DE GRUYTER OPEN
International Conference KNOWLEDGE-BASED ORGANIZATION
Vol. XXII No 3 2016

MILITARY MULTIPURPOSE PYLON DEIGNED FOR 100TF LOAD

Adrian POPA, Ionuț-Cristian SCURTU, Beazit ALI, Marian RISTEA

"Mircea cel Bătrân" Naval Academy, Constanța, Romania
adrian.popa@anmb.ro

***Abstract**: The breakthrough in static structural, buckling and modal computer-simulated results can be used for new military purpose applications. The development of hardware resources leads to better simulation in mechanical analysis. Based on the Ansys software, the values presented for the pylon result from a complex analysis. The present paper aims at defining the working parameters for a specific load of 100 tf, as well as presenting the actual software capabilities for military applications. The information provided is valuable for the emerging technologies and the military techniques, being based on the ANSYS simulation.*

Keywords: multipurpose, pylon, Ansys, stress, load design

Introduction

The multipurpose pylon presented in fig.1 was designed in the Ansys Design Modeler and can be used in applications for antenna support, launching site for military apps, and easy to use fixture for offshore installations.

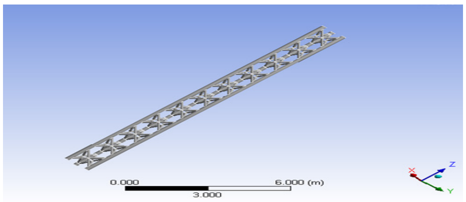

Fig.1. Geometry in Designmodeler

For the case study, we analysed the pylon, using the finite element method, with the software Ansys (fig.2), the height is 13.2 [m], the dimension used for triangle cross section is 1[m]. The material properties are presented in table 1.

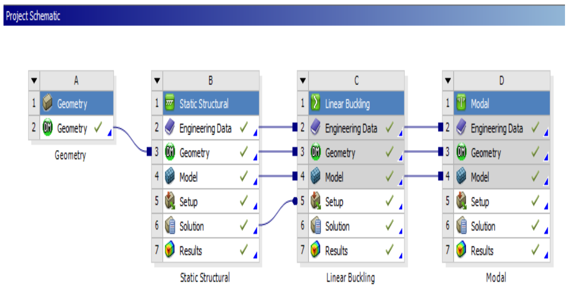

Fig.2. Project schematics

In this paper we will analyse the displacements and the von Mises Stress created by the load of 100 tf. The material used is naval steel with the characteristics presented in table 1.

Table 1 Material characteristics

Elastic modulus	2.10E+11	N/m^2
Poisson's ratio	0.28	NA
Shear modulus	2.35E+10	N/m^2
Mass density	7700	kg/m^3

Defining Geometry in Ansys software

The ANSYS software suite enables you to solve complex structural [1], [2] engineering problems such as linear buckling and vibration modes. Using the

DOI: 10.1515/kbo-2016-0115

finite element analysis (FEA) from Ansys software, you can calculate and simulate the multi purpose pylon. For pylon analysis in Ansys, the model geometry parameters are presented in table 2. The Static Structural software module provides realism [2] in predicting the behavior and performance of pylon under specified load of 100 tf in any military application.

Table 2 Geometry model

Object Name	*Geometry*
State	Fully Defined
Definition	
Type	DesignModeler
Length Unit	Meters
Element Control	Program Controlled
Display Style	Body Color
Bounding Box	
Length X	1.2 m
Length Y	1.066 m
Length Z	13.2 m
Properties	
Volume	0.23266 m^3
Mass	1826.4 kg
Scale Factor Value	1.
Statistics	
Bodies	1
Active Bodies	1
Nodes	1108129
Elements	558752
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

Geometric parameters are automatically calculated in Ansys software for each part and the results are presented in table.3.

