Flexible Pipe Pressure Armour: Stress and Fatigue Analysis

1- Introduction

The AAJL Engenharia EIRELI is a company that develops specific structural models based on the Finite Element Method. Thus, the AAJL Engenharia intends to develop models according to the necessity of the clients. The mean cost of a project in Brazil is about R$ 150,00/h, including invoice issuance.
In this article the AAJL Engenharia presents a simple example of a 2.5 in ID Flexible Pipe subjected to the self weight and wave loads for a Global Dynamic Analysis. Thus, the AAJL Engenharia has performed a Global Analysis and after a Local Stress Analysis. For the numerical analysis, the AAJL Engenharia used the in house software ANATEMP. This software has been developed and the AAJL Engenharia is inviting companies to develop together. See the link: www.aajl-engineering.com. For commercial works the AAJL Engenharia intends to use the OrcaFlex software.

2 - Global Analysis

2.1 - Input Data:
Structure: 2.5 in ID Flexible Pipe;
Water Depth: 1,000 m;
Total Riser Length: 2,605 m.

Layers:

<table>
<thead>
<tr>
<th>Carcass (Adpted from Figure 2)</th>
<th>Pressure Sheath</th>
</tr>
</thead>
<tbody>
<tr>
<td>Material: Stainless Steel AISI-304</td>
<td>Material: PA11</td>
</tr>
<tr>
<td>Longitudinal Young's Modulus: 205 GPa</td>
<td>Longitudinal Young's Modulus: 284 MPa</td>
</tr>
<tr>
<td>Poison: 0.3</td>
<td>Poison: 0.46</td>
</tr>
<tr>
<td>Outer Diameter: 70.2 mm</td>
<td>Outer Diameter: 80.1 mm</td>
</tr>
<tr>
<td>Laying Angle: 87.5° (Clockwise)</td>
<td>Thickness: 4.95 mm</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Pressure Armour (Adpted from Figure 3)</th>
<th>Inner Tensile Armour</th>
</tr>
</thead>
<tbody>
<tr>
<td>Material: Carbon Steel</td>
<td>Material: Carbon Steel</td>
</tr>
<tr>
<td>Longitudinal Young's Modulus: 205 GPa</td>
<td>Longitudinal Young's Modulus: 205 GPa</td>
</tr>
<tr>
<td>Poison: 0.3</td>
<td>Poison: 0.3</td>
</tr>
<tr>
<td>Outer Diameter: 92.5 mm</td>
<td>Outer Diameter: 101.5 mm</td>
</tr>
<tr>
<td>Laying Angle: 87.5° (Clockwise)</td>
<td>Laying Angle: 35° (Clockwise)</td>
</tr>
<tr>
<td>Profile: 6 mm x 3 mm</td>
<td>Profile: 6 mm x 3 mm</td>
</tr>
<tr>
<td>Wires: 40</td>
<td>Wires: 40</td>
</tr>
</tbody>
</table>
For each layer, it was calculated the weight per meter and the total weight of the pipe was estimated in 164.41 N/m.

**Outer Tensile Armour**
- Material: Carbon Steel
- Longitudinal Young's Modulus: 205 GPa
- Poison: 0.3
- Outer Diameter: 110.5 mm
- Laying Angle: 35° (Counter-Clockwise)
- Profile: 6 mm x 3 mm
- Wires: 44

**Outer Sheath**
- Material: PA11
- Longitudinal Young's Modulus: 284 MPa
- Poison: 0.46
- Outer Diameter: 120.5 mm
- Thickness: 5 mm

*Figure 2 - Carcass Profile.*

*Figure 3 - Pressure Armour Profile.*
2.2 - Analytical Results (Only self weight)

Table 1 - Equivalent pipe to the flexible pipe.

<table>
<thead>
<tr>
<th>Outer Diameter (m)</th>
<th>Inner Diameter (m)</th>
<th>Weight in Air (N/m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.1205</td>
<td>0.0805</td>
<td>164.41</td>
</tr>
</tbody>
</table>

Suspended Legth = 1,203.80 m

Horizontal Force = 34.92 kN

Vertical Force = 197.91 kN

Total Force = 200.97 kN

Vertical Angle = 10 degree

2.3 - Dynamic Analysis (Finite Element Analysis)

Table 2 - Equivalent pipe to the flexible pipe.

<table>
<thead>
<tr>
<th>Outer Diameter (m)</th>
<th>Inner Diameter (m)</th>
<th>Density (kg/m³)</th>
<th>Axial Stiffness (kN)</th>
<th>Flexural Stiffness (N.m²)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.1205</td>
<td>0.0805</td>
<td>2,654</td>
<td>112,730.4</td>
<td>1000</td>
</tr>
</tbody>
</table>

From the configuration presented on Figure 1, it was executed a dynamic analysis regarding the incidence of a regular wave (H = 2.73 m and T = 8.33 s), on the direction X. The top node (node 81) is located 10 m above mean water level. This node is free only on rotation about Z, and the node at the anchor is restricted on all 6 degree of freedom.

