FLOWStress v2.0.0

User manual

Oct. 27, 2025

Edited by KIM, Gitae and CHOUNG, Joonmo

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

Layout

1   Introduction

1.1   Motivation of development

The superstructure of a floating offshore wind turbine (FOWT) consists of slender members, such as the rotor and tower, and stocky members, which can be considered as point masses, such as the nacelle. These superstructure characteristics of FOWTs facilitate the two-way coupling of aerodynamic loads and structural elasticity through blade element momentum theory (BEMT) and the finite element method (FEM). In other words, aeroelasticity techniques have been applied to the superstructure of FOWTs.

The columns, pontoons, and decks of semi-submersible substructures, which are considered the most applicable substructures for FOWT platforms, are composed of members that are not sufficiently slender, thus wave diffraction effects cannot be ignored. For this reason, the consideration of diffraction forces is essential for the hydrodynamic loads acting on the substructure of the FOWT platform. Furthermore, since the substructure can be represented by the structural properties of plate members, most of the substructure can be modelled using shell elements. In other words, the substructure characteristics of the FOWT make it necessary to model the hydrodynamic force theory based on diffraction wave theory and the two-way coupling of hydrodynamic force and structural elasticity through finite element analysis (FEA) of the shell element.

This frequency-domain based hydrodynamic force theory can be extended to the time domain via the Cummins equation. However, the convolution integral of the hydrodynamic pressures on the side shell discretised as diffraction panel elements requires too much computational time. Therefore, it is customary to consider the substructure as a single mass point or rigid body. The time series motion response characteristics of the FOWT can then be obtained by substituting the tower base load, which takes into account the aeroelastic effects of the superstructure, into the Cummins equation as an external force.

However, while this coupled analysis technique can significantly reduce the computational load by not convolving the hydrodynamic pressures acting on the wet panel elements, it does not allow the structural elasticity of the substructure to be taken into account. For this reason, DNV has proposed a one-way coupled technique. Even though DNV's method is based on the one-way coupled technique, it applies transient response FEA based on direct integration or modal superposition, which creates a major bottleneck in the FEA phase to obtain the structural elasticity of the substructure (see Figure 1.1).

IEC 61400-3 requires around 30 design load cases (DLCs) for one new FOWT design. Each DLC must be subdivided into many sub-DLCs by wind direction, wave direction, wind speed, wave height, wave period, yaw misalignment, etc. One sub-DLC requires a 3-hour integrated load analysis (ILA), as the short-term sea state is typically known to be 3 hours which ensure statistical stability of the stationary and ergodic wave process. Furthermore, due to the nature of ILA, where wind speed and wave height are determined from wind and wave spectra that do not contain phase information, multiple wind and wave generations are required to ensure that the irregularities due to phase differences are captured. This is commonly referred to as 'the number of seeds'.

For example, DLC 6.2 is broken down into sub-DLCs: 3 wind directions, 3 wave directions, 3 current directions, 12 yaw misalignment angles, 3 wave height/wave period combinations, and 6 seeds increase the number of sub-DLCs to a minimum of 648 (1,944 hours) and a maximum of 5832 (17,469 hours) depending on the combination of wind direction, wave direction, and current direction (see Table 1.1). Although the computational time is highly dependent on the performance of the computer and the number of degrees of freedom of the substructure FEA model, a transient response FEA with one sub-DLC for a substructure with 30,000 elements would require more than 24 hours of computational time. Since the natural period of the superstructure is usually shorter than that of the substructure, a smaller time increment is required for transient response FEA, approximately 1/100th of a second, based on the natural period of the superstructure. The resulting output files would be several tens of GB in size. Therefore, the computation time and data storage required to run transient response FEAs for all DLCs is described as an impossible design process rather than a design bottleneck.
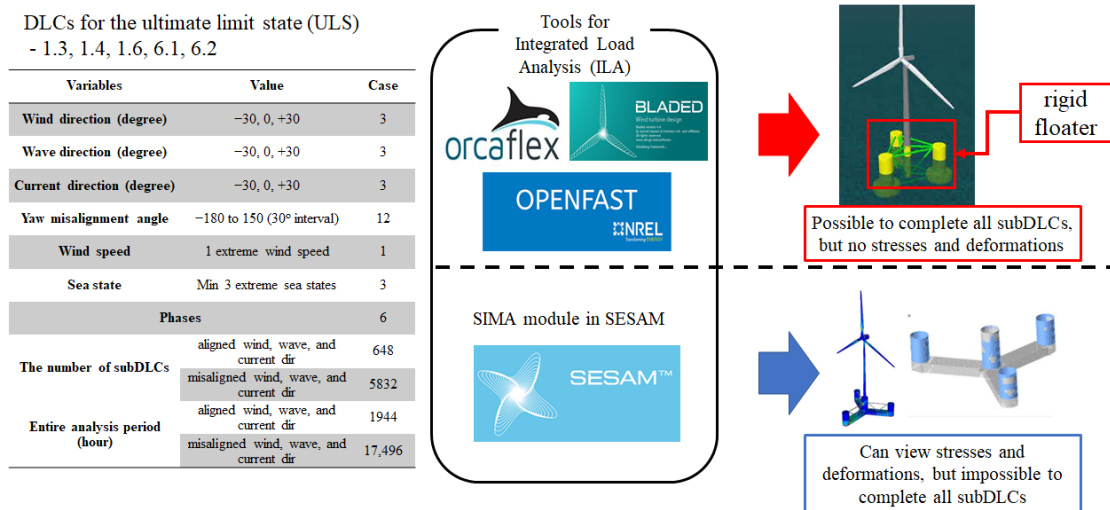
Figure 1.1 Bottlenecks in designing a FOWT unit platform

Table 1.1 The estimated number of sub-DLCs for DLC 6.2

Variables	Value	Case
Wind direction (degree)	−30, 0, +30	3
Wave direction (degree)	−30, 0, +30	3
Current direction (degree)	−30, 0, +30	3
Yaw misalignment angle	−180 to 150 (30º interval)	12
Wind speed	Min 1 extreme wind speed	1
Sea state	Min 3 extreme sea states	3
	The number of seeds	6
The number of subDLCs	aligned wind, wave, and current dir	648
	misaligned wind, wave, and current dir	5832
Entire analysis time (hour)	aligned wind, wave, and current dir	1944
	misaligned wind, wave, and current dir	17,496

1.2 Key algorithm of FLOWStress

This research was initiated to address these design bottleneck issues. The result of this research is the development of FLOWStress (FLoating Offshore Wind Stress analysis). FLOWStress assumes that the relationship between load and response is a linear system. FLOWStress is coupled with Wamit (or Nemoh), OpenFAST, and Abaqus/Standard. Ansys/Aqwa does not work with FLOWStress because it cannot individually produce the radiation- and wave excitation-induced hydrodynamic forces. FLOWStress utilizes the results of OpenFAST. ILA solvers such as Bladed, Hawc2, etc. can be linked with FLOWStress as long as those results are converted to OpenFAST result format. FLOWStress uses Abaqus/Standard results to obtain stress RAOs. Other FEA solvers such as MSC/Nastran, Altair/Optistruct, and Calculi-X will be developed in the future. FLOWStress derives time series stresses of shell elements belonging to substructure. The solvers required to run FLOWStress is summarized in Table 1.2 and Figure 1.2.

Table 1.2 Coupled solvers with FLOWStress

Solvers	Analysis type	Output	Remark
Nemoh	- Frequency domain hydrodynamic analysis	- Wave excitation-induced hydrodynamic force constants	Nemoh or Wamit should be alternatively used.
Wamit	- Frequency domain hydrodynamic analysis	- Wave excitation-induced hydrodynamic force constants	
OpenFAST	- Time domain integrated load analysis	- Wave elevation history (it is also used as input) - Motion history at CoM	
Abaqus/Standard	- Quasi-static linear stress analysis under nodal forces	- Stress distribution due to nodal forces (6 DoF CoM acceleration, 3 DoF fairlead tension, and 6 DoF tower base load)	
	- Frequency response analysis under hydrodynamic force RAOs	- Stress distribution due to hydrodynamic forces	

FLOWStress obtains stress distributions through quasi-static linear FEA under unit nodal forces. Typical nodal forces include the mooring tension on each fairlead, the tower base load, and the acceleration acting on the center of mass (CoM) of the FOWT. The time series stresses due to the nodal forces are then derived by multiplication with time domain nodal force data obtained through OpenFAST execution. FLOWStress linearly summarizes the stress history at each finite element due to the nodal forces, the radiation force, and the wave excitation force. The summation process of stresses is performed by stress components.

