FLOWStress v1.0.0

User 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

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.
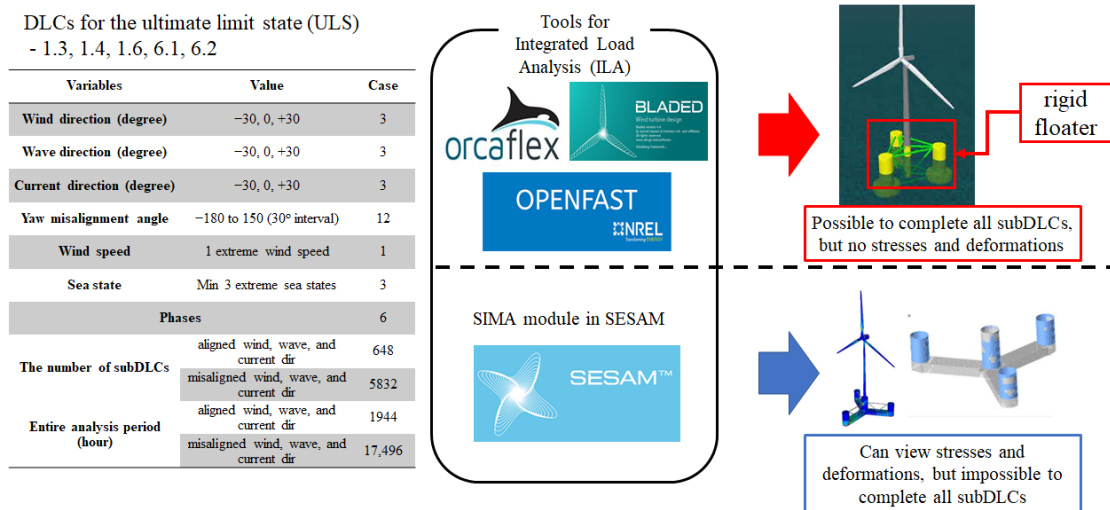

Figure 1.1 Bottlenecks in designing a FOWT unit platform

Table 1.1 The estimated number of subDLCs 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 works in conjunction with Wamit, OpenFAST, and Abaqus/Standard. Ansys/Aqwa does not work with FLOWStress because it produces the combined hydrodynamic force due to radiation and wave excitation. FLOWStress utilizes the results of OpenFAST. ILA S/W such as Bladed, Hawc2, etc. can be linked with FLOWStress as long as they are converted to OpenFAST result format. FLOWStress uses Abaqus/Standard results for FEA. Other FEA S/W interfaces such as MSC/Nastran and Altair/Optistruct will be developed in the future. FLOWStress derives time series stresses of shell elements belong to substructure. The S/W required to run FLOWStress is summarised in Table 1.2 and Figure 1.2.

Table 1.2 S/W for FLOWStress

S/W	Analysis type	Output
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 for unit nodal forces. Typical nodal forces are the mooring tension force on the fairlead, the tower base load, and the acceleration of the FOWT acting on the center of mass (CoM) of the substructure. The time series stresses due to nodal forces are then derived by multiplication with time domain time series data obtained through OpenFAST. FLOWStress summarizes the stress histories due to the nodal forces, the radiation force, and the wave excitation force for each finite element. The summation process of stresses is performed by stress component.

