From 3D modelling in CAD environment to the generation of computational models

Application to the case of hydraulic structures of complex geometry

Duarte José Saavedra Cardoso

duartesaavedra@tecnico.ulisboa.pt

Department of Civil Engineering, Architecture and Georesources, Instituto Superior Técnico, University of Lisbon

Av. Rovisco Pais, 1049-001 Lisboa, Portugal

May 2019

Abstract

Within the scope of Civil Engineering structures design, more specifically, of hydraulic structures, the development of three-dimensional models, required for the structural calculation, can prove to be a time-consuming task due to the inherent nature of the geometries normally involved.

Since a three-dimensional geometric model in CAD environment is usually available, there is the interest in obtaining the calculation model automatically from the corresponding geometric model, thus avoiding the extra work that the creation of a new one would imply.

The thesis focuses on the study of this problem, as well as on the development and operation of a new software tool built using *Python* programming language, which contributes to the increasing of the interoperability between *AutoCAD*, geometry modelling tool, and *SAP2000*, structural analysis and calculation application.

Keywords

Geometric model in CAD environment, Structural calculation model, Finite Element Method, Hydraulic structures.

1 Introduction

1.1 Background

Hydraulic structures are generally characterized by their complex spatial geometry (curves definition with plan and profile variability), considering the nature of surfaces in contact with the flow, the need to accommodate hydromechanical equipment and also aspects related to the excavation geometry.

Due to their particularities, their geometric models are regularly composed of solid elements, thus existing the interest in executing the Finite Element Method in order to generate 3D meshes and, in the calculation phase, obtain more accurate results.

Although some simplifications are admitted, it is important that these models remain representative without being inefficient computationally.

Usually it proves to be a time-consuming task, so it is important to obtain directly a calculation model, as reliable as possible, from the existing geometric model, avoiding repetitive procedures caused by the lack of compatibility between work tools of different natures.

1.2 Objectives

Within the scope of the conception, manipulation and analysis of relatively complex geometries, the thesis lays on the study and contribution to the development of new procedures for information transfer that ensure interoperability as well as a more automated and efficient process between three-dimensional modelling tools and structural analysis software.

Figure 1 illustrates the conceptual background of this work:

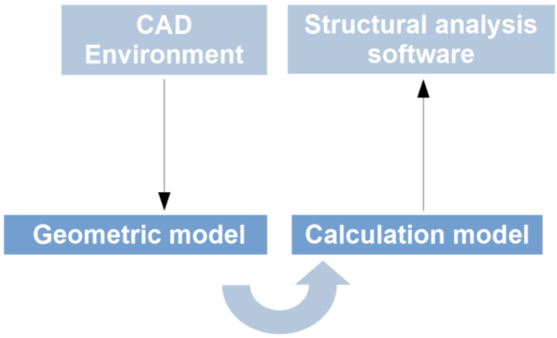


Figure 1: Conceptual background of this work

In addition to the above, it is also sought to demonstrate the potential of the development of new tools that guarantee the transfer of information between applications of different natures, especially in the Civil Engineering environment.

1.3 Case study - Interoperability between *AutoCAD* and *SAP2000*

Within the context of the mentioned problem, a practical case will be presented and explored that will allow to illustrate the potentialities under study, being investigated the interoperability between *AutoCAD* software (*Autodesk, Inc.*) and *SAP2000* (*Computers and Structures, Inc.*).

The first software is commonly used for the creation of geometric models, the starting point from which will be developed a new computing tool, using information transfer by means of files that can be imported and interpreted by the structural analysis software.

DXF file format was used for data transfer between the two computer programs, proving to be an adequate format.

2 Application development

2.1 Background

SAP2000 is able to import various format files for geometric models consisting of lines and surfaces (***Line*** and ***3D Face*** entities, respectively, in *AutoCAD* environment), so it would be expected that the same would also be possible for 3D models composed of solid elements (***3D Solid*** entities in *AutoCAD*). However, in the current development status of the structural analysis program, it is not able to correctly interpret the referred type of entities.

This represents an obstacle for the structural analysis of hydraulic structures (or other structures with similar geometric properties) when using *SAP2000*, since their *AutoCAD* models usually consist of 3D solid elements, as already mentioned in Subchapter **1.1**.

