FINITE ELEMENT TECHNIQUE FOR LIMIT ANALYSIS OF FRAME STRUCTURES ASSOCIATED TO A CAD 3D MODELER

José Ricardo Queiroz Franco, franco@dees.ufmg.br
Bruno Cesarino Soares, brunocesarino@hotmail.com
Felício Bruzzi Barros, felicio@dees.ufmg.br
Universidade Federal de Minas Gerais

Abstract. This paper presents a novel finite element technique for limit analysis of plane frames. A computational system has been constructed encompassing two modulus: a 3D CAD modeler and a FE processor for limit analysis of plane frames. An application for managing interaction between the two modulus and for automating data exchange has also been developed. Implementation of the technique algorithm required the construction of a FE code developed using the C++ programming language associated to the paradigms of OOP. The CAD system has been designed as a plug-in for the AutoCAD, using the API (Application Program Interface) ObjectARX (Autocad Runtime eXtension). Such an API consists of a set of C++ libraries, which can be used to customize AutoCAD graphical environment, allowing the creation of new entities and permitting access to native classes. The CAD application is used to execute pre-processing of the geometry and for post-processing the collapse mechanism. Finally, the solutions obtained are compared to other analytical and numerical solutions available in the literature.

Keywords: Limit Analysis, Finite Elements, Frame Structures, CAD, Object Oriented Programming (OOP)

1. INTRODUCTION

This work presents a finite element technique for the limit analysis of plane frames based on two papers by Franco and Ponter(1997) for finite element shakedown and limit analysis of axisymmetrical pressure vessels. A CAD application has been developed to automate the phases of the process, such as the pre-processing of the geometry, the graphical post-processing of the collapse mechanisms and so on. The CAD application uses C++ associated to the paradigms of the object-oriented programming (OOP) and to some software design patterns, Gamma, E. et al (1995, 2000).

An application for managing interacting data has also been developed, based on the concepts of BIM (Building Information Modeling) technologies, so that a database can store information generated in any phase of the process. Geometrical data, structural models with bars elements, nodes, materials and cross sections, collapse mechanisms models and others can be used to build information models, which serve for a variety of purposes, as in detailing, manufacturing and construction.

Upper bounds of the collapse load factor (κ_c) and the corresponding collapse mechanism are determined by applying the kinematical collapse theorem to the complete set of independent elementary mechanisms and their combinations. Bending actions concentrated at nodal points are assumed to be predominant and the behavior of the material is considered to be perfectly plastic.

The discretized structural problem is transformed into an optimization problem and solved by linear programming (LP). The structure is discretized in two-node finite elements with linear interpolation. Plastic hinges are admitted to occur at nodal points, where free rotations together with nodal displacement constitute the set of variables for the structural problem. The major set of constraint equations of the LP problem is defined by enforcing compatibility between the displacement rate fields and the plastic strain rate fields due to the formation of plastic hinges at nodal points during collapse.

The 3D CAD modeler was designed to be a plug-in for the AutoCAD, using an Application Program Interface (API) called ObjectARX (Autocad Runtime eXtension), Autodesk (2004). The API ObjectARX consists of a set of C++ libraries, that can be used to program and customize the AutoCAD graphical environment, which allows the creation of new entities and also permits access to the structure of native entities. The CAD application is used to pre-process the geometry and for post-processing the results and the collapse mechanisms.

2. FORMULATION OF THE NOVEL UPPER BOUND TECHNIQUE

Several methods for limit analysis of frames have already been proposed. In many cases the goal is to create optimized procedures to implement the method of combining mechanism (Deeks, 1996). The use of Linear Programming (LP) have had a great appeal to optimize upper and lower bound solutions as in the techniques presented by (Grierson & Gladwell, 1971) and (GriersonandAddel-Baset, 1977). The former work is limited by the need of an *apriori* procedure to separately obtain the independent mechanisms for the kinematical approach. The latter, as in most methods in the literature, presents a technique for obtaining lower bounds of the collapse load, where optimization of

the computational performance was considered, which no longer is a major issue since even PC's computers can now have high performance.

