CREATING A TOOL FOR STRESS COMPUTATION WITH RESPECT TO SURFACE DEFECTS

O. Sedova[1,a], O. Iakushkin[1,2], A. Kondratiuk

[1] *St Petersburg University*

[2] *Plekhanov Russian University of Economics Stremyanny lane, 36, Moscow, 117997, Russia*

E-mail: [a] o.s.sedova@spbu.ru

Consider the problem of calculating the stresses of a sphere with surface defects for a set of different initial conditions of the problem at hand. Varying materials, size and shape of defects have to be considered. We developed a system form open source components that combines CAD and CAE functions inside one user interface hosted inside portable Docker image.

Keywords: CAE, CAD, FEM, Docker, Jupyter, IDE

Olga Sedova, Oleg Iakushkin, Anna Kondratiuk

1. Introduction

Surface defects frequently lead to stress concentration in various devices and structures. The reasons for their appearance may be aggressive environmental impact during operation, as well as heavy workloads, some of the defects come as a result of the production process.

Local damage on the surface and inside the bodies acts as a stress concentrator, it accelerates the destruction of structural elements, leading to premature failure and / or the need for repairs.

To ensure the strength and reliability of any given structure, it is necessary to take into account the stress concentration near the defects. In this regard, a lot of theoretical, experimental and numerical studies are carried out that consider the effects of various defects on the stresses that arise in structures under various loads and boundary conditions.

A significant part of the work in this area is devoted to plates and rods with single defect in the form of a crack or periodic defects. Note that if there are several damages, but they are located far enough from each other, their number may not significantly affect the stress distribution compared to the case of a single defect. In such situations, it is allowed to consider each defect separately, in other words considering it to be unique in its vicinity.

For micro- and nanoscale products, the stress state in the body affects the change in the shape of the surface, which leads to the appearance of roughness, then the relief of the formed surface can be considered as a set of surface defects. It is important to take into account the effect of surface stresses on the physical properties of the material in case of nanoscale defects.

The finite element method is widely used for the numerical estimation of the stress-strain state in the vicinity of defects and in structures as a whole. It is important to note that some problems are solved analytically.

Our research is trying to bridge the gap between iterative analytical experiments, technical limitations of freely available production tools and opensource packages. To achieve this we are developing a Jupyter based CAE environment capable of covering analytical and practical tasks in this area.

2. Our CAE Environment

We develop a parametric approach to the creation and processing of geometry. Featuring a CAE ecosystem that provides FEM tools for working with GPGPU that allows minimization of the time needed to test an engineering idea under widely varying initial and boundary conditions.

For the end user experience, the system provides an interface that combines interactive web components based on the Jupyter Notebook platform and a programming environment based on the Python language. The system is opensource and can be deployed on to any Linux compatible system thanks to Docker containerization technology.

We build a Docker image environment composed of Conda with Jupyter and a set of OpenSource Python libraries for CAD and CAE with FEM operations [1-3]. Capabileties of our environment are shown in Table 1.

Creating geometry. We bundle Open Cascade [4] to allow user to input any geometry in form of program code. We have developed tools that provide facilities for in-place rendering.

Meshing geometry. We utilize GMSH and FEniCS-dolfin [5-6] to generate meshed geometry triangulation.

Applying forces. We bundle FEniCS to allow user to apply the forces needed for an experiment in form of program code. We have developed tools that provide facilities for in-place rendering.

Testing our system. We consider a linearly elastic thin-walled spherical vessel with an inner radius r and an outer one, R, to the inner surface of which pressure p is applied. On the inner surface of the vessel there are defects - spherical recesses of radius δ, which are immersed in the surface of the vessel to a depth of h, where $h \leq \delta$. The number of recesses on the surface is n. Defects are located along one of the circles of a large circle of the sphere, while they are evenly distributed around this circle. The considered values of n are in the range from 4 to 348.

We performed a task of assessing the stress state of the body near the defects for different numbers of defects n, different notch radii δ, and different depths h.

A finite element model of a thin-walled hollow sphere with radii r = 340 mm and R = 350 mm was constructed in our CAD environment. Spherical recesses were carved on the inner surface. The following notch radii were considered: $\delta = 4$, $\delta = 6$ and $\delta = 8$mm; depths: $h = 2$, $h = 3$ and $h = 4$mm; the number of defects: n - in the range from 4 to 348. For each fixed radius of the notch, a model was first built with the minimum number of defects ($n = 4$), after which their number gradually increased to such $n = N$ (depending on the radius of the notch δ: $N = N(\delta)$), at which the intersection of neighboring recesses was observed.

Since the recesses were located evenly along the circumference of a large circle, by virtue of symmetry, instead of the entire sphere, one eighth of the part was considered, enclosed between three mutually perpendicular planes passing through the center of the shell.

The constructed model geometry was prepared for finite element analysis such as applying loads, finite element breakdowns, and subsequent calculations.

Load is carried out by pressure p, which is applied to the inner surface of the shell: $p = 1$ MPa. As boundary conditions, symmetry conditions are imposed on all side faces of the element under consideration.

