Calculus of the geometrical characteristics of the sections using CAD/CAE commercial applications

To cite this article: E Oanta et al 2018 IOP Conf. Ser.: Mater. Sci. Eng. 400 042042

View the article online for updates and enhancements.
Calculus of the geometrical characteristics of the sections using CAD/CAE commercial applications

E Oanta, A Sabau and M Barhalescu
Constanta Maritime University, Faculty of Naval Electro-Mechanics, 104 Mircea cel Batran Street, 900663, Constanta, Romania
E-mail: barhalescum@yahoo.com

Abstract. Analytical and numerical models of the structures use the geometrical characteristics in most of the problems, such as: calculus of the stresses, of the displacements, in elastic stability problems and others. Being an important problem to be solved, we conceived general methods to compute the geometrical characteristics. Along the time, an interesting question for us was how our calculus methods may be linked to the existing commercial software applications. From this standpoint we started to extensively test the CAD/CAE applications in order to understand their generality and flexibility in the calculus of the geometrical characteristics and, further on, of the stresses. We followed several aspects: if the boundary of a section may be automatically loaded, if there are ‘tricks’ regarding the definition of the model in the current CAD application, if there are commands/functions for the automatic calculus of the geometrical characteristics and how the results may be exported. We tested AutoCAD, NX, FEMAP/NASTRAN and ANSYS, detailed information regarding the solutions being presented for each CAD. Last but not least, this study is interesting in order to test the results of our computing methods and to assess their accuracy and their level of generality.

1. Introduction

Automatic calculus in structural problems based on the analytical methods is a long run concern of the authors, [1], [2], along the time being conceived general methods to compute the geometrical characteristics. The first direction of generalization was more than 20 years ago and it considered that the cross section may be divided in ‘simple’ shapes whose sign may be either +1 or -1, in this way being defined a Boolean algebra, [3]. Moreover, we conceived an algorithm according to which the maximum stresses may be found by searching into a very small set of points in the section, in this way being defined a fast algorithm, [4]. The second generalization developed 4 years ago considered a calculus domain bordered by curves modelled as spline functions for which we computed the geometrical characteristics using the GMP arbitrary precision library, [5]. The third generalization presented 2 years ago considered that the domain may be divided in several polygons for which we have direct calculus methods, [6]. This method used linearization on subdomains of the initial calculus domain. After the methods were checked and double checked, last year we conceived an automatic calculus generalization method by defining the ‘calculus domain’ more general concept, according to which a domain may be divided in several generally defined sub-domains whose sign is +1 or -1, [7].
2. Motivation

Let us consider that we use a right hand system of axes, the $X$ axis being along the beam, the $Z$ axis is downwards and the $Y$ axis comes outside the vertical plane. The observer is located at the left end of the beam and in a cross section the $Z$ axis is downwards and the $Y$ axis oriented to the right, figure 1, [8]. This system of axes will be used when the values of the geometrical characteristics will be compared.

![Figure 1](image)

**Figure 1.** The system of axes on a beam and in the cross section of a beam.

As we mentioned before the geometrical characteristics are used in beam models for several purposes. In this way, the stresses produced by the bending moment and by the shear forces are computed using the relations:

$$
\sigma(y, z) = \frac{N}{A} + \frac{M_y \cdot I_z + M_z \cdot I_{yz} \cdot z}{I_y \cdot I_z - I_{yz}^2} \cdot y - \frac{M_y \cdot I_y + M_y \cdot I_{yz} \cdot y}{I_y \cdot I_z - I_{yz}^2} \cdot z, \text{ or }
$$

$$
\sigma(y, z) = \frac{N}{A} + \frac{I_z \cdot z - I_{yz} \cdot y}{I_y \cdot I_z - I_{yz}^2} \cdot M_y - \frac{I_y \cdot y - I_{yz} \cdot z}{I_y \cdot I_z - I_{yz}^2} \cdot M_z,
$$

(1)

respectively

$$
\begin{align*}
\tau^x &= \begin{pmatrix} T_z \\ I_y \end{pmatrix} \cdot \begin{pmatrix} S_y \\ b_y \end{pmatrix}, \\
\tau^y &= \begin{pmatrix} T_y \\ I_z \end{pmatrix} \cdot \begin{pmatrix} S_z \\ b_z \end{pmatrix}.
\end{align*}
$$

(2)