FLOWStress obtains the hydrodynamic forces from the Wamit analysis. The hydrodynamic pressure RAOs 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, the 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.
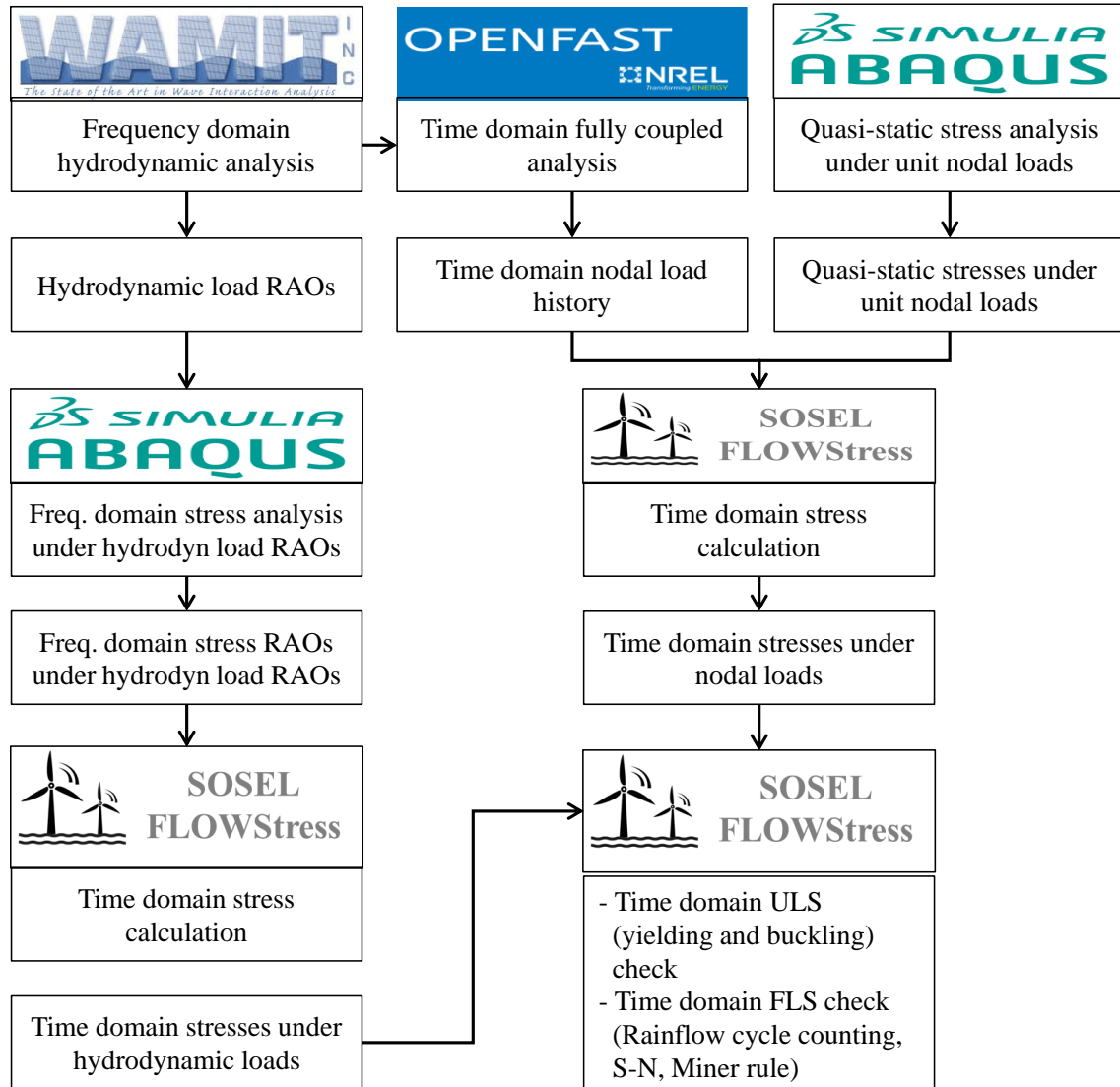