Since this implies the creation of the calculation model from scratch, it decreases significantly the efficiency of the whole process.

It was sought to understand the data structure interpreted by *SAP2000* as solid entities and, even though the available information on this matter is not much, it was possible to clarify, through the information available on *Computers and Structures, Inc.* website, that in order to import to *SAP2000* a 3D geometric model (directly from *AutoCAD*) that produces the intended effect, it is required that these elements are modelled respecting the *Polygon Mesh* topology, as it is known in *AutoCAD* environment.

2.2 *Polygon Mesh* elements topology

Polygon Mesh topological elements are represented geometrically in the most varied ways, being defined by a particular structure in *AutoCAD* library of objects.

In order to understand this structure, see **Figure 2**, which, on its left side, illustrates a generic *Polygon Mesh* topological element in its three-dimensional shape (in this example, an hexahedron) and, in its right side, illustrates the element's nodes and faces reduced to the same plan:

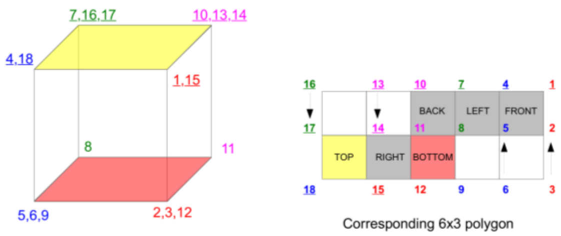


Figure 2: *Polygon Mesh* elements topology (Information available in *Computers and Structures, Inc.* website)

As can be seen, this topology requires an element to be described through a mesh. For the specific case of elements to be imported to *SAP2000*, this mesh must be a *6x3* mesh (on the right side of **Figure 2**, six columns of nodes are observed, in the horizontal direction, and three rows of nodes in the vertical direction) consisting of eighteen nodes.

In three-dimensional space, some of these nodes are collapsed. Examples are nodes 2, 3 and 12 or nodes 7, 16 and 17, as can be seen on the left side of **Figure 2**.

On the right side of the same figure, the collapsing of nodes that share the same coordinates in 3D space is indicated by the arrows.

2.3 Conversion tool

In order to contribute with a solution for this problem, it is proposed to use a tool, developed within the scope of the thesis, which allows the semi-direct conversion of geometric models composed of ***3D Solid*** entities (*AutoCAD*), in an importable DXF file (geometric model) to *SAP2000*. The entities in this file will be automatically interpreted by the structural analysis program as being solid elements.

Figure 3 shows the organization chart for the proposed procedure:

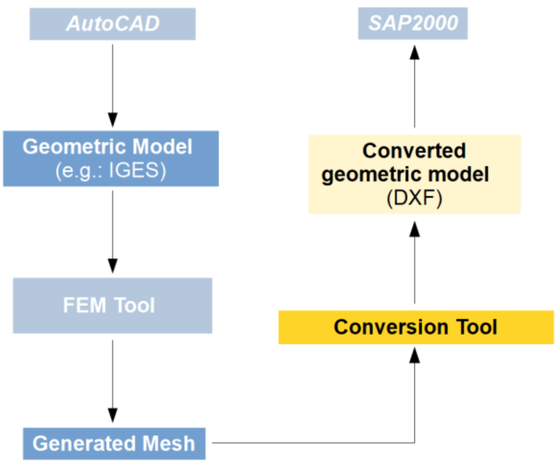


Figure 3: Organization chart for the proposed process

As can be seen in **Figure 3**, after exporting *AutoCAD*'s geometric model, it is required to generate a 3D mesh, using a Finite Element solver tool, that afterwards will be converted to a DXF file.

It is important to highlight the fact that the developed conversion tool is merely able to interpret 3D meshes composed of linear elements and that are exported under Gmsh format files or meshes generated using *Ansys*.

A three-dimensional mesh, only composed of linear elements, may consist of several different element shapes, being the most common:

- **Tetrahedrons** - Consisting of four nodes, four faces and six edges;
- **5-node pyramids** - Defined by five nodes, five faces and eight edges;
- **Triangular prisms** - Having six nodes, five faces and nine edges;
- **Hexahedrons** - Composed of eight nodes, six faces and twelve edges.

