

A customized AutoCAD environment for structural analysis using three bending plate finite elements

Raimundo Aprígio de Menezes Jr., Moisés Salvino Menezes
Laboratório de Ensaios de Materiais e Estruturas, CT, Universidade Federal da Paraíba, PB – Brazil

Faustino Sanches Junior
Departamento de Engenharia de Estruturas, EESC, Universidade de São Paulo, SP – Brazil

Ângelo Vieira Mendonça
Departamento de Engenharia Civil, CT, Universidade Federal da Paraíba, PB – Brazil

Abstract

This paper deals with an attempt to incorporate to standard drawing tools of Autocad package an evaluation tool related to Finite Element Analysis (FEA) of bending plate problems. Many aspects of development of the present customized environment called StructCAD are described. Firstly some remarks are addressing to pre-processing stage where analysis properties required by FEA can be added to drawing information. Then without quitting customized environment, the stiffness matrices and equivalent nodal forces can be evaluated. At last the post-processing libraries can be run in order to complete the analysis without quitting the customized environment. Numerical examples are shown in order to illustrate the feasibility of present customized package and to make comparisons among triangular bending plate elements.

Keywords: structural analysis, DKT, HSM, DST, OOP

1 Introduction

Engineering practice requires engineer to deal with structures containing usually high degrees of freedom so that analysis procedures can be become a cumbersome task if there is not an efficient and friendly tools to do the stages of analysis, which are defined from pre-processing to post-processing of numerical model. There are many commercial and academic packages available in engineering community and they can basically classified into two groups: a) single structural analysis systems and b) complete structural analysis systems. In first group the available tools are restricted to determining specifically quantities such as displacements and effort fields. Majority of analysis soft wares provide

Finite Element Method (FEM) calculation incorporating many mathematical models. In case of frame and slab problems, some academic [1–3] and commercial [4–7] packages can be addressed. The second group of structural analysis provides a wider spectrum of tools. In addition to calculation of quantity problem, the package is also able to check design criteria according to brazilian and/or international code for a specific material such as reinforced concrete and steel. In some packages are also available drawing tools using a CAD (Computer Aided Design) system in order to detailing stages of the structural members [8–13]. The majority of aforementioned 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. In contrast, oriented object programming strategies have been used in conjunction with drawing packages such as Autocad [14]. Santos & Devloo[15], Campagnã & Meneses[16], Maud et al.[17] describe an attempt of compatibilization between drawing environment of Autocad package and finite element analysis environment (classes in C++ for frame and bending plate problems) using ObjectArx library.

In this paper is described an attempt to incorporate to standard drawing tools of Autocad package an evaluation tool related to Finite Element Analysis (FEA) of bending plate problems. Hence, other properties than element geometry such as mechanical properties, element connectivity, and boundary constraints can be added to drawing information, providing a more friendly and efficient pre-processing environment. Next without quitting customized Autocad, the stiffness matrices and equivalent nodal forces related to bending plate finite elements can be evaluated. The evaluation functions are writing in oriented object language C++ and they are compiled as dll libraries, whose can be run from a menu of customized Autocad environment. At last the analysis is completed as soon as the post-processing libraries are run. The analysis results are shown in specific dialog boxes run from customized Autocad as well. Numerical examples are shown in order to illustrate the feasibility of present customized package and to make comparisons among triangular bending plate elements well-known in literature such as Hybrid Stress Model (HSM), Discrete Kirchhoff Theory (DKT) and Discrete Shear Theory (DST).

2 Pre-processing stage

The pre-processing is usually defined as a stage of the analysis where input data (such as geometry, node and element numbering, mechanical properties, boundary conditions, loading) are set to perform the calculation of the discretized problem. The original Autocad environment provides only drawing capabilities so that it is necessary to build intermediate tools in order to incorporate remaining FEM input data into drawing information. This procedure of compatibilization has usually been done using two strategies namely Autolisp Language [18] and ObjectARX library [19]. In first approach the source code must be shown in explicit form to the users because the Autolisp is an interpreted programming language. In second approach is used the Oriented Object Programming (OOP), where ObjectARX library with more than 200 classes and 3000 objects are written in C++ programming language. It should noted the ObjectARX library has a compiled form and therefore the source code is not necessarily to have be shown to the users.