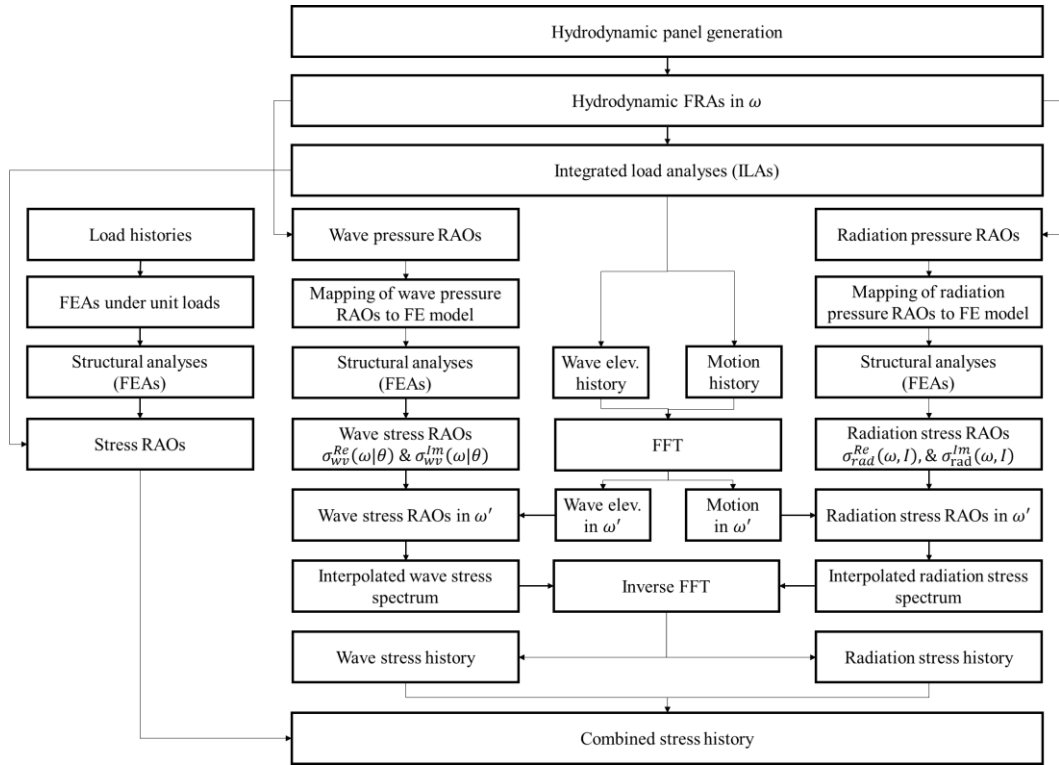Figure 1.2 Key algorithm of FLOWStress from S/W interface perspective

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.
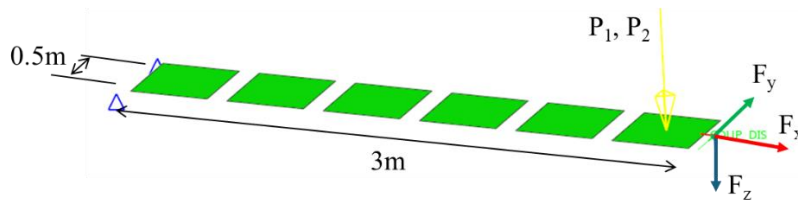
Density	7850 kg/m^3
Thickness	10 mm
E	206 GPa
Nu	0.3
Dload	Pressure 1, 2
Cload	F_x, F_y, F_z

Figure 2.1 Cantilever beam

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

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

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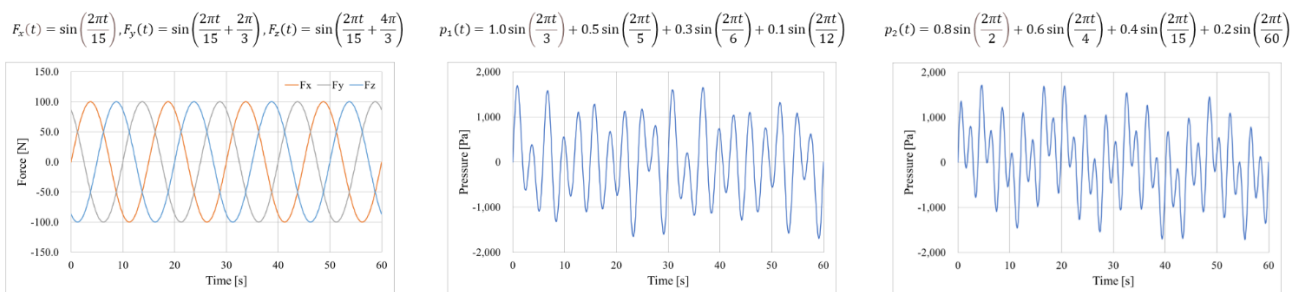

