STRESS-STRAIN ANALYSIS OF THE SPUR GEAR OF THE GEAR-BOX REDUCTION OF THE WORKING WHEEL OF THE EXCAVATOR SCHRS 1300 24/5.0 USING CAD/CAE SOFTWARE

[1]MR. SC. FATMIR AZEMI, [2]DR.SC. GANI PLLANA, [3]MSC. MUSAJ TERBUNJA, [4]MSC. BEKIM MALOKU

[1]PhD student at Josip Juraj Strossmayer University Of Osijek-Republic of Croatia, [2]Lecturer at Faculty of Mechanical Engineering-University of Prishtina-Republic of Kosovo, [3]Director in Department of Education in Municipality-Republic of Kosovo, [4]Teacher at Secondary Technical School "L.utfi Musiqi" Vushtrri-Republic of Kosovo
E-mail: [1]fatazemi@hotmail.com, [2]ganipllana44@gmail.com, [3]musajterbunja@postribe.com, [4]bekim.m70@hotmail.com

Abstract- Kosovo is a place with a rich natural resources, especially the industry of coal mine has the most impact in development of various sectors of industry and economy of Kosovo. One from other problems of production energy is the way of transportation of coal from mines to power plants. A lot of machines and equipments are used to get the lignite reserves from surface mines, one of them which will be our case study of this work is the Excavator SchRs 1300 24/5.0 located in open-cast coal mines in "Bardhi i Madh- Fushë Kosova". The paper presents the results of analysis of stress-strain of spur gear of the gear-box reduction of the Excavator. During Excavator arm movement from right side to left the bucket wheel works with maximum capacity, so in that case have been caused fracture in spur gear 9 and spur gear 10 of gear- box reduction of the Excavator. To calculate stress-strain structure of parts we are going to take parameters with maximum loads which are taken from experimental analysis in the entry of axis of gear-box reduction. In this paper we will use CATIA V5 (CAD- Computer Aided Design) software for design, and ANSYS WORKBENCH 14.0 (FEM- Finite Element Method) for stress-strain analysis.

Keywords- ANSYS WORKBENCH 14.0, CAD/CAE software, CATIA V5, Stress-Strain Analysis, Spur-Gear

I. INTRODUCTION

The need for visualizing, simulation, analysis and documenting mechanical parts prior to production has existed ever since human beings began creating machines, mechanisms, and products. Over the last decades, methods for achieving this function have been developed dramatically. Today, powerful computer hardware and software have supplanted the manual drafting and have advanced to the point of playing a pivotal role in not only improving design visualization but also in driving the entire manufacturing process.

Computer Aided Design (CAD) is assistance of computer in engineering processes such as creation, optimization, analysis and modifications. It is an integration of Mechanical and Computer technology to aid in the design process like Modeling, Assembly, Drafting, Die Design, Tool Design, Sheet metal, analysis of products. In this paper we have used CATIA V5 software as CAD software for modeling of spur gears as well as their assembly.

Computer Aided Engineering (CAE) is the use of Computer to support engineers in tasks such as analysis, simulation, design, manufacture, planning, diagnosis, and so on. ANSYS Mechanical and ANSYS Multiphysics software are non exportable analysis tools incorporating pre-processing (geometry creation, meshing), solver and post-processing modules in a graphical user interface.
These are general-purpose finite element modeling packages for numerically solving mechanical problems, including static/dynamic structural analysis (both linear and non-linear), heat transfer and fluid problems, as well as acoustic and electro-magnetic problems.

In our case ANSYS Workbench 14.0 is used for Stress-Strain Analysis of the spur gear of the gear-box reduction of the working wheel of the Excavator.

II. STATEMENT OF THE PROBLEM, OBJECTIVES AND PURPOSE OF THE STUDY

The Excavator works in located in open-cast coal mines in "Bardhi i Madh- Fushë Kosovë", and during Excavator arm movement from right side to left the bucket wheel works with maximum capacity and in that case have been caused damage in spur gear 9 and spur gear 10 of gear- box reduction of the Excavator.