In present work was used Autolisp language for pre-preprocessing programming in order to implement the drawing routines and to assign structural analysis data. Principal functions of pre-processing menu in StructCAD are: “Projetar Nós da Estrutura”, “Projetar Placas da estrutura”, “Define e Numera estrutura”, etc. For sake of conciseness, only Autolisp source code for function “Projetar Placas da estrutura” is shown in Table 1. For information about remaining Autolisp functions in StructCAD, the reader is addressed to Menezes Jr [20]

Table 1: Autolisp source code for “Projetar Placas da estrutura” function

```
1 ;;ProjetaPlacas
2 ;; (ProjetaPlacas)
3 (defun ProjetaPlacas ()
4
5 (setq dh (load_dialog "C:\\Amate\\Suporte\\m_placas.dcl"))
6 (if (not (new_dialog "ProjPlacas" dh))
7 (exit)
8 )
9
10 (setq OK 0)
11
12 (if (not $N_Placa)
13 (setq $N_Placa "1")
14 )
15
16 (if (not $N_elemX)
17 (setq $N_elemX "1")
18 )
19
20 (if (not $N_elemY)
21 (setq $N_elemY "1")
22 )
23
24 (set_tile "N_Placa" $N_Placa)
25 (set_tile "N_elemX" $N_elemX)
26 (set_tile "N_elemY" $N_elemY)
27
28 (action_tile "N_Placa" "(setq $N_Placa $value)")
29 (action_tile "N_elemX" "(setq $N_elemX $value)")
30 (action_tile "N_elemY" "(setq $N_elemY $value)")
31
32 (setq OK (start_dialog))
33 (unload_dialog DH)
34
35 (if (= OK 1)
36 (progn
37
```

Table 1: Autolisp source code for “Projetar Placas da estrutura” function (continued)

```

38 (princ "\nDefina a placa através da sua diagonal.")
39 (setq Vrtc1 (getpoint "\nPosicione o Vertice Inicial: "))
40 (initget 32)
41 (setq Vrtc2 (getcorner Vrtc1 "\nPosicione o Vertice Oposto: "))
42 (princ "\n")
43
44 (setq NelemX (atoi $N_elemX) NelemY (atoi $N_elemY)
45 )
46
47 (DefinePlaca)
48
49 )
50 )
51
52 (princ)
53 )
54 ;;end

```

When the user selects the menu option “projetar placas”, the dialog Box “Projetar elemento finito de placa” is shown in the customized environment, so that this corresponds to execute the line 5 instructions in source code shown in Table 1. After setting dialog box values, the user must press the “Ok” button to apply the input data (see Figure 1b). This procedure executes source code line 47 in Table 1, resulting in discretized form of plate (see Figure 2).

