FINITE ELEMENT MODELS FOR STRUCTURAL DESIGN OF POWER TRANSFORMERS - 5th IBERIAN CONFERENCE ON STRUCTURAL INTEGRITY

L. M. C. Seixas[1], S. M. O. Tavares[1, 2, *], P. M. S. T. de Castro[1]

[1]FEUP - Faculdade de Engenharia da Universidade do Porto, Rua Dr. Roberto Frias, Campus FEUP, 4200-465 Porto, Portugal
[2]Efacec, Parque Empresarial Arroteia Rua Poente, 4466-952 S. Mamede de Infesta, Portugal , Portugal
[*]Corresponding author: sergio.tavares@fe.up.pt

ABSTRACT

The communication concerns the implementation of open source Salome-Meca FEM software for structural calculation of transformers, including pre-processing, calculation and post-processing. The solver component is Code Aster. The main aim of the work is to develop routines for automatic calculation of FEM models with minimal interaction with the user. From the input data, this algorithm creates the necessary files for the study and performs the pre-processing, where the geometry and the mesh are generated from Python code in the environment of Salome-Meca. After this process, the calculation with functions of Code Aster is performed. Finally, the user is able to visualize the results in ParaView module or output data processing may be executed. This architecture is applied to panel analysis. In panel analysis the automatic calculation of simple and stiffened planar panels was studied. The comparisons of these results with theoretical ones or obtained using other software, show that this routine achieves results quickly and with good accuracy. Also, the analysis to structurally optimize panels was performed using these algorithms.

KEYWORDS: Power Transformers Tanks; Structural Design; FEM Models; Reinforced Panels

INTRODUCTION

Finite element models have a preponderant role today in the structural analysis of mechanical components. Simulation of the behaviour of different geometries, materials and types of application and their integration both at the academic and industrial levels allows for the large-scale development of products that in the past would be outside the application spectrum. Code Aster / AsterStudy is a free open source software and used by multiple industrial sectors. This communication explores its capabilities, implementing practical examples relevant for the transformers design.

The Code Aster software, where "Aster" stands for *Analyzes of Structures et Thermomechanique pour des Études et des Recherches* (Structural and Thermomechanical Analysis for Studies and Research) is a mechanical calculation program, developed by EDF (*Électricité de France*) since 1989, initially dedicated to structural calculation of nuclear reactors. Salome-Meca integrates the Salome pre- and post-processing features, with calculation by finite elements with Code Aster. This program is distributed by several platforms including by SimulEase that developed a free version compatible with the Windows 10 operating system, [1].

FRAMEWORK FOR A FINITE ELEMENT ANALYSIS ROUTINE

An architecture for an efficient analysis routine was developed in order to create finite element programs capable of giving the outputs without manual pre-processing operations. For repetitive analysis when only some parameters are changed in design, this type of approach allows faster and optimized finite element analysis.

Based on the same procedures that a typical analysis, this routine must follow input and preparation of the data, files management, pre-processing, solving and a post-processing. All these modules, except for the input, must be automatic and sequential. Figure 1 presents a flowchart describing all steps that is performed through the routine.
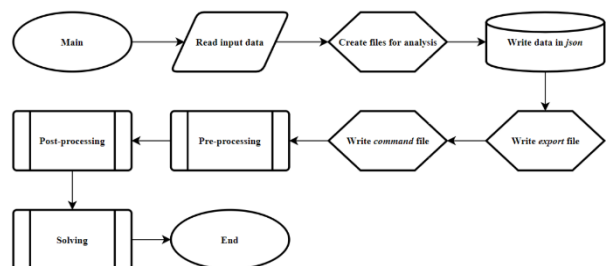

Figure 1. Flowchart for the implementation of the finite element analysis routine.

2.1. Input and files manager

The objects on the created routine are organized in classes, each class has his own attributes and methods. The classes necessary are dependent of the implementation and purpose of the routine, however, typical data are dimensions, mesh properties, material properties, loads and boundary conditions and criteria for design. This routine is capable of being automated mainly due to the management of files that the software Salome-Meca uses for analysis. The pre-processing and post-processing can be fully done with Python routines implemented on the software and the solving is achieved with a *command* file created with specific functions from Code Aster. There is also the necessity to identify the batch files to execute Salome-Meca and Code Aster.

