International Journal of Advanced Information Science and Technology (IJAIST) ISSN: 2319:2682
Vol.5, No.2, February 2016 DOI:10.15693/ijaist/2016.v5i2.83-87

Numerical simulations by finite element analisysis method of checking structure elements propulsion system components

Eng. Flaviu KMEN PhD student
Chief Engineer NS Mircea
Dunarea de Jos University, Galati, România

Prof.eng. Silviu MACUTA PhD
Professor/Mechanical EngineeringDepartment
Dunarea de Jos University, Galati, România

***Abstract*— In recent years , structural analysis , has emerged as a means of checking the current buildings eligible low-cost technique , including naval propulsion plants . For this reason, subject , in this article , shaft , part of the axial line , the torsion force , uniformly distributed over the entire surface of the flange connection point . For the analysis is considered recessed aft propeller propulsion blocking valid hypothesis using a CAD environment . SolidWorks CAD environment used is 2010, and the method of calculation method was used " finite element " (MEF) , stress analysis theory is determined by the resistance of the composite applications Huber - Hencky - Mises . The results are compared with the limit values of stresses and strain accepted .**

I. INTRODUCTION

The elements of design axial line require detailed knowledge of the design characteristics of the ship, transport capacity , climatic conditions , work schemes in which it operates.

For calculations were used elements like MATHCAD software - used in the thermal calculation to verify the shaft was used a CAD environment , SolidWorks 2010 .

SolidWorks software enables structural analysis of the elements constructed in this environment , highlighting through finite element method , the criterion von Mises equivalent stress and tangential movements of the material.

Modeling CAD environment , as discussed above , has become in recent years a practice mandatory engineering to check sizing carried out mathematically , this solution allows the display of resulting feedback very close to those of actual conditions , avoiding a heavily errors Major calculation.

Of great importance in achieving article were technical documentation on board , which were the basis for drawing up the scheme of the propulsion system , with all its elements Components .

II. RELATED WORK

A. Finite Element Method

FEM is a general equalization method based on virtual work equalizer conducted by external forces to that of inner forces .

It requires that "tool" standard (numeric) for engineering calculations .

Engineering practice assume issues tension and strain in more complex bodies , for which no analytical solutions whose solution is possible through digital electronic computers.

Function defines only for small areas . This allows them to describe behavior within the domain to be able to choose simple shape functions , inferior

Finite element method is based on the following assumptions:

1) The structure is discretized into finite elements connected to each other only at the knots. Items can take different forms, movements of nodes are selected as unknown.

2) connecting the elements and forces of external forces on the components selected only in knots.

3) strain, strains and stresses in any point of any element is expressed according to the movement of nodes.

4) forces the nodes, from the nodes movements of external forces on the elements and components must be in balance. The equations of balance of forces in all nodes of the mesh, leading to a resulting algebraic system of unknowns, ie nodes movements. Accuracy of results depends on how the conditions of continuity sides finite element, which depends on the fineness of the mesh and unknowns chosen for each item. The finite elements which fully comply with the terms of continuity sides are called conforming elements.

Fundamental equations and formulas of finite element method

Fundamentals of finite element will be set for a triangular finite element , the calculation methodology is the same for any type of item .

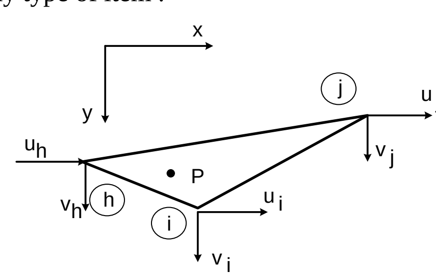


Figure. 1. Study triangular element

B. Expressions strain, strains and stresses on the specific nodes movements

For a node, as well as for a possible displacement body there are six , also known as degrees of freedom (DOF) , three translations, denoted u , v, w and three rotations , denoted rx , ry , rz . For particular cases the number of these movements is reduced.

