THE IMPACT OF THE SOLIDWORKS SIMULATION NETWORK QUALITY ON THE ACCURACY OF THE CALCULATIONS

Rudyk Oleksandr Yuhymovych,

Gonchar Volodymyr Antonovych,

Ph.D., Associate Professors

Khmelnytsky National University

Khmelnitsky, Ukraine

yuhymovych@gmail.com

Extreme working conditions of elements of modern structures make it difficult and expensive to perform a full-scale experiment. And setting limit (destructive) loads is a special case. It is not possible to create designs of this type without improving and automating the design process.

Today, the standard is three-dimensional computer aided design (CAD). But there is no universal CAD. Due to the different complexity of the software, the configurations can vary greatly [1]. One of the best varieties is SolidWorks. This is a fully featured CAD system. It is based on a parametric object-oriented methodology. This allows you to get a 3D model from a 2D sketch using simple and effective tools.

One of SolidWorks applications is SolidWorks Simulation. It is based on the finite element method (FEM). FEM is the standard for solving the problems of solids mechanics using numerical algorithms. He took the leading position with the ability to simulate a wide range of objects and phenomena. It is necessary to take into account the inevitable in any numerical approximation of convention and error. Therefore, the question of the correspondence between the computational model and reality is fundamental when using analysis programs. This implies a certain unpredictability of the results, as well as some differences in their interpretation.

FEM is a method of approximate numerical solution of physical problems. It is based on the sampling of an object into a finite set of elements. The finite elements

(FE) have different shapes and sizes. Sampling creates a grid of element boundaries. Their intersections form nodes. The set of all FE and nodes is a finite element model of a deformed body. The choice of the type, shape and size of the FE depends on the type of stress-strain state, shape and load of the body under study.

Within the FE, the following parameters are assigned to it: characteristics of hardness and strength of the material and others. The parameters of displacement, deformation, stress are assigned at the element nodes. With the help of interpolating functions, the values of the corresponding values can be calculated at any point inside the element or at its boundary.

The basic idea behind FEM is that any continuous value can be approximated by a discrete model. This model is built on many piece-continuous functions. They are defined on a finite number of subdomains. Piece-continuous functions are determined by means of values of continuous value in a finite number of points of the considered area. In the general case, the continuous value is unknown in advance and it is necessary to determine its value at some internal points of the region. The discrete model is easy to build. But first, let's assume that the numerical values of this value at each inner point of the area are known. Therefore, the peculiarity of calculating FEM contact problems is that the contact area width is proportional to the size of the FE grid.

The grid created depends on the following factors: the geometry of the part and the size of the model; active grid parameters and management characteristics; global element sizes and tolerances. Errors may occur when analyzing an object. They are related to the density of the grid. Therefore, it is necessary to choose the optimal number and size of FE - the quality of the grid plays a key role in the accuracy of the results.

Aim of the study: To evaluate the influence of the quality of SolidWorks Simulation grid on the accuracy of calculations of the valve of the hydraulic distributor of the Amkodor-208 loader.

At this time, small mobile forklifts are becoming more widespread at construction sites. This is due to their versatility. The presence of a large number of interchangeable working bodies ensures their wide application.

The authors of [2] conducted a static analysis of the valve of the valve. He confirmed his ability to work. Continuation of the study - predicting the loss of its stability [3]: it does not occur. But in these calculations the standard FE grid was applied (the algorithm for creating the Voronoi-Delaunay grid is activated for the subsequent operations of its creation - fig. 1a).

Сетка Детализация

Имя исследования	SimulationXpress_Study (-Default-)
Тип сетки	Сетка на твердом теле
Используемое разбиение	Стандартная сетка
Автоматическое уплотнение сетки	Выкл
Включить автоциклы сетки	Выкл
Точки Якобиана	16 точек
Размер элемента	2.27029 mm
Допуск	0.113515 mm
Качество сетки	Высокая
Всего узлов	82152
Всего элементов	53224
Максимальное соотношение сторон	6.6761
Процент элементов с соотношением сторон < 3	99.2
Процент элементов с соотношением сторон > 10	0
% искаженных элементов (якобиан)	0
Время для завершения сетки (hh:mm:ss)	00:00:09

