2014 3rd International Conference on Informatics, Environment, Energy and Applications
IPCBEE vol.66 (2014) © (2014) IACSIT Press, Singapore
DOI: 10.7763/IPCBEE. 2014. V66. 15

Dynamic Analysis of Supporting Structure of Linear Fresnel Solar Concentrator through FEM, OOP C++ and AutoCAD

Silvia Caroline [1], Tiago Freires [2] , Raimundo Menezes [3]

[1,2] UFPB-CEAR-DEER-Laboratory of Numerical Methods Distributed Processing

[3] UFPB–CEAR–Department of Renewable Energy Engineering

João Pessoa, Brazil

Abstract. This paper describes a AutoCAD tool used for dynamic analysis of a Supporting Structure of Linear Fresnel Solar Concentrator. The Supporting Structure is represented by space frames which the mathematical representation is done using finite element methodology to obtain the global stiffness and mass matrices from the contribution of each local equation of the structure elements. The processing stage has built into a DLL program with routines to calculate the local and global matrices and solve the resultant equilibrium linear system. This stage is written in C++ programming language and uses the strategy to link the DLL with other ObjectARX program, then AutoCAD can access and use functions from the DLL to make the process analysis without quit from your own environment. For pre-processing ObjectARX Library functions are used in order to customize AutoCAD and then provide the graphical construction for geometry mesh and other requirements of de model. The post-processing stage has built using specific dialogs box to present all results analysis. In post-processing the displacement results are visualized in the graphic interface to show the dynamic deformations. Numerical examples are shown in order to illustrate the feasibility of present academic customized package developed by CEAR-LMPD.

Keywords: Solar Concentrator, Fresnel Linear, AutoCAD, OOP C++, ObjectARX, FEM.

1. Introduction

The majority of packages handle directly with external files to import or export archives in order to provide data communication through pre-processing to processing and finally to post-processing stages when modeling structural problems with Finite Element Analysis. In this paper is described an attempt to incorporate to standard drawing tools of AutoCAD package an tool related to Finite Element Analysis (FEA) of space frames, used to dynamic analysis of solar fresnel linear concentrator. The mechanical properties of element geometry, element connectivity, and boundary constraints can be added to drawing elements, providing a more friendly and efficient pre-processing environment. Next without quitting from AutoCAD, the stiffness and consistent mass matrices of all elements and equivalent nodal forces can be evaluated. The functions of processing stage are writing in oriented object language C++ and they are compiled as dll libraries and link to AutoCAD through a ObjectARX program. The routines programmed to process and post-process the model can be run from a menu of customized AutoCAD environment.

2. Pre-Processing

In pre-processing analysis input data (such as geometry, node and element numbering, mechanical properties, boundary conditions, loading) are set to perform the calculation of the discretized problem. This procedure to build the intermediate tools in order to incorporate FEM input data information into drawing [3-6] through ObjectARX that uses the Oriented Object Programming (OOP), with more than 200 classes and 3000 objects written in C++ programming language [1-2]. After the user draw the basic concept of the structure using lines and points, the menu option “Define Structure” can be accessed and a dialog box is shown (See figure 1a). After setting dialog box values, and chose the kind “Space Frame” to discretize the model, the user must press the "Ok" button, and the lines and points selected are labelled and prepared to receive input data information to the process stage. The graphical representation is visualized in Fig 1.

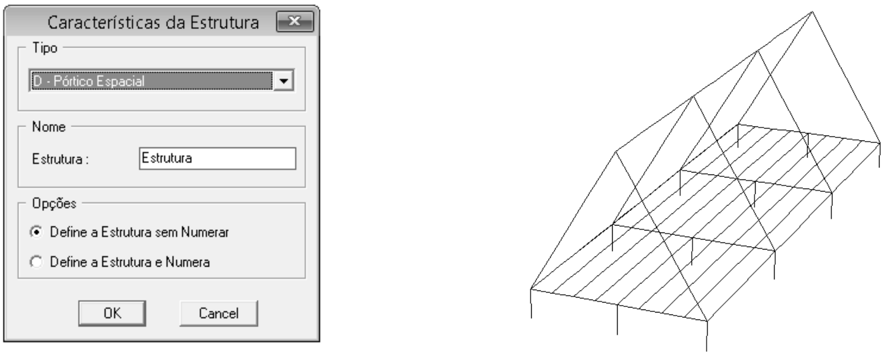


Fig 1: Pre-Processing "Define Structure Dialog"

In this point, the model is prepared to receiver another information like the boundary conditions that can be applied by selecting options in the dialog box, where all six degrees of freedom (displacements and rotations) can be accessed and one by one conveniently prescribed, see Fig. 2.

