

MATEST12, Educational Tool in Visual Basic .NET, for the Analysis of Structures Applying the Matrix Method

Fernando Jove Wilches*¹, Jose Rodrigo Hernández Avila², and Jose Ángel Padilla Castro³

^{1,2,3} *Department of Civil Engineering, Universidad de Sucre, Sincelejo, Sucre, Colombia.*

Abstract

The use of structural analysis methods brings with it the application of laborious mathematical calculations, which is why, in the case of matrix analysis of structures, this was put aside for some time, given that due to the complexity of its calculations, and it was very common to make mistakes. However, with the appearance of computers, this method has had an important resurgence, since through programming, very complex mathematical calculations can be performed in a reduced time. From this perspective, the MATEST12 software has been created, a computational tool that, in a Windows environment, has been developed under the Visual Basic.NET programming language and allows the application of the matrix method for the analysis of structures, such as trusses, grills, frames (in 2D and 3D), beams, frames with diagonal, combined systems, frames and shear walls (2D, only available for uniform loads). For the development of the MATEST12 tool, the steps to be followed when creating structural models were taken into account, as well as the variables related to each one of them, which led to a review of previous works on the subject of the matrix method of structures and its programming, resulting in the design and systematization of an interface that, thanks to its friendly and intuitive design, is easy for the user to use, clearly showing the insertion of the necessary data to the calculation and the results obtained. After being verified, the software yielded results with a difference of no more than 5% with respect to known analysis and design programs, such as SAP2000, a difference that is negligible when it comes to structural design.

Keywords: software, matrix analysis, structural analysis, programming language

I. INTRODUCTION

In recent years there has been an increase in the demand for engineering works, because the population is constantly increasing, resulting in a challenge for engineering and other parties involved [1]. As a consequence of this, there has been an accelerated growth in the construction sector, which has led to the need for tools that speed up and make the designs carried out in this field more reliable. The structural design of a civil engineering work deals with understanding, predicting and calculating the stability and resistance of a structural element.

All structures that are exposed to external loads, such as houses, buildings, bridges, viaducts, dams, among others, must be analysed and designed, in order to be able to understand, predict and calculate the stability and resistance of the elements that make up said structures. Generally, to achieve this end, commercial structural design software is used, where the analysis and design of the structures is essential to determine the significant threats to the integrity and stability of the same [2]. In past decades, the procedure of analysing and designing a structure was done manually and became a very tedious and time-consuming task. For this reason, currently most of the analysis and design work is carried out using specialized software for this task, with the advantage of saving time and minimizing errors [3], [4].

Through the analysis and design of a structure, it is possible to know how optimal the design is, both in terms of resistance-stability, and in economic terms [5]. The application of engineering software in current industrialization processes is an increasingly common practice and plays an important role in the conception, analysis and development of projects. There is no doubt that the use of software has simplified the procedures for analysis and design of structures [6], minimizing the possibility of making mistakes in the modelling or execution stages of mathematical calculations [3]. Thus, the results of the analysis can show the suitability of the structure for its use, through the analysis of the geometry of the structure, its sections, the materials that compose it and the loads to which they are exposed.

The structural analysis and design programs that are known today, allow to carry out the modelling of the geometry of the structure, the numerical processing of the data involved, as well as to carry out the analysis of the results. These three stages are known as pre-processing, processing, and post-processing, respectively. Commercial programs perform all these calculations by means of the finite element method [7], which has caused it to have gained a lot of ground over the years. Even so, matrix analysis continues to be widely accepted and taken as a reference in the methods used today in the teaching of civil engineering [8], [9], which is why it is a very useful tool for the structural analysis.

Regarding the implementation of computational tools for the analysis of structures, several authors can be found who have

carried out work and developed computational tools for the calculation of structures, such as Nicole Katherine Gordillo Bravo [10] and Fabiana Elizabeth Cunalata Vásquez [11], who developed interesting works to receive the title of civil engineers at the Technical University of Ambato (Ecuador). Mention should also be made of authors such as Jairo Uribe Escamilla, Jorge Eduardo Hurtado and Roberto Aguilar Falconi, who in their books present the bases for the systematization of the matrix method, making use of different computational tools; It is also worth mentioning the main commercial structural analysis and design software such as SAP2000, ETABS, SAFE, Robot Structural, among others, which have been taken as a basis for comparison by the aforementioned authors, for the tools designed in their works.

The present work aims to offer the MATEST12 computational tool, created in order to automate the use of the matrix analysis method of structures. Within the scope of the document, the considerations taken into account for the creation of the computer tool are established, as well as the objectives achieved, such as the type of structures that allow the tool to be analysed, the load cases, the common sections and the type of materials. In the same way, the methodology used in the programming of the tool, the guidelines followed, the software used and the programming language are shown. The main views or program windows are also presented and finally, a comparison is made for each of the types of structures sheltered within the software, carrying out an analysis between the developed program and a reference software in the field of structural analysis. For this purpose, the commercial use software SAP2000 owned by the American company Computers and Structures Inc. (CSI) [12] was chosen.

II. MATERIALS AND METHODS

II.I Software development methodology

For the development of the MATEST12 tool, the method of matrix analysis of structures was used, and more specifically, the method of rigidities, for which a programming code was developed under the Visual Basic.NET Language, which allowed the creation of a structural analysis tool.

The interface of the MATEST12 Tool has been developed through the VISUAL STUDIO 2015 programming IDE, making use of the Visual Basic.NET language, in such a way that the user, following the small help that the tool shows and without knowing in depth the procedures of calculation, you can define and obtain the results of any structure that you intend to solve, which is within the options that the tool allows to solve. In the case of the visualization of results, these are presented in a simple and understandable way, by means of tables that explain the values obtained. On the other hand, you have the possibility of exporting the calculation memories to Microsoft Excel and you can see a detailed step-by-step of the calculation procedure to obtain these results.