The communication between each step, is possible by different files with information. The *json* file is where all classes with respective attributes are saved and can be accessed by the different routines. Code Aster input and output files are called med files. The input *med* file contain information about the geometry and mesh and the output has the calculated results. Finally, there is also the *export* file that allows Code Aster to find the localization of each file.

2.2. Pre-processing

The pre-processing is done by a Python routine implemented in Salome-Meca environment. With a parametrized geometry, it is possible to create the model with the input data, and subsequently the mesh. The mesh model is saved on a *med* file that will later be read by Code Aster.

2.3. Solving

The solving is achieved by running a command routine in Code Aster environment. The *med* file from the preprocessing with mesh information is read by the software and a finite element model, loads, boundary conditions and material properties are applied to the model. After finishing the structural analysis, the routine writes for another *med* file the output results.

2.4. Post-processing

After the finite element analysis, several types of post-processes can be implemented. Typical examples are viewing the results graphically with Paraview module from Salome-Meca or filtering the results with some pre-imposed criteria. Either way, these processes are achieved with Python routines.

PANELS

The structural analysis of transformer tank panels was addressed, creating a routine to perform panel analysis using a Python algorithm integrating the Salome-Meca finite element software and performing, in a user-friendly way, the complete analysis of these components. It was also possible to implement the analysis of several models of panels simultaneously and a treatment of the results obtained in order to identify which model is preferable according to the applied criteria. In this way, it is possible to optimize both the analysis and the final model, since, easily and in a short time, the results of the finite element calculations are obtained, as well as the design that fits the indicated criteria. Different constructions of tank reinforced panels available in the implemented routine were analysed. The typology of these constructions considers, first, whether the panel is curved or flat and whether the panel includes vertical reinforcements or horizontal reinforcements. For each type of construction, various types of validation of examples will be presented and compared either with literature or with analysis in other finite element software. Parametric analyses were also carried out in order to assess the influence of different factors on results. Finally, some examples of panel optimization were carried out using the routine created in order to validate their ability to analyse different constructions, according to adopted criteria, which minimize the amount of material needed while maintaining a mechanical behaviour within the intended intervals. A routine was developed to obtain a finite element analysis of panels of several models simultaneously, in which the user will only have to supply the input data to obtain the displacement and stress fields. This algorithm allows to perform structural calculations more efficiently and in shorter time, since the implementation in the Salome-Meca software of each model separately and manually implied that the user performed all the analysis processes from the beginning. Therefore, from this routine, all pre-processing, calculation and post-processing steps are carried out automatically through already established algorithms and parametric modelling.

3.1. Formulation

The formulation of shell elements in Code Aster is similar to the classic shell elements; however, they account for the curvature of the geometry. In this case, Code Aster limits its use only to triangular geometry elements with seven nodes (TRIA7) or quadrangular geometry elements with nine nodes (QUAD9), [2]. This discretization is also carried out using Heterosis elements, avoiding the shear locking phenomenon that can happen when the thickness of the shell is much smaller than the radius of curvature. These elements are characterized by the border nodes having the three degrees of freedom of translation and rotation, while the middle node of the element only has degrees of freedom of rotation, [3]. The integration used in this modelling is the Hammer integration with seven points in the case of triangular elements and Gauss integration with nine points in the case of quadrangular elements. It is also noted that this algorithm uses selective reduced

integration to also prevent locking effects. Contrary to the uniform reduced integration, Gaussian integration suitable for the calculation of the stiffness matrix is used, however, it reduces the degree used for the calculation of deformations and stresses, [3].