The novel technique presented here propose a simplified formulation for calculating upper bounds on the ultimate load of a plane frames without taking into account the computational effort due to the great number of variables involved in the optimization problem. The structural problem is transformed into a LP problem by discretizing the frame with a two nodes bar element as in Fig. 1.
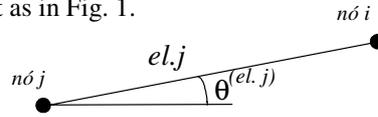

Figure 1. Two nodes bar element

The technique implemented in this work uses the upper bound theorem, in order to automate the assembling of the LP problem. Bending is assumed to occur only at nodal points of the element, where plastic hinges can form. The technique represents an extension of a Finite Element formulation proposed in two papers by Franco and Ponter (1997) for shakedown and limit analysis of axisymmetrical pressure vessels. The theory and fundaments of the technique are presented in Part I and complementary Part II add also important numerical solutions for pressure vessels.

A brief description of the upper bound formulation for the limit analysis of plane frame is given next:

The formulation seeks to finding the best mechanism corresponding to the smallest collapse load factor κ_c, among all independent and combined mechanisms.

The upper bound theorem for the general structural problem considers the velocity or displacement rates field $\dot{u}^c$ and the plastic deformation rates field $\dot{\varepsilon}^c$ during collapse. These fields are kinematically admissible and correspond to a possible collapse configuration. Eq.(1) describes the kinematical theorem for the general structural problem, which can be stated as: for a structure subjected to a set of loads defined by a load factor κ at collapse, which corresponds to any assumed mechanism, the work done by κ must be greater than or equal to the energy dissipation during collapse, ie, κ is either greater than o equal to the actual collapse load factor κ_c.

$$\kappa \int_S p\dot{u}^c \, dS \geq \int_V \sigma^c \dot{\varepsilon}^c \, dV \quad or \quad \kappa \geq \frac{\int_V \sigma^c \dot{\varepsilon}^c dV}{\int_S p\dot{u}^c dS} = \frac{W_i}{W_e} \tag{1}$$

where:

κ = an upper bound on the actual load factor

p = characteristic load

$\dot{u}^c$ = displacement rates at collapse

σ_c = stress at collapse

$\dot{\varepsilon}^c$ = plastic deformation rates at collapse

W_i = internal energy dissipation

W_e = external work

The kinematical collapse theorem equation for plane frames is more simply described by the inequality shown in Eq.(2), where the integrals in Eq. (1) can be transformed into sums. The external work is done by nodal loads and the internal energy dissipation is due to plastic hinges formation, when the plastic moment (M_p^i) is reached at nodal points producing free hinge rotation rates ($\dot{\theta}^c$) during the formation of the mechanism. For plane frames the variables for the optimization problem are the nodal displacements rates $\dot{u}^c$ and the free hinge rotation rates ($\dot{\theta}^c$).

$$\kappa \geq \frac{\sum_{i=1}^{m} M_{p,i} \dot{\theta}_i^c}{\sum_{i=1}^{n} p_i \dot{u}_i^c} = \frac{W_i}{W_e} \tag{2}$$

where: n = degrees of fredom

m = number of nodal points where hinge can form

M_p^i = nodal plastic bending moment

$\dot{\theta}^c$ = free hinge rotation rates

2.1. The General Linear Programming (LP) Problem

The standard form of the linear programming problem, Sousa, et al., (2005), is formulated to maximize (minimize) a generic objective function (*f*) described as:

$$f = \sum_{j=1}^{n} c_j x_j \tag{3}$$

subjected to the following restrictions:

$a_{ij}x_j \le b_j$
$x_j \ge 0$ *where*:
$i = 1, 2, \dots, m = m_1 + m_2$
$j = 1, 2, \dots, n$
$n = number\ of\ variables$
$m_1 = number\ of\ restrictions\ inequations$
$m_2 = number\ of\ restrictions\ equations$

The LP problem can be stated in matrix form as:

$$Maximize\ f = \{c\}^T \{x\} \quad subjected\ to\ [A]\{x\} \ge \{B\} \tag{4}$$

Vector $\{c\}$ stores the coefficients of the objective function f and vector $\{x\}$ contains the problem unknowns. Matrix $[A]$ and vector $\{B\}$ compose a system of equations and inequations that constrains the LP problem.

