THE IMPLEMENTATION OF THE DIRECT STIFFNESS METHOD IN AUTOCAD

Symeonidis Orestis
Msc Student in Applied Informatics at
University of Macedonia,
47 T. Oikonomidi, Thessaloniki, Greece

Samaras Nikolaos
Assistant Professor at University of Macedonia
Department of Applied Informatics
156 Egnatia str. ,Thessaloniki, Greece

ABSTRACT

This paper demonstrates an implemented add-in for AutoCAD which can be used as a structural analysis tool for two dimensional structures under static loading conditions. The method, which is based on, is the direct stiffness method, a method well known to all scientists and engineers involved in structural analysis. The main body of the paper consists of three parts. The first contains some introductory terms and concepts required for the understanding of the method, the second describes the method and the third demonstrates the functionality of the add-in. The paper concludes with some points regarding why the choice of implementing the method in AutoCAD was correct.

KEYWORDS

Direct Stiffness Method, Structural analysis, AutoCAD

1. INTRODUCTION

The analysis of structures is of great importance since it is an estimation of the true behavior of the building under different loading conditions. One of the most known and used methods of structural analysis is the direct stiffness method or in short D.S.M. whose final form dates back to 1959 [Turner M. J.,1959]. However it must be mentioned that other previous works contributed to great extends for Turner to present the method in its known form such as the work of John H. Argyris [Argyris J. H. et al, 1960] who systematized the concept of assembly of structural system equations from elemental components [Fellipa, Carlos A.,2001]. This method is considered to be particularly suited for implementation in the computer and that’s the reason most commercial FEM programs implement it. Our attempt was to implement the method in AutoCAD providing a tool for static analysis of two dimensional structures.

2. METHOD DESCRIPTION AND IMPLEMENTATION

It is important for the reader first to familiarize himself with some basic terms and concepts of structural analysis before reading the description and implementation of the method. Subsection 2.1 tries to explain them in a plain straightforward manner without requiring any special knowledge on the subject. Any reader familiar with them can proceed reading the description or even the implementation of the method. However he must take into consideration the coordinate systems defined in the first section and the fact that both the description and the implementation refers to two dimensional frames only.

2.1 Introductory terms and concepts

The ideal model of a structure is the first concept one has to bear in mind in his attempt to analyze the structure. This model should behave as close to the actual structure's behavior as the requirements of analysis dictate, while omitting details that have an insignificant factor of influence towards this direction. The model used in structural analysis is usually an assembly of line segments, called members or elements, connected to each other at special points, called joints or nodes. In figure 1 the model of an actual bridge is depicted.

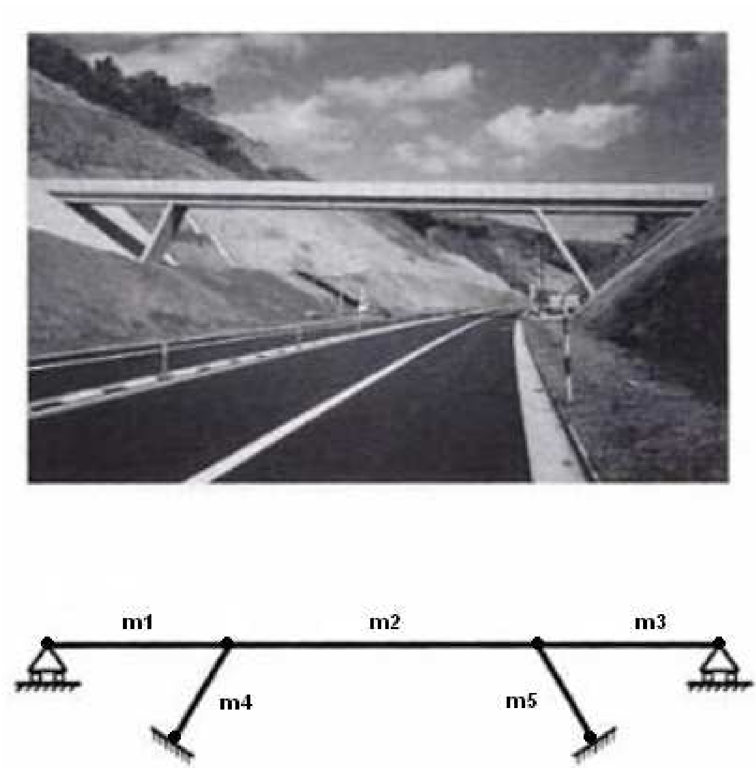


Figure 1. The ideal model of an actual bridge

In Figure 1 the joints are depicted as small circles and the members are numbered from 1 to 5. Furthermore the outer joints seem to carry some strange symbolic icons which represent the supports or else the boundary conditions of the problem. These supports dictate that their joints will not have a displacement towards a specific direction during the analysis of the structure.

In order for the D.S.M. to work the members must have some special properties. Each member must have a constant section along its length and its material cannot change also along its length. If an actual member does not have these properties it must be discretized into smaller members with constant section and material. Another important aspect about members is the way they are connected with the rest of the structure at joints. They may be firmly connected which implies that the displacements at their ends will be the same as the displacements of the joints or they may have releases. A release is the ability of a member's end not to follow the displacement of a joint towards a direction. For example an actual steel member connected with one bolt to the rest of the structure has a release, meaning that its end's rotation is not the same as the joint's rotation.