FLOWStress obtains hydrodynamic forces from the Wamit or Nemoh analysis. The hydrodynamic pressure RAOs are composed of pressures due to radiation force and 1st order wave excitation force. Each of these pressure RAOs is applied to

Abaqus to perform a frequency response FEA and obtain the stress RAOs. The motion history of the FOWT obtained from OpenFAST is subjected to a fast Fourier transform (FFT) to obtain the motion spectrum. Similarly, the wave elevation history of the FOWT applied to OpenFAST is FFTed to obtain the wave elevation spectrum. The radiation-induced stress RAO is interpolated or extrapolated to have the same frequency as the motion spectrum. Similarly, stress RAOs due to wave excitation force are interpolated or extrapolated to have the same frequency as the wave elevation spectrum. The stress spectrum attributable to each load is calibrated to have the same phases as the motion spectrum and the wave elevation spectrum, respectively. The two stress spectra are subjected to an inverse Fast Fourier Transform (IFFT) to obtain the radiation-induced stress history and the wave elevation-induced stress history, respectively. The time increment for the IFFT should be the same as that in OpenFAST. This process is illustrated in Figure 1.3.
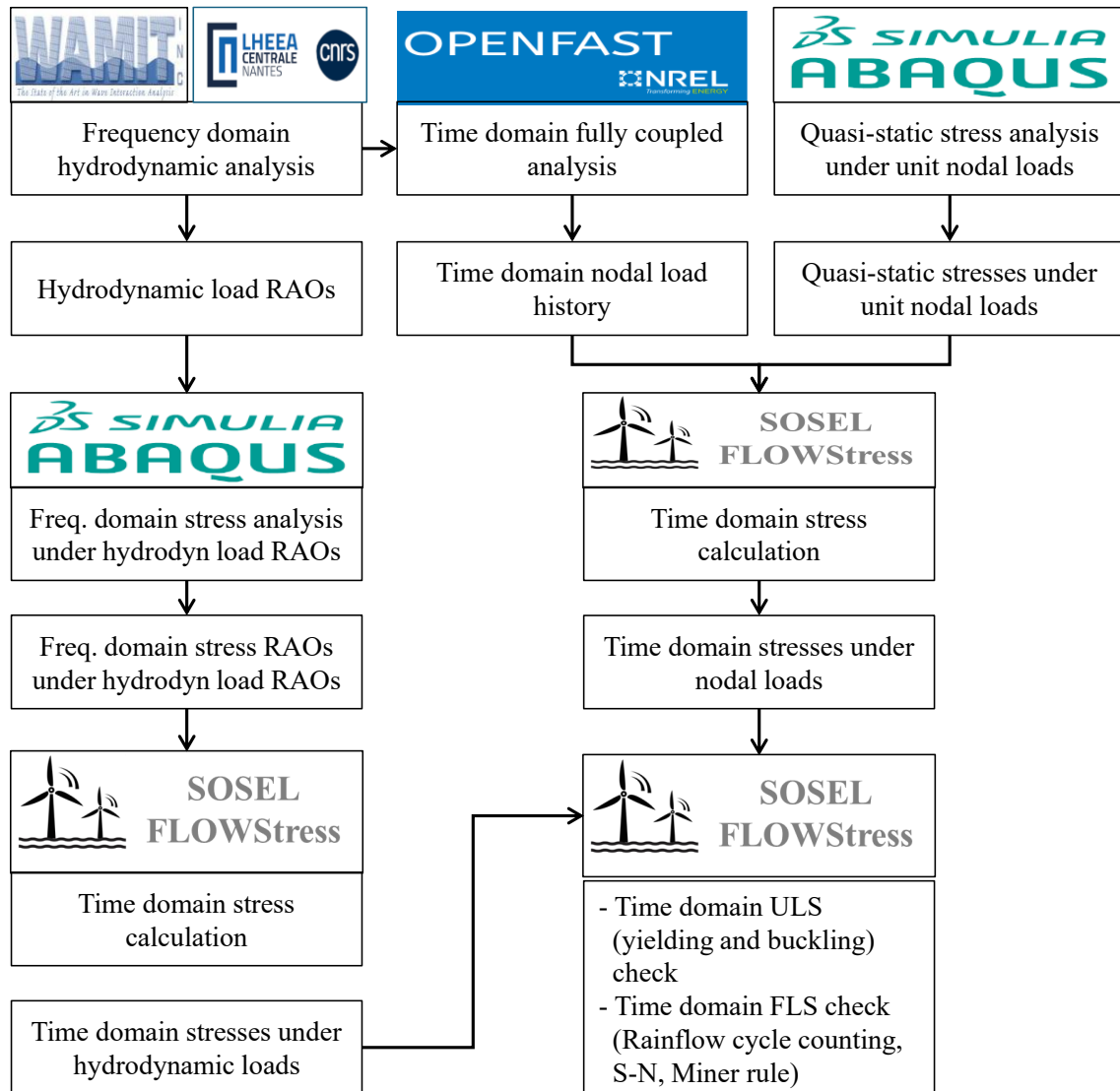

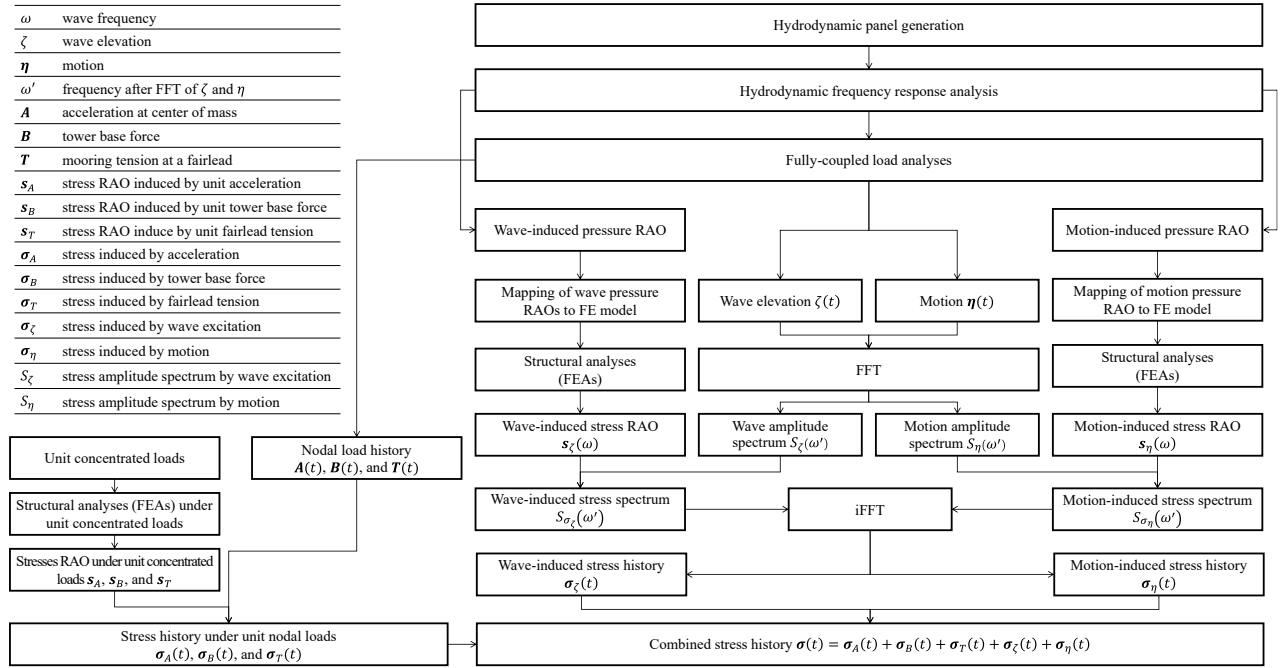Figure 1.2 Key algorithm of FLOWStress from S/W interface perspective

ω	wave frequency
ζ	wave elevation
$\boldsymbol{\eta}$	motion
ω'	frequency after FFT of ζ and η
$\boldsymbol{A}$	acceleration at center of mass
$\boldsymbol{B}$	tower base force
$\boldsymbol{T}$	mooring tension at a fairlead
$\boldsymbol{s_A}$	stress RAO induced by unit acceleration
$\boldsymbol{s_B}$	stress RAO induced by unit tower base force
$\boldsymbol{s_T}$	stress RAO induce by unit fairlead tension
$\boldsymbol{\sigma_A}$	stress induced by acceleration
$\boldsymbol{\sigma_B}$	stress induced by tower base force
$\boldsymbol{\sigma_T}$	stress induced by fairlead tension
$\boldsymbol{\sigma_\zeta}$	stress induced by wave excitation
$\boldsymbol{\sigma_\eta}$	stress induced by motion
S_ζ	stress amplitude spectrum by wave excitation
S_η	stress amplitude spectrum by motion

Flowchart boxes:

- Hydrodynamic panel generation
- Hydrodynamic frequency response analysis
- Fully-coupled load analyses
- Wave-induced pressure RAO
- Wave elevation $\zeta(t)$ — Motion $\boldsymbol{\eta}(t)$
- Motion-induced pressure RAO
- Mapping of wave pressure RAOs to FE model
- FFT
- Mapping of motion pressure RAO to FE model
- Structural analyses (FEAs)
- Structural analyses (FEAs)
- Wave-induced stress RAO $\boldsymbol{s_\zeta}(\omega)$
- Wave amplitude spectrum $S_\zeta(\omega')$ — Motion amplitude spectrum $S_\eta(\omega')$
- Motion-induced stress RAO $\boldsymbol{s_\eta}(\omega)$
- Wave-induced stress spectrum $S_{\sigma_\zeta}(\omega')$
- iFFT
- Motion-induced stress spectrum $S_{\sigma_\eta}(\omega')$
- Wave-induced stress history $\boldsymbol{\sigma_\zeta}(t)$
- Motion-induced stress history $\boldsymbol{\sigma_\eta}(t)$

Left column boxes:

- Unit concentrated loads
- Nodal load history $\boldsymbol{A}(t)$, $\boldsymbol{B}(t)$, and $\boldsymbol{T}(t)$
- Structural analyses (FEAs) under unit concentrated loads
- Stresses RAO under unit concentrated loads $\boldsymbol{s_A}$, $\boldsymbol{s_B}$, and $\boldsymbol{s_T}$
- Stress history under unit nodal loads $\boldsymbol{\sigma_A}(t)$, $\boldsymbol{\sigma_B}(t)$, and $\boldsymbol{\sigma_T}(t)$
- Combined stress history $\boldsymbol{\sigma}(t) = \boldsymbol{\sigma_A}(t) + \boldsymbol{\sigma_B}(t) + \boldsymbol{\sigma_T}(t) + \boldsymbol{\sigma_\zeta}(t) + \boldsymbol{\sigma_\eta}(t)$

Figure 1.3 Key algorithm of FLOWStress from stress history generation technology perspective

1.3 Version notation rule

| FLOWStress	v	1	0	0
S/W name	version	New module addition	Function update	Bug fix
		Significant module update	Keyword addition or update	Text change

2   Verification of FLOWStress

The stress derivation process using FLOWStress was validated on a cantilever beam and a stiffened plate used in a FOWT. The stress history using FLOWStress was compared to that using Abaqus linear transient response FEA with a direct integration scheme.