Our overall objective and purpose of this study is to determine stress-strain structure of spur gear 9 and spur gear 10 in order to find out if the gears accomplish working conditions.

III. METHODOLOGY AND MATERIALS

CAD is used mainly for the creation of detailed 3D solid or surface models, or 2D vector-based drawings of physical components. However, CAD is also used throughout the engineering process from conceptual design and layout of products, through strength and dynamic analysis of assemblies. In this case we use CATIA V5 software for modeling parts and their assembly as shown below in figure 1.

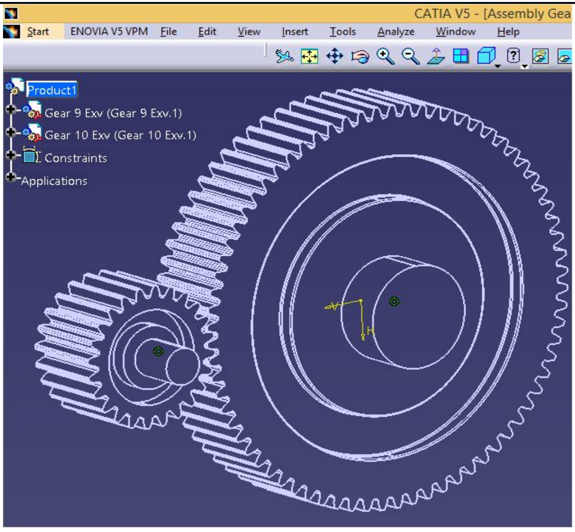

Fig. 1. Solid Modeling of assembly of parts with use of CATIA

Analysis stress-strain structure of assemble of parts are done with application of FEM method Ansys workbench 14.0 software, and modeling of parts are taken from CATIA V5, to open file in ANSYS needed that file in CATIA to saved as IGES file.

Stress-strain analysis passed through some steps:

Build Geometry

There are several ways to create the model geometry within ANSYS, some more convenient than others, but in this case the geometry is taken from CATIA software.

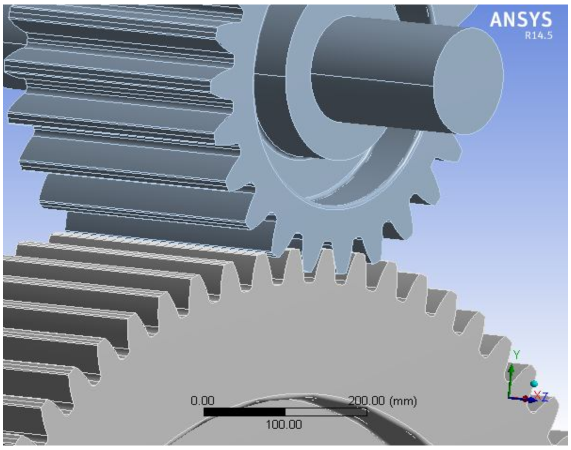

Fig. 2. Import file from CATIA V5 to ANSYS WORKBANCH 14.0

All data of geometry of parts are given below through tables.

Geometry

TABLE 1
Model (A4) > Geometry

Object Name	Geometry
State	Fully Defined
	Definition
Source	C:\Users\Fati\Desktop\Assembly Gear 9 with Gear 10 ver 3.igs
Type	Iges
Length Unit	Meters
Element Control	Program Controlled
Display Style	Body Color
	Bounding Box
Length X	1530.2 mm
Length Y	2068.3 mm
Length Z	700. mm
	Properties
Volume	6.0285e+008 mm³
Mass	4732.4 kg
Scale Factor Value	1.
	Statistics
Bodies	2
Active Bodies	2
Nodes	82624
Elements	48194
Mesh Metric	None
	Basic Geometry Options
Temporary Directory	C:\Users\Fati\AppData\Local\Temp
Analysis	3-D
Type	
Mixed Import Resolution	None
Decompose Disjoint Geometry	Yes
Enclosure and Symmetry Processing	Yes