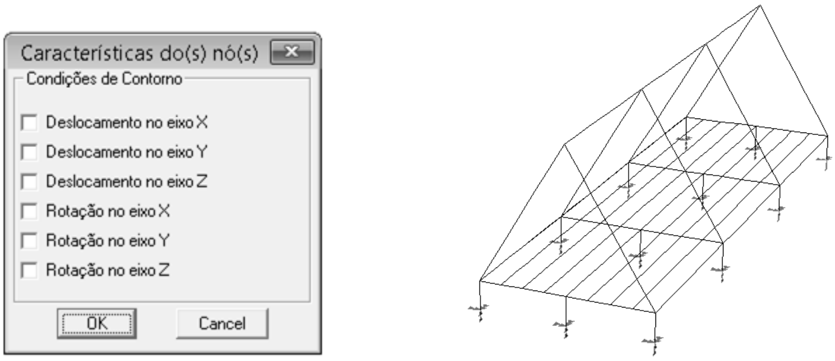


Fig 2: Boundary condition dialog box and updated model;

To complete the pre-processing stage concentrated and distributed loading must still be assigned. This can be done selecting the force dialog box, see Fig. 3.

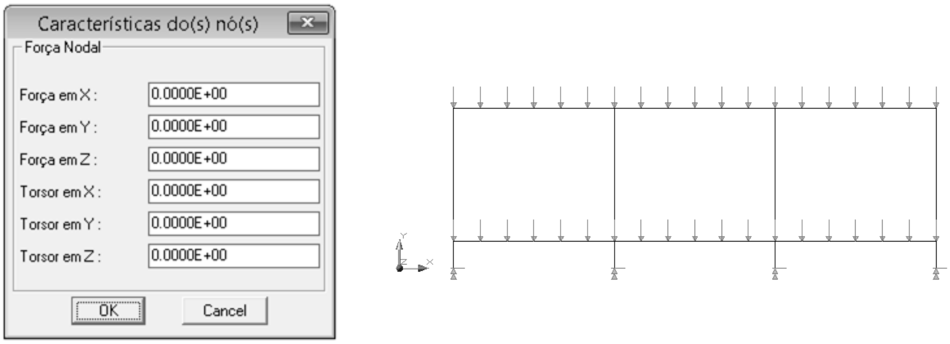


Fig. 3: Force dialog box and updated model;

Finally, the mechanical properties can be added to the model when the user fill the fields in dialog options to this propose. In Fig. 4, are shown all fields that must be setting by the user.

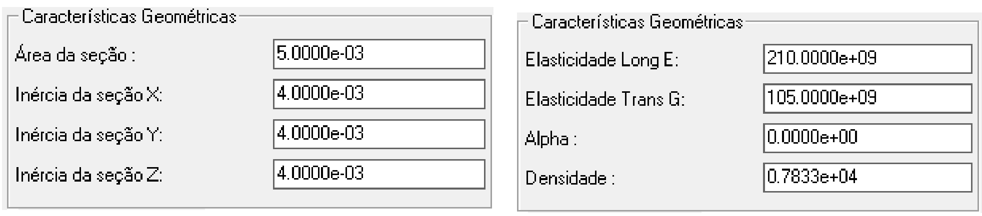


Fig. 4: Properties dialog box;

3. Processing

The processing is usually defined as a stage of the analysis in which main calculations (Elemental Stiffness and Mass matrix evaluation, structural Stiffness and Mass matrix assembling, nodal equivalent force vector evaluation, algebraic system solution, etc) are done. In present paper, the Fresnel Supporting Structure dynamic problem is modeled by reticulated elements using the Finite Element Method (FEM). The twelve final DOF of the element are shown in Fig. 5. The support structure analysis is made using the Amate.dll library, which have functions and classes implemented in oriented object language C++ compiled and linked in ObjectARX program, whose it can run from a menu of the AutoCAD. For sake of conciness, only principal classes and functions of the library are described too in Fig. 5.