2.1   Cantilever beam

The geometry and material properties of a cantilever beam with dimensions of 3,000 mm (length) x 500 mm (breadth) x 10 mm (height) are shown in Figure 2.1. A concentrated load F_x in the longitudinal direction, F_y in the breadth direction, and F_z in the height direction, representing the fairlead nodal force, was applied to the free end of the cantilever beam. The nodal force history was in the form of a sine wave as shown in Figure 2.2. Two pressure histories were applied to the free end element of the cantilever, one due to the wave excitation force and the other due to the radiation force. The pressure histories are shown in Figure 2.2 in the form of superimposed sine waves. After performing linear static FEA and linear transient response FEA on the cantilever under these loads, the stress history was derived at the fixed end element of the cantilever. After applying unit nodal forces and two pressures RAOs to the cantilever, the stress RAOs were then converted to time series stresses using FLOWStress.

It can be seen from Figure 2.3 that the linear static FEA results are in perfect agreement with the FLOWStress results. The linear transient response FEA results were in relatively good agreement with the FLOWStress results, but differences were observed due to the dynamic amplification factor (DAF). The cantilever's very slender dimensions may have contributed to the relatively large DAF.
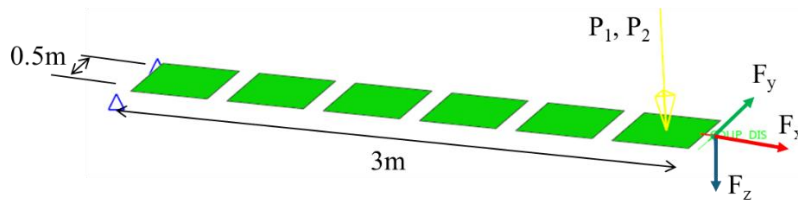
Density	7850 kg/m³
Thickness	10 mm
E	206 GPa
Nu	0.3
Dload	Pressure 1, 2
Cload	F_x, F_y, F_z

Figure 2.1 Cantilever beam

$$F_x(t) = \sin\left(\frac{2\pi t}{15}\right), F_y(t) = \sin\left(\frac{2\pi t}{15} + \frac{2\pi}{3}\right), F_z(t) = \sin\left(\frac{2\pi t}{15} + \frac{4\pi}{3}\right)$$

$$p_1(t) = 1.0\sin\left(\frac{2\pi t}{3}\right) + 0.5\sin\left(\frac{2\pi t}{5}\right) + 0.3\sin\left(\frac{2\pi t}{6}\right) + 0.1\sin\left(\frac{2\pi t}{12}\right)$$

$$p_2(t) = 0.8\sin\left(\frac{2\pi t}{2}\right) + 0.6\sin\left(\frac{2\pi t}{4}\right) + 0.4\sin\left(\frac{2\pi t}{15}\right) + 0.2\sin\left(\frac{2\pi t}{60}\right)$$
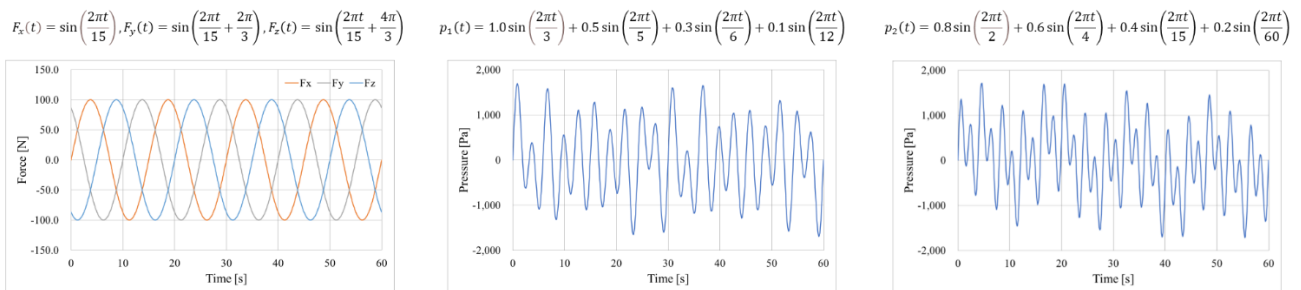

Figure 2.2 Point load history and pressure history
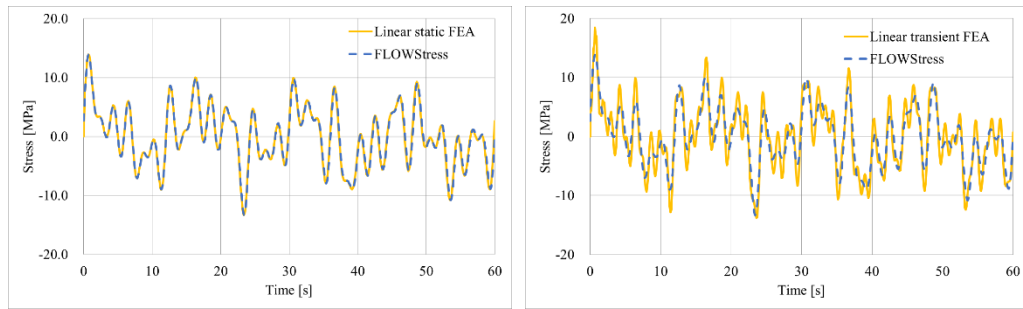
Figure 2.3 Result comparison; static analysis (left) and dynamic analysis (right)

2.2   Stiffened panel

Wamit V7 was used to derive the hydrodynamic coefficients for the FOWT substructure (see Figure 2.4), and ILA was performed using OpenFAST V3.3.0 to derive the motion histories. The radiation-induced hydrodynamic pressure RAO and wave elevation induced hydrodynamic pressure RAO were converted to their respective spectra. IFFT were performed to derive the wave-induced hydrodynamic pressure history and radiation-induced hydrodynamic pressure history. Each hydrodynamic pressure was applied to FEA model of Abaqus/Standard V2023 to perform direct integration-based linear transient response FEA.

In addition, the radiation-induced hydrodynamic pressure RAO and wave-induced hydrodynamic pressure RAO were applied to FEA model of Abaqus/Standard V2023 to obtain a linear static stress RAO. From this stress RAO, a stress time series was derived using FLOWStress. The stress history is shown in Figure 2.5. It can be seen that the high stiffness of the stiffener plate resulted in very little DAF and the two results are in good agreement.

Longi. Space	1 m
Density	7850 kg/m³
Stiffener	210 GPa
E	0.3
Dload	Wave excitation induced pressure
Cload	-
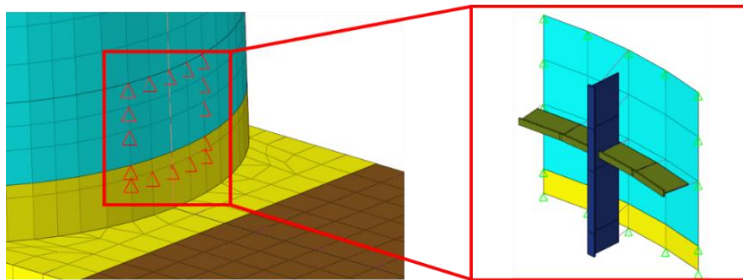
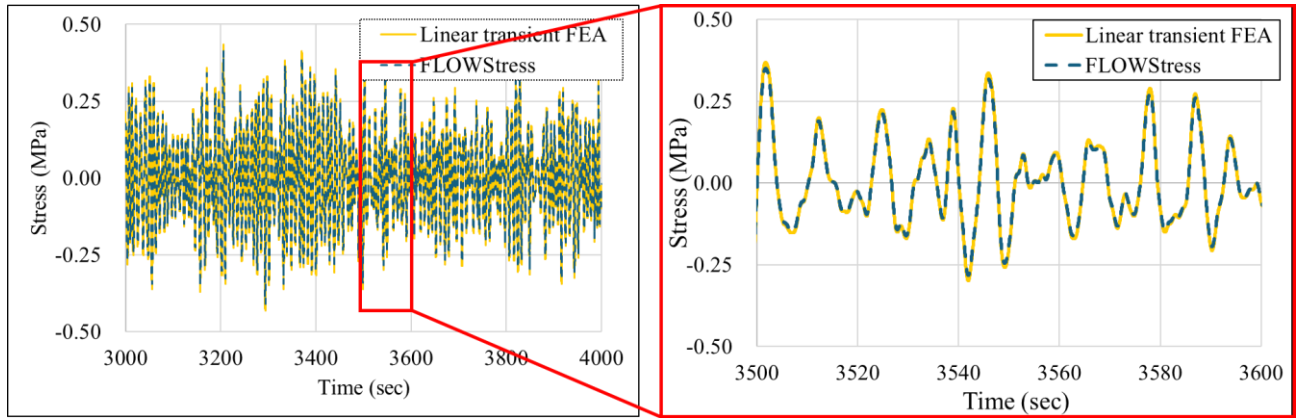Figure 2.4 Stiffened panel of FOWT