The forces at node 66, coordinate (13.37;-63.07;0), were extracted along the time, and after, the variation of force were obtained: ∆FX = 123.26 N and ∆FY = 17.32 N. The maximum applied forces at node 66 were used in the Local Analysis described on Section 3.

3 - Local Analysis of Pressure Armour (Finite Element Analysis)

3.1 - Input Data

Table 3 - Pressure armour data.

<table>
<thead>
<tr>
<th>Outer Diameter (m)</th>
<th>Density (kg/m³)</th>
<th>Longitudinal Young's Modulus (GPa)</th>
<th>Yield Stress (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0925</td>
<td>7,850</td>
<td>205</td>
<td>900(1)</td>
</tr>
</tbody>
</table>

Note 1: See reference [1].
3.2 - Mesh Data and Restraints in Nodes

- Elements: 1,288 (see Figure 5);
- Nodes: 625;
- Profile: See Figure 3;
- Profile Thickness: 5 mm.
- Special elements were used on the region of contact between the two profiles analysed.
3.3 - Load Case on the Flexible Pipe

Figure 6 - Scheme of the Load Case on the Flexible Pipe.
3.4 - Load Case on the Finite Element Model Developed

Internal Pressure = 20 MPa (See reference [1])
External Pressure = 10.06 MPa.

![Figure 7 - Load Case on the Finite Element Model Developed.](image)

3.5 - Deformed Shape

Figure 8 presents the deformed shape on a scale of 1:100. The major Von Mises Stress, 351.85 MPa, was located into the red circle on the Figure 8. It can be observed that the combination of forces on the X direction was more expressive than the combination on the Y direction.

![Figure 8 - Deformed Shape, Scale of 1:100.](image)
3.6 - Allowable Stress for Extreme Operation Condition

The Table 4 presents the Allowable Stress for Extreme Operation Condition. It is possible to identify that the maximum value calculated for the Von Mises Stress (351.85 MPa) is below the Allowable Stress (765 MPa), indicating that the forces applied on the pressure armour layer are in accordance to the structural limits. It is important to remember that this work is a simple example. For a real work, realistic data must be taken into account, and extreme environmental condition should be simulated. Furthermore, irregular waves in several directions should be evaluated.

Table 4 - Allowable Stress for Extreme Operation Condition.

<table>
<thead>
<tr>
<th>Element</th>
<th>Von Mises Stress (MPa)</th>
<th>Yield Stress (MPa)</th>
<th>Allowable Stress (MPa) - API 17J</th>
<th>Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>21</td>
<td>351.85</td>
<td>900</td>
<td>765&lt;sup&gt;(1)&lt;/sup&gt;</td>
<td>0.46</td>
</tr>
</tbody>
</table>

Note 1: API 17J: Allowable Stress = Uf x Yield Stress (Uf = 0.85).

3.7 - Deterministic Fatigue Analysis

According to the DNV rule RP-C203, Table 2.4.5, S-N Curve F, and the Stress Variation of 17.95 MPa (From numerical analysis), the number of cycles is $2.13 \times 10^{11}$. Thus, the fatigue life was estimated in $9.55 \times 10^5$ years. Regarding an operation life of 25 years, and a safety factor of 10, it can be concluded that the pressure armour modeled is in accordance to the limits of fatigue. See below the estimated S-N Curve, on Figure 9.

![Figure 9 - Estimated S-N Curve.](image)
4 - Conclusion

Figure 8 presents the deformed shape on a scale of 1:100. The major Von Mises Stress, 351.85 MPa, was located into the red circle on the Figure 8. It can be observed that the combination of forces on the X direction was more expressive than the combination on the Y direction.

The Table 4 presents the Allowable Stress for Extreme Operation Condition. It is possible to identify that the maximum value calculated for the Von Mises Stress (351.85 MPa) is below the Allowable Stress (765 MPa), indicating that the forces applied on the pressure armour layer are in accordance to the structural limits. It is important to remember that this work is a simple example. For a real work, realistic data must be taken into account, and extreme environmental condition should be simulated. Furthermore, irregular waves in several directions should be evaluated.

To be more realistic, numerical simulations regarding irregular waves must be accomplished in order to execute a stochastic fatigue analysis, instead of a deterministic one. Moreover, it must be regarded the imposed motions from the vessel and load condition concerning more than one direction. The AAJL Engenharia reminds that the presented work is only a simple example. For a contract work, other modeling considerations will be evaluated. Moreover, probably the OrcaFlex will be used to execute the dynamic offshore analysis, regarding irregular waves applied in different directions, for the global analysis. And for Local analysis the ANSYS or SAP2000 will be used.

10 - References


Aluizio de Amorim Pacheco
aluizio.pacheco@aajl-engineering.com

AAJL Engenharia EIRELI
www.aajl-engineering.com