without any modeling complexity. For example if we take thinner vessel geometry, the development of crack is not so frank, that means the modeling of crack on thinner surface is bit difficult than thicker surface, nevertheless the behavior of thicker geometry simulation is more appropriate than thinner geometry. However In real scenario, the deflection around small cracks could be found easily using manual equations, mean time, if crack length is more; the manual estimation of deflection might not give exact solution all time because the deflection in bigger crack is always higher. But the physical behaviors of vessel shapes of deflection and the deformations for taken body might not viable using manual calculation methods. For this kind of problems, the finite element simulation could be applied for solving critical problems. Thus in this case for getting exact results, we have performed this study using COSMOS simulation for setting up the guideline procedure for crack displacement. The taken modeling and messing of geometry are as follows in Table-1 and Figure [3]. And the considered applied loads on the vessel surface in mentioned the below Figure [3]. The methodology of simulation with respect to the model messing, loading, crack modeling and result interpretations are as follows below in details. Our realization through this study with respect to crack deflections and its behavior in solid body are also presented below.

2. GEOMETRY AND MODELING

The geometry for finite-element modeling has been generated along with internal cracks. The exact shape of the model considered for simulation is being taken from the original physical test model shown in the Figure No. [1]. The baseline dimensions including crack and vessel ID, OD and thickness were taken in accordance with the original physical model of the vessel. Please refer the Figure No [3] for details. The considered 3-D model for FEA simulation is also shown in the below Figure No’s. [2] and [3]. Since the model is axis symmetry, half portion of the model is good enough for simulation but for knowing the exact crack and deflection effects, we have considered 3D full model as a pattern for simulation. In contrast with half model, the full modal would give better stress distribution, deformation results. The solid modeling of crack inside the wall thickness has been employed with the following manipulations. Since the type of crack is MODE-I, Initially we generated lesser opening cracks which were lesser of 35 deg in angle. The considered crack angles are 15deg, 20deg,
25deg, 30deg, 35deg and 40deg. But from the FEM simulations, we understood that the load transformations inside small cracks (for above said crack dimensions) are difficult and the distribution of stresses could not be viewable in details. Thus for getting good result and optimization, we made 45 deg crack for analysis. The advantage of 45 deg crack is the load can be transferred easily in comparison with smaller cracks. Also the stresses and deflection findings can be viewable. It would be easy for FEM manipulations and experiments for this type of problems.

![Figure 1 The Physical Model of ASME Pressure Vessel Considered for Simulation](image)

### 3. FEM MESHING ON THE MODEL

The model has been messed using high quality COSMOS mid-side nodded elements. The details of the elements are given in the following Table No [1]. There was no surface contacts were considered in this analysis. The cracked areas are messed using same type of element but the sizes of element are smaller than general mesh. The reason is, since the cracked areas are critical, we can’t get the stresses linearly in the cracked surfaces. But, for messing using FEM would give better result than ordinary messing. Load transformation using fine mesh would be the best way for extracting stresses in the critical cracked surfaces. So we have selected small 50mm element size for solid body general mesh and 20mm size for cracked areas. The considered mesh tolerance is as 2.52mm. For checking the Jacobean transformation, we have selected 30 points throughout the solid body for checking Jacobean transformation of all elements. There were different types of quality of messing is available in COSMOS software. But the selection of quality is purely based on the type of the problem, if the problem is critical, for example FEM for cracked areas, shear areas and etc, high quality mesh is always advisable. So we have chosen high quality mesh for this analysis. The total number of elements in the messed model is 20586 and the total numbers of nodes are 41948. The details of meshing are as follows below for short reference.

<table>
<thead>
<tr>
<th>DESCRIPTION</th>
<th>TYPE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mesh Type</td>
<td>Solid Mesh</td>
</tr>
<tr>
<td>Jacobian Check</td>
<td>30 Points</td>
</tr>
<tr>
<td>Element Size</td>
<td>50mm</td>
</tr>
<tr>
<td>Mesh Tolerance</td>
<td>2.52mm</td>
</tr>
<tr>
<td>Quality of Mesh</td>
<td>High</td>
</tr>
<tr>
<td>No. of Elements</td>
<td>20586</td>
</tr>
<tr>
<td>No. of Nodes</td>
<td>41948</td>
</tr>
</tbody>
</table>

Table 1 Details of Meshing
4. MATERIAL PROPERTIES

The type of material is Linear Isotropic Plain Carbon Steel SA 516 Gr.60. The detailed properties of this material are as tabulated in the below Table-II. These properties were directly taken from ASME SECTION-II [2] and [3]. The wide-range of general properties of plain carbon steel has been listed in the below table for better understanding. We also
considered the properties of thermal expansion, thermal conductivity and specific heats for the taken material because we understood that the behavior of solid body under thermal loading would give greater effect on solid bodies. However in this analysis we have not focus the temperature distribution on cracked faces but we applied internal temperature effect on inside wall surfaces for knowing the exact peak stress effect on the cracked areas.

