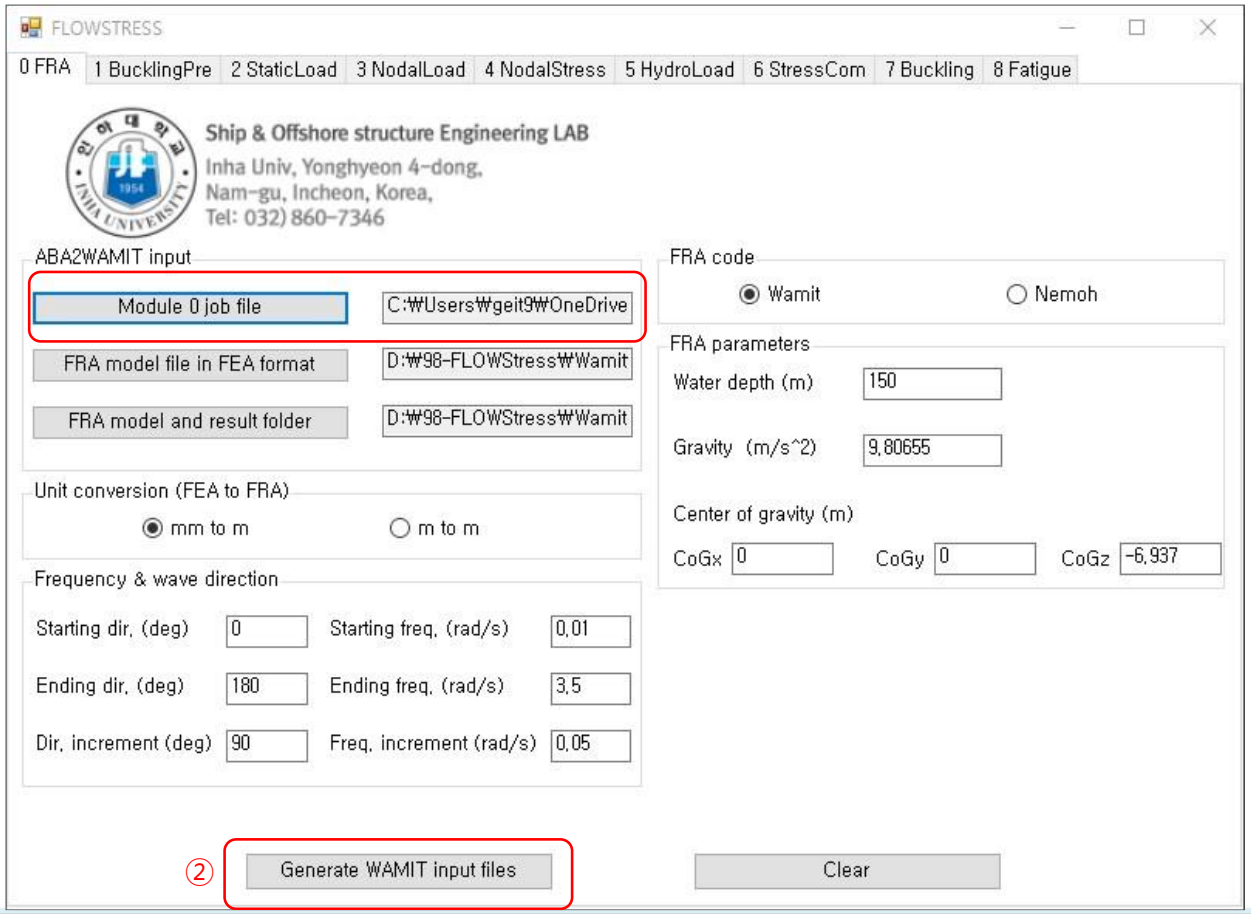
Tutorial Manual for FLOWStress 2.0.0

October 27, 2025

Edited by KIM, Gitae and CHOUNG, Joonmo

0FRA using Wamit (Frequency Response Analysis)

✓ How to prepare Wamit input files with JOB file?

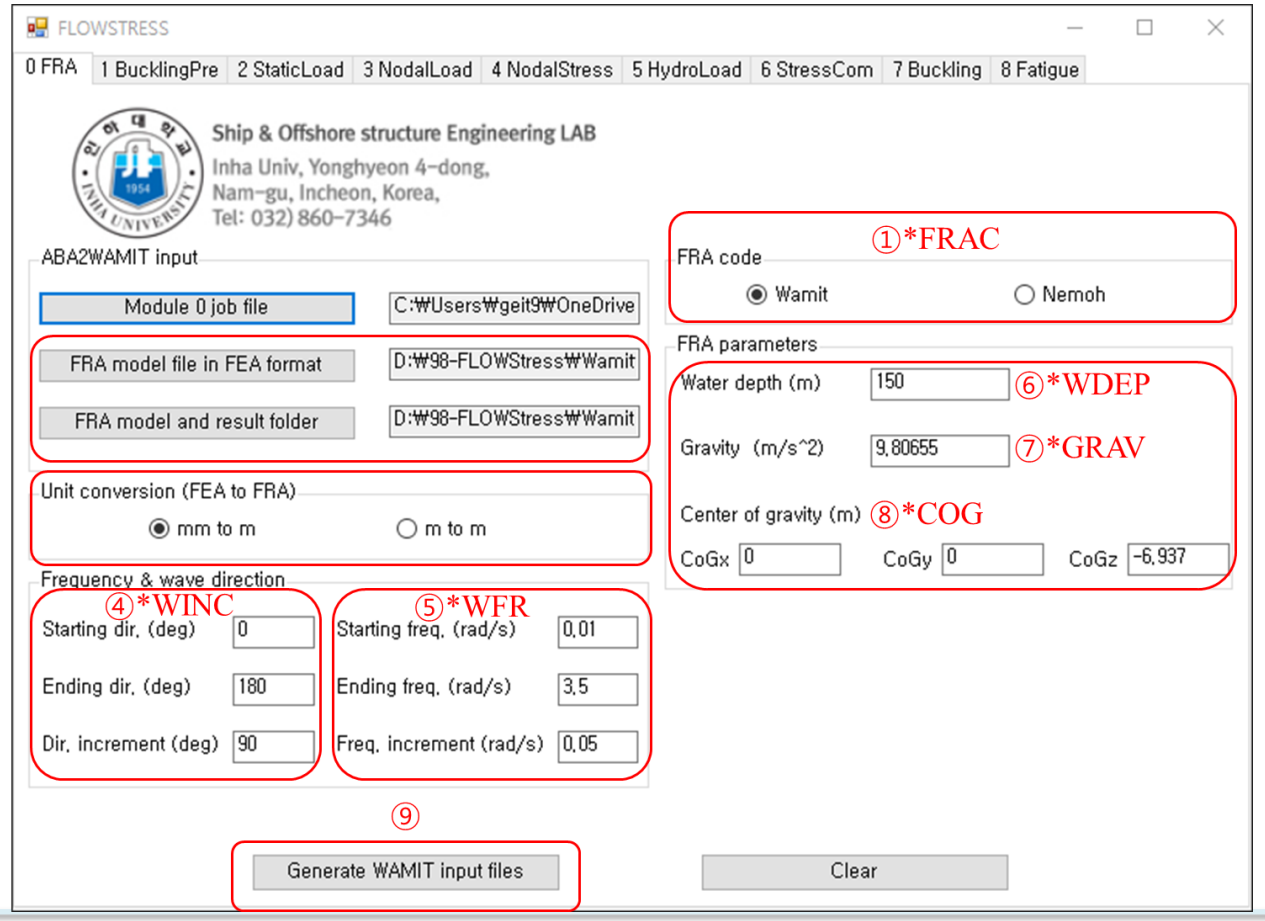


① Select JOB file
② Generate Wamit input files

*FRAC (Frequency response analysis code)
1,
*FAB0 (FEA model file for FRA transform)
D:\98-FLOWStress\Wamit\FRA.inp
*UNA0 (Unit system in FEA model for FRA transform)
,
*WINC (Wave incident angle)
0, 180, 90,
*WFR (Wave frequency range)
0.01, 3.5, 0.05,
*WDEP (Water depth)
150,
*GRAV (Gravity acceleration)
,
*COG (Center of gravity)
0, 0, -6.937,
*FA2W (path for ABA2WAMIT.exe)
C:\Program Files\Common Files\FLOWStress\ABA2WAMIT\ABA2WAMIT.exe

0FRA using Wamit (Frequency Response Analysis)

✓ How to prepare Wamit input files manually?



① Select FRA code 'Wamit'

②-1 Select a model file (PreWamit file)

*PreWamit file should be in Abaqus format

②-2 Select a folder where the transformed Wamit input and result files are located

③ Select unit conversion from PreWamit file to Wamit input file

*FLOWStress allows only SI unit for Wamit analysis

④ Assign angles of incident waves

*Must be in degree unit

⑤ Assign wave frequencies

*Must be in rad/s unit

⑥ Assign water depth

*Must be in meter unit

⑦ Assign gravity

*Must be in m/s^2

⑧ Assign center of gravity from origin

*Must be in m

*Origin are usually at CoGx, CoGy, and MSL

⑨ Generates Wamit input files based on the settings

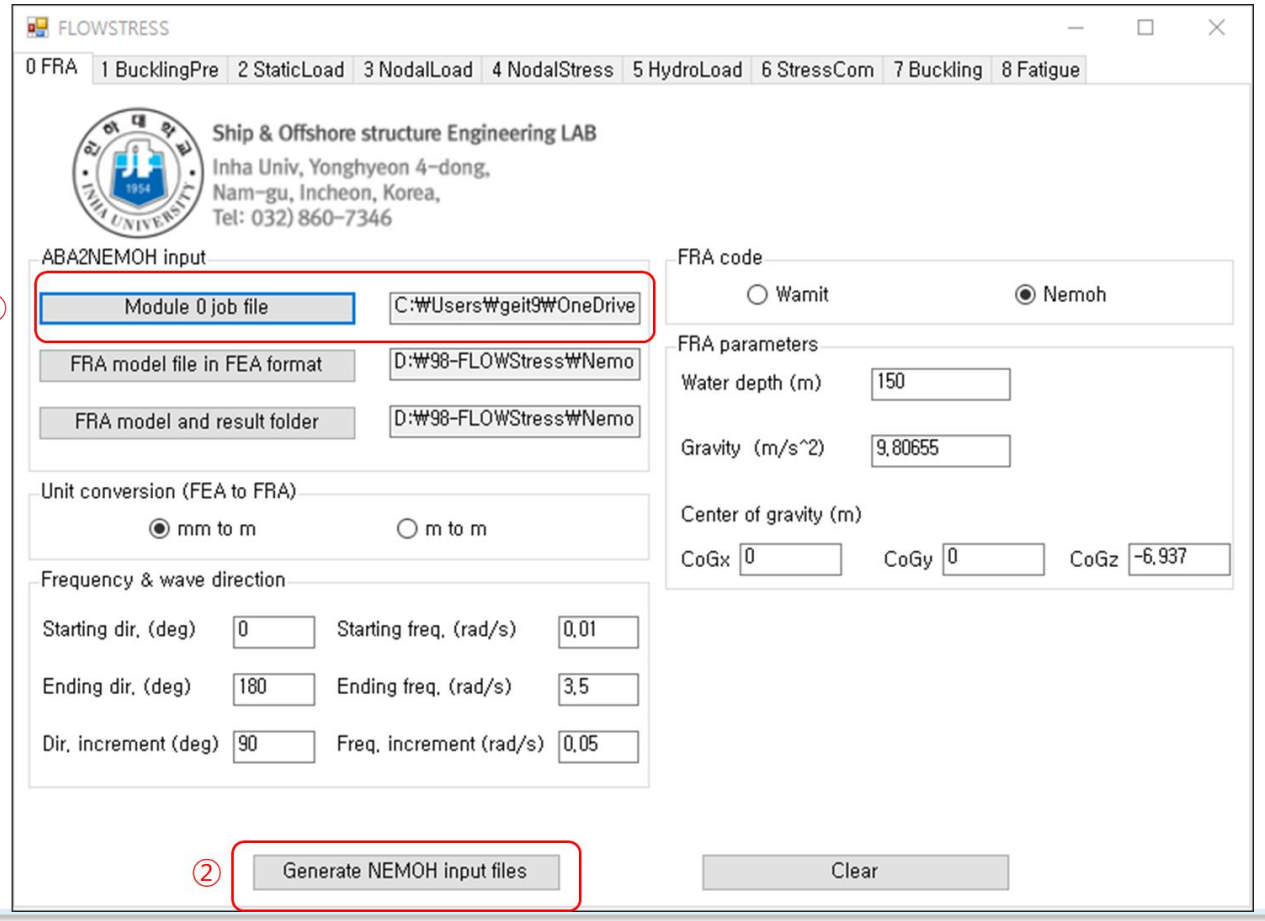
0FRA using Wamit (Frequency Response Analysis)

✓ How to execute Wamit?

1) Manually prepare Wamit analysis setup file (CFG) and mass information file (FRC), respectively.
2) Check if FLOWStress successfully generated panel file (GDF) and wave file (POT), respectively.
3) Manually prepare WAM file which collects four files (CFG, FRC, GDF, and POT).
4) Place five files (WAM, CFG, FRC, GDF, and POT) in the folder where Wamit execution files are located.
5) Execute Wamit.EXE on DOS command

0FRA using Nemoh (Frequency Response Analysis)

✓ How to prepare Nemoh input files with JOB file?