Movements element nodes clustered column vector

$$\widetilde{\Delta^{(e)}} = \left[u_h v_h u_i v_i u_j v_j\right]^T \quad \textbf{(1)}$$

T - marks transposition surgery

Movements in the current point

a) Moving a certain point "p" determines the vector

$$\widetilde{\Delta p} = [u \cdot v]^T \quad \textbf{(2)}$$

And the movements in a current point P on the travel element nodes

$$\widetilde{\Delta_P} = \tilde{N}\ \widetilde{\Delta^{(e)}} \quad \textbf{(3)}$$

Or,

$$\widetilde{\Delta_P} = \sum_h N_h \widetilde{\Delta_h} = \sum_h \begin{bmatrix} N_h & 0 \\ 0 & N_h \end{bmatrix} \begin{bmatrix} u_h \\ v_h \end{bmatrix} \quad \textbf{(4)}$$

b) a point deflection current

The finished item is deemed to be uniform , continuous and isotropic , the relationship between specific deformations and strain are given by:

$$\varepsilon_x = \frac{\partial u}{\partial x} \quad \textbf{(5)}$$

$$\varepsilon_y = \frac{\partial v}{\partial y} \quad \textbf{(6)}$$

$$\gamma_{xy} = \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \quad \textbf{(7)}$$

Assembling deflection vector - column

$$\tilde{\varepsilon} = \left[\varepsilon_x \varepsilon_y \gamma_{xy}\right]^T \quad \textbf{(8)}$$

So we can write as before:

$$\tilde{\varepsilon} = \begin{bmatrix} \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix} \begin{bmatrix} u \\ v \end{bmatrix} =$$

$$\begin{bmatrix} \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix} \begin{bmatrix} N_h & 0 & N_i & 0 & N_j & 0 \\ 0 & N_h & 0 & N_i & 0 & N_j \end{bmatrix} \widetilde{\Delta^{(e)}} \quad \textbf{(9)}$$

Matrix follows :

$$\tilde{B} = \begin{bmatrix} \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix} \cdot \begin{bmatrix} N_h & 0 & N_i & 0 & N_j & 0 \\ 0 & N_h & 0 & N_i & 0 & N_j \end{bmatrix} =$$

$$\frac{1}{2A_e} \begin{bmatrix} \beta_h & 0 & \beta_i & 0 & \beta_j & 0 \\ 0 & \gamma_h & 0 & \gamma_i & 0 & \gamma_j \\ \gamma_h & \beta_h & \gamma_i & \beta_i & \gamma_j & \beta_j \end{bmatrix} \quad \textbf{(10)}$$

Thus, the relationship becomes:

$$\tilde{\varepsilon} = \tilde{B} \cdot \widetilde{\Delta^{(e)}} \quad \textbf{(11)}$$

Therefore using matrix $\tilde{B}$, the vector specific strains at a point current is expressed according to the movement of nodes .

$$\varepsilon_x = \frac{1}{E}\left(\sigma_x - \mu\sigma_y\right) \quad \textbf{(12)}$$

$$\varepsilon_y = \frac{1}{E}\left(\sigma_y - \mu\sigma_x\right) \quad \textbf{(13)}$$

$$\gamma_{xy} = \frac{2(1+\mu)}{E}\tau_{xy} \quad \textbf{(14)}$$

If it is vector tensions

$$\tilde{\sigma} = \left[\tau_x \tau_y \tau_{xy}\right]^T \quad \textbf{(15)}$$

And note the initial deformations uniform vector

$$\widetilde{\varepsilon^0} = [\varepsilon_0 \varepsilon_0 0]^T \quad \textbf{(16)}$$

Prepared relationship matrix

$$\tilde{\sigma} = \tilde{D}\left(\tilde{\varepsilon} - \widetilde{\varepsilon^0}\right) = \tilde{D}\tilde{B}\widetilde{\Delta^{(e)}} - \tilde{D}\widetilde{\varepsilon^0} \quad \textbf{(17)}$$

In matrix

$$\tilde{D} = \frac{E}{1-\mu^2} \begin{bmatrix} 1 & \mu & 0 \\ \mu & 1 & 0 \\ 0 & 0 & \frac{1-\mu}{2} \end{bmatrix} \quad \textbf{(18)}$$

It represents metrics elasticity.