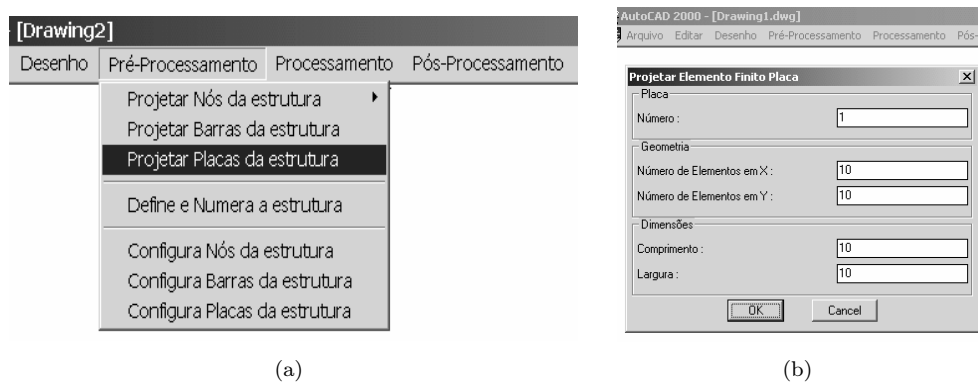


Figure 1: (a) Projetar Placas da estrutura; (b) Projetar Elemento Finito Placa

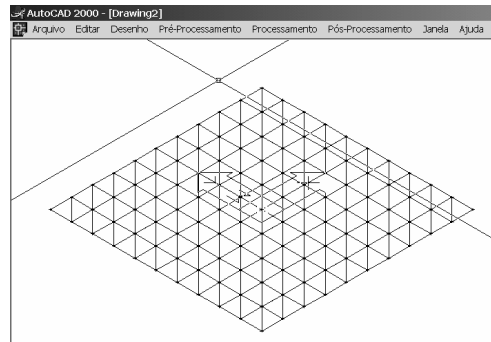


Figure 2: Discretized Model

Another step in StructCAD pre-processing is related to the definition of structural element type and element connectivity. These procedures can be done by selecting the StructCAD option menu “Pré-processamento->Define e Numera a Estrutura” (see Figura 3a) and by selecting the dialog Box “Tipo” (see Figura 3b).

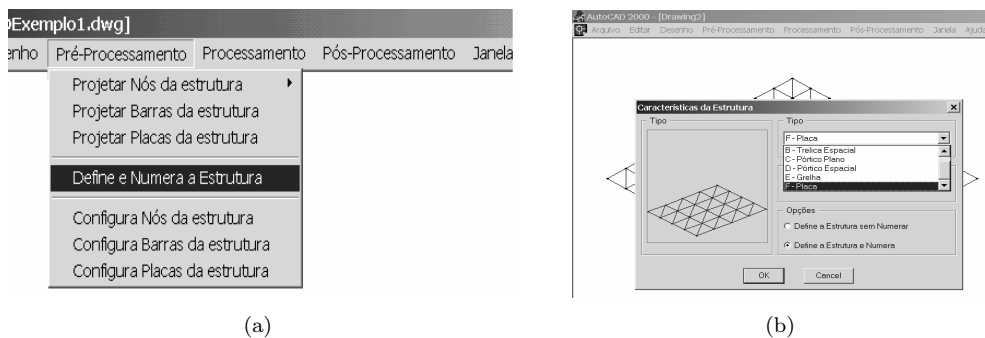


Figure 3: (a) “Define estrutura” function; (b) “Tipo” function.

3 Processing stage

The processing is usually defined as a stage of the analysis in which main calculations (such as elemental stiffness matrix evaluation, structural stiffness matrix assembling, nodal equivalent force vector evaluation, algebraic system solution, etc) are done. In present paper, the bending plate problem is

analysed by Finite Element Method (FEM) using three flat triangular elements namely DKT (discrete Kirchhoff Theory), HSM (hybrid stress model) and DST (Discrete Shear Theory). The first two elements were developed to deal with thin bending plate problems and their formulations have been thoroughly discussed elsewhere Batoz et al.[21]. The DST element takes into account the deformation by shear force and it was developed by Batoz & Lardeur[22] based on Refined Theory of Plate. The final nodal variables used in the formulation of three elements are represented by two rotations and one translation in each corner of the triangle as shown in Fig. 4.

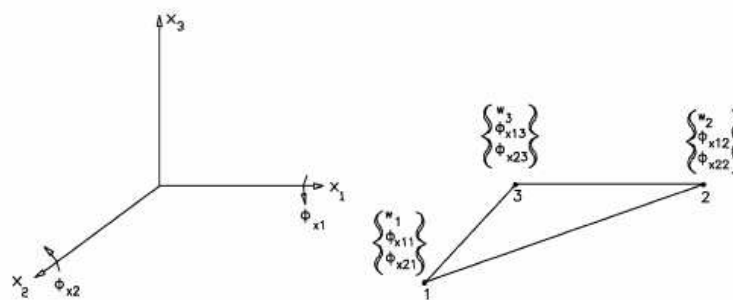


Figure 4: Nodal quantities of HSM, DKT and DST elements

The bending plate analysis is done using the CaeFep++ library, in which have functions and classes implemented in oriented object language C++ and they are compiled as ObjectARX program, whose it can run from a menu of the StructCAD. For sake of conciness, only principal classes and functions of the CaeFep++ library are described next.

