Applied Solid Mechanics Summative Part A

Reuben Marland 1935805

EXECUTIVE SUMMARY

To reduce environmental damaging, Liquified Natural Gas (LNG) carriers fit a large Gas Combustion Unit (GCU), which burns boil-off gas that cannot be either consumed by ship engines or re-liquified and returned to the tanks. Due to the scale of the GCU, the modules are constructed in a factory onshore, to later be lifted into place on the LNG carrier. At a mass of 36,000kg, the GCUs require strong attachment points to be lifted and installed. The design presented is a lifting lug. 6 lifting lugs are welded onto the GCU and will each be able to fit a lifting shackle. This report evaluates if one of the lifting lugs can withstand the distributed weight of the GCU. Using Finite Element Analysis (FEA) software Autodesk Inventor Nastran (AIN), the lifting lug is analysed under static stress. The maximum Von Mises stress, excluding the contact region and the vertical tip deflection, were calculated as 129.62MPa and 0.09mm respectively. Safety factor across the component is presented in Figure 11, highlighting multiple areas where the component does not meet typical equipment safety factor requirements. Due to standard requirements not being met, the design is currently not usable and must be adjusted further.

ANALYSIS METHODS

Autodesk Inventor with Nastran

AIN is analysis software previously used by NASA whilst testing out improved space vehicles and hence has a high degree of accuracy. Autodesk acquired the software to add to its powerful library of analysis tools. After acquisition, the program was highly tested through a comprehensive verification program and documentation set using NAFEMS (1). The Autodesk 3D modelling space is familiar to complete analysis under and was also chosen since the highest probability of having successful results relied heavily in a correct model, which Autodesk provides ease in construction of.

Inputs

AIN takes a design created in Autodesk Inventor and allows the user to input values, boundary conditions, loads and the type of stress situation in the analysis space, to output Von Mise's stress, displacement and the safety factor.

The constraint input for the model is shown as two fixed points in Figure 1. The material value inputs are shown in Figure 2. The loading input is shown in Figure 3.

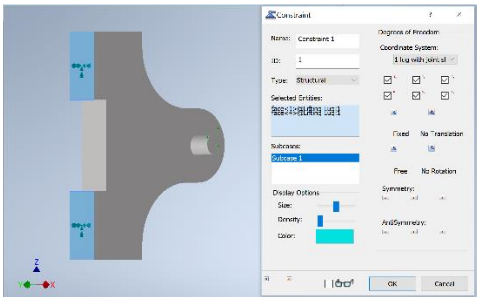

Figure 1

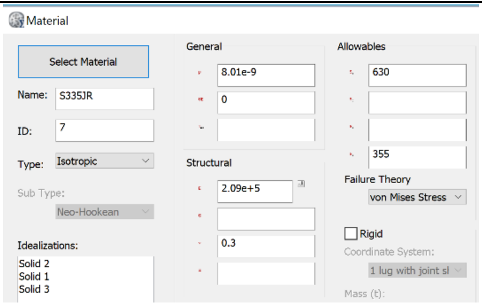

Figure 2

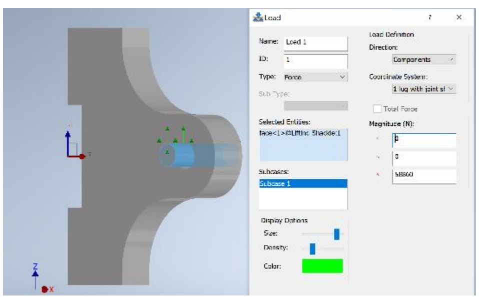

Figure 3

Loading scenario

A static simulation was chosen assuming the GCU is installed at a slow pace, so the cables attached to the lifting shackles are always at the same constant tension.

As shown in Figure 3, there is an additional component running through the lifting lug, the shackle. The shackle was not involved in results and is there to apply the load in a realistic scenario. The diameter of the connecting surface is 0.02 m, the diameter of shackle used was 0.018. This was an engineering estimation assuming tolerances of maximum 0.005 m in both components. The mass of the GCU is 36,000kg. This mass is distributed about 6 different lifting lugs. Assuming g = 9.81ms^{-2}, The force P acting on a singular lifting lug is calculated with equation

$$P = \frac{36000 * 9.81}{6} \qquad (1)$$

resulting in a force P of 58860N. The contact type between the shackle and lug was set as sliding with no separation, meaning the two components would continuously make contact as the force is applied. The shackle is thus allowed to slide around the connection surface so it can continuously act upon the most vertical points of the surface as the lug bends under stress.

To make sure the contact point between the lifting lug and the shackle was a smooth fit, project midside nodes was enabled as shown in figure (6), approximately curving the nodes around curvatures in the model, creating a more realistic contact between the shackle and lug.

Meshing

The universal element size for the model was chosen at 1mm as shown in Figure 5. This element size was chosen because there will be a high degree of accuracy as 1mm is a fine grain and when the overall model element size was decreased further the processing time for results was inadequate. The region of highest stress in the model resulted within section 3, Figure 8, of the lifting lug as shown in Figure 10. This area was determined previously by conducting initial FEA analysis with much larger grain sizes, though the resulting stress from these tests were not of optimum accuracy. An additional Mesh control was introduced surrounding this area where the element size was chosen to be 0.85mm which, as shown in figure (4), densely populated the region further whilst still allowing processing of the data.

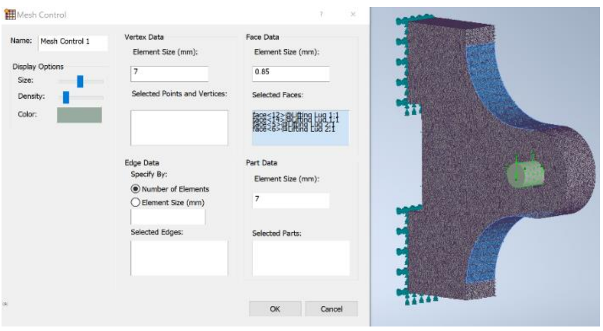

Figure 4

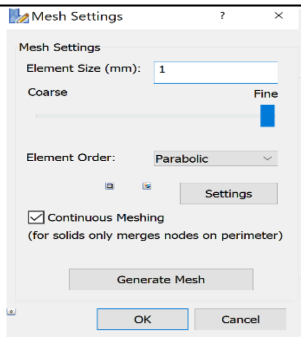

Figure 5

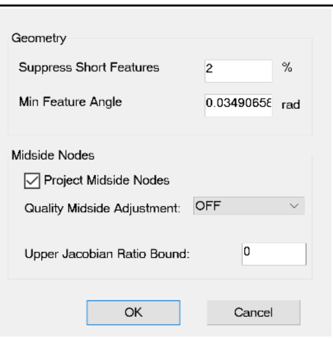

Figure 6

ANALYSIS VALIDATION

Prior use of AIN

The Helicopter Bracket Report by Reuben Marland (2) used AIN to complete analysis of a similar static stress situation with a helicopter bracket pad. The model had simpler geometry though the same static stress simulation was used whilst applying load at the end of a cantilever-based shape. This stress was validated using the analytical solution for the maximum tip deflection of a cantilever beam.