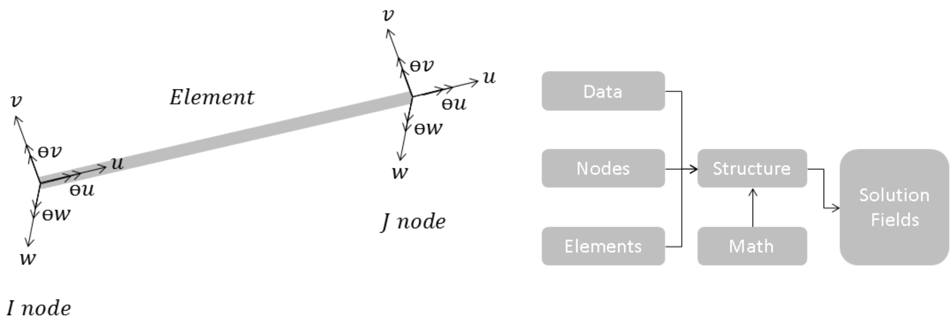


Fig. 5: Space Frame Element DOF and Amate.dll class diagram

In general, external functions can only be accessed from AutoCAD environment if these functions are compiled as ObjectARX format. In present work "solver" is a function defined as main function that uses methods and objects from Amate.dll. The "solver" function works in two steps: in the first the objects and functions of ObjectARX library collect the input data assigned in pre-processing stage, and store them in vectors. In the second step, these vectors are processed by objects and methods from Amate.dll to perform the structural calculations. From a point of view of user, all processing steps are done when he selects the menu option "processing >> Structural Calculus", and selects the Dynamic Analysis. In the same dialog box, the user can set up the frequency of load applied to the structure. See Fig. 6.

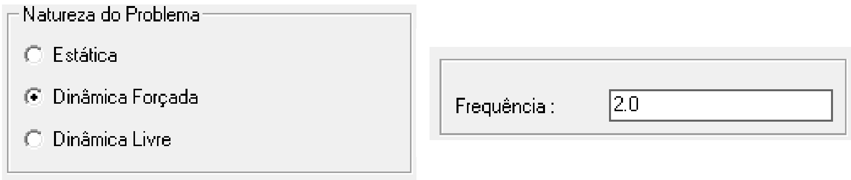


Fig. 6: "Structural Calculus" (Dynamic Analysis)

From the point of view of mathematics, the "solver" function uses the principle of Hamilton in the total energy. Assuming harmonic external excitation, the governing equation of the problem is.

$$[\mathrm{A}]\{\mathrm{D}\} = \{\mathrm{P}\} \quad (1)$$

Where $[\mathrm{A}] = [\mathrm{K}] - \varpi^2[\mathrm{M}]$, being ϖ vibration frequency; $[\mathrm{K}]$, $[\mathrm{M}]$ are the global stiffness and global consistent mass matrices from the structure. It should be noted, that $[\mathrm{K}]$ matrix is the accumulation of all stiffness element matrices in global coordinates.

$$[\mathrm{K}] = \sum_{i=1}^{n} [\widetilde{K}] \quad (2)$$

Where $[\widetilde{K}]$ given by $[\widetilde{K}] = [T][k][T]^T$. The same process is made for the consistent mass matrix [M], that is obtained by the accumulation of all mass element matrices.

$$[M] = \sum_{i=1}^{n} [\widetilde{M}] \quad (3)$$

Where $\left[\widetilde{M}\right] = [T][m][T]^T$ and $[m] = \rho AL[m']$. Below shows the stiffness matrix in (4) and consistent mass matrix $[m']$ in (5).

