

### Maritime Systems Engineering Department

Texas A&M University at Galveston

# A Guide on Using DNV Sesam for Design and Analysis of Ships



Author: Christopher Karl Micklitz

Instructor: Professor Masoud Hayatdavoodi

**Revision**: SUMMER\_2015

## Preface

This report provides a reference capable of enabling the user with knowledge and skills needed to perform a hydrostatic and structural analysis of a ship using DNV Sesam modules GeniE, HydroD, and Sestra. In order for this report to remain effective, it must be updated to the current revision of the DNV software that is available to the user, and tailored to the projects assigned within the department. The original report was constructed with GeniE V6.4-08, HydroD V4.6-03, and Sestra V8.5-01.

REVISION	AUTHOR/EDITOR	DATE	Contact Information
SUMMER_2015	Chris Micklitz/Prof. Hayatdavoodi	31-Jul-15	ckmicklitz@yahoo.com

### Abstract

The hydrostatic and structural analysis of a vessel is presented using DNV Sesam modules GeniE, HydroD, and Sestra. The report focuses on the design and modelling of the hull, primary, secondary, and tertiary structural elements of the ship. In addition to the physical modelling of the ship, the report will include the necessary steps required to migrate the model between software modules.

## Nomenclature

Acronym	Explanation							
ABS	American Bureau of Shipping							
FEM file	SESAM Input Interface File							
LMB	Left Mouse Button							
RMB	RMB Right Mouse Button							
PORT	PORT Left Side of Ship When Facing Forward							
STBD	STBD Right Side of Ship When Facing Forward (Starboard)							
DNV	DET NORSKE VERITAS							
XVERSE	Transverse							
Т	Draft (Ship Still Waterline)							
AP	Aft Perpendicular							
Fwd	Forward							
FP	Fwd Perpendicular							

# **Table of Contents**

Preface1				
Abstract	2			
Nomenclature	2			
Chapter 1- Introduction	5			
Chapter 2- Vessel Information	6			
Chapter 3- GeniE	7			
Section 1 – Panel Model	7			
Establishing a Workplace	7			
Define units:	7			
Define material				
Define section properties				
Define Plate Thicknesses				
Specifying Mesh Settings				
Poly-Curve Dialog				
Cover Curves/Plating				
Plate/Shell Orientation				
Guide Planes				
Dividing Plates				
Copy/Mirror				
Prepare Panel Model for HydroD				
Section 2- Structural Model				
Centerline Girder				
Beam Orientation				
Additional Bottom Longitudinal Beams				
Transverse Beams				
Longitudinal Stiffeners				
Prepare to Export to HydroD				
Meshing				
Chapter 4 – HydroD				
Stability Wizard				
Saving the Stability Analysis				
Chapter 5 – Sestra	45			

Creating a Load Analysis	
Establish Boundary Conditions	
Chapter 6 – Suggestions/Common Errors	55
Bibliography	56
Appendices	57
Appendix 1: Table of Offsets	57
Appendix 2: Ships Stations	58
Appendix 3: Hull Lines	59
Appendix 4: Stability Report	60
Appendix 5: Condensed Static Sestra Results	61
Appendix 6: Condensed Eigenvalue Sestra Results	64

## **Chapter 1- Introduction**

Student feedback as well as instructors observations have consistently expressed a need for supplemental training materials outside of those provided with the DNV Sesam software package. The materials provided by DNV presents a comprehensive reference needed to navigate and utilize the software when creating objects that comprise of basic and complex geometric shapes. However, student feedback as well as instructors observations have consistently noted that DNV's supplemental training guides could be shortened to a concise guide that better suits the needs of the department. It was also suggested that more focused information regarding the navigation between DNV modules would be beneficial too.

Prior to this guide, a formal mechanism in which senior generations of students could transfer knowledge and expertise related to the DNV Sesam software suite was not available. Through proper upkeep and maintenance, this document would facilitate a steeper learning curve with respect to DNV Sesam for upcoming students, thereby allowing them to update this document with further detail pertinent to department curricula, and thus the cycle continues.

This report utilizes DNV's "A2\_GeniE\_Semisub\_pontoon" PDF guide, "GeniE\_UM\_Vol3", "HydroD\_UM", and "Sestra\_UM", as well as information that was obtained from the first-hand use of DNV Sesam that was not explicitly delineated in their training materials. The main objectives of this guide is to perform and concisely document the steps used to perform the hydrostatic and structural analysis of a ship using DNV Sesam modules GeniE, HydroD, and Sestra.

Initially, this guide begins with steps taken from DNV's "A2\_GeniE\_Semisub\_pontoon" PDF guide. Beginning at the 'Poly-Curve Dialogue' section of Chapter 3, the images and text are almost entirely created by the author unless otherwise noted. The report is organized and structured following the general progression of steps taken in order to complete the stated goals. Chapter 3 is divided into two sections: Section 1, Panel Model, utilizes Sesam GeniE to create a panel model of the ship's hull and export it as a T1FEM file; Section 2, presents the generation of a structural model of the ship to be exported as a T3FEM file. It is important to understand that Chapter 3, Section 2 is written as a continuation of the previous section. In Chapter 4, the hydrostatic and stability analysis of the ship are performed using Sesam HydroD. Structural analysis of the ship is covered in Chapter 5 utilizing Sesam Sestra which is executed through the GeniE module. Chapter 6 contains Suggestions/Common Errors that pertain to the use of DNV Sesam in the context of this guide.

## **Chapter 2- Vessel Information**

The ship modelled in this report is based on the Armfield ship model located in the Naval Architecture laboratory. The laboratory model is a 1/70th scale model, and in this report it is modelled to full scale. The Table of Offsets of the ship are provided in Appendix 1. An image of the Armfield Ship Model is shown and resembles the model located in the Naval Architecture laboratory. In addition to the hull and bulkheads, the ship will be modelled with structural members determined utilizing American Bureau of Shipping (ABS) Rules for Building and Classing Steel Vessels 2009. In this guide, the ships: machinery, superstructure, and propeller are not modelled or analysed. However, in the complete study of a ship, the aforementioned elements of the ship should be evaluated.



http://discoverarmfield.com/media/filter/l/img/general\_cargo\_vessel.jpg

Bulk Head #	Distance From A.P. (meters)
1	12.53
2	33.53
3	53.48
4	74.34
5	95.2
6	116.27
7	136.92
8	156.38
Transverse Bulk Head	Centerline from Bulkhead 2 - 3

The hull lines of the vessel are provided in Appendix 3. There are 8 transverse bulkheads and 1 longitudinal bulk head. There locations in the vessel are annotated in the diagram below.

This guide was performed using a draft of 9.403 meters and a KG of 5.2 meters. This yielded a metacentric height of 1.101 meters

## **Chapter 3- GeniE**

In this guide, Sesam GeniE is used to generate and export two FEM files for use in HydroD, that consist of the hull (shell) and structural (beams) elements of the ship. The 'Panel Model' contains the shell only, while the 'Structural Model' consists of the hull and beams. The initial portion of the first section begins with steps taken from DNV's "A2\_GeniE\_Semisub\_pontoon" PDF guide. Beginning at the 'Poly-Curve Dialogue' section, the remainder of the images and text are entirely created by the author.

### Section 1 – Panel Model

#### **Establishing a Workplace**

• When GeniE is first opened, the user is prompted with the dialog box shown below. Ensure the boxes are selected as shown and select "OK"



#### **Define units:**

Meters and Newtons

- Start the program and open a new workspace File|New Workspace
- Specify name 'Panel\_Model' and use the default values for database units
- Click OK when done

Semi_pontoon						
Location:						
C:\DNV\Workspaces\Semi_pontoon						
Set Database Units 🔽 Enable tolerant modelling						
Length	m	~				
Force	Ν	~				
Temperature	delC	Ψ.				
Command I	nput File					

#### **Define material**

- Use the command Edit|Properties and select Material. Select Create/Edit Material to give the details for St52 (remember to tick "Allow edit").
- Set the material type to default
- Click OK when done.

<none></none>	Use Description   no material	Yield			
		Create/Edit Materia	I Properties		
		Linear Isotropic Material	Isotropic Shear M	laterial	
		St52	•		
		Existing material	Yield	235000000 Pa	[Pa]
		<i>σ</i>	Density	7850 Kg/m^3	 [Kg/m^3]
1		young	Young	2.1e+011 Pa	[Pa]
Create/Edit Mat	terial Set Default		Poisson	0.3	
Cicate/Eart Hat	Sebendar		Thermal	1.2e-005 delC^-1	[delC^-1]
	ок	Canc	Damping	0.03 N*s/m	[N*s/m]
			Tensile		[Pa]

#### **Define section properties**

The Section profiles required for the user's vessel should be determined by the appropriate "Code Manual" (ABS was used to determine the structural components utilized in this report.)

- Use the command Edit|Properties and select Section. Select Create/Edit section to start defining the sections. Select "Section Library"
- Section profiles Tbar425x120x12x25, Tbar575x150x12x25, and Tbar885x200x14x35 are found from section libraries.