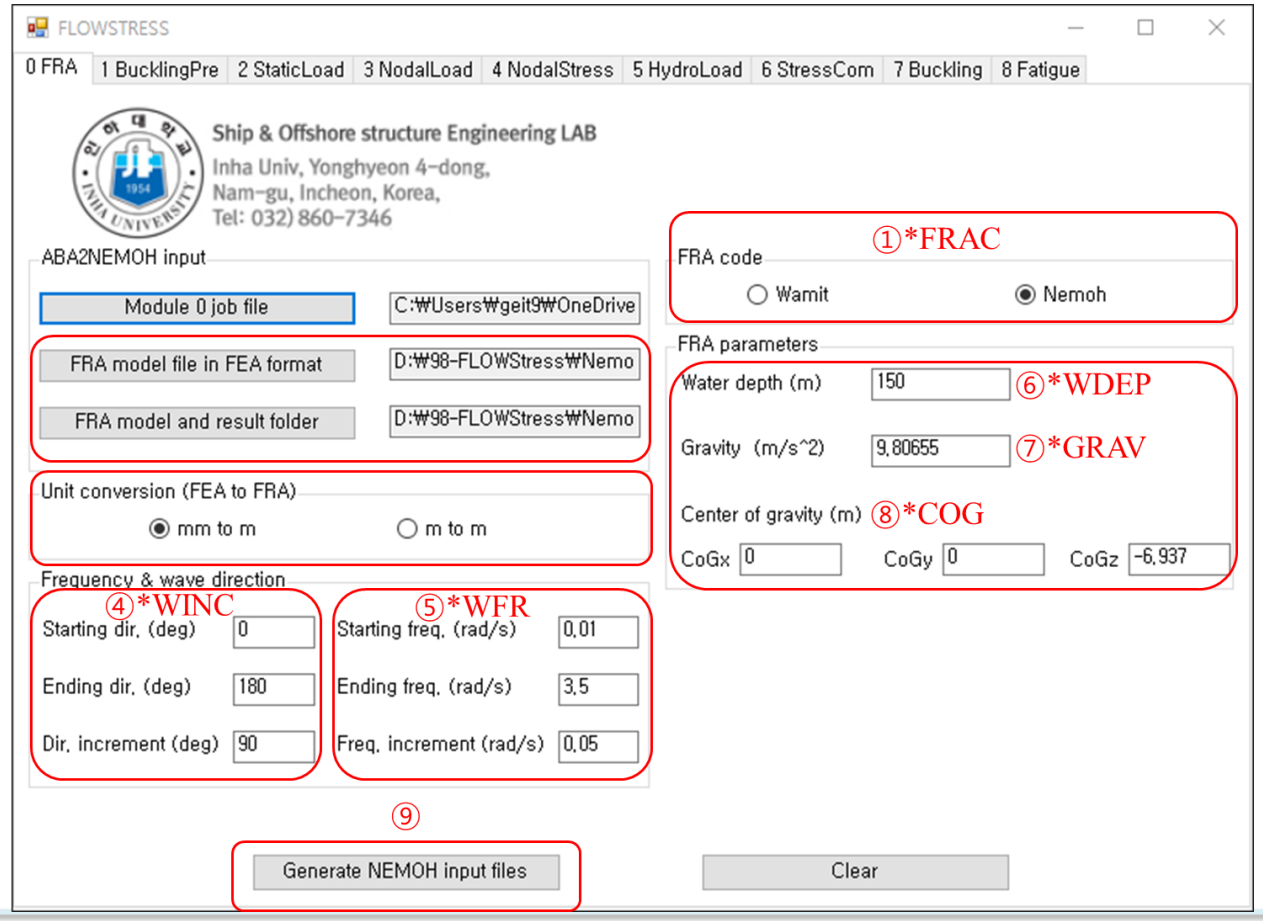


① Select JOB file
② Generate Nemoh input files

```
*FRAC (Frequency response analysis code)
2,
*FAB0 (FEA model file for FRA transform)
D:\98-FLOWStress\Nemoh\FRA.inp
*UNA0 (Unit system in FEA model for FRA transform)
,
*WINC (Wave incident angle)
0, 180, 90,
*WFR (Wave frequency range)
0.01, 3.5, 0.05,
*WDEP (Water depth)
150,
*GRAV (Gravity acceleration)
,
*COG (Center of gravity)
0, 0, -6.937,
*FA2N (path for ABA2NEMOH.exe)
C:\Program Files\Common Files\FLOWStress\ ABA2NEMOH\ABA2NEMOH.exe
```

0FRA using Nemoh (Frequency Response Analysis)

✓ How to prepare Nemoh input files manually?



① Select FRA code 'Nemoh'

②-1 Select a model file (PreNemoh file)

*PreNemoh file should be in Abaqus format

②-2 Select a folder where the transformed Nemoh input and result files are located

③ Select unit conversion from PreNemoh file to Wamit input file

*FLOWStress allows only SI unit for Nemoh analysis

④ Assign angles of incident waves

*Must be in degree unit

⑤ Assign wave frequencies

*Must be in rad/s unit

⑥ Assign water depth

*Must be in meter unit

⑦ Assign gravity

*Must be in m/s^2

⑧ Assign center of gravity from origin

*Must be in m

*Origin are usually at CoGx, CoGy, and MSL

⑨ Generates Nemoh input files based on the settings

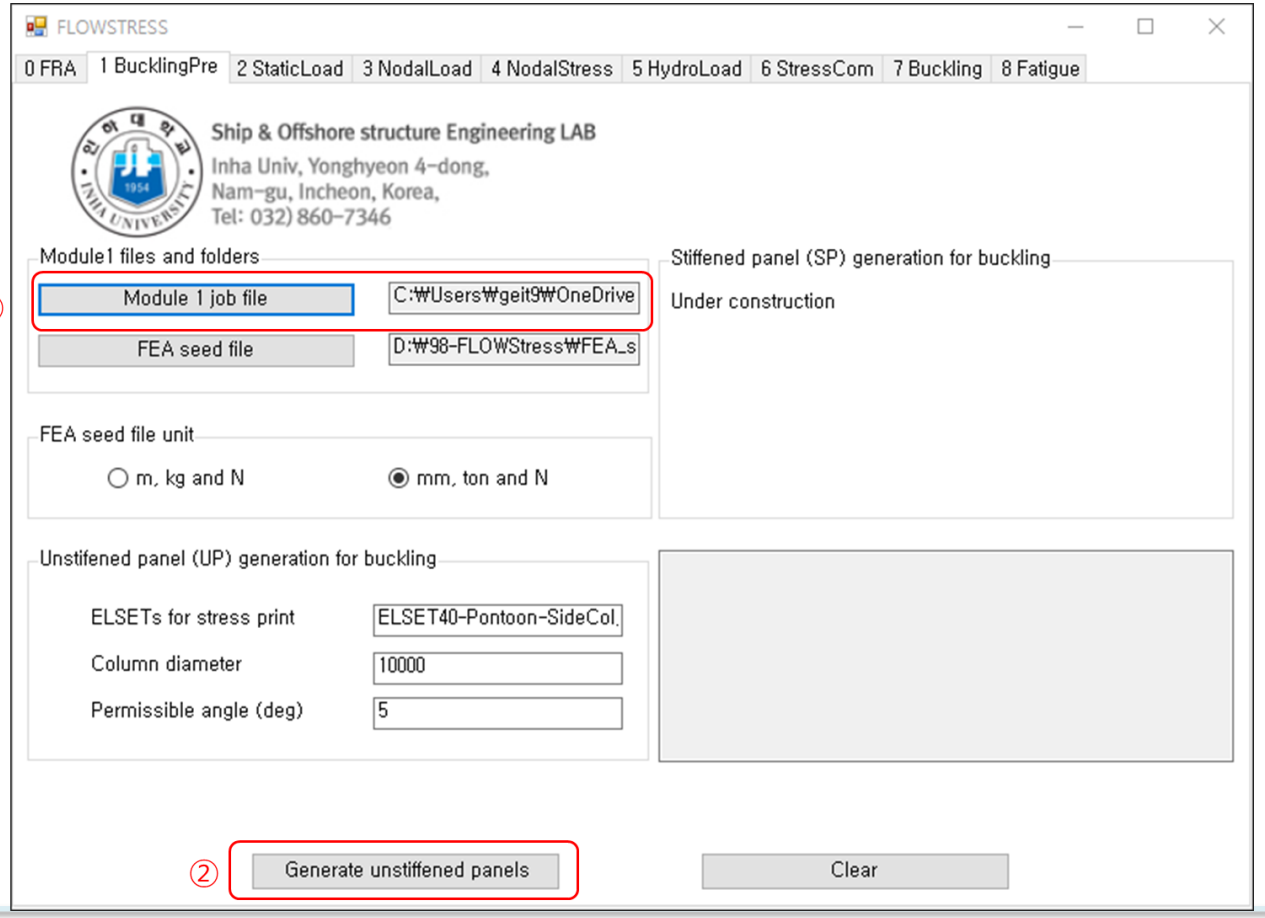
0FRA using Nemoh (Frequency Response Analysis)

✓ How to execute Nemoh?

1) Check if FLOWStress successfully generated setup file (Input_solver.TXT), mass information file (Mesh.CAL), panel file (FRA.DAT), and wave file (Nemoh.CAL), respectively.
2) Place four files (Input_solver.TXT, Mesh.CAL, FRA.DAT, and Nemoh.CAL) in the folder where Nemoh execution files are located.
3) Execute preProc.EXE on DOS command
4) Execute solver.EXE on DOS command

1BucklingPre (Buckling Preprocessor)

✓ How to prepare BucklingPre with JOB file?

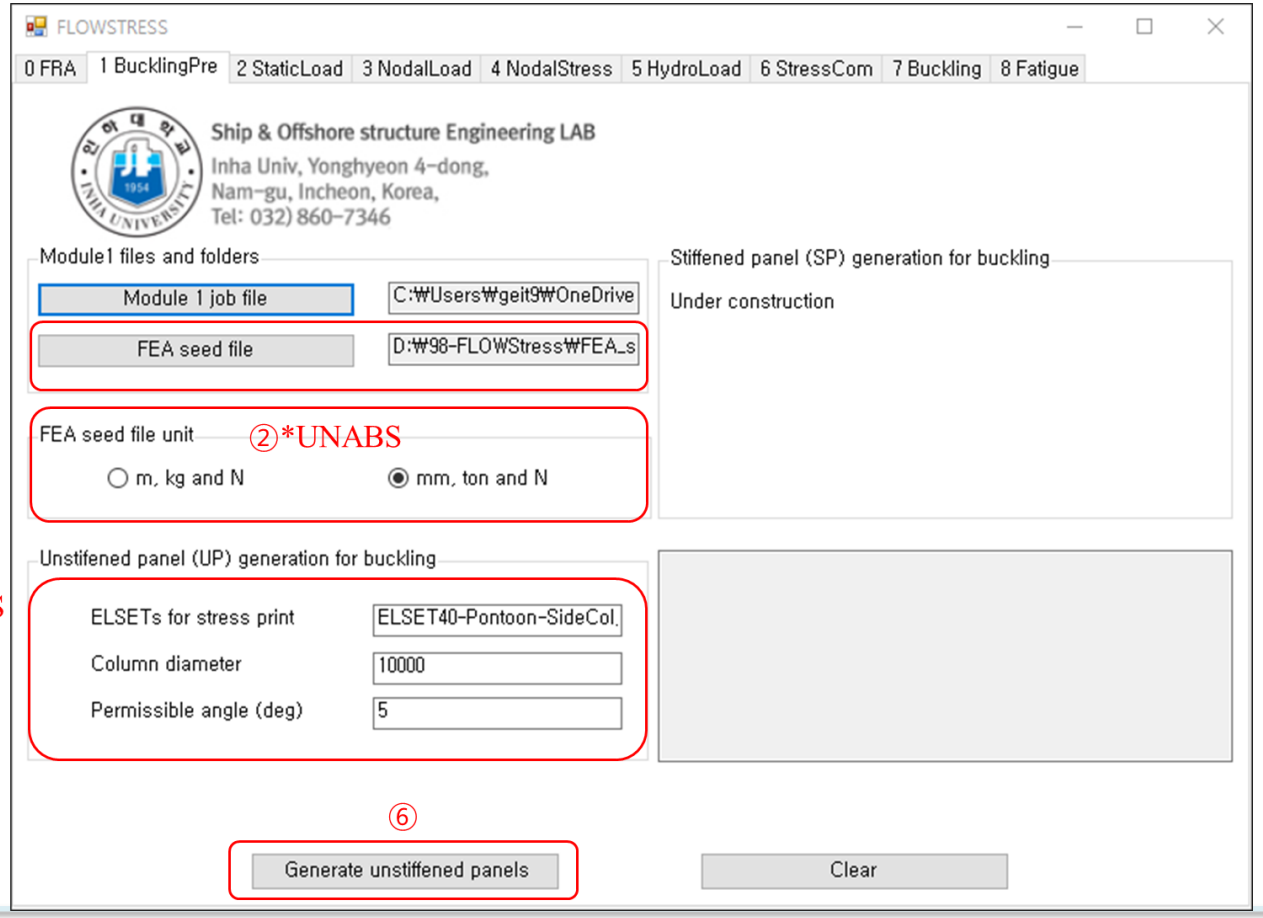


① Select JOB file
② Generate unstiffened panel

*FABS (FEA seed file)
D:\98-FLOWStress\FEA_seed\FEA_seed.inp
*UNABS (Unit system of FEA seed file)
2,
*ELSETS (Element set containing elements with stress printed)
ELSET40-Pontoon-SideCol,
ELSET40-Pontoon-CenCol,
*CDIA (Column diameter)
10000,
*PANG (Buckling panel tolerance angle in degrees)
5,

1BucklingPre (Buckling Preprocessor)

✓ How to prepare BucklingPre manually?



① Select FEA seed file
② Select unit system of FEA seed model
③ Assign element set with stress printed
④ Assign column diameter
*Must follow unit system defined in *UNABS
⑤ Assign buckling panel tolerance angle
*Must be in degree
⑥ Generate unstiffened panels based on the settings

1BucklingPre (Buckling Preprocessor)

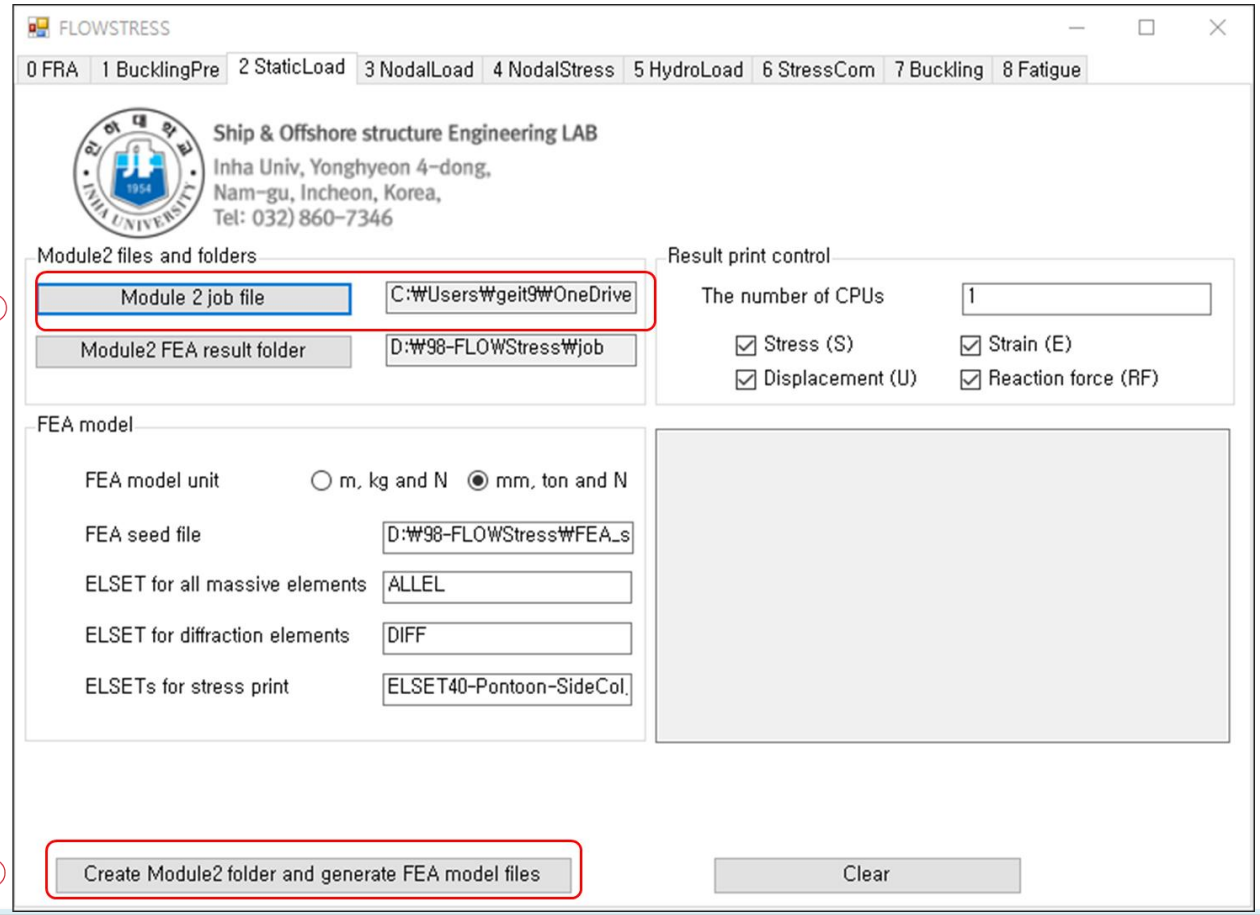
✓ How to prepare and check BucklingPre?