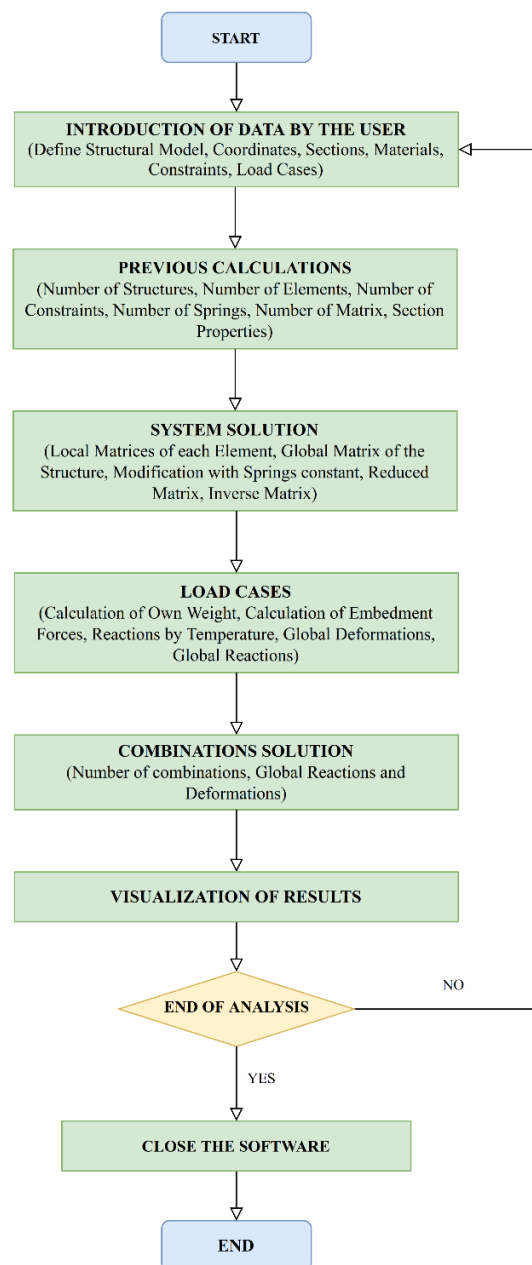
Internally, the tool has algorithms that, by defining the type of structure and inserting the necessary information about the geometry, the sections and the materials to be used, allow, from a structural model, to analyse the structure quickly and effectively. The importance of the tool in the academic field is

highlighted within the teaching-learning process of civil engineering, being possible through it, to carry out the comparison and verification of results, the analysis of structures with manual calculations or of other tools in which the same method is applied.

II.II Software calculation procedure

The MATEST12 tool follows the following logical sequence to perform the structural analysis calculations. In total, six stages are roughly described for the execution of the program: the introduction of the data by the user, the preliminary calculations, the system solution, the load cases, the combination solution and the visualization / export of results. In Figure 1, each of the stages carried out for the analysis of a structure is shown in detail.

Fig. 1. MATEST12 working procedure



Source: Self-Made

II.III Scope and limitations of the project

The MATEST12 Tool allows the analysis of structural systems, using the matrix analysis method, limiting its field of action to the following elements:

- Continuous beams
- Frames (2D and 3D).
- Trusses (2D and 3D).
- Grills (2D).
- Combination systems, frames and shear walls (2D, only available for uniform loads)

Additionally, the tool allows the possibility of working with shear deformations, temperature deformations and the option of entering elastic supports (springs). The program is also able to model and analyse several structures at the same time.

Among the characteristics that are excluded from the program and as points to work towards the future, there are: the integration of databases with more sections of structural elements, as well as material properties; be able to work with variable sections and compound sections; add static and dynamic seismic analysis or wind force analysis. On the other hand, it is necessary to clarify that improvements and additions can be made in the graphic part of the program, in such a way that it can be shown in the form of figures, in the same main window, the internal force diagrams of the elements (axial forces, shear forces, flexural moments and torsional moments) and the reactions, turns and displacements, as well as the deformed elements of the elements analysed in the program.

II.IV Software description

The MATEST12 application is composed of views or windows, which help in the introduction of the model data, the analysis of the introduced structures, as well as the understanding and application of the method.

Presentation window: provides the name of the software along with the data of the author of the same.

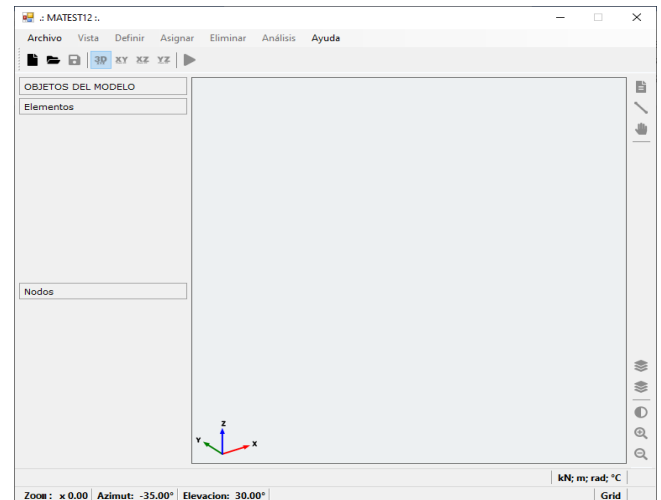
Fig. 2. Presentation window



Source: Self-Made

Main window of the program: presents the main interface of the program, in which are the access buttons to all its functions, together with the different work areas.

