STATIC ANALYSIS TO REDESIGN THE GRIPPER, USING CREO PARAMETRIC SOFTWARE TOOLS

Cristian VILAU[1], Nicolae BALC[1] , Dan LEORDEAN[1], Cosmin COSMA[1]

ABSTRACT: *Within this paper is presented the optimizing activity of a clamping jaw used for gripping and fixing revolution or/and prismatic work-pieces, by the help of CREO design software, using Creo Parametric and Creo Simulate modules. The article approaches a case study, within two loading situations of the gripper jaw, mounted on an industrial arm robot, are presented. Also, CAD and finite element method (FEM) advantages have been presented.*

KEY WORDS: design, FEM, Creo Parametric, Creo Simulate.

1 INTRODUCTION

Design of a complex model, supposes using a CAD software, having extended possibilities in order to allow modelling as accurate as possible of the 3D desired model. CREO package is a 3D integrated solution CAD/CAM/CAE (Computer Aided Design/Computer Aided Manufacturing/ Computer Aided Engineering), it offers all-steps development solutions of a new product, from design until manufacturing. The Parametric Module can create models that have completely parametrized features and the Simulate Module allows calculation of the forces, forces of reaction, moment of loading from coupling/bearings (yield supports), e.g. Creo Parametric and Creo Simulation are design and analysis software used in mechanical engineering, in order to virtually test a system that is mechanically, thermally, fatigue, shock, electrically, hydraulically stressed on.

Creo Parametric disposes of the most complex 3D CAD package, all these being completely integrated and associated. This it means that any change done on the model, in any module, has a further influence in the others associated modules (Creo parametric). The advantages were the used of CAD resources for transfer and interplay of the geometrical, technological and mechanical information.

A design process starts by identifying condition of a need and defining the problem and ends with a decision for a design process. After problem has been defined, the next job step is analysing and finding an optimal solution.

[1] Technical University of Cluj-Napoca, Department of Manufacturing Engineering,

E-mail: cristian.vilau@tcm.utcluj.ro, nicolae.balc@tcm.utcluj.ro, dan.leordean@tcm.utcluj.ro, cosmincosma@mail.utcluj.ro

Finding the optimal solution is not possible if this does not fit to the tender book of the product (Dan, 2013).

When all conditions of a design process are fulfilled, the final step is the assessment of the process, through experiments performed in lab with a prototype model.

General aspects about finite element method (FEM).

FEM is the most widespread method of solving engineering issues. A model subjected to a FEM analysis is meshed in subdomains of finite dimensions limited by straight-line or curved-lines frontiers.

The main advantages of FEM are:

- Flexibility (allows meshing of some complex shapes)

- It has the possibility of modelling the inhomogeneous bodies in terms of physical properties.

- It can be easily implemented in calculation programs.

In order to perform a FEM of a model (structures), it is need to perform the meshing of the body (structure). The model (structure) is a continuous one, having an endless number of points. The meshing consists in passing from continuous model a discreet one, meshed (having a finite number of points). Through meshing is understood dividing the model in elements or fragments, called finite elements. The peaks of the finite elements are linked each other through points. The points determined through meshing are called nodes, in these nodes are defined the unknown nodal (these unknown can be displacements or stresses) and represent result values AEF (Comsa, 2007), (Balc, 2004).

Finite elements divide into:

- Unidirectional finite elements (vary according to an axis).
- Bidirectional finite elements (vary according to two axles)
- Three-dimensional finite elements (vary according to three axles)

Creo Simulate module uses tetrahedral finite elements with superior functions.

2 STATIC ANALYSIS OF THE GRIPPING JAW, UNDER DIFFERENT LOADINGS

Model and material.

Within this paper, two loading situations have been considered, concerning the clamping areas of the jaw depending on the shape of the parts that is need to be handled (cylindrical or prismatic)

Work-piece material is an aluminium alloy Al2014, its main properties being presented within the table 1 (mat2014).

Table 1. Al2014 Mechanical properties

	U.M	Value
Density	g/cm^3	2.8
Ultimate Tensile Strength	MPa	483
Modulus of Elasticity	GPa	72.4
Poisson's Ratio		0.33
Shear Strength	MPa	290

Clamping jaw modelling has been realized by the help of Creo Parametric software. The model has a weight of m=23.56g and dimensions according to figure 1.

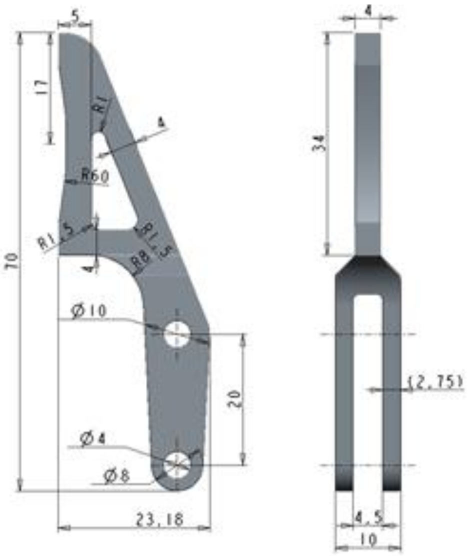

Figure 1. Design model.