• Find the right section library from "Browse" and select the library 'tbar' (a library containing typical Tbar ship profiles)

Create/Edit Cross Section	·	Open Section Li	hrany on t ci\nto	aram files\days\aeaie da	4-01\libraries	X
Pipe Bar Box I Section I Unsymmetrical I General Section Cone	Section Channel Section Library	Look in:	Libraries	gram mestarivsigene ad	- 0 p 🕫 🕬	
Library: × Browse	」)	(ca)	Name 🔺		Date modified	- Туре
Subset:		<b>S</b>	aisc_v3.kzy		2012-06-29 21:16	KZY File
		Recent Places	🖀 anglebar.xm	1	2012-06-29 21:16	XML Doci
Filter			BS4_Section	s_Part1_1993.xml	2012-06-29 21:16	XML Doci
🗖 Channel 🗖 Pipe		Dealdars	🖆 bulb.xml		2012-06-29 21:16	XML Doci
Section type E Box E I		Desktop	🖺 flatbar.xml		2012-06-29 21:16	XML Doci
E Ber EL			Hatenal_lion	aryiimi	2012-06-29 21:16	XML Doci
Name match			NSF_EN.KZY		2012-06-29 21:16	KZY File
		cionantes	📑 tbarxml		2012-06-29 21:16	XML Doci
L Regula expression a -						
Max height/diam [m]		SHAL0189				
Min height/diam [m]		<b>6</b>				
		N				
Number of sections to be imported:		Network	I			•
		F	le name.	tbarxmi		Open
_importC	AI C Selected		les of type:	Section library (*.KZY,*.X/		Cancel
ок	Cancel Apply					

### **Define Plate Thicknesses**

Plate thicknesses were determined following **ABS Rules for Building and Classing Steel Vessels 2009:** 5B, <u>Common Structural Rules for Bulk Carriers</u>: 5B Chapter 6 Section 1 Table 2 (Minimum net thickness of plating), and 5B, Chapter 3 Section 3 Table 1 (Corrosion addition on one side of structural members)

Define the plate thicknesses Pl12, Pl16 and Pl21 as follows:

- Use the command Edit|Properties and select Thickness.
- Select Create/Edit Thickness to start defining the thickness properties
- Click OK when the desired values have been entered

Properties				×				
Section Materia	Thickness	Effective Flange	Corrosion Addition	Plate Ty	•			
Name	Use	Description	Thickness					
<none></none>		no thickness						
					ļ ,			
						Create/Edit Thickness	N	-
						Thickness	63	
						P112	•	
	_					New thickness	$\frown$	
Course Kida Th		Lange al					hickness 0.012 m	[m]
Create/Edit In	iomess	et Default						
					1	Thickness		
		OK	Cancel	Apply				

#### **Specifying Mesh Settings**

The mesh settings for automatic mesh creation will be set next, but these values can be adjusted at any time prior to the mesh generation.

- Edit|Rules|Meshing. In addition to default settings select:
  - Use second order elements
  - Round off Mesh Density

eral   Max/Min Angle   Jacobi   Eliminate edge   Ch	ord Height
Ceneral FEM options Use second order elements Superelement type:  Model topology (Performed before meshing)  Always simplify topology  Split periodic geometry if needed  Element preferences  Prefer regular mesh (m x n)  Alway triangular elements  Define the mass a poole mass	Other preferences         Include unused properties         ✓ Automatic load combination FEM numbering         ✓ Round off Mesh Density         ✓ Adjust number of elements         ✓ Write loads separate         ✓ P?         Naming preferences         ✓ Use long LoadCase names         ✓ Use long Property names
□ Use drilling elements         ♥?           ▼ Use eccentric hinges         ♥?	Face mesher         C Advancing front quad mesher         C Advancing front triangle mesher
Scantling idealizations Thickness: msGross Ignore beam eccentricities Use co-centric beams 9?	Edge mesher ○ Uniform distribution ○ Linear distribution Idealisations I Remove internal vertices ♀? □ Remove internal edges ♀?

#### **Poly-Curve Dialog**

This will be the mechanism by which the ships offsets are transformed into Guiding Geometry.

• In the model tree, right-click on Curves and select 'New Poly Curve.'



One approach to creating the station curves involves utilizing the station line coordinates in the format shown in Appendix 2. Coordinates can be copied into the poly-curve dialogue box to generate the PORT half of the stations.

These values can be entered into the dialogue box manually from Appendix 1 as well.

G C:\User:	C:\Users\Student.PMEC142-05\Desktop\CKM_GENIE\g_ship.gni - Genie (DirectX9)								<u> </u>	×								
File Edit	ile Edit View Guiding Geometry Structure Loads Tools Help								Ho	me Ince	rt Page	Lavout	Eormu	lac Dati	a Peviev			
D 🚅 🛙	l e	× 🤋 👧 🗛	<u>ው ጉል ከተ</u>	8 1 🖌 🔜		a. a A	s - 27 - 17	afault digg	olav			inc insc	it iuge	Luyou	. Tonia	us Dutt		v
-0-0	Ĵ≛ P	oly Curve						23			& Cut		Calibri		* 11 *	A A	= =	39
15 <u>1</u> 2			-						bar-	Pas	te	y -	BZT	I -	- 3	- A - T	= = =	4
• • •	Nan	e: Curve7599	_	Fit Curve To V	iew	OK				· ·	V Forr	nat Painter			-	-		
1.		, Curve definition INVA	ALID	Auto Curve Ty	/pe	Cancel				-	Cipboard	191	(-	ru	nu	19		P
1		Label curve point	s				4		-		S61		0	Ĵx	16.016			
	Er	force tangent vectors									А	В	С	D	E	F	G	H
□ <b>□ □</b>	Г	End 1								1						F	ULL S	C/
	Г	End 2								1				1				-
		,	1							2		Station AF	)			Station 2		1
÷-		X [m]	Y [m]	Z	[m]		Curve Type			4	0	0	0		4 172	0	0	
	1						Spline			5	0	0	0.5005		4.172	0.42	0.5005	
	2						Spline	-		6	0	0	1.001		4.172	0.21	1.001	t
	3						Spline	•		7	0	0	2.002		4.172	0.14	2.002	
	4						Spline	-		8	0	0	3.003		4.172	0.07	3.003	Γ
	5						Spline	-		9	0	0	4.004		4.172	0.07	4.004	
	6						Spline	-		10	0	0	5.005		4.172	0.14	5.005	
	7						Spline	-		11	0	0	6.006		4.172	0.14	6.006	
	8						Spline	•		12	0	0	7.007		4.172	0.42	7.007	
-	9						Spline	-		13	0	0	8.008		4.172	1.4	8.008	
L	10						Spline	-		14	0	1.19	9.009		4.172	2.8	9.009	
	11						Spline	•		15	0	2.38	10.01		4.172	4.06	10.01	4
	12						Spline			16	0	3.5	11.011		4.172	5.18	11.011	-
	13						Spline			1/	0	4.41	12.012		4.172	6.23	12.012	-
	14						Spline			18	0	5.18	13.013		4.172	0.93	13.013	-
	15						Spline			20	0	5.74	15.015		4.172	7.45	15.015	-
	16						Seline	- <b>-</b>		20	0	6.58	16.015		4.172	9.22	16.015	+
	10						Calling			21	0	0.50	10.010		4.172	0.33	10.010	-
	10						Calling			23		Station 6				Station 7		T
	18						spiine			24	25.046	0	0		33.39	0	0	$\top$
	19						Spline			25	25.046	3.5	0.5005		33.39	6.02	0.5005	T
	20						Spline	-		26	25.046	4.76	1.001		33.39	7.28	1.001	T
										27	25.046	6.16	2.002	1	33.39	8.61	2.002	

Figure 1, resembles what should be visible after the initial 25 PORT stations have been entered (AFT perpendicular is on the left).



Figure 1: 25 PORT Stations

• Next, the forward most station labelled BOW Extra (Appendix 2) is added to the model using the method explained previously.

This is done to complete the fore section of the ship, beyond FP. At this point, the user should have the 26 polycurves comprising of the 25 stations from the table of offsets plus the additional BOW Extra station.



#### **Cover Curves/Plating**

The outer hull will be modelled for the panel model needed for later analysis in HydroD.

- This is accomplished by selecting two adjacent stations
- Right Mouse Button (RMB)
- Then selecting Cover Curves.



After plating the first set of stations, the model should look similar to the image shown below. This image will vary based on the order in which stations are plated.



• This method will be repeated until all stations are covered to include to extra BOW station.

At this point, the ship will resemble Fig. 2, shown below with the FWD perpendicular located on left.



Figure 2: Completed PORT-Side Plating

#### **Plate/Shell Orientation**

After creating all of the plates (panels) you may notice that some appear red as opposed to grey. The color represents the orientation of the plating (i.e. what is the front and back side). All of the plates should have the grey side facing outward, away from centerline.

- To change the orientation, Left Mouse Button (LMB) a plate
- Then RMB and select Flip Normal

Note that background, plating and beam colors may be modified by the user to enhance visibility.



• In the Plate/Shell Normal dialogue, select APPLY, then click cancel.



The plate should now appear grey.



• Perform this method until all plates are oriented properly (grey facing outward).

Next, create a model curve (curve following surface topology) along the top edge of the newly created PORT plates.

• Select Guiding Geometry| Curves on Surfaces |Model Curves

View	Guiding Geometry	Structure Loads Tools Help
	Points	<u> </u>
₩ •   • N ■ • ≉	Lines Polylines Conic Sections Free-Form Curves Split/Join Curves	
015 07	Curves on Surface	es 🕑 🖬 Model Curve
	Planes	Plate/Shell Intersection with Plane Plate/Shell Edges
	Transformations	•
	Advanced	•

• Click the upper most point of station, followed by the upper most point of an adjacent station (Example points shown in red).



- Once all of the curves have been created along the top edge of the PORT hull, select all of them.
  - First select the first line and press and hold the SHIFT key
  - While holding SHIFT, select the remaining lines until all are highlighted
  - Now, release the SHIFT key, RMB on the highlighted line and select JOIN



• Now complete the JOIN command by selecting Apply.



#### **Guide Planes**

Now, we will create horizontal guide planes that intersect the newly created hull at the T = 9.403 m waterline.

- In the model tree, RMB on Guiding Geometry and select New Guide Plane...
- Enter the coordinates for the T=9.403 m waterline, and create a guide plane.



The T = 9.403 meter draft is shown below in Fig. 3, and is specific to this vessel and guide.

] ∕ ▼   <b>□</b> ▼ <i>@</i> ▼   <u>▲</u> ▼   K → ▼	
/ •   <b>□</b> • <b>Φ</b> •   <b>Δ</b> •   <b>K</b> • • •   10 Jul 2015 04:34 (cm, hydrod Analysis1	Properties         Com           Object Properties         GuideRhane2           Type:
	$\begin{array}{c c c c c c c c c c c c c c c c c c c $
x	

Figure 3: 9.403m Waterline Guide Plane

#### **Dividing Plates**

- Next, the PORT hull plates will be divided at the 9.403m waterline
  - This is done in order to establish the ships draft for HydroD
    - Select all of PORT hull plates, RMB|Divide



- In the dialogue box, select divide plates with plane
  - Select 3 corners of the 9.403m guide plane that was just created
  - Click Apply then Cancel



• Now the plates have been divided at a draft of T = 9.403 m and should resemble the image below.



### **Copy/Mirror**

At this point, the PORT panel model is ready to be mirrored.

- Select all items created to this point
- RMB|Copy



In the Copy dialogue box.