The calculus of the displacements also uses the geometrical characteristics, no matter that we use the method of initial parameters

$$
\begin{align*}
\frac{d^2 u_z(x)}{dx^2} &= \frac{d\varphi_{xoz}(x)}{dx} = -\frac{1}{E \cdot I_y} \cdot M_y(x) \\
\frac{d^2 u_y(x)}{dx^2} &= \frac{d\varphi_{xoy}(x)}{dx} = +\frac{1}{E \cdot I_z} \cdot M_z(x)
\end{align*}
$$

(3)

or the strain energy methods:
\[ \delta_k = \sum_{i=1}^{n} \left[ \frac{1}{(E \cdot A)_i} \cdot \int_{L_i} (N_i \cdot n_i) \cdot dx \right] + \sum_{i=1}^{n} \left[ \frac{k_T}{(G \cdot A)_i} \cdot \int_{L_i} (T_{Zi} \cdot t_{Zi}) \cdot dx \right] + \sum_{i=1}^{n} \left[ \frac{1}{(G \cdot I_p)_i} \cdot \int_{L_i} (M_{Xi} \cdot m_{Xi}) \cdot dx \right] + \sum_{i=1}^{n} \left[ \frac{1}{(E \cdot I_{yi})_i} \cdot \int_{L_i} (M_{yi} \cdot m_{yi}) \cdot dx \right] \]

where the definition of the form factors for interval ‘i’, [9], is

\[ k_{yi} = \frac{A_{yi}}{I_{yi}} \cdot \int_{\lambda} \left( \frac{S_{yi}}{b_{yi}} \right)^2 \cdot dA, \quad k_{zi} = \frac{A_{zi}}{I_{zi}} \cdot \int_{\lambda} \left( \frac{S_{zi}}{b_{zi}} \right)^2 \cdot dA. \]

Moreover, for buckling problems, Euler's formula for long slender columns is

\[ F_b = \frac{\pi^2 \cdot E \cdot I_z}{\ell_b^2}, \]

where \( \ell_b = k \cdot \ell \) and \( k \) is a coefficient designated as the column effective length factor, its values depending on the types of supports at the ends of the column. It results that the geometrical characteristics are paramount for the structural calculus, being required accurate methods to compute their values, together with alternate methods to verify their correctness. The question is about what methods may be used to verify the values of the geometrical characteristics.

### 3. Analysis of the CAD commercial software capabilities

The answer may be offered by the commercial software applications which may be improved using the aforementioned original methods. From this standpoint we started to extensively analyse and test the CAD/CAE applications in order to understand their generality and flexibility in the calculus of the geometrical characteristics and, further on, of the stresses in beam models. We followed several aspects:

- if the sections’ boundary definitions may be automatically loaded;
- if there are ‘tricks’ regarding the definition of the model in the current CAD application;
- if there are commands/functions for the automatic calculus of the geometrical characteristics;
- methods to export the results, if any.

Several CAD applications were tested: AutoCAD, NX, FEMAP/NASTRAN and ANSYS.

The exploratory analysis of these CAD commercial applications is also useful in order to understand the computing methods and their level of generality, methods which may be used in other models. Other aspect concerned the methods to include the CAD capabilities in models having a higher level of generality.

#### 3.1. Solution in AutoCAD

Let us consider that the boundary of the cross section was automatically generated and it is expressed as a series of LINE commands stored in a SCR script file. The AutoCAD script files are text files which may be generated by any programming languages.

The information in the script file is loaded using the SCRIPT command. Using the REGION command and the ALL option, there is defined a region using all the lines on the boundary of the section. Next we use MASSPROP to automatically compute the geometrical characteristics. We are allowed to save the results in a text file having the MPR extension. Being a text file, it can be read
from any programming language, i.e. the values of the geometrical characteristics may be loaded in the upper level applications.

Two aspects must be reminded:

- when the script files are generated there should be paid attention to the blank characters because AutoCAD considers them <Enter> commands;
- the line segments defined in the script file must be defined one after the other, not randomly, in order to properly define the boundary of the cross section, otherwise the REGION command returns the ‘1 loop rejected’ error message.
Figure 5. The values of the geometrical characteristics saved in a MPR text file.