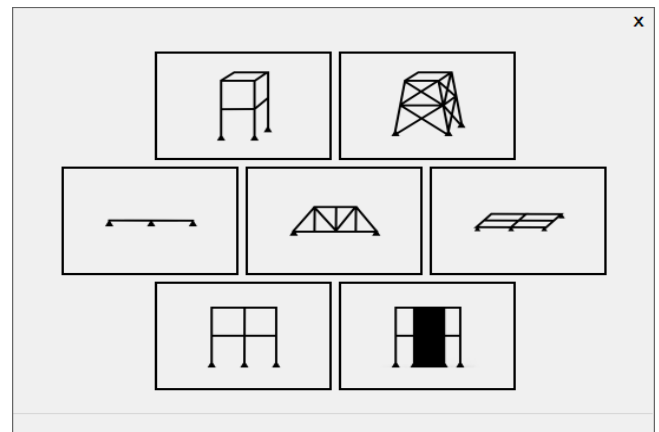
Fig. 3. Main window of the program



Source: Self-Made

Select structural model window: the accesses for the analysis of the different structures considered by the program are displayed.

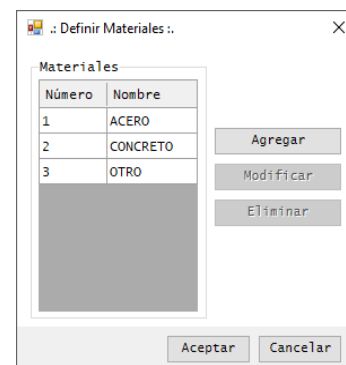
Fig. 4. Select structural model window



Source: Self-Made

Window to define materials: it contains the list of the predefined materials in the tool, giving the possibility of adding new ones and modifying or eliminating existing ones.

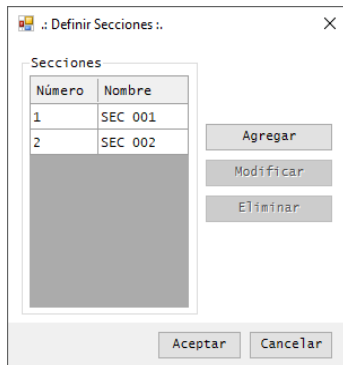
Fig. 5. Window to define materials



Source: Self-Made

Define/assign sections window: contains the list of sections present in the analysed structural model, it also shows the possibility of adding new cross sections and modifying or eliminating existing sections.

Fig. 6. Define/assign sections window



Source: Self-Made

Results window: set of two forms; the first, for simplified results of each node; the second, for the results of the entire structural model analysed.

Error messages: the tool has alerts, which are activated when the user enters any wrong value, or blocks the entry of characters that cause a malfunction.

III. COMPARISON RESULTS WITH COMMERCIAL SOFTWARE

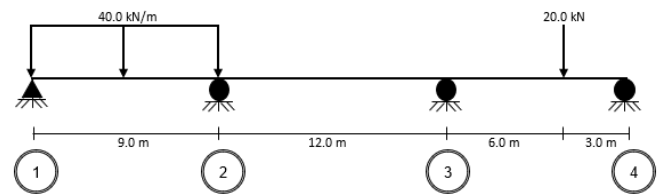
The MATEST12 software has been validated by applying various examples, comparing the results obtained through analytical calculations and other structural calculation programs such as SAP2000 v22, finding variations of less than 5%, considering the most significant differences to the mathematical approximations made, but that in the end, for the structural design of the element, they could be considered null.

Here are eight models that are within the possibilities that the software can analyse. These models are: continuous beams, beams with elastic supports, 2D frames, combined systems (frames-shear walls), grills, 2D trusses, 3D trusses and 3D frames. In the illustrated examples, a comparison is presented between the results obtained using the SAP2000 v22 program and the MATEST12 software.

III.I Application N ° 1 - Continuous Beam Analysis

A continuous beam of 3 sections is analysed, exposed to the loads shown in the figure, with a rectangular section 0.35 x 0.60 m, of concrete, with a modulus of elasticity, $E = 21'700,000 \text{ kN/m}^2$.

Fig. 7. Continuous beam for analysing



Source: Self-Made

Table 1. Strains for continuous beam

DESCRIPTION			SOFTWARES		
N	Dir.	Uds	MATEST12	SAP2000 v22	% VARIATION
1	Giro Y	rad	0.006872474	0.006872	0.0069%
2	Giro Y	rad	-0.00481858	-0.00481	0.0086%
3	Giro Y	rad	0.001489756	0.00149	0.0164%
4	Giro Y	rad	-0.00117943	-0.00117	0.0369%

Source: Self-Made

Table 2. Reactions for continuous beam

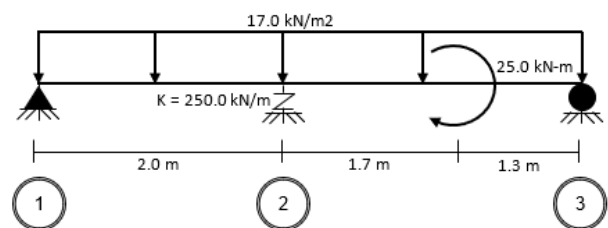
DESCRIPTION			SOFTWARES		
N	Dir.	Uds	MATEST12	SAP2000 v22	% VARIATION
1	Fz	kN	159.4638236	159.464	0.0001%
2	Fz	kN	219.3624769	219.362	0.0002%
3	Fz	kN	-16.7251912	-16.725	0.0011%
4	Fz	kN	17.89889079	17.899	0.0006%

Source: Self-Made

III.II Application N ° 2 - Analysis of beams with elastic supports

A continuous beam of 2 sections is analysed, with an elastic support, exposed to the loads shown in the figure, with a solid circular section of STEEL of radius $R = 0.25 \text{ m}$ with a modulus of elasticity, $E = 200'000.000 \text{ kN/m}^2$.