$$\begin{bmatrix}
\frac{EA}{L} & 0 & 0 & 0 & 0 & 0 & -\frac{EA}{L} & 0 & 0 & 0 & 0 & 0 \\
0 & \frac{12E\cdot Iz}{L^3} & 0 & 0 & 0 & \frac{6E\cdot Iz}{L^2} & 0 & -\frac{12E\cdot Iz}{L^3} & 0 & 0 & 0 & \frac{6E\cdot Iz}{L^2} \\
0 & 0 & \frac{12E\cdot Iy}{L^3} & 0 & -\frac{6E\cdot Iy}{L^2} & 0 & 0 & 0 & -\frac{12E\cdot Iy}{L^3} & 0 & -\frac{6E\cdot Iy}{L^2} & 0 \\
0 & 0 & 0 & \frac{G\cdot Ix}{L} & 0 & 0 & 0 & 0 & 0 & -\frac{G\cdot Ix}{L} & 0 & 0 \\
0 & 0 & -\frac{6E\cdot Iy}{L^2} & 0 & \frac{4E\cdot Iy}{L} & 0 & 0 & 0 & \frac{6E\cdot Iy}{L^2} & 0 & \frac{2E\cdot Iy}{L} & 0 \\
0 & \frac{6E\cdot Iz}{L^2} & 0 & 0 & 0 & \frac{4E\cdot Iz}{L} & 0 & -\frac{6E\cdot Iz}{L^2} & 0 & 0 & 0 & \frac{2E\cdot Iz}{L} \\
-\frac{EA}{L} & 0 & 0 & 0 & 0 & 0 & \frac{EA}{L} & 0 & 0 & 0 & 0 & 0 \\
0 & -\frac{12E\cdot Iz}{L^3} & 0 & 0 & 0 & -\frac{6E\cdot Iz}{L^2} & 0 & \frac{12E\cdot Iz}{L^3} & 0 & 0 & 0 & -\frac{6E\cdot Iz}{L^2} \\
0 & 0 & -\frac{12E\cdot Iy}{L^3} & 0 & \frac{6E\cdot Iy}{L^2} & 0 & 0 & 0 & \frac{12E\cdot Iy}{L^3} & 0 & \frac{6E\cdot Iy}{L^2} & 0 \\
0 & 0 & 0 & -\frac{G\cdot Ix}{L} & 0 & 0 & 0 & 0 & 0 & \frac{G\cdot Ix}{L} & 0 & 0 \\
0 & 0 & -\frac{6E\cdot Iy}{L^2} & 0 & \frac{2E\cdot Iy}{L} & 0 & 0 & 0 & \frac{6E\cdot Iy}{L^2} & 0 & \frac{4E\cdot Iy}{L} & 0 \\
0 & \frac{6E\cdot Iz}{L^2} & 0 & 0 & 0 & \frac{2E\cdot Iz}{L} & 0 & -\frac{6E\cdot Iz}{L^2} & 0 & 0 & 0 & \frac{4E\cdot Iz}{L}
\end{bmatrix}$$

(4)

$$\begin{bmatrix}
\frac{1}{3} & 0 & 0 & 0 & 0 & 0 & \frac{1}{6}L & 0 & 0 & 0 & 0 & 0 \\
0 & \frac{13}{35} & 0 & 0 & 0 & -\frac{11}{210}L & 0 & \frac{9}{70}L & 0 & 0 & 0 & -\frac{13}{420}L \\
0 & 0 & \frac{13}{35}L & 0 & \frac{11}{210}L & 0 & 0 & 0 & \frac{9}{70}L & 0 & \frac{13}{420}L & 0 \\
0 & 0 & 0 & \frac{1}{3}I_z & 0 & 0 & 0 & 0 & 0 & \frac{1}{6}I_z & 0 & 0 \\
0 & 0 & -\frac{11}{210}L & 0 & \frac{1}{105}L^2 & 0 & 0 & 0 & -\frac{13}{420}L & 0 & -\frac{1}{140}L^2 & 0 \\
0 & \frac{11}{210}L & 0 & 0 & 0 & \frac{1}{105}L^2 & 0 & \frac{13}{420}L & 0 & 0 & 0 & -\frac{1}{140}L^2 \\
\frac{1}{6} & 0 & 0 & 0 & 0 & 0 & \frac{1}{3} & 0 & 0 & 0 & 0 & 0 \\
0 & \frac{9}{70} & 0 & 0 & 0 & \frac{13}{420}L & 0 & \frac{13}{35} & 0 & 0 & 0 & -\frac{11}{210}L \\
0 & 0 & \frac{9}{70}L & 0 & -\frac{13}{420}L & 0 & 0 & 0 & \frac{13}{35} & 0 & \frac{11}{210}L & 0 \\
0 & 0 & 0 & \frac{1}{6}I_z & 0 & 0 & 0 & 0 & 0 & \frac{1}{3}I_z & 0 & 0 \\
0 & 0 & \frac{13}{420}L & 0 & -\frac{1}{140}L^2 & 0 & 0 & 0 & \frac{11}{210}L & 0 & \frac{1}{105}L^2 & 0 \\
0 & -\frac{13}{420}L & 0 & 0 & 0 & -\frac{1}{140}L^2 & 0 & -\frac{11}{210}L & 0 & 0 & 0 & \frac{1}{105}L^2
\end{bmatrix}$$