Figure 2.5 Result comparison; static analysis (left), dynamic analysis (right)

3   How to use FLOWStress

3.1   Installation of FLOWStress

- Download MSI file from GitHub

- Execute MSI file as an administrator

- Follow setup wizard with default options
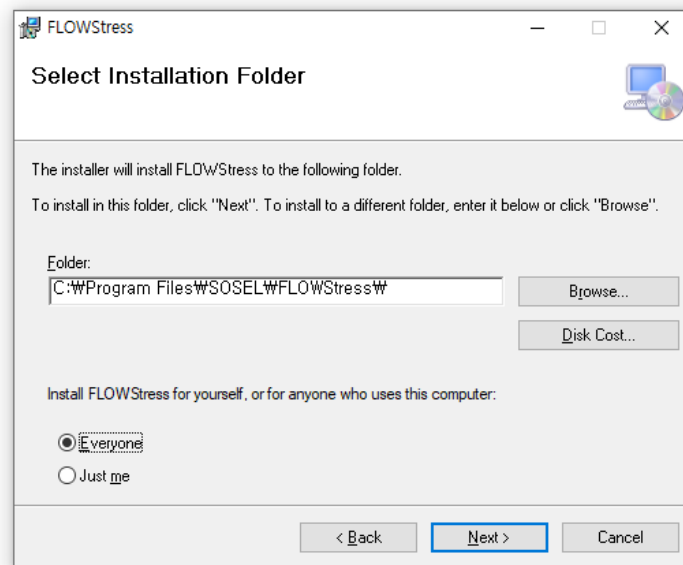
- No license required

Figure 3.1 FLOWStress installer

3.2   Data for FLOWStress

3.2.1   Wamit

Input

- Prepare an INP file in Abaqus format with panel elements and nodal points.

- The normal vector of the diffraction panel elements should point away from the structure towards the sea water.

- The origin of Wamit should be the same as that of OpenFAST. Since OpenFAST typically uses the tower base as the origin, it is recommended that the origin of Wamit be set to the same.

> x origin location: tower base
>
> y origin location: tower base
>
> z origin location: mean waterline plane

- 'INUMOPT5=1' must be set in Wamit's CFG file to write the wave-induced and radiation-induced pressure RAOs to the 5P file.

Execution

- Convert the Abaqus INP file to a Wamit GDF file using the '0 FRA' module in FLOWStress.

- The Wamit analysis is used to 1) obtain the hydrodynamic force coefficients for OpenFAST and 2) derive the panel pressure RAOs and accelerations at CoM for Abaqus FEA.

Output

- FLOWStress obtains the following necessary information from the Wamit FRC, GDF, POT files (input files) and 5P file (output file)

> FRC: Seawater density
>
> GDF: Panel elements, nodal coordinates, gravity acceleration
>
> POT: Wave directions and wave frequency
>
> 5P: Pressure due to hydrodynamic forces on panel elements

3.2.2 Nemoh

Input

- Prepare an INP file in Abaqus format with panel elements and nodal points.

- The normal vector of the diffraction panel elements should point away from the structure towards the sea water.

- The origin of Nemoh should be the same as that of OpenFAST. Since OpenFAST typically uses the tower base as the origin, it is recommended that the origin of Nemoh be set to the same.

> x origin location: tower base
>
> y origin location: tower base
>
> z origin location: mean waterline plane

Execution

- Convert the Abaqus INP file to a Nemoh DAT file using the '0 FRA' module in FLOWStress.

- The Nemoh analysis is used to 1) obtain the hydrodynamic force coefficients for OpenFAST and 2) derive the panel pressure RAOs and accelerations at CoM for Abaqus FEA.

Output

- FLOWStress obtains the following necessary information from two input files of Nemoh.CAL and Mesh.CAL and one output file of Pressure.#####.dat.

> DAT: Panel elements and nodal coordinates
>
> Mesh.CAL: Seawater density, CoM, and gravity
>
> Nemoh.CAL: Environmental data, wave directions, wave frequencies, etc
>
> Pressure.#####.DAT: Pressure due to hydrodynamic forces on panel elements

3.2.3   OpenFAST

Input

- The OpenFAST origin should be the same as the Wamit origin.

- FLOWStress obtains information from the DAT files of the following modules in OpenFAST.

> DAT file for HydroDyn module: wave directions
>
> DAT file for MoorDyn module: fairlead positions

Output

- FLOWStress obtains time series data of wave amplitude, 6 DoF motions at CoM, 6 DoF forces at tower base, acceleration at CoM, and tensions at fairleads from the OUT file obtained from the OpenFAST analysis.

- For this purpose, the physical quantities to be defined in the DAT file for each OpenFAST module are as follows, and the output keywords are as shown in Figure 3.2.

> HydroDyn module: wave amplitude & 6 DoF motions at CoM
>
> ElastoDyn module: 6 DoF forces at tower base & acceleration at CoM
>
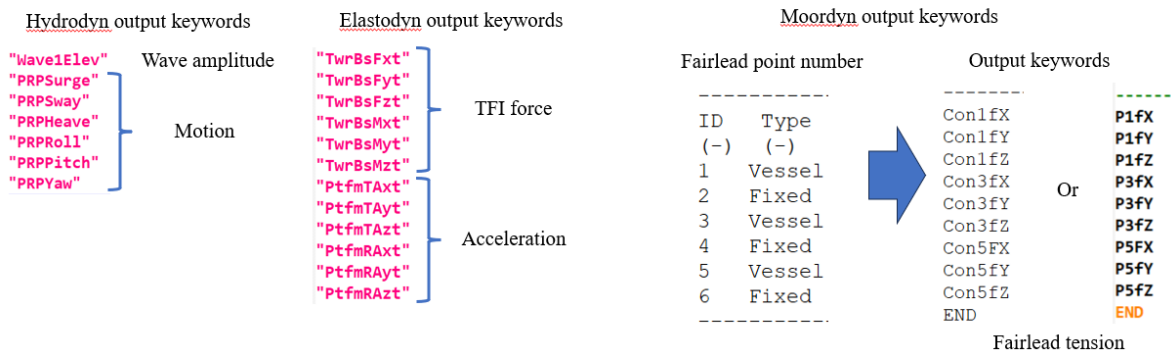> MoorDyn module: 3 DoF tensions at fairleads

Figure 3.2 OpenFAST keywords for output requests

3.2.4   Abaqus

Input

- If quadrilateral shell element with reduced integration scheme is used (S4R), the triangular shell element type must be S3R. On the other hand, if S4 are used, the triangular shell element type must be S3.

- The origin position of the Abaqus FEA model must be the same as the origin position used in OpenFAST and Wamit/Nemoh

- The normal vector of the wet side shell elements should point away from the structure towards the sea water[1].

---

[1] To distinguish between negative and positive pressure

- In the FEA model, a user needs to create one single NSET containing the tower base node and the fairlead nodes[2] (see Figure 3.3). This NSET name should be the same as the NSET name defined in the JOB file.
- The following ELSETs must be defined in the FEA model. This ELSET name must be the same as the ELSET name defined in the JOB file.

ELSET that acceleration loads should be applied
ELSET that plate pressure below the waterline should be applied
ELSET that stresses should be printed[3]
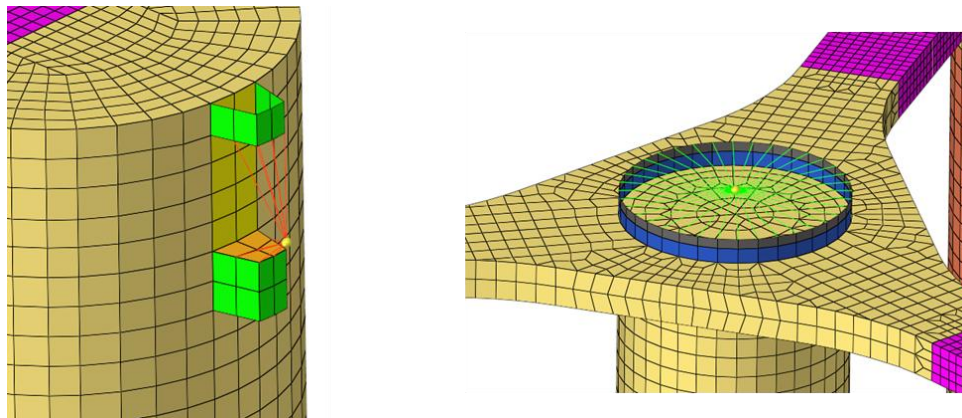
Figure 3.3 Fairlead node (left) and tower base node (right).