1) Prepare FEA seed file manually which should include model data only such as nodes, elements, section and material properties, and node and element sets (NSETs and ELSETs), respectively. History data of load steps and load cases should not be in FEA seed file.

2) Check if FLOWStress successfully created a panel information file (Panel_info.CSV) and a new FEA model file (FEA_seed_BucklingPre.INP) in which unstiffened panels are generated automatically in element sets (UP_00001 and UP_00002).
Free-license version supports maximum two unstiffened panels. In order to obtain all unstiffened panels, contact the developer (Prof Joonmo CHOUNG, jmchoung@inha.ac.kr).

2StaticLoad (Static load FEA)

✓ How to prepare StaticLoad with JOB file?

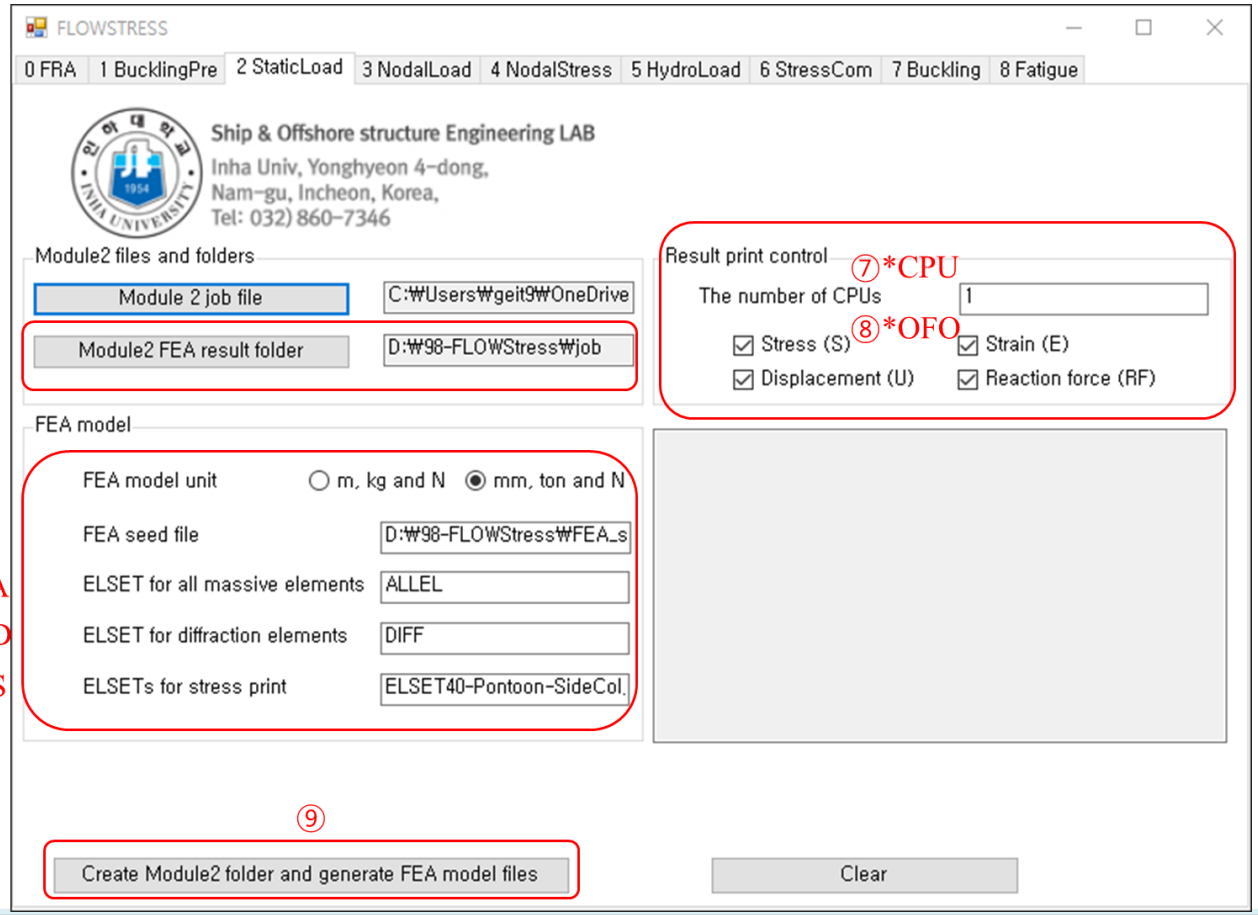


① Select JOB file
② Create Module2 folder and generate FEA model files

*DFLO (Folder path where FLOWStress generates results)
D:\98-FLOWStress\job
*UNABS (Unit system of FEA seed file)
2,
*FABS (FEA seed file)
D:\98-FLOWStress\FEA_seed\FEA_seed.inp
*ELSETA (Element set containing all massive elements which contribute the mass of structures)
ALLEL,
*ELSETD (Element set containing diffraction elements)
DIFF,
*ELSETS (Element set containing elements with stress printed)
ELSET40-Pontoon-SideCol,
ELSET40-Pontoon-CenCol,
*CPU (The number of CPUs for FEA analysis)
,
*OFO (ODB field output option)
S, E, U, RF

2StaticLoad (Static load FEA)

✓ How to prepare StaticLoad manually?



① Select Module2 FEA result folder
② Select unit system of FEA seed model
③ Select FEA seed file
④ Assign element set containing all massive elements which contribute the mass of structures
⑤ Assign element set containing diffraction elements
⑥ Assign element set containing elements with stress printed
⑦ Assign the number of CPUs for FEA analysis
⑧ Assign ODB field output option
⑨ Create Module2 folder and generate FEA model files

2StaticLoad (Static load FEA)

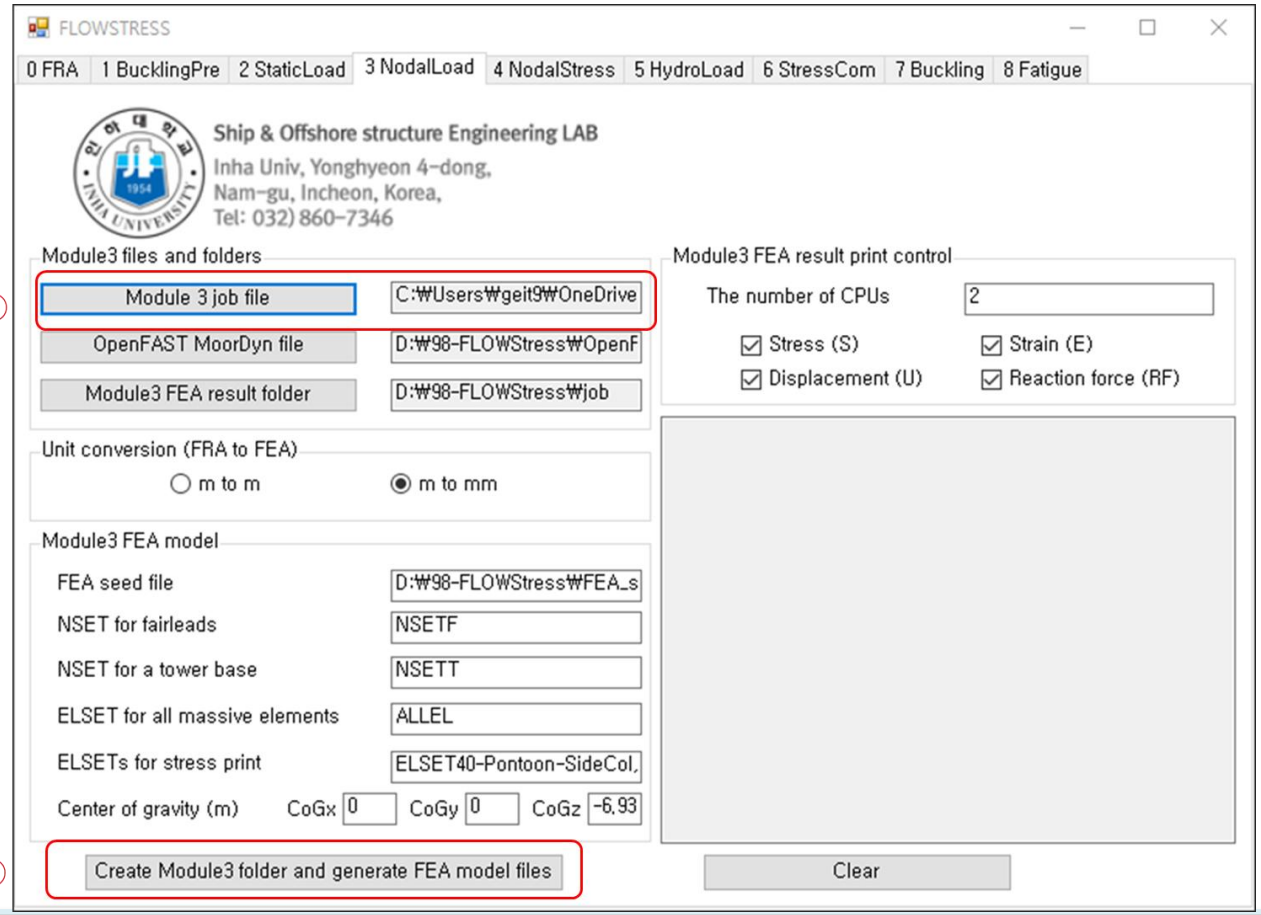
✓ How to execute StaticLoad?

1) Check if FLOWStress successfully created Module2 folder (\2_StaticLoad), load file(Static_load.INP), and batch file (ABAQUS_RUN.BAT).
2) Execute ABAQUS_RUN.BAT
3) Check if the FEA result file (FEA_seed.DAT) is successfully generated. A keyword 'ERROR' should not exist in FEA_seed.DAT file.

3NodalLoad (FEA under unit concentrated nodal load)

✓ How to prepare NodalLoad with JOB file?

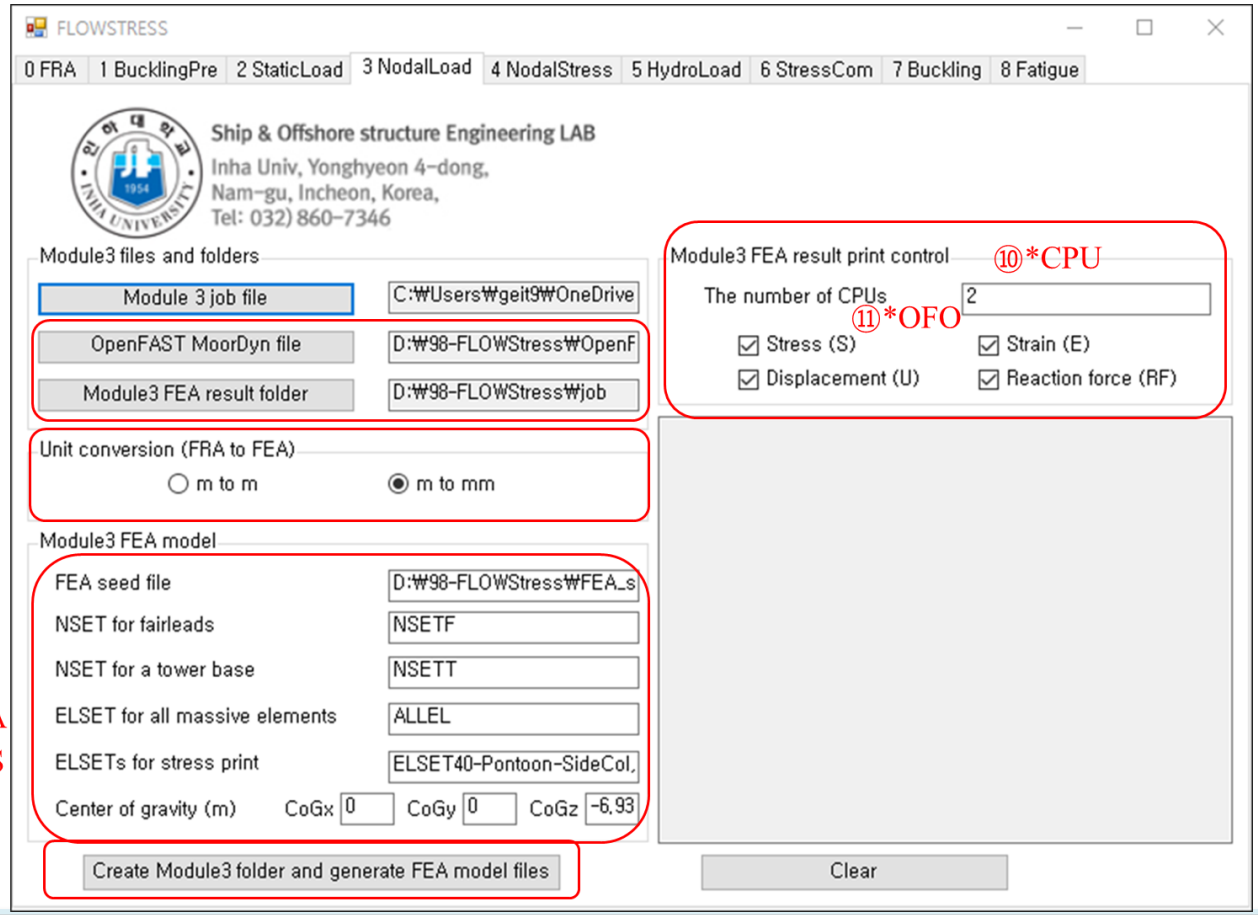
① Select JOB file
② Create Module3 folder and generate FEA model files




```
*FMOR (OpenFAST MoorDyn DAT file)
D:\98-FLOWStress\OpenFAST\IEA-15-240-RWT-UMaineSemi_MoorDyn.dat
*FABS (FEA seed file)
D:\98-FLOWStress\FEA_seed\FEA_seed.inp
*DFLO (Folder path where FLOWStress generates results)
D:\98-FLOWStress\job
*UNABS (Unit system of FEA seed file)
2,
*NSET1 (Node set cobtaining fairlead nodes)
NSETF,
*NSET2 (Node set containing a tower base node)
NSETT,
*ELSETA (Element set containing all massive elements which contribute the mass of
structures)
ALLEL,
*ELSETS (Element set containing elements with stress printed)
ELSET40-Pontoon-SideCol,
ELSET40-Pontoon-CenCol,
*COG (Center of gravity)
0, 0, -6.937,
*CPU (The number of CPUs for FEA analysis)
2,
*OFO (ODB field output option)
S, E, RF, U,
```

3NodalLoad (FEA under unit concentrated nodal load)

✓ How to prepare NodalLoad manually?