Table 3. Geometry parts

Object Name	*Solid*
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	1.2 m
Length Y	1.066 m
Length Z	13.2 m
Properties	
Volume	0.23266 m^3
Mass	1826.4 kg
Centroid X	2.1666e-007 m
Centroid Y	0.28868 m
Centroid Z	6.6 m
Moment of Inertia Ip1	26823 kg·m^2
Moment of Inertia Ip2	26823 kg·m^2
Moment of Inertia Ip3	450.19 kg·m^2
Statistics	
Nodes	1108129
Elements	558752

Mesh

Table 5. Model Mesh

Object Name	*Mesh*
State	Solved
Defaults	
Physics Preference	Mechanical
Relevance	100
Sizing	
Use Advanced Size Function	Off
Relevance Center	Coarse
Statistics	
Nodes	1108129
Elements	558752

Static structural

Table 8 Model accelerations

Object Name	*Standard Earth Gravity*
State	Fully Defined
Definition	

Coordinate System	Global Coordinate System
X Component	0. m/s² (ramped)
Y Component	0. m/s² (ramped)
Z Component	-9.8066 m/s² (ramped)
Suppressed	No
Direction	-Z Direction

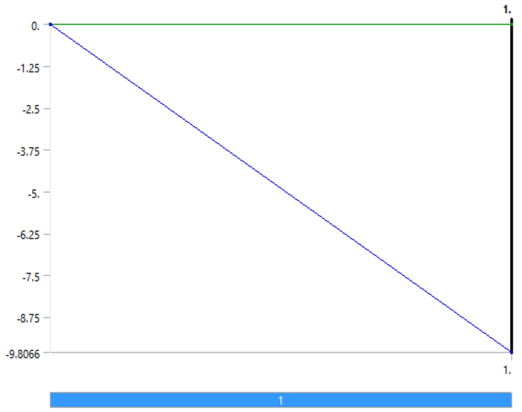

Fig.3 Standard Earth Gravity

Table 9 Static Structural Loads

Object Name	*Fixed Support*	*Force*	*Force 2*	*Force 3*
State	Fully Defined			
Scope				
Scoping Method	Geometry Selection			
Geometry	3 Faces	1 Face		
Definition				
Type	Fixed Support	Force 100000N		

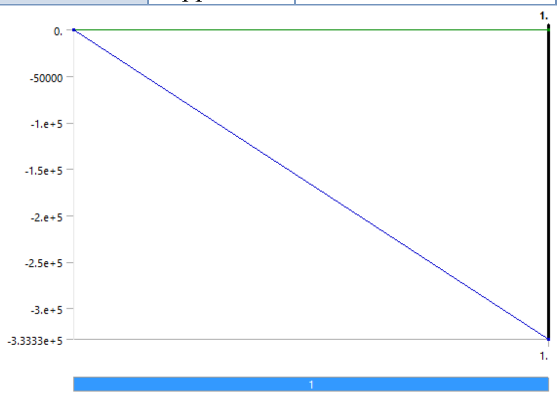

Fig.4 Force in Static Structural

Solution

Calculations in Ansys Static Structural set as a convergence criterion the reduction of the maximum difference between consecutive iterations to 10^{-6} [3], [4]. The mesh shown below will interfere in the solution calculus. The solver will exit calculations at the step where we will obtain numerical convergence and the desired solution is presented in outline bar.

Table 10. Static structural solution

Object Name	*Solution (B6)*
State	Solved
Adaptive Mesh Refinement	
Max Refinement Loops	1.
Refinement Depth	2.
Information	
Status	Done

Table 11. Solution information

Object Name	*Solution Information*
State	Solved
Solution Information	
Solution Output	Solver Output
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
Line Thickness	Single
Display Type	Lines

Linear Buckling Analysis

Buckling is caused by a bifurcation in the solution to the equations of static equilibrium and Ansys software is designed to identify values for defined loads and geometry. Under axial load the pylon is able to sustain increasing load in one of two states of equilibrium: a purely compressed state (with no lateral deviation) or a laterally-deformed state.

Pylon buckling is characterized by a sudden sideways failure of a structural member subjected to high compressive stress.

