"Mircea cel Batran" Naval Academy Scientific Bulletin, Volume XX – 2017 – Issue 1
The journal is indexed in: PROQUEST / DOAJ / Crossref / EBSCOhost / INDEX COPERNICUS / DRJI / OAJI / JOURNAL INDEX / I2OR / SCIENCE LIBRARY INDEX / Google Scholar / Academic Keys/ ROAD Open Access / Academic Resources / Scientific Indexing Services / SCIPIO / JIFACTOR

MESHING AND 3D MODELLING FOR SHIP CONSTRUCTION ELEMENTS

Adrian POPA[1]
Ionut-Cristian SCURTU[2]
Beazit ALI[3]
George NOVAC[4]
[1] Associate Professor PhD. Eng. "Mircea cel Batran" Naval Academy, Constanta, Romania, adrian.popa@anmb.ro
[2] Principal Instructor PhD. Eng."Mircea cel Batran" Naval Academy, Constanta, Romania, ionut.scurtu@anmb.ro
[3] PhD. Eng"Mircea cel Batran" Naval Academy, Constanta, Romania, beazit.ali@anmb.ro
[4] Associate Professor, PhD. Eng"Mircea cel Batran" Naval Academy, Constanta, Romania, george.novac@anmb.ro

Abstract: *All construction elements are subjected to immense forces during ship operation. The presented study is made using different mesh for high stress area on the transverse element presented. The paper presents static loading analysis for the transverse beam model based on Ansys software results.*
Key words: *transverse beam, statical loads, ansys software, meshing.*

1. Introduction

In this paper we analyzed the mechanical resistance of a tranverse beam structure used in naval construction. The content has exposed particularities of the model structure, but operations were performed and numerical computation of Ansys Workbench software. . Naval technology development has reached a very advanced nowadays and their development will continue for the purposes of weight reductions, which will have effects even of its design or working parameters. Based on ship loads on calm sea we conducted a series of simulations in order to determine which efforts and mesh is showing the best results. The paper updated the mechanical problem of the items ship static approach using modern software programs specially designed for field engineers.

Units

TABLE 1

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

Model (A4)

Geometry

TABLE 2
Model (A4) > Geometry

Object Name	*Geometry*
State	Fully Defined
Definition	
Source	K:\21_files\dp0\SYS\DM\SYS.agdb
Type	DesignModeler
Length Unit	Meters
Element Control	Program Controlled
Display Style	Body Color
Bounding Box	
Length X	42.017 m
Length Y	9.6403 m
Length Z	10. m
Properties	
Volume	290.41 m³
Mass	2.2798e+006 kg
Scale Factor Value	1.
Statistics	
Bodies	1
Active Bodies	1
Nodes	48248
Elements	26366
Mesh Metric	None
Basic Geometry Options	
Parameters	Yes
Parameter Key	DS
Attributes	No
Named Selections	No
Material	No

DOI: 10.21279/1454-864X-17-I1-044

Properties	
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
Temporary Directory	C:\Users\Jhon\AppData\Local\Temp
Analysis Type	3-D
Decompose Disjoint Geometry	Yes
Enclosure and Symmetry Processing	Yes

TABLE 3
Model (A4) > 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	42.017 m
Length Y	9.6403 m
Length Z	10. m
Properties	
Volume	290.41 m³
Mass	2.2798e+006 kg
Centroid X	1.0382 m
Centroid Y	0.87803 m
Centroid Z	0.35564 m
Moment of Inertia Ip1	2.4241e+007 kg·m²
Moment of Inertia Ip2	2.4699e+008 kg·m²
Moment of Inertia Ip3	2.5508e+008 kg·m²
Statistics	
Nodes	48248
Elements	26366
Mesh Metric	None

Coordinate Systems

TABLE 4
Model (A4) > Coordinate Systems > Coordinate System

Object Name	*Global Coordinate System*
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.]

Mesh

TABLE 5
Model (A4) > Mesh

Object Name	*Mesh*
State	Solved
Defaults	
Physics Preference	Mechanical
Relevance	0
Sizing	
Use Advanced Size Function	Off
Relevance Center	Coarse
Element Size	Default

DOI: 10.21279/1454-864X-17-I1-044

Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Fast
Span Angle Center	Coarse
Minimum Edge Length	0.50 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
Patch Conforming Options	
Triangle Surface Mesher	Program Controlled
Patch Independent Options	
Topology Checking	Yes
Advanced	
Shape Checking	Standard Mechanical
Element Midside Nodes	Program Controlled
Straight Sided Elements	No
Number of Retries	Default (4)
Extra Retries For Assembly	Yes
Rigid Body Behavior	Dimensionally Reduced
Mesh Morphing	Disabled
Defeaturing	
Pinch Tolerance	Please Define
Generate Pinch on Refresh	No
Automatic Mesh Based Defeaturing	On
Defeaturing Tolerance	Default
Statistics	
Nodes	48248
Elements	26366
Mesh Metric	None