3.2.5   FLOWStress

Input (JOB file)

- The JOB file is the input file for FLOWStress execution.
- The keywords used in the JOB file can be found in Chapter 4.
- Keywords must start with a '*' (single asterisk).
- Data that depends on the keyword must be defined on the line following the keyword.
- Lines with '**' (double asterisk) are considered unconditional comments.

---

[2] The distance difference between the fairlead nodes in FEA and MoorDyn cannot exceed 1m.
$(\sqrt{\Delta x^2 + \Delta y^2 + \Delta z^2} < 1.0 \text{ m})$
[3] ELSETs for stress print must consist of shell elements only. Multiple ELSETs can be defined for this purpose.

3.3   FLOWStress modules

FLOWStress consists of six modules. The functions of each module are summarized in Table 3.1.

Table 3.1 Summary of FLOWStress modules

Module	Purpose	Remark
0FRA	Aba2Wamit: converting Abaqus INP file to Wamit GDF and POT files Aba2Nemoh: converting Abaqus INP file to Nemoh DAT, Nemoh.CAL, and mesh.CAL files	
1BucklingPre	Define the type of limit states, FEA model file path, FEA unit system selection, ELSET generation for buckling, ELSETs & NSETs for load mapping and stress output, and FEA output options.	New module to prepare automatic ELSET generation[4] for buckling evaluation of unstiffened panels.
2StaticLoad	Generating Abaqus INP file to include quasi-static hydrostatic pressure and gravity loads. Other static loads can be manually added to Abaqus INP file	A new folder '2StaticLoad' is created where a Abaqus INP file is generated.

3NodalLoad	Generating Abaqus INP file to include unit nodal loads at a single tower base node and fairlead nodes.
	  - 6 DoF load components at a tower base node
	  - 3 DoF load components at each fairlead node
	  - 6 DoF acceleration components at a CoM node
		A new folder '3NodalLoad' is created where a Abaqus INP file is generated.

4NodalStress	Creating stress histories in substructure due to unit nodal loads, multiplying unit load-induced stress by nodal load history obtained from OpenFAST.
	  - Unit load-induced stresses obtained from Abaqus
	  - Nodal load history obtained from OpenFAST
		A new folder '4NodalStress' is created where unit load-induced stress history files with BIN extension are generated.

5HydroLoad	Generating Abaqus INP file to include wave excitation-induced panel pressure RAO and radiation-induced panel pressure RAO, respectively. Panel pressure RAOs obtained from Wamit output 5P file or Nemoh output Pressure.#####.DAT files.	A new folder '5HydroLoad' is created where a Abaqus INP files are generated.
6StressCom	Combining static load-, nodal load-, and hydrodynamic load-induced stress history.	A new folder '6StressCom' is created where combined stress history files with BIN extension are generated.
7Buckling	Evaluating buckling strength analyses.	A new folder '7Buckling' is created

---

[4] The number of buckling panels that can be generated at one time is limited to two in order to restrict commercial use. If you wish to expand to the full set of panels, contact developers or leave messages in GitHub.

		where buckling analysis result files with CSV extension are generated.
8Fatigue	Evaluating screening and refined fatigue strength analyses.[5]	A new folder '8Fatigue' is created where a fatigue analysis result file with OUT extension is generated.

3.4 Graphic user interface

3.4.1 ShortCuts

Table 3.2 Summary of shortcuts in FLOWStress GUI

Module	Purpose
↑, ↓, ←, →	Rotate by 15-degree increment
CTRL+ (↑, ↓)	Rotate by 180-degree increment
CTRL+ (←, →)	Rotate by 15-degree increment wrt the axis facing user
space bar	Play and pause stress history
ESC	Pause stress history
Mouse left button	Rotate object
Mouse center wheel	Zoom-in and zoom-out object
Mouse right button	Pan object

3.4.2 Loading of stress visualization interface

- The stress combinations are completed through '5 StressCom' module.

- GUI 3D Viewer can be loaded after executing the visualization button.

- On launching the 3D viewer, it automatically reads the JOB file, INP file, and BIN file (stress combination results).

3.4.3 Element selection zone

- Select visualize option (Global = overall FE model; Local = ELSETs that output requested in Abaqus INP file)

- Select Elset[6]

---

[5] To limit commercial use, fatigue strength evaluation is possible at only one point. If the hotspot type is 0, fatigue evaluation is performed on the two element set, and if the hotspot type is 1 or higher, fatigue evaluation is performed on the first designated hotspot.

[6] If stress output contained in a ELSET defined in INP file exceeds 8GB, a series of suffixes are added to the ELSET a series of suffixes are added to the ELSET to keep the stress output below 8 GB.
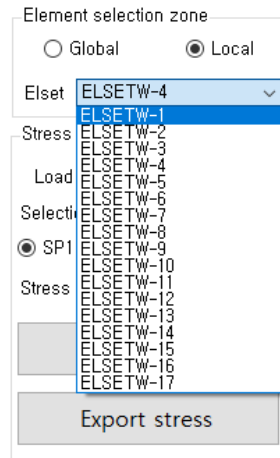
Figure 3.4 ELSETs with a series of suffixes that are automatically generated.

3.4.4 Stress display zone

- Select a load case (multiple load cases cannot be selected)
- Select the shell section stress integration point (SP1 = bottom layer; SP5 = top layer; MAX = max(SP1, SP5), AVG = average(SP1, SP5))
- Select 'Stress components' (S11 = normal stress in x-dir; S22 = normal stress in y-dir; S12 = shear stress; S1 = max principal normal stress; S2 = min principal normal stress; Max shear = principal shear stress; von Mises = von Mises equivalent stress)
- Run the 'Display' button to visualize the stresses.
- Run the 'Export'[7] button to export the stress selected.

3.4.5 Video zone

- Adjust the time (select the desired time; execute the time adjustment arrows; adjust the time slide bar)
- Play the video by executing the 'Play' button
- Adjust the 'Sampling freq' to speed up the video playback (if 5 is entered, the video will play using every 5th stress data)

---

[7] File name convention
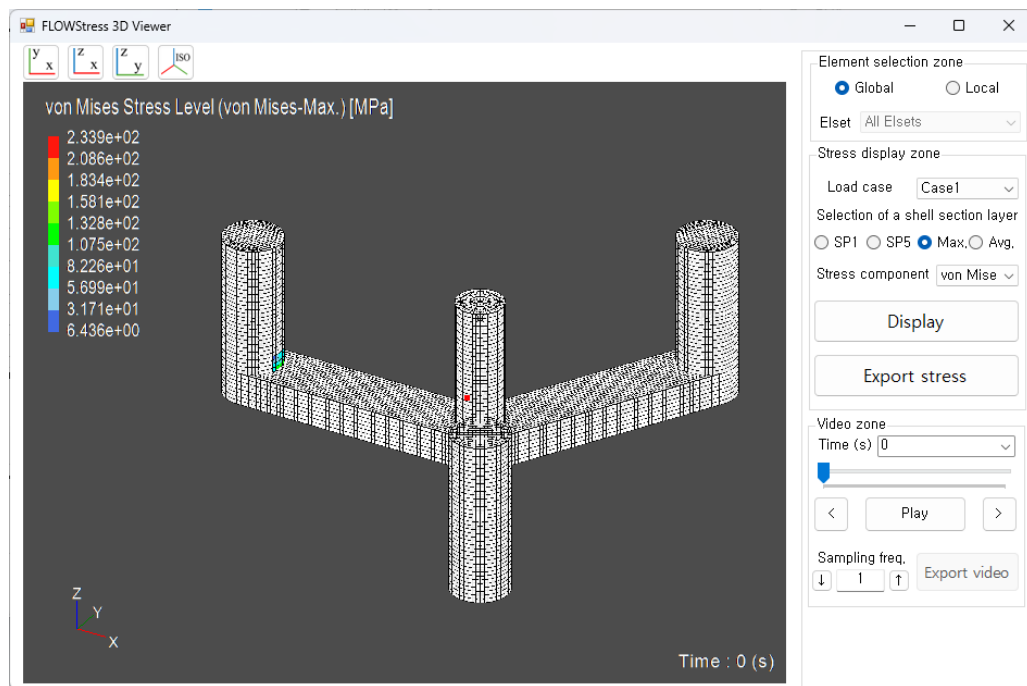ELSET_LoadCase_StressComponent_ElementSectionPoint.csv

Figure 3.5 Stress history visualization

3.4.6 Parallel computing

FLOWStress supports parallel computation through grouping of stress recovery elements. FLOWStress figures out the number of CPUs and the number of stress recovery elements and creates as many groups of stress recovery elements as the number of CPU cores. FLOWStress performs parallel computation for every group of stress recovery elements. For example, for 10,000 stress recovery elements on a 10-core CPU machine, it creates 10 groups of 1,000 elements and performs parallel computations.

FLOWStress allows not only parallel computation through grouping of elements, but also parallel computation through grouping of simultaneous load cases. In the design phase of a FOWT, many load cases will be required, and in this case, parallel computation through element grouping and load case grouping can dramatically reduce FLOWStress computation time. Parallelization with load case grouping is available upon individual request.

3.5 Limitations and errors

3.5.1 Size of RAM

- The number of elements, the number of time data, the number of stress components (σ_x, σ_y, τ_{xy}), and the number of shell stress recovery points in through-thickness direction (bottom and top) decide the computing load that is associated with the physical memory (RAM) size. FLOWStress is validated in Microsoft Windows 10 and 11 where RAM size of 32GB or greater size is recommended.