- Select the Mirror tab
- Fill in the values as depicted below
- Select Apply then Cancel



- On the AFT end of the ship, create a line connecting the uppermost points of the PORT & STBD aft most stations
- Select the newly XVERSE line and the PORT & STBD station curves.
  - RMB|Cover Curves





### **Prepare Panel Model for HydroD**

• Select all of the hull plates below the T=9.403 m draft



- RMB the highlighted region and select properties
  - $\circ$  Select the Wet Surface tab
  - Select WS1 and click Apply
  - Select the Front box and click OK
  - Click Cancel on the main dialogue box

Object Properties	Material	Thickness	Corrosion	Addition	Plate Type	Mesh Property	Local Syste	m Wet Surface	Mesh Option	Permeable	Structure
Name	Use	Descriptio	n								
<none></none>		no wet su	rface								
🔵 WS1	0	Wet Surfa	ace								
				Į	Select whi want the p Front Back	ch side(s) of the p property to be as	olate you signed to	OK Cancel			

Next, apply the load case

- Under the Load tab, select load case
- Complete the dialogue box as shown and select Apply then Cancel

File Edit View	Guiding Geometry Structure Loads Tools Help
🗅 🗳 🖬 🔤	🗙   💡 🕼 🕭 🖑 🗟 🏹 🖉 ! 🛷 🐨 🗗 🗊 🐨 🖌 🖉 🕶 Default display
] 15: 55   14: -	
	Insert Load Case       Name:       LC1       Image:       Image:

Apply the mesh

• Select the entire model and press Alt-M (this will mesh the model.)



Export the model as a FEM file

• File|Export|FEM File...

File Edit View Guiding Geometry	Structure Loads Tools Help
New Workspace       Ctrl+N	
Export •	<u>F</u> EM File
Read Command <u>F</u> ile	Results SIN File
Recent Co <u>m</u> mand Files Recent <u>W</u> orkspaces	<u>X</u> ML Concept Model <u>I</u> ntergraph PDS (SDNF file) <u>C</u> adCentre PDMS (SDNF file) <u>G</u> enie journal file (JS file)
	<u>R</u> ule Loads XML file

### Section 2- Structural Model

This section is written such that the user has the following initial conditions met:

- A panel model T1FEM file has been exported with mesh applied to LC1
- All of the previously defined: member sections, plate thicknesses, material properties and units have been selected and/or defined.
- Identical Mesh properties have been selected, with the exception of the Super Element type, which should be set to 3 vice 1.
- The original 25 PORT ship stations and Bow Extra poly-curves have been created with the value listed in Appendix 1.
- A complete and empty hull exists from the completion of the previous section.

The information needed to establish these initial conditions is provided in the previous section. Additionally, detailed instruction on how to navigate to functions or tools that were explained/demonstrated in the previous section will not be repeated.

In the previous section, the cover curves method was utilized to make panels (plates) that collectively represent the ship's hull. It is at this point that the inner structural members of the ship will be added to the model carried forward from the previous section.



#### **Centerline Girder**

- Double LMB the PORT hull
  - This will cause the model to change to show the topology of the structure to include connection lines
  - Select the centerline connection line
    - RMB|Create Beam



- Double LMB the PORT hull
- Select the beam and RMB|Edit Beam...



### **Beam Orientation**



The figure above shows that the T-Bar that was just created is upside down and protruding through the hull

- In the Edit Beams Dialogue, select Rotate around local X-axis and enter 180
  - o LMB|Apply
  - Select the Offset Vector tab
    - Check the Align Section box and LMB|Apply|OK



### **Additional Bottom Longitudinal Beams**

- Double LMB the PORT hull
  - This will cause the model to change to show the topology of the structure to include connection lines
  - Select the connection lines adjacent to the centerline between Station 24 Station 3
    - RMB|Create Beam
  - Double LMB the PORT hull
  - Select the new beam and RMB|Edit Beam...
- In the Edit Beams Dialogue, select Rotate around local X-axis and enter 180
  - o LMB|Apply
  - Select the Offset Vector tab
    - Check the Align Section box and LMB|Apply|OK

- Select the Local System tab
  - Check the Relative to plate circle
  - Click inside of the 'relative to the normal of' box
  - Click anywhere on the PORT hull
  - LMB|Apply|Cancel

Perform these steps for the next adjacent connection lines until you have attained a longitudinal beam layout similar to the one below.



- Now select all thesenewly created beams beams at once
  - RMB|Properties
  - Ensure the Tbar885x200x14x35 section is chosen
  - o LMB|Apply|Cancel

#### **Transverse Beams**

Create transverse beams on the PORT and STBD station poly-curves.

- Select a station line
  - RMB|Create Beam
- Orientate the beam properly using previously utilized methods
- When done, set all of them to be Tbar575x150x12x25 sections
- Perform the same steps to create a layout similar to the one shown below.





### Longitudinal Stiffeners

The longitudinal stiffeners are created by utilizing the same methods utilized to generate the previous beams. They are made up of Tbar425x120x12x25 sections.



#### Prepare to Export to HydroD

Prior to meshing the model various images of the models structural layout are shown to give the user an opportunity to review their model for accuracy.

Figure 4 is taken from the AFT perpendicular looking forward. The bulk heads are hidden in this view.



Figure 4: Scantling as Viewed From the Aft Perpendicular Centerline Facing the Bow

The next image is similar to Fig. 4, but it is taken from the FWD perpendicular looking aft.



This image shows the location of the bulk heads. The AP is located on the left.



### Meshing

The structural model has to be meshed in order for it to be able to be imported into HydroD

• Press Alt-M on the keyboard

Export the model as a FEM file

• File|Export|FEM File...



## Chapter 4 – HydroD

Generally speaking, HydroD is the software module that will be needed whenever a structure or vessels behaviour/response in water is desired. This includes, but is not limited to, static stability, hydrostatic properties, frequency or time-domain vessel response due to wind/waves/current, and select multiple body interactive response to dynamic forces.

HydroD will be used to perform the hydrostatic and stability analysis of the ship by utilizing the two model files that were created and exported in Chapter 3. The Panel and Structural model of the ship are brought together via the Stability Wizard to in order to perform these tasks.

- Start by making a new workspace in HydroD
  - $\circ$  File  $\rightarrow$  New Workspace
  - o Or by pressing Ctrl+N



• Name your workspace and press the OK button

iew workspa		
Workspace r	iame:	
Micklitz_Hyd	lroD	
Location:		
C:\DNV\Wo	rkspaces\HydroD\Micklitz_HydroD	
🔽 Set Data	pase Units	
Length	m 💌	
Force	N	
Temperatur	e delC 💌	
	OK	Cancel

#### **Stability Wizard**

• Select Tools| Stability Wizard



• Set up the settings tab as shown below and click OK

👪 Stability Wizard Settings		
📚 Wizard: StabilityWiza	rd1	
Panel Model C Morison M     Information Settings	1odel C Composite Model C Dual Model	
<ul> <li>Element model</li> <li>Section model</li> <li>Compartments</li> <li>Openings</li> </ul>	ନ ୧ ୧ ୧	
Heeling moment curve	&s	
<ul> <li>IMO general code check</li> <li>ABS MODU code check</li> <li>IBC damage code check</li> <li>IGC damage code check</li> <li>IMO MODU code check</li> <li>IMO MODU code check</li> <li>MARPOL code check</li> <li>NMD code check</li> <li>User defined code check</li> <li>Allowable VCG analysis</li> </ul>	85 85 85 85 85 85 85 85 85	
		OK Cancel

• Click First step on the Stability Wizard

👗 Stabilit	Wizard1			X
🃚 <	First ste	эр		<b>  &gt;&gt;</b>
Create	e location Cance			ancel

• Click Apply then Cancel

Create/Edit Loca	n1 •			X
Air Water			IK = 4-1	
Density:	1.226 Kg/m 3		[Kg/m	3] -
		OK	Cancel	Apply

• Click Next step on the Stability Wizard

👪 StabilityWizard1 📃 💻	x
🗞 <<   Next step (2 of 14)	_ ⊪>>
Create hydro model	Cancel

• Change FP position to 166.95 and select OK.

Note that 166.95 m is the length between perpendiculars for the specific ship model used in this guide.

🎎 Define Hydro Model				
📚 Hydro model:	HydroModel	1		
C Fixed 💿 Floa	ating		<mark>8</mark> 3	
Column stabilized	d unit		<mark>8</mark> ?	
Baseline z-position:	0 m	[m]	<mark>9</mark> ?	
AP x-position:	0 m	[m]	<del>8</del> ?	
FP x-position:	166.95	[m]	<mark>8</mark> ?	
	OK		ancel	

• Click Next step on the Stability Wizard

Ĵå St	tability	/Wizard1 📃 🗖		x
۲	<<	Next step (3 of 14)		<b>  &gt;&gt;</b>
	Create	e panel model	C	ancel

• Load the Panel model that was created in Chapter 3.1 and click OK

🛓 Define Panel Model			
Panel model: PanelModel1 File Type: T*.FEM file			
Name: C:/Users/Student.PMEC142-05/Desktop 9?			
Symmetry XZ-plane T YZ-plane			
Translation Translate model <b>9</b> ? Vector3d(0 m,0 m,0 m)			
OK Cancel			

- Click Next step on the Stability Wizard
- Click Apply then Cancel

👪 Define Load Cross Section
📚 🖸 New 🥂 Edit existing
LoadCrossSection1
Point: 🚱
Point(0 m,0 m,0 m)
Side: 🔗
• Positive
Section plane: 🔗
YZ-plane ▼
Input shear center Z-coordinate 💡?
[m]
OK Cancel Apply

- Click Next step on the Stability Wizard
- Load the Structural Model created in chapter 3.2 and click OK

Define Structural Model	
Model name:       StructureModel1         File	
Symmetry No symmetry is currently permitted on the structural model.	
Translation Translate model 💡?	
Vector3d(0 m,0 m,0 m)	
OK Cancel	

- Click Next step on the Stability Wizard
- Change the Z-waterline value to 9.403 and click OK

Note that 9.403 m is the draft that was chosen for analysis of the specific model considered here.

