XXXVII IBERIAN LATIN AMERICAN CONGRESS
ON COMPUTATIONAL METHODS IN ENGINEERING
BRASÍLIA - DF - BRAZIL

COMPARISON BETWEEN PROPRIETARY AND FREE PROGRAMS IN THE FIELD OF FEM STATIC SIMULATION

Guilherme Celeghini Ferreira

guilherme.celeghini@gmail.com

Pontifícia Universidade Católica de Minas Gerais

Av. Dom José Gaspar, 500 - Coração Eucarístico – Belo Horizonte – Minas Gerais - CEP 30535-901

Marilia Mansur Ferreira de Resende Chaves

marilia.chaves@sga.pucminas.br

Pontifícia Universidade Católica de Minas Gerais

Av. Dom José Gaspar, 500 - Coração Eucarístico – Belo Horizonte – Minas Gerais - CEP 30535-901

Jánes Landre Junior

janes@pucminas.br

Pontifícia Universidade Católica de Minas Gerais

Av. Dom José Gaspar, 500 - Coração Eucarístico – Belo Horizonte – Minas Gerais - CEP 30535-901

Abstract. *Programs that utilize the finite element method (FEM) have as purpose the virtual analysis of real problems, and there is a search for those with better results reliability at lower cost. Hence, the comparison between free and proprietary FEM platforms was proposed, using data from an experimental test to feed the analysis and compare it with the numerical results. It has been noted that in the studied point the variations in relation to experimental data were below 15% and under 6% in comparison to analytical data. Consequently, as to the accuracy of static analysis results, it was possible to verify that free programs have similar reliability as proprietary ones.*

Keywords: *finite element, free software, proprietary software.*

1 INTRODUCTION

The finite element method (FEM), as Rao (2004) states, ascended from the need to solve highly complex problems, exchanging these for simpler ones, but with similar results. Felippa (2004) notes that the FEM was only to have a commercial application in industry and in the solving of engineering problems in the late fifties and early sixties, with the developer Professor Ray Clough. Barkanov (2001) declares that in the beginning, such software was limited to computers in the state of the art, which were held only by large enterprises and aerospace, automotive, military and nuclear industries. However, in the late seventies, the FEM tools began to gain popularity due to technological development in the information area.

According to Hughes (2000), since the appearance of the first FEM program, which had a free distribution policy, several other commercial software based on this arose. Rolandsson et al. (2010) and Gaff et al. (2012), emphasize advantages attributed to free software due to its applicability in business and academia, and also for their low cost, ease of editing and support provided by user communities. Galeano et al. (2007) says that the solution of the complex linear system, which is a domain of multiple degrees of freedom, requires high processing capabilities, which inflates the cost for the method implementation; which shows another advantage of free programs. Its application allows a utilization flexibility, the FEM dissemination and consequently the reduction of licensing costs.

Brito (2011) shows in his study of free software, evidence of the importance of the search for knowledge and digital inclusion. Stefanuto and Salles (2005) addresses the philosophy of free software, which challenges the software industry for committing appropriation of knowledge. In this context, there is a loss attributed to the use of proprietary software by restricting the expansion of knowledge, whereas the free software distribution enables the exchange of knowledge among users, all of which can contribute to development of their own code; enhancing the knowledge exchange.

This paper aims to compare free and proprietary software, applied to the FEM, assessing their applicability in solving physical engineering everyday problems. The comparison was made for static analysis, with reference to experimental results. Thereupon, the results obtained are presented and the advantages and disadvantages associated with the use of each software are displayed.

1.1 FEM utilized software

Present in this article one finds the comparison between five different FEM programs, namely: CalculiX, Elmer, Z88, OptiStruct and NX Nastran; where the first three are free and the later proprietary. Dhondt and Wittig. (2016) claims that the CalculiX is a free program developed by Guido Dhondt and Klaus Witting, which enables the construction, calculation and post-processing of three-dimensional structural problems by MEF.

According to CSC - IT Center for Science (2016), Elmer is a program initiated in 1995, that turned into open source in 2005. The program's aim is to be a platform for multi-physics simulations, having structural mechanics, fluid dynamics, electromagnetics, heat transfer, acoustics and others solution modules.