The equation for the maximum tip defection of a cantilever beam for a point load at the end is

$$\delta_{max} = \frac{PL^4}{8EI} \tag{2}$$

where δ_{max} is the maximum tip deflection, P is the uniformly distributed load, 1635N, L is the length of the beam. 30m E is the Young's Modulus, 200MPa and I is the second moment of Area, $6.75 * 10^{-4}$.

The resultant δ_{max} = 1.226m. The same values were input into AIN which resulted in a displacement of 1211mm. The 1% validation provides validation for the use of AIN.

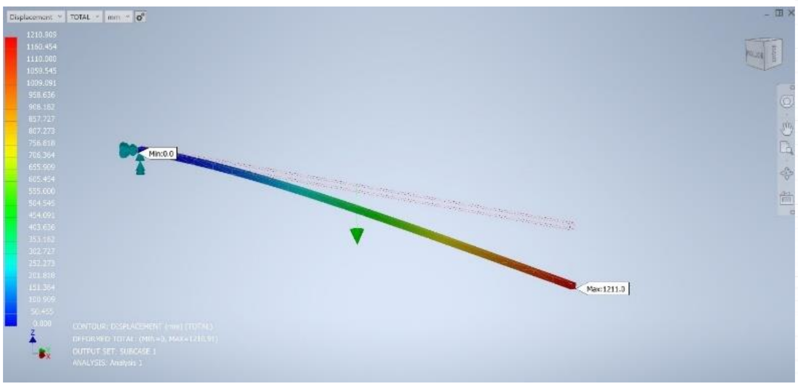

Figure 7

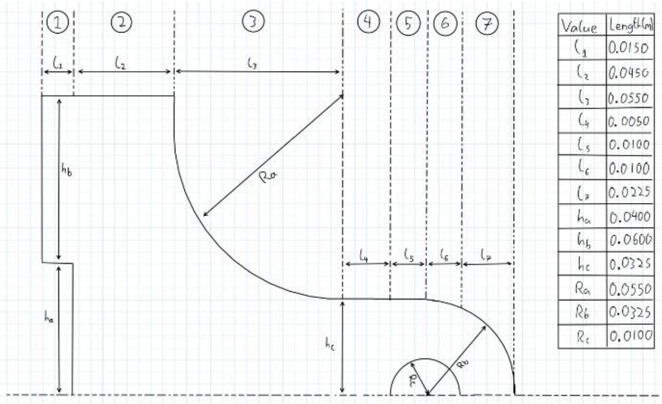

Figure 8

Analytical solution

The loading scenario is different for the lifting lug since the load is not being applied at the tip and the overall shape is more complicated in comparison to a basic cantilever beam. An estimated analytical solution was constructed using Castigliano's Second Theorem (CST).

CST defines that the deflection at the location of an applied load is equal to the partial derivative of the total strain energy with respect to that load with equation:

$$\delta_i = \frac{\partial U}{\partial P_i} \tag{3}$$

where δ_i is the deflection at position i along the beam's neutral axis, U is the total strain energy and P_i is the load at position i along the neutral axis. CST combines with the strain energy due to bending in a beam equation:

$$U = \int \frac{M^2}{2EI} dx \tag{4}$$

where M is the bending moment, x is the distance from the boundary to the applied load, E is the Young's Modulus of the material, and I is the second moment of area over the beam. The combined equation is:

$$\delta_i = \frac{1}{E} \int \frac{1}{I} M \frac{\partial M}{\partial P_i} dx. \tag{5}$$

Since the force was not being applied at the tip of the lifting lug, a fictitious load, Q, is applied at the furthest horizontal point on the lug. The moment and derivative moment equations of the system with both real and fictitious load are respectively

$$M(x) = ((l_1 + l_2 + l_3 + l_4 + l_5 - x) * P + ((l_1 + l_2 + l_3 + l_4 + l_5 + l_6 + l_7) - x) * Q \tag{6}$$

$$\frac{\partial M(x)}{\partial Q} = (l_1 + l_2 + l_3 + l_4 + l_5 + l_6 + l_7) - x. \tag{7}$$

Due to the cross-sectional area changing height throughout the length of the lifting lug, the second moment of area was formed into a function of x This had to be completed by splitting the lug into 5 sections as shown in Figure 8. Further Sections not needed due to load location. For the second moment of area per section to combine for I(x), the Heaviside function is used.

Assuming the cross section will be split up into infinitesimal rectangles integration, the second moment area calculates as:

$$I = \frac{b * h^3}{12} \tag{8}$$

where b = t is the width of the cross section, 0.3m, and h is the height of the cross section, which is a function of x.

The Heaviside function (3) defines as

$$H(x) = \begin{cases} 0, & x < 0 \\ 0.5, & x = 0. \\ 1, & x > 0 \end{cases} \tag{9}$$

The equation for the second moment of area for each section, A, B C, D, E corresponds to equations, 10, 11, 12, 13, 14 ;

$$I_1 = \frac{t * ((2 * (h_a + h_a))^3 - (2 * h_a)^3)}{12} * H(l_1 - x) \tag{10}$$

$$I_2 = \frac{t * (2 * (h_a + h_b))^3}{12} * H(x - l_1) * H(l_1 + l_2 - x) \tag{11}$$

$$I_3 = \frac{t * \left(2 * \left(h_c + \sqrt{{R_a}^2 - (x - (l_1 + l_2))^2}\right)\right)^3}{12} * H(x - l_1 - l_2) * H(l_1 + l_2 + l_3 - x) \tag{12}$$

$$I_4 = \frac{t * (2 * h_c)^3}{12} * H(x - l_1 - l_2 - l_3) * H(l_1 + l_2 + l_3 + l_4 - x) \tag{13}$$

$$I_5 = \frac{t * \left((2 * h_c)^3 - \left(2 * \sqrt{{R_c}^2 - (x - (l_1 + l_2 + l_3 + l_4 + l_5))^2}\right)^3\right)}{12} * H(x - l_1 - l_2 - l_3 - l_4). \tag{14}$$

The second moment of areas for each section combine for I as a function of x

$$I(x) = I_1 + I_2 + I_3 + I_4 + I_5. \tag{15}$$

The integral is formed using equations 5, 6, 7 and 15

$$\delta_{tip} = \frac{1}{E}\int_0^{l_1+l_2+l_3+l_4+l_5} \frac{1}{I(x)} M(x) \frac{\partial M(x)}{\partial Q} dx \tag{16}$$

where δ_{tip} is the vertical deflection at the tip of the bracket and E is the Youngs Modulus, 209GPa.

