NUMERICAL ANALYSIS ON NAVAL APLICATION M7-T36 GEAR BASED ON COMMERCIAL SOFTWARE

Şef lucr. dr. ing. **Ionuţ Cristian SCURTU,** Şef lucr. dr. ing. **Adrian POPA**

Academia Navală „Mircea cel Bătrân“, Constanţa, România

REZUMAT. Experienţa arată că există mai multe frecvenţe de operare la bordul navelor şi unele elemente pot fi deteriorate de situaţiile de funcţionare proaste. Datorită complexităţii geometrice a roţii dinţate, analiza cu element finit este necesară pentru a determina toate frecvenţele proprii pentru roata dinţată prezentată. Lucrarea prezintă o analiză modală pentru roata dinţată M7-T36, pe baza rezultatelor software Ansys. Ochiurile 3D într-o astfel de analiză poate fi complexă şi consumatoare, în scopul de a obţine cel mai bun rezultat din analiza modal. Discretizarea prezentată în acest articol poate fi îmbunătăţită pentru model în simulările ANSYS.

Cuvinte cheie: Roata dinţată M7-T36, modal, Analiză navală , soft Ansys.

ABSTRACT. Experience indicates that there are many operating frequencies onboard vessels and gears can be damaged by bad operating situations. Due to geometrical complexity of gears, finite element analysis is needed in order to determine all frequencies related to presented gear. The paper presents modal analysis for M7-T36 gear based on Ansys software results. The 3D mesh in such analysis can be complex and time consuming in order to get the best result in modal analysis. The mesh presented in this article can be improved for a model in ANSYS simulations .

Keywords: M7-T36 gear, modal, Analysis on naval aplication, Ansys software.

1. INTRODUCTION

Gears are relatively common in the naval industry and a finite element analysis done for specific gear types is relevant to detection of faults discovered during inspections onboard.

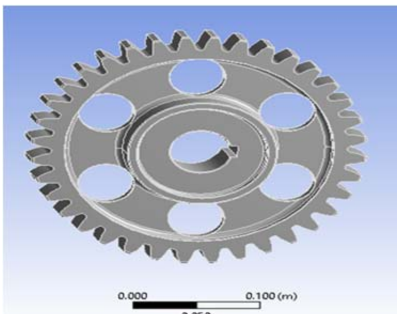

Fig. 1. M7-T36 gear Cad geometry.

2. PROBLEM DESCRIPTION

The main idea of Ansys numerical analysis is to determine modal frequencies generated for M7-T36 gear. ANSYS Modal can use parametric design language (APDL) commands necessary to build variable geometry and meshes. When the report of analysis is complete, the user can modify input the data used in parametric design. Ansys software is widely used in industry and manufacturing process via geometry modeler and static and dynamic load calculation while other sources cause failure during operation. In naval sector, there are many old vessels fitted with wide range of gears. To avoid gears failure production plants have to investigate all related failure sources.

3. ANALYSIS SETTINGS AND NUMERICAL SOLUTION

Settings used in mesh will presented below for the numeric solution calculated for M7-T36 gear Cad geometry. All calculated and presented data is solved in the metric system as presented in table 1.

Table 1. **Units**

Unit System	Metric (m, kg, N, s, V, A) Degrees rad/s Celsius
Angle	Degrees
Rotational Velocity	rad/s
Temperature	Celsius

Table 2. **Geometry**

Object Name	*Geometry*
State	Fully Defined
Definition	
Type	DesignModeler
Length Unit	Meters
Element Control	Program Controlled
Display Style	Body Color
Bounding Box	
Length X	9.1e-002 m
Length Y	0.2654 m
Length Z	0.2654 m
Properties	
Volume	1.7375e-003 m³
Mass	13.639 kg
Scale Factor Value	1.
Statistics	
Bodies	1
Active Bodies	1
Nodes	67880
Elements	38427
Mesh Metric	None
Basic Geometry Options	
Parameters	Yes
Parameter Key	DS
Attributes	No
Named Selections	No
Material Properties	No
Advanced Geometry Options	
Use Associativity	Yes
Coordinate Systems	No
Reader Mode Saves Updated File	No
Use Instances	Yes
Smart CAD Update	No
Compare Parts On Update	No
Attach File Via Temp File	Yes
Analysis Type	3-D
Decompose Disjoint Geometry	Yes
Enclosure and Symmetry Processing	Yes

