13th International Research/Expert Conference
"Trends in the Development of Machinery and Associated Technology"
TMT 2009, Hammamet, Tunisia, 16-21 October 2009

SOLID MODELING AND STRESS ANALYSIS OF THE LOWER SHEAVE ASSEMBLY

Adil Muminović
Mirsad Čolić
Faculty of Mechanical Engineering
Vilsonovo šetalište 9, Sarajevo
Bosnia and Herzegovina

Meho Gutić
BEGroup d.o.o.
Bistrik do br.11, Sarajevo
BiH

ABSTRACT
The lower sheave assembly is a series of elements that make the whole, over which load is transferred from the hooks on the crane. Taking into account responsibility and consequences of these elements (injury of people and economic damage from the fall of cargo) that are possible in the case of a collapse of the element or assembly as a whole it is quite justifiable request for a detailed scrutiny of their validity and security. Classical (conventional) methods include the making of necessary number of prototype elements, which after computation checks are additionaly analysed what includes certain economic and time costs. This paper gives an overview of geometrical modeling of parts and the formation of the lower sheave assembly in CAD/CAE software CATIA V5. Engineering (stress deformation) analysis of carrier sheet metal is made. The aim is to present the methods that provide additional security at the planning and contribute to faster and cheaper development of new products.
Keywords: Crane, lower sheave assembly, geometric modeling, stress analysis

1. INTRODUCTION

The technical product is the result of a series of analysis, synthesis and test processes. The design methodology helps to structure these individual steps and to evaluate the products according to different features. The tool CAD (Computer Aided Design) gives an important assistance in this process. This computer support should, in general, raise the information level of the designer and decrease the, at the moment, considerable time required for obtaining information [4]. The use of the CAD systems focuses on automation of the creation of the CAD geometry, the engineering analysis, and generation of the support information. In order to have an useful system, and demonstrate its functionality, the system will have to operate within an integrated design environment. CAD is becoming a necessary tool for any engineering task. The computers graphics capability and computing power allow designers to fashion and test their ideas interactively in real time without having to create real prototypes as in conventional approaches to design. A typical CAD system involves both design and manufacturing operations (CATIA V5). This paper gives an overview of geometrical modeling of parts and the formation of the structure of the lower sheave assembly in CAD/CAE software CATIA V5. Engineering (stress deformation) analysis of carrier sheet metal is made. The aim is to present the methods that provide additional security at the planning and contribute to faster and cheaper development of new products.

2. MODELING ELEMENTS OF THE LOWER SHEAVE ASSEMBLY

Solid modeling, formation of assembly as well as stress analysis is conducted using CAD / CAE software package CATI V5 in the *Part Design, Assembly Design and Generative Structural Analysis module* respectively. Forming assembly in the CATI software V5 includes previously individually modeling all parts of assembly. Formation of 3D solid model is made on the basis of 2D sketches, which is used to obtain a 3D body by one of the basic functions: Extrude (*Pad*), Revolve (*Shaft*), Sweep (*Rib*) i Loft (*Multi-selections Solid*). Example of the carrier sheet modeling on the basis of the initial 2D sketches feature Extrude *(Pad)* is shown in Figure 1.

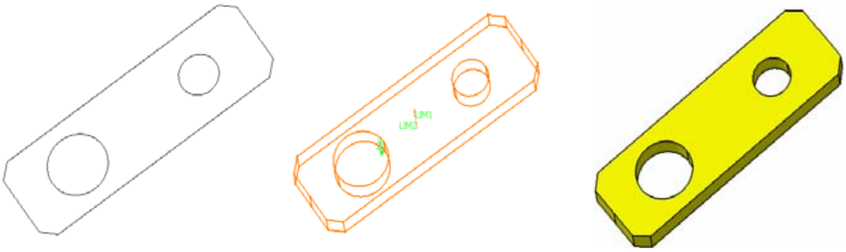


Figure 1. Modeling of the carrier sheet on the basis of the initial 2D sketches

Unambiguously defining of the position of the elements that which form the lower sheave assembly is realized by the function *Coincidence, Offset and Angle*. It is recommended that the listed status should be defined in relation to existing or needed by newly co-ordinate systems of single parts of assembly in case of changes of geometry of parts, which are, as usual, one in the process of construction. Components of assembly are stored as separate files with extension CATPart, while the assembly is stored by the extension .CATProduct for which the links and position of the components are previously defined. The principle forming of the lower sheave assembly in CATI software V5 is shown in Figure 2.

Figure 2. Formation of the lower sheave assembly

Formed 3D assembly model allows the checking of the accuracy of the parts that make the assembly by geometric criteria of mutual relations with the function *Clash*, and in this way, it is possible to just eliminate the errors in the construction. Also, the function *Sectioning*, for section creating, is used for a better analysis of the shape and mutual position of the elements. In the module *Assembly Design*, composed of solid models of parts, it is possible to quickly determine the geometric characteristics and measures inertia using the function *Measure (Between, Item, Inertia)* for single parts or assembly. These data can be used for predicting the amount of material required (for cast parts) or amount of material which will be removed by mechanical treatment (combined with the Bool's operations), and the way of transport, cost of production prices (eg, the length of the contour laser edge), the use of planning standard elements of the weight of single parts or assembly (carrying screws, spring elements, Fuses), etc.. The lower sheave assembly is shown in Figure 3.