Rieg (2015) reports that the Z88 was created at the University of Bayreuth, Germany, and is a complete FEM platform, having solvers, pre- and postprocessors. This, however, does not have an open license and it is only free distributed. It has a friendly and easy to use interface, managing to perform linear and nonlinear static, free vibration, and thermal analysis.

2 METHODOLOGY

In order to grant adequate reliability, parameters of nine cantilever tests were used in the comparison between the FEM platforms. Three bars of the same material and of similar geometry were machined. Two rosette gauges were placed on each one of the bars, these being positioned in a region that is greatly influenced in flexion, however, away from a possible FEM singularity zone. After the surfaces and strain gages preparation, installation was performed at 45 mm from the second constraint hole. The tests were performed with the bars constrained by two screws and another was applied on the free end, for fixing the masses as shown in Fig. 1. Three masses were selected (*a*: 0.271 kg, *b*: 1.338 kg and *c*: 2.502 kg), they were applied one at a time, making at least sixty seconds of data gathering each time.

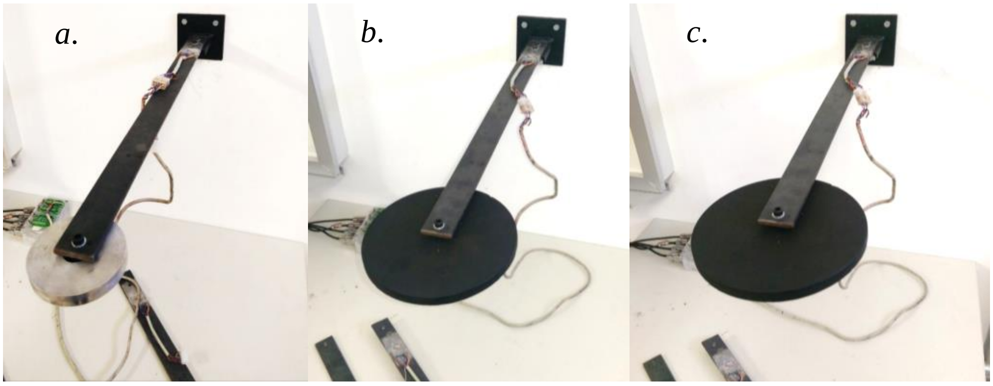

Figure 1. Experimental tests

The experiments provided stresses for comparison with the theoretical and numerical values, and fed simulations with the Poisson ratio and bulk density of the material. To construct the model, the three bars were dimensionally checked, accordingly to the JCGM (2008), thus the measures of a virtual medium bar was achieved. Hence, with all data necessary to carry out the analytical and FEM analysis, the theoretical and numerical portion of the work began.

The analytical calculus was held as shown in Beer, Johnston and DeWolf (2009).

In order to reduce inequities between programs, a single mesh with the same boundary conditions for all analyzes was used. A convergence study, with elements between 10 mm and 0.5 mm, was conducted to determine the optimum element size. Through this study and targeting an improvement in results and computation time, a mesh adopting symmetry conditions was created, using second order tetrahedral elements with 1 mm dimensions.

After a boundary conditions review, it was concluded that as the objective of this study was the stress comparison, the consideration that the bar was constrained only in the screw holes proved to be appropriate; what was done directly to the nodes, without using rigid elements. The restriction was made for three translational degrees of freedom for the bolt holes, translation in Z axis and rotation of X and Y in the symmetry face; and the mass load value in Newtons divided equally in the load application hole.

Relaying on the work of Galeano et al. (2007) and practical experience of the authors, it was decided to use for the comparison OptiStruct and NX as proprietary and CalculiX, Elmer and Z88 Aurora as free programs. After the solution of the three cases for each of the five platforms, the results were acquired and worked up in order to quantitatively compare them.

3 RESULTS AND DISCUSSION

In these topic the results obtained during the work are presented.

3.1 Beam dimensional analysis

The dimensional analysis resulted in the bar in Fig. 2 and its measures are shown in Table 1. In the row "Real" the dimensions of the physical bar are presented, and in the "FEM" row measures adopted in the virtual model are displayed.