Using the inputs shown in Figure 8, the tip deflection results as $1.9206 * 10^{-5}$.

The vertical tip deflection given using AIN results as $9.097 * 10^{-5}m$ as shown in figure 9. The two results fall within the same order of magnitude, showing that the results presented with AIN will have a large degree of accuracy. There are multiple reasons to explain the possible difference in value: the sudden change in shape after section 2, along with its curvature in section 3 cannot be accounted for in CST, as only an approximation is produced as the entire shape is estimated to be of a sum of varying second moments of area.

RESULTS

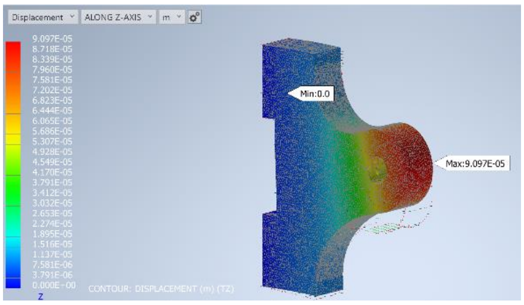

Figure 9

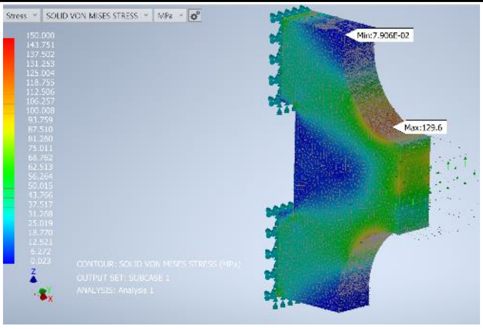

Figure 10

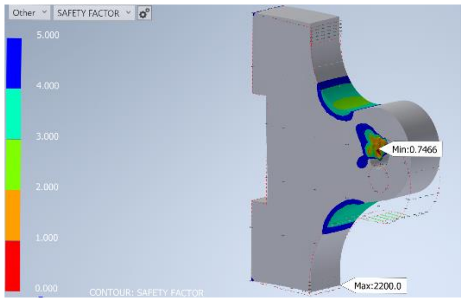

Figure 11

The maximum vertical deflection at the end of the lifting lug that would be expected during the lifting operation is $9.097 * 10^{-5}m$ as shown in Figure 9. The maximum Von Mises Stress experienced, excluding the contact region between the lug and the lifting shackle, is $129.6MPa$ as shown in Figure 10. Assuming the material will fail once the Yield point of Steel EN 10025-2 – S355J, $355MPa$, is reached, Figure 11 highlights the areas of the lug that will experience a safety factor of 5 or less. Any area that is coloured will not meet the typical safety factor requirement of 5 for this type of equipment.

RESULTS ANALYSIS + RECOMMENDATIONS

The most conclusive evidence to use when measuring whether the lifting lug is adequate for the operation is the safety factor results within figure 11. The design details states (4): It is essential that the GCU can be lifted safely during installation, and that the lifting lugs will not break. For lifting equipment of this type, a typical safety factor is: 5. There are multiple areas within the lifting lug that fail to reach this safety factor, specifically the contact boundary between the lifting lug and the shackle would presumably fail under the current loading conditions, with safety factor 0.7644, therefore I must recommend that the lug is not used with its current design of contact with the shackle.

If this issue were resolved, the safety factor in section B, figure (7), still falls within the failing range, approximately 2.5, though the lug would presumably be able to withstand carrying the weight of the GCU since the maximum Von Mises stress in the area is 129.6MPa, though this would not be recommended since if for multiple reasons the stress increased, either due to structural deformity in the shackle or an inappropriate loading operation, the stress could easily increase above the materials Yield Limit and plastically fail.

If the lug must be used in a lifting operation with the knowledge it will exceed its Yield Limit, the alternate failure situation is when the material reaches its Ultimate Tensile strength, 630MPa, this would put the stress within section b, figure 8, above a safety factor of 5.

I would conclusively not recommend this equipment to be used within the operation specified, whilst it may work an infrequent number of times, it is not safe.

REFERENCES

1. Autodesk,"AUTODESK NASTRAN, overview", https://www.autodesk.com/products/inventor-nastran/overview?term=1-YEAR&tab=subscription&plc=PDCOLL
2. Marland, R, "Helicopter Bracket report", https://uob-my.sharepoint.com/:w:/g/personal/op20532_bristol_ac_uk/EV18KjptpztBns74FCUgJEQBK7W4p10CuP8letgweZ5tKw?e=bkOyQP
3. Weisstein, Eric W. "Heaviside Step Function." https://mathworld.wolfram.com/HeavisideStepFunction.html
4. Coules, H, Velichko, A "MENG30011 Applied Solid Mechanics AY 2022/23 coursework assessment", https://www.ole.bris.ac.uk/bbcswebdav/pid-6782762-dt-content-rid-30108780_2/xid-30108780_2

Applied Solid Mechanics Summative Part B

Reuben Marland 1935805

QUESTION 1

How many: a.) elements, b.) nodes and c.) degrees-of-freedom does the mesh in Figure 1 have?

Part a.)

48 Elements.

Part b.)

83 Nodes

Part c.)

2 Degrees of Freedom (DoF) per node, therefore $83 * 2 = 166$ DoF.

QUESTION 2

What is the stiffness matrix of the element shown in Figure 2? You do not need to show a derivation of formulae for the stiffness matrix, but you should show your working.

Coordinates for each node within the Constant Strain Triangle (CST):

Node 1 (x_1, y_1) = (-4.99,-0.83).

Node 1 (x_2, y_2) = (-3.46,0.18).

Node 1 (x_3, y_3) = (-6.18,0.73).

Poison ratio (ν) = 0.35.