The relationship matrix $\tilde{\sigma} = \tilde{D}\left(\tilde{\varepsilon} - \widetilde{\varepsilon^0}\right) = \tilde{D}\tilde{B}\widetilde{\Delta^{(e)}} - \tilde{D}\widetilde{\varepsilon^0}$ determines the current tensions at a point $P(x, y)$ according to the movement of the element in the element nodes .

C. An element stiffness matrix

H,i,j element with simple connections after x , y (pendula) is a basic form that allows independent variation of parameters , ie moving nodes . Work on those links nodes travel products are grouped into vectors

$$\widetilde{Q_h^\Delta} = \left[Q_h^x Q_h^y\right]^T \quad \textbf{(19)}$$

$$\widetilde{Q_i^\Delta} = \left[Q_i^x Q_i^y\right]^T \quad \textbf{(20)}$$

$$\widetilde{Q_j^\Delta} = \left[Q_j^x Q_j^y\right]^T \quad \textbf{(21)}$$

$$\widetilde{Q^{(e)\Delta}} = \left[Q_h^x Q_h^y Q_i^x Q_i^y Q_j^x Q_j^y\right]^T = \left[\widetilde{Q_h^\Delta}\widetilde{Q_i^\Delta}\widetilde{Q_j^\Delta}\right]^T \quad \textbf{(22)}$$

The relationship between the vector $\widetilde{Q^{(e)\Delta}}$ and vector $\widetilde{\Delta^{(e)}}$ is carried out through a matrix $\widetilde{K^{(e)}}$ so that

$$\widetilde{Q^{(e),\Delta}} = \widetilde{K^{(e)}} \cdot \widetilde{\Delta^{(e)}} \quad \textbf{(23)}$$

Known matrix rigidity of the element (e), which is a square matrix symmetric , if the element in question.

Given the vector components $\widetilde{Q^{(e)}}, \widetilde{\Delta^{(e)}}$ stiffness matrix $\widetilde{K^{(e)}}$ to customize the form :

$$\widetilde{K^{(e)}} = \begin{bmatrix} \widetilde{K_{hh}} & \widetilde{K_{hi}} & \widetilde{K_{hj}} \\ \widetilde{K_{ih}} & \widetilde{K_{ii}} & \widetilde{K_{ij}} \\ \widetilde{K_{jh}} & \widetilde{K_{ji}} & \widetilde{K_{jj}} \end{bmatrix}^{(e)} \quad \textbf{(24)}$$

Given the matrix $\widetilde{K_{hi}}$ with composition :

$$\widetilde{K_{hi}^{(e)}} = \begin{bmatrix} K_{hi}^{xu} & K_{hi}^{xv} \\ K_{hi}^{yu} & K_{hi}^{xv} \end{bmatrix}^{(e)} \quad \textbf{(25)}$$

They K_{hi}^{xu} represents the effort of connection h after the x direction due to movement of $u_i = 1$ in base form , while all other movements are void. Similarly K_{hi}^{xv} efforts in connection represents the direction *x* due to displacement of $v_i = 1$.

D. Establishing the system of equations of the finite element method

Consider a finite network elements and node h thereof, are connected more items.