① Select OpenFAST MoorDyn DAT file
② Select Module3 FEA result folder
③ Select unit system of FEA seed model
④ Select FEA seed file
⑤ Assign node set cobtaining fairlead nodes
*MoorDyn fairlead nodes must match
*Must be pre-defined in FEA seed file
⑥ Assign node set containing a tower base node
*Must be pre-defined in FEA seed file
⑦ Assign element set containing all massive elements which contribute the mass of structures
⑧ Assign element set containing elements with stress printed
⑨ Assign center of gravity from origin
⑩ Assign the number of CPUs for FEA analysis
⑪ Assign ODB field output option
⑫ Create Module3 folder and generate FEA model files

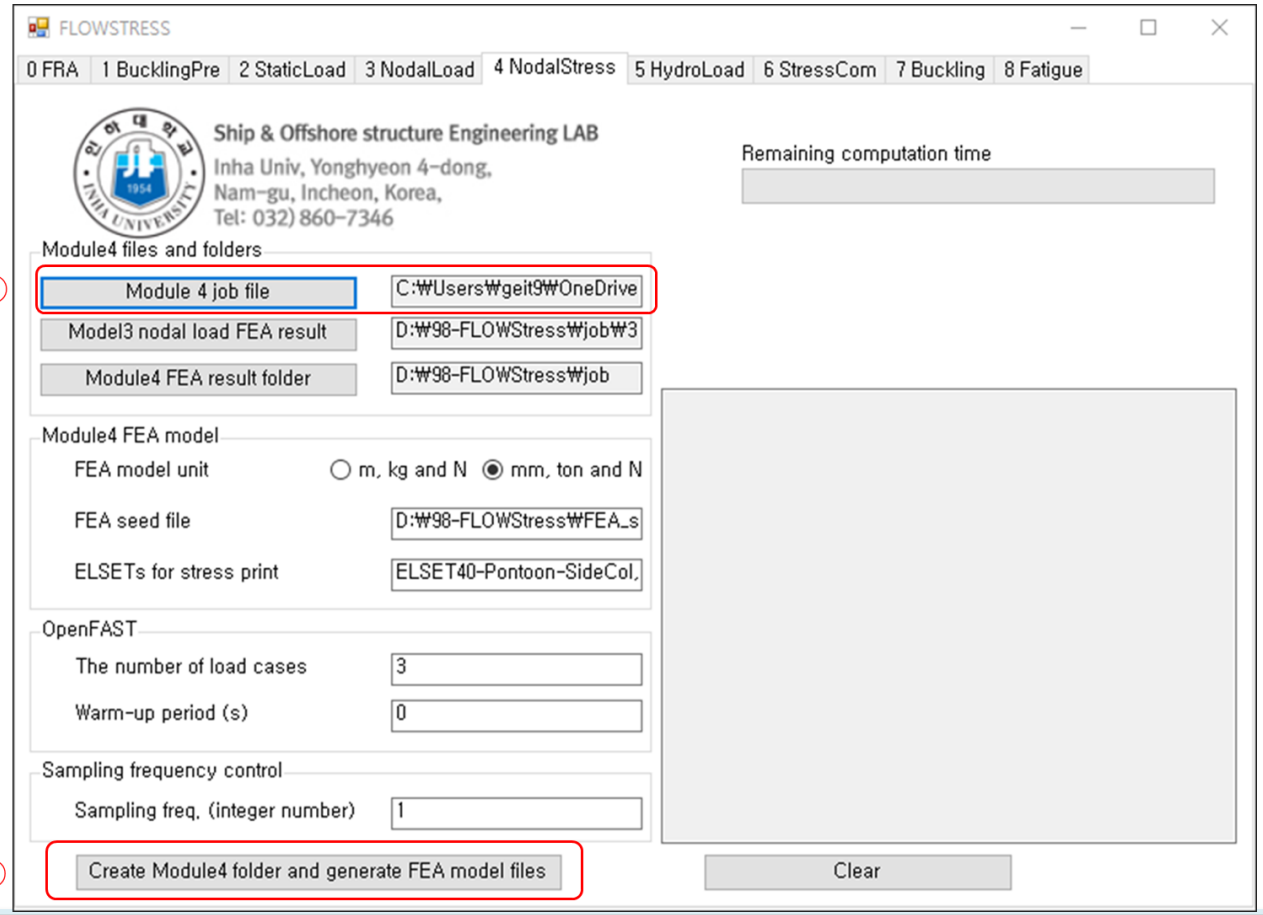
3NodalLoad (FEA under unit concentrated nodal load)

✓ How to execute NodalLoad?

1) Check if FLOWStress successfully created Module3 folder(\3_NodalLoad), load file(Nodal_load.INP) and batch file (ABAQUS_RUN.BAT).
2) Execute ABAQUS_RUN.BAT
3) Check if the FEA seed file.DAT is valid after FEA analysis

4NodalStress (Stress history under concentrated nodal load)

✓ How to prepare NodalStress with JOB file?

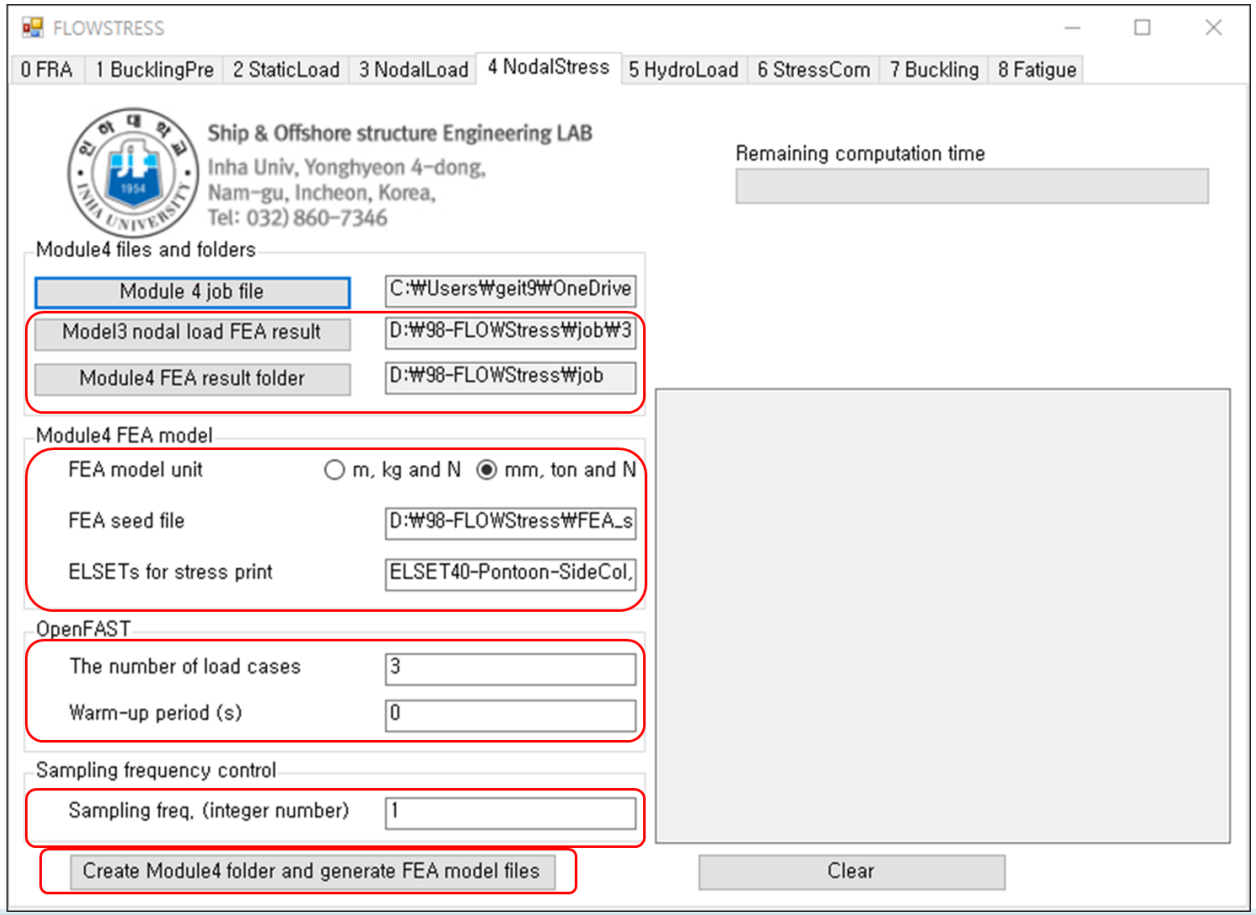


① Select JOB file
② Create Module4 folder and generate FEA model files

*DFLO (Folder path where FLOWStress generates results)
D:\98-FLOWStress\job
*UNABS (Unit system of FEA seed file)
2,
*FABS (FEA seed file)
D:\98-FLOWStress\FEA_seed\FEA_seed.inp
*ELSETS (Element set containing elements with stress printed)
ELSET40-Pontoon-SideCol,
ELSET40-Pontoon-CenCol,
*FOO (OpenFAST OUT file(s))
D:\98-FLOWStress\OpenFAST\IEA-15-240-RWT-UMaineSemi1.out,
D:\98-FLOWStress\OpenFAST\IEA-15-240-RWT-UMaineSemi2.out,
D:\98-FLOWStress\OpenFAST\IEA-15-240-RWT-UMaineSemi3.out,
*WUP (Warm-up period to be removed in OpenFAST results)
0,
*SFQ (Sampling frequency for stress history)
1,

4NodalStress (Stress history under concentrated nodal load)

✓ How to prepare NodalStress manually?



①-1 Select Module3 FEA result folder
* FEA_seed.DAT file
①-2 Select Module4 FEA result folder
② Select unit system of FEA seed model
③ Select FEA seed file
④ Assign element set containing elements with stress printed
⑤ Assign OpenFAST OUT file(s)
⑥ Assign Warm-up period to be removed in OpenFAST results
*Must be in second
⑦ Assign Sampling frequency for stress history
*Must be an integer
⑧ Create Module4 folder and generate FEA model files

4NodalStress (Stress history under concentrated nodal load)

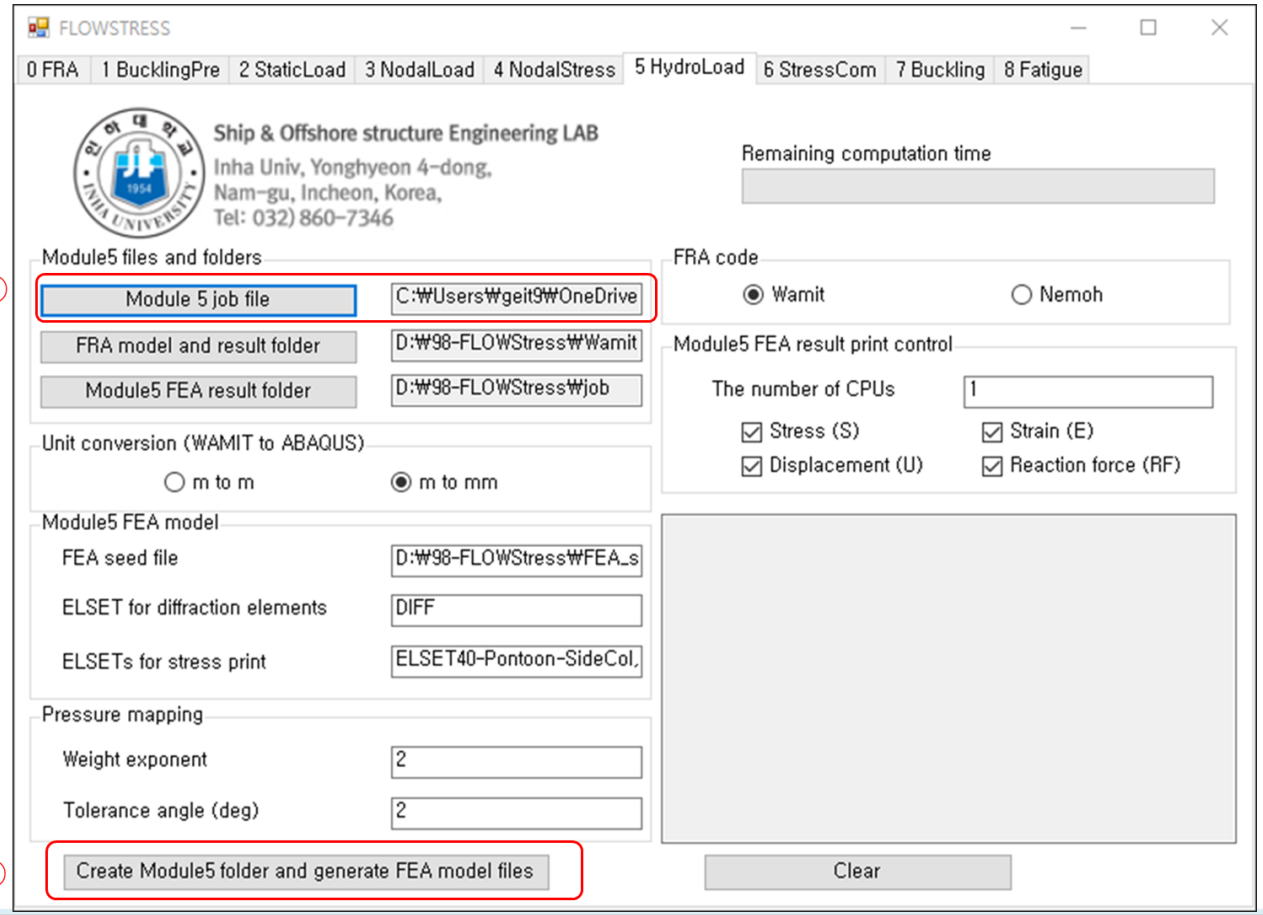
✓ How to check NodalStress results?