(5)

Where I_x, I_y, I_z are inertial moments from section of the finite element. E is elasticity module, L is the length of the element and A is the Area of the section. In both cases, static and dynamic, the $[T]$ matrix is present. This matrix represent the transformation coordinate system from local to global coordinates. For the space frame case its form is:

$$[\beta] = \begin{bmatrix} [B_{pe}] & [0] & [0] & [0] \\ [0] & [B_{pe}] & [0] & [0] \\ [0] & [0] & [B_{pe}] & [0] \\ [0] & [0] & [0] & [B_{pe}] \end{bmatrix} \quad (6)$$

Where $[B_{pe}]$ is $\begin{bmatrix} C_x & C_y & C_z \\ -(C_x.C_y.cos\alpha + C_z.sen\alpha)/C_{xz} & cos\alpha.C_{xz} & (-C_y.C_z.cos\alpha + C_x.sen\alpha)/C_{xz} \\ (C_x.C_y.sen\alpha + C_z.cos\alpha)/C_{xz} & -C_{xz}.sen\alpha & (-C_y.C_z.sen\alpha + C_x.cos\alpha)/C_{xz} \end{bmatrix}$

The cosines are: $C_x = \frac{x_j - x_i}{L}$; $C_y = \frac{y_j - y_i}{L}$; $C_z = \frac{z_j - z_i}{L}$; $C_{xz} = \sqrt{C_x^2 + C_z^2}$;

And the length of each element is: $L = \sqrt{(x_j - x_i)^2 + (y_j - y_i)^2 + (z_j - z_i)^2}$

If C_{xz}=0, then $[B_{pe}] = \begin{bmatrix} 0 & C_y & 0 \\ -C_y.cos\alpha & 0 & sen\alpha \\ C_y.sen\alpha & 0 & cos\alpha \end{bmatrix}$

4. Post-Processing

The Post-processing is usually defined as a stage of the analysis in which the results are shown. The customized AutoCAD, allows graphical outputs for displacements and deformed shapes by selecting the menu Post-Processing (see Fig. 8). In this case, two basic displacements modes are visualized. It should be noted that maximum values of the displacements are oscillating as a time function.

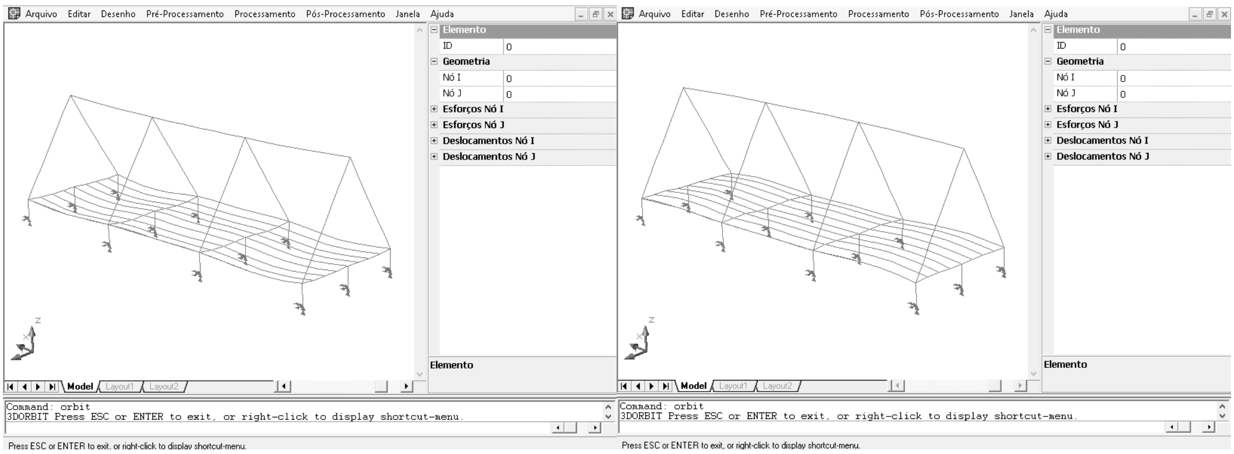


Fig. 8: Negative and Positive Dynamic Displacements

The analysis of the displacement fields for different frequencies of the loads present in the support structure is given by equation.