Youngs Modulus (E) = 202GPa.

The equation for the area of a triangle is:

$$\Delta = \frac{(x_2y_3 - x_3y_2) + (x_3y_1 - x_1y_3) + (x_1y_2 - x_2y_1)}{2}.$$

Inputting coordinate values results $\Delta = 2.7458$

Matrix **B**(r) is defined as:

$$\boldsymbol{B}(r) = \begin{Bmatrix} y_2 - y_3 & 0 & y_3 - y_1 & 0 & y_1 - y_2 & 0 \\ 0 & x_3 - x_2 & 0 & x_1 - x_3 & 0 & x_2 - x_1 \\ x_3 - x_2 & y_2 - y_3 & x_1 - x_3 & y_3 - y_1 & x_2 - x_1 & y_1 - y_2 \end{Bmatrix}$$

resulting in

$$\boldsymbol{B}(r) = \begin{Bmatrix} -0.1002 & 0 & 0.2841 & 0 & -0.1839 & 0 \\ 0 & -0.4953 & 0 & 0.2167 & 0 & 0.2786 \\ -0.4953 & -0.1002 & 0.2167 & 0.2841 & 0.2786 & -0.1839 \end{Bmatrix}.$$

The digression matrix for an isotropic material in plane strain is:

$$\boldsymbol{D} = \frac{E}{(1+\nu) * (1-2\nu)} \begin{Bmatrix} 1-\nu & \nu & 0 \\ \nu & 1-\nu & 0 \\ 0 & 0 & \frac{1}{2} - \nu \end{Bmatrix}$$

resulting in

$$\boldsymbol{D} = 10^{10}.* \begin{Bmatrix} 3.2420 & 1.7457 & 0 \\ 1.7457 & 3.2420 & 0 \\ 0 & 0 & 0.7481 \end{Bmatrix}.$$

For a CST the stiffness matrix can be calculated using:

$$\boldsymbol{K} = \Delta \boldsymbol{B}^T \boldsymbol{D} \boldsymbol{B}$$

where

$$\boldsymbol{B^T} = \left\{ \begin{matrix} -0.1002 & 0 & -0.4953 \\ 0 & -0.4953 & 0.1002 \\ 0.2841 & 0 & 0.2167 \\ 0 & 0.2167 & 0.2841 \\ -0.1839 & 0 & 0.2786 \\ 0 & 0.2786 & -0.1839 \end{matrix} \right\}$$

$$\boldsymbol{K} = 2.7485.* \left\{ \begin{matrix} -0.1002 & 0 & -0.4953 \\ 0 & -0.4953 & 0.1002 \\ 0.2841 & 0 & 0.2167 \\ 0 & 0.2167 & 0.2841 \\ -0.1839 & 0 & 0.2786 \\ 0 & 0.2786 & -0.1839 \end{matrix} \right\} * 10^{10}.* \left\{ \begin{matrix} 3.2420 & 1.7457 & 0 \\ 1.7457 & 3.2420 & 0 \\ 0 & 0 & 0.7481 \end{matrix} \right\} \ldots$$

$$* \left\{ \begin{matrix} -0.1002 & 0 & 0.2841 & 0 & -0.1839 & 0 \\ 0 & -0.4953 & 0 & 0.2167 & 0 & 0.2786 \\ -0.4953 & -0.1002 & 0.2167 & 0.2841 & 0.2786 & -0.1839 \end{matrix} \right\}$$

which is solved to result:

$$\boldsymbol{K} = 10^{11} * \left\{ \begin{matrix} 0.5932 & 0.3397 & -0.4737 & -0.3931 & -0.1195 & 0.0534 \\ 0.3397 & 2.2044 & -0.7190 & -1.0139 & 0.3793 & -1.1905 \\ -0.4737 & -0.7190 & 0.8148 & 0.4215 & -0.3411 & 0.2975 \\ -0.3931 & -1.0139 & 0.4215 & 0.5838 & -0.0284 & 0.4301 \\ -0.1195 & 0.3793 & -0.3411 & -0.0284 & 0.4606 & -0.3509 \\ 0.0534 & -1.1905 & 0.2975 & 0.4301 & -0.3509 & 0.7604 \end{matrix} \right\}.$$

QUESTION 3

Consider the element in Figure 2 alone. Imagine the situation where:

• Nodes 2 & 3 are restrained against movement in both the x and y directions. Node 1 remains unrestrained.

• Forces $ff1xx$ and $ff1yy$ are applied at Node 1 in the x and y directions. The values of these forces per unit thickness of material are given by the MATLAB script.

What is the magnitude of the reaction force at Node 2? Show your working

The force and displacement of each node in a constant strain triangle are related by the equation:

$$\boldsymbol{F} = \boldsymbol{k\,u}$$

where forces f1x and f1y are applied forces at node one x and y respectively, f2x f2y f2x and f3y are reaction forces at nodes 2 and 3 respectively. u_{1x} and u_{1y} are the unknown displacements of node 1.

$$\begin{bmatrix} 102.1 * 10^6 \\ 191.3 * 10^6 \\ F_{2x} \\ F_{2y} \\ F_{3x} \\ F_{3y} \end{bmatrix} = \begin{bmatrix} 0.5932 & 0.3397 & -0.4737 & -0.3931 & -0.1195 & 0.0534 \\ 0.3397 & 2.2044 & -0.7190 & -1.0139 & 0.3793 & -1.1905 \\ -0.4737 & -0.7190 & 0.8148 & 0.4215 & -0.3411 & 0.2975 \\ -0.3931 & -1.0139 & 0.4215 & 0.5838 & -0.0284 & 0.4301 \\ -0.1195 & 0.3793 & -0.3411 & -0.0284 & 0.4606 & -0.3509 \\ 0.0534 & -1.1905 & 0.2975 & 0.4301 & -0.3509 & 0.7604 \end{bmatrix} \begin{bmatrix} u_{1x} \\ u_{1y} \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix}.$$

Initially u_{1x} and u_{1y} are solved by multiplying matrices **k** and **u** together to form the pair of simultaneous equations

$$102.1 * 10^6 = 59324 * 10^6 * u_{1x} + 33967 * 10^6 * u_{1y}$$

and

$$191.3 * 10^6 = 33967 * 10^6 * u_{1x} + 22044 * 10^7 * u_{1y}.$$

These simultaneous equations solve for $u_{1x} = 0.0013m$ and $u_{1y} = 0.0007m$.