Fig. 8. Beams with elastic supports for analysing



Source: Self-Made

Table 3. Strains for beam with elastic supports

DESCRIPTION			SOFTWARES		
N	Dir.	Uds	MATEST12	SAP2000 v22	% VARIATION
1	Giro Y	rad	0.000272064	0.000272	0.0237%
2	Des Z	m	-0.000411000	-0.00041	0.0242%
	Giro Y	rad	8.05434E-05	0.000081	0.5637%
3	Giro Y	rad	-0.000272090	-0.00027	0.0360%

Source: Self-Made

Table 4. Reactions for beam with elastic supports

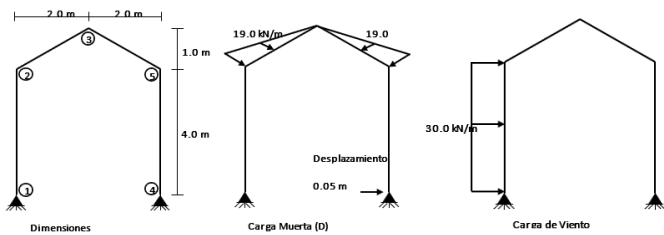
DESCRIPTION			SOFTWARES		
N	Dir.	Uds	MATEST12	SAP2000 v22	% VARIATION
1	Fz	kN	80.14689884	80.147	0.0001%
2	Fz	kN	0.102975067	0.103	0.0242%
3	Fz	kN	80.16749386	80.167	0.0006%

Source: Self-Made

III.III Application N ° 3 - Analysis of 2D frames

The analysis of a 2D frame exposed to wind loads and dead load is developed. For this purpose, there is a rectangular section for the columns and beams of 0.30 x 0.30 m, made of concrete, and a modulus of Elasticity, $E = 24'821.128 \text{ kN/m}^2$.

Fig. 9. Frame in 2D for analysing



Source: Self-Made

Table 5. Strains for beam with elastic supports

DESCRIPTION			SOFTWARES		
N	Dir.	Uds	MATEST12	SAP2000 v22	% VARIATION
1	Giro Y	rad	0.033721175	0.033721	0.0005%
2	Desp. X	m	0.093906445	0.093906	0.0005%
	Des Z	m	5.73297E-05	0.000057	0.5784%
	Giro Y	rad	0.007628391	0.007628	0.0051%
3	Des X	m	0.094857164	0.094857	0.0002%
	Des Z	m	-0.0018562	-0.00185	0.0108%
	Giro Y	rad	-0.00254034	-0.00254	0.0135%
4	Des X	m	0.0500000	0.05000	0.0000%
	Giro Y	rad	0.015412895	0.015413	0.0007%
5	Des X	m	0.09564039	0.09564	0.0004%
	Des Z	m	-0.00015754	-0.00015	0.2905%
	Giro Y	rad	0.003306441	0.003306	0.0133%

Source: Self-Made

Table 6. Reactions for beam with elastic supports

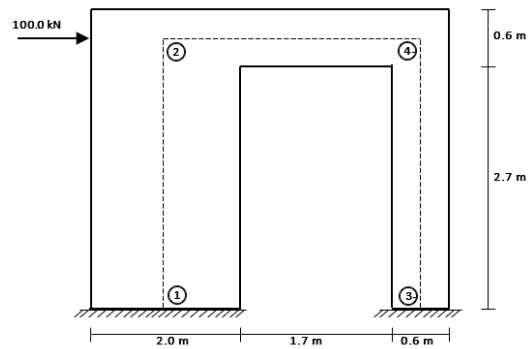
DESCRIPTION			SOFTWARES		
N	Dir.	Uds	MATEST12	SAP2000 v22	% VARIATION
1	Fx	kN	-94.6456643	-94.646	0.0004%
	Fz	kN	-27.7761434	-27.776	0.0005%
4	Fx	kN	-25.3543356	-25.354	0.0013%
	Fz	kN	92.22385653	92.224	0.0002%

Source: Self-Made

III.IV Application N ° 4 -Analysis of combined systems (frames-shear walls)

Considering a plate gantry system, like the one shown in the figure. Having a plate of 0.30 x 2.00 m, a column of 0.30 x 0.60 m and a beam of 0.30 x 0.60 m. For analysis purposes, the self-weight of the structure is considered null, of concrete with a modulus of elasticity, $E = 2.50 \times 10^8 \text{ kN/m}^2$.

Fig. 10. Frames-shear walls for analysing



Source: Self-Made

To perform this comparison, it is recommended to modify the Cross-section (axial), Factor Area to 1000000, in all the sections in the SAP2000 v22 software, to observe more homogeneity in the compared results.

Table 7. Strains for beam with elastic supports

DESCRIPTION			SOFTWARES		
N	Dir.	Uds	MATEST12	SAP2000 v22	% VARIATION
2	Des X	m	1.7392E-05	0.000017	2.3103%
	Des Z	m	3.9502E-13	3.95E-13	0.0052%
	Giro Y	rad	5.4922E-06	0.000005492	0.0052%
4	Des X	m	1.7392E-05	0.000017	2.3103%
	Des Z	m	-1.316E-12	-1.317E-12	0.0201%
	Giro Y	rad	-1.758E-06	-0.00000175	0.0091%