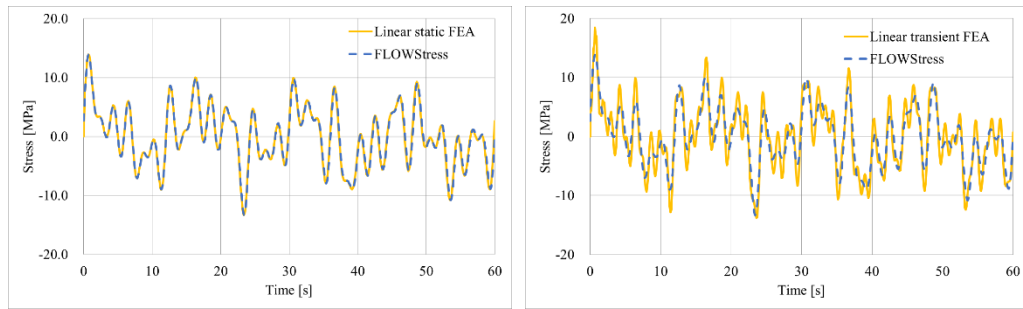Figure 2.2 Point load history and pressure history

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.
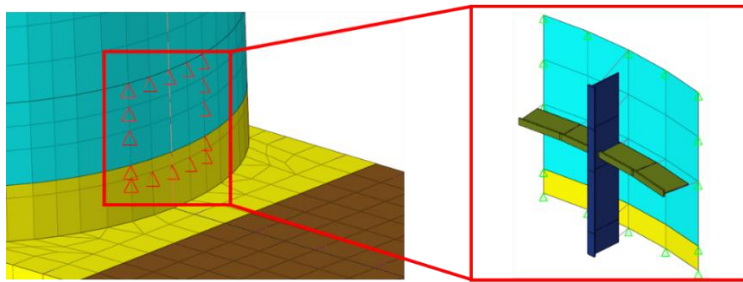

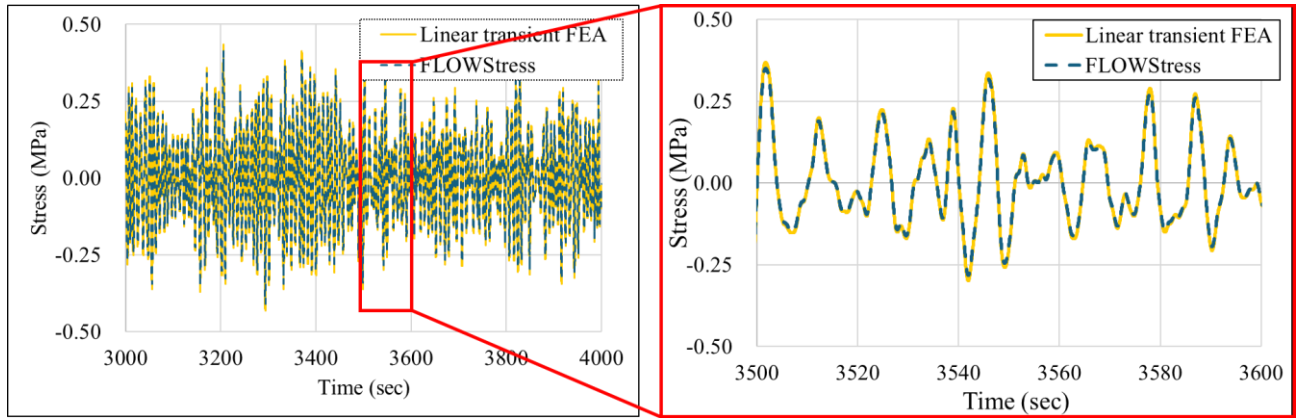| Longi. Space	1 m
Density	7850 kg/m³
Stiffener	210 GPa
E	0.3
Dload	Wave excitation induced pressure
Cload	-