TABLE 2
Model (A4) > Geometry > Parts

Object Name	Part 1	Part 2
State	Meshed	
	Graphics Properties	
Visible	Yes	
Transparency	1	
	Definition	
Suppressed	No	
Stiffness Behavior	Flexible	
Coordinate System	Default Coordinate System	
Reference Temperature	By Environment	
	Material	
Assignment	Structural Steel	
Nonlinear Effects	Yes	
Thermal Strain Effects	Yes	
	Bounding Box	
Length X	1530.2 mm	603.1 mm
Length Y	1530.2 mm	603.1 mm
Length Z	700. mm	

Properties		
Volume	5.4151e+008 mm³	6.1336e+007 mm³
Mass	4250.9 kg	481.49 kg
Centroid X	147.12 mm	146.38 mm
Centroid Y	-2266.5 mm	-1264.9 mm
Centroid Z	-440.22 mm	-440.2 mm
Moment of Inertia Ip1	6.0192e+008 kg·mm²	1.3469e+007 kg·mm²
Moment of Inertia Ip2	6.0192e+008 kg·mm²	1.3468e+007 kg·mm²
Moment of Inertia Ip3	1.0974e+009 kg·mm²	1.4242e+007 kg·mm²
Statistics		
Nodes	50357	32267
Elements	29799	18395
Mesh Metric	None	

Define Materials

Data of materials are given through tables.

Material Data

Structural Steel

TABLE 3
Structural Steel > Constants

Density	7.85e-006 kg mm^-3
Coefficient of Thermal Expansion	1.2e-005 C^-1
Specific Heat	4.34e+005 mJ kg^-1 C^-1
Thermal Conductivity	6.05e-002 W mm^-1 C^-1
Resistivity	1.7e-004 ohm mm

TABLE 5
Structural Steel > Compressive Yield Strength

Compressive Yield Strength MPa
250

TABLE 6
Structural Steel > Tensile Yield Strength

Tensile Yield Strength MPa
250

TABLE 7
Structural Steel > Tensile Ultimate Strength

Tensile Ultimate Strength MPa
460

TABLE 8
Structural Steel > Strain-Life Parameters

Strength Coefficient MPa	Strength Exponent	Ductility Coefficient	Ductility Exponent	Cyclic Strength Coefficient MPa	Cyclic Strain Hardening Exponent
920	-0.106	0.213	-0.47	1000	0.2

TABLE 9
Structural Steel > Isotropic Elasticity

Temperature C	Young's Modulus MPa	Poisson's Ratio	Bulk Modulus MPa	Shear Modulus MPa
	2.e+005	0.3	1.6667e+005	76923

Generate Mesh

Mesh

TABLE 10
Model (A4) > Mesh

Object Name	Mesh
State	Solved
Defaults	
Physics Preference	Mechanical
Relevance	0
Sizing	
Use Advanced Size Function	On: Proximity and Curvature
Relevance Center	Coarse
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Fast
Span Angle Center	Coarse
Curvature Normal Angle	Default (70.3950 °)
Proximity Accuracy	0.5
Num Cells Across Gap	Default (3)
Min Size	Default (1.3290 mm)
Proximity Min Size	Default (1.3290 mm)
Max Face Size	Default (132.90 mm)
Max Size	Default (265.80 mm)
Growth Rate	Default (1.850)
Minimum Edge Length	2.16460 mm
Statistics	
Nodes	82624
Elements	48194
Mesh Metric	None