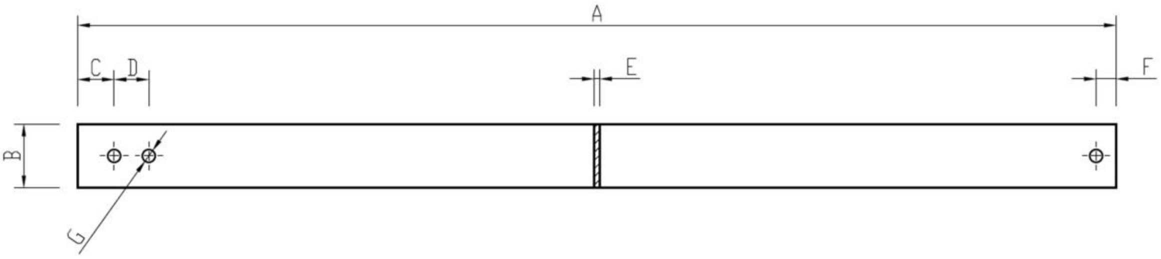

Figure 2. Virtual medium bar

Table 1. Real and FEM beam dimensions

	A	B	C	D	E	F	G
Real (mm)	570 ± 3	38,00 ± 0,29	20,05 ± 0,76	20,32 ± 0,52	6,27 ± 0,27	12,50 ± 0,20	7,00 ± 0,23
FEM (mm)	570,0	38,0	20,0	20,0	6,3	12,5	7,0

Subsequent to the dimensional analysis, the mass of the three bars were measured and with the coupling of these results to the bars volume, the material bulk density of 7879 ± 118 kg/m³ was obtained.

3.2 Experimental static analysis

The rosettes provided the bars specific deflection, and with the 0° and 90° direction, the Poisson's ratio of 0.26 was found. With these data the von Mises stress was achieved. Hence, the reference stresses for the FEM programs were 5.07 ± 0.31 MPa for the experiment "a", 26.88 ± 1.10 MPa for the "b" and 50.13 ± 1 47 MPa for test "c".

3.3 Mesh convergence study

The second order tetrahedral mesh convergence study, with elements between 10 mm and 0.5 mm, generated in NetGen is shown in Fig 3, which compares the results with the analytical response. The graph shows the variation of the bar stress between the fixing and loading screws when subjected to a loading in its free end; for each one of the six meshes and for analytical data. Due to the singularity problem presented by FEM stresses calculation in corners, the stress values until a distance of 20 mm were ignored.

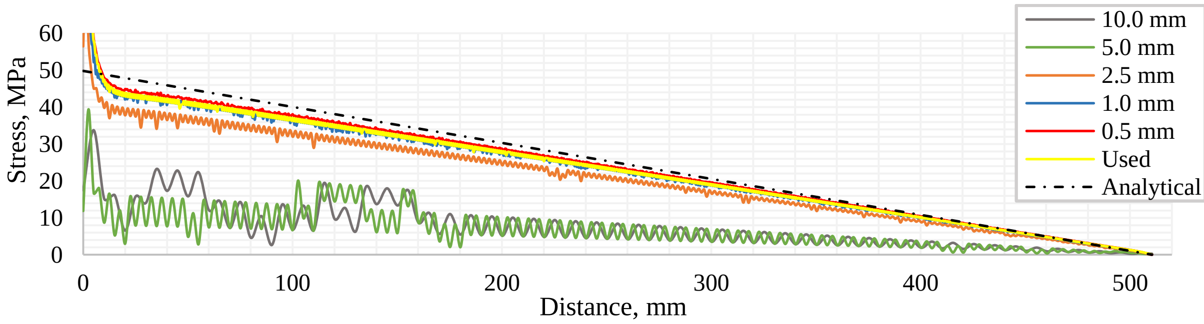

Figure 3. Elemental stress result along bar for same load and different meshes

Observing the results on the bar's region of interest, in meshes with elements sizes of 5 and 10 mm errors were higher than 60.0% and 18.9% for 2.5 mm elements. The 0.5 mm elements mesh showed the best results, 6.0% error, and 1 mm the second best (8.95%). Due to this difference of less than 3.0% and processing costs, the latter was adopted, and at it symmetry was applied, improving the results, 7.8% error, and simplifying processing.