- Check if FLOWStress successfully created Module4 folder (\4_NodalStress), nodal stress history files (e.g. NodalStress_Case1.BIN, NodalStress_Case2.BIN…) and OpenFAST stress RAOs files (Acceleration.CSV, FairleadTension.CSV, TowerBase.CSV)

5HydroLoad with Wamit results (FEA under pressure RAO)

✓ How to prepare HydroLoad with JOB file?

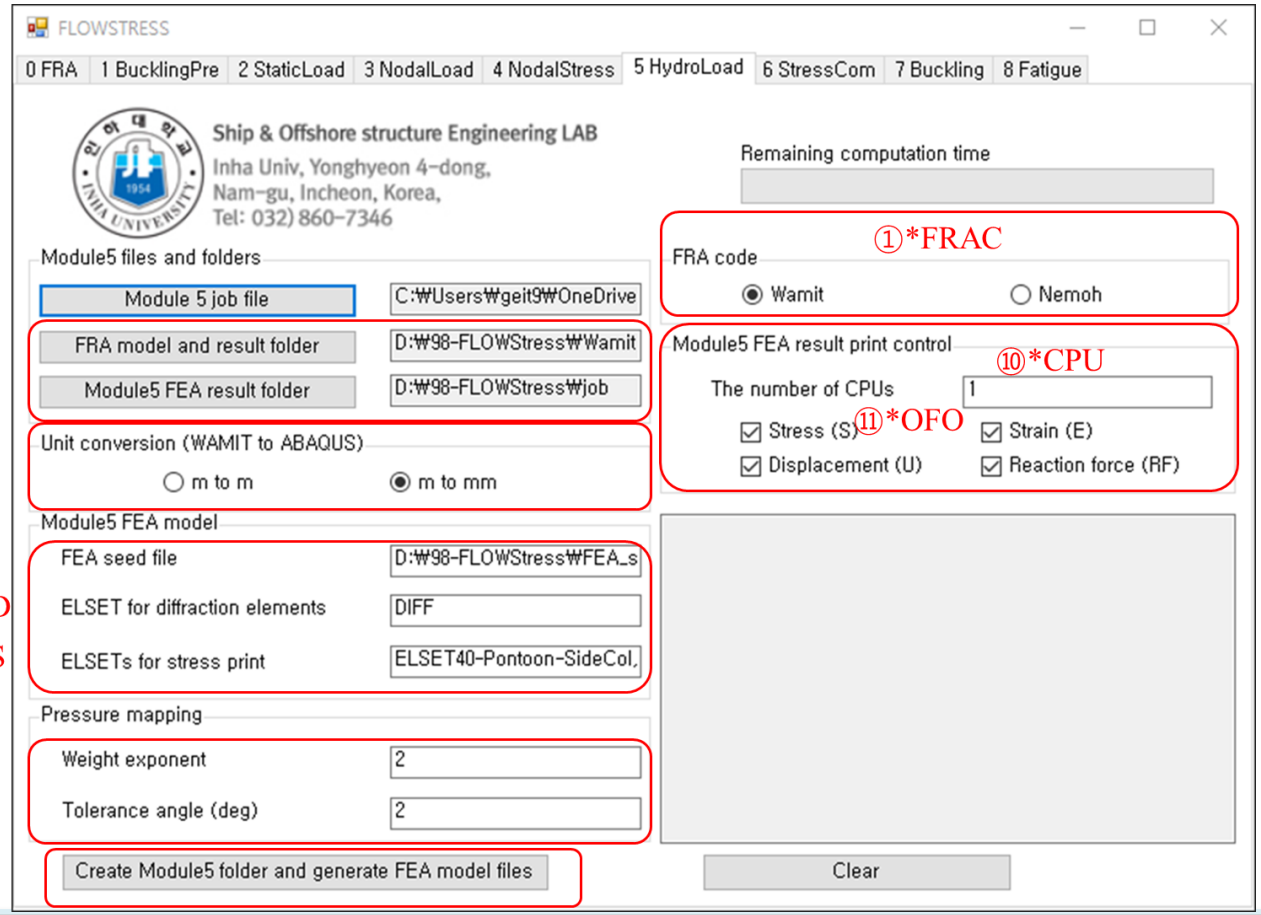
① Select JOB file
② Create Module5 folder and generate FEA model files




```
*FRAC (Frequency response analysis code)
1,
*DFRA (Folder path for frequency response analysis model and result files)
D:\98-FLOWStress\Wamit
*DFLO (Folder path where FLOWStress generates results)
D:\98-FLOWStress\job
*UNABS (Unit system of FEA seed file)
,
*FABS (FEA seed file)
D:\98-FLOWStress\FEA_seed\FEA_seed.inp
*ELSETD (Element set  containing diffraction elements)
DIFF,
*ELSETS (Element set containing elements with stress printed)
ELSET40-Pontoon-SideCol,
ELSET40-Pontoon-CenCol,
*EXP (Pressure mapping exponent)
,
*PTA (Pressure mapping tolerance angle (deg) to identify elements in a same plane)
,
*CPU (The number of CPUs for FEA analysis)
1,
*OFO (ODB field output option)
S, E, RF, U,
```

5HydroLoad with Wamit results (FEA under pressure RAO)

✓ How to prepare HydroLoad manually?



① Select FRA code 'Wamit'
② Select folder path for frequency response analysis model and result files(5P)
③ Select Module5 FEA result folder
④ Select unit conversion from Wamit file to FEA seed file
⑤ Select FEA seed file
⑥ Assign element set containing diffraction elements
⑦ Assign element set containing elements with stress printed
⑧ Assign pressure mapping exponent
⑨ Assign pressure mapping tolerance angle to identify elements in a same plane
*Must be in degree
⑩ Assign the number of CPUs for FEA analysis
⑪ Assign ODB field output option
⑫ Create Module5 folder and generate FEA model files

5HydroLoad with Wamit results (FEA under pressure RAO)

✓ How to execute HydroLoad with Wamit results?

1. Manually prepare Wamit result file (5P), respectively.
2. Check if FLOWStress successfully created a Module5 folder (\5_HydroLoad), diffraction load files (DIFF-D001-F001.INP, DIFF-D001-F002.INP…), diffraction model files (FE-DIFF-D001.INP, FE-DIFF-D002.INP…), radiation load files (RAD-D001-F001.INP, RAD-D001-F002.INP…), radiation model files (FE-RAD-D001.INP, FE-RAD-D002.INP…) and batch files (ABAQUS_RUN_DIFF.BAT and ABAQUS_RUN_RAD.BAT).
3. Execute ABAQUS_RUN_DIFF.BAT and ABAQUS_RUN_RAD.BAT.
4. Check if the diffraction result files (FE-DIFF-D001.DAT, FE-DIFF-D002.DAT…) and radiation result files (FE-RAD-D001.DAT, FE-RAD-D002.DAT …) are successfully generated. A keyword 'ERROR' should not exist in DAT files.

5HydroLoad with Wamit results (FEA under pressure RAO)

✓ File structures in HydroLoad

- Diffraction load file (e.g. DIFF-D001-F001.INP) and radiation load file (e.g. RAD-D001-F001.INP) are containing wave excitation-induced pressure RAOs and radiation-induced pressure RAOs.
- A diffraction model file (e.g. FE-DIFF-D001.INP) or a radiation model file (e.g. FE-RAD-D001.INP) includes diffraction or radiation load files same as the number of wave frequencies.
- For example, file structure is shown for 2 wave incident angles and 3 wave frequencies

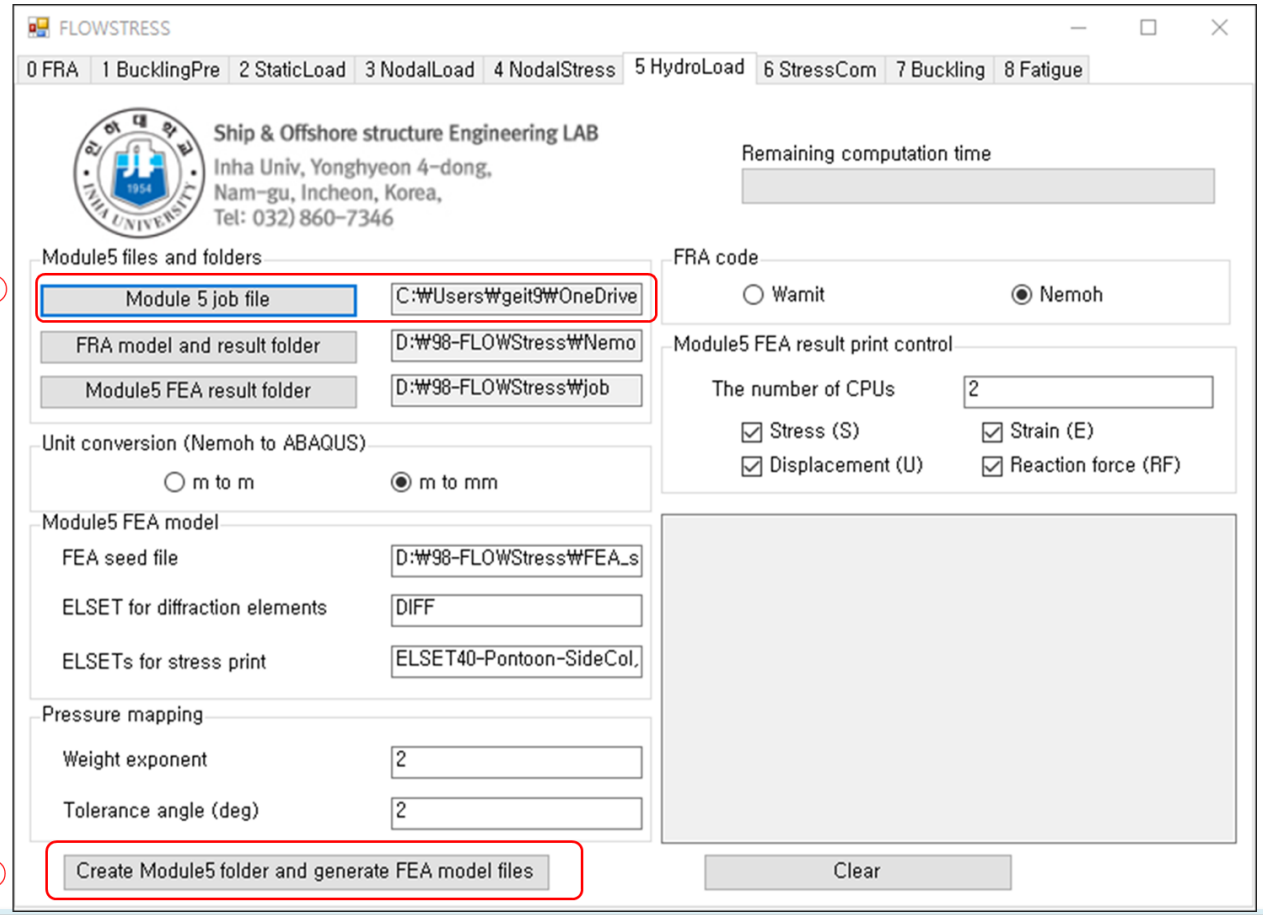
Diffraction model file	Diffraction load file	Diffraction result file
FE-DIFF-D001.INP	DIFF-D001-F001.INP	FE-DIFF-D001.DAT
	DIFF-D001-F002.INP	
	DIFF-D001-F003.INP	
FE-DIFF-D002.INP	DIFF-D002-F001.INP	FE-DIFF-D002.DAT
	DIFF-D002-F002.INP	
	DIFF-D002-F003.INP	

Radiation model file	Radiation load file	Radiation result file
FE-RAD-D001.INP	RAD-D001-F001.INP	FE-RAD-D001.DAT
	RAD-D001-F002.INP	
	RAD-D001-F003.INP	
FE-RAD-D002.INP	RAD-D002-F001.INP	FE-RAD-D002.DAT
	RAD-D002-F002.INP	
	RAD-D002-F003.INP	

5HydroLoad with Nemoh results (FEA under pressure RAO)

✓ How to prepare HydroLoad with JOB file?