It is already stated that joints can be restrained by supports, meaning that their displacements towards some direction are zero-valued. The displacements of a joint when studying a two dimensional frame can be only three, two vertical to each other dispositions and a rotation. These displacements which are also called degrees of freedom are the unknowns the D.S.M. attempts to estimate.

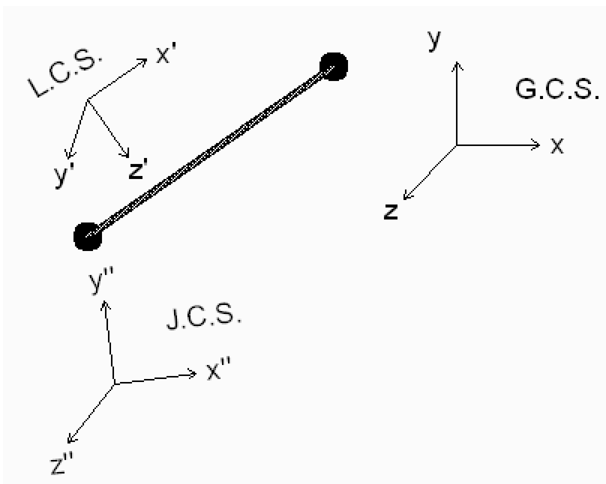


Figure 2. The required coordinate systems

Another important aspect of D.S.M. is the coordinate systems. Firstly a global coordinate system (G.C.S.) is required to define the coordinates of each joint. Its x-axis is usually the horizontal of the plane with direction from left to right, its y-axis is the vertical with direction from down to up and the z-axis is vertical to the plane and has the direction dictated by the clockwise rule. Its origin can be anywhere on the plane since we are only interested in the relative distance between the joints. Secondly each member has its own local coordinate system (L.C.S.). Its origin is the start joint of the member and its x'-axis is parallel to the member's length. The vertical to the x'-axis can be either the y'-axis or the z'-axis. This is a matter of convention and we considered it to be the z'-axis with a downwards direction as one observes the member from its start joint towards the end joint and y' resulted to be on the same direction as the z-axis of G.C.S. Finally in some cases a third coordinate system is needed which is called joint coordinate system (J.C.S.). It is required mainly when there is a support of a joint towards a direction that is not parallel to any axis of the global coordinate system. In these cases the rotation angle of the support's direction is needed to be known and the joint coordinate system is just the global one rotated by this angle.(fig. 2)

To complete the picture of the ideal model one must refer to the loads, the burdens this structure must carry. The tool we implemented is analyzing structures under static loading conditions and for the rest of this document when referring to loads static loads are implied. Loads fall into two broad categories, forces and bending moments which can be applied either on the members or on the joints of the structure. Member forces may also be distributed along the member, meaning that they are exercised along the member's length in contrast to concentrated forces which are imposed on a single point.

After describing the model the D.S.M. is able to solve the model, i.e. determine the unknowns which are the non-zero displacements of the joints. After a solution has been reached the analysis of the structure completes with the estimation of internal forces and bending moments of the members.

If we imaginary cut a member into two parts with a plane vertical to its length we should be able to imagine that there are small forces that act on the section of one part that hold it together with the other one. The resultant of these small forces are the internal forces, one along the member's length which is called axial force and one tangent to the section which is called shear force, and the bending moment. The final outcome of the implemented add-in are three diagrams which show the axial force, the shear force and the bending moment distribution along the members.

2.2 The Direct Stiffness Method

The method can be divided into ten steps: i) Data preparation, ii) Calculation of each member's local stiffness matrix, iii) Calculation of each member's local force vector, iv) Calculation of each member's transformation matrix, v) Globalization of each member's local stiffness matrix and force vector, vi) Assembly of the whole structure's stiffness matrix, vii) Assembly of the whole structure's force vector, viii) Boundary conditions, ix) Solution of the system and x) Calculation of internal forces.

The core of the method is included in the steps from two to nine while the first and last step can be considered as pre- and post-processing procedures.

2.2.1 Data preparation

It includes the definition of all the required for the method input data. For each member the following must be known: i) its start and end joint, ii) its length, iii) its direction on the plane, iv) its section, v) its material, vi) its releases and vii) the loads that are applied on it. For each joint the following must be known: i) its coordinates in the global coordinate system, ii) its supports if there are any, iii) its coordinate system's rotating angle and iv) the loads applied on it.

2.2.2 Calculation of each member's local stiffness matrix

Each member imposes some degree of difficulty on their start and end joint to displace. This difficulty is expressed by the member's stiffness matrix. For a member without any releases at its ends the matrix is of the following form:

$$K_L = \begin{bmatrix} \frac{EA}{L} & 0 & 0 & -\frac{EA}{L} & 0 & 0 \\ 0 & \frac{12EI}{L^3} & -\frac{6EI}{L^2} & 0 & -\frac{12EI}{L^3} & -\frac{6EI}{L^2} \\ 0 & -\frac{6EI_i}{L^2} & \frac{4EI}{L} & 0 & \frac{6EI}{L^2} & \frac{2EI}{L} \\ -\frac{EA}{L} & 0 & 0 & \frac{EA}{L} & 0 & 0 \\ 0 & -\frac{12EI}{L^3} & \frac{6EI}{L^2} & 0 & \frac{12EI}{L^3} & \frac{6EI}{L^2} \\ 0 & -\frac{6EI}{L^2} & \frac{2EI}{L} & 0 & \frac{6EI}{L^2} & \frac{4EI}{L} \end{bmatrix} \quad (1)$$