Source: Self-Made

Table 8. Reactions for beam with elastic supports

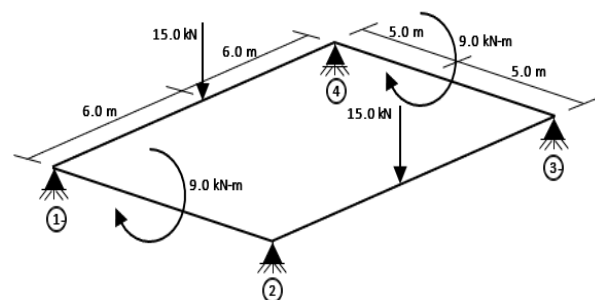
DESCRIPTION			SOFTWARES		
N	Dir.	Uds	MATEST12	SAP2000 v22	% VARIATION
1	Fx	kN	-89.2234608	-89.223	0.0005%
	Fz	kN	-19.7510330	-19.751	0.0002%
	My	kN-m	-225.373265	-225.3733	0.0000%
3	Fx	kN	-10.7765391	-10.777	0.0043%
	Fz	kN	19.75103301	19.751	0.0002%
	My	kN-m	-15.3736365	-15.3736	0.0002%

Source: Self-Made

III.V Application N ° 5 - Analysis of grills

It analyses a grid subjected to point loads and moments as shown in the figure. The sections of the elements are rectangular 0.40 x 0.45 m, made of CONCRETE that has a modulus of Elasticity $E = 275'000,000 \text{ kN/m}^2$, with a poisson's ratio of 0.25.

Fig. 11. Grills for analysing



Source: Self-Made

Table 9. Strains for grills

DESCRIPTION			SOFTWARES		
N	Dir.	Uds.	MATEST12	SAP2000 v22	% VARIATION
1	Giro X	rad	0.000211552	0.000212	0.2113%
	Giro Y	rad	0.000527178	0.000527	0.0338%
2	Giro X	rad	0.000211552	0.000212	0.2113%
	Giro Y	rad	-0.00052717	-0.00052	0.0338%
3	Giro X	rad	-0.00021155	-0.00021	0.2113%
	Giro Y	rad	-0.000527178	-0.000527	0.0338%
4	Giro X	rad	-0.000211552	-0.000212	0.2113%
	Giro Y	rad	0.000527178	0.000527	0.0338%

Source: Self-Made

Table 10. Reactions for grills

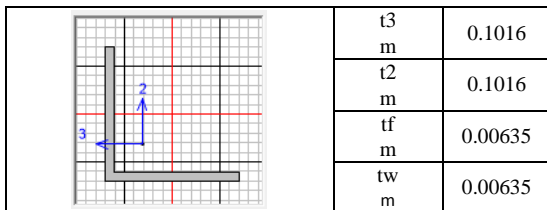
DESCRIPTION			SOFTWARES		
N	Dir.	Uds.	MATEST12	SAP2000 v22	% VARIATION
1	Fz	kN	54.151968	54.152	0.0001%
2	Fz	kN	54.151968	54.152	0.0001%
3	Fz	kN	54.151968	54.152	0.0001%
4	Fz	kN	54.151968	54.152	0.0001%

Source: Self-Made

III.VI Application N ° 6 - Analysis of 2D trusses

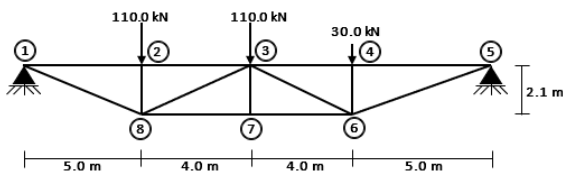
There is a reinforcement with the loads shown, with an angular profile of steel, with the dimensions shown in Figure 12, with a modulus of elasticity $E = 200'000,000 \text{ kN/m}^2$. For analysis purposes, the self-weight of the structure is considered null.

Fig. 12. Angular profile dimensions



Source: Self-Made

Fig. 13. Truss in 2D for analysing



Source: Self-Made

Table 11. Strains for 2D truss

DESCRIPTION			SOFTWARES		
N	Dir.	Uds.	MATEST12	SAP2000 v22	% VARIATION
2	Des X	m	-5.38937E-06	-0.00000538	0.0069%
	Des Z	m	-0.00024270	-0.000243	0.1214%
3	Des X	m	-9.70087E-06	-0.00000970	0.0013%
	Des Z	m	-0.00032006	-0.00032	0.0187%
4	Des X	m	-5.38937E-06	-0.00000538	0.0069%
	Des Z	m	-0.00019019	-0.00019	0.1031%
6	Des X	m	3.77342E-05	0.000038	0.6994%
	Des Z	m	-0.00018859	-0.000189	0.2161%
7	Des X	m	-3.25196E-06	-0.00000325	0.0012%
	Des Z	m	-0.00032006	-0.00032	0.0187%
8	Des X	m	-4.42381E-05	-0.000044	0.5412%
	Des Z	m	-0.00023682	-0.000237	0.0749%

Source: Self-Made

Table 12. Reactions for 2D truss

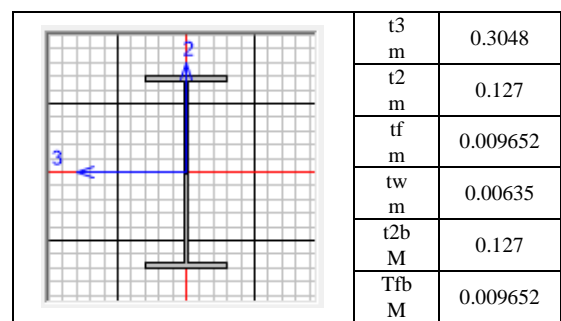
DESCRIPTION			SOFTWARES		
N	Dir.	Uds.	MATEST12	SAP2000 v22	% VARIATION
1	Fx	kN	-297.619047	-297.619	0.0000%
	Fz	kN	142.7777778	142.778	0.0002%
5	Fx	kN	297.6190476	297.619	0.0000%
	Fz	kN	107.2222222	107.222	0.0002%