Loading and fixing conditions.

Loading force „F" applied in static conditions, has a value of 300N, it means a maximum load of 30Kg. Loading conditions are for clamping revolution parts and prismatic parts. In figure 2 is presented the clamping area for cylindrical parts and in figure 3 are presented the clamping areas for clamping prismatic part.

In order to perform the FEM, in addition to loading conditions, it is need to establish also the fixing conditions. In figure 4 are shown the fixing conditions.

Figure 2. Loading surfaces for clamping revolution parts.

Figure 3. Loading surfaces for clamping prismatic parts.

Figure 4. Fixing surfaces.

2.1 FEM for clamping rotational parts

In order to realize FEM, the work-piece has been meshed using automatic meshing option, this option having a more rarefied structure, this it means a lower quality mesh. The analysis results are affected by the meshing elements, the more refined the meshing elements is, the higher the results accuracy is, but the needed time is longer. In order to get more accurate results, on the important surfaces (holes and clamping surfaces), the work-piece has been meshed with a network that has max. 2 mm the size of meshed elements.

After meshing, have been achieved the elements presented within the table 2.

Table 2. Meshing elements

	Tetrahedral	Nodes	Edge	Face
Automatic meshing	2236	793	3708	5147
Customized meshing	8532	2345	12512	18695

In order to realize the FEM, it was need to apply the force (loading condition) according to figure 2 and fixing of the jaw (fixing condition) according to figure 4. After running FEM, have been achieved the results presented in table 3. In figure 5 is presented the distribution of equivalent von Mises stresses that belong to the case of cylindrical gripping parts.

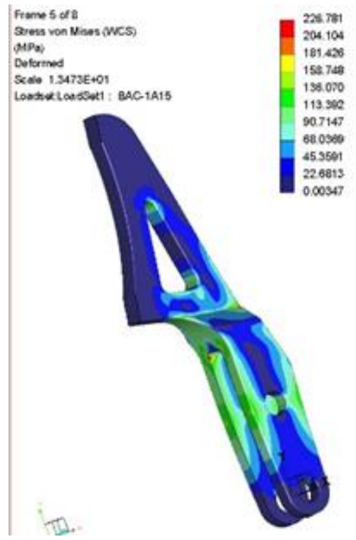

Figure 5. Stress Von Mises, while clamping cylindrical parts (revolution).

2.2 FEM for clamping prismatic parts

As presented before, it has been used the same meshing network, but applying the force as shown in figure 3. After FEM, in this case, have been achieved the values of stresses and distortions as shown in table 3.

In figure 6 is presented the distribution of equivalent von Mises stresses that appear in this case.

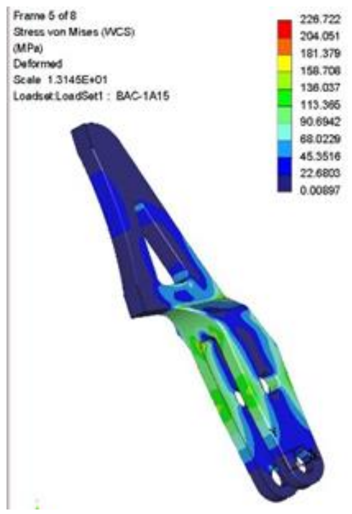

Figure 6. Von Mises stresses, while clamping prismatic parts

From the table 3 it can be seen that von Mises stresses are much under the allowable stress of chosen material (483 Mpa), the optimizing decision of the jaw through redesigning has been taken.

Table 3. Stresses and displacements while clamping cylindrical and prismatic parts

The area of clamping	Stress von Mises [MPa]	Stress shear [MPa]	Strain energy [MPa]	Displacement [mm]				
				Total	Axis Y		Axis Y	
					negative	positive	negative	positive
cylindrical	**222.562**	113.243	0.35	**0.4136**	-0.0086	0.384	-0.0178	0.156
cylindrical	**222.562**	113.243	0.35	**0.4136**	-0.0086	0.384	-0.0178	0.156

3 REDESIGN THE CLAMPING JAW

Through redesign, reducing the weight and optimizing the model were the goals that have been established. In figure 7 is the redesigned version of the jaw, having a mass, m=13,79g. Redesign of the jaw focused on maximum 1mm distortion values and the von Mises stresses being under the allowable material stresses.

Further are presented the two cases of clamping, cylindrical and prismatic.