It can be interpreted as follows. Each column contains as entries the internal forces at the member's ends that result from the corresponding unitary displacement of one of the joints while all other displacements are considered zero-valued. (Mitsopoulou, 2009)

So each column refers to a corresponding displacement which is in terms of the local coordinate system. These displacements are six, three for each joint. For the matrix shown at formula 1 the first column corresponds to a disposition of the first joint aligned to the x-axis of the local coordinate system, the second to a disposition of the same joint aligned to z-axis of the L.C.S. and the third to the rotation about the y-axis of the L.C.S. The fourth, fifth and sixth column correspond to the same displacements of the other joint considering them in the same order. Because both internal forces and displacements refer to the local coordinate system this stiffness matrix is called local which is shown in the formula by indexing K with L.

$$K_L^{ref} = \begin{bmatrix} \frac{EA}{L} & 0 & 0 & -\frac{EA}{L} & 0 & 0 \\ 0 & \frac{3EI}{L^3} & 0 & 0 & -\frac{3EI}{L^3} & -\frac{3EI}{L^2} \\ 0 & 0 & 0 & 0 & 0 & 0 \\ -\frac{EA}{L} & 0 & 0 & \frac{EA}{L} & 0 & 0 \\ 0 & -\frac{3EI}{L^3} & 0 & 0 & \frac{3EI}{L^3} & \frac{3EI}{L^2} \\ 0 & -\frac{3EI}{L^2} & 0 & 0 & \frac{3EI_4}{L^2} & \frac{3EI}{L} \end{bmatrix} \quad (2)$$

Without explaining how these internal forces result it is stated that they depend on the length L of the member, the modulus of elasticity E of the material, the area A of the section, and the moment of inertia I of the section.

A reform of the matrix is needed when there are releases at its ends since a release implies that the member won't impose any difficulty during the corresponding displacement of the corresponding joint. An example of a reformed matrix is shown above.

The above matrix shows a local stiffness matrix of a member with a release aligned to the y-axis of the local coordinate system (a pin) at the start joint.

2.2.3 Calculation of each member's force vector

As already stated each member may have some loads imposed on it. These loads produce also internal forces at the ends of the member. The internal forces are calculated considering the member disconnected from the rest of the structure and bi-fixed, i.e. displacements of both of its ends are restrained. The reactions that result from the application of the load on this equivalent member can be determined by the use of the force method (whose explanation is beyond the scope of the paper). Tables 1 and 2 show the formulas used for calculating the internal forces for each loading type.

Table 1. Reactions of different loads not parallel to the member

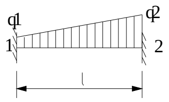
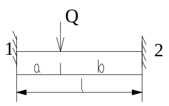
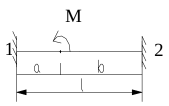
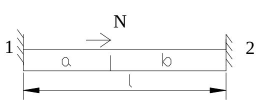
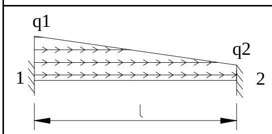
Loading type	Bending moment at end 1 M_1	Bending moment at end 2 M_2	Shear force at end 1 Q_1	Shear force at end 2 Q_2
	$+\frac{l^2}{60}(3q_1+2q_2)$	$-\frac{l^2}{60}(2q_1+3q_2)$	$-\frac{l}{20}(7q_1+3q_2)$	$-\frac{l}{20}(3q_1+7q_2)$
	$+Q\cdot\alpha\cdot\left(\frac{b}{l}\right)^2$	$-Q\cdot b\cdot\left(\frac{a}{l}\right)^2$	$-\frac{Q\cdot b^2}{l^3}(l+2\alpha)$	$-\frac{Q\cdot\alpha^2}{l^3}(l+2b)$
	$M\cdot\frac{b}{l}\left[3\cdot\left(\frac{\alpha}{l}\right)-1\right]$	$M\cdot\frac{\alpha}{l}\left[3\cdot\left(\frac{b}{l}\right)-1\right]$	$-\frac{6M}{l^3}\cdot\alpha\cdot b$	$+\frac{6M}{l^3}\alpha\cdot b$

Table 2.Reactions of different loads parallel to the member

Loading type	Axial force at end 1 N_1	Axial force at end 1 N_1
	$-N\cdot b/l$	$-N\cdot\alpha/l$
	$-q_1\cdot l/3-q_2\cdot l/6$	$-q_1\cdot l/6-q_2\cdot l/3$

The signs in the above table follow the sign convention the defined local coordinate system dictates. If there are more than one loads upon the member the internal forces come up as the sum of each load's resulting reactions. The resulting six values N_1, Q_1, M_1, N_2, Q_2 and M_2 are the entries of the local force vector.

$$F_L = \begin{bmatrix} N_1 \\ Q_1 \\ M_1 \\ N_2 \\ Q_2 \\ M_2 \end{bmatrix} \qquad (3)$$

In case there are releases at the member's ends a reformation of the vector is applied.

2.2.4 Calculation of each member's transformation matrix

Both the stiffness matrix and the force vector refer to the member's local coordinate system. In order to assembly the whole structure's stiffness matrix and to use equilibrium equations they must be transformed to refer to the joint's coordinate system. This task is assigned to the member's transformation matrix.