The resultant forces acting on node 2 are the forces F_{2x}, calculated with

$$F_{2x} = -47373 * 10^6 * u_{1x} - 71899 * 10^6 * u_{1y}$$

resulting in $F_{2x} = -111.12MPa$ and the force F_{2y}, calculated with

$$F_{2y} = -39306 * 10^6 * u_{1x} - 10139 * 10^7 * u_{1y}$$

resulting in F_{2y} = -119.78MPa.

QUESTION 4

Consider the domain shown in Figure 1 again. If you knew:

• The distribution of mass in the domain.

• Its boundary conditions.

• That any material damping was negligible.

describe how you could find its natural frequencies of vibration. You do not need to determine these frequencies.

In a dynamic model, $\boldsymbol{f_a}$ and **u** are functions of time, since damping is negligible, global damping matrix **C** can be removed to form expression:

$$f(t) = \boldsymbol{M} * \ddot{u}(t) + \boldsymbol{k} * u(t)$$

where **M** is the global mass matrix and **k** is the global stiffness matrix.

The global stiffness matrix is defined by

$$\boldsymbol{K} = \iiint_V \boldsymbol{B^T}(\boldsymbol{r})\boldsymbol{D}\boldsymbol{B}(\boldsymbol{r})dV(r)$$

where V is the volume of element, **D** is the material stiffness matrix and **B(r)** is defined as

$$\boldsymbol{B}(\boldsymbol{r}) = \boldsymbol{\nabla N}(\boldsymbol{r})$$

where **N(r)** contains shape functions which give displacement **r** in terms of nodal displacements **u**.

The global mass matrix, **M**, is formed using the expression:

$$\boldsymbol{M} = \rho \iiint_V \boldsymbol{N^T}(\boldsymbol{r})\boldsymbol{N}(\boldsymbol{r})dV(r).$$

Assuming that the $f(t)$ is harmonic at frequency ω, the force vector can be written as

$$f(t) = \tilde{f}e^{-i\omega t}$$

where

$$\tilde{f} = \begin{Bmatrix} \tilde{f}_1 \\ \tilde{f}_2 \\ \vdots \end{Bmatrix}$$

where f_1, f_2 etc. are complex numbers describing the amplitude and phase of the force applied at each node.

The displacement vector can be written as

$$u(t) = \tilde{u}e^{-i\omega t}$$

with nodal velocities

$$\dot{u}(t) = -i\omega\tilde{u}e^{-i\omega t}$$

and nodal accelerations

$$\ddot{u}(t) = \omega^2\tilde{u}e^{-i\omega t}$$

where

$$\tilde{u} = \begin{Bmatrix} \tilde{u}_1 \\ \tilde{u}_2 \\ \vdots \end{Bmatrix}$$

where $\tilde{u}_1$, $\tilde{u}_2$ etc. are complex numbers describing the amplitude and phase of the displacement at each node.

Tidying up and inserting expressions into the force displacement equation:

$$\tilde{f} = (\boldsymbol{K} - \omega^2\boldsymbol{M})\,\tilde{u}.$$

At resonance, the system undergoes harmonic oscillation at unknown frequency ω with no external forces, f = 0, therefore

$$(\boldsymbol{K} - \omega^2\boldsymbol{M})\,\tilde{u} = 0.$$

In this eigenvalue problem, the eigenvalues ω_j^2 are the resonant frequencies that satisfy

$$\left|\boldsymbol{K} - \omega_j^2\boldsymbol{M}\right| = 0.$$

Applied Solid Mechanics Summative Part C

Reuben Marland 1935805

QUESTION 1

a.) What is the smallest number of strain measurements required to determine σ_x and σ_y?

Since there are only normal forces applied, there is no shear stress within the element, if we choose the strain gauge directions to be in the same direction as both σ_x and σ_y, we will only need to use 2 strain gauges.

b.) Calculate applied stresses σ_x and σ_y from strain measurements.

A strain gauge rosette is formed with the first strain gauge in line with the σ_x direction, resulting a strain of ε_a and one in line with the σ_y direction, ε_b.

It is assumed that the plate is in plane stress as there are no stresses acting in the z direction since the plate is thin. Aluminium can also be assumed to be an isotropic material.

The strain in each direction is described by equations, note the expression for ε_{xy} would be 0 in this case.

$$\varepsilon_x = \varepsilon_a$$

$$\varepsilon_y = \varepsilon_b$$

$$\varepsilon_{xy} = \varepsilon_c - \frac{\varepsilon_a + \varepsilon_b}{2}$$

where $\varepsilon_a = -0.0013$ and ε_b = 0.0021 are calculated values from the strain gauges, which are used to solve for ε_{11} = -0.0013 and $\varepsilon_{22} = 0.0021$.

The strain tensor is thus formed as

$$\varepsilon = \begin{bmatrix} -0.0013 \\ 0.0021 \\ 0 \end{bmatrix}.$$

Since the material is considered isotropic and plate in plane stress, hooks law can be written in a reduced matrix form:

$$\begin{bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{bmatrix} = \frac{E}{1 - v^2} \begin{bmatrix} 1 & v & 0 \\ v & 1 & 0 \\ 0 & 0 & \frac{1 - v}{2} \end{bmatrix} \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{bmatrix}$$

where E is the Youngs Modulus, 400MPa and v is the poison ratio, 0.3.

Applying the matrix transformation, the resultant stress matrix is:

$$\sigma = \begin{bmatrix} -50.00 \\ 130.00 \\ 0 \end{bmatrix} MPa$$

where σ_x solves as a compressive stress of magnitude 50MPa and σ_y solves as a tensile stress of magnitude 130MPa.

c.) Provide analysis of the possible component's failure mechanisms as a function of stress σ_p and crack orientation

To form the total stress tensor for the system including various values of σ_p, we must find the current stress tensor for the system where the normal stresses are both perpendicular and parallel to the direction of σ_p.

A strain gauge rosette is formed at angles $30°, 75°$ and $120°$, with respective strains resulting as

Following the previously stated strain gauge equations, the strain matrix is formed:

$$\varepsilon = \begin{bmatrix} -0.0004 \\ 0.0012 \\ 0.0014 \end{bmatrix}$$

using hooks law, the stress matrix is calculated as:

$$\sigma = \begin{bmatrix} -5.00 \\ 85.00 \\ 77.94 \end{bmatrix} MPa.$$

with equivalent stress tensor (in this case a 2x2 matrix since in plane stress $\sigma_{xz}, \sigma_{yz}\ and\ \sigma_{zz}$ are all 0):

$$\sigma = \begin{bmatrix} -5.00 & 77.94 \\ 77.94 & 85.00 \end{bmatrix} 10^6.$$

σ_p is parallel to the normal stress σ_x in this situation, summing onto that individual component of the stress tensor, hence the overall stress tensor for the plate becomes

$$\sigma = \begin{bmatrix} -5.00 + \sigma_p & 77.94 \\ 77.94 & 85.00 \end{bmatrix} 10^6$$

where σ_p is a stress that has magnitude $|\sigma_p| \leq 400MPa$.