Source: Self-Made

III.VII Application N ° 7 - Analysis of 3D trusses

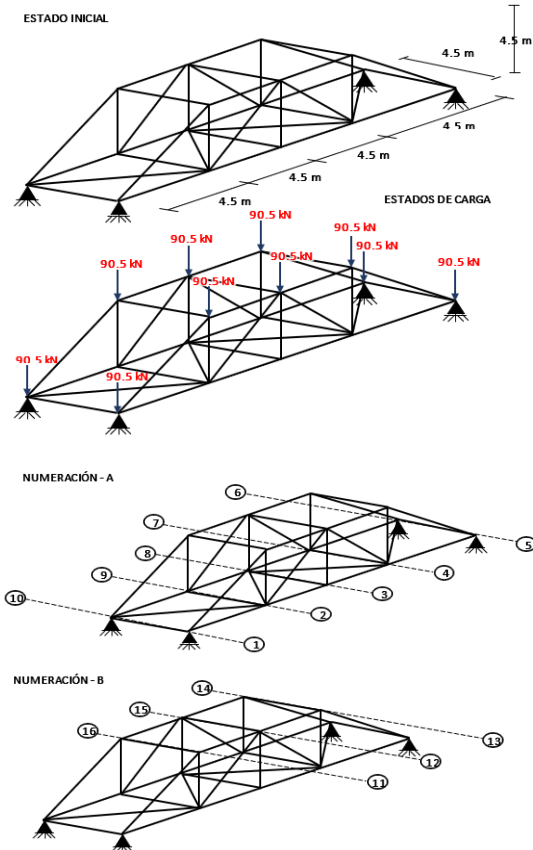
We want to build a bridge like the one shown in the figure 14, for which use is made of steel I-profiles, with Modulus of Elasticity 210000000 kN/m^2 , with bolted joints. After carrying out the load analysis, it was determined that a force of 90.5 kN is applied to each node. For analysis purposes, the self-weight of the structure is considered null.

Fig. 14. Profile in I dimensions



Source: Self-Made

Fig. 15. Truss in 3D for analysing



Source: Self-Made

Table 13. Strains for 2D truss

N	Dir.	Uds.	SOFTWARES		
			MATEST12	SAP2000 v22	% VARIATION
2	Des X	m	-0.00011367	-0.000114	0.2856%
	Des Y	m	-0.00011367	-0.000114	0.2856%
	Des Z	m	-0.00283851	-0.002839	0.0171%
3	Des Y	m	-0.00022734	-0.000227	0.1537%
	Des Z	m	-0.00359522	-0.003596	0.0215%
4	Des X	m	0.000113674	0.000114	0.2856%
	Des Y	m	-0.00011367	-0.000114	0.2856%
	Des Z	m	-0.00283851	-0.002839	0.0171%
7	Des X	m	0.000113674	0.000114	0.2856%
	Des Y	m	-0.00011367	-0.000114	0.2856%
	Des Z	m	-0.00283851	-0.002839	0.0171%
8	Des Y	m	-0.00022734	-0.000227	0.1537%
	Des Z	m	-0.00359522	-0.003596	0.0215%
9	Des X	m	-0.00011367	-0.000114	0.2856%
	Des Y	m	-0.00011367	-0.000114	0.2856%
	Des Z	m	-0.00283851	-0.002839	0.0171%
11	Des X	m	0.000682047	0.000682	0.0068%
	Des Z	m	-0.00261116	-0.002611	0.0063%
12	Des Z	m	-0.00359522	-0.003596	0.0215%
13	Des X	m	-0.00068204	-0.000682	0.0068%
	Des Z	m	-0.00261116	-0.002611	0.0063%
14	Des X	m	-0.00068204	-0.000682	0.0068%
	Des Z	m	-0.00261116	-0.002611	0.0063%
15	Des Z	m	-0.00359522	-0.003596	0.0215%
16	Des X	m	0.000682047	0.000682	0.0068%
	Des Z	m	-0.00261116	-0.002611	0.0063%