- For 10,000 elements and three hours ILA with Δt = 0.1 s, assuming one single float-type data takes 4 bytes, then required

data storage is determined as follows:

$$\frac{3h \times 3,600s}{0.1s} \times 10,000\text{elms} \times 3 \text{ stresses} \times 2 \text{ layers} \times 4 \text{ bytes} = 24.14\text{ GB}$$

- FLOWStress calculates and generates the stress history file (BIN file) per each load case for every shell element specified in INP file, simultaneously. Graphical visualization has been possible by automatically dividing stress data into 8GB that is the maximum array size in C#.

3.5.2   ABAQUS INP file format

- ELSET must be defined in "*ELSET" keyword and ELSET must be composed of element numbers.

** Readable format**
*ELSET, ELSET=Pontoon
1,   2,   3,   4,   5,   6,   7,   8,
9,   10,   11,   12,   13,   14,   15,   16,

** Unreadable format**
*ELSET, ELSET=Pontoon
Deck, Bottom, Tower

4 Keywords for JOB file

Table 4.1 Summary of JOB file keywords

Keyword	Description	Default value	Example with user-defined value	Module number used
*CDIA	- Column diameter - When *UNABS=1, default value is 10,000.0mm - When *UNABS=2, default value is 10.0m	Depends on *UNABS	*CDIA 10,000.0	1, 7
*COG	- Center of gravity (m) - It is a distance vector from the origin of global coordinate system - The origin of global coordinate system is commonly set as (CoGx, CoGy, Mean water level)	n/a	*COG 0.0, 0.0, -5.0	0 (Nemoh only), 3
*CPU	- The number of CPUs for FEA analysis - 1 (default)	*CPU ,	*CPU 10,	2, 3, 5
*DFLO	- Folder path where FLOWStress generates results - For example, *DFLO in Module2 automatically produces "2_StaticLoad" folder - For Modules 3, 4, 5, 6, 7, and 8, folder names that *DFLO produces are designated as "3_NodalLoad", "4_NodalStress", "5_HydroLoad", "6_StressCom", "7_Buckling", and "8_Fatigue"	n/a	*DFLO D:\FLOWStress\	2, 3, 4, 5, 6 ,7 ,8
*DFRA	- Folder path for frequency response analysis model and result files - Wamit input files (GDF, POT, CFG and FRC) must be in this folder before Wamit execution - Nemoh input files (DAT, Mesh.CAL, Input_solver.TXT and Nemoh.CAL) must be in this folder before Nemoh execution	n/a	*DFRA D:\WAMIT\ or *DFRA D:\NEMOH\	5, 6
*ELSETA	- Element set containing all massive elements which contribute the mass of structures	n/a	*ELSETA ELSET-All	2, 3
*ELSETB	- Element set containing identified buckling panels	n/a	*ELSETB ELSET-UP	7
*ELSETD	- Element set containing diffraction elements	n/a	*ELSETD ELSET-SideShell	2, 5
*ELSETS	- Element set containing elements with stress printed	n/a	*ELSETS ELSET-Stress	1, 2, 3, 4, 5, 6
*EXP	- Pressure mapping exponent - 2.0 (default)	*EXP ,	*EXP 8.0	5
*FA2N	- File "ABA2NEMOH.exe" to transform FEA model file to Nemoh model (DAT, Mesh.CAL, Input_solver.TXT and Nemoh.CAL) file - It is located at installation folder that a user assigned	*FA2N C:\ProgramFiles\CommonFiles\FLOWStress\ABA2NEMOH\ABA2NEMOH.exe	User-defined folder in which ABA2NEMOH.exe is placed	0 (Nemoh only)
*FA2W	- File "ABA2WAMIT.exe" to transform FEA model file to Wamit model (GDF and POT) file - It is located at installation folder that a user assigned	*FA2W C:\ProgramFiles\CommonFiles\FLOWStress\ABA2WAMIT\ABA2WAMIT.exe	User-defined folder in which ABA2WAMIT.exe is placed	0 (Wamit only)
*FAB0	- FEA model file which will be transformed to Wamit or Nemoh model file - It should contain only diffraction and non-diffraction panel elements - FRA model and result files should be in a same folder	n/a	*FAB0 D:\Example\WAMIT\Wamit.inp	0
*FABB	- FEA element set file in which identified buckling panels are defined	n/a	*FABB D:\Example\Abaqus\Abaqus_ELSET.inp	7
*FABS	- FEA seed file - FEA seed file includes only model data such as nodes, element, section and material properties, and pre-assigend NSETs and ELSETs, while no load case data (history data) are included.	n/a	*FABS D:\Example\Abaqus\Abaqus.inp	1, 2, 3, 4, 5, 6, 8
*FAT	- Fatigue assessment parameters - Hotspot type, ELSET, m1, log(a1), m2, log(a2), t_ref, k, DFF, SCF	n/a	*FAT 0, ELSET, 3.0, 12.164, 5.0, 15.606, 25, 0.2, 1, 1	8
*FHYD	- OpenFAST HydroDyn DAT file(s) - For multiple load cases with a same wave incident angle, single or multiple datalines are available - For multiple load cases with each wave incident angle, multiple dataline must be defined	n/a	*FHYD D:\OpenFAST\HydroDyn_Case01.dat D:\OpenFAST\HydroDyn_Case02.dat	6
*FMOR	- OpenFAST MoorDyn DAT file	n/a	*FMOR D:\OpenFAST\MoorDyn.dat	3
*FOO	- OpenFAST OUT file(s)	n/a	*FOO D:\OpenFAST\Case01.out D:\OpenFAST\Case02.out	4, 6, 8
*FRAC	- FRA code or potential code - 1: Wamit (default) - 2: Nemoh	*FRAC ,	*FRAC 2,	0, 5, 6
*GRAV	- Gravity in m/s^2 - 9.80655 (default)	*GRAV ,	*GRAV 9.81,	0, 3
*LIF	- Design life in year - 20.0 (default)	*LIF ,	*LIF 20,	8
*NSET1	- Node set cobtaining fairlead nodes - It must be pre-defined in FEA seed file.	n/a	*NSET1 NSETF	3
*NSET2	- Node set containing a tower base node - It must be pre-defined in FEA seed file.	n/a	*NSET2 NSETT	3
*OFO	- ODB field output option - S: stress, E: strain, U: disp, RF: reaction force - S, E, U, and RF (default)	*OFO ,	*OFO S, E, U, RF	2, 3, 5
*PANG	- Buckling panel tolerance angle in degrees - 5.0 (default)	*PANG ,	*PANG 2.0,	1, 7
*PROB	- Probability of each load case - The number of dataline should be same as the number of load cases in OpenFAST - Sum of probability should be 1.0	n/a	*PROB 0.3, 0.7,	8
*PTA	- Pressure mapping tolerance angle (deg) to identify elements in a same plane - Must be ranged 0.0° < PTA < 10° - 2.0 (default)	*PTA ,	*PTA 3.0,	5
*SFQ	- Sampling frequency for stress history - Default sampling frequency is 1 so that the number of stress history is same as that in OpenFAST result - For example, it is set 2, then the number of stress history is reduced to the half of that in OPENFAST results	*SFQ ,	*SFQ 2,	4, 6
*SRO	- Stress print option according to load sources - WEX: wave excitation, RAD: radiation, PL: nodal load, SL: static load - WEX, RAD, PL, and SL (default)	*SRO ,	*SRO WEX, RAD, SL	6
*UNA0	- Unit system of FEA model for FRA transform - 1: m (default) - 2: mm	*UNA0 ,	*UNA0 2,	0
*UNABS	- Unit system of FEA model containing seed model - 1: m - 2: mm (default)	*UNABS ,	*UNABS 1,	1, 2, 3, 4, 5, 6, 7
*WDEP	- Water depth (m) - Infinite (default)	*WDEP ,	*WDEP 150,	0
*WFR	- Wave frequency (rad/s) - From, To, Increment	n/a	*WFR 0.01, 3.51, 0.05	0
*WINC	- Wave incident angle (deg) - From, To, Increment	n/a	*WINC 0.0, 180.0, 30.0	0
*WUP	- Warm-up period to be removed in OpenFAST results - 600.0 (default)	*WUP ,	*WUP 300.0,	4, 6

*EXP

First job is to find the five panel elements that lie in the same plane and are closest to the center of the jth finite element (the closest one is P5 and the others are P2, P4, P6, and P8 in Figure 4.1). For each of the five panel elements, find the relative distance w_i (the relative distance between the finite element center and the panel element center) using Equation (1). The pressure F_j acting on the jth finite element is given by Equation (2) where α is called the distance weight exponent for the pressure, and *EXP is the corresponding value. In general, α = 2.0 is recommended.