**TABLE 2** Summary of Material Properties for SA-516 GR.60

<table>
<thead>
<tr>
<th>Property</th>
<th>Description</th>
<th>Value</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ex</td>
<td>Elasticity Modulus (1 St Direction)</td>
<td>30463138</td>
<td>Psi</td>
</tr>
<tr>
<td>NUXY</td>
<td>Poisson’s Ratio (XY Direction)</td>
<td>0.28</td>
<td>NA</td>
</tr>
<tr>
<td>Gxy</td>
<td>Shear modulus (XY Direction)</td>
<td>11459943</td>
<td>Psi</td>
</tr>
<tr>
<td>Dens</td>
<td>Mass Density</td>
<td>0.007800</td>
<td>Kg/cm³</td>
</tr>
<tr>
<td>SigXT</td>
<td>Tensile Strength (X Direction)</td>
<td>57999.785</td>
<td>Psi</td>
</tr>
<tr>
<td>Sig YLD</td>
<td>Yield Strength</td>
<td>31999.931</td>
<td>Psi</td>
</tr>
<tr>
<td>ALPx</td>
<td>Coefficient of Thermal Expansion</td>
<td>1.3 exp-005</td>
<td>degC</td>
</tr>
<tr>
<td>KX</td>
<td>Thermal Conductivity</td>
<td>0.43</td>
<td>W/(cm.K)</td>
</tr>
<tr>
<td>C</td>
<td>Specific Heat</td>
<td>105.162</td>
<td>Cal/Kg.C</td>
</tr>
</tbody>
</table>

5. LOADS AND BOUNDARY CONDITIONS

The bottom portion of the model has been arrested in all X, Y, Z directions. The translations as well as rotations along with these directions were also arrested. The top of the model is arrested in X, Z directions and the Y direction is free to transfer cylindrical thrust force. However, we allowed translation as well as rotation in the top Y portion. Please refer Figure Numbers [4] and [5]. The Figure [4] is showing the restrained areas and Figure number [5] is showing the way how the software was restrained the degrees of freedom. Figure [6] is showing the pressure loads wherever we applied inside the cracked cylinder for simulation. Also we can see from Figure [6] how the loads were distributed on the cracked areas. Figure [7] is the 3D view of the load applied areas. Since we arrested all directions in bottom, the developed Stress Intensity distribution on the bottom would not be linear. Thus we decided to negligence the bottom Stress Intensity value of these models.

![Figure 5 Boundary Conditions of the Vessel](image-url)
6. BASIS FOR MODE-I TYPE CRACK SIMULATION

The basis for Mode-I type crack simulation is derived and compared with various fracture mechanics research papers as well as from existing journals. C. E. Inglish had used the elliptical crack model in his fracture mechanics research paper shown in Figure [9] and the Equations [1]. Later this was adopted by A. A. Griffith for his research work in the same field. He considered the case in which c=0 when the ellipse degenerates to a straight line of length 2a. He used the equation [2] to estimate the stress intensity factor. After Griffith, Irwin has extended Griffith approach into metals including the energy by the local plastic flow. Irwin developed energy release rate concept which is related to Griffith theory using the below Equation [3].

\[
\left(\sigma_\eta\right)_{\eta=0,\pi}^{\text{max}} = 2\sigma_0(a/b) \\
\left(\sigma_\eta\right)_{\eta=(\pi/2)(3\pi/2)}^{\text{min}} = 2\sigma_0(b/a)
\]

\[
K_1 = P_0 \sqrt{a}
\]
In the extension of above, the linear elastic fracture mechanics approach for evaluating stresses, deformations and displacements associated with each fracture mode follows the Griffith-Irwin theory [10], [12] based on an elastic solution derived by Westergaard [9] approach. Their approach, the stress field near a crack tip can be divided into three basic types, each associated with a kinetic movement of two crack surfaces. These displacement modes are illustrated in Figure [8], [9] and [10] denoted as an Opening Mode-I, an edge Sliding Mode-II and a tearing Mode-III.