To construct it three angles are needed, the angle of the member on the plane θ and the rotation angles of the joint coordinate systems of the two joints of the member θ_1^r and θ_2^r. Its form is the following:

$$T = \begin{bmatrix} \cos(\theta - \theta_1^r) & \sin(\theta - \theta_1^r) & 0 & 0 & 0 & 0 \\ \sin(\theta - \theta_1^r) & -\cos(\theta - \theta_1^r) & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & \cos(\theta - \theta_2^r) & \sin(\theta - \theta_2^r) & 0 \\ 0 & 0 & 0 & \sin(\theta - \theta_2^r) & -\cos(\theta - \theta_2^r) & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix}$$

(4)

2.2.5 Globalization of each member's local stiffness matrix and force vector

After having constructed the transformation matrix for each member the global stiffness matrix and the global force vector of each member are:

$$K_{GL} = T^T \cdot K_L \cdot T \qquad (5)$$

$$F_{GL} = T^T \cdot F_L \qquad (6)$$

2.2.6 Assembly of the whole structure's stiffness matrix

Having the stiffness matrices of the members in terms of the joints' coordinate systems one can calculate the stiffness matrix of the whole structure which is an 3nx3n matrix where n stands for the total number of joints. It will be denoted as K_{tot} and its physical interpretation is the difficulty which all members impose on their joints to displace.

To construct it the "difficulties" described by each member's K_{GL} must be "summed". This is achieved through a mapping of each K_{GL} entries to the corresponding places of K_{tot} and finally by adding these entries to these initially zero-valued entries of K_{tot}.

The six by six global stiffness matrix is divided into four three by three submatrices as shown below:

$$K_{GL} = \begin{bmatrix} K_{bb} & K_{be} \\ K_{eb} & K_{ee} \end{bmatrix} \qquad (7),$$

where K_{bb} contains entries which are the reactions on the start joint due to unitary displacements of the same joint. K_{be} contains entries which are the reactions on the start joint due to unitary displacements of the end joint. K_{eb} contains entries which are the reactions on the end joint due to unitary displacements of the start joint and K_{ee} contains entries which are the reaction on the end joint due to unitary displacements of the end joint.

Considering that the start joint is numbered i and the end joint j the submatrices of K_{tot} that correspond to K_{bb}, K_{be}, K_{eb} and K_{ee} are the following: The first is formed by the three successive rows which lie after the row numbered 3·(i-1) and the three successive columns next to the column numbered 3·(i-1). The second is formed by the three successive rows which lie after the row numbered 3·(i-1) and the three successive columns next to the column numbered 3·(j-1). The third is formed by the three successive rows which lie after the row numbered 3·(j-1) and the three successive columns next to the column numbered 3·(i-1). Finally the fourth is formed by the three successive rows which lie after the row numbered 3·(j-1) and the three successive columns next to the column numbered 3·(j-1).

In mathematical terms the four submatices are:

i) $K_{tot[3\cdot(i-1)+1:3(i-1)+3,\ 3\cdot(i-1)+1:3(i-1)+3]}$	corresponds to	K_{bb}
ii) $K_{tot[3\cdot(i-1)+1:3(i-1)+3,\ 3\cdot(j-1)+1:3(j-1)+3]}$	corresponds to	K_{be}
iii) $K_{tot[3\cdot(j-1)+1:3(j-1)+3,\ 3\cdot(i-1)+1:3(i-1)+3]}$	corresponds to	K_{eb}
iv) $K_{tot[3\cdot(j-1)+1:3(j-1)+3,\ 3\cdot(j-1)+1:3(j-1)+3]}$	corresponds to	K_{ee}

With the above described mapping the K_{tot} can be assembled if we add the entries of the submatrices of each member's global stiffness matrix to the corresponding submatrix of K_{tot}.

2.2.7 Assembly of the whole structure's force vector

The whole structure's force vector is a vector of length 3xn where n stands for the total number of joints of the structure. Its entries contain the loads that are applied on the structure's joints expressed in terms of each joint coordinate system decreased by the resulting reactions of the member's loads. It is denoted as F_{tot}.

It is constructed in a similar but easier to explain manner as the K_{tot}. First the loads of the joints are added to the corresponding entries of F_{tot} which are initially zero-valued. Considering that the joint with number i has a force F_1 which is aligned to the x-axis of the joint coordinate system than this force is added to the first entry after the entry with number 3·(i-1). If it was aligned to the y-axis than it would be added to the second entry after the one with number 3·(i-1). Finally if it was a bending moment it would be added to the third one.

Secondly the reactions of the each member's loads which are contained in the member's global force vector should be subtracted from the corresponding entries of F_{tot}. Again each F_{GL} is divided into two vectors, one containing the reactions for the start joint and the other containing the reactions for the end joint of the member. The first vector's entries are subtracted from the entries which lie successively after the entry with number 3·(i-1) where i is the number of the start joint and the other vector's entries are subtracted from the entries which lie successively after the entry with number 3·(j-1) where j is the number of the end joint.

2.2.8 Boundary conditions

In order to impose the boundary conditions the lines and rows of the whole structure's stiffness matrix that correspond to the restrained degrees of freedom of the joints must be deleted. So if joint i has a restrain that imposes zero-valued disposition along the x-axis of the J.C.S. the row and the column with number 3·(i-1)+1 must be deleted. If the restrain imposed a zero-valued disposition along the y-axis of the J.C.S. than the row and the column with number 3·(i-1)+2 should have been deleted. If the restrain imposed a zero-valued rotation about the z-axis of the J.C.S. then the row and column with number 3·(i-1)+3 should have been deleted.