Figure 4 illustrates all the above listed elements that usually compose 3D linear meshes (from left to right, tetrahedron, 5-node pyramid, triangular prism and hexahedron):

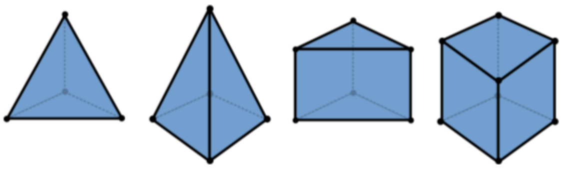


Figure 4: Types of linear elements generally used in 3D mesh generation

Due to the fact that *SAP2000* only interprets as solid entities, *6x3 Polygon Mesh* elements and that 3D meshes may contain any of the linear element types illustrated in **Figure 4**, during the development of the conversion tool, it was necessary to add a function to artificially convert each of these typological elements into *6x3 Polygon Mesh* entities.

2.3.1 Tetrahedrons

Considering **Figure 2**, which illustrates the representative hexahedron of the *Polygon Mesh* structure (*6x3* polygonal mesh), it is possible to conclude that if all the nodes of one of the faces of this hexahedron, as well as two nodes of the opposite face, were collapsed, from this transformation would result a tetrahedral element.

In **Figure 5**, the resulting tetrahedron is conceptually represented. Nodes 1, 4, 7, 10, 13, 14, 15, 16, 17 and 18 all share the same coordinates in three-dimensional space. The same for nodes 8 and 11.

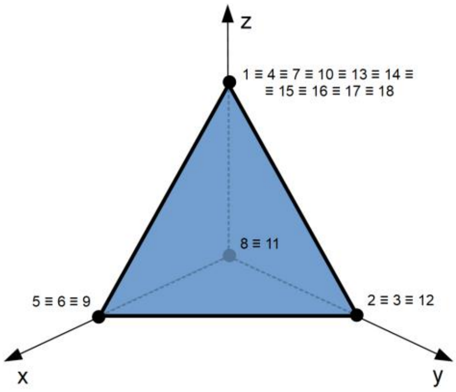


Figure 5: Tetrahedron, *6x3 Polygon Mesh* element

After implementing this transformation in the code, in order to test this hypothesis, a mesh composed of tetrahedral elements was generated. Then, the conversion tool was executed and the resulting DXF file was produced. It was possible to verify that the importation of these elements into *SAP2000* was successfully done.

It should be noted that other points could have been collapsed. Therefore, the hypothesis shown in **Figure 5** does only represent one of the possible combinations of node collapsing that would culminate in the same solution.

Analogously, the same is true for the following presented cases of 5-node pyramids, defined by five nodes, and triangular prisms, defined by six nodes.

2.3.2 5-node pyramids

For the case of 5-node pyramids, the collapsing of the face where points 1, 4, 7, 10, 13, 14, 15, 16, 17 and 18 all share their Cartesian coordinates is enough. All other points maintain the coordinate relationship previously presented in **Figure 2**. This transformation is illustrated in **Figure 6**:

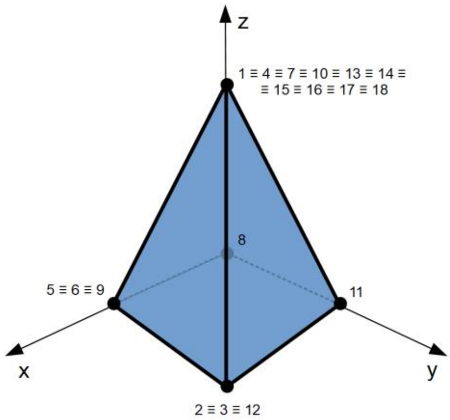


Figure 6: 5-node pyramid, *6x3 Polygon Mesh* element

2.3.3 Triangular prisms

In order to obtain a triangular prism, defined by six nodes, it is required to carry out the collapse of two nodes on two opposite faces.