Table 3. **Geometry parts**

Object Name	*M7-T36 Gear*
State	Meshed
Graphics Properties	
Visible	Yes
Transparency	1
Definition	
Suppressed	No
Stiffness Behavior	Flexible
Coordinate System	Default Coordinate System
Reference Temperature	By Environment
Behavior	None
Material	
Assignment	Structural Steel
Nonlinear Effects	Yes
Thermal Strain Effects	Yes
Bounding Box	
Length X	9.1e-002 m
Length Y	0.2654 m
Length Z	0.2654 m

Table 3 (continued)

Properties	
Volume	1.7375e-003 m³
Mass	13.639 kg
Centroid X	-3.4664e-003 m
Centroid Y	-7.944e-005 m
Centroid Z	-4.1814e-008 m
Moment of Inertia Ip1	0.11301 kg·m²
Moment of Inertia Ip2	6.1288e-002 kg·m²
Moment of Inertia Ip3	6.1261e-002 kg·m²
Statistics	
Nodes	67880
Elements	38427
Mesh Metric	None

Table 4. **Coordinate Systems**

State	Fully Defined
Definition	
Type	Cartesian
Coordinate System ID	0.
Origin	
Origin X	0. m
Origin Y	0. m
Origin Z	0. m
Directional Vectors	
X Axis Data	[1. 0. 0.]
Y Axis Data	[0. 1. 0.]
Z Axis Data	[0. 0. 1.]

Table 5. **Mesh**

Object Name	*Mesh*
State	Solved
Display	
Display Style	Body Color
Defaults	
Physics Preference	Mechanical
Relevance	100
Shape Checking	Standard Mechanical
Element Midside Nodes	Program Controlled
Sizing	
Size Function	Adaptive
Relevance Center	Coarse
Element Size	Default
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Fast
Span Angle Center	Coarse
Automatic Mesh Based Defeaturing	On
Defeaturing Tolerance	Default
Minimum Edge Length	7.0711e-004 m
Inflation	
Use Automatic Inflation	None
Inflation Option	Smooth Transition
Transition Ratio	0.272
Maximum Layers	5
Growth Rate	1.2
Inflation Algorithm	Pre
View Advanced Options	No

Table 5 (continued)

Advanced	
Number of CPUs for Parallel Part Meshing	Program Controlled
Straight Sided Elements	No
Number of Retries	Default (4)
Extra Retries For Assembly	Yes
Rigid Body Behavior	Dimensionally Reduced
Mesh Morphing	Enabled
Triangle Surface Mesher	Program Controlled
Topology Checking	No
Pinch Tolerance	Please Define
Generate Pinch on Refresh	No
Statistics	
Nodes	67880
Elements	38427
Mesh Metric	None

Table 6. **Analysis**

Object Name	*Modal (A5)*
Definition	
Physics Type	Structural
Analysis Type	Modal
Solver Target	Mechanical APDL

Table 7. **Condition**

Object Name	*Pre-Stress (None)*
State	Fully Defined
Definition	
Pre-Stress Environment	None

Table 8. **Analysis Settings**

Object Name	*Analysis Settings*
State	Fully Defined
Options	
Max Modes to Find	20
Limit Search to Range	No
Solver Controls	
Solver Type	Program Controlled
Output Controls	
Stress	No
Strain	No
Nodal Forces	No
Calculate Reactions	No
General Miscellaneous	No
Analysis Data Management	
Scratch Solver Files Directory	
Save MAPDL db	No
Delete Unneeded Files	Yes
Solver Units	Active System
Solver Unit System	mks

4. RESULTS

Presented results for mesh(fig.2) and for total deformation (fig.3-10) are also supported with data tables and commands history. The results are presented for M7-T36 gear and can be easily updated for any other gear.