A similar elimination of entries must be applied to the entries of the structure's force vector. The mapping is the same but instead of deleting rows entries are deleted.

The outcomes of the above procedures are the reduced structure's stiffness matrix K_{tot}^{red} and the reduced structure's force vector F_{tot}^{red}.

2.2.9 Solution

Having calculated K_{tot}^{red} and F_{tot}^{red} a system of equations can be written in the following form:

$K_{tot}^{red} \cdot U_{tot}^{red} = F_{tot}^{red}$, where U_{tot}^{red} is the vector of all unknown displacements of the joints.

In the implementation the system was solved using Gauss-Jordan elimination through partial pivoting to invert K_{tot}^{red} and the outcome was multiplied with F_{tot}^{red}. The outcome, meaning U_{tot}^{red}, may be called partial displacement solution, since it does not include the known zero-valued displacements of the structure's joints. If we add the zero-valued displacements as well we get the complete displacement solution, a vector with 3·n entries where n stands for the number of joints, which will be denoted as U_{tot}.

2.2.10 Calculation of internal forces

The last described step concludes the core of the method. Having calculated U_{tot} one can identify the internal forces at each member's ends and then use simple equilibrium equations to identify the internal forces at each point of the member on which the internal forces is required to be known.

In order to identify the internal forces of each member's ends a six entry vector denoted as u_{GL} must be constructed for each member. This vector's entries will be the displacements of the member's start and end joint in terms of joint coordinate system.

A transformation of this vector is needed so that its displacements refer to the local coordinate system of the member. This is achieved by multiplying the transformation matrix of each member with the corresponding u_{GL}. The transformed vector will be denoted as u_L

$u_L = T \cdot u_{GL}$

The internal forces of each member's ends yield by multiplying each member's local stiffness matrix with u_L and adding to the result the local force vector. To be in accordance with classical statics' sign convention the signs of the internal forces of the member's start must change. To draw the diagrams of axial force, shear force and bending moment equilibrium equations are used. For an in detail description on drawing these diagrams the reader is referred to the citations. [Komodromos P.,2009] [Mitsopoulou E.,2009]

2.3 The implementation

Our attempt was to implement an add-in for AutoCAD that would be able to perform the D.S.M. on two dimensional frames and produce the final diagrams, meaning axial force shear force and bending moment diagrams. We chose not to implement as a stand alone application because we wanted to exploit the powerful mechanisms AutoCAD provides for defining, i.e. drawing, the model and because this way mechanisms such undo, redo and save didn't need to be implemented from scratch.

2.3.1 A few words about AutoCAD

First of all the version of AutoCAD for which the add-in was implemented was AutoCAD 2009. As it is well known it is very powerful software which allows computer aided design. Describing all functionality of AutoCAD is not essential. However it must be pointed out that it provides many different APIs that allow extending its functionality, such as VBA, AutoLISP ,ObjectARX etc. Among them ObjectARX was chosen mainly because it is the API that allows defining custom objects, i.e. objects that behave according to the programmer's wishes when an AutoCAD command is invoked.

2.3.2 The tools used

The ObjectARX programming environment provides an object-oriented C++ application programming interface for developers to use, customise, and extend AutoCAD. It includes C++ libraries that are the building blocks one can use to develop AutoCAD applications, extend AutoCAD classes and protocol, and create new commands that operate in the same manner as built-in AutoCAD commands. An ObjectARX application is simply a dynamic link library (DLL) that shares the address space of AutoCAD and makes direct function calls to AutoCAD. Furthermore, Visual Studio 2005 SP1 was used to develop and compile the application.

2.3.3 Objects and entities defined in the application

AutoCAD's database may contain two broad categories of custom items, namely objects and entities. Objects have not a graphical representation on AutoCAD's model space whereas entities have. In the application eight custom entities were defined, namely frElement which represented a member of the structure, joint which represented a joint of the structure, five entities regarding the different types of loads and an entity called diagram that was used to display the diagrams that yield from the structure's analysis.

The objects that were defined were four representing the sections that were provided to assign on each member. These sections can be parameterised, e.g. the rectangular section has as parameters the height and the width of the section and it calculates automatically the necessary for the method values of area and moment of inertia. The other section types were a circular section, an I-section and a custom section for which the user is allowed to input directly the moment of inertia and the area.

2.3.4 The application's menu

After the add-in is loaded a pop-up menu is added to AutoCAD's menu bar which is displayed at figure 3

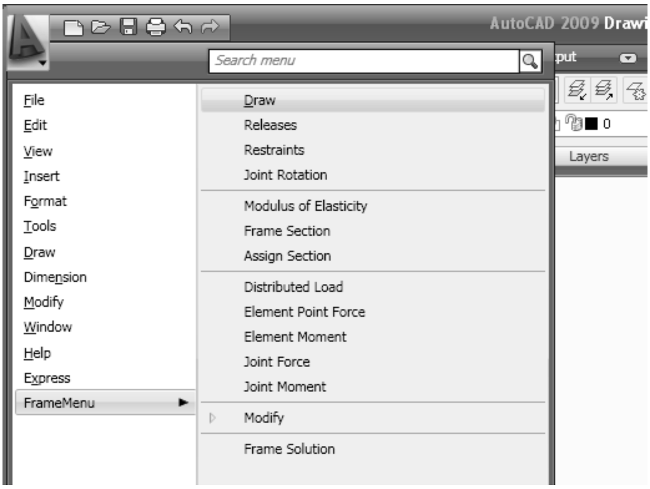


Figure 3. The Frame Menu

The menu consists of 12 menu items which correspond to the registered by the application commands that are necessary to be issued in order for the user to describe the ideal model of the structure. The 13th menu item is chosen when the solution for the described model is needed. The menu also includes a submenu with items that when chosen allow the user to modify the loads and the sections he had previously defined.