Among several methods, the Simplex Method was adopted, which requires the variables x_j to be positives. Since, in general, such restrictions do not apply to engineering problems the following algebraic transformations are applied, Eq. (5):

$$x_j = x_j^+ - x_j^- \tag{5}$$

where

x_j^+ *and* x_j^- *are positive values that satisfy the conditions*:

$x_j^+ = x_j$ *and* $x_j^- = 0$, for $x_j \ge 0$

$x_j^- = -x_j$ *and* $x_j^+ = 0$, for $x_j \le 0$

2.2. The structural problem for plane frames reduced to a LP problem

Eq.(2) shows that the relative values of W_i and W_e are significant but not the absolute values. Therefore if W_e is assigned the unity value, which just scales the relative size of the mechanism, the structural problem can be reduced to a simplified optimization problem formulated as follows:

$$Minimize\ \kappa = \{c\}^T \{x\} = \sum_{i=1}^{s} M_{p,i}\dot{\theta}_i \tag{6}$$

$$subjected\ to\ [A]\{x\} \ge \{B\} \begin{cases} \sum_{i=1}^{e}(\dot{e}_i = 0)\ (inextensibility\ conditions) \\[2mm] \sum_{i=1}^{s}(\dot{\theta}_i - \dot{\theta}_i^N = \dot{w}_i')\ (nodal\ rotation\ compatibility\ conditions) \\[2mm] \sum_{i=1}^{r}(\dot{u}_i = 0)\ (boundary\ conditions) \\[2mm] \sum_{i=1}^{g} p_i\dot{u}_i = 1\ (general\ constraint\ equation\ involving\ all\ elements) \end{cases} \tag{7}$$

where:
κ = load factor
s = number of critical sections where plastic hinge can occur
e = number of elements
r = number of boundary conditions
g = number of degrees of freedom
p = nodal load
$\dot{u}$ = nodal displacement rate at collapse
M_p = plastic moment

$\dot{\theta}$ = relative rotation rate of the critical section at collapse

$\dot{\theta}^N$ = nodal rotation rate due to joint mechanism

$\dot{e} = \dot{u}_f - \dot{u}_i$ = elongation rate of the finite element

$\dot{w}^{'} = \dfrac{d\dot{w}}{dx}$ = nodal rotation rate of the finite element

Reduction of the structural problem into a LP problem requires the definition of the objective function Eq. (6) and the restriction equations, Eqs. (7). A general constraint equation is defined by assigning the unity value to the external work W_e. Boundary conditions and compatibility between free hinge rotation rates ($\dot{\theta}^c$) and the nodal displacements rates $\dot{u}^c$ complete the set of constraint equations of the LP problem.

The objective function can now be written as in Eq. (8), where the coefficients c_i for "g" nodal displacements rates $\dot{u}_i$ are null and the coefficients for the "s" free hinge nodal rotation rates $\dot{\theta}_i$ are the element plastic bending moments M_{pi}.

$$f = \kappa = \sum_{i=1}^{g}(c_i)\dot{u}_i + \sum_{i=1}^{s}(c_i)\dot{\theta}_i \ \ \text{or}$$

$$f = \kappa = \sum_{i=1}^{g}(0)\dot{u}_i + \sum_{i=1}^{s}(M_{p,i})\dot{\theta}_i$$

In terms of global displacement rates and non negative variables the cost function gives

$$f = \kappa = \sum_{i=1}^{n}(0)(\dot{U}_i^+ + \dot{U}_i^- + \dot{W}_i^+ + \dot{W}_i^-) + \sum_{i=1}^{e}(M_{p,i})(\dot{\theta}_{1,i}^+ + \dot{\theta}_{1,i}^- + \dot{\theta}_{2,i}^+ + \dot{\theta}_{2,i}^-) \tag{8}$$

Eqs. (7) show that the problem is subjected to no constraint inequation, ie, $m_1 = 0$ and $m_1 > 0$ in Eqs. (3a).

The general constraint equation involving all elements $\sum_{i=1}^{g} p_i \dot{u}_i = 1$ in Eqs. (7) represents the work done by the nodal equivalent loads during collapse. For simplicity, such external work is assigned the unity value. Eq. (9) illustrate the format of the general constraint equation for the following applied load components at node 1; (10) unities of force in the global direction x and (-5) unities of force in the global direction y.

$$(10)U_1^+ + (-10)U_1^- + (-5)W_1^+ + (5)W_1^- + ... + (0)\theta_1^+ + (0)\theta_1^- + ... + (0)\theta_n^+ + (0)\theta_n^- = 1 \tag{9}$$

The boundary conditions $\sum_{i=1}^{r}(\dot{u}_i = 0)$ define a subset of constraint equations related to nodal displacements. For example, a translation restriction at node i in the x direction implies in two constraint equations for the problem global variables; $U_i^+ = 0$ and $U_i^- = 0$.