Young's modulus of the used material $E = 2.1 \cdot 10$ ^ 5MPa, Poisson's ratio $\vartheta = 0.3$.

It was shown that for all considered δ, with an increase in the number of recesses on the surface, the stresses increase. Moreover, with an increase in the number of defects, the growth of stresses becomes more rapid. However, this pattern is violated with such a large number, at which neighboring recesses intersect.

This results obtained correlate with behavior predicted by analytical calculations and previously performed experiments in closed source alternative software packages, showing correctness of our CAE system behavior. In the Table 2 we compare our opensource solution with commercial alternatives.

3. Drawbacks

For a user of WYSIWYG IDE such tool where coding is required for any shape creation may seem too complicated. Yet for any complicated scenario ability to create objects in a reproducible manner from code may be beneficial.

4. Conclusion

The system we presented allows creation of complex CAD and CAE scenarios. It is easily deployable and allows processing of simple and complicated geometry.

Table 1. A set of various geometry examples with applied forces

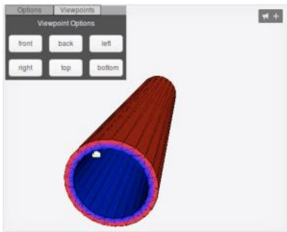
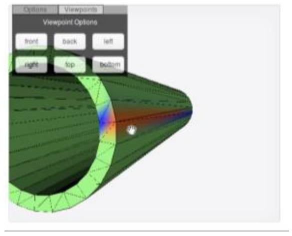
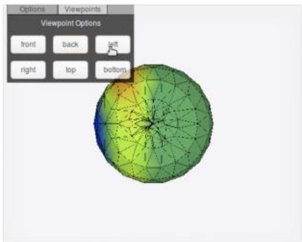
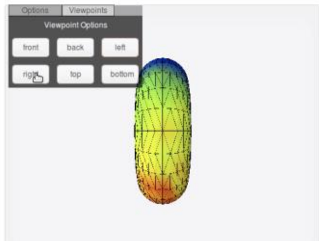
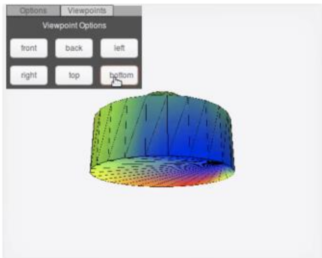
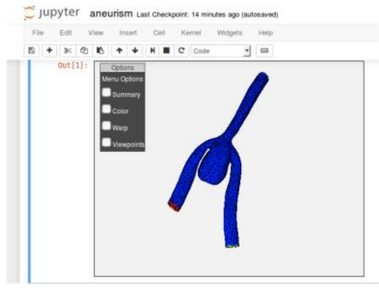
<table>
<tr><td>Pipe geometry</td><td>
</td><td>
</td></tr>
<tr><td>Simple geometry</td><td>
</td><td>
</td></tr>
<tr><td>Complex geometry</td><td>
</td><td>
</td></tr>
</table>

Table 2. Comparison of various CAD tools with CAE capabilities

Compas	SimScale	Solidworks	Our solution	
-	+	-	+	Browser integration
+	-	+	+	Geometry and conditions from code
+	-	+	-	Stand-alone application focus
-	+	-	+	Cloud computation offload

5. Acknowledgement

This research was partially supported by the Russian Foundation for Basic Research grants (projects no. 18-71-00071). The authors would like to acknowledge the Reviewers for the valuable recommendations that helped in the improvement of this paper.

References

[1] Malevanniy D., Sedova O., Iakushkin O. Controlled Remote Usage of Private Shared Resources via Docker and NoVNC. In: Misra S. et al. (eds) Computational Science and Its Applications – ICCSA 2019. ICCSA 2019. Lecture Notes in Computer Science, vol 11622. Springer, pp. 782-791

[2] Iakushkin, Oleg, Anna Kondratiuk, Alexey S. Eremin, and Olga Sedova. "Development of a containerized system to build geometric models and perform their strength analysis." In 3rd International Conference on Applications in Information Technology, ICAIT 2018, pp. 146-149. Association for Computing Machinery, 2018.

[3] Iakushkin, Oleg O., and Olga S. Sedova. "Creating CAD designs and performing their subsequent analysis using opensource solutions in Python." In AIP Conference Proceedings, vol. 1922, no. 1, p. 140011. AIP Publishing, 2018.

[4] T. Paviot, pythonOCC, 3D CAD/CAE/PLM development framework for the Python programming language, PythonOCC – 3D CAD Python

[5] M. S. Alnaes, J. Blechta, J. Hake, A. Johansson, B. Kehlet, A. Logg, C. Richardson, J. Ring, M. E. Rognes and G. N. Wells, The FEniCS Project Version 1.5, Archive of Numerical Software, vol. 3, 2015

[6] A. Logg, G. N. Wells and J. Hake DOLFIN: a C++/Python Finite Element Library in Automated Solution of Differential Equations by the Finite Element Method, Volume 84 of Lecture Notes in Computational Science and Engineering, Edited by A. Logg, K.-A. Mardal and G. N. Wells, Springer, chapter 10, 2012