The opening mode Mode-I is associated with local displacements in which the crack surfaces move directly apart. The edge-sliding mode Mode-II is characterized by displacements in which crack surfaces slide over one another perpendicular to the leading edge of the crack. The tearing mode is Mode-III is defined by the crack surface sliding with respect to one another parallel to the leading edge. Mode I and II can be regarded as plane extensional problems of the theory of elasticity and mode III, the pure shear problem. Elastic fracture mechanics theory postulates that a crack begins to propagate when the stress concentration at the crack tip reaches a critical value. This value can be reflected in a parameter called SIF (Stress Intensity Factor), usually denoted as K. The plane elastic state of stresses in the vicinity of the crack tip can be expressed in terms of a local polar co-ordinate system, r, θ in terms of mode I and II [13].
Figure 10 Detail Modes of Crack Displacements

Figure [10] is showing the behavior of crack which is propagated due to pulling tension load. In the above Figure you can identify the effect of tension load and its subsequent moment direction. With respect to the loads, the equations for stresses along xx, yy and xy can be extracted as follows:

\[
\sigma_{xx} = K_I / \sqrt{2r} \cos(\theta/2)(1 - \sin(\theta/2)\sin(\theta/2)) - K_{II} / \sqrt{2r} \sin(\theta/2)(2 + \cos(\theta/2)\cos(\theta/2))
\]

\[
\sigma_{yy} = K_I / \sqrt{2r} \cos(\theta/2)(1 + \sin(\theta/2)\sin(3\theta/2)) - K_{II} / \sqrt{2r} \cos(\theta/2)\sin(\theta/2)\cos(\theta/2)
\]

\[
\sigma_{xy} = K_I / \sqrt{2r} \sin(\theta/2)(\cos(\theta/2)\cos(3\theta/2)) - K_{II} / \sqrt{2r} \cos(\theta/2)(1 - \sin(\theta/2)\cos(3\theta/2))
\]

\[
\sigma_{zz} = \nu(\sigma_{xx} = \sigma_{yy})
\]

\[
\sigma_{yy} = \sigma_{zz} = 0
\]

In the plane stress state of displacements in the vicinity of the crack tip can be expressed as using the following equations [8] and [9].

\[
u = K_I / G \sqrt{r/2} \cos(\theta/2)(1 - \nu/1 + \nu + \cos^2(\theta/2)) + K_{II} / G \sqrt{r/2} \sin(\theta/2)(2/1 + \nu - \cos^2(\theta/2))\sin(\theta/2)
\]

\[
v = K_I / G \sqrt{r/2} \sin(\theta/2)(2/1 + \nu - \cos^2(\theta/2)) + K_{II} / G \sqrt{r/2} \cos(\theta/2)(1 - \nu/1 + \nu + \sin^2(\theta/2))
\]

Apart from the above equations [8] and [9], in the elastic plastic regimes, the J-contour integral is extensively used in fracture mechanics analysis, as both the energy and stress intensity based criteria’s, for determining the onset of the crack growth. J-integral for the line counter surrounding the crack tip can be rewritten as using the Equation [10] and the appropriate moment direction is as shown in Figure [11]. In this Figure we can see the direction of crack as well as moments in accordance with the loading condition.
In the above equation \( w = \int \sigma_{ij} \varepsilon_{ij} \) is the strain energy density \( \sigma_{ij} \) and \( \varepsilon_{ij} \) as stress and strain tensors), \( T_i = \sigma_{ij} n_j \) are the components of the traction vector which are acting on the counter, \( u_i \) are the displacement components and \( d_s \) is the length increment along the counter \( \Gamma \). The J-integral is nothing but a non linear energy release that is defined by

\[
J = -\frac{d\Pi}{dA}
\]

Where \( \Pi = U - V \)

In which \( \Pi \) is the total potential energy \( U \) is the strain energy release, and \( V \) is the external work.

7. J-INTEGRATION ON CRACKED SURFACES

From the understanding of above elastic plastic fracture research work; we decided to use J-estimation method as a basis for checking our FEM simulation in cracked area. For any application of J-integral, Romberg-Osgood [4] relation can be used to solve the simulations. The equation of Romberg Osgood can be written as follows [12].

\[
\frac{\xi}{\xi_0} = \frac{\sigma}{\sigma_0} + \alpha \left[ \frac{\sigma}{\sigma_0} \right]^n
\]

\[
\xi = (\frac{\sigma}{E}) + (\frac{\sigma_y}{F})
\]