$$w_i = 1 - \frac{d_i}{d_{max}} \tag{1}$$

$$F_j = \sum_{i=1}^{n} p_i \times w_i^{\alpha} / \sum_{i=1}^{n} w_i^{\alpha} \tag{2}$$
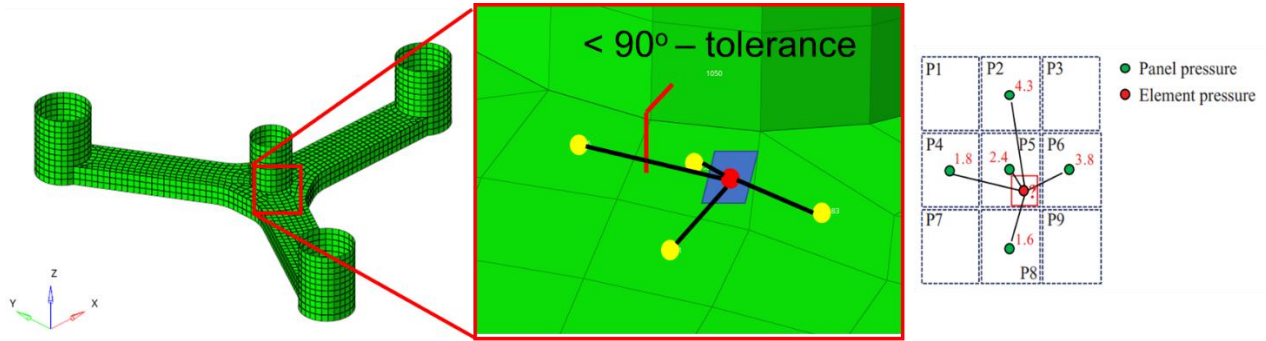
Figure 4.1 Pressure mapping

*FAB1

This Abaqus INP file is reference file, so it must not include HISTORY part but only MODEL part. The keyword '*step' separate MODEL part and HISTORY part in Abaqus. From '*step' keyword, data corresponds to HISTORY part.

*PTA

The condition for the panel to be in the same plane as the element is determined using the norm of the normal vectors of the panel and the element. *PTA is the difference between the normal vectors. For small diameter circular columns, this angle must be increased to be recognized as one single plane. *PTA of 2 degrees is generally recommended.

*PANG

*PANG is used for the automatic recognition of an unstiffened panel for buckling evaluation. For this purpose, as shown in Figure 4.2(a), FLOWStress automatically recognizes an unstiffened panel consisting of multiple elements, bounded by stiffeners and web frames modeled with beam elements and shell elements. If the ith element belonging to the unstiffened panel and the ith element's unit normal vector $\boldsymbol{n}_i$ have an angular difference of θ_i (see Equations (3)-(4)) that is smaller than the tolerance angle defined in *PANG, the two elements are finally assigned to this unstiffened panel. If θ_i is larger than the tolerance angle, as shown in Figure 4.2(b), two unstiffened panels are generated. The tolerance angle of 5 degrees is

generally recommended. In Equation (3), α_i, β_i, and γ_i are the direction cosines of ith element.

(a) Separated by stiffeners and web frames modeled with beam and shell elements   (b) Separated by element unit normal vectors
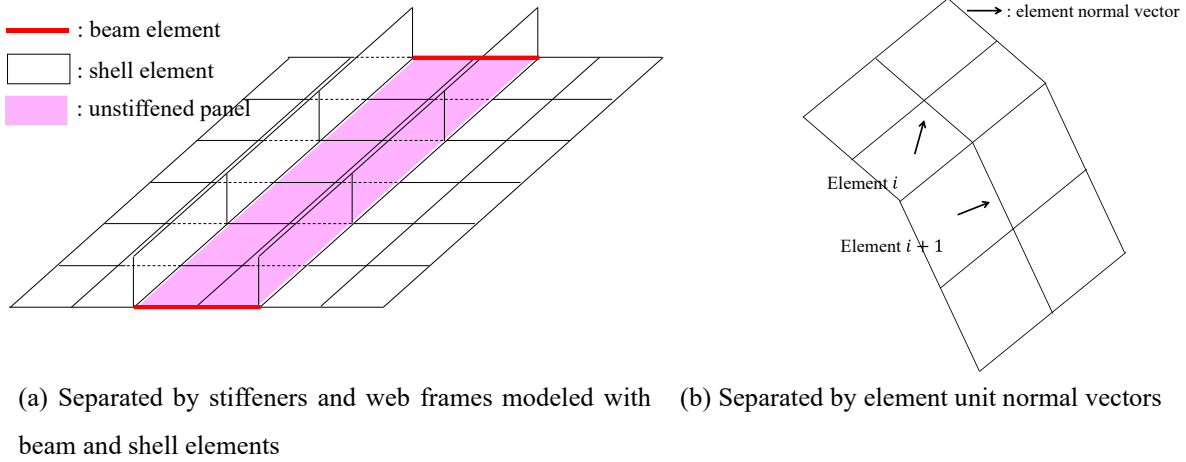
Figure 4.2 Reading of unstiffened panels

$$\boldsymbol{n}_i = \cos\alpha_i\,\boldsymbol{i} + \cos\beta_i\,\boldsymbol{j} + \cos\gamma_i\,\boldsymbol{k} \tag{3}$$

$$\theta_i = \cos^{-1}\frac{\boldsymbol{n}_i \cdot \boldsymbol{n}_{i+1}}{\|\boldsymbol{n}_i\|\|\boldsymbol{n}_{i+1}\|} \tag{4}$$

*CDIA

Since the buckling ultimate strengths of a flat unstiffened panel and a curved unstiffened panel are evaluated using different formulas, the keyword *CDIA is required to distinguish between the two types. In the FOWT substructure shown in Figure 4.3, three main columns and one tower column are the cylindrical structures. For example, in the case of a floating substructure as shown in Figure 4.3, the minimum diameter $D_{\min}$ defined in *CDIA is the diameter of the tower column. FLOWStress calculates the angle θ_j between the unit normal vectors $\boldsymbol{n}_j$ and $\boldsymbol{n}_{j+1}$ of the jth and $j+1^{th}$ unstiffened panels. As presented in Equation (5), if the width b_j of the jth unstiffened panel is greater than the product of θ_j and the circumference generated by the minimum diameter, the panel is regarded as a flat unstiffened panel. Through this criterion, the type of unstiffened panel can be determined without being affected by the absolute value of the minimum diameter, the spacing of stiffeners, or the spacing of web frames.

The unit of $D_{\min}$ defined in the keyword *CDIA must be consistent with the unit system used in the FEA model. Assuming that the length unit of the FEA model is millimeters, the default value of $D_{\min}$ is set to 10,000.

$$\begin{aligned} b_j &\geq \pi D_{\min} \cdot \theta_j \quad \text{for flat unstiffened panel} \\ b_j &< \pi D_{\min} \cdot \theta_j \quad \text{for curved unstiffened panel} \end{aligned} \tag{5}$$
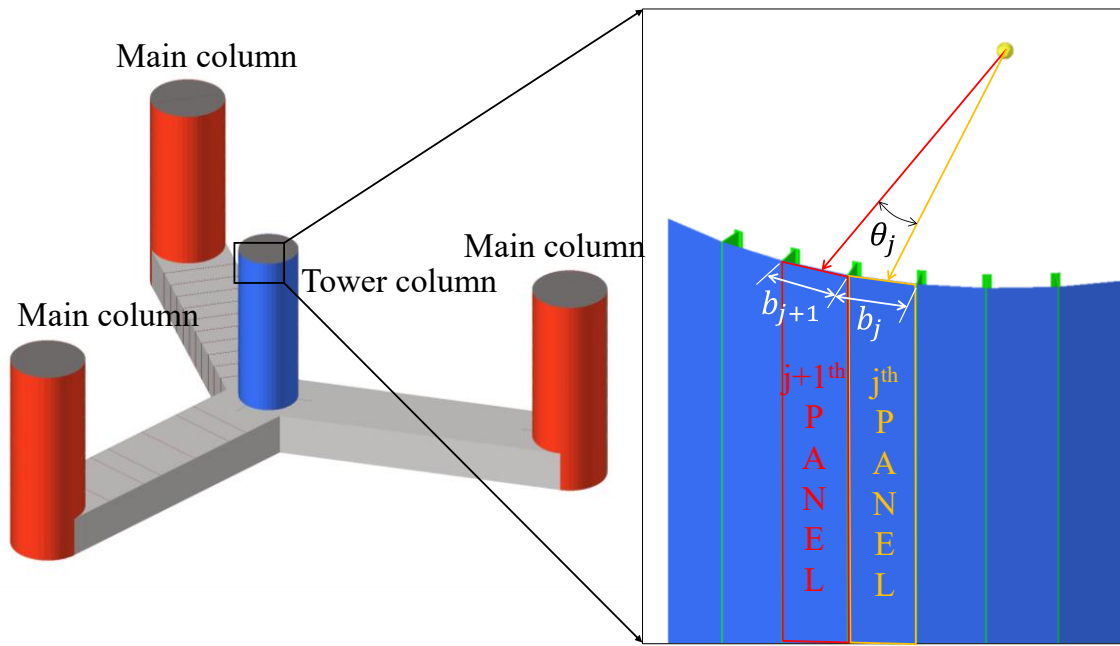
Figure 4.3 Judgement of flat or curved panel

*PROB

The occurrence probability of each load case is defined through the *PROB keyword. In fatigue damage assessment, it indicates the occurrence probability of each FLS load case, and the sum of all probabilities must be 1.0. Also, the number of defined probabilities must match the number of load cases for OpenFAST analysis.

*LIF

The design life (unit: year) is set through the *LIF keyword. The default value for the design life is 20 years.