2.3.5 Drawing of a member

The first menu item "draw" issues the command "frElement" which allows the user to define a member in model space. When issued the user is prompted to select the first point of the member.

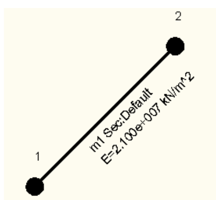


Figure 4. A member with its joints drawn in model space

The second point can be chosen using relative orthogonal or polar coordinates or it can simply be chosen by clicking on a point of the model space. Then a member with two joints at its ends is drawn (fig. 4).

The user is allowed to define the joint of another member as the starting or ending point of the member he wants to draw. Each member will eventually only two joints one at each of its ends. To make it easy for the user to draw a member beginning from another member's end the entity is supplied with two snap points at its ends (fig.5)

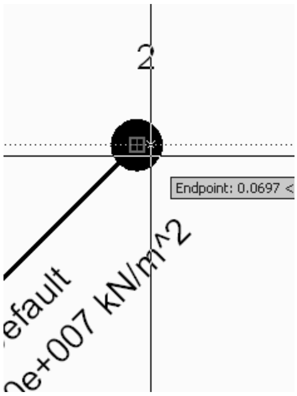


Figure 5. A snap point of a member

Another important functionality feature provided for this entity is the ability to modify it through grip points. Grip points or grips are special points most entities in AutoCAD are equipped with. They become visible when the user selects the entity and allow him to modify the entity's shape through various operations such as stretching, moving etc. The grip points provided for the member allow the user to stretch it from one of its ends (fig. 6).

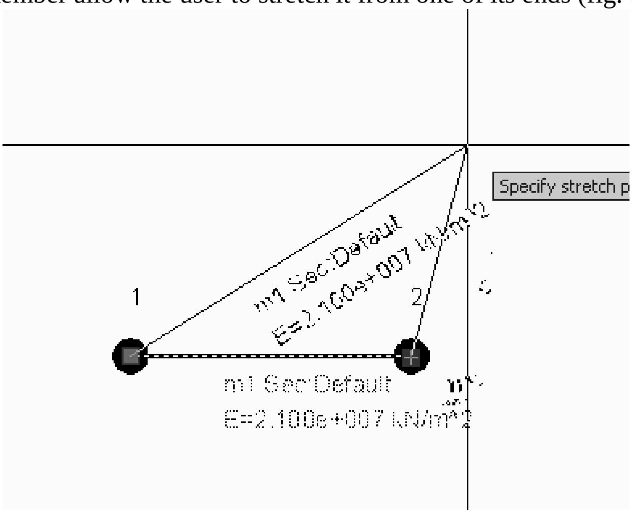


Figure 6. Using grip points to stretch a member

2.3.6 Releases, restrains and joint coordinate system

Releases of a member can be defined through the given choice of the menu, named releases. When the corresponding command is issued the user is prompted to select a member. Then a dialog box appears that give the ability to the user to define which end of the member will have a release and the type of the release through checkboxes (fig.7).

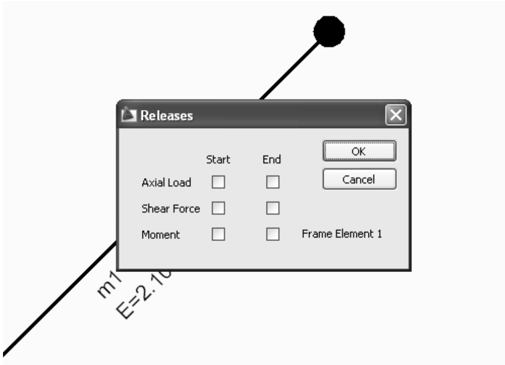


Figure 7. Releases' dialog box

Each type of release has its own symbol drawn at the member's end. For example if a member has a release of type moment meaning that it can have different rotation than the joint two circles, one inside the other, are drawn on the member's end next to the joint, depicting a pin.

Restrains are defined again through dialog box only now the user is prompted to select a joint. Again each type of restrain has its own symbol, e.g. a fully fixed joint has an upside down T drawn beneath it (fig. 8)

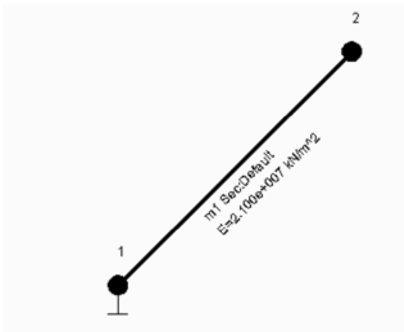


Figure 8. A fully fixed joint

A rotated joint coordinate system can be defined when the user chooses the menu item "Joint Rotation" where he must input the clockwise rotation angle of the joint C.S.

2.3.7 Materials and sections

In the second part of the paper it was mentioned that the material of each member must be known. Actually the only property of the material that is required for the method is the modulus of elasticity E. In the application the user must input the value of E with the help of the menu item "Modulus of Elasticity". When the corresponding command is issued the user is prompted to choose a member and type the value of E. The application regards the input value measure in kN/m^2. Each member drawn for the first time has a default value of E equal to 21000000 kN/m^2.