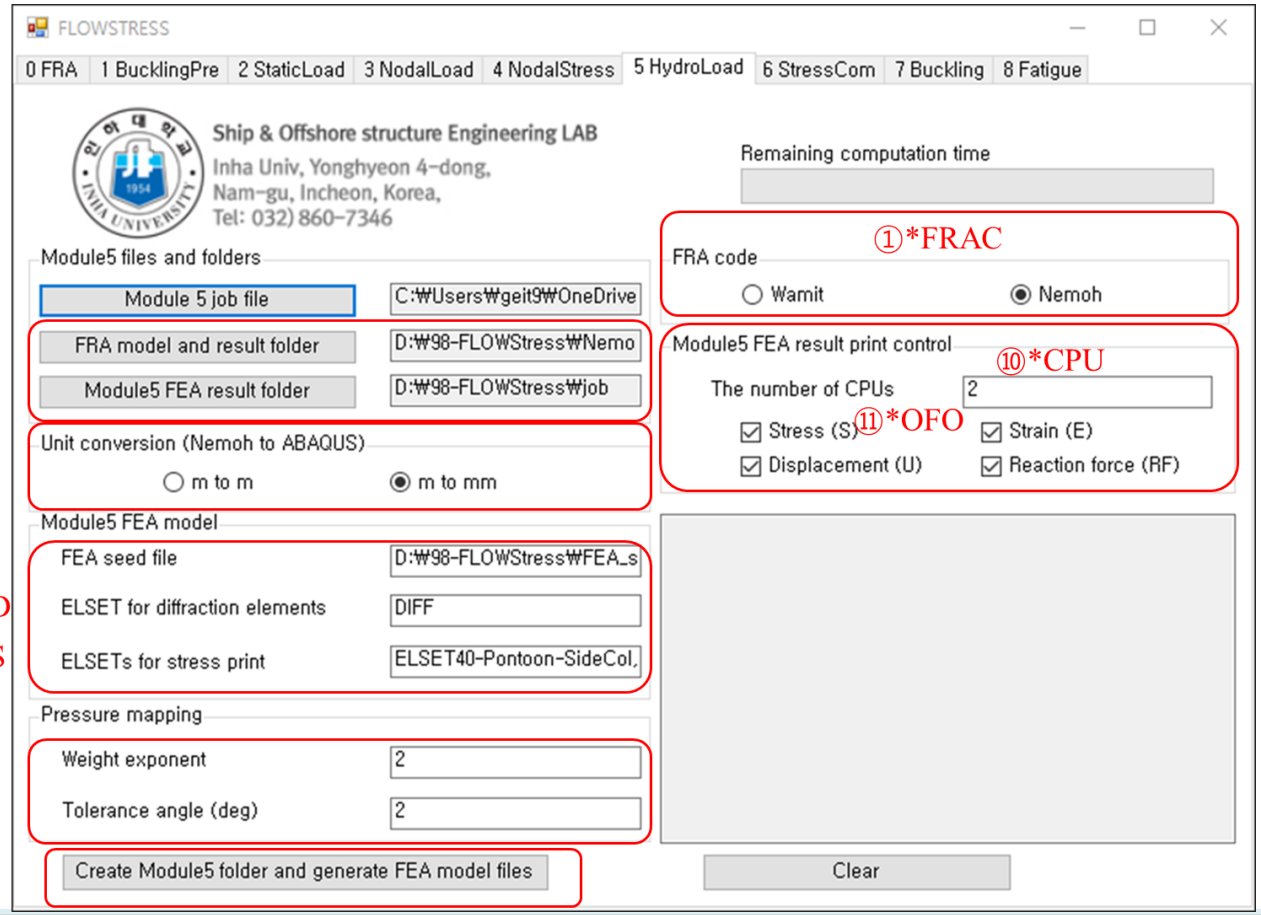
① Select JOB file
② Create Module5 folder and generate FEA model files




```
*FRAC (Frequency response analysis code)
2,
*DFRA (Folder path for frequency response analysis model and result files)
D:\98-FLOWStress\Nemoh
*DFLO (Folder path where FLOWStress generates results)
D:\98-FLOWStress\job
*UNABS (Unit system of FEA seed file)
2,
*FABS (FEA seed file)
D:\98-FLOWStress\FEA_seed\FEA_seed.inp
*ELSETD (Element set  containing diffraction elements)
DIFF,
*ELSETS (Element set containing elements with stress printed)
ELSET40-Pontoon-SideCol,
ELSET40-Pontoon-CenCol,
*EXP (Pressure mapping exponent)
2,
*PTA (Pressure mapping tolerance angle (deg) to identify elements in a same plane)
2,
*CPU (The number of CPUs for FEA analysis)
2,
*OFO (ODB field output option)
S, E, RF, U,
```

5HydroLoad with Nemoh results (FEA under pressure RAO)

✓ How to prepare HydroLoad manually?



① Select FRA code 'Nemoh'
② Select folder path for frequency response analysis model and result files(pressure.00001.DAT)
③ Select Module5 FEA result folder
④ Select unit conversion from Nemoh file to FEA seed file
⑤ Select FEA seed file
⑥ Assign element set containing diffraction elements
⑦ Assign element set containing elements with stress printed
⑧ Assign pressure mapping exponent
⑨ Assign pressure mapping tolerance angle to identify elements in a same plane
*Must be in degree
⑩ Assign the number of CPUs for FEA analysis
⑪ Assign ODB field output option
⑫ Create Module5 folder and generate FEA model files

5HydroLoad with Nemoh results (FEA under pressure RAO)

✓ File structures in HydroLoad

- Diffraction load file (e.g. DIFF-D001-F001.INP) and radiation load file (e.g. RAD-D001-F001.INP) are containing wave excitation-induced pressure RAOs and radiation-induced pressure RAOs.
- A diffraction model file (e.g. FE-DIFF-D001.INP) or a radiation model file (e.g. FE-RAD-D001.INP) includes diffraction or radiation load files same as the number of wave frequencies.
- For example, for 2 wave incident angles and 3 wave frequencies

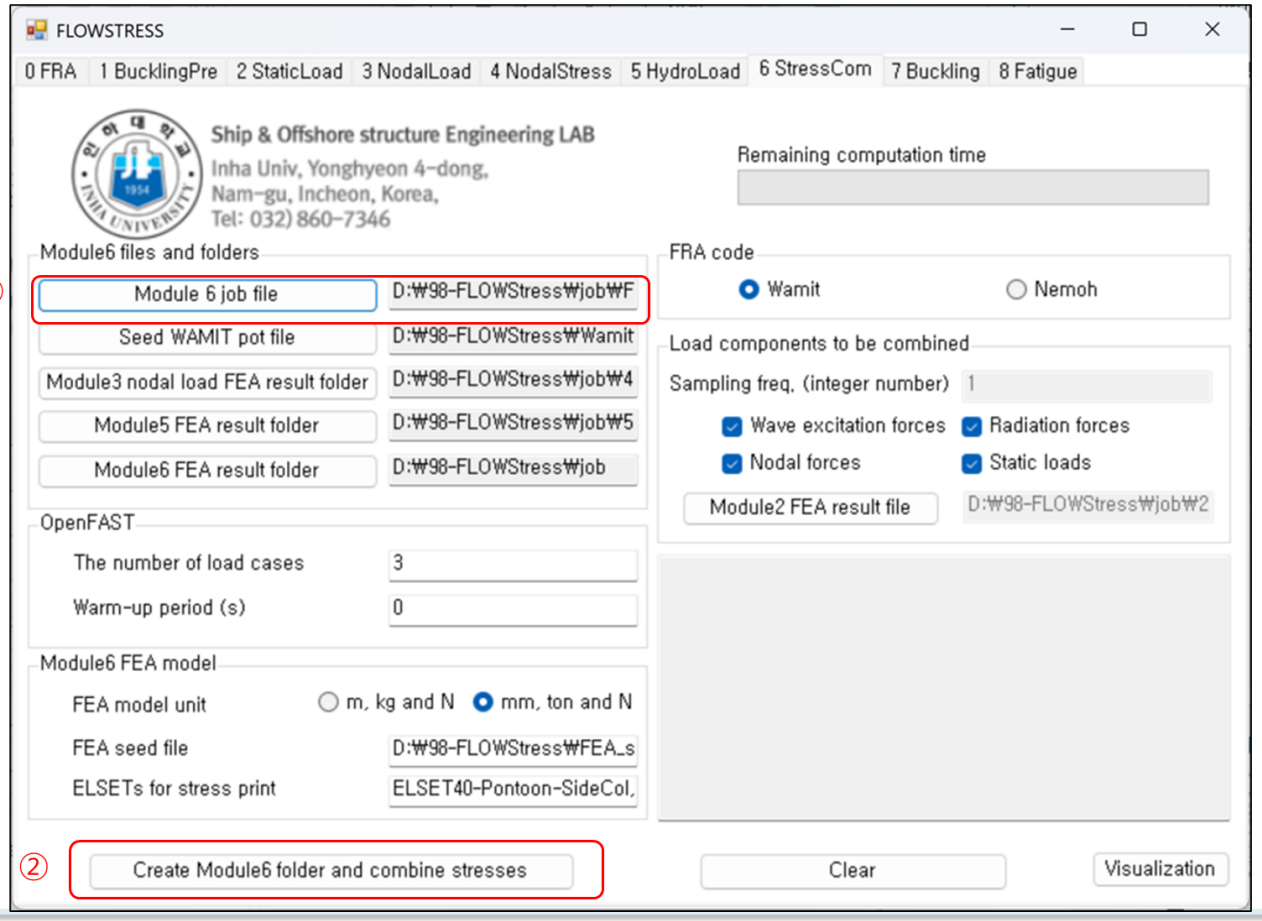
Diffraction model file	Diffraction load file	Diffraction result file
FE-DIFF-D001.INP	DIFF-D001-F001.INP	FE-DIFF-D001.DAT
	DIFF-D001-F002.INP	
	DIFF-D001-F003.INP	
FE-DIFF-D002.INP	DIFF-D002-F001.INP	FE-DIFF-D002.DAT
	DIFF-D002-F002.INP	
	DIFF-D002-F003.INP	

Radiation model file	Radiation load file	Radiation result file
FE-RAD-D001.INP	RAD-D001-F001.INP	FE-RAD-D001.DAT
	RAD-D001-F002.INP	
	RAD-D001-F003.INP	
FE-RAD-D002.INP	RAD-D002-F001.INP	FE-RAD-D002.DAT
	RAD-D002-F002.INP	
	RAD-D002-F003.INP	

6StressCom with Wamit results (Stress combination)

✓ How to prepare StressCom with JOB file?

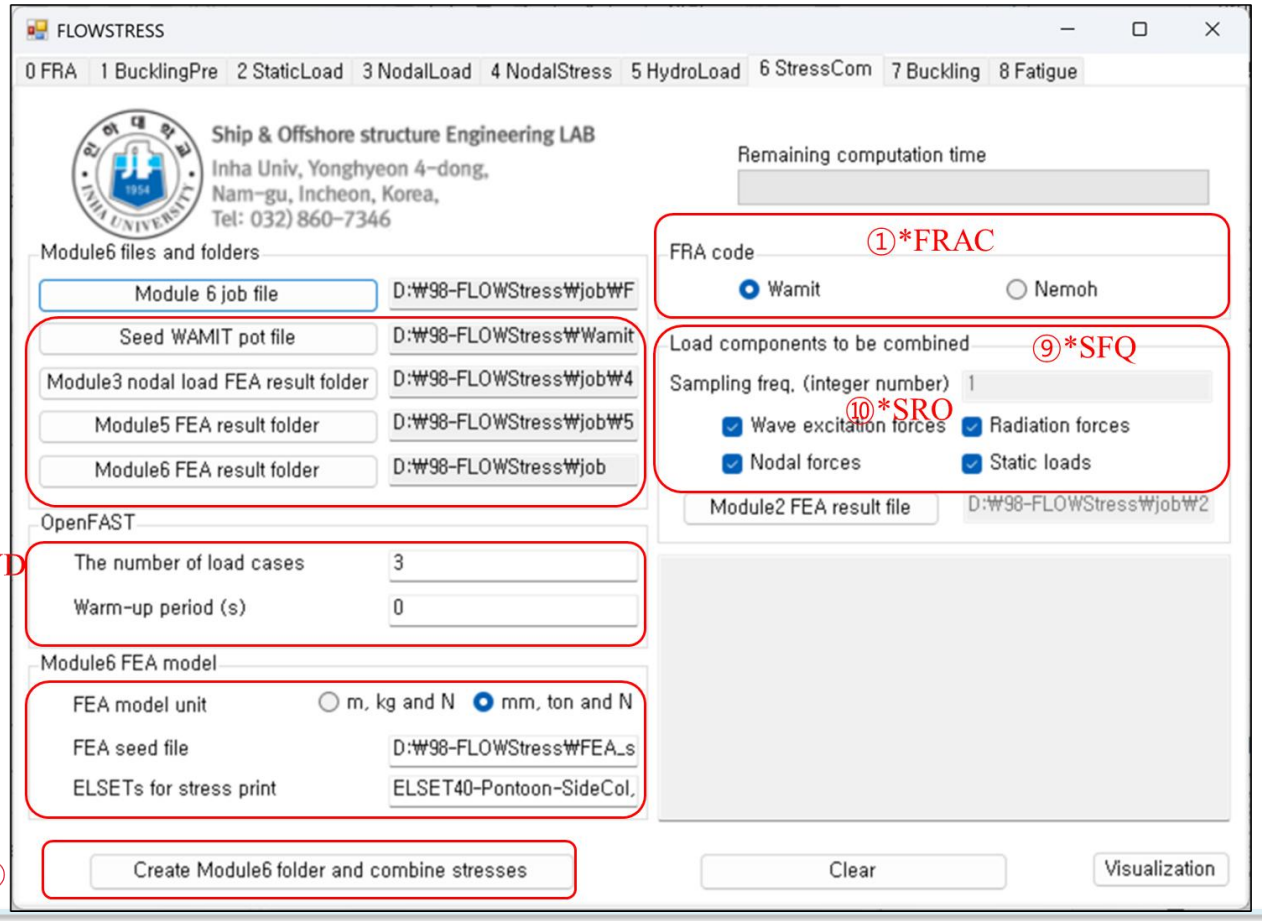
① Select JOB file
② Create Module6 folder and combine stresses




```
*FRAC (Frequency response analysis code)
1,
*DFRA (Folder path for frequency response analysis model and result files)
D:\98-FLOWStress\Wamit
*DFLO (Folder path where FLOWStress generates results)
D:\98-FLOWStress\job
*UNABS (Unit system of FEA seed file)
2,
*FABS (FEA seed file)
D:\98-FLOWStress\FEA_seed\FEA_seed.inp
*ELSETS (Element set containing elements with stress printed)
ELSET40-Pontoon-SideCol
ELSET40-Pontoon-CenCol
*WUP (Warm-up period to be removed in OpenFAST results)
0,
*SFQ (Sampling frequency for stress history)
,
*SRO (Stress print option according to load sources)
, , , ,
*FOO (OpenFAST OUT file(s))
D:\98-FLOWStress\OpenFAST\IEA-15-240-RWT-UMaineSemi1.out,
D:\98-FLOWStress\OpenFAST\IEA-15-240-RWT-UMaineSemi2.out,
D:\98-FLOWStress\OpenFAST\IEA-15-240-RWT-UMaineSemi3.out,
*FHYD (OpenFAST HydroDyn DAT file(s))
D:\98-FLOWStress\OpenFAST\IEA-15-240-RWT-UMaineSemi_HydroDyn1.dat,
D:\98-FLOWStress\OpenFAST\IEA-15-240-RWT-UMaineSemi_HydroDyn2.dat,
D:\98-FLOWStress\OpenFAST\IEA-15-240-RWT-UMaineSemi_HydroDyn3.dat,
```

6StressCom with Wamit results (Stress combination)

✓ How to prepare StressCom manually?