The above equation has four material constants \( n, \alpha, \sigma_0, \xi \) in a place of the three constants. Here \( E \) is the young’s modulus of the material and \( n, F \) are the other two material constants. This kind of equation formation is acceptable because \( \alpha \) is the dimensional number. If the equation expressed in another form cannot have more independent constants, there should be some relation between all constants. In fact, \( \sigma_0 \) can be chosen anywhere on the elastic portion of the \( \sigma - \xi \) curve and the corresponding \( \xi_0 \) is evaluated by the relation using \( \sigma_0 = E \xi_0 \). Many investigators prefer to choose \( \sigma_0 \) as the yield stress \( (\sigma_{ys}) \) of the material. Realizing a relation exists between \( \sigma_0 \) and \( \xi_0 \) and based on the equations [12] and [13] we could obtain:
With respect to the equations [12] and [13], the equation of \( J_p \) can be expressed as follow equation [14]. This equation could be identified as simple power law equation of material stress strain curve. The variations of pressure load and its effects on the crack also can be estimated using the same equation \( P/P_o \). So the considered equations for our simulations for checking also included the following equation [14].

\[
J_p = \alpha * \sigma_0 * \xi_0 * b_{gl} * h_i [P/P_o]^{\frac{b_{gl}}{n+1}}
\]

\[
P/P_o = (J_p)/\alpha * \sigma_0 * \xi_0 * b_{gl} * h_i (1/n + 1)
\]

So according to old referred research papers, the above mentioned equations would be useful [14] for estimating stresses and load confrontations for the above considered axially cracked pressure vessel. Please see Figure No. [3]. But here the decision is, these equations may not give exact load confrontation and stress extraction values if the crack is very longer in size. So the combination of manual equations using these formula as well as FEM calculation would be the exact choice for finding solution for these kinds of problems. Thus, in this study, we have used both the options to evaluate the exact solution.

**Table 3** \( H_1 \) Value for Internally Pressurized Cylinder with Axial Crack

<table>
<thead>
<tr>
<th>( a/w )</th>
<th>( w/R_i )</th>
<th>( H_1 )</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>1</td>
</tr>
<tr>
<td>1/8</td>
<td>1/5</td>
<td>6.32</td>
</tr>
<tr>
<td></td>
<td>1/10</td>
<td>5.22</td>
</tr>
<tr>
<td></td>
<td>1/20</td>
<td>4.50</td>
</tr>
<tr>
<td>1/4</td>
<td>1/5</td>
<td>7.00</td>
</tr>
<tr>
<td></td>
<td>1/10</td>
<td>6.16</td>
</tr>
<tr>
<td></td>
<td>1/20</td>
<td>5.57</td>
</tr>
<tr>
<td>1/2</td>
<td>1/5</td>
<td>9.79</td>
</tr>
<tr>
<td></td>
<td>1/10</td>
<td>10.5</td>
</tr>
<tr>
<td></td>
<td>1/20</td>
<td>10.8</td>
</tr>
<tr>
<td>3/4</td>
<td>1/5</td>
<td>11.00</td>
</tr>
<tr>
<td></td>
<td>1/10</td>
<td>16.1</td>
</tr>
<tr>
<td></td>
<td>1/20</td>
<td>23.1</td>
</tr>
</tbody>
</table>

For manual checking, we considered the hardening exponent of the material \( n \) is 7 for the material SA-516 Gr.60, and the material constant of Ramberg-Osgood relation \( \alpha \) is 6.2. And the \( h_1 \) value is considered as per the above fracture mechanics Table-III as well as earlier research paper guidelines [12]. With the help of all these values and estimations plus our FEM simulations, the obtained load confrontation, stress intensity, deflection and deformation values were plotted in the below graphs Figure 25 and Figures [12]-[23]. We found that the obtained FEM results were exact reasonable with respect to the manual calculation using the equations [14] with respect to the old research papers. So we concluded that the approach of obtaining stresses and load confrontation using FEM is reasonable. For example the range of stresses obtained through manual are around 360 to 380MPa however the allowable stress limit of the material with the taken temperature condition is only 220MPa. The variations of stresses like 360 to 380 are due to the over load application on the vessel and due to crack.
The obtained FEM stresses through COSMOS are around also in the ranges of 370 to 380. The developed stresses along the directions XX, YY, ZZ and XY and displacements u and v also were verified using the equations [1], [2], [3], [4], [5], [6], [7], [8], [9], [10], [11], [12], [13] and [14]. The obtained stresses and displacements values analytical as well as manual methods are very close to each other and the percentage of the error is also less. So the results are acceptable. However, we performed our study only based on the FEM results which we obtained from COSMOS only in top portion of vessel. We neglected the bottom portion results as we arrested bottom in all directions X,Y,Z so the developed stress in bottom are not real. The recommendations from this study is only based on the results we obtained in below mentioned Figures [12], [13], [14], [15], [16], [17], [18], [19], [20], [21], [22] and [23].