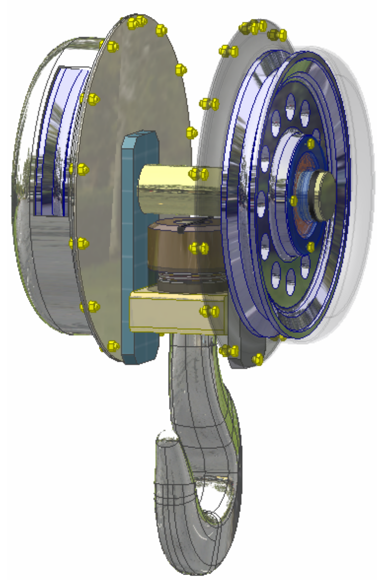
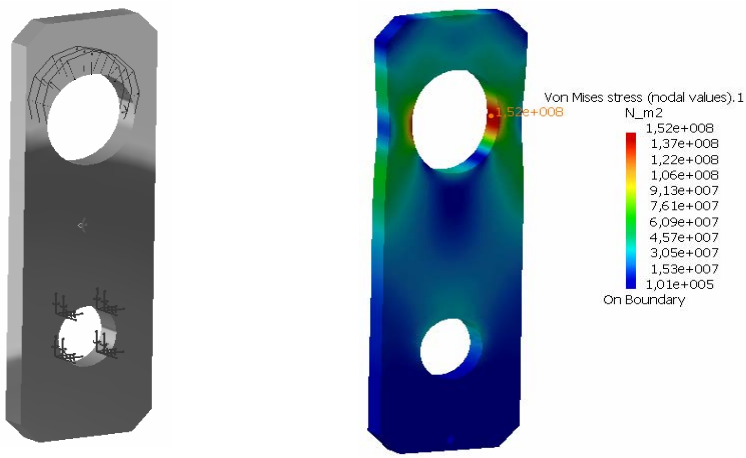


Figure 3. The lower sheave assembly

3. STRESS WARPED ANALYSIS OF THE CARRIER SHEET

Stress warped analysis includes the following steps: pre-processing, processing and post-processing. Pre-processing includes defining the characteristics of material, type and size of finite elements, load and the connection to the environment. Defining characteristics of the material includes the input of the values for Young's elasticity module and Poisson's ratio. Material of the carrier sheet is a steel tag EN C22E with the following characteristics: Young's modulus of elasticity Y = 2.1 · 1011 Pa, Poisson's ratio of 0.266. Parabolic tetrahedron, which dimension is 4 mm and its accuracy of the actual contours monitoring is 2mm, is used as the finite element. Using of these values for the size of the finite element does not require a long time or a lot of computer resources for processing, and because of that, the value of finite element can be assigned to the overall element (the global value of the finite elements). Choice of types of the finite element is executed in accordance with the characteristic geometry analyzed places stress analysis carrier sheet, and the parabolic tetrahedron elements, available in the module *Generative structural analysis,* best suits for its form. Load in the bearings is defined by function *Load Bearing* for the place of a larger hole in the amount of Q/2, where is Q - nominal capacity hooks. Connection with the environment is defined on the surface of a smaller opening, using the *User-defined Restraint* (general connection with the allowed rotation). A model of the carrier sheet, with the symbolic visualization of boundary conditions explained in the previous text, is shown in Figure 4a, and that is the final phase of pre-processing.



a) *b)*

Figure 4. Model of the carrier sheet: a) boundary conditions, b) the state stress

Stress of the carrier sheet, with a scale value of stress, is shown in Figure 4b. The deformed state of the carrier sheet, for which the increasing factor is 200, is, also, shown in the figure. One can see that the maximum value of stress in a opening is 152 Mpa. Analytical value of stress on this place is:

$$\sigma_z = \alpha_k \cdot \sigma_{zn} = \alpha_k \cdot \frac{Q}{2\cdot(b-D_2)\cdot s} = 2{,}1\cdot\frac{156960}{2\cdot(135-90)\cdot 24} = 152{,}46[MPa] \text{... (1)}$$

where: α_k – stress concentration factor; b, D_2, s – sheet width, diameter of a larger opening, thickness of sheet. On the basis of the displayed results, it can be seen that the maximum values of the stress obtained by FEM-based analysis and by analytical way are the approximately equal. Analysis of the overall stress state of the element shown in Figure 5b allows the optimization of machine elements according to the criteria of equality of stress. Although this is not the direct method, high-developed possibilities of parameterizations and functional connecting of the elements characteristics in the software CATI V5 through all the modules in which is the element, as well as the ability of the current changes update, greatly facilitate the proper construction of machine elements with respect to achieving an optimal form of the element according to the criteria of stress equality or according to some other criteria.

4. CONCLUSION

The shown way of presenting the machine elements and assemblies by using computers, enables the accurate analysis of their dimensions and shape, especially taking into account the impact of these characteristics on the validity of the assembly. Modern CAD systems offer the functionality of advanced analysis of the single elements in the standby mode as well as in a state of movement (a simulation of moving parts). Noticed errors could be corrected relatively quickly, and in this way it is currently being updated through all the modules of there software in which the mechanical element or the assembly is placed, and in that way the reflection of the implemented changes on the assembly could be analyzed. Stress warped analysis enable timely detection of errors and help in optimizing of the form and dimensions of the mechanical elements.

5. REFERENCE

[1] http://www.metalravne.com/selector/steels/CK22.html
[2] Nedžad Repčić, Adil Muminović, Alija Zuko: Mašinski Elementi – Priručnik za izradu projektnih zadataka.
[3] Nedžad Repčić, Mirsad Čolić: Transportna Sredstva
[4] Amirouche F. M. L.: Higher-Principles of Computer-Aided Design and Manufacturing, 2nd edition, Prentice Hall, Upper Saddle River, New Jersey, 2004.