Сетка Детализация

Имя исследования	SimulationXpress_Study (-Default-)
Тип сетки	Сетка на твердом теле
Используемое разбиение	Сетка на основе кривизны
Точки Якобиана	В узлах
Макс. размер элемента	2.27029 mm
Мин. размер элемента	0.454058 mm
Качество сетки	Высокая
Всего узлов	96403
Всего элементов	62737
Максимальное соотношение сторон	8.4954
Процент элементов с соотношением сторон < 3	99.4
Процент элементов с соотношением сторон > 10	0
% искаженных элементов (якобиан)	0
Время для завершения сетки (hh:mm:ss)	00:00:11

a b

Fig. 1. Grid options

High quality grid is recommended for reliable results. And to create more elements in the area of higher curvature, it is necessary to activate the algorithm of creating a grid based on curvature – fig. 1, b. And for the solution of static problems it is recommended to set the Jacobian check parameter “In knots” [4] – fig. 1, b. When you need to choose the optimal number and size of finite elements. The dependence of the maximum stress in the spool on the size of the grid element is shown in fig. 2.

As can be seen from fig. 2, it is difficult to determine clearly the size of the grid element for calculating the strength of the spool: if it is lowered we will get

unreliable voltages, but the actual ones are governed by the capabilities of the computer - in this case, when the grid element size is 0.5675725 mm, it “hangs”.

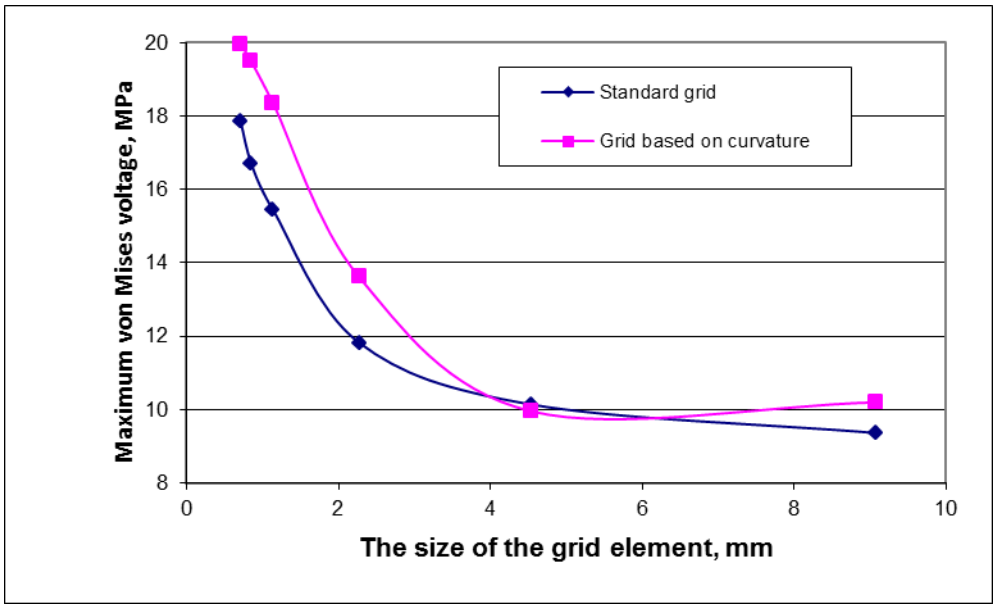

Fig. 2. The dependence of the maximum von Mises stress in the spool on the size of the grid element

What is significant in the performance of this study: the minimum factor of safety margin has decreased by 1.65 times, ie the spool will withstand less load than calculated in [2].

Thus, the thickening of the grid improves the results. However, this results in a sharp increase in computing costs. And at some point, your computer is running out of resources. Therefore, it is necessary to choose tailor-made grids.

REFERENCES

1. Borovyk L.V. Designing by SolidWorks Applications [Electronic resource]. – Access mode: http://elar.khnu.km.ua/jspui/handle/123456789/8462

2. Rudyk O. Yu. Use of information technology for engineering tasks [Electronic resource]. – Access mode: http://elar.khnu.km.ua/jspui/handle/123456789/8628

3. Rudyk O. Yu. Prediction of loss of stability of the valve of the valve of the distributor [Electronic resource]. – Access mode: http://elar.khnu.km.ua/jspui/handle/123456789/6453

4. Blinnikov G.P Information support for calculations in SolidWorks Simulation [Electronic resource]. – Access mode: http://elar.khnu.km.ua/jspui/handle/123456789/8461