Figure 7 illustrates the transformation adopted for this element type while implementing the conversion tool:

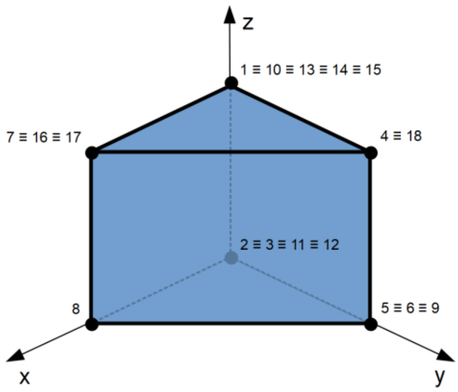


Figure 7: Triangular prism, *6x3 Polygon Mesh* element

By comparing **Figure 2** and **Figure 7**, it is possible to verify that the transformation adopted, comprises the collapsing of nodes 1, 10, 13, 14 and 15, as well as, on the opposite face, the collapsing of nodes 2, 3, 11 and 12.

All other nodes keep the coordinate relationship previously shown in **Figure 2**.

2.4 Additional considerations

For the case of hexahedral elements, whether they are regular or not, due to their linear geometry and to the fact that they are composed of eight points in three-dimensional space, no additional operation is required.

Therefore, the definition of points coordinates for these typological elements is performed by the conversion tool, as already illustrated in **Figure 2**.

It should be emphasized that any element of order higher than one, or that presents a different geometry from those above referred, cannot be interpreted by the developed conversion tool. Consequently, if this is the case, the proposed process (illustrated in the chart of **Figure 3**) will not be successfully concluded.

3 Application examples

This section illustrates the use of the developed conversion tool applied to structures with complex geometry.

Two examples of geometric models, created in *AutoCAD*, going through the referred process (previously illustrated in **Figure 3**) will be shown.

Static simulations, where only the model's deadweight has been considered, were carried out with the purpose of proofing both models.

This way, it is possible to observe the applicability of the developed conversion tool when executed within the scope of geometric models of increasing complexity.

It should also be noted that both models analysed were simulated (in *SAP2000* environment), considering the material properties and joint restraints (at the base of each of the structures), respectively, indicated in **Figure 8** and **Figure 9**.

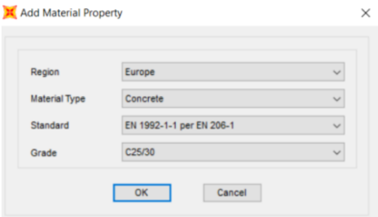


Figure 8: Material properties considered (*SAP2000*)

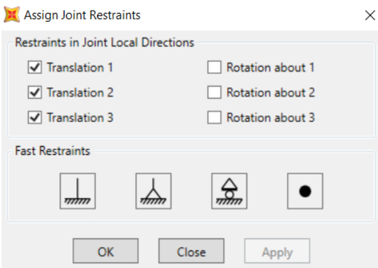


Figure 9: Joint restraints considered (*SAP2000*)

The geometric models presented will be the following:

- E1 - Spiral case of hydropower plant;
- E2 - WES Spillway.

Both examples will be presented with the following sequence:

- Presentation of the geometric model (*AutoCAD*);

- Calculation of model's total volume, using ***MASSPROP*** command (*AutoCAD*);
- Illustration of the generated mesh (using *Netgen* or *Ansys Student*);
- Execution of the conversion tool;
- Checking on the number of elements imported to *SAP2000* program;
- Vertical stresses for Finite Element model (*SAP2000*);
- Vertical reactions sum for *SAP2000* model.

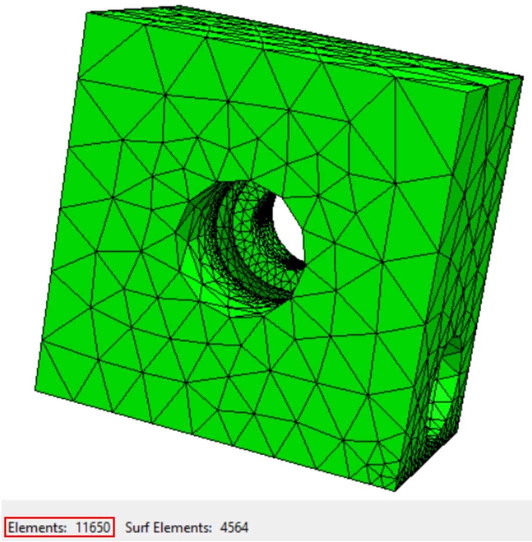


Figure 11: Spiral case 3D mesh composed of 11650 tetrahedral elements (*Netgen*)