Boundary conditions related to free nodal rotation can also be imposed to a pinned joint. A joint free to rotate can be considered by assuming that no plastic hinge occurs at this node and therefore it does not contribute to the energy dissipation during collapse. To illustrate such a case, consider node 2 of a frame as pinned, so that vector $\{c\}$ would have the following configuration:

$$\{c\} = \begin{bmatrix} 0\,0\,0...M_{p1}\,M_{p1}\,0\,0\,M_{p3}\,M_{p3}...M_{pn}\,M_{pn} \end{bmatrix}^T \tag{10}$$

The major set of constraint equation for the LP problem is derived from the compatibility between the displacement rates field $\dot{u}^c$ and the plastic deformation rates field $\dot{\varepsilon}^c$ during collapse. Since the effects of normal forces are neglected, the axial deformation rate field, $\dot{\varepsilon}_{xi}^c$ for a general finite element i with linear interpolation, Fig. 2, constitute a group of constraint equations, which can be set to zero as follows:

$$\varepsilon_{xi} = \frac{u_j^i - u_i^i}{L_i} = 0 \,(\textit{axial strain on element i}) \tag{11}$$

where: $L_i = length\ of\ element\ i$

$u^i{}_i$ = axial displacement of node i

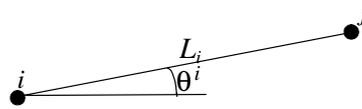$u^i{}_j$ = axial displacement of node j

Figure 2. General finite element i with linear interpolation

As an example consider the horizontal element *"1"* oriented from node 1 to node 2 in the discretized structure, Fig.3. The restraint equation representing the compatibility between axial deformation and the global displacements at the element nodes can be written as:

$$(1/L_1)U_1^+ + (-1/L_1)U_1^- + (0)W_1^+ + (0)W_1^- + ... + (0)\theta_i^+ + (0)\theta_i^- + ... + (0)\theta_n^+ + (0)\theta_n^- = 0 \qquad (12)$$

The major set of nodal restraint equations relates the displacement rates field $\dot{u}^c$ and the plastic deformation rates field $\dot{\varepsilon}^c$ during the formation of collapse mechanisms. Such restrictions are defined in terms of nodal rotations. Since several elements can concur in a single node and each element has an independent description for the node rotation, it is necessary to make those descriptions compatible. Rotations along each element can be geometrically defined through nodal displacements. For example, the two nodes element *"1"* in Fig. (3), behaves as a rigid bar, in which a constant rotation along its length can be describe by the Eq. (13).

$$\theta_1 = \frac{w_2 - w_1}{L_1} \qquad (13)$$

where:
  L_1 = length of element 1
  w_1 = local displacement at the initial node of element 1
  w_2 = local displacement at the end node of element 1
  θ_1 = element 1 nodal rotation

The global rotation of a general node i, θ_i^G, can be defined as the sum of the contribution $\theta_i^{(el.j)}$ of each element intercepting the node. This restriction equation must also allow the independent node mechanism, as defined by Neal (1977), to occur. For implementation reasons an additional variable θ_i^{node} had to be created for each node to represent the independent node rotations, which are not associated to the external work W_e. Therefore, these node rotations are not included either in the composition of the cost function. These additional variables can then composed the constraint equations, which represent the contribution of each element to the node rotation as in Fig. 3 for the three elements joint at node 2.
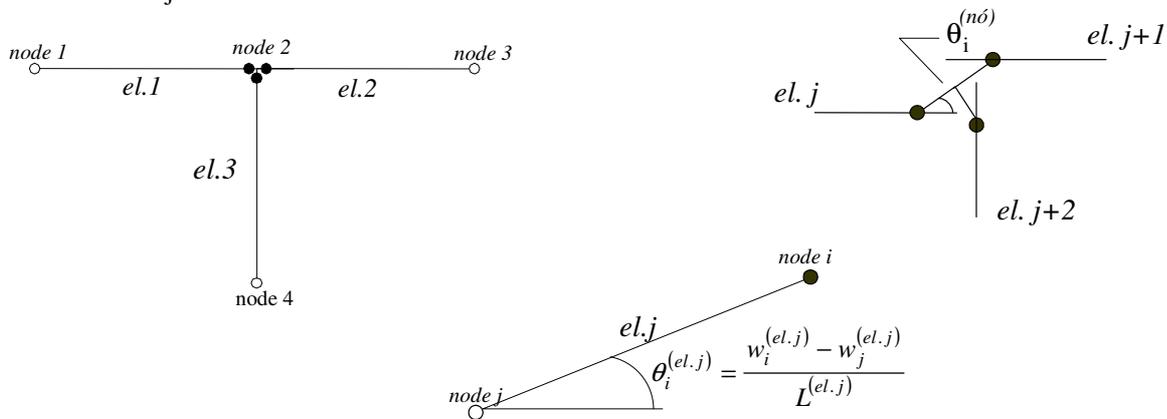