$$f(t) = d\ .\ sin(w\ .\ t) \quad (7)$$

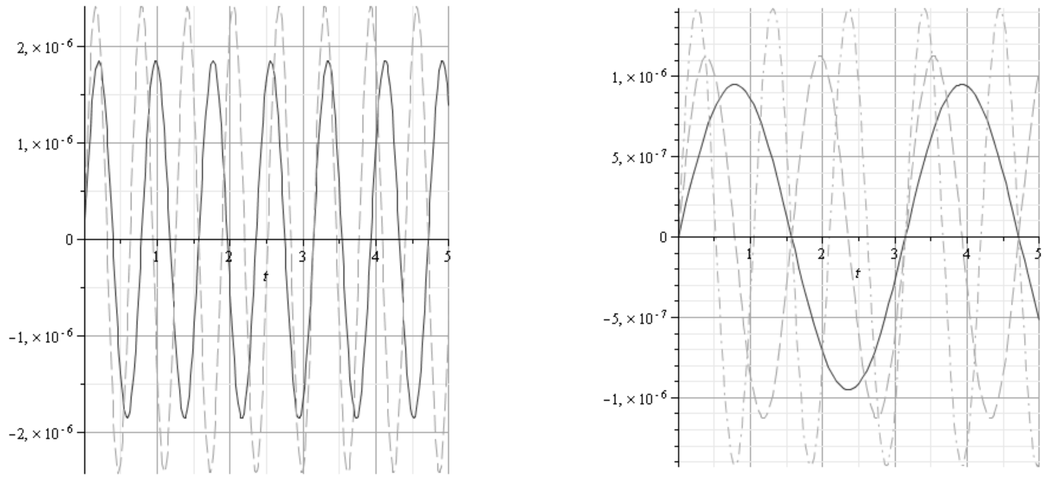
Where:

d = Displacement obtained from the equation (1).
w = Frequency of the loads in the support structure
t = Time in Seconds

Different frequencies for the load applied in middle of the support structure model are tested, see table 2.

Table 2 – Tested Frequencies and Displacements	
w (Hertz)	d (m)
2.0	9.491660e-007
4.0	1.125750e-006
6.0	1.424010e-006
8.0	1.850114e-006
10.0	2.413293e-006

The graphical results for the displacement variation in the time for the frequencies 2.0, 4.0, 6.0, 8.0 and 10.0 hertz are shown in figure 15.



W1 = 2.0 hertz, W2 = 4.0 hertz W3 = 6.0 hertz W4 = 8.0 hertz and W5 = 10.0 hertz

Figure 15 – Time and Frequency Displacement results

Conclusions

In this paper a parametric tool to dynamic analysis of solar fresnel linear concentrator was was presented. The main attractive feature of this structural analysis tool is the user-friendly environment implemented using ObjectARX library to provide the graphical construction for geometry, mesh orientation, and other requirements of the finite element model. Another important contribution is the DLL program for the processing stage implemented in C++ language for the space frame finite element. All stages of analysis namely pre-processing, processing and post-processing are completely integrated within a single tool for static and dynamic analysis.

Acknowledgment

The first two authors are grateful to the National Council For scientific and technological development (CNPQ) for the financial support for this research.

References

[1] Stein, D M. The Visual LISP Developer's Bible, Autodesk (2011).

[2] Autodesk Inc. AutoCAD 2014 - User’s Guide, (2013).

[3] Bathe, K.J. Finite Element Procedures. Cambridge, MA: Klaus-Jürgen Bathe., (2006).

[4] Reddy, J.N. An Introduction to the Finite Element Method (Third ed.). McGrawHill, (2005).

[5] Chaskalovic, J. Finite Elements Methods for Engineering Sciences, Springer Verlag, (2008).

[6] Zienkiewicz, O. C. R. L. Taylor, J. Z. Zhu : The Finite Element Method: Its Basis and Fundamentals, Butterworth-Heinemann, (2005).