3.1 Spiral case of hydropower plant

Figure 10 illustrates a spiral case solid model as well as its volume, in m^3:

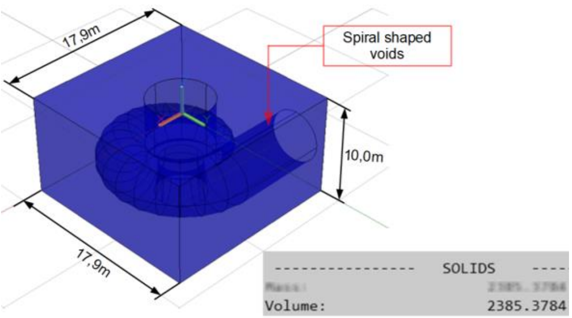


Figure 10: Spiral case solid model and its volume, approximately equal to 2385,4m^3 (*AutoCAD*)

In order to obtain a three-dimensional mesh in Gmsh file format for the geometry presented in **Figure 10**, *Netgen* application has been used, since it is an *Open source* application as well as free when used for non-commercial purposes. However, it is possible to use any other FE solver that also consents exporting 3D meshes in Gmsh format.

In **Figure 12** the execution of the developed conversion tool can be seen. Note for the creation of 11650 solid elements:

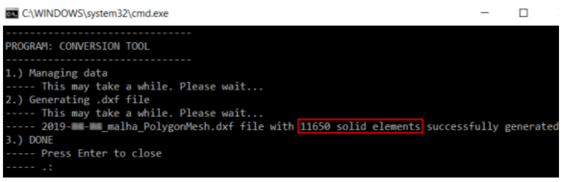


Figure 12: Conversion tool execution (11650 solid elements created)

Figure 13 shows the importation of the generated DXF file to *SAP2000*:

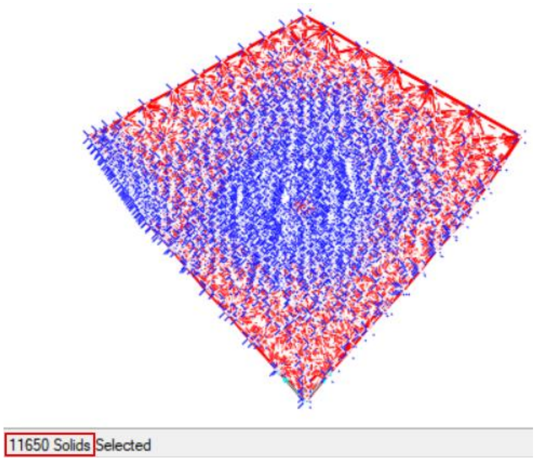


Figure 13: Converted mesh imported to *SAP2000* (11650 solid elements selected)

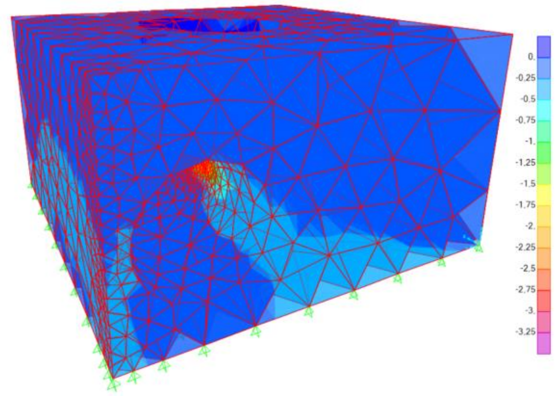


Figure 14: Vertical stresses for FE model, in MPa (*SAP2000*)

In **Figure 15**, the red marked result reveals itself to be very close from the expected one, 2385,4m^{3}x25kN/m^3=59635kN (0,993 ratio):

Base Reactions

File View Edit Format-Filter-Sort Select Options

Units: As Noted
Filter:

OutputCase	CaseType Text	GlobalFX KN	GlobalFY KN	GlobalFZ KN
DEAD	LinStatic	1,651E-11	-3,605E-11	60057,361

Figure 15: Finite Element model's vertical reactions sum (*SAP2000*)

3.2 WES Spillway

Figure 16 illustrates a WES Spillway solid model (geometry considered as referred in Rosa, 2015) as well as its volume, in m^3:

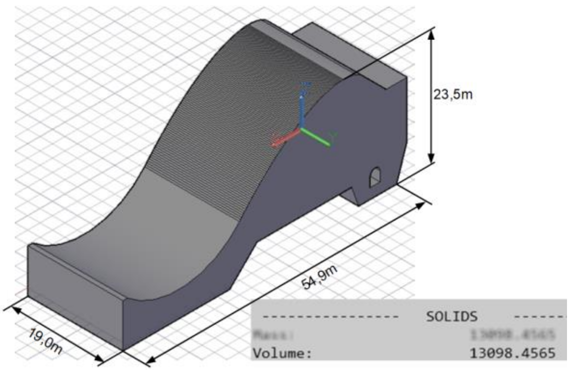


Figure 16: WES spillway solid model and its volume, approximately equal to 13098,5m3 (*AutoCAD*)

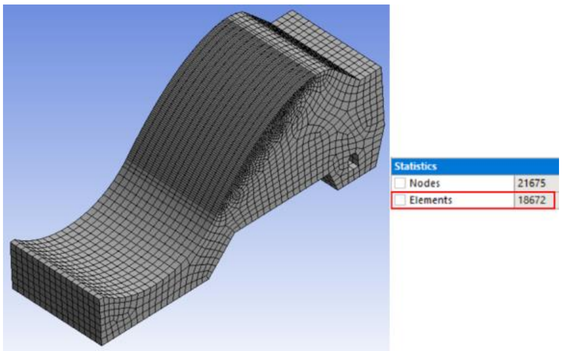


Figure 17: Mesh composed of 18672 triangular prismatic and hexahedral elements (*Ansys Student*)

In **Figure 18** the execution of the developed conversion tool can be seen. Note for the creation of 18672 solid elements:

```
C:\WINDOWS\system32\cmd.exe
-----------------------------
PROGRAM: CONVERSION TOOL
-----------------------------
1.) Managing data
----- This may take a while. Please wait...
2.) Generating .dxf file
----- This may take a while. Please wait...
----- 2019-[redacted]_PolygonMesh.dxf file with 18672 solid elements successfully generated
3.) DONE
----- Press Enter to close
----- .:
```

Figure 18: Conversion tool execution (18672 solid elements created)

Figure 19 shows the importation of the generated DXF file to *SAP2000*:

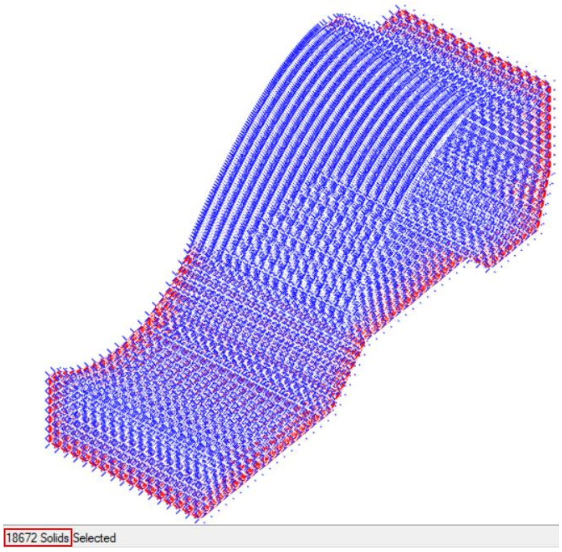


Figure 19: Converted mesh imported to *SAP2000* (18672 solid elements selected)

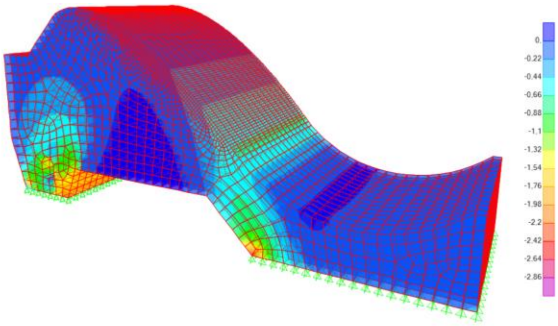
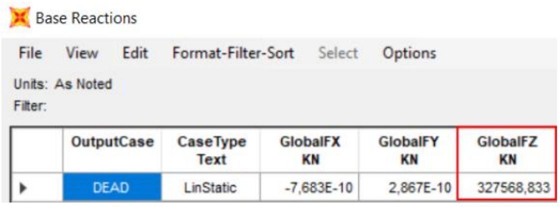