Isolate node vector *h* and placed both outside forces applied directly to the node $(\mathcal{F}_h)$ and efforts from elements connected to that node . Since the balance of forces to the node *h*.

$$\widetilde{\mathcal{F}_h} - \sum_{e_h} \widetilde{Q_h^{(e_h)}} = 0$$

Knowing that

$$\widetilde{Q_h^{(e_h)}} = \sum_i \widetilde{K_{hi}} \widetilde{\Delta_i^{(e_h)}} - \widetilde{F_h^{e_h q}} - \widetilde{F_h^{e_h \varepsilon^0}} \quad \textbf{(27)}$$

And,

$$\widetilde{Q_h^{\Delta}} = \sum_i \widetilde{K_{hi}} \widetilde{\Delta_i} \quad \textbf{(28)}$$

This gives the matrix equation :

$$\sum_{e_h} \sum_i \widetilde{K_{hi}^{e_h}}\, \widetilde{\Delta_i^{(e_h)}} = \widetilde{\mathcal{F}_h} + \widetilde{F_h^{e_h q}} + \widetilde{F_h^{e_h \varepsilon^0}} \quad \textbf{(29)}$$

noting

$$\widetilde{F_h} = \widetilde{\mathcal{F}_h} + \widetilde{F_h^{e_h q}} + \widetilde{F_h^{e_h \varepsilon^0}} \quad \textbf{(30)}$$

And writing equilibrium equation for all nodes of the system of equations is obtained :

$$\sum_{e_h} \sum_i \widetilde{K_{hi}^{e_h}}\, \widetilde{\Delta_i^{(e_h)}} = \widetilde{F_h}; \;\; h = 1,2,\ldots,n \quad \textbf{(31)}$$

Where *n* is the number of nodes of the system of equations called *Finite Element Method* . This is an algebraic system in which the unknowns are the strain of nodes , which are assembled in the vector .

$$\tilde{\Delta} = \left[\widetilde{\Delta_1}, \widetilde{\Delta_2}, \ldots, \widetilde{\Delta_h}, \ldots, \widetilde{\Delta_n}\right]^T \quad \textbf{(32)}$$

Similarly , the forces $\widetilde{F_h}$ of nodes up vector :

$$\tilde{F} = \left[\widetilde{F_1}, \widetilde{F_2}, \ldots, \widetilde{F_h}, \ldots, \widetilde{F_n}\right]^T \quad \textbf{(33)}$$

With this notation system of equations is written as :

$$\tilde{R}\tilde{\Delta} = \tilde{F} \quad \textbf{(34)}$$

Wherein $\tilde{R}$, , has the constitution

$$\tilde{R} = \begin{bmatrix} \widetilde{R_{11}} & \widetilde{R_{12}} & \ldots & \widetilde{R_{1i}} & \ldots & \widetilde{R_{1n}} \\ \vdots & \ldots & \ldots & \ldots & \ldots & \ldots \\ \widetilde{R_{h1}} & \widetilde{R_{h2}} & \ldots & \widetilde{R_{hi}} & \ldots & \widetilde{R_{hn}} \\ \vdots & \ldots & \ldots & \ldots & \ldots & \ldots \\ \widetilde{R_{n1}} & \widetilde{R_{n2}} & \ldots & \widetilde{R_{ni}} & \ldots & \widetilde{R_{nn}} \end{bmatrix} \quad \textbf{(35)}$$

E. Methods for solving problems by FEM

The main stages of the application of finite element calculation are:

1. The finite element discretized structure in which only the nodes to be interconnected, whose movements are chosen as unknowns.
2. Choose interpolation functions and is calculated for each element matrix B
3. Determine the matrices of rigidity and stiffness matrix elements of the mesh.
4. Determine the nodal forces vector, write the system of equations, and determine the unknowns.
5. Calculate movements of the points of interest on any item and then tensions.
6. Given the tensions and movements in the required check points of strength and stiffness conditions

Accuracy of results depends on the fineness of the mesh and how the conditions of continuity on the sides of the elements. Their fulfillment is influenced by the interpolation functions in areas with high gradient variation must be taken to a higher degree, which leads to higher order elements.

III. Intermediate Shaft. Predimensioning shaft

For analytical expression of the rotating machine torque is

$$M_t = \frac{30}{\pi} \cdot \frac{P}{n} \quad \textbf{(36)}$$

It adopts

$$P = 8825.6[kW], n = 122\ [rot/min].$$

$$M_t = 6.908 \cdot 10^2\ [kN \cdot m]$$
$$M_t = 6.908 \cdot 10^8\ [N \cdot mm]$$

Choose as a building material shaft OLC35, normal tension the tension limit flow $\sigma_{flow} = 320\ [MPa]$.