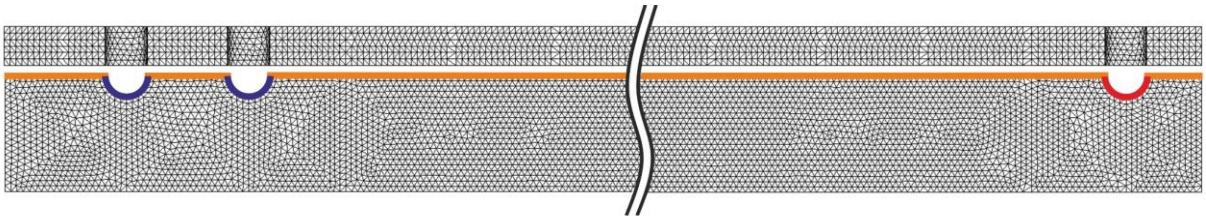

Figure 4. FEM mesh generated for all platforms through NetGen

Hence, the final mesh was achieved with 382,077 second order tetrahedral elements with 1 mm and 576,264 nodes, as presented on Fig. 4. In this the blue portion denotes the nodes that have all degrees of freedom restricted, orange are the symmetry condition nodes and in red are the load application ones.

3.4 Calculated stresses via FEM programs

Using the experimental data, the mesh created and elastic modulus of 200 GPa (ASM International, 2005), the simulations were held and the stress distribution along the bar was reached. The stresses acquired in OptiStruct, NX and CalculiX were nodal stresses, oppose to Elmer and Z88, whose elemental stresses were obtained. In the graph of Fig. 5, one finds these values at the upper side along the bar symmetry profile, starting from the second fixing hole until the load application hole. Three groups of curves are presented a, b and c, where these relates to the masses used on the experiments, as stated earlier in this paper.

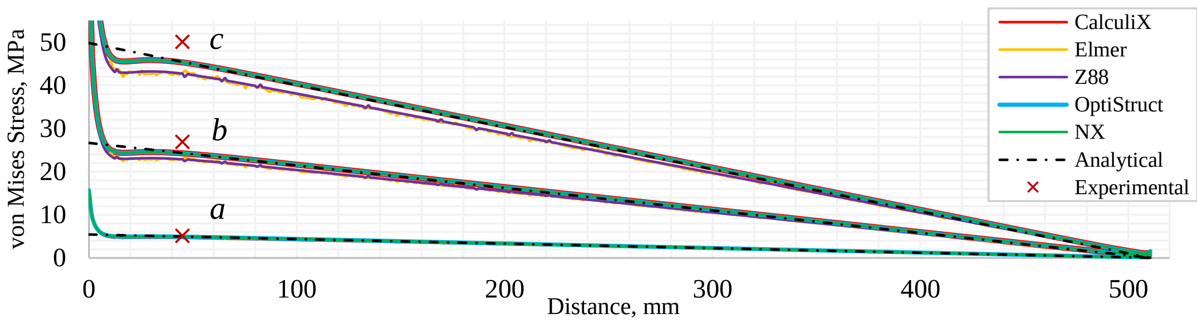

Figure 5. Stress at the upper side along the bar symmetry profile

Due to the difficulty in interpreting the results, thanks to the low variance in results, a comparison between the data was then performed. The divergence between the calculated analytical and FEM stress in relation to the experimental values at a distance of 45 mm, strain gage position, are presented for the three experiments in Table 2.

Table 2. Divergence between calculated analytical and FEM data relative to experimental stress

	Analytical	OptiStruct	NX	CalculiX	Elmer	Z88
a	2.82 %	3.04 %	3.10 %	3.04 %	8.50 %	8.52 %
b	9.57 %	9.77 %	9.83 %	9.77 %	14.86 %	14.88 %
c	9.33 %	9.53 %	9.59 %	9.53 %	14.63 %	14.66 %

4 CONCLUSION

From the research that has been conducted, one concludes that free FEM programs can be so assertive as proprietary programs regarding linear static simulations. In Table 2 one perceives that in all cases the deviation was less than 15% in comparison to the experimental result, noting that Elmer and Z88 got a greater divergence because the elemental stress was utilized, instead of nodal. It’s also notable that when comparing the results from FEM against the analytical, the divergence in all three cases - a, b and c - were equal for each software; 0.22% for OptiStruct; 0.28% for NX; 0.22% for CalculiX; 5.85% for Elmer and 5.86% for Z88. These comparisons endorse the applicability of free FEM software.