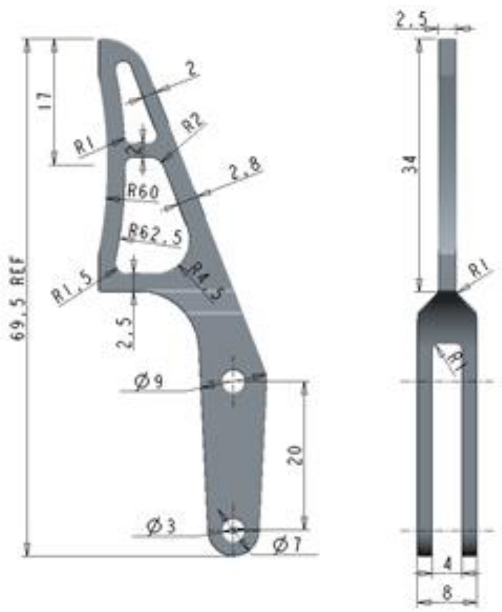

Figure 7. Redesigned version

3.1 FEM for clamping improved rotational parts

In order to perform FEM, the work-piece has been redesigned and meshed and the values for meshing elements are presented in table 4. After loading the jaw on the cylindrical clamping surface (according to figure 2) and fixing according to figure 4, the FEM has been done. After this, the results presented in table 5 were achieved for von Mises stresses and displacements.

Table 4. Meshing elements

	Tetrahedral	Nodes	Edge	Face
Automatic meshing	2087	739	3467	4810
Customized meshing	6445	1912	9800	14324

In figure 8 is presented the distribution of the equivalent stresses while clamping cylindrical parts.

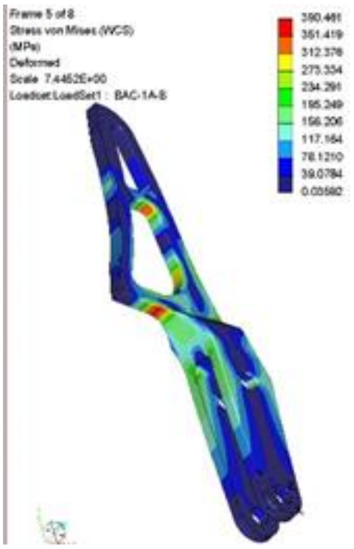

Figure 8. Equivalent stresses distribution

3.2 FEM for clamping improved prismatic parts

In order to perform FEM for prismatic parts, the same steps as for the cylindrical parts were followed, only the force has been applied according to the figure 3 and fixing according to the figure 4. After FEM, the results achieved are shown in table 5. In figure 9 is presented the distribution of von Mises stresses.

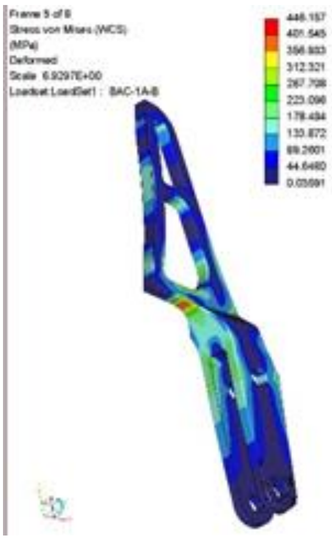

Figure 9. Equivalent stresses distribution

Table 5. Stresses and displacements while clamping prismatic parts

The area of clamping	Stress von Mises [MPa]	Stress shear [MPa]	Strain energy [MPa]	Displacement [mm]				
				Total	Axis Y		Axis Y	
					negative	positive	negative	positive
cylindrical	**390.461**	199.682	1.06	**0.93**	-0.017	0.86	-0.027	0.364
cylindrical	**446.157**	228.252	1.385	**1.00**	-0.017	0.928	-0.027	0.388

From table above it can be seen a difference relatively high between the equivalent stresses, while clamping cylindrical parts and the value of prismatic clamping stresses. Also, a small difference between overall distortions can be seen.

4 CONCLUSIONS

The Creo parametric analysis method and software tools are efficient and accurate in estimating both the deformations and stresses caused by different loads and working condition.

The results obtained could be used immediately to improve the shape of the analysis structure.

An interactive redesign and analysis procedure could be rapidly used, in order to check the benefits of the shape modification.

Gripper could be optimised according to the shape of the parts to be handled, in order to decrease the weight of the grippers and their cost.

Future research will make dynamic analysis and shock.

5 ACKNOWLEDGEMENTS

This work was partially supported by the strategic grand POSDRU/159/1.5/S/137070 (2014) of the European Social Fund – Investing in People, within the Sectorial Operational Programme Human Resources Development 2007-2013. Also, this paper was supported by the PCCA 115/2014 research project and by the European Social Fund through POSDRU Program, DMI 1.5, ID 137516 PARTING.

6 REFERENCES

►http://www.inas.ro/ro/products/ptc/3d-design/creo-parametric
►Dan Leordean, Nicolae Balc, (2013). *Proiectare Industriala. Aplicatii PTC: Creo Parametric*, Alma Mater, Cluj-Napoca. ISBN 978-606-504-152-3.
►Comsa Sorin, (2004). *Metoda elementelor finite: curs introductiv*, Editura U.T. Pres, Cluj-Napoca, 2007.
►Nicolae Balc, Petru B,*Sisteme CAD/CAM/CAPP – teorie si practica*, Editura Tehnica, Bucuresti.
►http://asm.matweb.com/search/SpecificMaterial.asp?bassnum=MA2014T6

7 NOTATIONS

The following symbols are used in this paper:
CAD = Computer Aided Design;
CAM = Computer Aided Manufacturing;
CAE = Computer Aided Engineering;
FEM = Finite element method.
m = weight
F = Force