Table 13. Linear Buckling Analysis

Object Name	*Linear Buckling (C5)*
State	Solved
Definition	
Physics Type	Structural
Analysis Type	Linear Buckling
Solver Target	Mechanical APDL
Options	
Generate Input Only	No

Table 14. Initial Condition Linear Buckling

Object Name	*Pre-Stress (Static*

	Structural)
State	Fully Defined
Definition	
Pre-Stress Environment	Static Structural
Pre-Stress Define By	Program Controlled
Reported Loadstep	Last
Reported Substep	Last
Reported Time	End Time
Contact Status	Use True Status

Table 15.Linear Buckling Analysis Settings

Object Name	*Analysis Settings*
State	Fully Defined
Solver Type	Program Controlled
Output Controls	
Delete Unneeded Files	Yes
Solver Units	Active System
Solver Unit System	mks

Solution

Table 16 Linear Buckling Solution

Object Name	*Solution (C6)*
State	Solved
Adaptive Mesh Refinement	
Max Refinement Loops	1.
Refinement Depth	2.

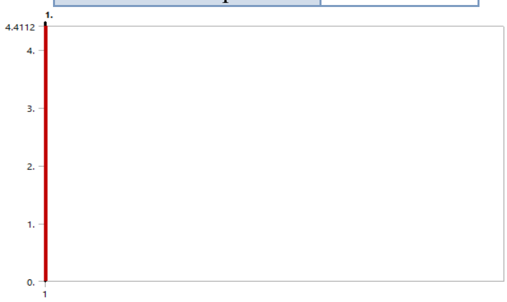

Fig.5. Linear Buckling Solution

Table 18. Linear Buckling Solution

Table 27. Modal solution

Object Name	*Solution (D6)*
State	Solved
Adaptive Mesh Refinement	
Max Refinement Loops	1.
Refinement Depth	2.
Information	
Status	Done

The following bar chart indicates the frequency at each calculated mode.

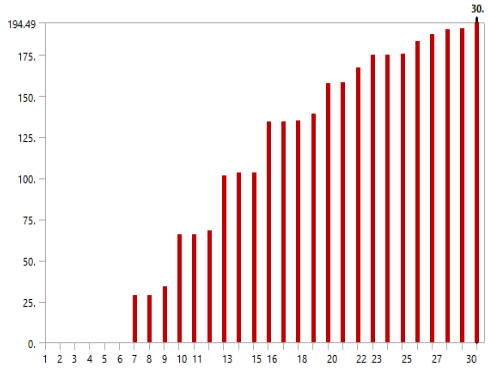

Fig. 6 Modal solution

TABLE 28 Modal Solution

Mode	Frequency [Hz]
7.	28.527
8.	28.558
9.	34.227
10.	65.49
11.	65.646
12.	68.059
13.	101.36
14.	103.19
15.	103.39
16.	134.27
17.	134.43
18.	134.6
19.	138.87
20.	157.64
21.	157.82
22.	166.99
23.	174.61
24.	174.96
25.	175.29
26.	182.89
27.	187.22
28.	190.4
29.	190.72
30.	194.49