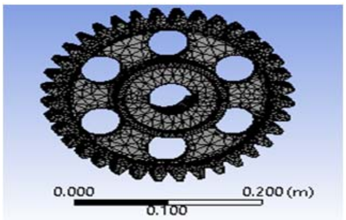

Fig. 2. M7-T36 gear Cad mesh.

Table 9. **Modal solution**

Object Name	*Solution (A6)*
State	Solved
Adaptive Mesh Refinement	
Max Refinement Loops	1.
Refinement Depth	2.
Information	
Status	Done
MAPDL Elapsed Time	
MAPDL Memory Used	
MAPDL Result File Size	
Post Processing	
Calculate Beam Section Results	No

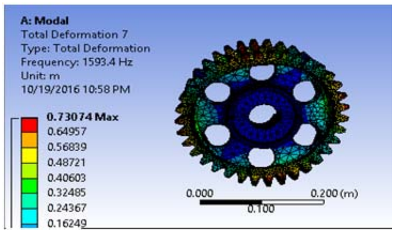

Fig. 3. M7-T36 gear total deformation at 1593 Hz.

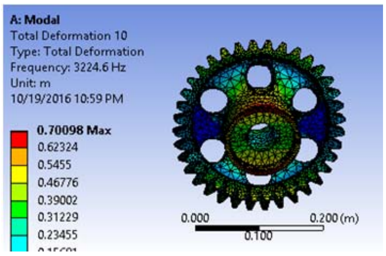

Fig. 4. M7-T36 gear total deformation at 3224 Hz.

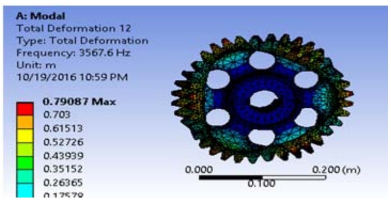

Fig. 5. M7-T36 gear total deformation at 3567 Hz.

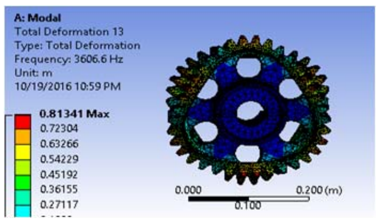

Fig. 6. M7-T36 gear total deformation at 3567 Hz.

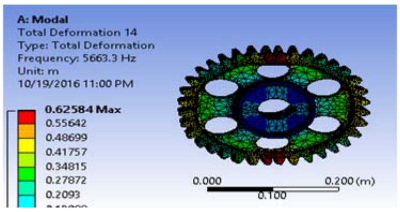

Fig. 7. M7-T36 gear total deformation at 5663 Hz.

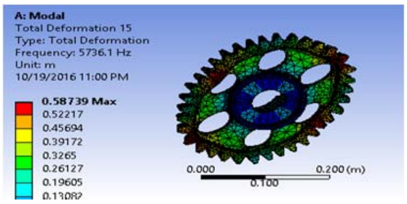

Fig. 8. M7-T36 gear total deformation at 5736 Hz.

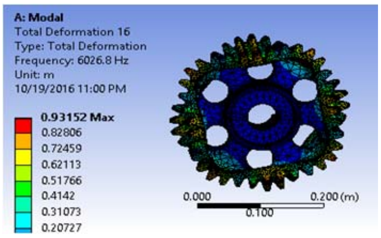

Fig. 9. M7-T36 gear total deformation at 6026 Hz.

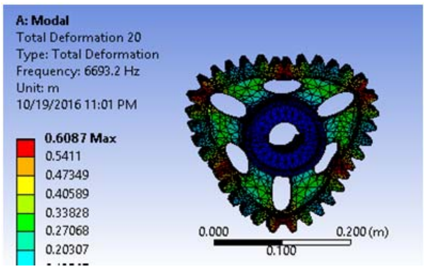

Fig. 10. M7-T36 gear total deformation at 6693 Hz.