3.2. Panels subjected to internal pressure

A particular case of panels subjected to internal pressure, without stiffeners, as schematically represented in Figure 2 is used in this case study. This example is well documented in the literature for several boundary conditions, [4,5]. Equation (1) allows to calculate analytically the displacement perpendicular to the plane of the plate, [5]. From this equation, it is possible to estimate the moments and stress, used to compare with the finite element results in this case study.

$$w = \frac{16p}{\pi^6 D} \sum_m^\infty \sum_n^\infty \frac{1}{mn[(m/a)^2 + (n/b)^2]^2} \sin\frac{m\pi(x+a/2)}{a} \sin\frac{n\pi(y+b/2)}{b} \quad (1)$$
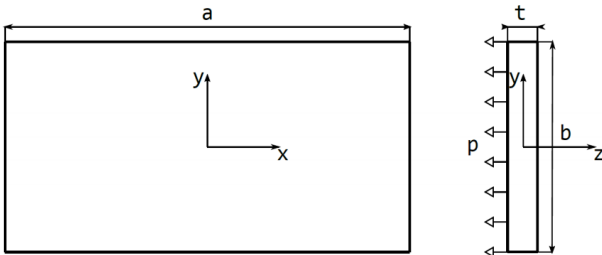
Figure 2. Geometry of the panel subjected to internal pressure.

Using the implemented routine these panels were analyzed and compared with theoretical results for different values of ratio between the length and the width of the panel ($k = a/b$). The displacements and von Mises stresses obtained using finite element analysis and analytical solutions are represented graphically in Figure 3 and Figure 4.
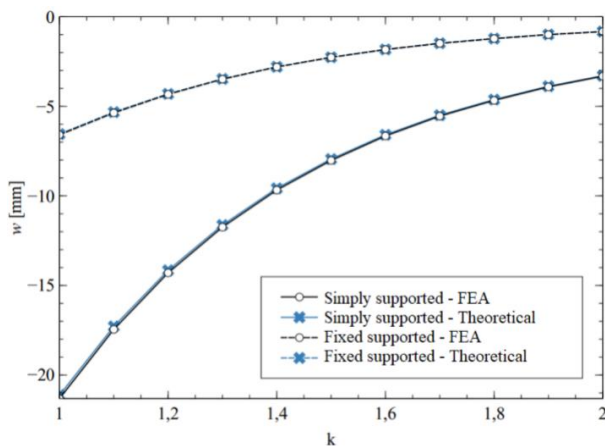

Figure 3. Comparison of displacements for simple panels.
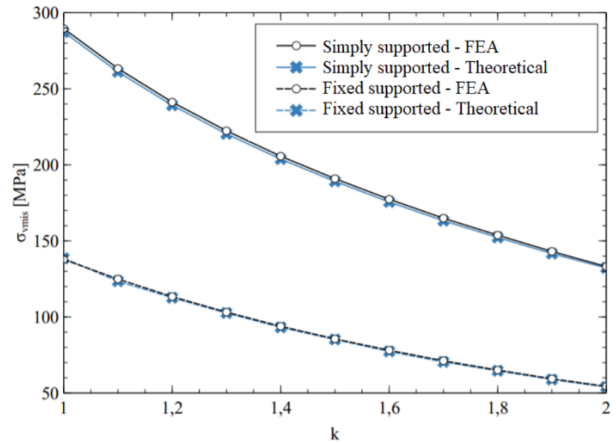

Figure 4. Comparison of equivalent stress for simple panels.

The finite element model achieve very good agreement with the theory, with the relative difference being less than 1% for all ratios both in displacements and von Mises stresses.

3.3. Stiffened panels subjected to internal pressure

Tank transformers rely on stiffened panels in order to reduce the necessary thickness to overcome the loads. Using the routine created for panels structural analysis, it is possible to model several geometries, adding and removing stiffeners, with diverse cross-sections for the stiffeners. Figure 5 gives schematic representation of base geometry, used in the developed routines.

The developed routines are able to create the FEM model for any configuration of a stiffened panel with orthogonal stiffeners. With this approach is possible to estimate the best solution of reinforcement arrangement for a given geometry. The geometries differ in number, thicknesses and height of the stiffeners and using an optimized model one can find a combination of these parameters with good mechanical behaviour and less material.
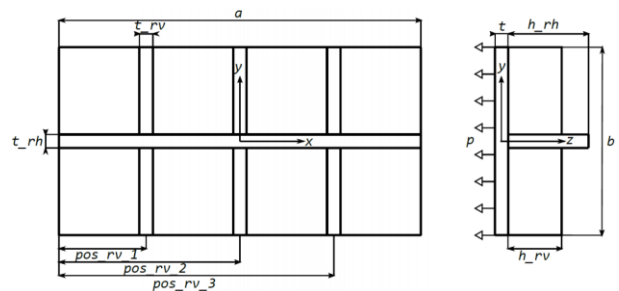

Figure 5. Reinforced panel - generic geometry characterization.

