FLOWStress

Example Manual

25 July 2024

Developed by PARK, Sungjun
Supervised by CHOUNG, Joonmo

Ship and Offshore Structure Engineering Lab (SOSEL)
Department of Naval Architecture and Ocean Engineering, Inha University,
Incheon, Republic of Korea

1 FOWT Model

The example FOWT was prepared based on the specifications of the UMaine VolturnUS-S[1]. The number of shell and beam elements is 46,188 without coupling elements. The internal stiffeners and web frame were determined in their dimensions without the strength consideration. It is not recommended to use this model in specific projects.

[1] https://www.nrel.gov/docs/fy20osti/76773.pdf

2 Files for FOWT model

	Path	File	Remark
Input	.../Example	UMaine VolturnUS-S Technical report.pdf	
		FlowStress_40elems.job	FLOWStress job file with stress recovery ELSET[2] of 40 elements
		FlowStress_5000elems.job	FLOWStress job file with stress recovery ELSET of 5,000 elements
	…/Example/Abaqus	FEA_model.inp	Reference Abaqus INP file without HISTORY part for structural analysis
	…/Example/OpenFAST	OpenFAST execution files (EXE, DLL…), module input files (DAT), result files (OUT), etc.	Module input files include HydroDyn DAT, SurvoDyn DAT, ElastoDyn DAT, MoorDyn DAT, Inflow DAT, AeroDyn DAT, FAST FST, Airfoil DAT, and Wind BTS
	…/Example/Wamit	VolturnUS-S.cfg	Wamit configuration file
		VolturnUS-S.frc	Wamit force control file
		VolturnUS-S.gdf	Wamit geometric data file
		VolturnUS-S.inp	Abaqus INP file to convert Wamit GDP and POT files
		VolturnUS-S.pot	Wamit potential control file
		VolturnUS-S.5p	Wamit result file containing pressure RAOs
Output	.../Example/Results/40elems/1_StaticLoad	FEA_model.inp	Abaqus INP file for static load analysis
		Static_load.inp	Abaqus INP file with static loads only
		FEA_model.dat	Abaqus result file
		ABAQUS_RUN.bat	Batch file to run Abaqus
	.../Example/Results/40elems/2_UnitLoad	FEA_model.inp	Abaqus INP file for unit nodal load analysis
		Unit_load.inp	Abaqus INP file with unit nodal loads only
		FEA_model.dat	Abaqus result file
		ABAQUS_RUN.bat	Batch file to run Abaqus
	.../Example/Results/40elems/3_NodalStress	UnitStress/Acceleration.csv UnitStress/FairleadTension.csv	Unit stress result files under unit nodal load

[2] ELSET that includes shell elements for stress output and visualization

	UnitStress/TowerBase.csv	
	StressHistory/NodalStress_Case1.bin StressHistory/NodalStress_Case2.bin StressHistory/NodalStress_Case3.bin	Nodal stress history files due to nodal loads in binary format
…/Example/Result/40elems/4_HydroLoad	FE-DIFF-D000.inp FE-RAD-DOF0.inp	Abaqus INP files to obtain stress RAOs
	DIFF-D000-F000.inp RAD-DOF0-F000.inp	Abaqus INP files with hydrodynamic loads
	FE-DIFF-D000.dat FE-RAD-DOF0.dat	Abaqus result files
	ABAQUS_RUN_DIFF.bat ABAQUS_RUN_RAD.bat	Batch files to run Abaqus
…/Example/Result/40elems/5_StressCom	StressCom_Case1.bin StressCom_Case2.bin StressCom_Case3.bin	Combined stress history files in binary format

3 Wamit model

Wamit v7 was used for Wamit analysis. The Wamit analysis model in Figure 3.1 was created in Abaqus INP file format and converted to the Wamit GDF (geometric data file) file using the '0 Aba2Wamit' module. The Wamit analysis conditions performed are shown in the following Table 3-1.