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 administer

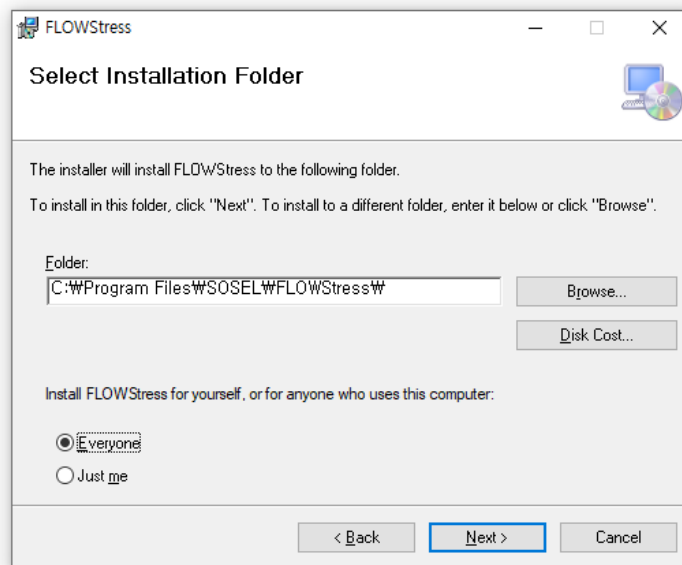- Follow setup wizard with default options

- No license required

Figure 3.1 FLOWStress installer

3.2　Data for FLOWStress

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.

Wamit run

- Convert the Abaqus INP file to a Wamit GDF file using the '0 Aba2Wamit' 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.

Wamit 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

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

OpenFAST 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
>
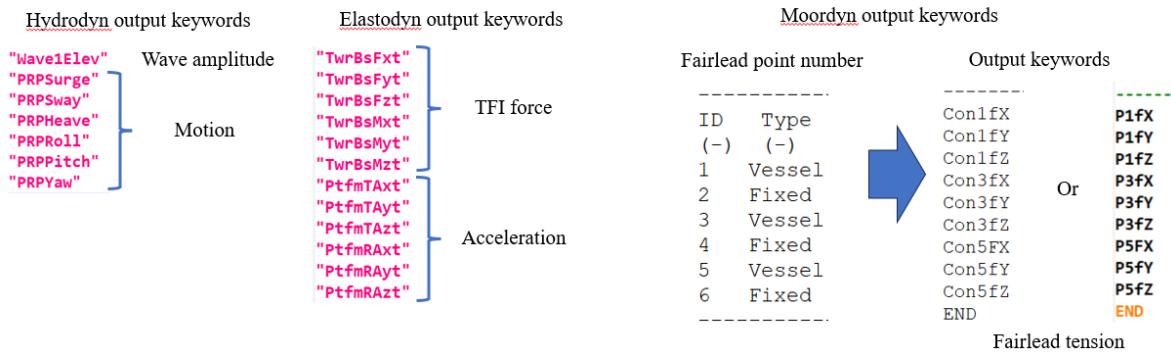> MoorDyn module: 3 DoF tensions at fairleads

Figure 3.2 OpenFAST keywords for output requests

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.

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

- 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.

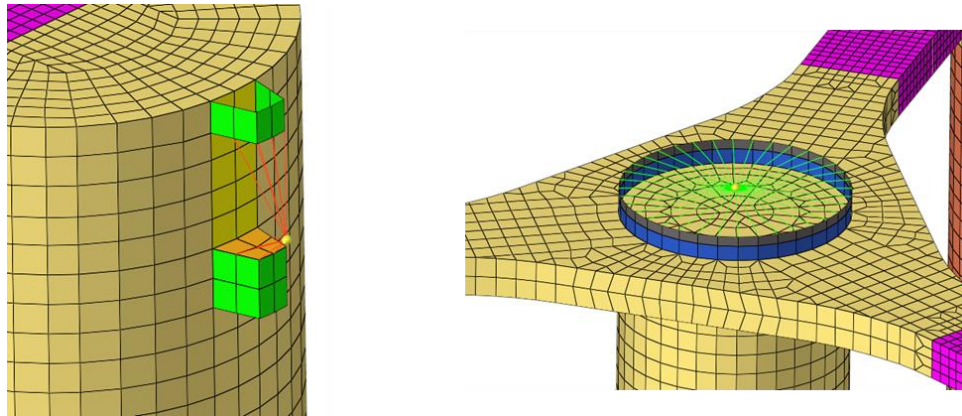| ELSET that acceleration loads should be applied |
| ELSET that plate pressure below the waterline should be applied |
| ELSET that stresses should be printed[3] |

Figure 3.3 Fairlead node (left) and tower base node (right).

JOB file

- The JOB file is the input file for running FLOWStress.

- The keywords used in the JOB file can be found in Chapter 4.