Table 10. **Modal Solution**

Mode	Frequency [Hz]
1.	0.
2.	
3.	
4.	9.1223e-004
5.	2.2801e-003
6.	2.8428e-003
7.	1593.4
8.	1594.4
9.	2487.3
10.	3224.6
11.	3226.3
12.	3567.6
13.	3606.6
14.	5663.3
15.	5736.1
16.	6026.8
17.	6030.
18.	6331.8
19.	6333.2
20.	6693.2

Table 11. **Solution Information**

Object Name	*Solution Information*
State	Solved
Solution Information	
Solution Output	Solver Output
Newton-Raphson Residuals	0
Identify Element Violations	0
Update Interval	2.5 s
Display Points	All
FE Connection Visibility	
Activate Visibility	Yes
Display	All FE Connectors
Draw Connections Attached To	All Nodes
Line Color	Connection Type
Visible on Results	No
Line Thickness	Single
Display Type	Lines

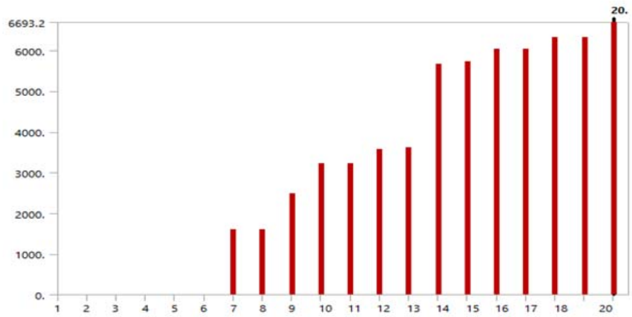

Fig. 10. Modal frequency solutions.

Table 12. **Modal Solution Results**

Minimum	0.1136 m	9.6024e-002 m	0.11885 m	0.25663 m	0.103 m
Maximum	0.5392 m	0.54626 m	0.40542 m	0.28759 m	0.41387 m

Table 13 **Model Modal Solution Results**

Frequency	3567.6 Hz	3606.6 Hz	5663.3 Hz	5736.1 Hz	6026.8 Hz

CONCLUSION

Using ANSYS modal analysis and different naval parts or structures, it was possible to determine own frequencies.

Cad tests were performed for the presented gear, and the results showed good agreement between the numeric and expected solutions, indicating that the mesh and input data were correct.

5 REFERENCES

[1] Chioibas A. 1, *The softer cosmos using in drawing process simulation*, „Mircea cel Batran" Naval Academy Scientific Bulletin, Volume XVII – 2014 – Issue 2

[2] Oanta E., Panait C., Lazaroiu G, Dascalescu, A., "*Analytic Model of the Mobile Half-Bridge of a Circular*

Settling Tank", Proceedings of the 11th WSEAS International Conference on Environment, Ecosystems and Development (EED '13), Brasov, Romania, June 1-3, 2013, ISSN 2227-4359, ISBN 978-1-61804-188-3, pg. 21-26.

[3] Velcea D.-D., Radu I., Calimanescu I., *Stress fatigue life optimization of an airplane landing gear lever arm using ANSYS nCODE*, Constanta Maritime University Annals Year XVI, Vol.24

[4] Velcea D.-D., Radu I., Calimanescu I., *Damage analysis of a gear box shaft under upset working conditions using ANSYS nCODE*, Constanta Maritime University Annals Year XVI, Vol.24

[5] CAD model available at www.grabcad.com

Despre autori

Șef lucr. dr. ing. **Ionuț Cristian SCURTU**
Academia Navală"Mircea cel Bătrân" Constanța

Absolvent al Academiei Navale „Mircea cel Bătrân", promoția 2010, doctor inginer din anul 2015. În prezent, şeful biroului tehnico-administrativ cercetare științifică al Academiei Navale „Mircea cel Bătrân" și cadru didactic asociat. Domenii de competență: Analiza CFD software Ansys, Analiză software structuri mecanice, dinamica structurilor plutitoare, construcția navelor.

Șef lucr. dr. ing. **Adrian POPA**
Academia Navală „Mircea cel Bătrân" Constanța

Absolvent al Academiei Navale „Mircea cel Bătrân", locțiitor al comandantului (prorector) pentru formarea continuă în cadrul Academiei Navale „Mircea cel Bătrân" și cadru didactic asociat. Domenii de competență: Analiza software Ansys, Analiză software structuri mecanice, dinamica structurilor plutitoare.