👪 Define Load	ding Condition	<b></b>	
🔖 Loading c	ondition: LoadingConditi	ion1	
Compute fro	m mass 🛛 🔗 ?		
C-waterline:	<b>양</b> ? 9.403 m	[m]	
Trim:	<mark>ଡ</mark> ଼ 0 deg	[deg]	
Heel:	ତ? Odeg	[deg]	
O Draft AP:	<b>9</b> :403 m	[m]	
Draft FP:	<b>9</b> :403 m	[m]	
	<b>⊗</b> ? AP=0m		
	FP = 166.95 m Baseline = 0 m		
		Grand	
		Lancel	

- Click Next step on the Stability Wizard
- Uncheck 'Add mass of compartment content'
  - Click Homogenous Density Panel Model
    - Click OK
  - Click Fill from buoyancy
    - Click OK
    - Click OK again
  - Click OK

Define Mass Model	×
Mass model: MassModel1	
Add mass of compartment content	
Update stiffness matrix with free surface effects 🛛 😵	
□ Include dynamics of internal fluid ♀?	
C From File 💿 User Specified C Matrix C Morison Model	
Coordinate system: 8?	
C O G Centered Coordinate System	
Automatic computation:	
Fill from buoyancy         9?         Buoyancy volume:         26812.04181 m^3	
Homogeneous Density Panel Model 😵 Center of bouyancy: 84.85651176 m, 0.001051634879 m, 5.012456875 m	
Mass: 8?	
Total mass: 27482342.85 K [Kg]	
Center of gravity:	
Х: <mark>84.85651176 г</mark> [m] Y: <mark>0.0010516348;</mark> [m] Z: <mark>8.012456875 г</mark> [m]	
Radius of gyration: 😵	
RX: 6.990737577 m [m] RY: 34.38548374 m [m] RZ: 34.94811809 m [m]	
Specific product of intertia: 8?	
RXY: -0.2458996476 [m] RXZ: -4.797009116 r [m] RYZ: -0.0252447022 [m]	
OK Car	ncel

Note that the 'Add mass of compartment content was deselected, because the ship is analysed absent of contents such as crude oil or iron ore for example. Further information regarding 'Compartment Contents' is located in the HydroD\_User\_Manual, 5.3.16.5 Compartment Contents.

- Click Next step on the Stability Wizard
- Select Loading Condition1 and Location1
  - Click Apply then cancel

👪 Define Stability Analysis	×
StabilityAnalysis1	Mallow edit
Loading condition:	LoadingCondition1
Location:	Location1
Auto detect rotation axis	83
Auto detect damage rotation axis	83
Rotation axis: 💡	Vector3d(1 m,0 m,0 m))
☐ Iterate on trim ♀?	
Curve angle range	
Start of angle interval:	-180 deg [deg]
End of angle interval:	180 deg [deg]
Angle step:	1 deg [deg]
Cut off:	NoCutOff
OK	Cancel Apply

- Click Next step on the Stability Wizard
- Input 10m, 2, and 1 (m/s) as shown
  - Click Apply then Cancel

🕍 Create/Edit Wind Profile							
Wind profile IMO	MODU Wind profile						
• New C Edit	existing WindProfile1			Allow edit			
Vo Z	Reference height: Wind profile exponent: Average wind velocity	10 2 1	Dim. [m]				
-		Can	cel	Apply			

- Click Next step on the Stability Wizard
- Click Apply then Cancel



- Click Next step on the Stability Wizard
- Click Apply then Cancel

👗 Defi	ne Drag Block Coefficient Curve	×
ء 📚	New C Edit existing	🔽 Allow edit
D	ragBlockCoefficientCurve1	
<mark>8</mark> ? (	Clear	💥 🌇 👪
	Block Co	oefficient Curve_
1		
2	0.9	
3	1 <u> </u>	
4		
•	nal Block Coeffic	ient 0.6 0.8 1
	ОК	Cancel Apply

- Click Next step on the Stability Wizard
- Populate the dialogue box as shown
  - Click OK

👪 Define Heeling Moment Cur	ve		x
🗞 Name: WindHeelingMomer	RI		
● Empiric Flow Grid ● User D	efined		
Wind profile:	WindProfile1	-	
Drag coefficient curve:	DragCoefficientCurve1	-	
Drag block coefficient curve:	DragBlockCoefficientCurve1	-	
Search grid resolution:	100		
Angle step:	5 deg		[deg]
	F	OK C	
			ancel

- Click Next step on the Stability Wizard
- Click Start

Ĵå	Activi	ity monitor					-				- 0	×
3	•	Show scripting commands						Start	Abo	rt	Clo	se
	Τŗ	ying to abort							Threads:	83	1	•
	Activiț	ý	Duration	Status Running	Status Completed	Computer	Progress	Ongoing	g work			
	🗹 🛷	1 - StabilityAnalysis1	Os	Not Started								
	$\checkmark$	1.1 - Stability	0s	Not Started								
	$\checkmark$	1.2 - Crossections	0s	Not Started								
	$\checkmark$	1.3 - Heeling moment	Os	Not Started								

• Once the file is done running click Close

Ĵ.	Activit	ty monitor					100				- 0	×
	_	Show scripting commands						Start	Abo	rt	Cle	ose
									Threads:	83	1	•
	Activity	ı	Duration	Status Running	Status Completed	Computer	Progress	Ongoing	g work			
	🖸 🛷 1	1 - StabilityAnalysis1	19s	Finished	Success		100 %					
		1.1 - Stability	1s	Finished	Success		100 %					
		1.2 - Crossections	Os	Finished	Success		100 %					
		1.3 - Heeling moment	18s	Finished	Success		100 %					

• Click Last step on the Stability Wizard



The Stability Wizard is now complete. Further hydrostatic information can be obtained by clicking on tabs of the Stability Report. Some of the basic hydrostatic data that was generated by the Stability Wizard is provided in Appendix 4.

#### Saving the Stability Analysis

Saving the entire stability report is done in a manner that is notably different than normal methods utilized in DNV Sesam. The report will be saved as an HTML file that will be accessible later with an internet browser.

- RMB HydroStaticAnalysis in the model browser
- Select Save Report from Branch



- Choose a file location and name for the file
- Select HTML as the report type
- Click Save then Close

🞎 Save Report		x
<ul> <li>File name: Stability_F</li> <li>Save</li> <li>All</li> <li>Selected items in 3D/browser</li> <li>Report type:</li> <li>HTML</li> </ul>	Results	View
	Save	Close

## Chapter 5 – Sestra

The Superelement Structural Analysis Program (Sestra) functions differently than GeniE and HydroD in that it does not have a its own standalone interface. Sestra is interfaced with various Sesam modules and performs calculations in the background. Real-time executions of these calculations are transparent to the user, and are only visible once Sestra has finished its analysis. At which point they can be viewed in .txt format in Notepad via the Sestra.LIS file.

Sestra is capable of Static, Free vibration, and Dynamic structural analysis of the hull, beam, and solid components that form the structural model of a vessel. Static and free-vibration analyses of the ship in Sesam GeniE are demonstrated below.

This chapter utilizes GeniE, and begins were chapter 3.2 left off. If the GeniE program was exited, simply reopen the GeniE workspace that was used to create the files that were imported into HydroD. Once this workspace has been opened, begin performing the steps below.

### **Creating a Load Analysis**

- LMB|Activities
- Select "New Analysis"



• Select OK

Create Linear Static Analysis           Name:         Analysis1           Image:         Automatically import global loadca           Available activities         Image: Comparison of the second se	ases
Meshing     Wave Load Activity     Wave Load Activity     Linear Structural Analysis     Tension/Compression Analysis     Pile Soil Analysis     Load Results	<ul> <li>● Static</li> <li>○ Eigenvalue</li> </ul>
ОК	Cancel

• Now Select Loads | New Loadcase



Click OK



- Double LMB
- Then RMB LC2 and select properties
- Populate the dialogue box as shown
- Click
- Apply|Cancel

Load Case Properties: LC2		23	and the second s			Σ
Load Case Properties: LC2     General Equipment Loads Ro     Environment     Acceleration field: Vector3dd     Structural Analysis Load and Ma     Delete Explicit Loads Ge     @ Represent Equipment as loa     @ Represent Equipment as loa     @ Include structure self-weigh     Sum over Equipments     Mass (Kol):	Itation Field Design Condition (0 m/s^2,0 m/s^2,-9.80665 m/s^2) ass management anerate Applied Loads ads adcase-independent mass:	22	drake     Analysis     Activities	Name	Description ResultCase ResultCase ResultCase Meshing (Always Regenerate) Linear Structural Analysis, Static Load Results Reference to Dummy Hydro Pressure LoadCas LoadCase	e 1
COG [m]: Applied load [N]:	(0, 0, 0) Fx=0, Fy=0, Fz=0 Fx=0, Fy=0, Fz=0					
Explicit conceptual load [N]: Explicit conceptual load [N]: Total applied load [N]: FEM Loadcase number:	FX=0, FY=0, FZ=0       No loads       Fx=0, Fy=0, FZ=0       2       C       Display in Input Units       C       Display in Database Units					
	OK Cancel A	Apply				

The dummy hydro load 'LC1' and the structure self-weight 'LC2' need to be combined so that both can be analyzed in one analysis.

• Select Loads | Load Combination

G C:\Users\Student.PMEC142-05\Desktop\Drake\Drake\drake.gni -	Genie (DirectX9)
File Edit View Guiding Geometry Structure Loads Tools	Help
Image: Structure control structure control structure control structure control structure control contrel contro control contrel control control control control control	ination

• Click OK

[	Insert Load Con	nbination			8
	Name: LC3				
	Load Case	Factor	Phas	Description	
	🗹 \varTheta LC1	1	0	Reference to	D
	Rikr LC2	1	0	LoadCase	
			OK	C	ancel



### **Establish Boundary Conditions**

Boundary conditions for the model need to be set for the ship.

• RMB Structure |Support |Support Point



• Click on the bottom of the hull centerline (longitudinal distance does not matter).