The principal stresses in plane stress are defined as

$$\sigma_1, \sigma_2 = \frac{\sigma_{11} + \sigma_{22}}{2} \pm \sqrt{\left(\frac{\sigma_{11} - \sigma_{22}}{2}\right)^2 + \sigma_{12}^2}.$$

Setting up a yield surface for σ_1 and σ_2 using, respectively, the Von Mise's and Tresca maximum stress $\bar{\sigma}$

$$\bar{\sigma} = \sqrt{\frac{1}{2}[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2)]}$$

which under plane stress conditions simplifies to

$$\bar{\sigma} = \sqrt{\sigma_1^2 + \sigma_2^2 + \sigma_1\sigma_2}$$

and for Tresca:

$$\bar{\sigma} = \max\left[|\sigma_1 - \sigma_2|, |\sigma_2 - \sigma_3|, |\sigma_1 - \sigma_3|\right]$$

which under plane stress conditions simplifies to

$$\bar{\sigma} = \max[|\sigma_1 - \sigma_2|, |\sigma_2|, |\sigma_1|].$$

The simplified equations are used to build the yield surfaces, given a σ_y of 400MPa.

An array of σ_1 and σ_2 values corresponding to each value of σ_p can be constructed and plot upon the yield surfaces as shown in Figure 1.

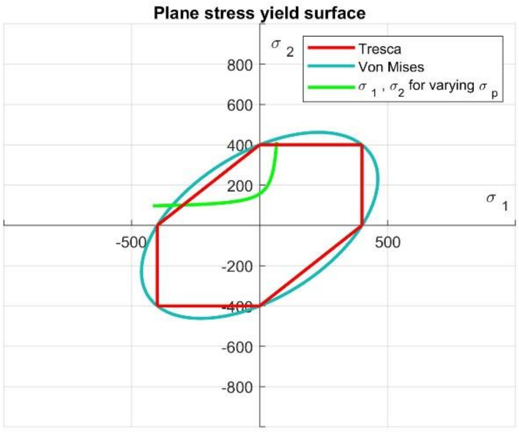

Figure 1

There are σ_1 and σ_2 values outside of the yield surface, the von mises criterion corresponds to σ_p values of $\sigma_p \leq\approx 322MPa$ and the Tresca criterion corresponds to $\approx -287MPa \leq \sigma_p \leq 387MPa$ that cause failure of the plate under a Yield Criterion.

A central crack can form in any arbitrary orientation, in this circumstance, crack orientation, θ, is defined as the angle that the normal stress on the crack makes as shown in Figure 2. Following the same initial orientation given in the figure handed with the question.

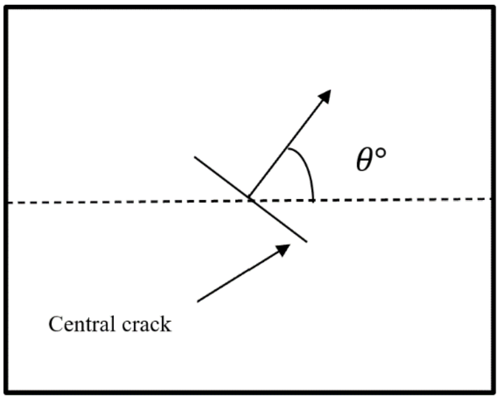

Figure 2

For each σ_p the normal force acting on each orientation of the crack can be solved by rotating the overall stress tensor for the corresponding σ_p using the rotational transformation:

$$\sigma' = r\sigma r^T.$$

Where r is a 2x2 rotation matrix:

$$r = \begin{bmatrix} \cos(\alpha) & \sin(\alpha) \\ -\sin(\alpha) & \cos(\alpha) \end{bmatrix}.$$

A large array of values is created expressing the normal stress acting on the crack σ_n as a function of θ and σ_p. The surface plot, Figure 3, shows how σ_n relates to θ and σ_p.

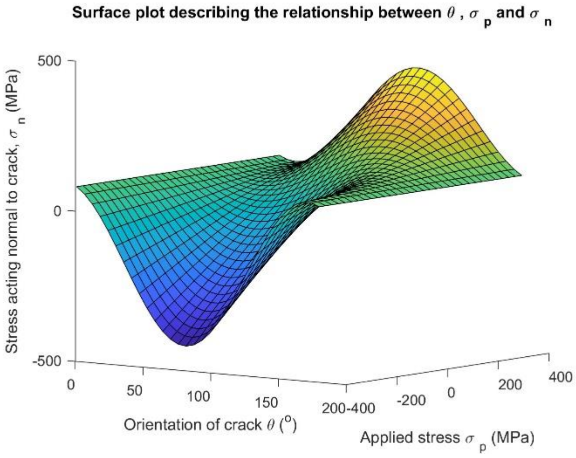

Figure 3

Since the plate is large and the crack is central, we assume the Geometrical Correction Factor, Y, to be 1.

Where the fracture toughness, K_{IC}, is 20MPa$\sqrt{m}$ and α, the NDT detection limit, 5mm, with crack radii detection limit of 2.5mm, the design stress can be calculated using:

$$\sigma = \frac{K_{IC}}{\sqrt{\frac{\pi\alpha}{2}}} = 225.67MPa.$$

Any crack with a radius greater than 2.5mm will in theory be detected from routine inspection and the material will be repaired.

However, for cracks with radii less than this, there is a possibility of failure.