Source: Self-Made

Table 14. Reactions for 2D truss

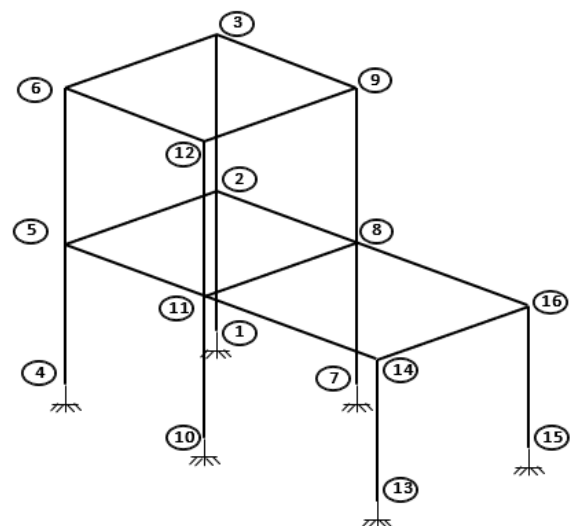
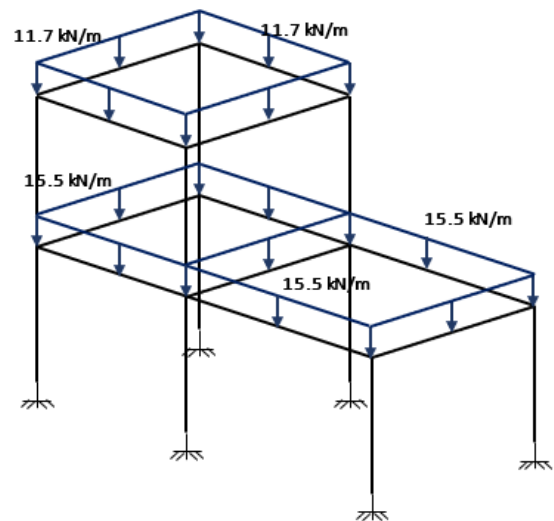
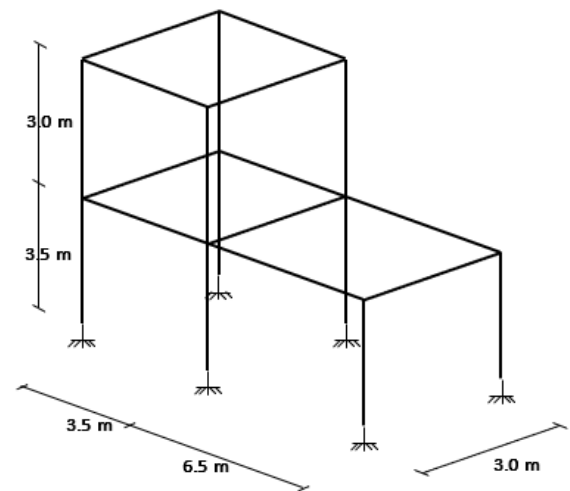
N	Dir.	Uds.	SOFTWARES		
			MATEST12	SAP2000 v22	% VARIATION
1	Fx	kN	158.375	158.375	0.0000%
	Fz	kN	226.25	226.25	0.0000%
5	Fx	kN	-158.375	-158.375	0.0000%
	Fz	kN	226.25	226.25	0.0000%
6	Fx	kN	-158.375	-158.375	0.0000%
	Fz	kN	226.25	226.25	0.0000%
10	Fx	kN	158.375	158.375	0.0000%
	Fz	kN	226.25	226.25	0.0000%

Source: Self-Made

III.VIII Application N ° 8 - Analysis of 3D frames

It is proposed to carry out the analysis of a space frame, where the sections used are 0.30 x 0.40 for the beams and 0.30 x 0.30 for the Columns, with a concrete with a modulus of elasticity of 28'000,000 kN / m². It is proposed to apply the loads shown with a factor of 1.4 and neglecting the structure's own weight, in order to determine the displacements and reactions indicated in Tables 15 and 16.

Fig. 16. Frame in 3D for analysing



Source: Self-Made

Table 15. Strains for 2D truss

DESCRIPTION			SOFTWARES		
N	Dir.	Uds.	MATEST12	SAP2000 v22	% VARIATION
2, 5	Des X	m	-0.000667825	-0.000668	0.0262%
	Des Y	m	1.25981E-06	0.00000126	0.0152%
	Des Z	m	-0.000150589	-0.000151	0.2720%
	Giro X	rad	0.000186081	0.000186	0.0438%
	Giro Y	rad	4.67055E-05	0.000047	0.6266%
	Giro Z	rad	2.94595E-08	2.95E-08	0.0018%
3, 6	Des X	m	5.35165E-06	5.35E-06	0.0065%
	Des Y	m	-2.01534E-06	-2.02E-06	0.0170%
	Des Z	m	-0.000212639	-0.000213	0.1694%
	Giro X	rad	0.000184705	0.000185	0.1593%
	Giro Y	rad	0.000482373	0.000482	0.0773%
Giro Z	rad	1.81116E-09	1.81E-09	0.0089%	
8, 11	Des X	m	-0.000670845	-0.000671	0.0231%
	Des Y	m	1.2603E-06	0.00000126	0.0234%
	Des Z	m	-0.000301266	-0.000301	0.0885%
	Giro X	rad	0.000190897	0.000191	0.0538%
	Giro Y	rad	0.000711942	0.000712	0.0081%
Giro Z	rad	-1.11733E-07	-1.117E-07	0.0299%	
9, 12	Des X	m	4.32729E-06	4.33E-06	0.0068%
	Des Y	m	-2.03692E-06	-2.04E-06	0.0038%
	Des Z	m	-0.000365966	-0.000366	0.0092%
	Giro X	rad	0.000183848	0.000184	0.0825%
	Giro Y	rad	-0.000342121	-0.000342	0.0354%
Giro Z	rad	-1.57655E-08	-1.58E-08	0.0284%	
14, 16	Des X	m	-0.000697565	-0.000698	0.0624%
	Des Y	m	1.25248E-06	0.000001252	0.0386%
	Des Z	m	-0.000133033	-0.000133	0.0250%
	Giro X	rad	-0.00031037	-0.00031	0.1193%
	Giro Y	rad	-0.00192221	-0.001922	0.0109%
Giro Z	rad	2.24178E-07	2.242E-07	0.0099%	