Figure 3. Three elements joint - Node 2

Equations (14) define the element individual contribution for the rotation at node 2, $\theta_2^{(el.j)}$, which sum together to give the global rotation θ_2^G.

$$\left.\begin{array}{l}\theta_2^{(el.1)} = \dfrac{w_2^{(el.1)} - w_1^{(el.1)}}{L^{(el.1)}} + \theta_2^{(nó)} \\[2em] \theta_2^{(el.2)} = \dfrac{w_2^{(el.2)} - w_1^{(el.2)}}{L^{(el.2)}} + \theta_2^{(nó)} \\[2em] \theta_2^{(el.3)} = \dfrac{w_2^{(el.3)} - w_1^{(el.3)}}{L^{(el.3)}} + \theta_2^{(nó)} \end{array}\right\} \theta_2^{(G)} = \theta_2^{(el.1)} + \theta_2^{(el.2)} + \theta_2^{(el.3)} \tag{14}$$

where:
$L^{(el.j)}$ = length of element j
$w_1^{(el.j)}$ = local displacement at the initial node of element j
$w_2^{(el.j)}$ = local displacement at the end node of element j
$\theta_i^{(el.j)}$ = end node i rotation of element j
$\theta_i^{node)}$ = independent rotations at node i
$\theta_i^{(G)}$ = global rotation at node i

Equation (14) defines a nodal constraint for the LP problem, representing a node rotation, which is described by the sum of the rotation at the extreme of each element concurring at that node together with the independent nodal rotation. The possibility of the formation of an independent node mechanism is then contemplated by the LP procedures, which determine the global nodal rotation that minimizes the dissipation energy during collapse.

Finally, with the addition of new variables representing nodal independent rotations, vectors $\{c\}$ and $\{x\}$ take, respectively, the forms:

$$\{c\} = \begin{bmatrix} 0\,0\,0...M_{p1}\,M_{p1}M_{p2}\,M_{p2}\,...M_{pn}\,M_{pn}\,0\,0...0\,0 \end{bmatrix}^T \tag{15}$$

$$\{x\} = \begin{bmatrix} U_1^+\,U_1^-\,W_1^+\,W_1^-\,U_2^+\,U_2^-\,W_2^+\,W_2^-\,...\theta_1^+\,\theta_1^-\,...\theta_n^+\,\theta_n^-\,\theta^{(node\,1)+}\,\theta^{(node\,1)-}\,...\theta^{(node\,n)+}\,\theta^{(node\,n)-} \end{bmatrix}^T \tag{16}$$

3. A THREE-DIMENSIONAL CAD MODELER

The CAD modeler works as a graphical interface to the limit analysis code processor to perform pre and post processing of the geometry and collapse mechanism of frames structures in 3D. The CAD system serves to automate the modeling, the processing of limit analysis of frame structures and the graphical post-processing of the results. AutoCAD was used as the graphical platform and the CAD application works as a plug-in, which was implemented using the AutoCAD programming interface (API), ObjectARX. This API consists of a set of libraries in C++, which allows personalization and customization of AutoCAD graphical environment. Such technology also allows the creation of new entities, including access to and extension of the native classes. The new entities, objects and classes allow the frame structures to be represented by basic elements such as: `nodes, bars, materials, and cross sections`.

The new and the customized entities, objects and classes can inherit native classes functionalities and can also define their own functionalities to become specialized in a specific job. They can also store the necessary data for modeling structural elements and attributes. The classes hierarchy provides essential information for the 3D graphical representation of structural elements. The basic functionality structural of an application developed using the ObjectARX API, according to Malard (1998), is schematized in Fig. 4
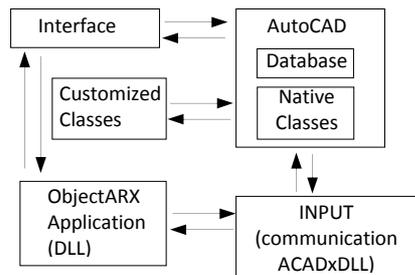
Figure 4. ObjectARX Application Structure