In order to validate the result obtained from Salome-Meca, these were compared to the commercial software ANSYS in the same condition. Figure 6 to 9 presents the results for fixed supporting condition in each software.
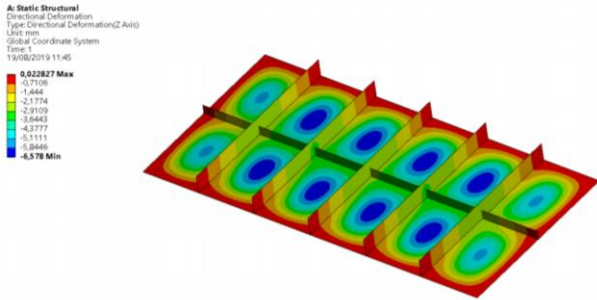
352

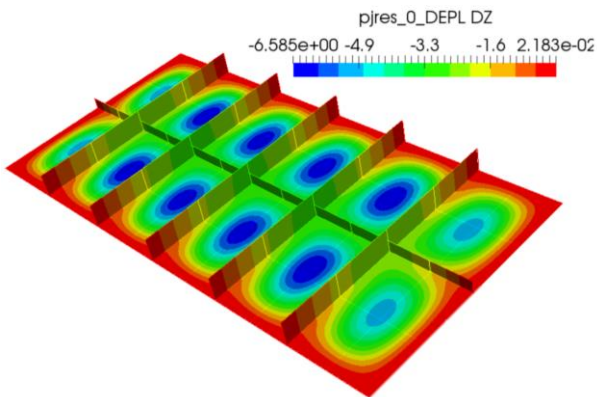*Figure 6. Displacements in ANSYS for stiffened panels.*

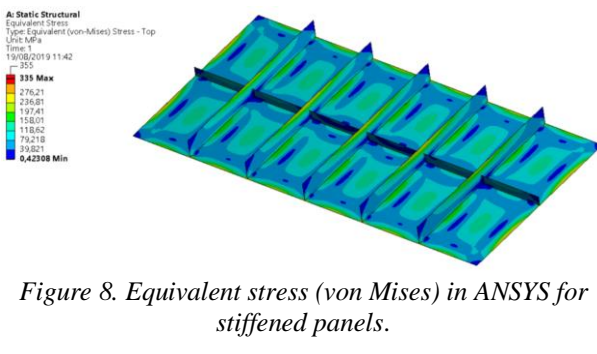*Figure 7. Displacements in Salome-Meca for stiffened panels.*

Figure 8. Equivalent stress (von Mises) in ANSYS for stiffened panels.
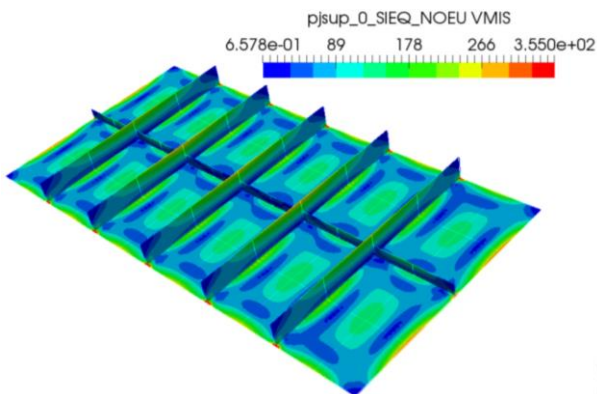

Figure 9. Equivalent stress (von Mises) in Salome-Meca for stiffened panels.