The values of the geometrical characteristics presented in figures 4 and 5 are computed in a system of axes whose origin is in the left-down corner of the bulb flat. In order to compute the centroid geometrical characteristics, we must use Steiner’s theorem, a.k.a. the parallel axes theorem, [8]:

\[
I_y = 189749.6606 \frac{\left( 7.4890 \right)^2 \cdot 1933.9209}{A} = 81285.47 \text{ mm}^4,
\]

\[
I_z = 21404482.2849 \frac{\left( 92.6297 \right)^2 \cdot 1933.9209}{A} = 4810935.59 \text{ mm}^4,
\]

\[
I_{yz} = -1628117.1596 + \left( 7.4890 \right) \frac{\left( 92.6297 \right) \cdot 1933.9209}{A} = -286548.84 \text{ mm}^4.
\]

In the last relation we use ‘+’ for \( \varepsilon_y \cdot \varepsilon_z \cdot A \) because the product moment of area must have a minimum absolute value in the centroid of the section.

The values of the geometrical characteristics resulted from the analytical model which approximates the HP160x10 bulb flat using several polygonal subdomains, the calculus relations being presented in [6], are: \( I_y = 81234.7925 \text{ mm}^4 \), \( I_z = 4.81094 \times 10^3 \text{ mm}^4 \) and \( I_{yz} = -2.84269 \times 10^3 \text{ mm}^4 \), where \( Y \) axis is horizontal and \( Z \) axis is vertical as it was previously presented in figure 1.

The according errors with respect to the values resulted from the analytical model are: \( \varepsilon_{I_y} = -0.00083 \% \), \( \varepsilon_{I_z} = 0.00009 \% \) and \( \varepsilon_{I_{yz}} = -0.80200 \% \).

3.2. Solution no. 1 in NX

We consider the parametric design of a bulb flat, as it is presented in the following figure. The dimensions are for the HP160x10 bulb flat.
Figure 6. Parametric design of a bulb flat.

Figure 7. Calculus of the geometrical characteristics of a closed 2D solid curve.

The parametric design allows us to modify a few parameters of the model in order to create another bulb flat. Some of the dimensions are automatically changes with respect to the actual values of the main dimensions.

By selecting ‘Menu’→‘Analysis’→‘Section Inertia’→‘Section Type = Solid’ → ‘Select Curve’ → ‘Show Information Window’ the geometrical characteristics are stored in a text file which may be saved. Then, by selecting ‘Apply’ the geometrical characteristics are displayed in the previously selected layer, as it is presented in the previous figure.

The according principal second moments of area of the analytical model which approximates the HP160x10 bulb flat with polygons are: \( I_1 = 4.8280 \times 10^6 \text{ mm}^4 \), \( I_2 = 6.4261 \times 10^4 \text{ mm}^4 \). The relative errors are very small, i.e. \( \varepsilon_{I_1} = -0.052 \% \) and \( \varepsilon_{I_2} = 0.167 \% \).
3.3. Solution no. 2 in NX

Basic idea is: starting from the sketch presented in figure 6, we use the ‘Extrude’ command in order to create a 3D domain which is included in a finite element method analysis (structural), where we define a ‘1D Element Section’ whose cross section is actually the initial sketch; then we ‘Evaluate Section Properties’.

In order to create the 3D mechanical part using the ‘Extrude’ command, the chain of commands is:
Menu: Insert → Design Feature → Extrude → Select Curve → (Sketch: HP160x10) → Specify Vector → (The axis perpendicular on the geometric plane that includes the HP160x10 sketch) → End: 2000 →<OK>.

The 3D part is positioned in an appropriate central location of the window using the FIT, PAN, and ROTATE commands. The results are presented in the following figure where the following chain of commands is presented: Menu: File → Utilities → New FEM and Simulation.

![Image of NX software interface](image)

**Figure 8.** The 3D domain and the options to be selected to define a new FEM simulation.
The appropriate options of the FEM solution are presented in figure 9 and the chain of selections used to start the definition of a ‘1D Element Section’ are presented in figure 10: Mesh → More → ‘1D Element Section’. The ‘Beam Section Manager’ window is started, figure 11. In this window we select the ‘Create Section’ option, as it is presented in the following figure.
Figure 11. Option used to define the cross section of the ‘1D Element Section’.

Figure 12. Selection of the face defined as the cross section and selection of the axis of reference of the cross section.

Once the ‘Create Section’ option was selected, there must be defined the face which is the cross section and the axis of reference of the cross section, in order to have appropriate notations of the geometrical characteristics. The aforementioned selections are presented in the previous figure. The results are presented in the following figure.
Figure 13. Results of the chain of selections meant to define a ‘1D Element Section’ of the FEM simulation.

Figure 14. Information displayed when the ‘Evaluate Section Properties’ option is selected.

If we select the ‘Evaluate Section Properties’ option, the ‘Information’ window will be opened, where the geometrical characteristics, among other information, are displayed. The contents of this window may be saved either using the html format, or as a simple text file.