3.1 Basic Functionalities of the Structural Modeler

Previous works have been developed by Malard (1998) and Hütner (1998), which form the basis for the present structural modeler. The modeler runs by loading AutoCAD with three DLL *(Dynamic Link Libraries),* files with extension (.ARX and (.DBX). The file "`CdTcAcadUtils.arx`" is a basic modulus, which all the other *DLL* depend upon and contain generic functionalities to access the graphical environment. The modulus containing the customized classes and the application global variables is run by loading the file "`CdTcModelador.dbx`" to AutoCAD. The application group of commands for creating entities, configuring drawing parameters, editing and managing data and other commands are added to the application environment by loading the modulus "`CdTcModAcad.arx`". Figure 5 illustrate the content of a pull down menu, where several commands created for the application have been joined together, which are used to edit and manage the model data.
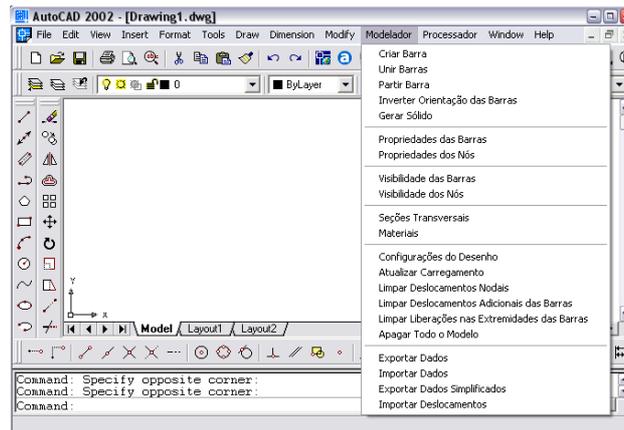

Figure 5. Menu do modelador estrutural.

The instances of the customized classes bars and nodes are defined as "entities" because they have graphical representation. On the other hand, classes that perform control, such as numbering, layers information storage, drawing parameters, cross sections and materials, are instances denominated "objects", since they have no graphical representation.

Editing the application model follows the usual procedure of AutoCAD native entities, such as lines, dots and circles. Nevertheless, the new and the customized entities require consistency checks to verify that the structural model can be processed to perform a structural analysis. It is imperative to have bars and nodes numbers under control. For example, if a bar is removed from the model, the bar number must be preserved in the database to be used when a new bar is created. Controlling the consistency of data is also important when creating a new bar. If the bar is isolated, two new nodes are created at the ends of the bar. Differently, if one create a new bar from the extreme of another existing bar it is necessary to associate the new bar to the existing node, so that consistency of the structural data model is preserved. Another important consistency check is required when intermediate nodes need to be created at points where bars cross. In this case discretizing the bar into two bars is also an option. The application requires other more complex consistency control verifications when editing the structural design and drawings using the CAD graphical platform environment commands *(AutoCAD commands)*, such as *copy, mirror, rotate, array* and other. These resources and functionalities differentiate the CAD application here from the simple use of a CAD environment to produce drawings of a structural project.

Figure 6 shows the result of a wireframe model of a plane frame containing numbered bars and nodes, the distributed loading and the clamped supports. Configuring the visualization of the structural elements can also be control by the application, which allows exhibiting the model partially or whole and in different scales. For example, visualization of bars and nodes can be configured individually or for the model as a whole. Since each graphical element is drawn is a specific layer, it is possible to modify the element style, to freeze or to turn off the layer, which are just native AutoCAD options. The size of the texts identifying attributes of bars and nodes, such as their numbers, can also be altered by customized commands. Bars are, for example, represented by a standard axis, nevertheless, their attributes and additional data can have the graphical representation visible or frozen. Depending on what is desired to be visible, texts, numbers, labels, displaced axes, local coordinate systems, applied loads representation and solids associated to cross sections may be set to show up in the model. Nodes, for instance, are derived from the AutoCAD native entity point; therefore maintain all the attributes and visual configurations.

The model data are automatically saved with the drawing file (*.dwg)* and are automatically recovered when the file is opened again, even if the application modeler is not loaded in the CAD environment. In this case it is only possible to visualize de model without editing. All functionality is recovered when the application is loaded. Commands to import and export data from the model to a text (ASCII) file has also been constructed.
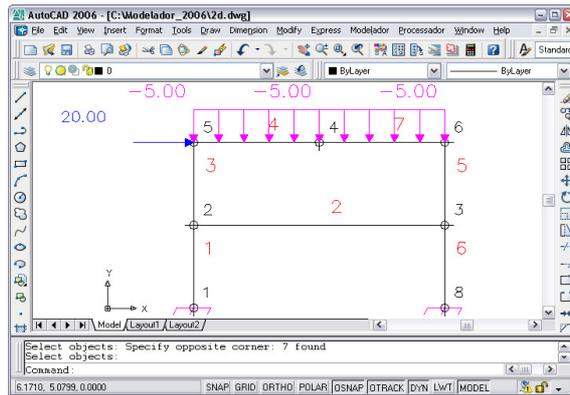
Figure 6. Plane Frame Model