Predimensioning shaft will be made solely by considerations of torque . It shall be admissible tangential $\tau_a = 100[MPa]$using the above mentioned values are determined W_p polar section modulus .

$$W_p = \frac{M_t}{\tau_a} \quad \textbf{(37)}$$
$$W_p = 6.908 \cdot 10^6\ [mm^3]$$

It adopts the embodiment of a hollow shaft report

$$k = \frac{d}{D} = 0.41$$

The expression literal torsional modulus is

$$W_p = \frac{\pi \cdot [D^4 - d^4]}{16D}, \quad \textbf{(38)}$$

which results in expression of the diameter D

$$D = \sqrt[3]{\frac{16 \cdot W_p}{\pi \cdot (1 - k^4)}} \quad \textbf{(39)}$$
$$D = 330.819\ [mm]$$

It adopts$D = 400\ [mm]$

and result: $d = k \cdot D = 0.41 \cdot 400$

and the inner diameter , according to the calculations we obtain:

$$d = 164\ [mm]$$

It adopts $d = 160\ [mm]$

The most important characteristics of shaft , outer diameter and were determined mathematically in the previous section .

Intermediate shaft was built to withstand the maximum possible torque , at the embedding the propeller flange ; This situation corresponds to the specific case of blocking the propeller from external causes , and maximum torque is developed from the engine.

For simplicity, intermediate shaft is built without radius. Shaft construction complies with quotas set

International Journal of Advanced Information Science and Technology (IJAIST) ISSN: 2319:2682
Vol.5, No.2, February 2016 DOI:10.15693/ijaist/2016.v5i2.83-87

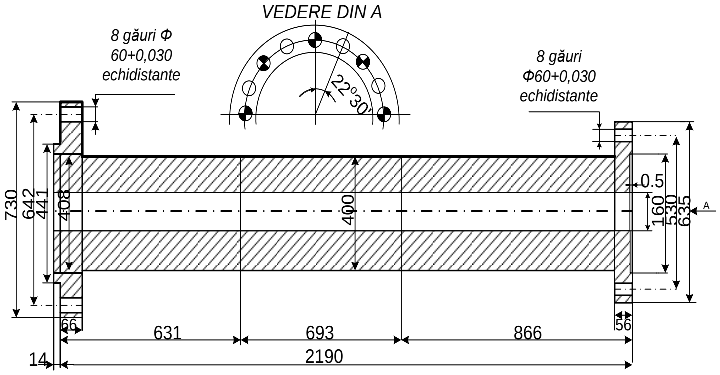


Figure 2 Intermediate Shaft

Accessories part of the propulsion plant structure intermediate shaft was subjected to stress to stress, travel and effort.

The piece which forms part of the structure of the propulsion plant , shaft was subjected to stress to stress, travel and effort.

As part of this verification , he was tried shaft radius of curvature as the coupling flange connection , and without it , observing substantial differences .

For this to be conclusive verification was considered one of the ends recessed , which is comparable to the practical case lock the propeller , so maximum torque developed by the propulsion system .

The moment the applicant used in the study is :

$$M_t = 6.908 \cdot 10^8 \; [N \cdot mm]$$

And tension flow limits is :

$$\sigma_{flow} = 320 \; [MPa]$$

In a first stage of verification shaft model built in the previous section it was built in Solidworks , strictly respecting its constructive dimensions .

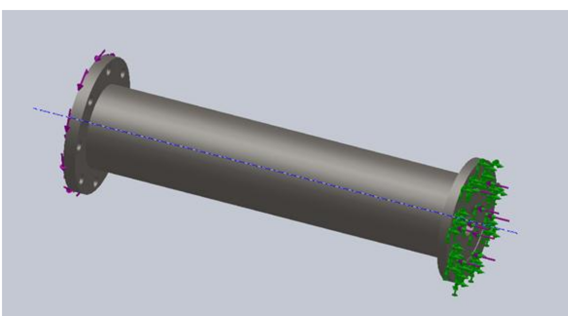


Figure 3 Intermediate shaft

The finite element method, the intermediate shaft is meshed in elements linked only to nodes.