• Do this 2 more times on the bottom of the hull and centerline



- Exit the Support Curve function
- Select the 1st support point

- LMB|Properties
- Adjust the dialogue box to match the image below and click Apply|Cancel (Fix Z)

Name :       Sp4         Position :       Point(50.085 m,0 m,0 m)         (* Boundary Condition C Boundary Stiffness Matrix C Boundary Stiffness Per Length         Boundary stiffness per length         If Let x change y and z         Fixed       Free         Prescribed       Dependent Super         Spring       stiffness         x	Properties			20
Boundary stiffness per length         Image: Let x change y and z       Spring         Fixed       Free       Prescribed Dependent Super       Spring         y	Name :     Sp4       Position :     Point(50.085 m,0 m,0 m)	ffness Per Len	gth	
Image: symmetry in the symmetry of the symmetry in the symmet	Boundary stiffness per length       Image: Boundary stiffness per length	Spring stiffness 0 N/m 0 N/m 0 N/m	[N/m] - [N/m] - [N/m]	
	Image: rx change ry and rz       Fixed     Free       Prescribed     Dependent Super       ry	Spring stiffness 0 N*m 0 N*m 0 N*m	[N*m] [N*m] [N*m]	

• Adjust the 2nd support point to match the image below and click Apply/Cancel (Fix Y & Z)

ect Properties Support Local System			 
ame: Sp5			
sition : Point(74.34 m.0 m.0 m)			
Boundary Condition C Boundary Stiffness Matrix C Boundary Sti	ffness Per Len	gth	
Boundary stiffness per length			
✓ Let x change y and z	Spring		
Fixed Free Prescribed Dependent Super Spring	stiffness		
x			
y	U N/M	[IV/m]	
z	0 N/m	[N/m]	
✓ Let rx change ry and rz	Spring		
rx	0 N*m	 [N*m]	
	0 N*m	[N*m]	
	0 N*m	[N*m]	
	1	14 mg	

• Adjust the 3rd support point to match the image below and click Apply|Cancel (Fix X, Y & Z)

Object Properties Support Local System		
Name : Sp6 Position : Point(95.2 m.0 m.0 m)		
Boundary Condition     C Boundary Stiffness Matrix     C Boundary Stiffness Per length	iffness Per Length	
Image: Let x change y and z         Fixed       Free       Prescribed       Dependent       Super       Spring         x	Spring stiffness 0 N/m (1 0 N/m (1 0 N/m (1	V/m] V/m]
Image: Tree       Image: Tree       Prescribed       Dependent       Super       Spring         rx	Spring stiffness         Image: Comparison of the second ON*m         Image: Comparison of the second ON*m	V*m] V*m]
		OK Cancel Apply

The symbols should now appear similar to the image below. The order that the symbols occur along the centerline is not important.



• RMB Analysis 1 and select Activity Monitor



#### Click Start

Ĵå.	Activity Monitor	-	-	22
3	Net Generation Elements: 5405 Nodes: 5324		Abort	Cancel
	Journal activity executions			
	Activity	Duration	Status	Generate Input
[	🗹 🎭 1 - Analysis 1 - Analysis		Running	
[	🗹 🥬 🛛 1.1 - Meshing (Always Rege		Running	
[	<ul> <li>1.1.1 - Delete loads</li> </ul>	0s	Success	
[	<ul> <li>1.1.2 - Generate loads</li> </ul>	0s	Success	
	<ul> <li>1.1.3 - Delete mesh</li> </ul>	0s	Success	
	<ul> <li>1.1.4 - Generate mesh</li> </ul>		Running	
	✓ <sup>K</sup> <sub>E</sub> =R 1.2 - Linear Structural Analy			Yes
[	R 1.3 - Load Results			

- Once it is finished running, RMB step 1.2
- Select Sestra.LIS (Open file with Notepad)
- Save the file as Sestra\_Static.LIS for later viewing

Activity Monitor	×
Start Cancel	
	_
Journal activity executions	
Activity Duration Status Generate Input	
🗹 🌯 1 - Analysis 1 - Analysis 26s Warnings	
🗹 🤁 1.1 - Meshing (Always Rege 12s Success	
✓ 1.1.1 - Delete loads Os Success	
✓ 1.1.2 - Generate loads Os Success	
✓ 1.1.3 - Delete mesh Os Success	
1.1.4 - Generate mesh 11s Success	
✓ Kr = R 1.2 - Linear Structural Analy 14s Warnings Kr = 10 and 1	
R 1.3 - Load Results     Os     Success     Success     Edit activity	
sestra.inp	
sestra.lis	
sestra.mnt	

This is a text file and contains the static structural analysis of the ship. Condensed contents of this file are included as Appendix 5.

- RMB Analysis 1
- Select Edit Analysis
- Select the Eigenvalue option then click OK

👪 Modify Linear Static Analysis	23
Name: Analysis1 Automatically import global loadcas Available activities Meshing Wave Load Activity Linear Structural Analysis Tension/Compression Analysis Pile Soil Analysis Load Results	ses O Static O Eigenvalue
ОК	Cancel

- RMB Analysis 1 and select Activity Monitor
- Click Start
- Once it is finished running, RMB step 1.2
- Select Sestra.LIS
- Save the file as Sestra\_Eigen.LIS for later viewing

This is a text file and contains the Eigenvalue Frequencies of the ship. Condensed contents of this file are included as Appendix 6.



54

## **Chapter 6 – Suggestions/Common Errors**

- Organizing model elements using 'Named Sets' will allow the user to more effectively navigate a complex model
- File naming convention dictates that the file must begin with a Letter and the string cannot contain any spaces.
- It is important that the user remembers to close dialogue boxes when done. This often means clicking cancel. If a dialogue box is left open and the user proceeds to open other unrelated dialogue boxes, program crash is likely.
- Often times the software will display the 'Not Responding' caption in the header. More often than not, the program has not crashed. Simply doing nothing until the message clears is the recommended approach to this error.
- If the material/section library is located in a drive that the user doesn't have write access to, the library will not import as shown in this guide as well as the guides provided by DNV.
  - The user must Copy|Paste the desired library to a location on the computer where they possess write privileges. Then import the library from its new location.
- When working with larger models, like the ship in this guide, increasing the zoom gain can have a dramatic positive affect on the user's ability to efficiently navigate the model
- The GeniE user manual mentions that the ship's hull is generally modelled in software outside of the Sesam suite. There are various help files that explain imported outside generated hulls from: Rhino, Sacs, and general Parametric software programs

## Bibliography

- American Bureau of Shipping (ABS)—ABS Plaza, 16855 Northchase Drive, Houston, TX 77060 ABS Rules for Building and Classing Steel Vessels 2009
- SESAM User Course in Curved Structure Modelling Genie Workshop: Modelling pontoon and lower column C:\Program Files (x86)\DNVS\GeniE V6.4-08\Examples\A2\_Semisub\_Pontoon
- SESAM User Manual GENIE VOL. III MODELLING OF PLATE/SHELL STRUCTURES C:\Program Files (x86)\DNVS\GeniE V6.4-08\Help\pdf\GeniE\_UM\_Vol3

SESAM User Manual HydroD

C:\Program Files (x86)\DNVS\HydroD V4.6-03\User Documentation\pdf\HydroD\_UM

SESAM User Manual Sestra

C:\Program Files (x86)\DNVS\Sestra V8.5-01\Doc\Sestra\_UM

## Appendices Appendix 1: Table of Offsets

_	-	_	·	_	-	_	·	_		_	-	_	·	_	·	_					
F.P.	9.75	9.5	9.25	9	8.5	∞	7	6	σ	4	ω	2	1.5	1	0.75	0.5	0.25	A.P.	Station Number		
166.95	162.78	158.60	154.43	150.26	141.91	133.56	116.87	100.17	83.48	66.78	50.09	33.39	25.04	16.70	12.52	8.35	4.17	0.00	Longitude (m)	Station	
0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.0 (m)		
0.00	0.00	0.70	1.68	2.87	5.88	8.54	10.64	10.85	10.85	10.85	10.01	6.02	3.50	1.75	1.12	0.70	0.42	0.00	0.5005 (m)		
0.00	0.07	1.19	2.45	3.85	6.93	9.31	11.06	11.27	11.27	11.27	10.64	7.28	4.76	2.52	1.61	0.91	0.21	0.00	1.001 (m)		
0.00	0.49	1.89	3.43	5.04	8.12	10.08	11.27	11.27	11.27	11.27	11.13	8.61	6.16	3.64	2.38	1.26	0.14	0.00	2.002 (m)		
0.00	0.77	2.45	4.13	5.81	8.75	10.50	11.27	11.27	11.27	11.27	11.27	9.38	7.07	4.41	2.94	1.54	0.07	0.00	3.003 (m)		_
0.00	1.05	2.87	4.69	6.44	9.17	10.71	11.27	11.27	11.27	11.27	11.27	9.87	7.77	5.04	3.43	1.82	0.07	0.00	4.004 (m)		able of
0.00	1.54	3.50	5.46	7.21	9.66	10.92	11.27	11.27	11.27	11.27	11.27	10.50	8.82	6.16	4.41	2.45	0.14	0.00	6.006 (m)	Waterlines	Offset
0.00	1.96	4.06	6.02	7.77	9.94	10.99	11.27	11.27	11.27	11.27	11.27	10.85	9.52	7.28	5.60	3.71	1.40	0.00	8.008 (m)		S
0.21	2.59	4.76	6.65	8.19	10.15	10.99	11.27	11.27	11.27	11.27	11.27	11.06	10.15	8.40	7.07	5.60	4.06	2.38	10.01 (m)		
0.77	3.43	5.53	7.28	8.61	10.29	10.99	11.27	11.27	11.27	11.27	11.27	11.20	10.64	9.45	8.61	7.49	6.23	4.41	12.012 (m)		
1.75	4.55	6.51	7.98									11.27	10.99	10.29	9.73	8.82	7.49	5.74	14.014 (m)		
3.15	5.81	7.56	8.82									11.27	11.20	10.78	10.29	9.52	8.33	6.58	16.016 (m)		
3.92	6.30	7.77	7.84	8.96	10.36	11.06	11.27	11.27	11.27	11.27	11.27	11.27	11.27	10.85	10.43	9.66	8.54	6.93	Half Breadth (m)	DE	
17.01	16.66	16.38	16.03	13.51	13.44	13.44	13.37	13.37	13.37	13.37	13.37	16.38	16.45	16.66	16.73	16.80	16.87	17.01	Height (m)	Ŕ	