8. FEM ASSESSMENT OF DEFLECTION AROUND THE CRACKED SURFACE

![Image 12](http://www.iaeme.com/IJMET/index.asp)

**Figure 12** Deflection plot for 10Kgf/cm² Internal Pressure Load

![Image 13](http://www.iaeme.com/IJMET/index.asp)

**Figure 13** Deflection Plot for 20Kgf/cm² Internal Pressure Load
Figure 14 Deflection Plot for 30Kgf/cm$^2$ Internal Pressure Load

Figure 15 Deflection Plot for 50Kgf/cm$^2$ Internal Pressure Load

Figure 16 Deflection Plot for 60Kgf/cm$^2$ Internal Pressure Load

Figure 17 Deflection Plot for 65Kgf/cm$^2$ Internal Pressure Load

9. FEM ASSESSMENT OF DEFORMATION AROUND THE CRACKED CYLINDER
Figure 18 Deformation Plot for 10Kgf/cm² Internal Pressure Load

Figure 19 Deformation Plot for 20Kgf/cm² Internal Pressure Load

Figure 20 Deformation Plot for 30Kgf/cm² Internal Pressure Load

Figure 21 Deformation Plot for 50Kgf/cm² Internal Pressure Load
10. RESULTS AND REALIZATION OF MODE-I TYPE OF CRACK UNDER DEFLECTION AND DEFORMATION

A complete parametric study has been made using axially cracked pressure vessel. The stress intensity, load confrontation, deformation and deflection shapes of the taken vessel were extracted from the analysis around the cracked surfaces. From the result, we found that the direction of stress intensity and load confrontations are as purely depends on the direction of applied load. If we increase the load, subsequently stresses and crack propagation also would increase. With respect to deformation and deflection shapes, both are also depends on the application of loads. The considered load for our simulation is 10, 15, 20, 25, 30, 40, 50, 55, 60 and 65Kgf/cm$^2$ and we found the taken model would withstand maximum of 65Kgf/cm$^2$ of internal pressure load. A dynamic fracture mechanics model was developed to predict the crack deflection vs. penetration at an interface in carbon steel material [8]. The homogeneous material results is also compared with this SA 516 Gr.70 based on the research study material [8]. To determine whether an incident crack will penetrate the interface, the normalized energy release rate for crack deflection and penetration is expressed as a function of incident angle $\beta$. An appropriate criterion for dynamic crack penetration through the interface is a ratio comparison between the dynamic energy release rates (driving force) and the fracture toughness’s (material resistance).

$$G^d(\beta, \nu_j)/G^d(\nu_i) \leq \Gamma^IT(\nu_j)/\Gamma^IT(\nu_i)$$

Figure 22 Deformation Plot for 60Kgf/cm$^2$ Internal Pressure Load

Figure 23 Deformation Plot for 65Kgf/cm2 Internal Pressure Load

Figure 24 Crack Penetration and Crack Deflection
Where \( v_1 \) and \( v_2 \) are the incident crack speed and the possible deflected crack speed respectively, is the dynamic fracture toughness of the chosen material while is the interfacial fracture toughness. We found the movement of crack and its deflection based on the Figure [24].

In this case, the bond involved in the inner material and its interfacial fracture toughness is around 130MPa. The incident crack speed was about 300 m/s. The fracture toughness of at this crack speed is about 380MPa, Figures [12] to [23] gives graphic representation of the inequality due to higher crack speed.

Indeed, according to the criterion, deflection into the interface will take place at \( 0<\beta<60^\circ \) while the interface will be penetrated for \( 59<\beta<90^\circ \) during various loading conditions. It should be also noted that all load cases displayed in Figure [12] to [24] are consistent in this study.

![Figure 25 The Obtained Load Confrontation and Intensity](image)

**Table 4 Summary of Stress Intensity and Load Confrontations**

<table>
<thead>
<tr>
<th>Load Confrontations in Kgf/Cm²</th>
<th>Stress Intensity in Mpa on the Cracked Areas</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>54</td>
</tr>
<tr>
<td>20</td>
<td>109</td>
</tr>
<tr>
<td>30</td>
<td>164</td>
</tr>
<tr>
<td>50</td>
<td>273</td>
</tr>
<tr>
<td>60</td>
<td>327</td>
</tr>
<tr>
<td>65</td>
<td>355</td>
</tr>
</tbody>
</table>
REFERENCES


http://www.iaeme.com/IJMET/index.asp 911  editor@iaeme.com