TABLE 6
Model (A4) > Mesh > Mesh Controls

Object Name	*Face Sizing*	*Face Sizing 2*
State	Fully Defined	
Scope		
Scoping Method	Geometry Selection	
Geometry	4 Faces	1 Face
Definition		
Suppressed	No	
Type	Element Size	
Element Size	0.1 m	
Behavior	Soft	

Static Structural (A5)
TABLE 7
Model (A4) > Analysis

Object Name	*Static Structural (A5)*
State	Solved
Definition	
Physics Type	Structural
Analysis Type	Static Structural
Solver Target	Mechanical APDL
Options	
Environment Temperature	22. °C
Generate Input Only	No

TABLE 8
Model (A4) > Static Structural (A5) > Analysis Settings

Object Name	*Analysis Settings*
State	Fully Defined
Step Controls	
Number Of Steps	1.
Current Step Number	1.
Step End Time	1. s
Auto Time Stepping	Program Controlled
Solver Controls	
Solver Type	Program Controlled
Weak Springs	Program Controlled
Large Deflection	Off
Inertia Relief	Off
Restart Controls	
Generate Restart Points	Program Controlled
Retain Files After Full Solve	No
Nonlinear Controls	
Newton-Raphson Option	Program Controlled
Force Convergence	Program Controlled
Moment Convergence	Program Controlled
Displacement Convergence	Program Controlled
Rotation Convergence	Program Controlled
Line Search	Program Controlled
Stabilization	Off

DOI: 10.21279/1454-864X-17-I1-044

Output Controls	
Stress	Yes
Strain	Yes
Nodal Forces	No
Contact Miscellaneous	No
General Miscellaneous	No
Store Results At	All Time Points
Analysis Data Management	
Solver Files Directory	K:\21_files\dp0\SYS\MECH\
Future Analysis	None
Scratch Solver Files Directory	
Save MAPDL db	No
Delete Unneeded Files	Yes
Nonlinear Solution	No
Solver Units	Active System
Solver Unit System	mks

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

Object Name	*Fixed Support*	*Force*
State	Fully Defined	
Scope		
Scoping Method	Geometry Selection	
Geometry	3 Faces	1 Face
Definition		
Type	Fixed Support	Force
Suppressed	No	
Define By		Components
Coordinate System		Global Coordinate System
X Component		0. N (ramped)
Y Component		-2121. N (ramped)
Z Component		0. N (ramped)

FIGURE 1
Model (A4) > Static Structural (A5) > Force

Solution (A6)

TABLE 10
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

TABLE 11
Model (A4) > Static Structural (A5) > Solution (A6) > Solution Information

Object Name	*Solution Information*
State	Solved
Solution Information	
Solution Output	Solver Output
Newton-Raphson Residuals	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

TABLE 12
Model (A4) > Static Structural (A5) > Solution (A6) > Results

Object Name	*Total Deformation*	*Equivalent Elastic Strain*
State	Solved	
Scope		
Scoping Method	Geometry Selection	
Geometry	All Bodies	
Definition		
Type	Total Deformation	Equivalent Elastic Strain
By	Time	
Display Time	Last	
Calculate Time History	Yes	
Identifier		
Suppressed	No	
Results		
Minimum	0. m	1.8291e-020 m/m
Maximum	4.8344e-009 m	2.8581e-009 m/m

DOI: 10.21279/1454-864X-17-I1-044

Minimum Value Over Time		
Minimum	0. m	1.8291e-020 m/m
Maximum	0. m	1.8291e-020 m/m
Maximum Value Over Time		
Minimum	4.8344e-009 m	2.8581e-009 m/m
Maximum	4.8344e-009 m	2.8581e-009 m/m
Information		
Time	1. s	
Load Step	1	
Substep	1	
Iteration Number	1	
Integration Point Results		
Display Option		Averaged
Average Across Bodies		No

Material Data

Structural Steel

TABLE 13
Structural Steel > Constants

Density	7850 kg m^-3
Coefficient of Thermal Expansion	1.2e-005 C^-1
Specific Heat	434 J kg^-1 C^-1
Thermal Conductivity	60.5 W m^-1 C^-1
Resistivity	1.7e-007 ohm m