The class diagram of CaeFep++ (Computer Aided Engineer for Finite Element Plate) library is shown in Figure 5. It should be noted that the CaeFep++ has functions and classes associated with three plate bending finite elements namely DKT, HSM and DST.

For sake of conciseness, only fields and methods related to the classes CaeFepDST and CaeStructDST are respectively shown in Table 2 and Table 3.

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 ObjectARX library and CaeFep++. 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 CaeFep++ to perform the structural calculations. It should be noted that CaeFep++ must be compiled in conjunction with ObjectARX library in order to create the function “ solver”, who after compilation process, it is stored in a file called “Aeplaca.arx”. Then, for the structural tool to be run from the StructCAD, the file “Aeplaca.arx” must to be loaded to it. It should be noted that at moment of loading “Aeplaca.arx” file, the method “entrypoint” of the objectARX library is

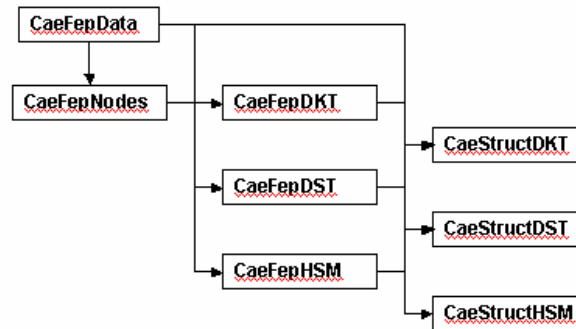


Figure 5: CaeFep++ class diagram

Table 2: The CaeFepDST class

Class	::CaeFepDST
Object	void CalculaAreaDST(int NumDst)
Object	void GeraGeometriaDST (int NumDst)
Object	void GeraIncidenciaDSTs ()
Object	void GeraMatrizRigidezLocalDST (int NumDst)

Table 3: The CaeStructDST class

Class	::CaeStructDST
Object	void AplicaCondicaoDeContorno ()
Object	void EliminatorDeGauss ()
Object	void EsforcosNoDST ()
Object	void GerarMatrizDeRigidezDaEstrutura ()
Object	void GerarVetorForcasNodais ()
Object	void ResolveEstrutura ()

called in order to become accessible the function “solver” to Autocad environment. The source code of “entrypoint” function can be found in ref.[19].

Form a point of view of user, all processing steps are done when he selects the menu option “processamento -> Calcular Estrutura” (see figure 6a). It should be noted that processing stage is done

without quitting customized Autocad environment StuctCAD. In addition, a warning flag is displayed to the user as soon as processing is done (see figure 6b).

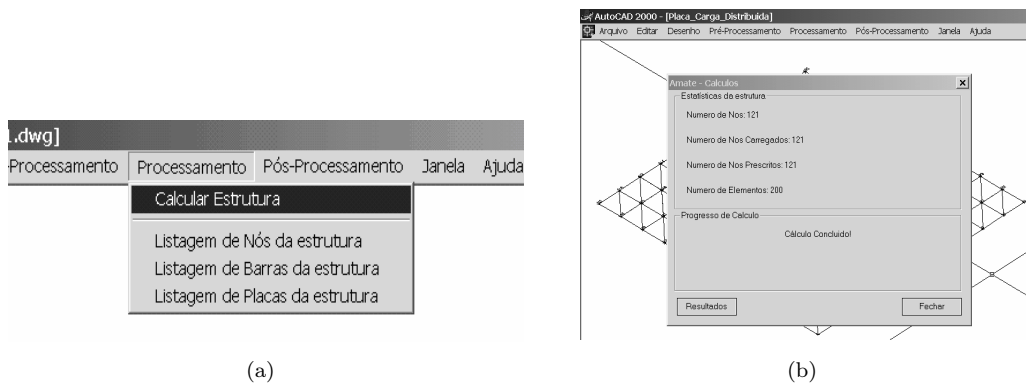


Figure 6: (a) “Calcular Estrutura” function; (b) Warning flag

4 Post processing

The Post-processing is usually defined as a stage of the analysis in which the results are shown. In StructCAD the results can be accessed by selecting the menu option “ Pós-Processamento -> Esforços e Deslocamentos da Estrutura” (see Figure 7a) and the values are shown in dialog box (see Figure 7b)