Source: Self-Made

Table 16. Reactions for 2D truss

DESCRIPTION			SOFTWARES		
N	Dir.	Uds.	MATEST12	SAP2000 v22	% VARIATION
1	Fx	kN	3.882857707	3.883	0.0037%
	Fy	kN	-1.693415739	-1.693	0.0246%
	Fz	kN	108.4243002	108.424	0.0003%
	Mx	kN-m	1.958637516	1.9586	0.0019%
	My	kN-m	6.542791239	6.5428	0.0001%
	Mz	kN-m	-0.00011202	-0.000112	0.0175%
4	Fx	kN	3.882857707	3.883	0.0037%
	Fy	kN	1.693415739	1.693	0.0246%
	Fz	kN	108.4243002	108.424	0.0003%
	Mx	kN-m	-1.958637516	-1.9586	0.0019%
	My	kN-m	6.542791239	6.5428	0.0001%
	Mz	kN-m	0.00011202	0.000112	0.0175%
7	Fx	kN	9.929090151	9.929	0.0009%
	Fy	kN	-1.737074474	-1.737	0.0043%
	Fz	kN	216.9117326	216.912	0.0001%
	Mx	kN-m	2.009035336	2.009	0.0018%
	My	kN-m	13.53142104	13.5314	0.0002%
	Mz	kN-m	0.000424866	0.0004249	0.0079%
10	Fx	kN	9.929090151	9.929	0.0009%
	Fy	kN	1.737074474	1.737	0.0043%
	Fz	kN	216.9117326	216.912	0.0001%
	Mx	kN-m	-2.009035336	-2.009	0.0018%
	My	kN-m	13.53142104	13.5314	0.0002%
	Mz	kN-m	-0.000424866	-0.0004249	0.0079%
13	Fx	kN	-13.81194786	-13.812	0.0004%
	Fy	kN	2.8071165	2.807	0.0042%
	Fz	kN	95.78396728	95.784	0.0000%
	Mx	kN-m	-3.236456608	-3.2365	0.0013%
	My	kN-m	-13.79097552	-13.791	0.0002%
	Mz	kN-m	-0.000852436	-0.0008524	0.0042%

DESCRIPTION			SOFTWARES		
N	Dir.	Uds.	MATEST12	SAP2000 v22	% VARIATION
15	Fx	kN	-13.81194786	-13.812	0.0004%
	Fy	kN	-2.8071165	-2.807	0.0042%
	Fz	kN	95.78396728	95.784	0.0000%
	Mx	kN-m	3.236456608	3.2365	0.0013%
	My	kN-m	-13.79097552	-13.791	0.0002%
	Mz	kN-m	0.000852436	0.0008524	0.0042%

Source: Self-Made

IV. CONCLUSIONS

The software developed called MATEST12, allows the automation of the calculation processes in the matrix analysis, allowing the calculation of the displacements, deformations, forces and reactions, of the main types of elements and structural systems, both in two and three dimensions. Among the advantages of using computer tools such as MATEST12, there is greater efficiency and time savings in the analysis of structures, as well as the reduction of errors in the modeling of the structure or in the performance of calculations. Additionally, the MATEST12 computational tool has been validated by applying various structural models, comparing the results obtained through the software with analytical calculations and with the results of other structural calculation programs such as SAP2000 v22. Variations of less than 5% were found between the results obtained by the MATEST12 program and those of the SAP2000 software, which makes MATEST12 a tool of great interest, both for academic purposes and for its application in the professional field by students and structural engineers, since it is a free software.

ACKNOWLEDGMENTS

The authors thank the engineer Edilberto Elias Contreras Sierra for the contributions made in the development of the manuscript and also thank the SIES research hotbed and the GIMAGUAS research group.

REFERENCES

- [1] Sabeer M & Gouse D. Comparison design result of RCC building using Staad and Etabs software. International Journal of Innovative Research in Advanced Engineering (IJIRAE). 2015;8(2):92-97.
- [2] Ramanjaneyulu V, Dharmesh M & Chiranjeevi V. Comparative study on design results of a multi-storied building using Staad Pro and Etabs for regular and irregular plan configuration. International Research Journal of Engineering and Technology (IRJET). 2018;5(1):676-681.
- [3] Raj Y. Comparison of Analysis & Design Results of a Structural Elements using STAAD-Pro, STRUDS and ETABS Software. International Journal of Scientific Progress and Research (IJSPR). 2015;14(1):50-52.
- [4] Mittal P, Kad N, Chutani S & Tangri A. A Comparison of the Analysis and Design Results of 4 Story Using STAAD Pro and ETABS Software. International Journal of Research (IJR). 2016;3(5):69-73.

- [5] Hiwase PD, Joshi A, Keshariya A. Comparison between Manual and Software Approach towards Design of Structural Elements. The International Journal of Engineering and Science (IJES). 2018:54-56.
- [6] Nemo L, Prakash J. Comparative Study of Structural Software Sap2000 and Staad Pro. International Journal of Engineering Science Invention (IJESI). 2019:8(3):37-43.
- [7] Vélez W, Gómez D & Thomson P. Ajuste de modelos de elementos finitos: revisión. DYNA. 2009:76(158):177-189. Retrieved from http://www.scielo.org.co/scielo.php?script=sci_arttext&pid=S0012-73532009000200017&lng=en&tlng=es.
- [8] Durán M. Análisis matricial de estructuras. Formación universitaria. 2020:13(2). <https://dx.doi.org/10.4067/S0718-50062020000200001>
- [9] Paico G & Segundo A. El “método matricial” en la enseñanza del análisis estructural: ¿una realidad! UCV-HACER. Revista de Investigación y Cultura. 2013:2(1):151-160. Disponible en: <https://www.redalyc.org/articulo.oa?id=521752180019>
- [10] Cunalata FE. Desarrollo De Software Para Cálculo De Vigas Continuas Simplemente Apoyadas, Empotradas Y En Volado. Tesis, Universidad Técnica De Ambato. Perú, 2016. [In Internet] Available in: <https://repositorio.uta.edu.ec/jspui/handle/123456789/24783>
- [11] Gordillo NK. Desarrollo De Un Software Para Cálculo De Deflexiones En Vigas Según La Norma Ecuatoriana De La Construcción (Nec). Tesis, Universidad Técnica De Ambato. Perú, 2016. [In Internet] Available in: <https://repositorio.uta.edu.ec/jspui/handle/123456789/23407>
- [12] Download page Sap2000 trial version: <https://www.csiamerica.com/products/sap2000/trial>