The distribution of failure stress, σ_f and crack radius, r are related by the equation:

$$\sigma_f = \frac{K_{IC}}{\sqrt{2\pi r}}$$

which can be shown in Figure 4

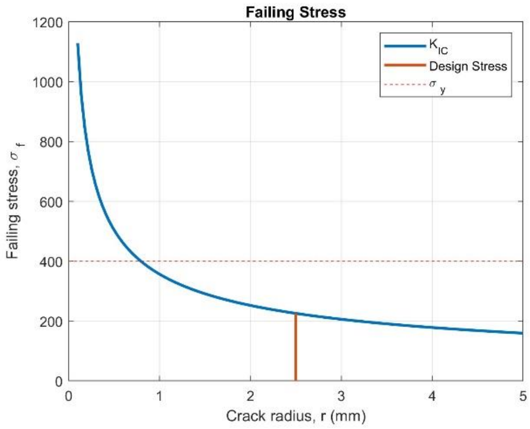

Figure 4

The range of σ_n values that exceed the design stress but do not exceed the yield stress are: $225.68MPa \leq \sigma_f < 400$ these values fracture under brittle failure before yield. Sectioning Figure 3, we can see the distribution of σ_p and θ that produce a σ_n that falls within this range, as shown in figure 5. The area where there are values is where the σ_n adheres to the range of σ_f.

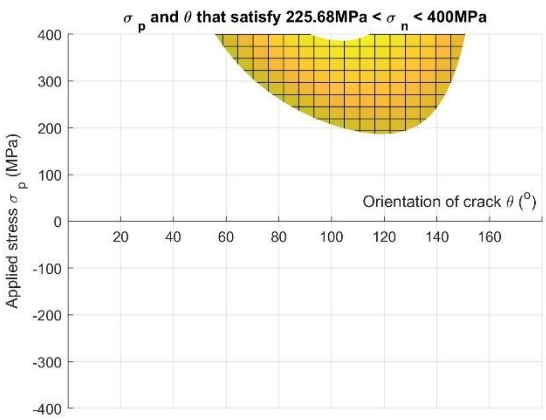

Figure 5

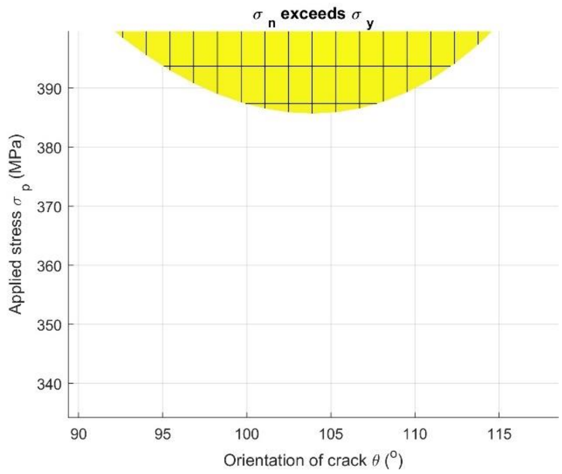

Figure 6

For crack sizes that have a failure stress more than yield stress, as shown above the dotted red line in figure 4, will undergo local crack yielding. The distribution of σ_p and θ that produce a $\sigma_n \geq 400MPa$ is shown in figure 6

The critical plastic zone size of a crack under plane stress can be described by equation:

$$r_p(\theta) = \frac{1}{2\pi}\left(\frac{K_{IC}}{\sigma_y}\right)^2 \cos^2\left(\frac{\theta}{2}\right)\left[1 + \sin\left(\frac{\theta}{2}\right)\right]^2.$$

This distribution of the plastic zone at the crack tip can be shown in figure 7

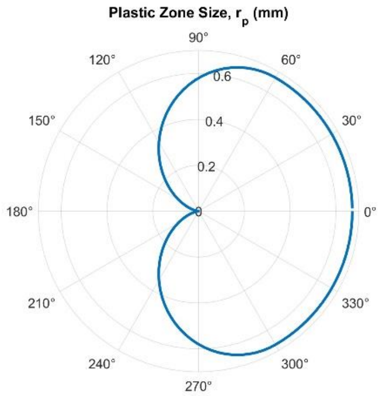

Figure 7

The plastic zone radius directly ahead of the crack tip, r_p, creates a new effective stress intensity factor for the crack, K_{eff}, as calculated by:

$$K_{eff} = \sigma_y\sqrt{\pi(r_y + r_p(\theta))} = 31.08MPa\sqrt{m}$$

where r_y is calculated from the failure stress = σ_y

$$r_y = \frac{1}{\pi}\left(\frac{K_{ic}}{\sigma_y}\right)^2 = 7.95 * 10^{-4}.$$

The plot of stress against crack radius with K_{eff} is displayed in figure 8

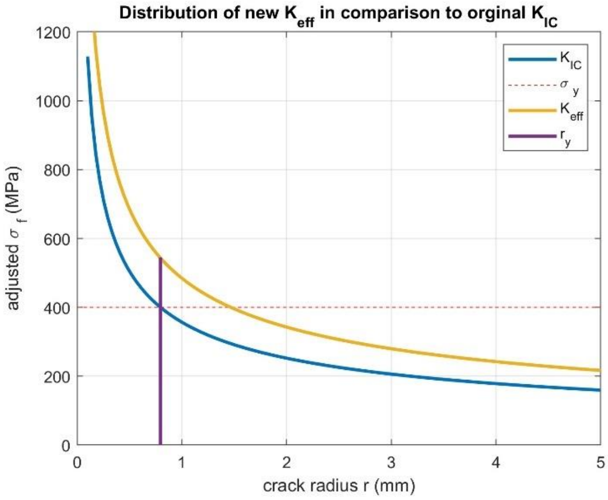

Figure 8

The maximum possible stress acting on a crack in any orientation and with any value of P, σ_{max}, is $\approx 413.46MPa$, this occurs at a crack angle of $\approx\ 101°$.

Any crack with radius less than r_y has failure stress higher than this and hence do not fail yet have undergone local plastic deformation around the crack tip

The safety factors for yield and brittle failure criteria are respectively calculated as:

$$S_f = \frac{\sigma_y}{\bar{\sigma}} = \ 1.77$$

and

$$S_f = \frac{\sigma_{max}}{\bar{\sigma}} = 1.83$$

where $\bar{\sigma}$ is design stress.

QUESTION 2

a.) How does the pipe's probability of failure depend on the interval between pipe inspections?

The crack is orientated normal to the axial stress. We can calculate this stress with a thin-walled tube approximation:

$$\sigma_{axial} = \frac{pr}{2t}$$

where p is the internal pressure ranging from 0 to 40MPa, r is the outer radius of the cylinder, 0.5m and t is the wall thickness 0.01m.

If the fracture crack length, α_f, is greater than t, then the crack can be approximated as a central crack in a plate where Y can be approximated as 1.

$$\alpha_f = \frac{1}{\pi}\left(\frac{K_{IC}}{\sigma_{axial}}\right)^2$$

where K_{IC} is the fracture toughness, 90MPa$\sqrt{m}$

The maximum pressure, without valve fault, σ_{axial} = 200MPa, calculates an α_f of 0.0644m. This value is greater than the wall thickness and failure by leakage occurs.