For the displacements normal to the panel, the relative difference between ANSYS and Salome-Meca is about 0.1%. Regarding the von Mises stresses, the distribution of this result seems similar for both softwares. Figure 10 presents the values obtained along a stiffener from the

panel for each software. These results present again small differences, therefore it can be concluded that finite element models implemented in Salome-Meca offers a good solution when compared to other commercial FEM package.
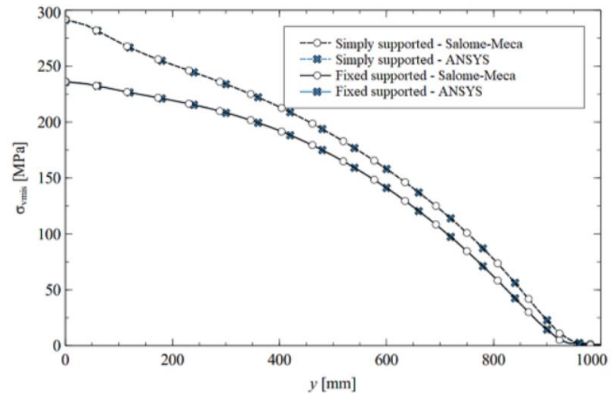

Figure 10. Distribution of von Mises stresses along vertical stiffener.

The post-processing of the panel structural analysis routine is capable of filtering, from the results obtained for different combinations of geometries studied simultaneous, only the ones that meet the design criteria defined. With this feature it is possible to find an optimized model from the models studied with less material in a relatively fast analysis. Figure 11 presents graphically the maximum von Mises equivalent stresses for different combinations of stiffeners where is possible to easily access the ones where the stresses are lower than the limit (yield strength) and have less mass.
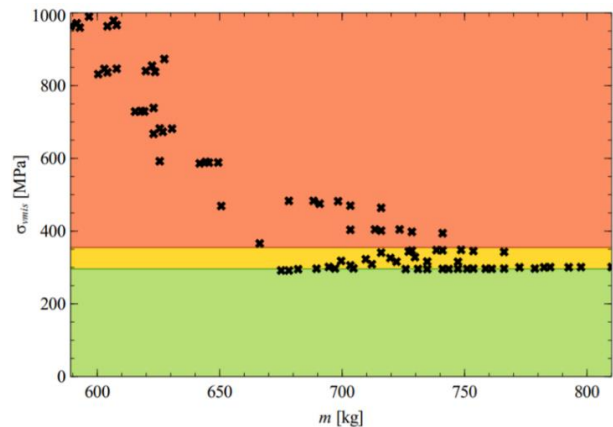

Figure 11. Maximum von Mises stress for different combinations of stiffeners.

CONCLUDING REMARKS

Code Aster and Salome-Meca allow to include routines that perform pre-processing, finite element calculation and post-processing. A generalist architecture of a routine of this type was created, which allows the implementation of different geometries, different meshes, different analysis conditions for calculation, such as formulations, types of analysis, design

constraints and different boundary conditions. An automatic calculation routine was implemented for transformer tanks panels, in which it was observed that it is possible to analyze these models more efficiently and faster. As this routine allows the simultaneous analysis of several geometric models, it is possible to create parametric analyzes and structural optimizations in a short time. Two examples of this routine were validated, the simple flat panels and the reinforced flat panels. In the case of reinforced flat panels, the parameters of the vertical reinforcements and horizontal reinforcement were varied and from this analysis it was possible to proceed with an optimization in which a model with less mass was obtained while maintaining the mechanical behaviour within the allowable limits.

A completely automated approach to structural analysis by finite elements completely using open source resources was demonstrated. With computers allowing ever faster analyses, these routines may be part of the industrial routine and thus allow finite element analysis to be frequent procedures in the design of any component.

REFERENCES

[1] Salome-Meca 2018 for Windows. https://code-aster-windows.com. Assessed on 13-08-2019. (2018)

[2] A.-D. Kudawoo. Note of use of the voluminal elements plates, hulls, hulls SHB, grids and membranes, no. U2.02.01. EDF – Électricité de France. Available https://code-aster.org/doc/default/en/man u/u2/u2.02.01.pdf. (2017).

[3] T. De Soza. Modeling COQUE_3D, no. U3.12.03. EDF - Électricité de France. Available https://www.codeaster.org/V2/doc/v13/en/man u/u3/u3.12.03.pdf. (2015).

[4] S. Timoshenko and S. Woinowsky-Krieger. Theory of plates and shells. Engineering Societies Monographs. McGraw-Hill. (1959).

[5] A.C. Ugural. Plates and shells: Theory and analysis. Fourth Edition, CRC Press. (2017).