In this way, the geometrical characteristics of the ‘1D Element Section’ cross section may be found and saved.
The values of the geometrical characteristics resulted from the second NX model are: 
\( I_Y = 81234.7715 \, mm^4 \), \( I_Z = 4809755.79 \, 26 \, mm^4 \) and \( I_{YZ} = -286442.34 \, 38 \, mm^4 \). The according errors with respect to the values resulted from the analytical model are: \( \varepsilon_{I_Y} = 0.0615 \% \), \( \varepsilon_{I_Z} = 0.0246 \% \) and \( \varepsilon_{I_{YZ}} = -0.7645 \% \).

3.4. Solution in Femap/Nastran

In Femap/Nastran the basic idea is to use the facilities specific to the beam elements used in the finite element analysis. There are several stages in the geometrical characteristics calculus process, depending on the final problem we must solve. However, we consider the most general case regarding the use of a general shape of the beams’ cross section in a finite element analysis.

First stage is to set the layer in which we draw the general shape of the beam’s cross section. In this way, we use the commands ‘Tools’ → ‘Layers’ → ‘New Layer…’ and we designate layer number 2 as ‘Boundary_Curve_Definition’. The following operation is to define this layer as the unique layer which is visible during the definition of the general section. We select ‘View’ → ‘Visibility’ → Layer (Tab) → View Multiple Layer = ON (radio button) → Active Layer = 2 (dropdown list) → check box is set to ‘ON’ for layer 2, ‘Boundary_Curve_Definition’ → ‘Done’ button. By the end of this selection process the user may see the information presented in the following figure. Once we complete the definition of the beam’s property (geometrical characteristics and material constants) this layer must become invisible.

![Figure 15. Setting the active layer where the general shape is defined.](image-url)

The next stage is to load the necessary information to create the curve which is the boundary of the beam type element whose property must be defined. We use the original API programming interface presented in [10], where the information regarding the geometry was structured in:

- data regarding the points (geometry) or nodes (mesh), consisting of \( ID \_ Point \_ X \), \( Y \), and \( Z \);  
- data regarding the ‘curves’ (geometry) as they are defined in Femap/Nastran or elements (mesh), consisting of \( ID \_ Group \_ ID \), \( ID \_ Elem \_ ID \), \( Node \_ I \) and \( Node \_ J \).
If the data was loaded as geometry, there was possible to create a more refined mesh. In this case we load the data as geometry in order to define the curve which is the boundary of the beam element.

![Figure 16. Using the Femap/Nastran API Programming facilities in order to load the curve information, necessary to define the boundary.](image)

![Figure 17. Results of the curve’s data loading process.](image)

We use Tools → Programming → API Programming, as it is presented in the left side of figure 16. Next we load the appropriate BAS application, as it is presented in the centre of figure 16. If we want more space to visualize the curve we may close the ‘Model Info’ window, which may be reopened using the icon pointed out by the green arrow in the centre of figure 16. Next we run the BAS application by pressing the ‘Play’ button, as it is presented in the right side of figure 16.

By the end of this stage the curve is defined. In order to visualize it, we use the commands View → Autoscale → Visible, the result being presented in the right side of figure 17.

Next we define the boundary, as it is presented in the following figure, using the commands Geometry → Boundary Surface → From Curves. In the ‘Entity Selection - Select Curve(s) on Closed
Boundary’ window, we select the current curve presented in its own visible layer using the mouse (or if there is only one curve we simply press the ‘Select All’ button) and then ‘OK’.

Figure 18. Definition of the boundary using the previously defined curve.

Figure 19. Creation of the current library of materials which will be assigned to various properties.

To select the materials used to define the properties, we use either the commands ‘Model’ → ‘Material…’ or we simply right click on the materials in the ‘Model Info’ window, as it is presented in the left side of the previous figure. In the ‘Define Material - ISOTROPIC’ window we click on the ‘Load’ button and the ‘Select From Library’ window is activated, as it is presented in the centre of the previous figure. Next we select a material, we press ‘OK’ and in the upper level window, i.e. ‘Define Material - ISOTROPIC’ we are allowed to customize the material’s constants. We are also allowed to
save the material in the library of materials in order to reuse it, as it is presented in the right side of the previous figure.

Figure 20. Selection of the ‘Element / Property Type’.