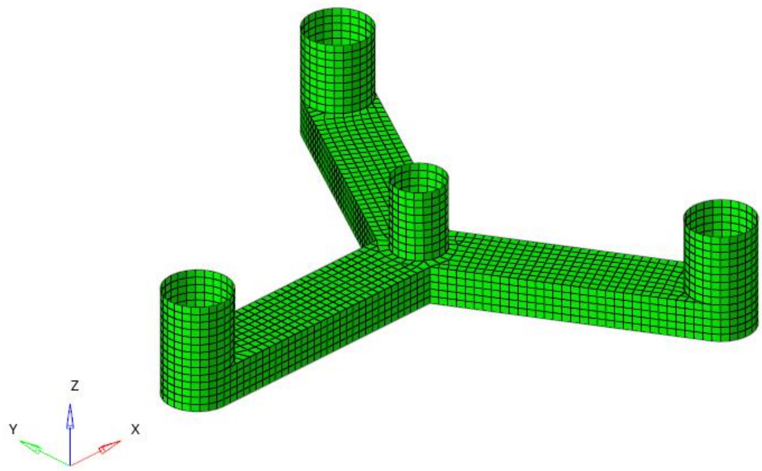


Figure 3.1 Wamit analysis model

Table 3-1 Wamit analysis conditions

Item	Value
Element size	1.5 m
The number of diffraction elements	3,624
Water depth	200 m
Frequency range	0.01 rad/s ~ 3.5 rad/s by 0.05 rad/s
The number of frequencies	71
Incident wave angles	0, 90, and 180 degrees

4 OpenFAST model

Load analyses were performed for DLC 1.6 by creating cases with three irregular phases. The analysis time for each case was 10 minutes. The tidal current speed was not taken into account. The conditions used in the OpenFAST analyses are shown in Table 4-1.

Table 4-1 Design load case

DLC 1.6			
Load	Item	Value	Note
Wind	Wind speed at hub height	11.4 m/s	10 min average
	Wind shear exponent	0.14	
	Wind model	NTM	
	Turbulence intensity	14%	
	Wind direction	0 deg	Upwind
Wave	Significant wave height	6.68 m	
	Wave peak period	10 s	
	Peakness parameter	3.73	
	Wave direction	0 deg	
Number of phases		3	
Analysis time per case		10 min	
Time increment Δt		0.0125 s	

5 Abaqus model

The FEA model for the Abaqus analysis is shown in Figure 5.1. The side shell plates of the pontoons and columns were modeled as shell elements. The bulkheads and web frames were also modeled as shell elements. The deck connecting the center and side columns and internal longitudinal stiffeners were modeled with beam elements.

The number of elements is given in Table 5-1. The applied material constants are presented in Table 5-2. The dimensions of the main structural members are summarized in Table 5-3.

The minimum constraints to restrain the six degrees of freedom motion of the FOWT were applied to the FEA model. The constraints were applied to the bottom center of each side column that are located furthest from the hotspots of the tower base and fairleads. The applied boundary conditions are shown in Figure 5.2.

Figure 5.3 and Figure 5.4 show the ELSETs for monitoring stresses for 40 elements and 5,000 elements, respectively.