Figure 20: Vertical stresses for FE model, in MPa (*SAP2000*)

In **Figure 21**, the red marked result reveals itself to be very close from the expected one, 13098,5m^{3}x25kN/m^3=327462,5kN:



Base Reactions

File View Edit Format-Filter-Sort Select Options

Units: As Noted
Filter:

	OutputCase	CaseType Text	GlobalFX KN	GlobalFY KN	GlobalFZ KN
▸	DEAD	LinStatic	-7,683E-10	2,867E-10	327568,833

Figure 21: Finite Element's model vertical reactions sum (*SAP2000*)

4 Final considerations

4.1 Conclusion

The thesis was intended to study the interoperability between *AutoCAD* and *SAP2000* for the case of models whose geometries are composed of three-dimensional solid elements.

A new conversion tool was implemented in the scope of the thesis, using the programming language *Python*, as well as DXF file format, for the data transfer between drawing software *AutoCAD* and structural analysis software *SAP2000*. This proved to be an efficient tool in the design procedure of complex structures, thus contributing to reducing overall design time and improving accuracy of computation models.

4.2 Future work

In future studies and work to come, the following points are to be considered:

- Make the developed conversion tool more efficient, lowering its computing time;
- Extend the application domain of this tool to meshes whose elements are of greater order than one (quadratic meshes);
- Increase the range of file formats that can be interpreted by the conversion tool;
- Evaluate in a quantitative way the degree of quality of the created meshes.

References

ANSYS, Inc., *Workbench User's Guide*, SAS IP, Inc., United States of America, 2013.

Araújo, R.R., *Elementos finitos convencionais versus MITC na análise estrutural de lajes moderadamente espessas*, Dissertação de Mestrado, Instituto Superior Técnico, Lisboa, 2013.

Arnold, N., *NETGEN/NGSolve Manual*, 2013.

Autodesk, Inc., *AutoCAD 2013 User's Guide*, Autodesk, Inc., 2012.

Autodesk, Inc., *AutoCAD 2012 DXF Reference*, Autodesk, Inc., San Rafael, United States of America, 2011.

Computers and Structures, Inc., *CSI Analysis Reference Manual*, Computers and Structures, Inc., Berkeley, California, United States of America, 2011.

Geuzaine, C., Remacle, J.F., *Gmsh Reference Manual*, 2011.

Liu, G.R., Quek, S.S., *The Finite Element Method*, *1st edition*, Butterworth-Heinemann, Singapore, 2003, ISBN 0 7506 5866 5.

Novak, P. *et al*., *Hydraulic Structures*, 4th edition, Taylor & Francis, New York, United States of America, 2007, ISBN 978-0-203-96463-7.

Pereira, O.J.B.A., *Introdução ao Método dos Elementos Finitos na Análise de Problemas de Planos de Elasticidade*, Análise de Estruturas II, Instituto Superior Técnico, Lisboa, 2005.

Reis, L.M.R., *Aplicação de Algoritmos para Geração Automática de Malhas de Elementos Finitos Hexaédricos*, Dissertação de Mestrado, Instituto Superior Técnico, Lisboa, 2014.

Rosa, R.H., *Estabilidade e Dimensionamento Estrutural de um Descarregador de Cheias em Soleira Espessa do Tipo WES*, Dissertação de Mestrado, Instituto Superior Técnico, Lisboa, 2015.

Smith, I.M., Griffiths, D.V., *Programming the Finite Element Method*, 2nd edition, John Wiley & Sons, Manchester, England, 1988, ISBN 0 471 91552 1.

Web pages:

Ondrej, Computers and Structures, Inc., *Required topology of 6x3 open polygon mesh in AutoCAD for importing as solid element into SAP2000*, 2014, https://wiki.csiamerica.com/display/sap2000/Required+topology+of+6x3+open+polygon+mesh+in+AutoCad+for+importing+as+solid+element+into+SAP2000, consulted in November 2018.