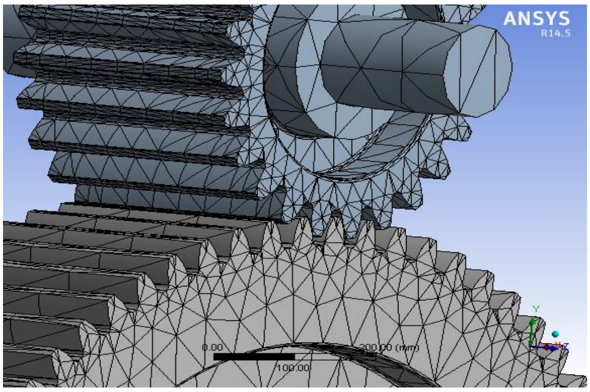

Fig. 3. Generating MESH

Apply Loads

TABLE 11
Model (A4) > Static Structural (A5) > Loads

Object Name	Frictionless Support	Moment
State	Fully Defined	
Scope		
Scoping Method	Geometry Selection	
Geometry	8 Faces	2 Faces
Definition		
Type	Frictionless Support	Moment
Suppressed	No	
Define By		Components
Coordinate System		Global Coordinate System
X Component		0. N·mm (ramped)
Y Component		0. N·mm (ramped)
Z Component		-1.9735e+008 N·mm (ramped)
Behavior		Deformable
Advanced		
Pinball Region		All

IV. RESULTS AND DISCUSSION

After the program executed we obtain results of Stress-strain

Results are given through tables.

Obtain Solution

Solution (A6)

TABLE 9
Model (A4) > Static Structural (A5) > Solution

Object Name	Solution (A6)
State	Solved
Adaptive Mesh Refinement	
Max Refinement Loops	1.
Refinement Depth	2.
Information	
Status	Done

Review Results

TABLE 10
Model (A4) > Static Structural (A5) > Solution (A6) > Results

Object Name	Equivalent Elastic Strain	Equivalent Stress
State	Solved	
Scope		
Scoping Method	Geometry Selection	
Geometry	All Bodies	
Definition		
Type	Equivalent Elastic Strain	Equivalent (von-Mises) Stress
By	Time	
Display Time	Last	
Calculate Time History	Yes	
Identifier		
Suppressed	No	
Integration Point Results		
Display Option	Averaged	
Results		
Minimum	2.4689e-008 mm/mm	2.7365e-003 MPa
Maximum	3.8289e-003 mm/mm	662.33 MPa
Minimum Occurs On	Part 1	
Maximum Occurs On	Part 2	
Information		
Time	1. s	
Load Step	1	
Substep	1	
Iteration Number	1	

Tables showed that the minimum of stress is 0.0027365 MPa and maximum of stress is 662.33 MPa.

The minimum of Equivalent Elastic Strain is 2.4689e-008 mm/mm and maximum of Equivalent Elastic Strain is 3.8289e-003 mm/mm.

Graphically are presented in figure 4 and 5.

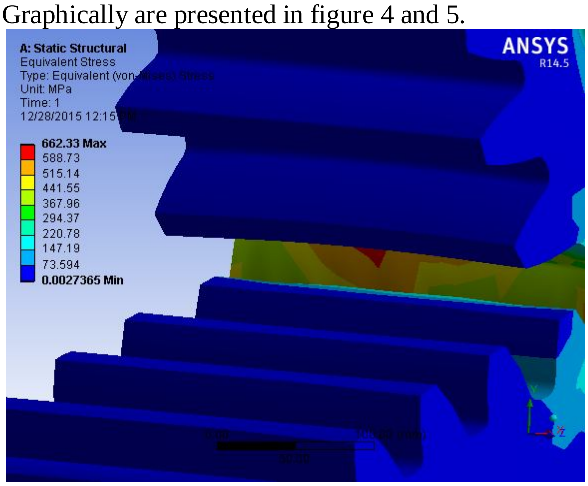

Fig. 4. Plot the von Mises equivalent stress.