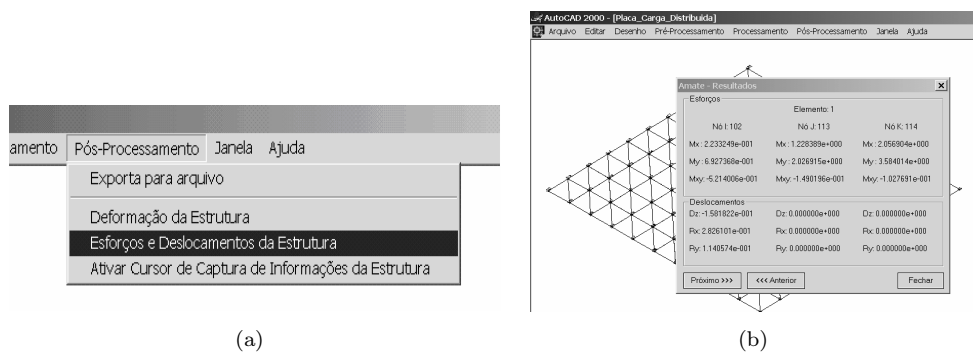


Figure 7: (a) “Esforços e Deslocamentos da Estrutura” function; (b) Dialog Box of results

It should be noted that buttons “>>> Próximo” and “<<< Anterior” provide the result selection for a next or previous element to be shown in dialog Box (see Figure 7b).

The StrucCAD provides a graphical output for displacements by selection of menu option “ Pós-Processamento-> Deformação da Estrutura” (see figure 8a) and the deformed structure form is shown in Figure 8b.

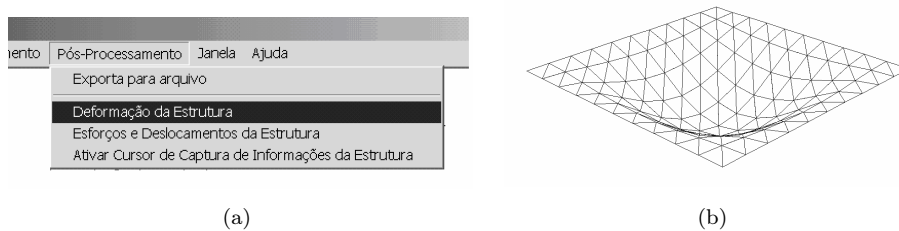


Figure 8: (a) “Deformação da estrutura” function; (b) Deformed shape.

5 Numerical analysis

In this section a simply supported square plate under concentrated loading is analysed. The plate has unitary side and Young modulus $E = 1092MPa$ and Poisson ratio $\nu = 0.3$ for mechanical properties and a concentrated load $P = 1kN$ is applied at the plate centre. The plate discretization and node numbering is shown in Fig. 9.

For concentrated load at node 41, the transverse displacements w and bending moments m_x along direction x and central line of plate for different span/thickness ratios are shown in Tables 4 and 5.

It should be noted that in Classical Plate Theory the bending moments are insensitive to flexural rigidity $D = Et^3/[12(1 - \nu^2)]$, therefore in the Table 5 only DKT and HSM bending moment values with respect to span/thickness ratio ($L/t = 10$) are shown.

In contrast, the DST results are changed for both displacements and bending moments when the L/t ratio is modified.