The next stage is to define the properties, either by selecting ‘Model’ → ‘Property...’ or by pressing a right click on ‘Properties’ in the ‘Model Info’ window, as it is presented in the left side of figure 20. Next we define the title of the property, i.e. ‘Beam Section no 1’ and we press the ‘Elem/Property Type’ button in order to select ‘Beam’ from the ‘Line Elements’ set, as it is presented in the centre of figure 20. After we complete these selections we return to the upper level window which is now designated ‘Define Property - BEAM Element Type’. In this window we press the ‘Shape’ button, as it can be noticed in the right side of figure 20.

Figure 21. Selection of the Element / Property Type.

In this stage, the ‘Cross Section Definition’ window is opened and from the dropdown list we select ‘General Section...’ and then we click on the ‘Surface...’ button, as it is presented in figure 21.
Figure 22. Selection of the previously defined curve and definition of the axis of reference.

Once the ‘Select Surface to Check’ window is opened we either input the identifier of the surface, or we simply click on it. Next, in the ‘Vector Locate – Define Section Y Axis’ we click on the ‘Preview’ button and we accordingly modify the coordinates of the ‘Tip’. Finally, we press on the ‘OK’ button, as it may be noticed in the previous figure.

Figure 23. Geometrical characteristics presented in two distinct windows of Femap/Nastran.

Now the ‘Cross Section Definition’ window is active again, so we click on the ‘Draw Section’ button and then on the ‘Flip Vert’ button, the results being presented in the left side of figure 23. Once we press the ‘OK’ button, the ‘Define Property – BEAM Element Type’ window is again visible and we are able to see the values of the geometrical characteristics, as they are presented in the right side of figure 23.
The values of the geometrical characteristics resulted from the Femap/Nastran model are: $I_y = 81407.33 \text{ mm}^4$, $I_z = 4811862. \text{ mm}^4$ and $I_{yz} = 286878.7 \text{ mm}^4$. The according errors with respect to the values computed using the analytical model are: $\varepsilon_{I_y} = -0.1507 \%$, $\varepsilon_{I_z} = -0.0192 \%$ and $\varepsilon_{I_{yz}} = -0.9180 \%$.

### 3.5. Solution in DS ANSYS DesignModeler

In ANSYS DesignModeler the input data structure must be carefully filtered. Once the input data is loaded there are fewer requirements.

For all the ANSYS geometry modellers the input data must define closed contours, otherwise the program is generating inconsistent surfaces / bodies and errors may occur.

The input data are imported as a 3D curve using the commands ‘Concept’ → ‘3D Curve’, as it is presented in the following figure.

![Figure 24. Import of the input data in ANSYS DesignModeler.](image.png)

In the ‘Tree Outline’ window we right click on the ‘Plane6’ icon and then we use the commands ‘Insert’→‘Sketch Projection’ in order to project the curve on a plane, as it is presented in figure 25.

Next we define a section having a general shape for which the commands are ‘Concept’→‘Cross Section’→‘User Defined’ and we copy the first sketch (the intersection between the curve and the plane) in the section's plane, figure 26. There must be reminded that a problem related to the system of axes of the new section may occur, depending on the point of reference used for the 'Paste' command.
Figure 25. Projection of the curve on a plane.

Figure 26. Defining a section having a general shape and copying the previous sketch in the section.

Using the commands ‘Create’→‘Body Transformation’→‘Mirror’ with respect to the axis of symmetry we draw the entire section. The ‘Mirror’ command and its results may be found in figure 27, where the values of the geometrical characteristics can be noticed.

Following the same procedure previously presented for a bulb flat, except the ‘Mirror’ command that is useless for a section without an axis of symmetry, we have the results presented in figure 28.
**Figure 27.** The ‘Mirror’ command and its results.

**Figure 28.** Results for a bulb flat and the values of the geometrical characteristics.
The values of the geometrical characteristics resulted from our analytical model of the HP120x8 bulb flat are: $I_y = 30891.3 \text{ mm}^4$, $I_z = 1634230 \text{ mm}^4$ and $I_{yz} = -103035 \text{ mm}^4$. The ANSYS DesignModeler results are: $I_y = 29478 \text{ mm}^4$, $I_z = 1563100 \text{ mm}^4$ and $I_{yz} = 99335 \text{ mm}^4$. The according errors with respect to the values resulted from the analytical model are: $\varepsilon_{iy} = 4.5707 \%$, $\varepsilon_{iz} = 4.3500 \%$ and $\varepsilon_{yz} = 3.5910 \%$.