*FAT

Through the FAT keyword, the hotspot type, ELSET, S-N curve slope and intercept, reference thickness, thickness correction exponent, design fatigue factor, and stress concentration factor are defined in sequence on a single line for fatigue assessment. If fatigue strength is to be evaluated at multiple points, the same definition can be continued on subsequent lines.
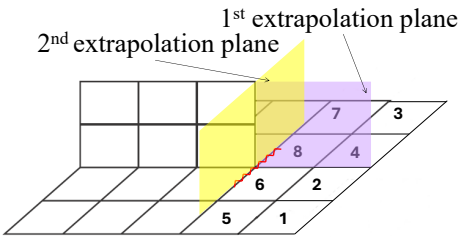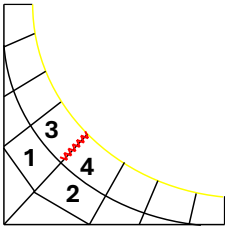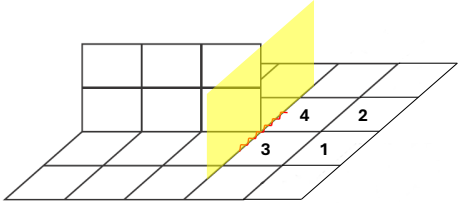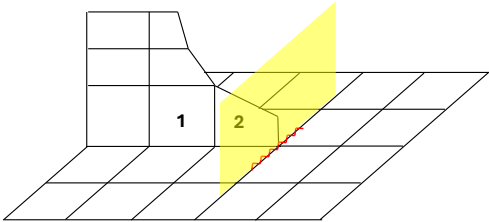
hotspot type	Elset	1st slope	1st intercept	2nd slope	2nd intercept	Ref. thick	Thick. exp.	DFF	SCF
0~4	Text	m_1	$\log a_1$	m_2	$\log a_2$	t_{ref}	k	real num	real num

The hotspot type can be selected from Table 4.2. If hotspot type = 0 is selected, hotspot stress is not used, and the maximum principal stress is extracted at the element centroid. The stress history obtained by multiplying this principal stress by the SCF is used for fatigue analysis. If hotspot type = 1 or higher is selected, hotspot stress is used. For example, when hotspot type = 1, the hotspot stress is calculated by extrapolating principal stresses at the centroid of each element to the 1st extrapolation plane (purple plane), averaging two extrapolated stresses, and finally re-extrapolating the averaged stresses to the 2nd extrapolation plane (yellow plane).

DNV determines the reference points as positions located 0.5t and 1.5t away from the 2nd extrapolation plane. The stresses at these two points are $\sigma_{0.5t}$ and $\sigma_{1.5t}$, and the hotspot stress is determined by extrapolating these two stresses.

If hotspot type = 0, specify the ELSET containing the elements on which fatigue evaluation is to be performed. If the hotspot type is greater than 1, specify ELSETs that sequentially define the number of elements presented in Table 4.2. For example, if hotspot type = 1, eight elements must be sequentially defined in the ELSET.

Table 4.2 Summary of hotspot type IDs

| Hotspot type	Sketch	Remark
0	n/a	Stress at element centroid is used to calculate elemental fatigue damage. If hotspot type is equal to 0, then SCF is multiplied to the element stress. Usually, hotspot type = 0 is used to detect fatigue-sensitive areas (screening analysis).
1		Weld toe crack along base plate with 8 element stresses 1) Extrapolation on to purple plane $\sigma_1 \rightarrow \sigma_2 = \sigma_a$, $\sigma_3 \rightarrow \sigma_4 = \sigma_b$, $\sigma_5 \rightarrow \sigma_6 = \sigma_c$, & $\sigma_7 \rightarrow \sigma_8 = \sigma_d$ 2) Stress averaging $(\sigma_a, \sigma_b) = \bar{\sigma}_A$ & $(\sigma_c, \sigma_d) = \bar{\sigma}_B$ 3) Extrapolation on to yellow plane for hotspot stress $\bar{\sigma}_A \rightarrow \bar{\sigma}_B = \sigma_{hs}$
2		Base plate crack with 4 element stresses 1) Stress averaging $(\sigma_1, \sigma_2) = \bar{\sigma}_A$ & $(\sigma_3, \sigma_4) = \bar{\sigma}_B$ 2) Extrapolation on to yellow plane for hotspot stress $\bar{\sigma}_A \rightarrow \bar{\sigma}_B = \sigma_{hs}$
3		Weld toe crack along base plate with 4 element stresses 1) Stress averaging $(\sigma_1, \sigma_2) = \bar{\sigma}_A$ & $(\sigma_3, \sigma_4) = \bar{\sigma}_B$ 2) Extrapolation on to yellow plane for hotspot stress $\bar{\sigma}_A \rightarrow \bar{\sigma}_B = \sigma_{hs}$
4		Weld toe crack along base plate with 2 element stresses 1) Extrapolation on to yellow plane for hotspot stress $\sigma_1 \rightarrow \sigma_2 = \sigma_{hs}$
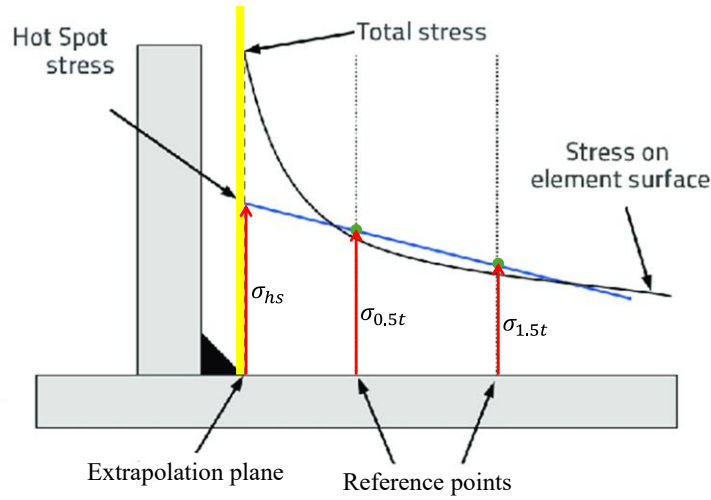
Figure 4.4 Hotspot stress

The slope m and intercept $\log a$ of the S-N curve in Equation (6) are material constants. For an S-N curve with a single slope, only $m_1/\log a_1$ are defined. For an S-N curve with two slopes, both $m_1/\log a_1$ and $m_2/\log a_2$ must be defined. Reference thickness t_{ref} is used to correct the effect of reduced fatigue strength in thick plates, and the default value is 25 mm. The thickness correction exponent k typically ranges from 0.0 to 0.3, with a default value of 0.2.

$$\log N = \log a - m \log \left[\Delta\sigma \times \left(\frac{t}{t_{ref}} \right)^k \right] \qquad (6)$$

Table 4.3 Recommended design fatigue factors

Structural element	Typical structural components	DFF
Internal structure, accessible and not welded directly to the submerged shell.	Frames, bulkheads, stringers in voids, ballast tanks and tower.	1.0
External structure, accessible for inspection and repair in dry and clean conditions.	Tower and substructure above lowest inspection or maintenance draught.	1.0
Internal structure, accessible and welded directly to the submerged shell.	Stiffeners, frames, bulkheads welded to external shell plate below highest loaded draught in voids, ballast tanks and tower.	2.0
External structure not accessible for inspection and repair in dry and clean conditions.	External shell plate below splash zone, bilge keel, fairlead structure.	2.0
Non-accessible areas, areas not planned to be accessible for inspection and repair during operation, and structures with permanent ballast.	Spaces with solid ballast, void spaces, sea chests, small cofferdams, external shell plate in splash zone.	3.0

The design fatigue factor (DFF) is determined by environmental exposure, human accessibility, contamination level, etc.,

and must be input according to the criteria presented in Table 4.3. The cumulative fatigue D_i of the ith sub-DLC is determined by Equation (7). Here, n is the number of stress ranges counted in the ith sub-DLC, and N is the fatigue life corresponding to each stress range. Using the occurrence probability p_i of the ith sub-DLC and DFF, the cumulative fatigue for all sub-DLCs can be obtained from Equation (8). A DFF of 3.0 is applied to ballast spaces filled with iron ore or concrete, void spaces and small cofferdams difficult for human entry, and external shell plates intermittently exposed to seawater in the splash zone. Since the default value of DFF is 1.0, an appropriate DFF should be selected following Table 4.3. The default value of DFF is 1.0.

$$D_i = \sum \frac{n}{N} \tag{7}$$

$$D = DFF \sum_{i=1}^{n} p_i D_i \tag{8}$$

5   Version update log

Version	Update date	Major changes and update
V1.0.0	25th July, 2024	
V1.1.0	6th August, 2024	- Updated to export stress history for selected element set
V1.2.0	4th November, 2024	- Updated stress history derivation algorithm to calculate for all shell element and improve the sorting Abaqus output algorithm.
V2.0.0	20th July, 2025	- Open source code NEMOH is available in "0FRA" module, which can be used alternatively with the commercial code Wamit. - "1BucklingPre" module that is to manage Abaqus INP file to insert NSETs and ELSETs for load mapping and stress output and to automatically generate unstiffened buckling panels is added. - "7Buckling" module is added. The maximum number of unstiffened panels to write the new INP file is limited to two. On private request, buckling evaluation on the full number of panels are available. - The module numbers are re-ordered due to the new modules of "1BucklingPre" and "7Buckling".