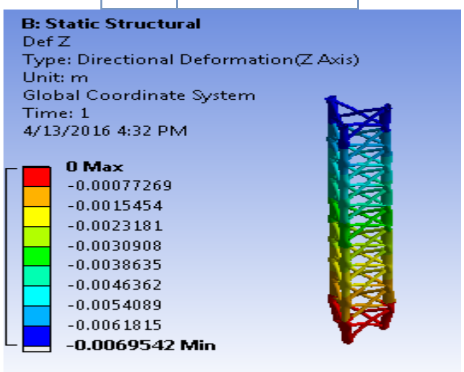

Fig.7 Results in static structural on Z axis

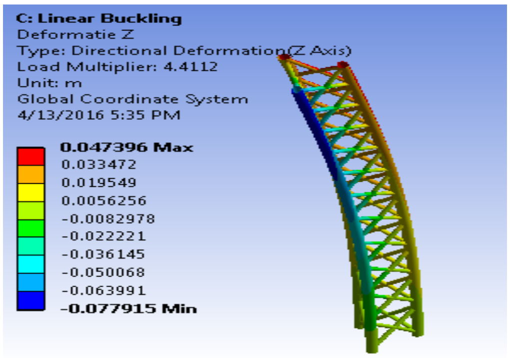

Fig. 8 Results in linear buckling on Z axis

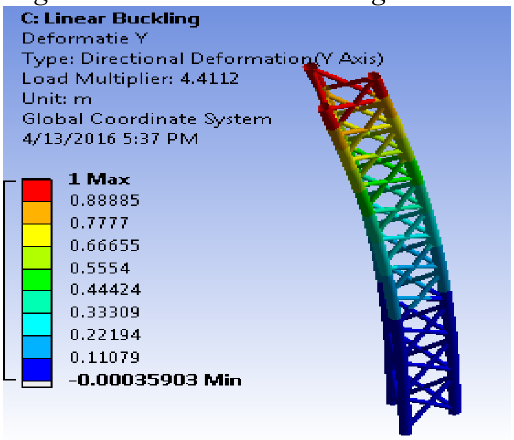

Fig.9 Results in linear buckling on Y axis

Modal analysis results

The modal analysis is used to determine the natural frequencies and mode shapes of the multipurpose pylon. We present several modes of vibration according to operational frequencies, but continuous like pylons have an infinite number of degrees of freedom. All data presented below in figures are natural frequencies and are a starting point for a transient or harmonic analysis [5].