3.6. Solution in SC ANSYS SpaceClaim
In ANSYS SpaceClaim there are fewer requirements regarding the condition to be fulfilled by the input data, but errors may be generated at a later stage.

![Figure 29. The structure of the input file and the SpaceClaim command.](image)

![Figure 30. The import is done using implicit coordinates.](image)
The curve was imported using the input data file. The input file has a header consisting of commands and the coordinates are in millimetres, starting with the Z axis. Figures 29 and 30 are presenting the structure of the input file and the commands used to import the data.

**Figure 31.** After the definition of the axis of symmetry a ‘Mirror’ operation is used to complete the generation of the entire section.

**Figure 32.** Using the ‘Fill’ command a surface was generated.
Figure 33. Using the ‘Push’ command we generate a 3D domain starting from the previously defined surface.

Figure 34. Using the ‘Mass Properties’ command we compute the geometrical characteristics.

The ANSYS SpaceClaim results of the HP120x8 bulb flat are: $I_x = 30891.46 \text{ mm}^4$, $I_y = 1614083 \text{ mm}^4$ and $I_{xy} = -105020 \text{ mm}^4$. The according errors with respect to the values computed using the analytical model are: $\varepsilon_{I_x} = -0.0005 \%$, $\varepsilon_{I_y} = 1.2300 \%$ and $\varepsilon_{I_{xy}} = 1.9264 \%$. 
4. Conclusions
The CAD commercial applications may be used to compute the geometrical characteristics of the beams’ cross section. In this way the results may be used for cross checking and to validate the results of our original applications.

The high level of detail of the procedures’ presentations is useful for several reasons:

- the presentation is explicit and useful for the CAD analysts;
- we have reliable methods to solve the geometrical characteristics problem, methods that may be reused;
- we are able to synthesize the methods used in the commercial CAD applications presented in the paper and to draw general conclusions regarding the algorithms;
- the methods previously presented may be used at a conceptual level when new CAD applications are studied;
- the methods used to define the general shape of the beams’ cross sections are useful for the finite method analysis;
- being conceived general methods to define the cross sections’ shape, we identified automatic CAD procedures, to be studied at a later stage;
- accuracy of the results may be the subject of a follow up study;
- acquiring knowledge regarding the calculus methods in CAD applications we are able to offer original methods to improve such calculus procedures.

The paper is a component of the computer based analytical models development direction, which is a part of the computer based hybrid models development strategy. The various ideas provided by many academic disciplines employed in our models have a synergic effect which is useful for the rapid development of a computer based research strategy.
Acknowledgement
This work was supported by a grant of the Romanian Ministry of Research and Innovation, CCCDI-UEFISCDI, project number PN-III-P1-1.2-PCCDI-2017-04-04/31PCCDI/2018, acronym HORESEC, within PNCDI III, [11].

References
[1] Oanta E 1994 Proceedings of the RezMat Conference (Timisoara, March 05, 1994) pp 36-38
[2] Oanta E1999 Proceedings of the RezMat6 Conference (Constanta, June 03, 1999) pp 81-86
[3] Oanta E 1998 Proceedings of TEHNONAV 1998: International Symposium on Marine Technologies and Management (Constanta, May 21-23, 1998) II pp 159-166
[4] Oanta E 1998 Proceedings of TEHNONAV 1998: International Symposium on Marine Technologies and Management (Constanta, May 21-23, 1998) II pp 143-150
[5] Oanta E, Constantinescu E, Raicu A and Axinte T 2014 Scientific.Net - Advanced Materials Research, 1036 pp 697-702
[6] Oanta E, Panait C, Raicu A, Barhalescu M and Axinte T 2016 IOP Conference Series: Materials Science and Engineering 145 082011
[7] Raicu A, Oanta E and Barhalescu M 2017 IOP Conference Series: Materials Science and Engineering 227 012109
[8] Oanta E 2014 Basic Knowledge in STRENGTH OF MATERIALS Applied in Marine Engineering for Maritime Officers Vol 1 (Constanta: Nautica Publishing House)
[9] Constantinescu E, Oanta E and Panait 2017IOP Conference Series: Materials Science and Engineering 227 012031
[10] Oanta E, Panait C and Raicu A 2016 Proceedings of SPIE, Advanced Topics in Optoelectronics, Microelectronics, and Nanotechnologies VIII, 10010 (December 14, 2016)
[11] Panait C 2017 HORESEC project, Romanian Ministry of Research and Innovation, CCCDI-UEFISCDI, project PN-III-P1-1.2-PCCDI-2017-04-04/31PCCDI/2018