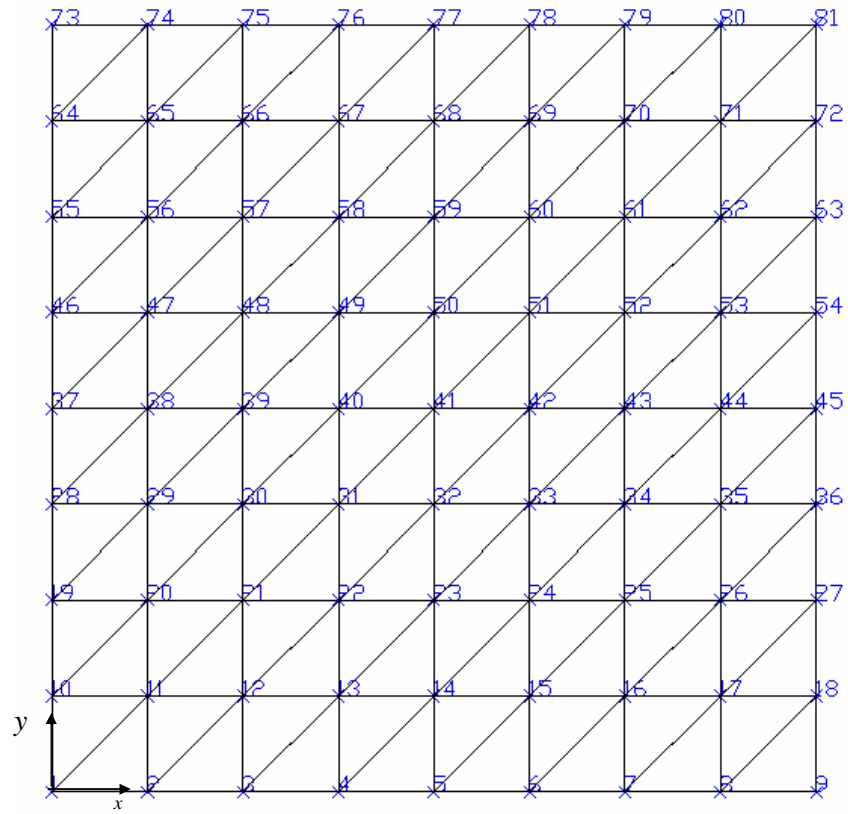


Figure 9: Plate discretization

Table 4: Transverse displacements along direction x (central line of plate)

L/t	Node	Transverse displacements (m)		
		HSM	DKT	DST
10	37	.0000E+00	.0000E+00	.0000E+00
	38	.3658E-04	.3681E-04	.3842E-04
	39	.7100E-04	.7169E-04	.7522E-04
	40	.1001E-03	.1013E-03	.1081E-03
	41	.1139E-03	.1164E-03	.1302E-03
30	37	.0000E+00	.0000E+00	.0000E+00
	38	.9879E-03	.9941E-03	.9989E-03
	39	.1917E-02	.1936E-02	.1946E-02
	40	.2702E-02	.2736E-02	.2755E-02
	41	.3077E-02	.3145E-02	.3184E-02
50	37	.0000E+00	.0000E+00	.0000E+00
	38	.4572E-02	.4601E-02	.4609E-02
	39	.8874E-02	.8961E-02	.8977E-02
	40	.1251E-01	.1266E-01	.1269E-01
	41	.1424E-01	.1455E-01	.1462E-01
70	37	.0000E+00	.0000E+00	.0000E+00
	38	.1254E-01	.1262E-01	.1264E-01
	39	.2435E-01	.2459E-01	.2461E-01
	40	.3432E-01	.3474E-01	.3478E-01
	41	.3908E-01	.3994E-01	.4002E-01

6 Conclusions

In this paper a customized Autocad environment called StructCAD was presented. The main attractive feature of this structural analysis for three bending plate finite elements, in which all stages of analysis namely pre-processing, processing and post-processing are completely integrated within a single analysis environment.