On the other hand sections are first defined and then assigned to a member. Through a combo box the user can choose the type of section he wants to define. Then a new dialog box appears according to the user's previous choice that requests the user to input the necessary parameters of the section. As an example figure 9 shows the dialog box for an I-section.

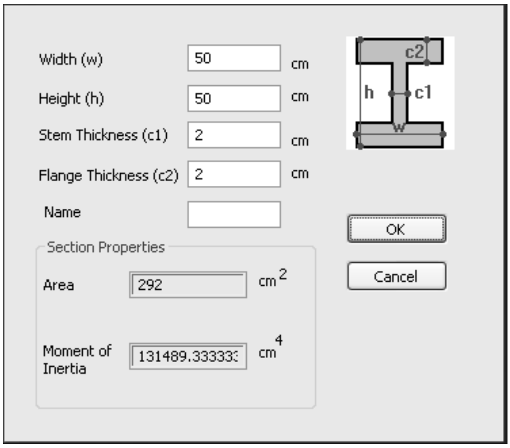


Figure 9. Dialog box for the definition of an I-section

As one can see the user is requested to enter the parameters of the section in the appropriate fields as well as a name which is obligatory to be given for the definition of the section. The section properties are calculated automatically and cannot be altered by the user.

After the definition of the sections the user must assign the sections to the members otherwise each member will have a default rectangular section.

2.3.8 Loads

The types of loads that are available for the user to apply on the structure are five, namely distributed trapezoidal load consisting of forces acting on the whole length of the member, member point forces, member bending moments, joint forces and joint bending moments.

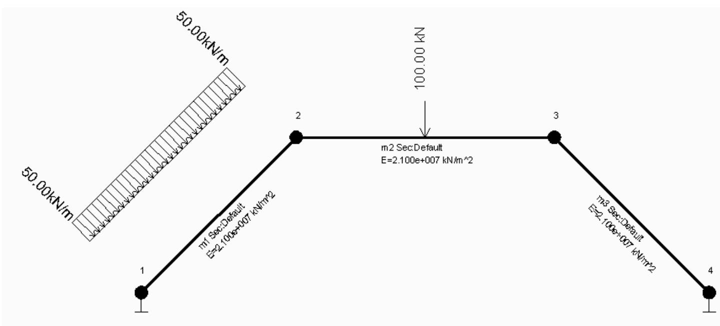


Figure 10. Distributed load and a member point force

Figure 10 displays some of the previously mentioned types of loads. The input parameters of the loads are entered with the help of dialog boxes that appear when the user issues the corresponding to the type of load command and after he has chosen the entity (member or joint) which the load will be applied on. An example of the dialog boxes use is displayed by figure 11.

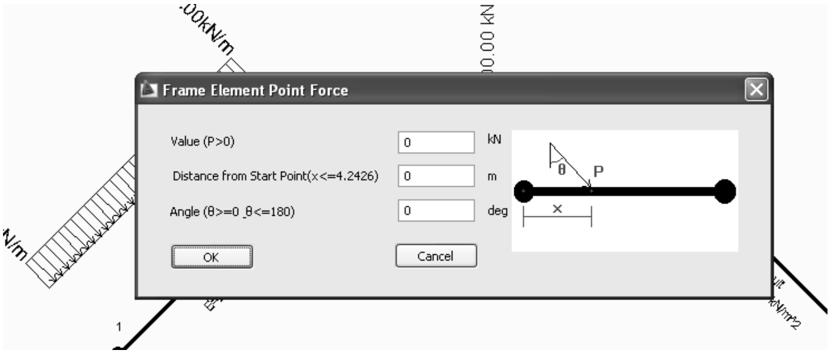


Figure 11. Dialog box for a member's point force

For all types of load the loading value is required. If it is a distributed load the value at the start and the end of the member measured in kN/m are requested. If it is a bending moment the value inserted is considered to be measured in kN·m. Finally on the case of forces the value is measured in kN. Other required inputs are for member forces and member bending moments the distance from the start of the member they are applied on and for all types of forces and the distributed loading their direction.

2.3.9 Modification and saving mechanisms

The user is able to modify any of the entities drawn besides joints since the latter to be attached at a member's end. This means that joints should always be drawn as part of the members and to behave according to the member's behaviour. A member on the other hand can be modified with the use of grip points which were described in section 2.3.5. In most cases the joint "follows" the stretching of the member.

The sections and the loads are modified through registered by the add-in commands which are included in the submenu modify. This submenu consists of items that let the user choose among section modification or any type of loads modification.

To erase an entity the erase command of AutoCAD can be issued. Only joints can't be erased directly but only after erasing the members they are attached on.

Undoing and redoing any command issued is also an ability provided by AutoCAD. The only task needed to be performed was to define the behaviour of the custom objects and entities through the use of virtual methods in the relative classes.

At any stage of the description of the model the user is also able to save his work in dwg format and load it the next time he uses AutoCAD. Again through virtual methods of the relative classes the behaviour of the custom objects and entities has been defined.

To summarise all this for undoing, redoing, saving and loading no extra commands were needed to be registered by the application. For all other modifications the user can use the submenu modify.

2.3.10 Solution

Finally after the model has been described the user can choose the menu item "Frame Solution" which starts the analysis of the structure. Special consideration should be taken in order to describe a model that is well supported meaning it is both a determinate or indeterminate structure and not a mechanism. Otherwise the application will print a message that informs the user of the insufficient support of the structure.