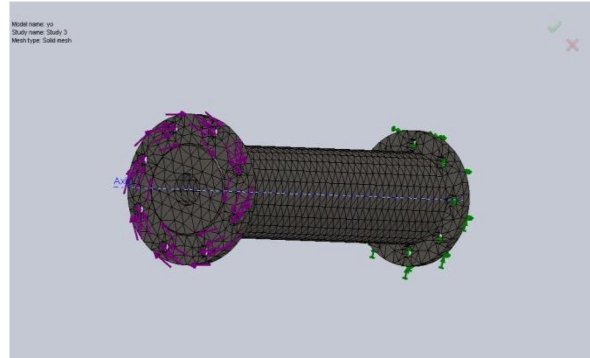


Figure 4. Intermediate shaft meshed finite element

IV. Stress Study

For stress analysis, intermediate shaft is subjected to torque from the engine, considering the recessed propeller shaft flange .

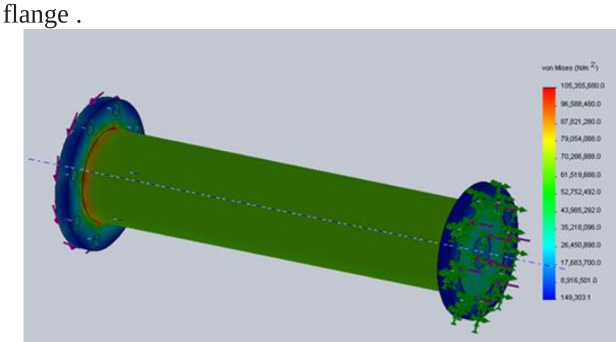


Figure 5. Stress Study

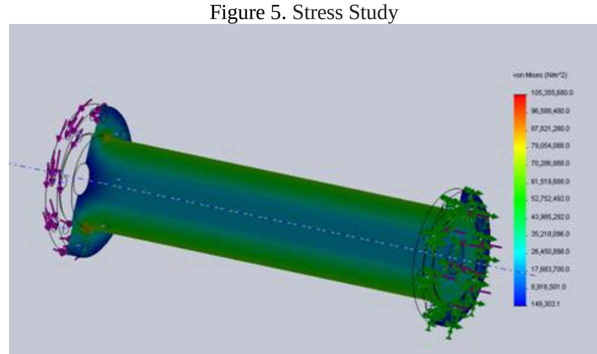


Figure 6. Sectio stress study

Following this analysis notes that the equivalent stress $\sigma = 106 \; [MPa]$ von Mises criterion, this is an acceptable solution because $\sigma_{flow} = 320 \; [MPa]$, especially for the steel , OLC 35 .

Requiring most pronounced is in the red , between the cylinder flange and shaft .

V. Study Displacement

Making this study strain absolutely necessary to know the deformations that occur in the material it is made of shaft .

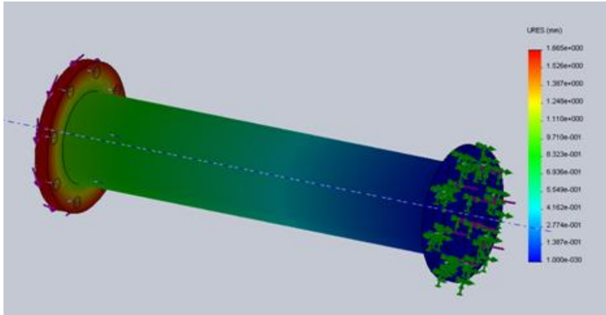


Figure 7. Strain Study

According to this study Figure 7, there is a resultant from translational 1.665 [mm] on the exterior of the flange for connection with the engine block.