① Select FRA code 'Wamit'
② Select folder path for frequency response analysis model and result files(5P)
③-1 Select Module3 FEA result folder
③-2 Select Module5 FEA result folder
③-3 Select Module6 FEA result folder
④ Select OpenFAST OUT file(s) and HydroDyn DAT file(s)
⑤ Assign Warm-up period to be removed in OpenFAST results
⑥ Select unit system of FEA seed model
⑦ Select FEA seed file
⑧ Assign element set containing elements with stress printed
⑨ Assign Sampling frequency for stress history
⑩ Assign stress print option according to load sources
⑪ Create Module6 folder and combine stresses

6StressCom with Wamit results (Stress combination)

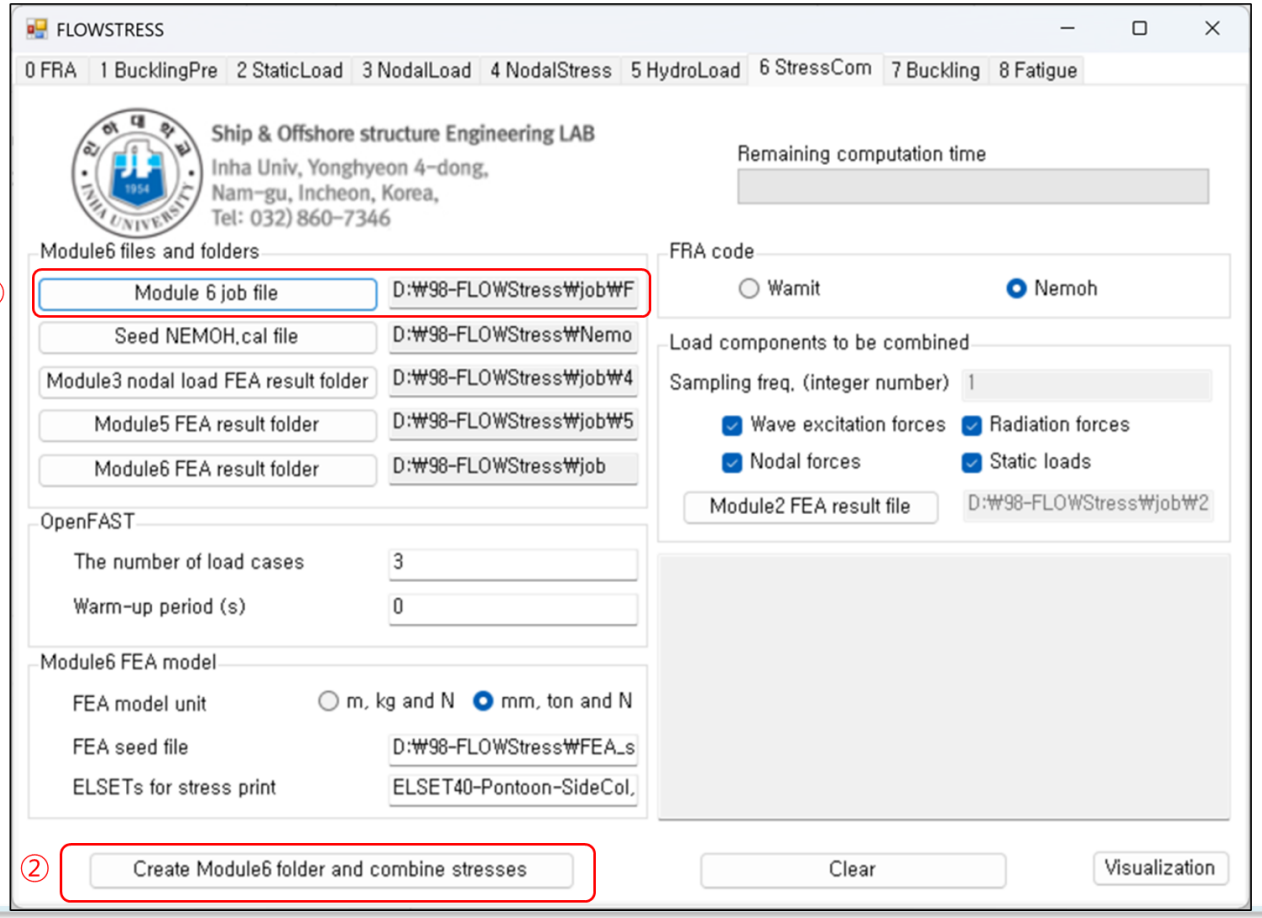
✓ How to view StressCom results?

1) Before Module6 execution, Module0, Module3, Module5 analyses should be finished.
2) Check if FLOWStress successfully created Module6 folder (\6_StressCom) and stress history files (e.g. StressCom_Case1.BIN, StressCom_Case2.BIN…).
3) By clicking 'visualization,' you can view the time-domain stress history at a selected element set.

6StressCom with Nemoh results (Stress combination)

✓ How to prepare StressCom with JOB file?

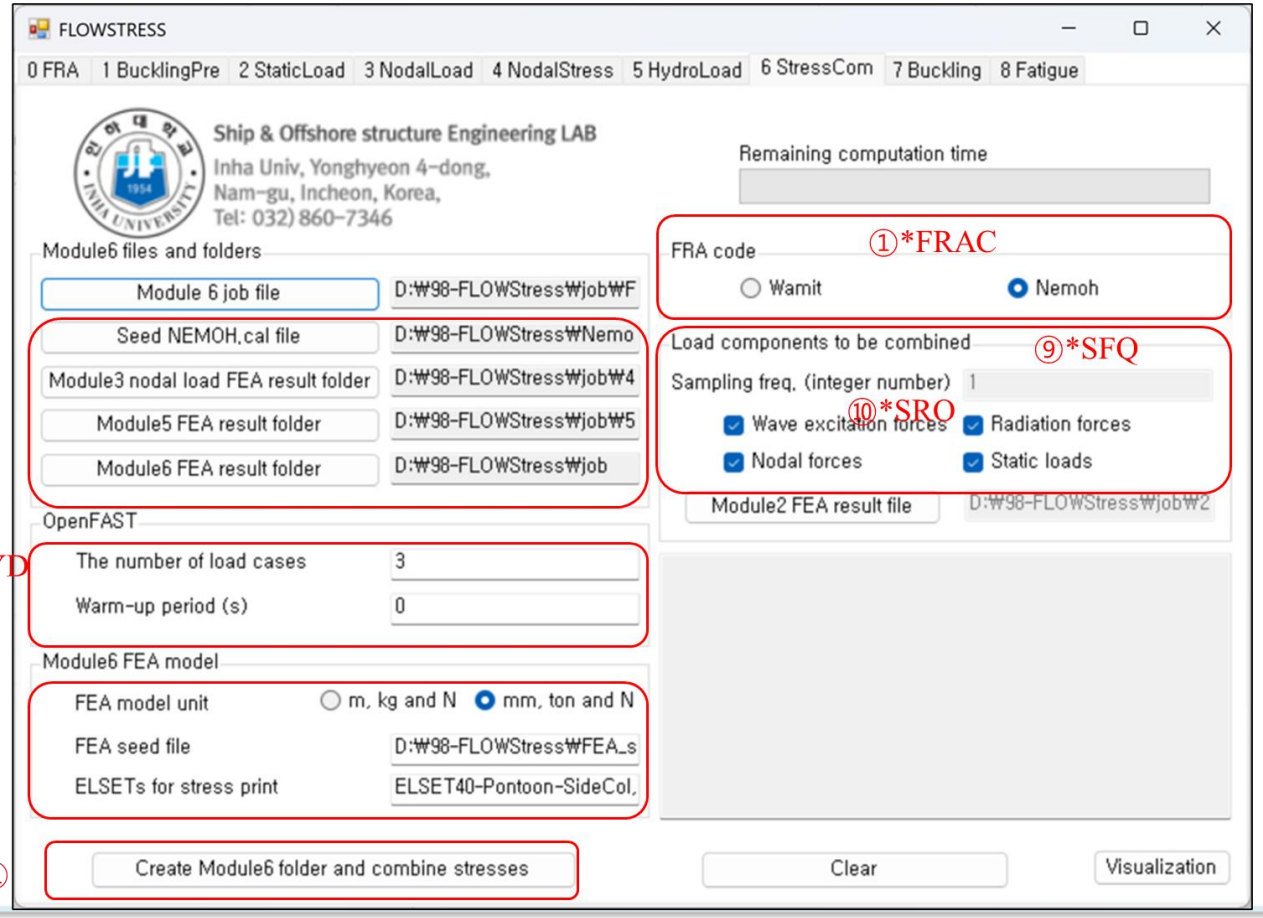
① Select JOB file
② Create Module6 folder and combine stresses




```
*FRAC (Frequency response analysis code)
2,
*DFRA (Folder path for frequency response analysis model and result files)
D:\98-FLOWStress\Nemoh
*DFLO (Folder path where FLOWStress generates results)
D:\98-FLOWStress\job
*UNABS (Unit system of FEA seed file)
2,
*FABS (FEA seed file)
D:\98-FLOWStress\FEA_seed\FEA_seed.inp
*ELSETS (Element set containing elements with stress printed)
ELSET40-Pontoon-SideCol,
ELSET40-Pontoon-CenCol,
*WUP (Warm-up period to be removed in OpenFAST results)
0,
*SFQ (Sampling frequency for stress history)
,
*SRO (Stress print option according to load sources)
, , , ,
*FOO (OpenFAST OUT file(s))
D:\98-FLOWStress\OpenFAST\IEA-15-240-RWT-UMaineSemi1.out,
D:\98-FLOWStress\OpenFAST\IEA-15-240-RWT-UMaineSemi2.out,
D:\98-FLOWStress\OpenFAST\IEA-15-240-RWT-UMaineSemi3.out,
*FHYD (OpenFAST HydroDyn DAT file(s))
D:\98-FLOWStress\OpenFAST\IEA-15-240-RWT-UMaineSemi_HydroDyn1.dat,
D:\98-FLOWStress\OpenFAST\IEA-15-240-RWT-UMaineSemi_HydroDyn2.dat,
D:\98-FLOWStress\OpenFAST\IEA-15-240-RWT-UMaineSemi_HydroDyn3.dat,
```

6StressCom with Nemoh results (Stress combination)

✓ How to prepare StressCom manually?



① Select FRA code 'Nemoh'
② Select folder path for frequency response analysis model and result files(Pressure_00001.DAT)
③-1 Select Module3 FEA result folder
③-2 Select Module5 FEA result folder
③-3 Select Module6 FEA result folder
④ Select OpenFAST OUT file(s) and HydroDyn DAT file(s)
⑤ Assign Warm-up period to be removed in OpenFAST results
⑥ Select unit system of FEA seed model
⑦ Select FEA seed file
⑧ Assign element set containing elements with stress printed
⑨ Assign Sampling frequency for stress history
⑩ Assign stress print option according to load sources
⑪ Create Module6 folder and combine stresses

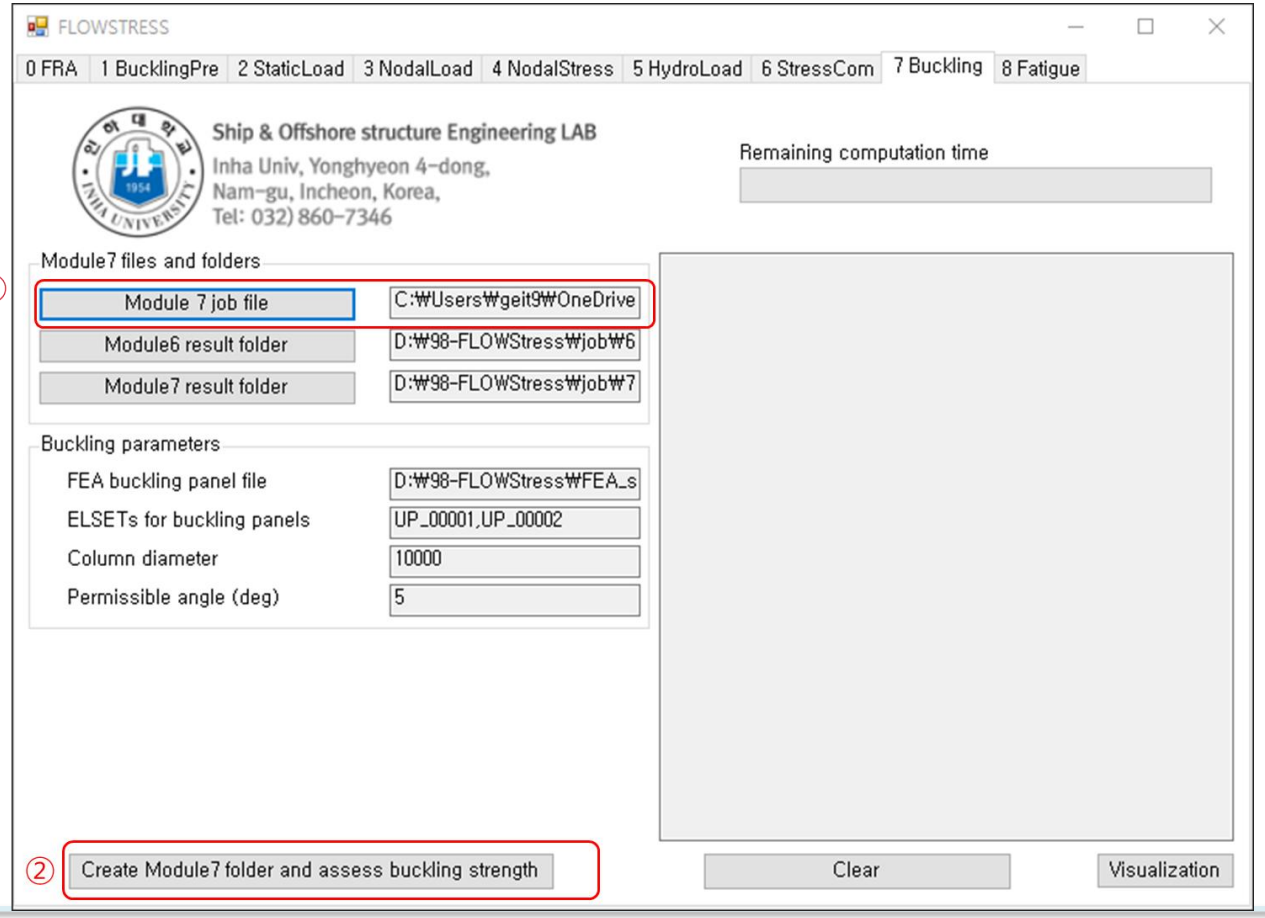
6StressCom with Nemoh results (Stress combination)

✓ How to view StressCom result?

1) Before Module6 execution, Module0, Module3, Module5 analyses should be finished.
2) Check if FLOWStress successfully created Module6 folder (\6_StressCom) and stress history files (e.g. StressCom_Case1.BIN, StressCom_Case2.BIN…).
3) By clicking 'visualization,' you can view the time-domain stress history at a selected element set.

7Buckling (Buckling ULS assessment)

✓ How to prepare Buckling with JOB file?