Figure 7 shows an example of the application representation of a 3D model spatial frame, which illustrates solids I shape profiles vesting the wireframe model.
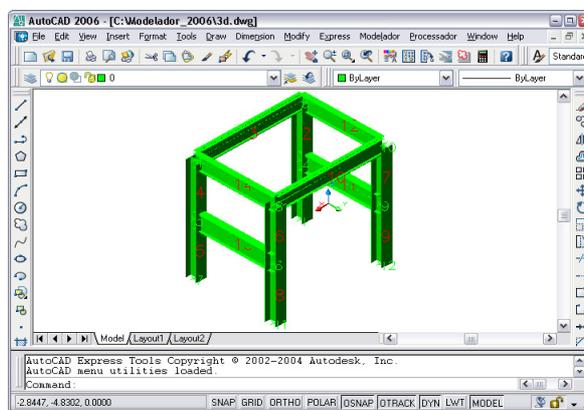

Figure 7. 3D Solid Model

3.2 Structural Modeler Post-Processing

The structural modeler works also as a post-processor providing specific functionalities to graphically display the collapse mechanisms resulting from the limit analysis. Entities and objects have been personalized by additional attributes and new commands to meet the functionalities requirements for the post-processing.

During collapse the formation of plastic hinges at nodal points can be graphically represented by free rotations, as shown in Fig. 8.
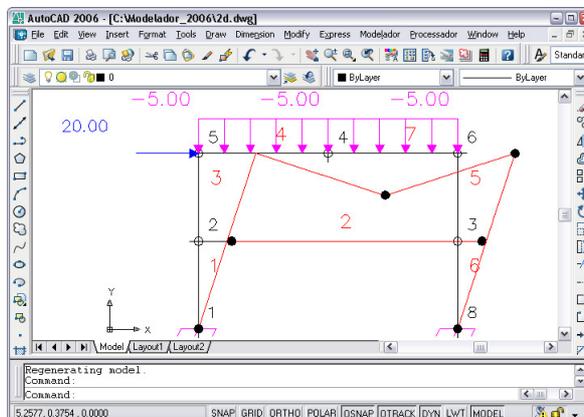

Figure 8. Collapse Mechanism with Nodal Plastic Hinges

The deformed wireframe structure model shown in Fig. 8, allows the CAD modeler to extrude cross sections shapes along the displaced axes to transform it into solids profiles as shown in Fig. 9.
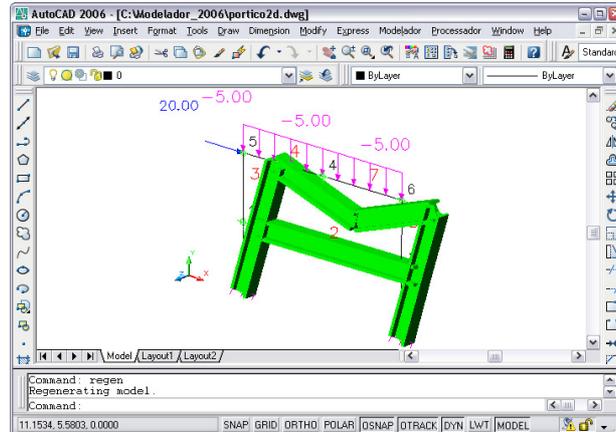

Figure 9. Solid Representation of the Collapse Mechanism

4. EXAMPLE

The example chosen to validate this work is the plane frame, Fig. 10, analyzed by Horne (1971), which has been model by the CAD application and integrated with the processor developed to implement the technique proposed here. The solution obtained is compared with that produced analytically by Horne (1971). The two-storey and two spans plane frame model, shown in Fig. 10, is discretized in 14 elements. Elements and nodes numbers are also indicated in Fig. 10. The geometry illustrates the clamped supports, the beams and columns dimensions and the applied loads at nodal points. The elements plastic moments are given in Table 1.