## Appendix 2: Ships Stations

					I	FULLS	SCA		MENS	IONS	(me	eters)						
	Station_AF		1		Station_2				Station 3				Station 4				Station 5	
0	0	8.008		4.172	0	0		8.351	0	0		12.523	0	0		16.695	0	0
0	1.19	9.009		4.172	0.42	0.5005		8.351	0.7	0.5005		12.523	1.12	0.5005		16.695	1.75	0.5005
0	2.38	10.01		4.172	0.21	1.001		8.351	0.91	1.001		12.523	1.61	1.001		16.695	2.52	1.001
0	3.5	11.011		4.172	0.14	2.002		8.351	1.26	2.002		12.523	2.38	2.002		16.695	3.64	2.002
0	4.41 5.18	13.012		4.172	0.07	3.003		8 351	1.54	3.003		12.523	3.43	4 004		16.695	5.04	3.003
0	5.74	14.014		4.172	0.14	5.005		8.351	2.1	5.005		12.523	3.92	5.005		16.695	5.6	5.005
0	6.23	15.015		4.172	0.14	6.006		8.351	2.45	6.006		12.523	4.41	6.006		16.695	6.16	6.006
0	6.58	16.016		4.172	0.42	7.007		8.351	3.01	7.007		12.523	4.97	7.007		16.695	6.72	7.007
-				4.172	1.4	8.008		8.351	3.71	8.008		12.523	5.6	8.008		16.695	7.28	8.008
				4.172	2.8	9.009		8.351	4.55	9.009		12.523	6.3	9.009		16.695	7.84	9.009
			-	4.172	4.06	11.011		8.351	5.0	11.011		12.523	7.07	11.011		16.695	8.4	11.011
-				4.172	6.23	12.012		8.351	7.49	12.012		12.523	8.61	12.012		16.695	9.45	12.012
				4.172	6.93	13.013		8.351	8.26	13.013		12.523	9.24	13.013		16.695	9.94	13.013
				4.172	7.49	14.014		8.351	8.82	14.014		12.523	9.73	14.014		16.695	10.29	14.014
				4.172	7.98	15.015		8.351	9.17	15.015		12.523	10.01	15.015		16.695	10.57	15.015
				4.172	8.33	16.016		8.351	9.52	16.016		12.523	10.29	16.016		16.695	10.78	16.016
	Station 6		-		Station 7				Station 9				Station 0				Station 10	
25.046	0	0		33 39	0	0		41 741	0	0		50.085	0	0		58 436		0
25.046	3.5	0.5005		33.39	6.02	0.5005		41.741	8.33	0.5005		50.085	10.01	0.5005		58.436	10.85	0.5005
25.046	4.76	1.001		33.39	7.28	1.001		41.741	9.31	1.001		50.085	10.64	1.001		58.436	11.2	1.001
25.046	6.16	2.002		33.39	8.61	2.002		41.741	10.22	2.002		50.085	11.13	2.002		58.436	11.27	2.002
25.046	7.07	3.003		33.39	9.38	3.003		41.741	10.71	3.003		50.085	11.27	3.003		58.436	11.27	3.003
25.046	7.77	4.004		33.39	9.87	4.004	-	41.741	10.99	4.004		50.085	11.27	4.004		58.436	11.27	4.004
25.046	8.4	5.005	-	33.39	10.22	5.005		41.741	11.13	5.005		50.085	11.27	5.005		58.436	11.27	5.005
25.046	0.82	7.007	-	33 39	10.5	7,007		41.741	11.2	7.007		50.085	11.27	7.007		58,436	11.27	7.007
25.046	9.52	8.008		33.39	10.85	8.008		41.741	11.27	8.008		50.085	11.27	8.008		58.436	11.27	8.008
25.046	9.87	9.009		33.39	10.92	9.009		41.741	11.27	9.009		50.085	11.27	9.009		58.436	11.27	9.009
25.046	10.15	10.01		33.39	11.06	10.01		41.741	11.27	10.01		50.085	11.27	10.01		58.436	11.27	10.01
25.046	10.36	11.011		33.39	11.13	11.011		41.741	11.27	11.011		50.085	11.27	11.011		58.436	11.27	11.011
25.046	10.64	12.012		33.39	11.2	12.012		41.741	11.27	12.012		50.085	11.27	12.012		58.436	11.27	12.012
25.046	10.85	13.013		33.39	11.2	13.013		41.741	11.27	13.013		50.085	11.27	13.013		58.436	11.27	13.013
25.040	11 13	15.015		33.39	11.27	15 015		41.741	0	15 015		50.085	0	15.015		58 436	0	15 015
25.046	11.15	16.016		33.39	11.27	16.016		41.741	0	16.016		50.085	0	16.016		58.436	0	16.016
-																		
	Station 11				Station 12				Station 13				Station 14				Station 15	
66.78	0	0		75.131	0	0		83.475	0	0		91.826	0	0		100.17	0	0
66.78	10.85	0.5005		75.131	10.85	0.5005		83.475	10.85	0.5005		91.826	10.85	0.5005		100.17	10.85	0.5005
66.78	11.2	1.001		75.131	11.2	1.001		83.475	11.2	1.001		91.826	11.2	1.001		100.17	11.2	1.001
66.78	11.27	3.003		75.131	11.27	3.003		83.475	11.27	3.003		91.826	11.27	3.003		100.17	11.27	3.003
66.78	11.27	4.004		75.131	11.27	4.004		83.475	11.27	4.004		91.826	11.27	4.004		100.17	11.27	4.004
66.78	11.27	5.005		75.131	11.27	5.005		83.475	11.27	5.005		91.826	11.27	5.005		100.17	11.27	5.005
66.78	11.27	6.006		75.131	11.27	6.006		83.475	11.27	6.006		91.826	11.27	6.006		100.17	11.27	6.006
66.78	11.27	7.007		75.131	11.27	7.007		83.475	11.27	7.007		91.826	11.27	7.007		100.17	11.27	7.007
66.78	11.27	8.008		75.131	11.27	8.008		83.475	11.27	8.008		91.826	11.27	8.008		100.17	11.27	8.008
66.78	11.27	9.009		75.131	11.27	9.009		83.475	11.27	9.009		91.826	11.27	9.009		100.17	11.27	9.009
66.78	11.27	11.011		75.131	11.27	11.011		83.475	11.27	11.011		91.826	11.27	11.011		100.17	11.27	11.011
66.78	11.27	12.012		75.131	11.27	12.012		83.475	11.27	12.012		91.826	11.27	12.012		100.17	11.27	12.012
66.78	11.27	13.013		75.131	11.27	13.013		83.475	11.27	13.013		91.826	11.27	13.013		100.17	11.27	13.013
					o:						-		<u></u>				<u></u>	
100 501	Station 16	0		116.965	Station 1/	0		125 216	Station 18	0		122.56	Station 19	0		141 011	Station 20	0
108.521	10.85	0 5005		116.865	10.64	0 5005		125.216	10.01	0 5005		133.50	8 54	0 5005		141.911	5.88	0 5005
108.521	11.2	1.001	1	116.865	11.06	1.001		125.216	10.57	1.001		133.56	9.31	1.001		141.911	6.93	1.001
108.521	11.27	2.002		116.865	11.27	2.002		125.216	11.06	2.002		133.56	10.08	2.002		141.911	8.12	2.002
108.521	11.27	3.003		116.865	11.27	3.003		125.216	11.2	3.003		133.56	10.5	3.003		141.911	8.75	3.003
108.521	11.27	4.004		116.865	11.27	4.004		125.216	11.27	4.004		133.56	10.71	4.004		141.911	9.17	4.004
108.521	11.27	5.005		116.865	11.27	5.005		125.216	11.27	5.005	-	133.56	10.85	5.005		141.911	9.45	5.005
108.521	11.27	7.007		116 865	11.27	7.007		125.216	11.27	7.007		133.50	10.92	7.007		141.911	9.00	7.007
108.521	11.27	8.008	1	116.865	11.27	8.008	1	125.216	11.27	8.008	1	133.56	10.92	8.008		141.911	9.94	8.008
108.521	11.27	9.009		116.865	11.27	9.009		125.216	11.27	9.009		133.56	10.99	9.009		141.911	10.08	9.009
108.521	11.27	10.01		116.865	11.27	10.01		125.216	11.27	10.01		133.56	10.99	10.01		141.911	10.15	10.01
108.521	11.27	11.011		116.865	11.27	11.011		125.216	11.27	11.011		133.56	10.99	11.011		141.911	10.22	11.011
108.521	11.27	12.012		116.865	11.27	12.012		125.216	11.27	12.012		133.56	10.99	12.012		141.911	10.29	12.012
108.521	11.27	13.013		116.865	11.27	13.013		125.216	11.27	13.013		133.56	11.06	13.013		141.911	10.36	13.013
	Station 21		1		Station 22	1	1		Station 23	l			Station 24	l			Station FP	
150.255	0	0		154.427	0	0		158.606	0	0		162.778	0	0		166.95	0	9.009
150.255	2.87	0.5005		154.427	1.68	0.5005		158.606	0.7	0.5005		162.778	0	0.5005		166.95	0.21	10.01
150.255	3.85	1.001		154.427	2.45	1.001		158.606	1.19	1.001		162.778	0.07	1.001		166.95	0.42	11.011
150.255	5.04	2.002	-	154.427	3.43	2.002		158.606	1.89	2.002		162.778	0.49	2.002		166.95	0.77	12.012
150.255	5.81	3.003	-	154.427	4.13	3.003	-	158.606	2.45	3.003	-	162.778	0.77	3.003		166.95	1.19	13.013
150.255	6.86	4.004 5.005	-	154.427	4.69	5.005	-	158.606	2.8/	5.005		162.778	1.05	5.005		166.95	2.75	14.014
150.255	7,21	6,006	-	154.427	5.46	6,006		158.606	3.5	6,006	-	162.778	1.20	6,006	-	166.95	3,15	16,015
150.255	7.49	7.007	1	154.427	5.74	7.007	1	158.606	3.78	7.007		162.778	1.75	7.007			0.10	
150.255	7.77	8.008		154.427	6.02	8.008		158.606	4.06	8.008		162.778	1.96	8.008				
150.255	7.98	9.009		154.427	6.37	9.009		158.606	4.41	9.009		162.778	2.24	9.009			BOW Extra	
150.255	8.19	10.01		154.427	6.65	10.01		158.606	4.76	10.01		162.778	2.59	10.01		167.4	0	10.01
150.255	8.4	11.011	-	154.427	6.93	11.011		158.606	5.11	11.011		162.778	2.94	11.011		167.65	0	11.011
150.255	8.61	12.012	-	154.427	7.28	12.012		158.606	5.53	12.012		162.778	3.43	12.012		167.9	0	12.012
150.255	0.02	14.014	-	154.427	7.98	14.014		158.606	6.51	14.014		162.778	4.55	14.014		168.48	0	14.014
150.255	0	15.015	1	154.427	8.4	15.015	1	158.606	7.07	15.015	1	162.778	5.11	15.015		168.9215	0	15.015
150 255	0	16.016	1	154 427	0 07	16.016	1	159 606	7 56	16.016	1	162 779	E 01	16.016	1	160 6922	0	16.016