The maximum pressure, with valve fault, σ_{axial} = 400MPa, calculates an α_f of 0.0161m. This value is greater than the wall thickness and failure by leakage occurs.

Since maximum pressure constitutes a failure by leakage, the overall α_f will be the wall thickness and the failure mechanism will be leak before break.

The growth of a crack from arbitrary size α_i to α_f is described by Paris' law

$$\frac{dN}{d\alpha} = \frac{1}{C(\Delta K)^m}$$

where

$$\Delta K = Y(a)\Delta\sigma\sqrt{\pi\alpha}$$

where α is various cracks ranging from 0.1mm to 10mm, $\Delta\sigma$ is the difference between σ_{max} = 200MPa and σ_{min} = 0MPa and

$$Y(a) = 0.728 + 0.393\left(\frac{\alpha}{t}\right)^2 - 0.029\left(\frac{\alpha}{t}\right)^4.$$

Combining equations, we get an integral for the number of cycles, N it takes to increase a crack radius from α_i to α_f:

$$N = \int_{\alpha_i}^{\alpha_f} \frac{d\alpha}{C * \left(\left(0.728 + 0.393\left(\frac{\alpha}{t}\right)^2 - 0.029\left(\frac{\alpha}{t}\right)^4\right)\Delta\sigma\sqrt{\pi\alpha}\right)^m}$$

where C = $10^{-12}\frac{m/cycle}{(MPa\sqrt{m})^m}$ and m = 4.

It is assumed that after every inspection, if any cracks are found, they are fixed without any deformity affecting the pressure-vessel in the future.

If you take a number of cycles N, the array of cracks that have reached failing radius are the cracks with radius larger than the crack that took precisely N_f cycles to reach failing radius. The probability that all cracks with larger radii than the radius corresponding to precise failure under N_f can be detected using the probability of detection function p_{pod}.

This distribution can be shown in Figure 9

Probability of failure corresponds to the radii of cracks that would have reached α_r and have gone undetected. This relationship can be shown in Figure 10.

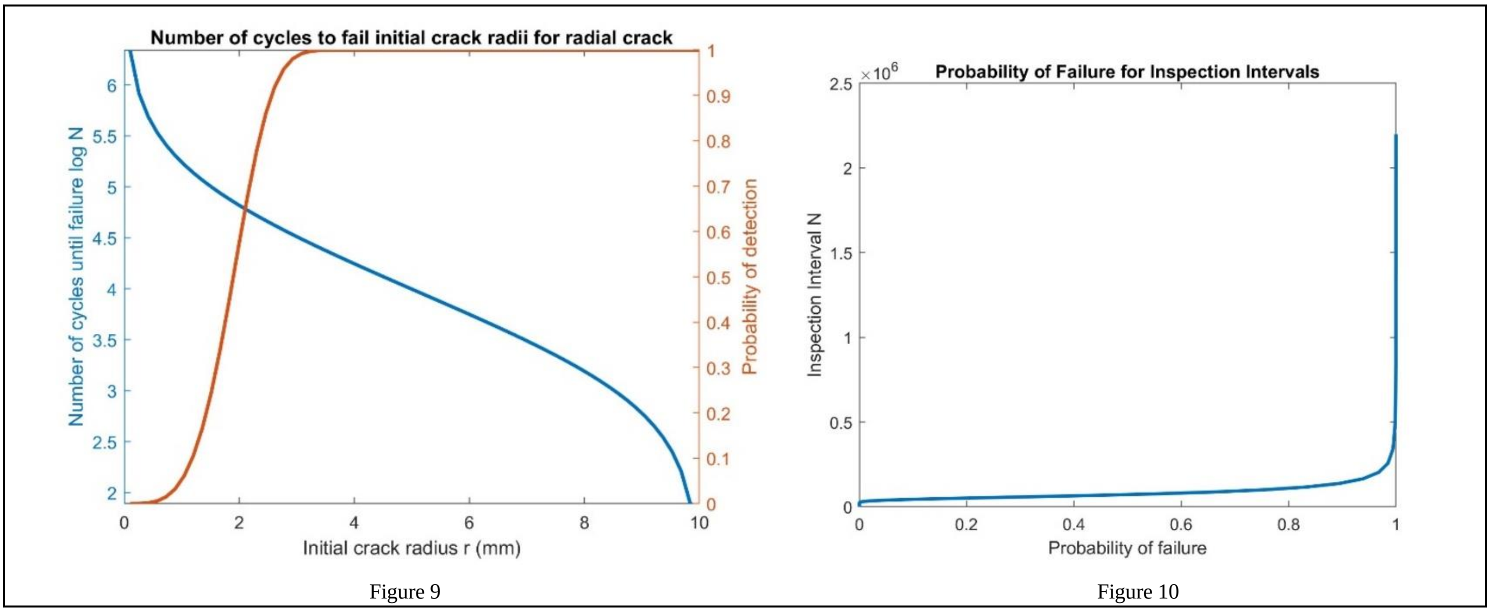

Figure 9 Figure 10

b.) Find the inspection interval, corresponding to the probability of failure equal to 0.01.

Using a scaled in version of figure 2, figure 3, a probability of failure of 0.01 corresponds to where you can predict 99% of all cracks that would have reached failure stress have been detected.

This corresponds to a Inspection Interval of ≈ 31544.

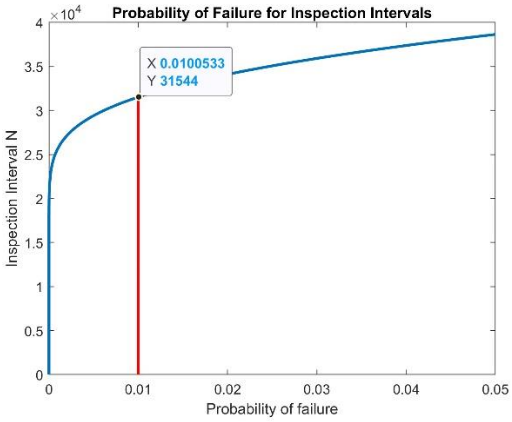
Probability of Failure for Inspection Intervals
×10^4
X 0.0100533
Y 31544
Inspection Interval N
Probability of failure
0
0.5
1
1.5
2
2.5
3
3.5
4
0.01
0.02
0.03
0.04
0.05

Figure 11