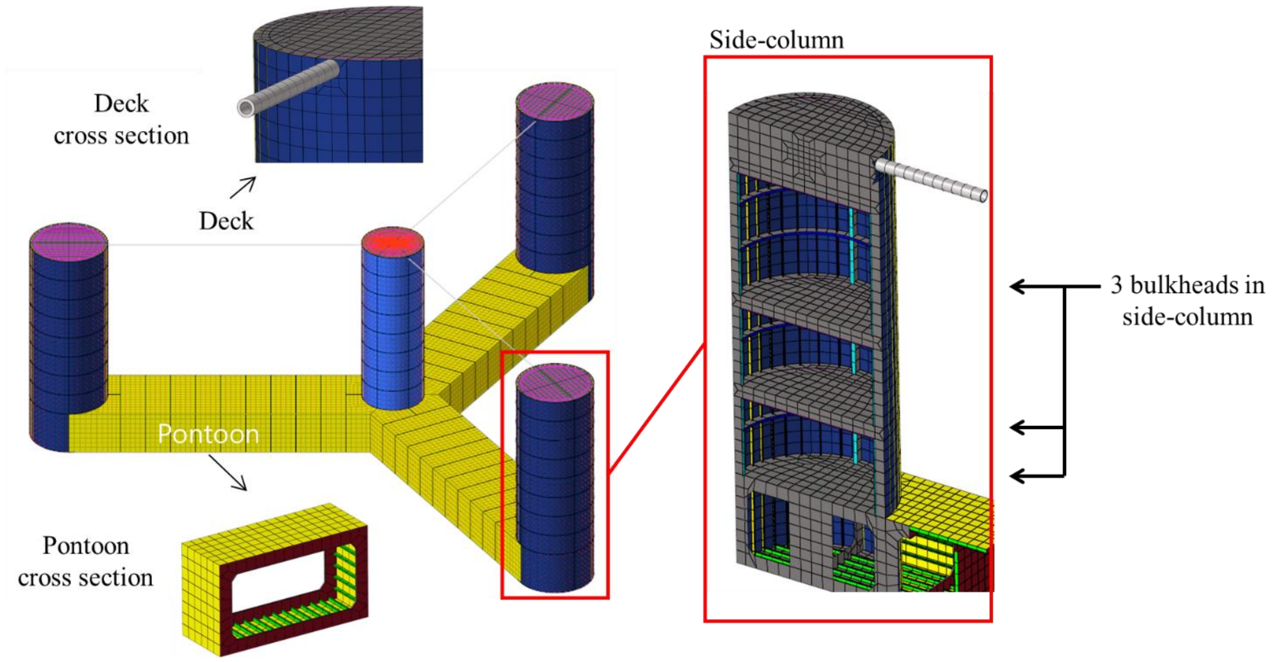


Figure 5.1 FEA model.

Table 5-1 Element information for FEA.

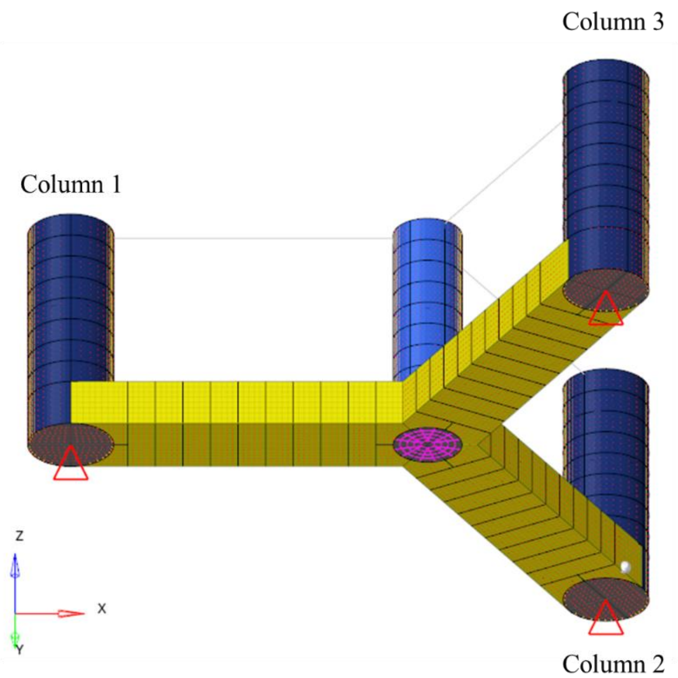
Element	Members	Number
B31 (beam)	Stiffeners	19,494
S3R & S4R (shell)	Columns, pontoons, web frames, and stringers	26,694
Distributing coupling	Tower base force distribution	1
Sum		46,189

Table 5-2 Material constants.

Density (ton/mm^3)	Young modulus (GPa)	Poisson ratio	Yield strength (MPa)
7.85E-9	206.0	0.3	n/a