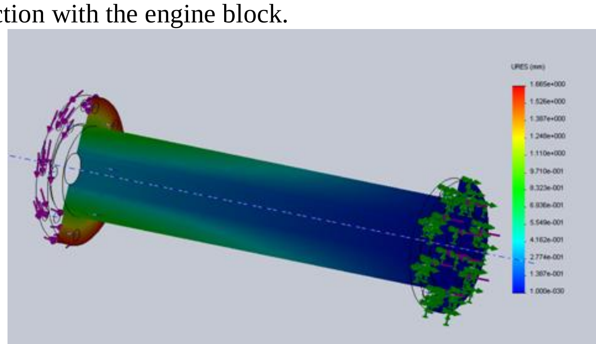


Figure 8. Section strain study

VI. CONCLUSION

Encapsulation of the lower surface is identical situation or lock the crankshaft axial line

Shaft component of the axial line, which transmits rotation of the drive shaft propeller thermal, imply the propellant.

Sizing shaft was done analytically, using the relationships of strength of materials classic knowing the torque $M_t = \frac{30}{\pi} \cdot \frac{P}{n}$, depending on engine power and speed it was adopted tangential admissible $\tau_a = 100[MPa]$ to determine polar section modulus $W_p = \frac{M_t}{\tau_a} = \frac{\pi \cdot [D^4 - d^4]}{16D}$thus obtained shaft diameter is $D = 330.819\ [mm]$. In operation shall be adopted value of 400 [mm] .

To check the values previously determined, shaft was modeled and analyzed with FEM in SolidWorks 2010. It is the subject of drive torque developed thermal conditions under which an intermediate shaft flange is recessed (does not have any degree of freedom) , it simulating extreme load - lock the propeller.

After verification of stres is observed in the connection between the flange and shaft, so to decrease the value of these stresses to build a connection between the two.

Checking values are imposed under the conditions of strain shall be admissible maximum 1 [mm]. They develop in the peripheral area of the flange connection, seeing a drop in this strain when using the shoulder portion .

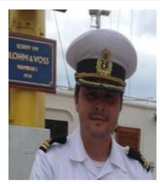
SolidWorks users fully resistance theory for calculating energy equivalent stress is the theory of variation in shape, Huber - Hencky - Mises .

Adopted annular sectional shape is more advantageous stress and strain observing only the outer fibers of the material.

REFERENCES

[1]. Blumenfelt Maty „Introducere în metoda elementelor finite”, Editura Tehnică, Bucureşti, 1995.
[2]. Buzdugan Gheorghe „Rezistenţa materialelor”, Editura Academiei Republicii Socialiste România, Bucureşti, 1986.
[3]. Gârbea Dan „Analiză cu elemente finite”, Editura Tehnică, Bucureşti, 1990.
[4]. Pescariu Ion „Elemente finite. Concepte. Aplicaţii”, Ediura Militară, Bucureşti, 1985.
[5]. Kmen Flaviu “Elemente de proiectare motorului principal al unui mineralier de 25.000 TDW și analiza structurală a unui element component” dissertation , Constanța, 2012
[6]. PRICOP MIHAIL ş.a „Structuri elastice. Elemente de teoria elasticităţii. Vibraţii”, Editura Ex ponto, Constanţa, 2001.
[7]. www.ipa.ro/AMTRANS , October 2015
[8]. www.omtr.pub.ro , October 2015

Authors Profile



Kmen Flavius is a PhD candidate in mechanical engineering at the University of Galati Lower Danube, under the guidance Prof.Univ.Dr.eng Silviu Măcuţă. Current studies are focused on the phenomenon of low cycle fatigue for the materials used in the shipping industry. He is currently chief engineer at N.S. Mircea, România.

Silviu Macuta is professor in Mechanical Engineering Departament/Engineering Faculty/. the University of Galati Lower Danube, Coordonator PhD student.

Prof.Silviu Macuta have 2 patents,135 scientific papers,14 books publish, and is International expert evaluator.