Table 5: Bending moments along direction x (central line of plate)

		Bending moments (kNm)		
L/t	Node	HSM	DKT	DST
10	37	-.3263E-03	.6069E-02	-.2173E-02
	38	.3135E-01	.2814E-01	.3231E-01
	39	.5982E-01	.5904E-01	.6300E-01
	40	.1282E+00	.1228E+00	.1307E+00
	41	.3049E+00	.3266E+00	.3202E+00
	Node	DST (L/t = 30)	DST (L/t = 50)	DST (L/t = 70)
	37	.2804E-02	.4590E-02	.5256E-02
	38	.2980E-01	.2890E-01	.2856E-01
	39	.6042E-01	.5964E-01	.5937E-01
	40	.1257E+00	.1241E+00	.1235E+00
	41	.3241E+00	.3255E+00	.3260E+00

Acknowledgements

The Scientific Initiation Scholarships (PIBIC) sponsored by Brazilian National Council of Research (CNPq) are gratefully acknowledged by first two authors.

References

- [1] www.tecgraf.puc-rio.br/ftool, Accessed in 4/20/2006.
- [2] www.set.eesc.usp.br/acadframe, Accessed in 4/20/2006.
- [3] Ramalho, M.A., *Sistema para análise de estruturas considerando interação com o meio elástico*. Master's thesis, Escola de Engenharia de São Carlos, Universidade de São Paulo, 1990.
- [4] www.ansys.com, Accessed in 4/20/2006.
- [5] www.abaqus.com, Accessed in 4/20/2006.
- [6] www.lusas.com, Accessed in 4/20/2006.
- [7] *CYPECAD - Manual do usuário*. São Paulo, 1997.
- [8] AltoQI Tecnologia em Informática Ltda, Florianópolis/SC, *AltoQi Eberick versão 2002*, 2002.
- [9] Sistrut Software e Tecnologia S/C, *Sistrut 4.0 - Manual do usuário*, 1992.
- [10] TQS Informática Ltda, São Paulo, *TQS - Manual do usuário*, 1997.
- [11] *Strad-Manual do usuário*, 2003.
- [12] CSI - Computer and Structures, Inc., Berkeley/CA, *SAP 2000 - Integrated finite elements analysis and design of Structures*, 1997.
- [13] Peixoto, R.H., *Um sistema gráfico-iterativo para análise e dimensionamento de pórticos planos de aço*.

- Master's thesis, PUC-Rio, 2001.
- [14] AUTODESK INC, *AutoCAD 2000i - User's Guide*, 1999.
 - [15] Santos, E.S.R. & Devloo, P.R.B., Desenvolvimento de classes viga e pilar para autocad utilizando linguagem c++ e objectarx. *IX Congresso Interno de Científica da Unicamp*, 2001.
 - [16] Campagnã, B.C.D. & Menezes, F.A., Customização do autocad utilizando-se a linguagem c++ e o objectarx - dimensionamento de lajes de concreto. *IX Congresso interno de iniciação científica da Unicamp*, 2001.
 - [17] Mauad, F.H.C., Santos, E.S.R. & Devloo, P.R.B., Integração de métodos numéricos no autocad, utilizando o objectarx e o ambiente de elementos finitos. *Sul-Americanas De Engenharia Estrutural*, Universidade de Brasília - Brasil: Brasília, volume 30, 2002.
 - [18] Kramer, W. & Kramer, D., *Programando em AutoLISP*. MAKRON Books do Brasil Editora Ltda: São Paulo - SP, 1995.
 - [19] McAley, C., *Programming AutoCAD 2000 Using ObjectARX*. Autodesk Press, 2000.
 - [20] Menezes Jr., R.A., Análise da interação dos efeitos placa-pórtico em problemas de edifícios utilizando-se programação orientada ao objeto em c++. Pibic, Universidade Federal da Paraíba, João Pessoa, PB., 2006.
 - [21] Batoz, J.L., Bathe, K.J. & Ho, L.W., A study of three-node triangular plate bending elements. *Int J Numer Meth Engng*, **15**, pp. 1771–1812, 1980.
 - [22] Batoz, J.L. & Lardeur, P., A discrete shear triangular nine d.o.f. element for the analysis of thick to very thin plates. *Int J Numer Meth Engng*, **28**, pp. 533–560, 1989.