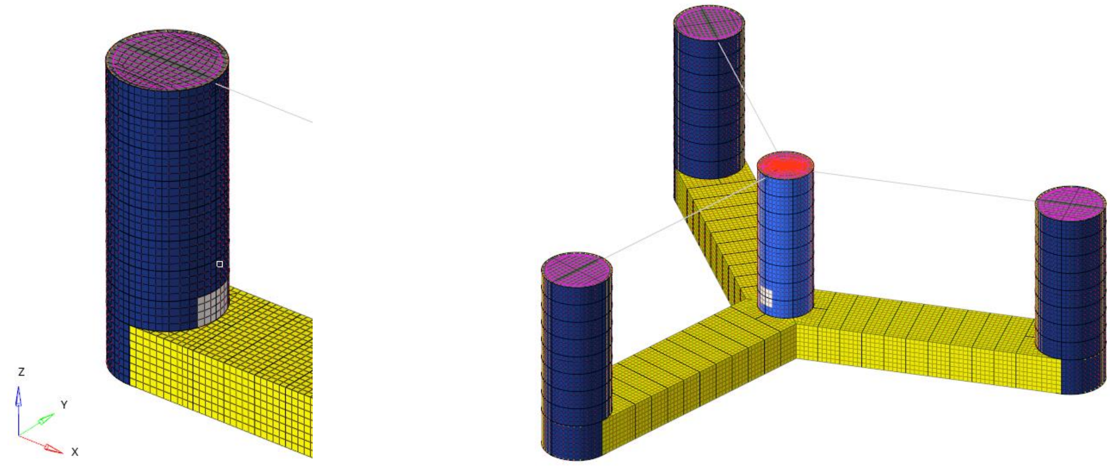
Table 5-3 Structural member dimensions.

Item	Property	Note
Brace (Deck)	OD455 x 55t	
Column BHD	30t	
Column shell	30t	
Pontoon shell	30t	
Pontoon BHD	30t	
Column vertical stiffener	300 x 20/100 x 15	
Column BHD stiffener	150 x 15	
Column girder face plate	300 x 20	
Column stringer	1000 x 30	
Pontoon web face plate	100 x 20	
Pontoon stiffener	300 x 15/150 x 15	
Pontoon web	1000 x 30	
Stringer face plate	200 x 20	



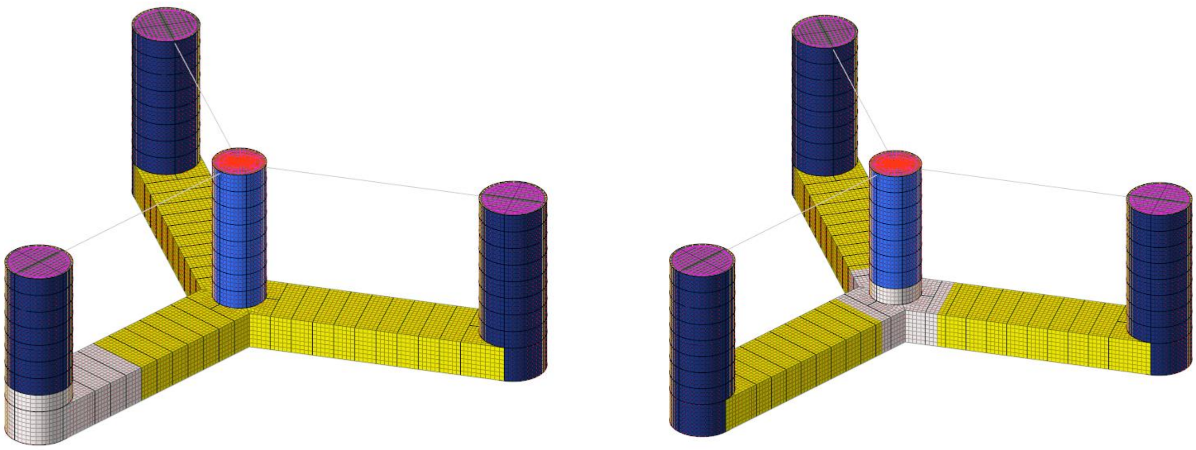
Location	Constraint DOF
Column #1	3
Column #2	1, 3
Column #3	1, 2, 3

Figure 5.2 Boundary condition.



(a) ELSET40-Pontoon-SideCol (b) ELSET40-Pontoon-CenCol

Figure 5.3 Stress recovery ELSETs for 40 elements.



(c) ELSET5000-Pontoon-SideCol (d) ELSET5000-Pontoon-CenCol

Figure 5.4 Stress recovery ELSETs for 5,000 elements.