### Appendix 3: Hull Lines





### Appendix 4: Stability Report

	Result Variable	Value (X)	Y	Z
1	Metacentric Height GM (wet)	1.100871913 m		
2	Metacentric Height GM (dry)	1.100871913 m		
3	Free surface correction FSC	0 m		
4	Total mass (wet) (including compartment contents)	1.797693135e+308 Kg		
5	Total mass (without compartment contents)	1.797693135e+308 Kg		
6	Center of gravity (wet)	1.797693135e+308 m	1.797693135e+308 m	1.797693135e+308 m
7	Center of gravity (without compartment contents)	1.797693135e+308 m	1.797693135e+308 m	1.797693135e+308 m
8	Center of gravity (with compartment contents in metacenter)	1.797693135e+308 m	1.797693135e+308 m	1.797693135e+308 m
9	Buoyancy volume	+In finity		
10	Buoyancy mass	+In finity		
11	Center of buoyancy	1.797693135e+308 m	1.797693135e+308 m	1.797693135e+308 m
12	Center of flotation	1.797693135e+308 m	1.797693135e+308 m	1.797693135e+308 m
13	Trim moment	1.797693135e+308 N*m		
14	Panel model block coefficient	1.797693135e+308		
15	Projected XZ area above waterline	305.5958295 m^2		
16	Center projected XZ area above waterline	69.16595658 m		3.82323368 m
17	Projected XZ area below waterline	572.16347 m^2		
18	Center projected XZ area below waterline	80.19379193 m		-2.953307962 m
19	Deck immersion heel angle negative side	-89 deg		
20	Deck immersion heel angle positive side	90 dea		



Righting Moment ----

Heel Angle [deg]

### Appendix 5: Condensed Static Sestra Results

Input from CMAS Command :

ANTYP = 1 Static Analysis MSUM > 0 Calculation of Sum of Masses and Centroid

The singularity constant for membrane and shell elements CSING = 1.0000E-08

Lowest accepted condition number in reduction EPSSOL= 1.1102E-14

Input from RSEL Command :

Data types selected for storing on Results File :

- Input Interface File Records,

- displacements, sequence:

all nodes for the first resultcase, all nodes for the second resultcase, etc.

 forces and moments for beam, spring and layered shell elements, sequence: all elements for the first resultcase, all elements for the second resultcase, etc.

 stresses (not for beam or spring elements), sequence: all elements for the first resultcase, all elements for the second resultcase, etc.

PAGE: 3

#### \*\*\* SUMMARY OF DATA FROM INPUT AND LOAD INTERFACE FILES \*\*\* FOR SUPERELEMENT TYPE 1 ON LEVEL 1

The superelement has

13209 subelements
11311 nodes
6 specified (fixed) degrees of freedom
67860 internal (free) degrees of freedom
67866 degrees of freedom

2 loadcases

Side information for hydropressure is given

The following kinds of loads are given: gravitational load

The following basic elements are given: 1797 2 node beam elements BEAS 11128 4 node flat shell elements FQUS 284 3 node flat shell elements FTRS

Eccentricities are given

PAGE: 6

DATAGENERATION - SUPERELEMENT TYPE 1 SUB PAGE: 6

\*\*\* SUM OF LOADS AND MOMENTS FOR SUPERELEMENT TYPE 1 ON LEVEL 1 \*\*\*

X-LOAD = SUM OF GIVEN LOADS IN GLOBAL X-DIRECTION Y-LOAD = SUM OF GIVEN LOADS IN GLOBAL Y-DIRECTION Z-LOAD = SUM OF GIVEN LOADS IN GLOBAL Z-DIRECTION X-MOM = SUM OF LOCAL MOMENTS ABOUT GLOBAL X-AXIS Y-MOM = SUM OF LOCAL MOMENTS ABOUT GLOBAL Y-AXIS Z-MOM = SUM OF LOCAL MOMENTS ABOUT GLOBAL Z-AXIS X-RMOM = SUM OF MOMENTS ABOUT GLOBAL X-AXIS FROM GIVEN LOADS AND MOMENTS Y-RMOM = SUM OF MOMENTS ABOUT GLOBAL Y-AXIS FROM GIVEN LOADS AND MOMENTS Z-RMOM = SUM OF MOMENTS ABOUT GLOBAL Y-AXIS FROM GIVEN LOADS AND MOMENTS Z-RMOM = SUM OF MOMENTS ABOUT GLOBAL Z-AXIS FROM GIVEN LOADS AND MOMENTS

LOADCASE X-LOAD Y-LOAD Z-LOAD X-MOM Y-MOM Z-MOM X-RMOM Y-RMOM Z-RMOM 1 0.0000E+00 0.0000E+00 0.0000E+00 0.0000E+00 0.0000E+00 0.0000E+00 0.0000E+00 0.0000E+00 0.0000E+00 2 5.6245E-13 -7.2831E-14 -1.4419E+07 -6.7387E+01 -3.5514E+02 -5.3707E-14 1.1082E+02 1.1732E+09 -4.8588E-12

DATAGENERATION - SUPERELEMENT TYPE 1 SUB PAGE: 7

### Appendix 5: Condensed Static Sestra Results (continued)

\*\*\* SUM OF MASSES AND CENTROID FOR SUPERELEMENT TYPE 1 ON LEVEL 1 \*\*\*

#### MASS MATRIX IN GLOBAL COORDINATE SYSTEM (OF THE SUPERELEMENT):

1.47032E+06 0.00000 0.00000E+00 1.47032	)E+00 0.000 2E+06 0.000	000E+00 000E+00	0.00000E+00	7.92487E+06	4.56943	E+01 E+08			
0.00000E+00 0.00000	)E+00 1.470	032E+06	-1.81717E+01	-1.19631E+08	0.00000	)E+00			
0.00000E+00 -7.9249	6E+06 -1.81	1717E+01	1.62273E+08	-4.86859E+02	-6.2039	7E+08			
7.92487E+06 0.0000	E+00 -1.19	000E+08	-4.86859E+02	2 1.27987E+10	-1.89948	8E+02			
4.50943E+01 1.1963	1E+08 0.000	000E+00	-6.20397E+08	-1.89948E+02	1.28108	SE+10			
COORDINATES OF C	ENTROID:								
8.1364E+01 -2.1718E	3-05 5.3899	9E+00							
MASS MATRIX AT C	ENTROID:								
1.47032E+06 0.00000	)E+00 0.000	000E+00	0.00000E+00	-4.18065E+01	1.37613	E+01			
0.00000E+00 1.47032	2E+06 0.000	000E+00	-4.18065E+01	0.00000E+00	1.42121	E+01			
0.00000E+00 0.00000	E+00  1.470	032E+06	1.37613E+01	1.42121E+01	0.00000	E+00			
$-4.18065E\pm01 - 0.00000E$	0E+01  1.37	013E+01	1.1955/E+08	-1.96539E+03	2.44104	+E+07 7E±02			
1.37613E+01 1.4212	IE+01 0.000	000E+00	2.44104E+07	-4.36237E+02	3.07705	5E+09			
DATE: 31-JUL-2015 TI	ME: 12:12:23	3 ******	********* SE	STRA ******	******			PAGE:	8
DA	ATAGENERA	ATION - S	UPERELEME	NT TYPE 1					
There are 6 3-nod	ed shell or mo	embrane el	lements with b	ad element shape	e.				
The ratio of the larges	st edge to the	smallest ne	eight is 4.0 or i	arger.					
- COMPUTATION	I IS CONTIN	IUED.							
*** Estimated size of st	tiffness matrix	x for super	element 1:	12140874 vari	ables				
*** Estimate of total siz	e of stiffness	matrices for	or new superel	ements: 1214	40874 vari	ables			
DATE: 31-JUL-2015 TI	ME: 12:12:23	3 ******	********* SE	STRA ******	******			PAGE:	9
RE	DUCTION N	MODULE	- SUPERELEN	MENT TYPE 1					
- 5	STIFFNESS F	FACTORIZ	ZATION PERI	FORMED BY M	ULTIFRO	ONT EQUA	TION SOLVE	R -	
- I	LOAD SUBS	TITUTION	N PERFORME	D BY MULTIF	RONT EQ	QUATION S	SOLVER -		
DATE: 31-JUL-2015 TI	ME: 12:12:24	4 ******	********* SE	STRA ******	******			PAGE:	10
	STATIC AN	NALYSIS (	OF STRUCTU	RE					
Results file name: 20 PAGE: 11	150731_1212	201_R1.SIN	N D	ATE: 31-JUL-20	)15 TIME	: 12:12:25	****	**** SES	TRA *************
RE TH	TRACKING	G MODULE URE	E - SUPERELI	EMENT TYPE	1 SUB	PAGE: 2	!		
REACTION FORCES I NODES MARKED WI	N NODES W ΓΗ AN ASTE ******	VITH SPEC ERISK (*) ********	CIFIED (FIXE) TO THE RIGH *******	D) DEGREES O IT HAVE A LO **********	F FREED CAL COC	OM. DRDINATE ******	SYSTEM.		
LOADCASE (INDEX)	NODE NO.	Х	Y Z	RX	RY	RZ			