- Keywords must start with a '*' (single asterisk).

- Data that depend on the keyword must be defined on the line following the keyword.

- Lines with '**' (double asterisk) are considered unconditional comments.

---

[1] To distinguish between negative and positive pressure

[2] The difference between the fairlead nodes in FEA and MoorDyn cannot exceed 1m ($\sqrt{\Delta x^2 + \Delta y^2 + \Delta z^2} < 1.0$ 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
0 Aba2Wamit	Converting Abaqus INP file to Wamit GDF and POT files	
1 StaticLoad	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 '1 StaticLoad' is created where a Abaqus INP file is generated.
2 UnitLoad	Generating Abaqus INP file to include unit nodal loads at a single tower base node and fairlead nodes. Other unit loads than below unit loads must be not manually defined. - 6 DoF load components at a tower base - 3 DoF load components at each fairlead - 6 DoF acceleration components at a CoM	A new folder '2 UnitLoad' is created where a Abaqus INP file is generated.
3 NodalStress	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 stress obtained from Abaqus - Nodal load history obtained from OpenFAST	A new folder '3 NodalStress' is created where unit load-induced stress history files with BIN extension are generated.
4 HydroLoad	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.	A new folder '4 HydroLoad' is created where a Abaqus INP files are generated.
5 StressCom	Combining static load-, nodal load-, and hydrodynamic load-induced stress history.	A new folder '5 StressCom' is created where combined stress history files with BIN extension are generated.
6 Fatigue	Evaluating screening and refined fatigue strength analyses.	This module is not included in the GitHub upload version and can be provided upon request.
7 Buckling	Evaluating buckling strength analyses.	This module is not included in the GitHub upload version and can be provided upon request.

3.4   Graphic user interface

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

Loading of stress visualization interface

- The stress combinations are completed through '5 StressComb' 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).

Display zone

- Select ELSET to visualise (Global = overall FE model; Local = ELSETs that output requested in Abaqus INP file)

Stress 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

History 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)
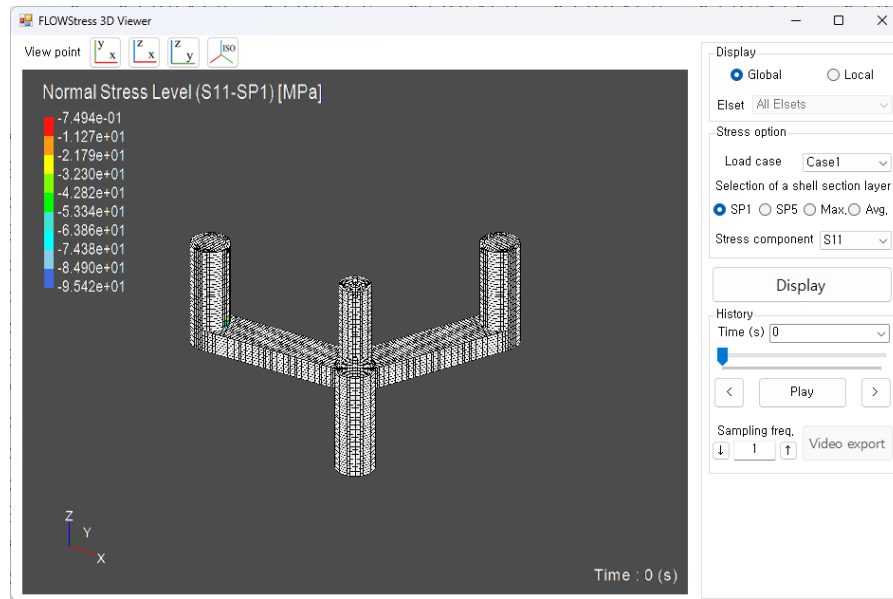
Figure 3.4 Stress history visualization

3.5   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.

3.6   Limitations and errors

Size of RAM

- The number of elements, the number of time data, the number of stress components ($\sigma_x,\ \sigma_y,\ \tau_{xy}$), 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.