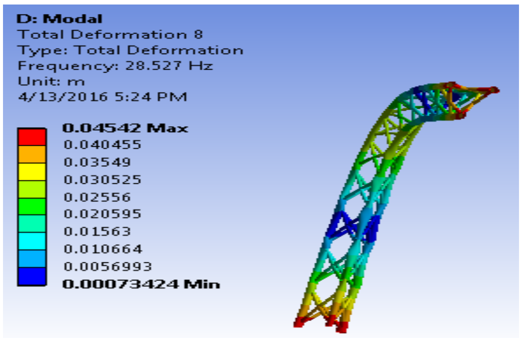

Fig.10 Total deformation 8 results in modal analysis

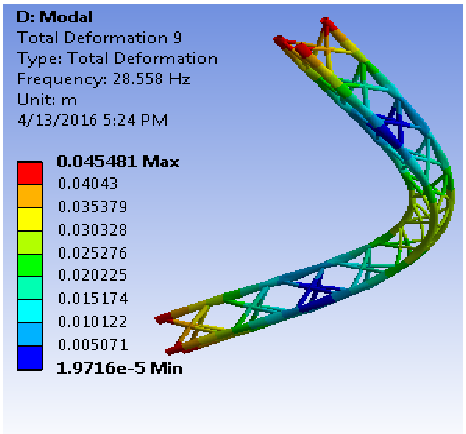

Fig.11 Total deformation 9 results in modal analysis

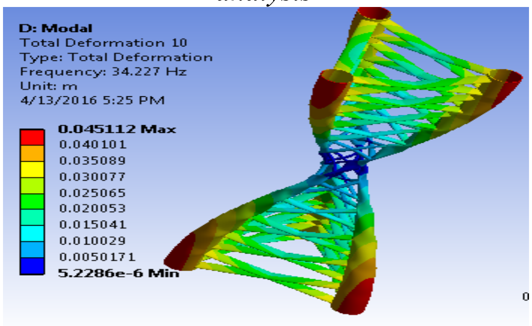

Fig.12 Total deformation 10 results in modal analysis

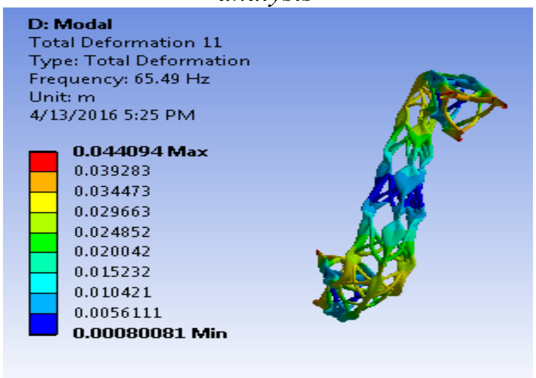

Fig.13 Total deformation 11 results in modal analysis

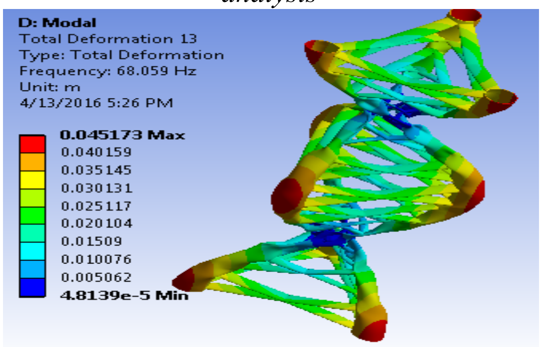

Fig.14 Total deformation 13 results in modal analysis

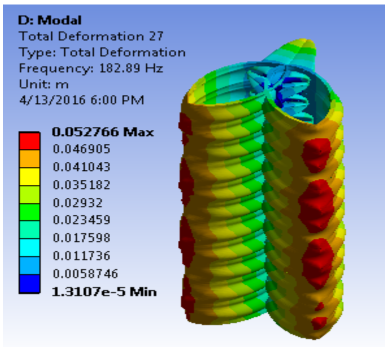

Fig.15 Total deformation 27 results in modal analysis

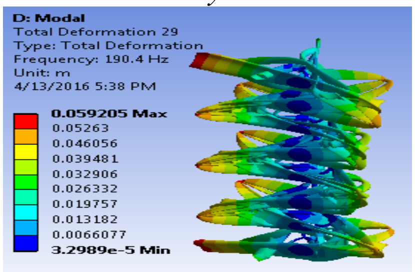

Fig. 16 Total deformation 29 results in modal analysis

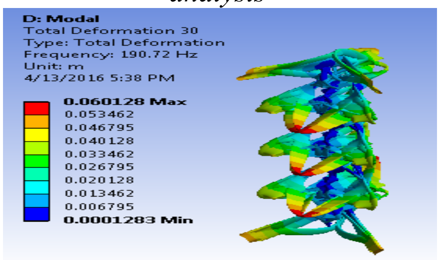

Fig. 17 Total deformation 30 results in modal analysis

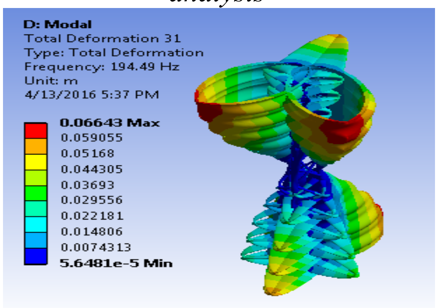

Fig.18 Total deformation 31 results in modal analysis

Conclusions

This work has reviewed and reported the state-of-the-art research in 2016 on pylon design and software analysis in Naval Academy. In this analysis we have shown that the multi purpose pylon design is an evolutionary process on design parameters and strong software analysis.

The analysis presented in this paper can be further explored to develop multipurpose pylons for naval applications.

This paper has reviewed and reported the state-of-the-art research in ANSYS simulations for pylon analysis. In this analysis we have shown static structural, linear buckling and modal based on the presented input geometry parameters and strong software analysis.

The values presented for a triangular-shaped pylon based on simulated Ansys are well-defined parameters required in specific load analysis.

In this case the structure is subjected to the considered axial force of 1000 [kN] and all data presented for each simulation case can be used in further analysis. Considering stresses values, the configurations will withstand any military loads.

References

[1] Scurtu I.C., *Manufacturing and design of the offshore structure Froude scale model related to basin restrictions*, Paper ID: A-30, *Modtech Conference2015,* The Third International Conference on Modern Manufacturing Technologies in Industrial Engineering, Mamaia, Romania, June17-20, 2015.

[2] Scurtu I.-C., Pricop M., Babiuc B., *Study of offshore structure design related to Ansys stress, displacement and vibration modes,* Constanţa Maritime University Annals, Year XIV, Vol.22, ISSN 1582-3601, Constanţa, 2014.

[3] STAN L.-C., *Offshore marine energy in the European area*, Constanţa Maritime University Annals, Year XIV, Vol.24, ISSN 1582-3601, Constanţa, 2015.

[4] STAN L.-C., *Seas and oceans, suppliers of the new and innovative renewable energy*, Constanţa Maritime University Annals, Year XIV, Vol.24, ISSN 1582-3601, Constanţa, 2015.
[5] http://www.ansys.com/Products