 2659
 0.00000E+00

 4782
 0.00000E+00
 0.00000E+00

 6487
 0.00000E+00
 0.00000E+00
 0.00000E+00

1

2 2659 5.35821E+06 4782 -8.70852E+00 -2.02488E+06 6487 3.42754E-04 8.70869E+00 1.10856E+07	
DATE: 31-JUL-2015 TIME: 12:12:25 ***********************************	
RETRACKING MODULE - SUPERELEMENT TYPE1THE STRUCTURESUB PAGE:3	
SUM OF REACTION FORCES FROM SPECIFIED DEGREES OF FREEDOM. THE FORCES AND MOMENTS ARE REFFERED TO THE COORDINATE SYSTEM OF THE ACTUAL SUPERELEMENT. Appendix 5: Condensed Static Sestra Results (continued)	
**********************	
LOADCASE (INDEX) X Y Z RX RY RZ	
1 0.0000E+00 0.0000E+00 0.0000E+00 0.0000E+00 0.0000E+00 0.0000E+00 2 3.4275E-04 1.7241E-04 1.4419E+07 -4.3453E-08 -1.1732E+09 1.8168E+02	
SUPERELEMENT TYPE: 1 ACTUAL ELEMENT: 1 HAS BEEN STORED ON RESULT FILE SUB PAGE: 1	
SUM OF GLOBAL LOADS AND MOMENTS ************************************	
LOADCASE (INDEX) X Y Z RX RY RZ	
1 0.0000E+00 0.0000E+00 0.0000E+00 0.0000E+00 0.0000E+00 0.0000E+00 2 5.6245E-13 -7.2831E-14 -1.4419E+07 1.1082E+02 1.1732E+09 -4.8588E-12	
DATE: 31-JUL-2015 TIME: 12:12:26 ***********************************	
RETRACKING MODULE - GLOBAL DATA SUB PAGE: 2	
SUM OF REACTION FORCES AND MOMENTS	
LOADCASE (INDEX) X X 7 RX RX R7	
1 0.0000E+00 0.0000E+00 0.0000E+00 0.0000E+00 0.0000E+00 0.0000E+00 2 34275E-04 1.7241E-04 1.4419E+07 -4.3453E-08 -1.1732E+09 1.8168E+02	
DATE: 31-JUL-2015 TIME: 12:12:26 ***********************************	
RETRACKING MODULE - GLOBAL DATA SUB PAGE: 3	
DIFFERENCES BETWEEN SUMMED LOADS AND REACTION FORCES ************************************	PONENTS
LOADCASE (INDEX) X Y Z RX RY RZ	
2 3.4275E-04 1.7241E-04 3.7532E-05 1.1082E+02 -6.4972E+02 1.8168E+02	
TOTAL TIME CONSUMED IN SESTRA CPU TIME: 8.38 CLOCK TIME: 5.16 CHANNEL TIME: 0.00	

#### Appendix 6: Condensed Eigenvalue Sestra Results

Type of Analysis :

Eigenvalue Solution by Lanczos Method Retracking Input from CMAS Command : ANTYP = 2 Dynamic Analysis MSUM > 0 Calculation of Sum of Masses and Centroid

The singularity constant for membrane and shell elements CSING = 1.0000E-08

Lowest accepted condition number in reduction EPSSOL= 1.1102E-14

Input from EIGL Command :

Specification of eigenvalues to be calculated: ENR = 10 eigenvalues are demanded.

MAXO50Maximum number of iterations.NBLO2Block size.NFIG5No. of digits of accuracy.IU=0The stiffness matrix is triangularised.PRIN0Print of eigenvalues.

Input from RSEL Command :

INTERPRETATION OF ANALYSIS CONTROL DATA FOR SUPERELEMENT TYPE 1 ON LEVEL 1

Input from DYMA Command :

IMAS = 1 Consistent mass matrices from the subelements are demanded.

\*\*\* SUMMARY OF DATA FROM INPUT AND LOAD INTERFACE FILES \*\*\* FOR SUPERELEMENT TYPE 1 ON LEVEL 1 The superelement has 13209 subelements 11311 nodes 6 specified (fixed) degrees of freedom 67860 internal (free) degrees of freedom totally 67866 degrees of freedom 2 loadcases Side information for hydropressure is given The following kinds of loads are given: gravitational load The following basic elements are given: 1797 2 node beam elements BEAS 11128 4 node flat shell elements FQUS 284 3 node flat shell elements FTRS

Eccentricities are given

\*\*\* SUM OF LOADS AND MOMENTS FOR SUPERELEMENT TYPE 1 ON LEVEL 1 \*\*\*

X-LOAD = SUM OF GIVEN LOADS IN GLOBAL X-DIRECTION Y-LOAD = SUM OF GIVEN LOADS IN GLOBAL Y-DIRECTION Z-LOAD = SUM OF GIVEN LOADS IN GLOBAL Z-DIRECTION X-MOM = SUM OF LOCAL MOMENTS ABOUT GLOBAL X-AXIS Y-MOM = SUM OF LOCAL MOMENTS ABOUT GLOBAL X-AXIS Z-MOM = SUM OF LOCAL MOMENTS ABOUT GLOBAL Z-AXIS X-RMOM = SUM OF MOMENTS ABOUT GLOBAL X-AXIS FROM GIVEN LOADS AND MOMENTS Y-RMOM = SUM OF MOMENTS ABOUT GLOBAL X-AXIS FROM GIVEN LOADS AND MOMENTS Z-RMOM = SUM OF MOMENTS ABOUT GLOBAL Z-AXIS FROM GIVEN LOADS AND MOMENTS Z-RMOM = SUM OF MOMENTS ABOUT GLOBAL Z-AXIS FROM GIVEN LOADS AND MOMENTS

LOADCASE X-LOAD Y-LOAD Z-LOAD X-MOM Y-MOM Z-MOM X-RMOM Y-RMOM Z-RMOM 1 0.0000E+00 0.0000E+00 0.0000E+00 0.0000E+00 0.0000E+00 0.0000E+00 0.0000E+00 0.0000E+00 0.0000E+00 2 -9.1138E-12 4.8423E-12 -1.4419E+07 -8.3358E+01 -3.5649E+02 2.3386E+01 2.0254E+02 1.1732E+09 7.0997E+01

PAGE: 7

\*\*\* SUM OF MASSES AND CENTROID FOR SUPERELEMENT TYPE 1 ON LEVEL 1 \*\*\*

#### Appendix 6: Condensed Static Sestra Results (continued)

MASS MATRIX IN GLOBAL COORDINATE SYSTEM (OF THE SUPERELEMENT):		
1.47032E+06       1.25933E-12       9.48871E-13       1.39642E+01       7.98233E+06       9.95754E+00         1.25933E-12       1.47032E+06       -7.03162E-14       -7.98239E+06       -7.45105E+00       1.19631E+08         9.48871E-13       -7.03162E-14       1.47032E+06       -2.06529E+01       -1.19631E+08       -7.23972E+00         1.39642E+01       -7.98239E+06       -2.06529E+01       1.61611E+08       -5.49188E+02       -6.25039E+08         7.98233E+06       -7.45105E+00       -1.19631E+08       -5.49188E+02       1.27983E+10       1.38814E+03         9.95754E+00       1.19631E+08       -7.23972E+00       -6.25039E+08       1.38814E+03       1.28098E+10		
COORDINATES OF CENTROID:		
8.1364E+01 -1.0409E-05 5.4290E+00		
MASS MATRIX AT CENTROID:		
1.47032E+06       1.25933E-12       9.48871E-13       1.39642E+01       -3.00585E+01       -5.34768E+00         1.25933E-12       1.47032E+06       -7.03162E-14       -3.00585E+01       -7.45105E+00       2.46832E+01         9.48871E-13       -7.03162E-14       1.47032E+06       -5.34768E+00       2.46832E+01       -7.23972E+00         1.39642E+01       -3.00585E+01       -7.03162E-14       1.47032E+06       -5.34768E+00       2.46832E+01       -7.23972E+00         1.39642E+01       -3.00585E+01       -5.34768E+00       1.18274E+08       -2.34586E+03       2.44419E+07         -3.00585E+01       -7.45105E+00       2.46832E+01       -2.34586E+03       3.02124E+09       1.35128E+03         -5.34768E+00       2.46832E+01       -7.23972E+00       2.44419E+07       1.35128E+03       3.07613E+09		
DATE: 31-JUL-2015 TIME: 12:53:13 ***********************************	PAGE:	8
DATAGENERATION - SUPERELEMENT TYPE 1		
*** Estimated size of stiffness matrix for superelement 1: 107895744 variables		
*** Estimate of total size of stiffness matrices for new superelements: 107895744 variables		
DYNAMIC ANALYSIS OF STRUCTURE		
- EIGENVALUEPROBLEM SOLVED BY LANCZOS METHOD DATE: 31-JUL-2015 TIME: 12:54:32 ************************************	PAGE:	11
DYNAMIC ANALYSIS OF STRUCTURE		
* EIGENVALUES * * *		
* FREQUENCIES *		
ALL FIGENVALUES BEING CAL-		
CULATED ARE PRINTED TOGE-		
THER WITH THE CORRESPONDING		
FREQ = SQRT(EIGENV)/(2.*PI)		
PERI = 1./FREQ		
TTTTTT		
I NO. I EIGENVALUE UNIT: (SEC)-2 I FREQUENCY UNIT: HERTZ I PERIOD UNIT: SEC I I I I I I I I		
++ +		
I I 0.7362895E-05 I 0.000 I 2315.55742 I		
I 2 I 0.2850339E+01 I 0.269 I 3.72162 I		
I 3 I 0.2859708E+01 I 0.269 I 3.71552 I		
4   0.3059905E+01   0.278   3.59191     5   0.2460260E+01   0.206   2.27774		
I 6 I 0.3503785E+01 I 0.250 I 3.37774 I		
I 7 I 0.5178583E+01 I 0.362 I 2.76105 I		
I 8 I 0.7079450E+01 I 0.423 I 2.36146 I		
I 9 I 0.1366105E+02 I 0.588 I 1.69996 I		
I 10 I 0.1415003E+02 I 0.599 I 1.67033 I		
*		
DATE: 31-JUL-2015 TIME: 12:54:32 ************************************	PAGE:	12
DYNAMIC ANALYSIS OF STRUCTURE		
SUB PAGE: 3		

Results file name: 20150731\_125244\_R1.SIN

SUPERELEMENT TYPE: 1 ACTUAL ELEMENT: 1 HAS BEEN STORED ON RESULT FILE TOTAL TIME CONSUMED IN SESTRA CPU TIME: 119.72 CLOCK TIME: 89.78 CHANNEL TIME: 0.00