- For 10,000 elements and three hours ILA with Δt = 0.1 s, assuming one single float-type data takes 4 bytes, then required memory size is determined as follows: $\frac{3h \times 3,600s}{0.1s} \times 10,000\text{elms} \times 3 \text{ stresses} \times 2 \text{ layers} \times 4 \text{ bytes} = 24.14 \text{ GB}$

- With RAM size of 16GB, FLOWStress can arithmetically process 6,600 elements in a single execution of FLOWStress.

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

Process stop in running "4 HydroLoad" module

- Abaqus/Standard under hydrodynamic load RAOs does not produce the element stresses where they are extremely small (less than 1.0×10^{-10}).

- A very small stress occurs when structural stiffness is unrealistically large. For example, shell element property has very huge thickness, all nodes of a shell element are tied with displacement coupling element (KINEMATIC COUPLING), all nodes of a shell element are completely constrained with boundary conditions, etc.

- When a user assigns these elements into an stress output ELSET, this will cause the FLOWStress processing to stop.

4　　Keywords for JOB file

Table 4.1 Summary of JOB file keywords

Keyword	Description	Example with default value	Example with user-defined value	Module number used
*A2W	- File path for "0 Aba2Wamit" module execution	*A2W C:\ProgramFiles\CommonFiles\FLOWStress\ABA2WAMIT\ABA2WAMIT.exe	Assigned folder when installing FLOWStress	0
*CPU	- The number of CPUs for Abaqus analysis - 1 (default)	*CPU default	*CPU 10	1, 2, 4
*DFLO	- Folder path where FLOWStress generates results	n/a	*DFLO D:\FLOWStress\	1, 2, 3, 4, 5, 6
*DWAM	- Folder path for Wamit in/out files - Wamit input files (GDF, POT, and FRC) must be in this folder before Wamit execution	n/a	*DWAM D:\Wamit\	0, 2, 4, 5
*ELSETA	- Element set for all massive elements	n/a	*ELSETA ELSET-All	1, 2
*ELSETD	- Element set for diffraction elements - Used for "0 Aba2Wamit" module execution	n/a	*ELSETD ELSET-SideShell	1, 4
*ELSETS	- Element set for stress recovery elements (element set that a user requested to print stress results)	n/a	*ELSETS ELSET-Stress	1, 2, 3, 4, 5, 6
*EXP	- Pressure mapping exponent - 2.0 (default)	*EXP default	*EXP 8.0	4
*EXT	- Thickness exponent in S-N curve - ELSET, thickness exponent - Increase dataline as may as user defined	n/a	*EXT ELSETS, 0.2 ELSETB, 0.25	6
*FAB0	- File path for hydrodynamic model in Abaqus format - Abaqus model should contain only diffraction and non-diffraction side shell elements	n/a	*FAB0 D:\Example\WAMIT\Wamit.inp	0
*FAB1	- File path for Abaqus INP file - Abaqus reference model file to which FLOWStress applies unit nodal loads and hydrodynamic load RAOS	n/a	*FAB1 D:\Example\Abaqus\Abaqus.inp	1, 2, 3, 4, 5, 6
*FHYD	- File path for 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	*FOUT D:\OpenFAST\HydroDyn_Case01.dat D:\OpenFAST\HydroDyn_Case02.dat D:\OpenFAST\HydroDyn_Case03.dat	5
*FMOR	- File path for OpenFAST MoorDyn module DAT file	n/a	*FMOR D:\OpenFAST\MoorDyn.dat	2
*FOUT	- File path for OpenFAST OUT files	n/a	*FOUT D:\OpenFAST\Case01.out D:\OpenFAST\Case02.out D:\OpenFAST\Case03.out	3, 5
*GRAV	- Gravity in m/s^2 - Default is 9.80655 m/s^2	*GRAV default	*GRAV 9.81	0
*HSP	- Extrapolation method to determine hotspot stress - 4 types of hotspots - type number, EID, EID…up to eight element	n/a	*HSP 1, 10, 12, 13, 24, 25, 26, 27, 28 2, 11, 12, 13, 15 3, 11, 12, 13, 15 4, 55, 66	6
*LIF	- Design life in year - 20.0 (default)	*LIF default	*LIF 20	6
*NSET1	- Node set for fairlead nodes	n/a	*NSET1 NSET-FCS	2
*NSET2	- Node set for a tower base node	n/a	*NSET2 NSET-TBase	2
*NSET3	- Node set for other nodes where external load history can be provided - Not available in GitHub upload version	n/a	n/a	n/a
*OFO	- ODB field output - S: stress, E: strain, U: disp, RF: reaction force - Default includes S, E, U, and RF	*OFO default	*OFO S, E, U, RF	1, 2, 4
*PROB	- Probability of each load case - The number of dataline should be same as the load cases in OpenFAST - Sum of probability should be 1.0	n/a	*PROB 0.3 0.5 0.2	6
*PTA	- Pressure mapping tolerance angle in degree to search elements in a same plane - Must be ranged 0.0^o < PTA < 10° - 2.0 (default)	*PTA default	*PTA 3.0	4
*SCI	- Stress cut-in time (s) from OpenFAST - 0.0 (default)	*SCI default	*SCI 100.0	3, 5
*SFF	- Safety factor of fatigue (design fatigue factor) - Increase dataline as may as user defined	n/a	*SFF ELSETS, 5.0 ELSETB, 10.0	6
*SNC	- S-N curve - ELSET, m1, log(a1) , m2, log(a2) - m: slope & log(a): intercept - Increase dataline as may as user defined	n/a	*SNC ELSETS, 3.0, 12.164, 5.0, 15.604 ELSETB, 3.0, 12.164	6
*SRO	- Stress recovery option according to load types - WEX: wave excitation, RAD: radiation, PL: nodal load, SL: static load - Default includes WEX, RAD, PL, and SL	*SRO dafault	*SRO WEX, RAD, SL	5
*STIM	- Δt (sampling time) - Default Δt is same as that in OpenFAST	*STIM default	n/a	3, 5
*TYP	- Fatigue assessment type - SCR (screening) or REF (refined)	n/a	*TYP SCR	6
*UNA0	- Abaqus FE model unit system - Used to convert Abaqus INP to Wamit GDF - m (default) or mm	*UNA0 default	*UNA0 mm	0
*UNA1	- Abaqus FE model unit system - m or mm (default)	*UNA1 default	*UNA1 2	1, 2, 3, 4, 5
*UNW	- Wamit model unit system - m (default) or mm	*WUN default	*WUN mm	0, 2, 4
*WDEP	- Water depth in m - Default is infinite depth	*WDEP default	*WDEP 200	0
*WFR	- Wave frequency - Min, Max, Increment	n/a	*WFR 0.01, 3.5, 0.05	0