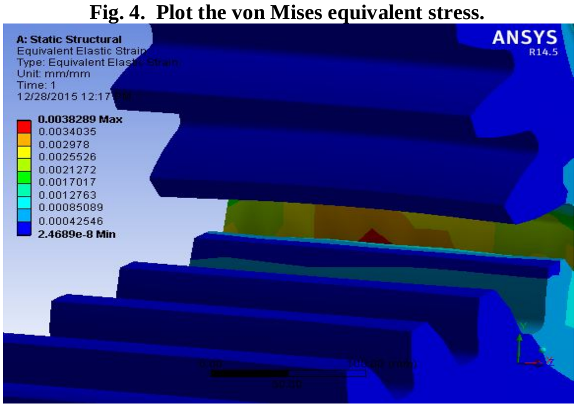

Fig. 5. Plot the Equivalent Elastic strain

CONCLUSION AND FUTURE WORK

This study has first described how using CATIA V5 for modeling parts and their assemble, and how files can import from other software to ANSYS WORKBENCH 14.0 for analysis of stress-strain structure. FEA method are used for analysis of spur gears and based on results from software we have found out that the material that have been used for gears don't fulfill conditions. Need to change the material of gears with a high-quality. However, further studies still need to analyze the geometry of gears in order to redesign parts and to optimize process.

REFERENCES

[1]. Ziethen, R. D.: "CATIA V5-Effiziente Konstruktion mit Makros" , Carl Hanser Verlag München Wien, 2003.

[2]. Koehldorfer, W.: "Finite-Elemente-Methoden mit CATIA V5", Carl Hanser Verlag München Wien, 2004.

[3]. Schmid, D.: "Konstruktionslehre Maschinenbau", VERLAG EUROPA-LEHRMITTEL, 2011.

[4]. Dobrovolsky, V. etc.: "Machine elements" (Translated from the Russian), MIR PUBLISHERS MOSCOW, 1977.

[5]. Ochęduszko, K.: "Koła zębate", Warszawa, 1976.

[6]. Tochtermann/Bodenstein: "Konstruktionselemente des Maschinenbaues",Teil 2, Springer-Verlag Berlin Heidelberg New York, 1979.

[7]. Niemann, G.; Winter, H.: "Maschinenelemente" Band II, pringer-Verlag Berlin Heidelberg New York Tokyo, 1985.

[8]. F. P. Beer, E. R. Johnston, J. T. DeWolf,x and D. F. Mazurek, Mechanics of Materials, 6th ed.(McGraw-Hill, New York, 2011).

[9]. J. E. Shigley, C. R. Mischke, and R. G. Budynas, Mechanical Engineering Design, 7th ed. (McGraw-Hill, New York, 2003).

[10]. ANSYS Help System, Design Exploration User Guide, Release 14.5, ANSYS, Inc., 2012.

[11]. ANSYS Help System, CFX—Modeling Guide, Release 14.5, ANSYS, Inc., 2012.

[12]. Y. Nakasone, S. Yoshimoto, T. A. Stolarski; Engineering Analysis With ANSYS Software-Printed and bound by MPG Books Ltd., Bodmin, Cornwall.

[13]. The Finite Element Method for Mechanics of Solids with ANSYS Applications, Ellis H. Dill; CRC Press Taylor & Francis Group 6000 Broken Sound Parkway NW, Suite 300 Boca Raton, FL 33487-2742, 2011.

[14]. OSITA D. I. NWOKAH YILDIRIM HURMUZLU: THE MECHANICAL SYSTEMS DESIGN, Southern Methodist University, Dallas, Texas, 2002.

[15]. Cornelius, L.: "COMPUTER INTEGRATED MANUFACTURING", Boca Raton, London New York Washington, D.C., 2001.

[16]. Derl, Peter J.:"CAD-CAM-Systeme", Wien, 1987.

[17]. Radhakrishnan, P.; Subramanyan, S.; Raju, V.: "CAD/CAM/CIM", New Delhi, 2008.

[18]. F. Azemi, "The analysis and optimization of the factors that indicate in the construction of machines with the aid of computer", Prishtine 2004 .

[19]. Richard G. Budynas and J. Keith Nisbett:" Mechanical Engineering Design", The McGraw-Hill Companies, 2008.

[20]. G. Pllana , I. Jashari, "Detalet e makinave", Prishtine 1994.

[21]. Sham Tickoo, "CATIA V5-6R2014 for Designers", CAD/CIM Technologies Schererville, Indiana 46375, USA.

★★★