Table 1. Plastic Moments for Elements 1-14

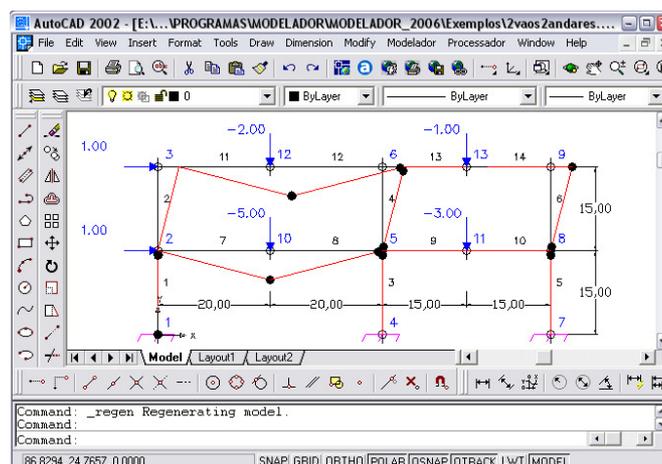| Element	1	2	3	4	5	6	7	8	9	10	11	12	13	14
Plastic Moment (kN.m)	50	60	100	40	50	20	180	180	160	160	80	80	60	60

Figure 10. Two-storey and two spans plane frame model

Figure 10 also illustrates the post-processing of the CAD modeler, where the collapse mechanism and the plastic hinges formed at nodal points are shown.

Table 4 shows coincident upper bound values on the limit load factor obtained analytically by Horne (1971) and numerically by the novel technique presented here.

Table 2. Results.

Upper Bound	(Horne, 1971)	Novel Technique
κ	6,0	6,0

5. CONCLUDING REMARKS

A novel finite element technique for the limit analysis of frame type structures integrated with a CAD modeler provided a strong tool to model and to analyze complex structures without great computational efforts. The simplified element model adopted produced solutions consistent with those obtained in the literature. The system code has been developed using the paradigm of object oriented programming, which allowed the implementation of the technique in independent and extensible modulus. The use of object-oriented programming and software design patterns brought important contributions to the development of computer codes. The novel technique can now be easily improved using a more robust element with better constitutive laws.

The CAD 3D modeler can also serve to interface several other structural analysis applications.

6. ACKNOWLEDGEMENTS

7. REFERENCES

Autodesk, Inc., 2004. ObjectARX Developer's Guide. Manual on line.

Deeks, A.J., 1996. Automatic computation of collapse loads for frames. *Computers & Structures*, vol. 60, n. 3, pp. 391–402.

Franco, J.R.Q., Ponter, A.R.S, 1997. A General Approximate Technique for the Finite Element Shakedown and Limit Analysis of Axisymmetrical Shells – Part I – Theory and Fundamental Relations, *International Journal for Numerical Methods in Engineering*, Vol. 40, 3495-3513 (1997)

Franco, J.R.Q., Ponter, A.R.S, 1997. A General Approximate Technique for the Finite Element Shakedown and Limit Analysis of Axisymmetrical Shells – Part II – Numerical Applications, *International Journal for Numerical Methods in Engineering*, Vol. 40, 3515-3536 (1997)

Gamma E., R. Helm, R. Johnson and J. Vlissides, *Design patterns: Elements of reusable object-oriented software*, Addison Wesley, Reading, Massachusetts, (1995).

Gamma, E. et al, 2000. *Padrões de Projeto: Soluções Reutilizáveis de Software Orientado a Objetos*, Bookman, Trad.: Salgado, Luiz. A. Meirelles.

Grierson, D.E., Gladwell, G.M.L., 1971. Collapse load analysis using linear programming. *Journal of Structural Division ASCE 97*, pp. 1561–1573.

Grierson, D.E., Abdel-Baset, S.B., 1977. Plastic analysis under combined stresses. *Journal of the Engineering Mechanics Division*, pp. 837–854.

Horne, M.R., 1971. *Plastic Theory of Structures*. Pergamon International Library.

Hütner, A., 1998. *Modelador 3D para Estruturas Via CAD – Engenharia/Visualização/Banco de Dados. Belo Horizonte*. Dissertação de mestrado, Escola de Engenharia da Universidade Federal de Minas Gerais (UFMG).

Malard, F. P., 1998. *Modelador 3D para Estruturas Via CAD – Arquitetura do Sistema/Banco de Dados/Visualização*. Belo Horizonte, 208f. Dissertação de mestrado, Escola de Engenharia da UFMG.

Neal, B.G., 1977. *The Plastic Methods of Structural Analysis*. Chapman and Hall.

Sousa R.S., Silva C.T.L. and Arenales M.N., "Métodos do tipo dual simplex para problemas de otimização linear canalizados", *Pesquisa Operacional*, Vol.25, n. 3, pp. 349-382, (2005).