*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.

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

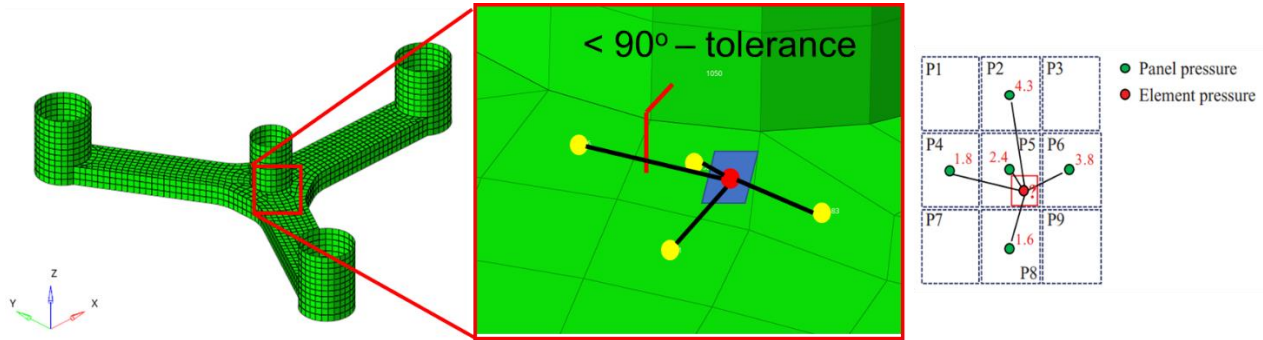$$F_j = (\sum_{i=1}^{n} p_i \times w_i^{\alpha})/ \sum_{i=1}^{n} w_i^{\alpha} \tag{2}$$

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 correspond 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.

5   Acknowlegment