In the following pages four figures, namely figures 12 to15, display the model, the axial force, the shear force and the bending moment diagram of a simple structure produced by the application.

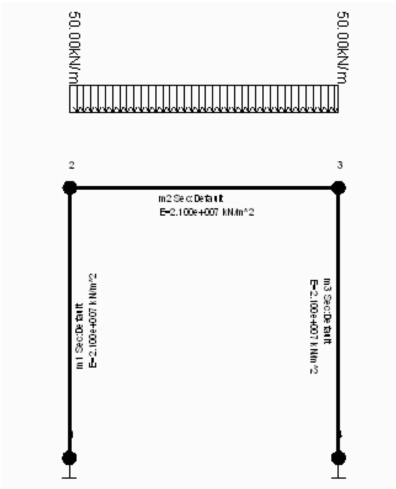


Figure 12. The model

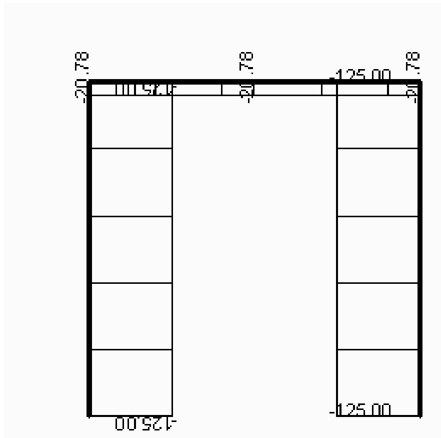


Figure 13. The axial force diagram

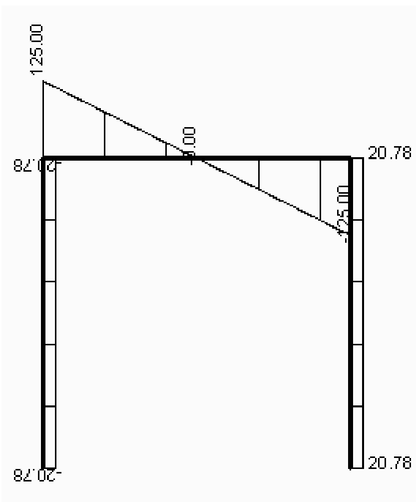


Figure 14. The shear force diagram

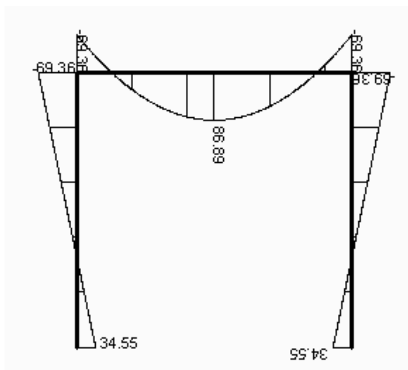


Figure 15. The bending moment diagram

3. CONCLUSION

The implemented add-in can be used to solve and analyze any two-dimensional determinate or indeterminate structure. This means that it will be rather useful for a student of structural or civil engineering who wishes to study the behavior of two dimensional frames.

Limitations regarding the application are the fact that is for two dimensional structures only and that it doesn't analyze the structure for dynamic loading nor supports inelastic analysis of the structure. These features can be provided by much larger applications that may include additional functionality as well. However this small to medium scale application shows why it is preferable to develop an add-in for a CAD software and specifically AutoCAD and not a stand alone application as a structural analysis tool.

To be more specific two aspects of the described add-in can be considered as its main advantages, its relatively easy implementation and the functionality that provides to the user to describe the model. Both of them are due to the fact that it was implemented as an add-in for another application, specifically AutoCAD, and not as a stand alone application.

As an add-in we didn't have to invent mechanisms for saving, undoing and redoing since these were provided by AutoCAD. Nor did we have to build a database that would maintain all application's objects. All that was required to be done is to define the behavior of the custom objects and entities during the execution of these commands. This was done through virtual methods of the base classes from which the custom objects and entities derived.

Finally AutoCAD's model space worked perfectly as a graphical user interface providing great extends of flexibility for the user to describe the geometry of the structure's model. Determining points with the use of polar or orthogonal relative coordinates or with the use of snaps are features that were provided by AutoCAD and which either they wouldn't be implemented in a stand alone application or it would be a real nuisance to provide them. In addition to that grips can make the modification of a drawn member look like a toy.

To conclude the choice of implementing the method as an add-in for AutoCAD must be considered correct especially if one takes into consideration that it is the CAD software that most engineers use.

REFERENCES

Argyris J. H et al., 1960. Energy Theorems and Structural Analysis, *Butterworths*, London, England; Part I reprinted from Aircraft Engrg. 26, Oct-Nov 1954 and 27, April-May 1955.

Felippa, Carlos A., 2001. A historical outline of matrix structural analysis: a play in three acts *In Computers & Structures*, 79 (14), pp 1313–1324

Komodromos P., 2009. *Structural Analysis: Modern Methods with the Use of Personal Computers,* Papasotiriou Publishers, Athens, Greece.

Mitsopoulou E., 2009. *Statics of Linear Structures: Classic and Matrix Methods of Analysis,* Sofia Publishers, Thessaloniki, Greece

Turner M. J., 1959. The direct stiffness method of structural analysis. Structural and Materials Panel Paper, *AGARD Meeting*, Aachen, Germany.