The use of FEM programs is increasing, and these are no longer limited to engineering applications. For instance, Lotti et al. (2006) points out how FEM can be applied in the field of Orthodontic analysis. Aiming at a greater method popularization, it is important to spread the usage of free FEM software as a mean to enable researches of all areas to benefit of such tool.

ACKNOWLEDGEMENTS

The authors gratefully acknowledge Vibracon Engenharia for their resources support, FAPEMIG (Research Support Foundation of Minas Gerais) and CNPq (National Council for Research and Development) for their financial support.

REFERENCES

ASM International, 2005. *ASM Handbook Properties and Selection: Irons, Steels, and High-Performance Alloys*, Vol. 1, 1618 p.

Barkanov, E., 2001. *Introduction to the Finite Element Method*, Institute of materials and structures, Faculty of civil engineering, Riga, 70 p.

Beer, F. P., Johnston, E. R., & DeWolf, J. T., 2009. *Pure bending*. In: Beer F. P., Johnston E. R., DeWolf J. T. *Mechanics of Materials*. Ed. 5. London: McGraw-Hill, pp. 208-306.

Brito, Antônio Carlos de Sousa, 2011. *O software livre gráfico na arquitetura: uma experiencia de avaliação*. Dissertation, Universidade Federal da Bahia.

CSC - IT Center for Science, 2016 *Elmer*, Available at <csc.fi/elmer>, Accessed in 11 July 2016.

Dhondt, G. & Wittig, K., 2016. *CalculiX: A Three-Dimensional Structural Finite Element Program*, Available at <calculix.de>, Accessed in 11 July 2016.

Felippa, C., 2004. *Introduction to Finite Element Method*, Department of Aerospace Engineering Sciences and Center for Aerospace Structures, University of Colorado, Boulder, EUA.

Gaff, B. M. & Ploussios, G. J., 2012. Open Source Software, *Computing and the Law*, IEEE Computer Society, pp. 9-11

Galeano, C., Mantilla, J., Duque C. & Mejía, M., 2007. Herramientas de Software con Licencia Pública General para el Modelado por Elementos Finitos, *DYNA – J. of the Facultad de Minas*, Universidad Nacional de Colombia, Vol. 153, Medellin, Colombia, pp. 313-324.

Hughes, T. J. R., 2000. *The finite element method: Linear static and dynamic finite element analysis*, Dover Publications INC, Mineola, Nova York.

Joint Committee for Guides in Metrology, 2008. *JCGM 100:2008: Evaluation of measurement data - Guide to expression of uncertainty in measurement*, Ed. 1.

Lotti, R. S. Machado, A. W., Mazzieiro, E. T., & Landre Jr, J., 2006. Aplicabilidade científica do método dos elementos finitos, *Revista Dental Press Ortodontia e Ortopedia Facial*, Maringá, Vol. 11, No. 2, pp. 35-43.

Rao, S. S., 2004. *The Finite Element Method in Engineering*, 4 ed, Ed. Elsevier Science & Technology Books, Miami, 663 p.

Rieg, F., 2015. *Z88Aurora: Theory manual*, V. 3, 183 p.

Rolandsson, B., Bergquist, M. &Ljungberg, J., 2010. Open Source in the Firm: Opening up Professional Practices of Software Development, *Research Policy*, Elsevier, pp. 576-587.

Salles Filho, S., De Lucca, J. E., Stefanuto, G. N., & Alves, A. M. 2005. O impacto Software Livre e de Código Aberto (SL/CA) nas Condições de Apropriabilidade na Indústria de Software Brasileira. *XI Seminário Latino-Iberoamericana de Gestión Tecnológica*.