TABLE 14
Structural Steel > Compressive Ultimate Strength

Compressive Ultimate Strength Pa
0

TABLE 15
Structural Steel > Compressive Yield Strength

Compressive Yield Strength Pa
2.5e+008

TABLE 16
Structural Steel > Tensile Yield Strength

Tensile Yield Strength Pa
2.5e+008

TABLE 17
Structural Steel > Tensile Ultimate Strength

Tensile Ultimate Strength Pa
4.6e+008

TABLE 18
Structural Steel > Isotropic Secant Coefficient of Thermal Expansion

Reference Temperature C
22

TABLE 19
Structural Steel > Alternating Stress Mean Stress

Alternating Stress Pa	Cycles	Mean Stress Pa
3.999e+009	10	0
2.827e+009	20	0
1.896e+009	50	0
1.413e+009	100	0
1.069e+009	200	0
4.41e+008	2000	0
2.62e+008	10000	0
2.14e+008	20000	0
1.38e+008	1.e+005	0
1.14e+008	2.e+005	0
8.62e+007	1.e+006	0

TABLE 20
Structural Steel > Strain-Life Parameters

Strength Coefficient Pa	Strength Exponent	Ductility Coefficient	Ductility Exponent	Cyclic Strength Coefficient Pa	Cyclic Strain Hardening Exponent
9.2e+008	-0.106	0.213	-0.47	1.e+009	0.2

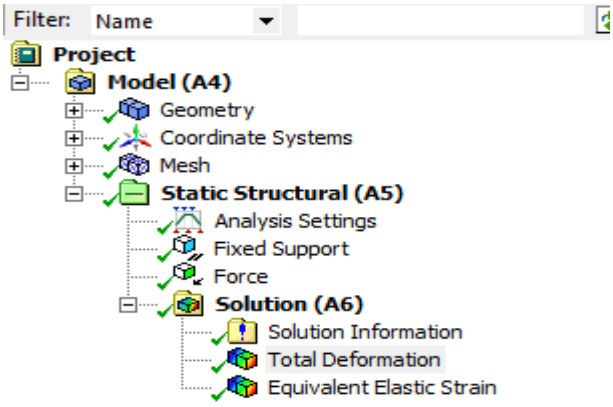
TABLE 21
Structural Steel > Isotropic Elasticity

Temperature C	Young's Modulus Pa	Poisson's Ratio	Bulk Modulus Pa	Shear Modulus Pa
	2.e+011	0.3	1.6667e+011	7.6923e+010

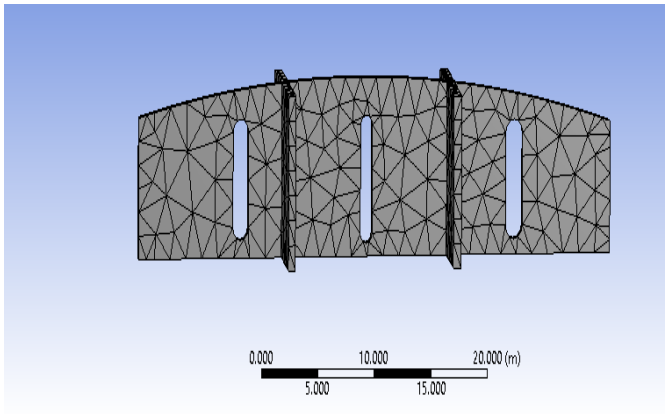
TABLE 22
Structural Steel > Isotropic Relative Permeability

Relative Permeability
10000

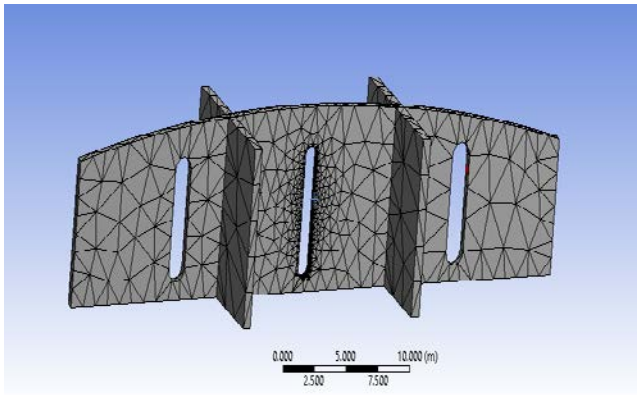
DOI: 10.21279/1454-864X-17-I1-044



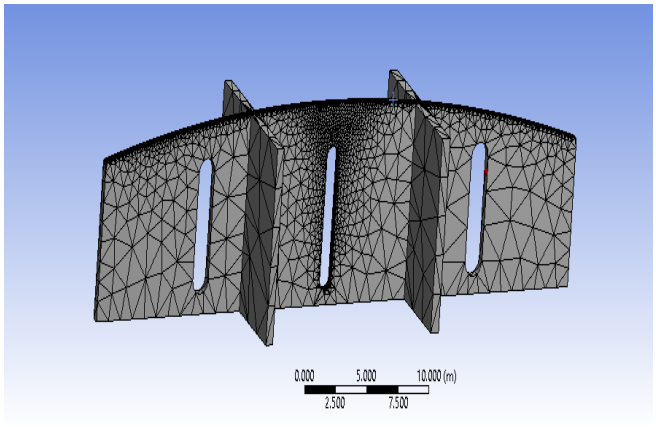
Settings within Ansys Workbench Program Outline are made and the steps required to achieve the Ansys Workbench correct simulation. You can manage the taught elements considered for analysis and geometry, how it will achieve meshing and managing various requests that appear on track in the study.