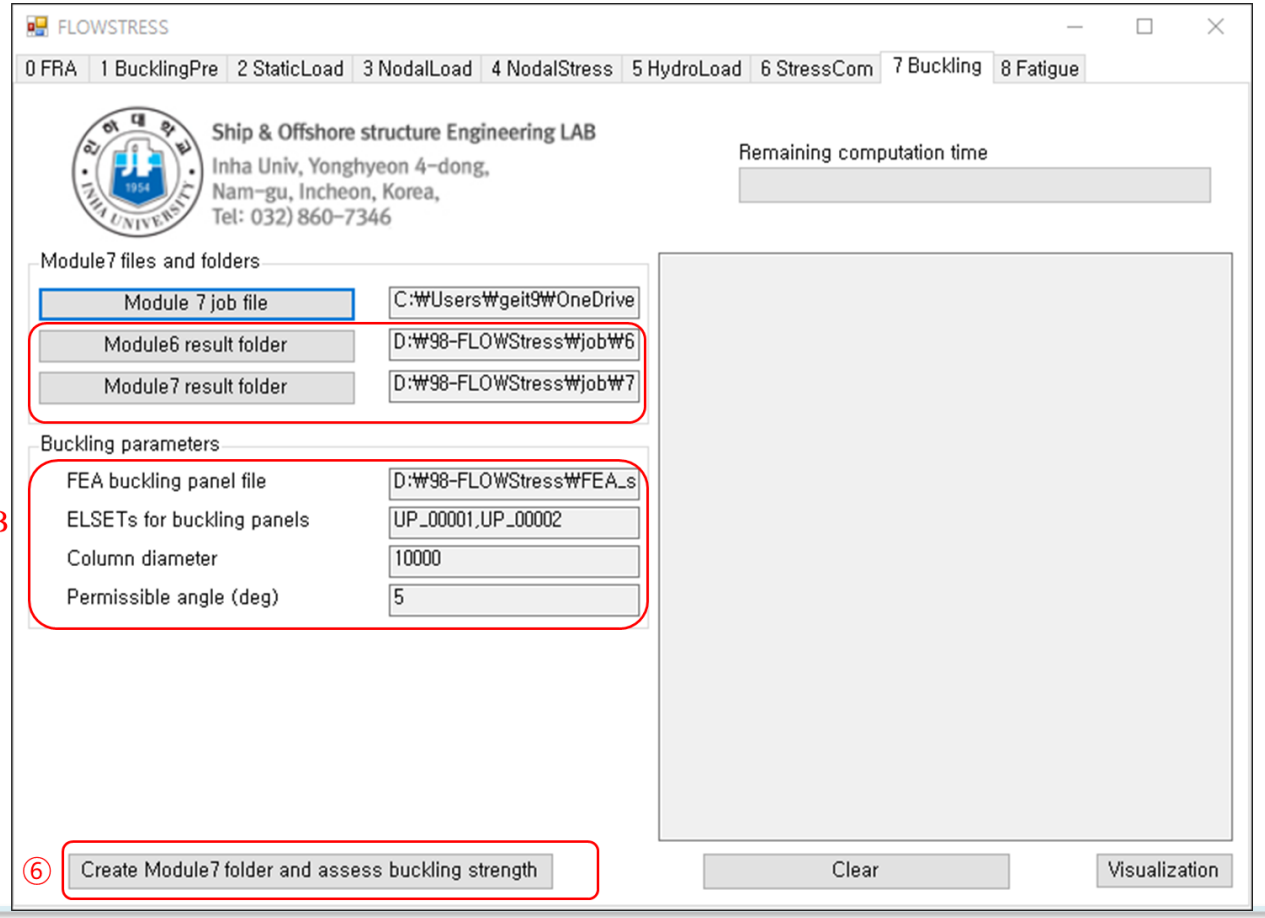


① Select JOB file
② Create Module7 folder and assess buckling strength

*DFLO (Folder path where FLOWStress generates results)
D:\98-FLOWStress\job
*FABB (FEA element set file in which identified buckling panels are defined)
D:\98-FLOWStress\FEA_seed\FEA_seed_BucklingPre.inp
*ELSETB (Element set containing identified buckling panels)
UP_00001,
UP_00002,
*UNABS (Unit system of FEA model containing seed model)
2,
*CDIA (Column diameter)
,
*PANG (Buckling panel tolerance angle in degrees)
,

7Buckling (Buckling ULS assessment)

✓ How to prepare Buckling manually?



①*DFLO

②*FABB
③*ELSETB
④*CDIA
⑤*PANG

①-1 Select Module6 result folder
①-2 Select Module7 result folder
② Select a FEA element set file in which identified buckling panels are defined
③ Assign element set containing identified buckling panels
④ Assign column diameter
⑤ Assign buckling panel tolerance angle
⑥ Create Module7 folder and assess buckling strength

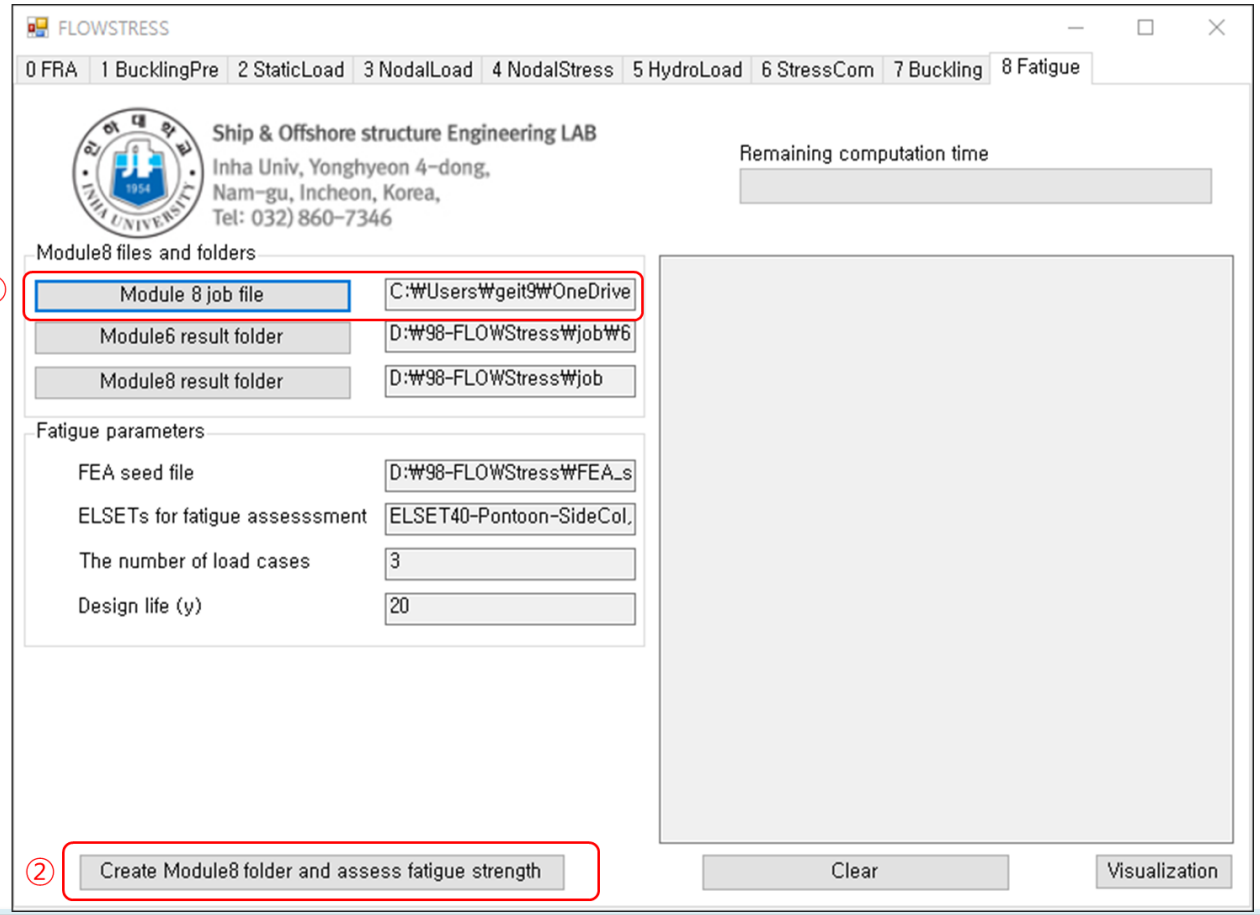
7Buckling (Buckling ULS assessment)

✓ How to view Buckling result?

1) Before Module7 execution, Module6 analyses should be finished.
2) Check if FLOWStress successfully created Module7 folder (\7_Buckling), a panel information file (Panel_info.CSV) and buckling result file (e.g. Buckling_Case1.CSV, Buckling_Case2.CSV…).
3) By clicking 'visualization,' you can view the time-domain buckling utilization factor history at unstiffened panels.

8Fatigue (FLS assessment)

✓ How to prepare Fatigue with JOB file?



① Select JOB file
② Create Module8 folder and assess fatigue strength

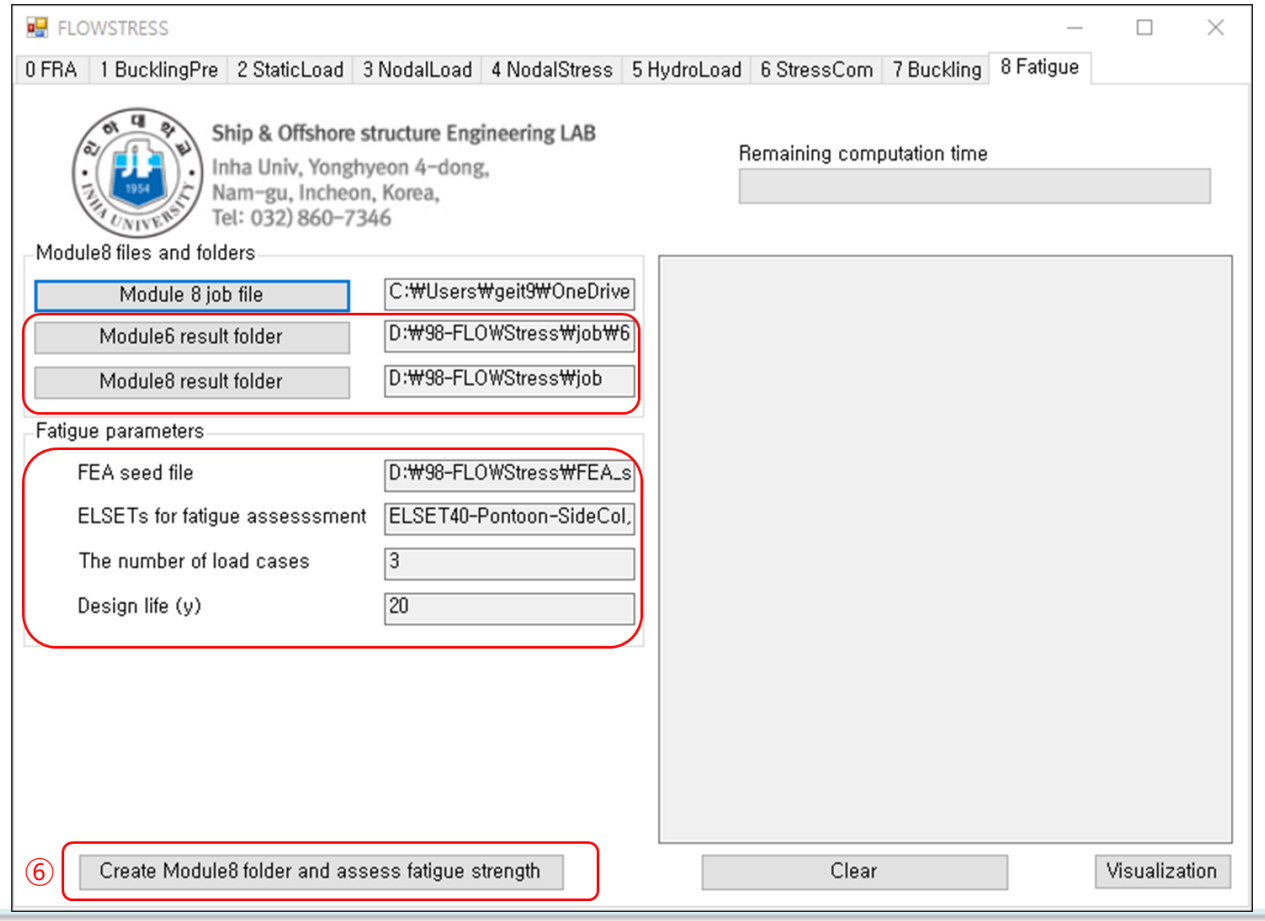
*DFLO (Folder path where FLOWStress generates results)
D:\98-FLOWStress\job
*FABS (FEA seed file)
D:\98-FLOWStress\FEA_seed\FEA_seed.inp
*FAT (Fatigue assessment parameters)
0, ELSET40-Pontoon-SideCol, 3.0, 12.164, 5.0, 15.606, 25, 0.2, 1, 1.12,
0, ELSET40-Pontoon-CenCol, 3.0, 11.764, 5.0, 15.606, 25, 0.2, 1, 1.12,
1, HOTSPOT1, 3.0, 12.164, 5.0, 15.606, 25, 0.2, 1, 1,
*FOO (OpenFAST OUT file(s))
D:\98-FLOWStress\OpenFAST\IEA-15-240-RWT-UMaineSemi1.out,
D:\98-FLOWStress\OpenFAST\IEA-15-240-RWT-UMaineSemi2.out,
D:\98-FLOWStress\OpenFAST\IEA-15-240-RWT-UMaineSemi3.out,
*LIF (Design life in year)
20,
*PROB (Probability of each load case)
0.34,
0.33,
0.33,

8Fatigue (FLS assessment)

✓ How to prepare Fatigue manually?

①*DFLO

②*FABS
③*FAT
④*FOO
⑤*LIF



①-1 Select Module6 result folder
①-2 Select Module8 result folder
② Select a FEA seed file
③ Assign fatigue assessment parameters
*Hotspot type, ELSET, m1, log(a1), m2, log(a2), t_{ref}, k, DFF, SCF
④ Assign OpenFAST OUT file(s)
⑤ Assign Design life in year
⑥ Create Module8 folder and assess fatigue strength

8Fatigue (FLS assessment)

✓ How to view Fatigue result?

1) Before Module8 execution, Module6 analyses should be finished
2) Check if FLOWStress successfully created Module8 folder (\8_Fatigue), screening fatigue result file (Fatigue.OUT), and refined fatigue result file (Fatigue_REF.OUT).
   Free-license version supports maximum lines of fatigue assessment parameters, so FLS assessment is available for two locations. In order to obtain FLS results for more than two locations, contact the developer (Prof Joonmo CHOUNG, jmchoung@inha.ac.kr).
3) By clicking 'visualization,' you can view the time-domain fatigue damage history visualized in a 3D viewer.