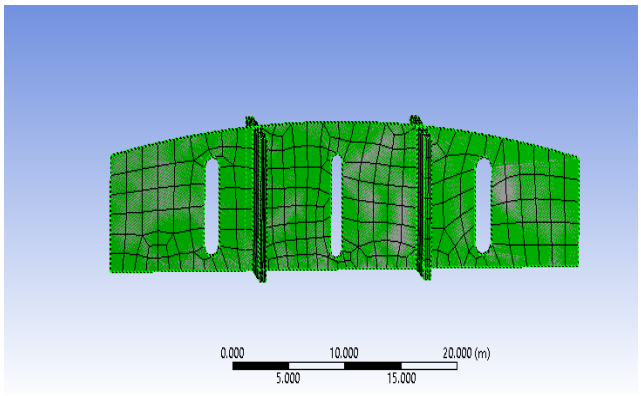
3D geometry and automatic mesh



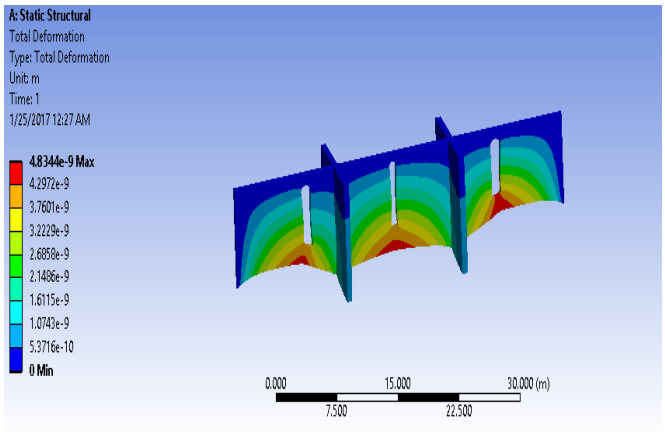
3D geometry transverse element of the vessel.
Meshing controlled situation with refining zone 1



Controlled meshing situation with refining zone 2

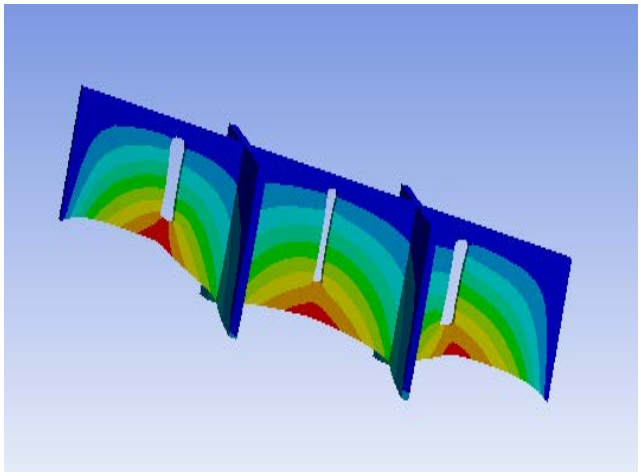


Presentation working method



Results 3D to the ship structure (deformations 1)

DOI: 10.21279/1454-864X-17-I1-044



Results 3D to the ship structure (deformations 2)

Conclusion

In this study, Ansys statical loads method is used for an actual simulation case study based on a transverse ship beam. Due to impossibility of experimental tests, the presented beam model was analised in Ansys numerical simulation. All result presented as beam deformation and stress for different mesh use. The Ansys simulation presented for ship beam is a fast way to solve static load analysis for any ship element.

Bibliography

[1] Ilie Patrichi, Ali Beazit, Iordan Novac, *The strenght structure and the stress of shell plating made of fibreglass reinforced plastics FRP*, Analele Universităţii Maritime din Constanţa, anul VI, vol. 8, 2005, pag. 265 - 274, ISSN 1582-3601.

[2] Călimănescu I., Stan L. C., *Computer fluid dynamics (CFD)study of a micro annular gear pump*, Atom 2016, Conference Paper.

[3] Stan L. Călimănescu I.,. C.A *New innovative turbocharger concept numerically tested and optimised with cfd*, 2016, Conference Paper.

[4] *** Ansys software documents and help

DOI: 10.21279/1454-864X-17-I1-044