Sesam

Buckling Analysis of EMULF Delta Floater using the Time Domain Response Reconstruction Method

Sesam Tutorial

Date: 26 June 2024

Prepared by: Digital Solutions at DNV

E-mail support: software.support@dnv.com

E-mail sales: digital@dnv.com

Table of contents

1 INTRODUCTION

This is a tutorial covering a floating OWT structure time history buckling analysis. The plate verification is done according to the DNV RP-C201 using the stiffener capacity model therein referred to as SCM2.

This tutorial covers the use of the Time Domain Response Reconstruction method. The following versions (or higher) of official Sesam applications are required to complete the tutorial: GeniE 8.10, HydroMesh 4.0, HydroD 7.0, Wadam 10.04, Sestra 10.18, Application Version Manager 3.2. In addition, a pre-release version of Sesam Core 3.1 is required.

This document explains how to perform a **buckling** analysis in **Sesam Core** though **OneWorkflow** for a selection of **design load cases (DLC)** using the **Response Reconstruction method**.

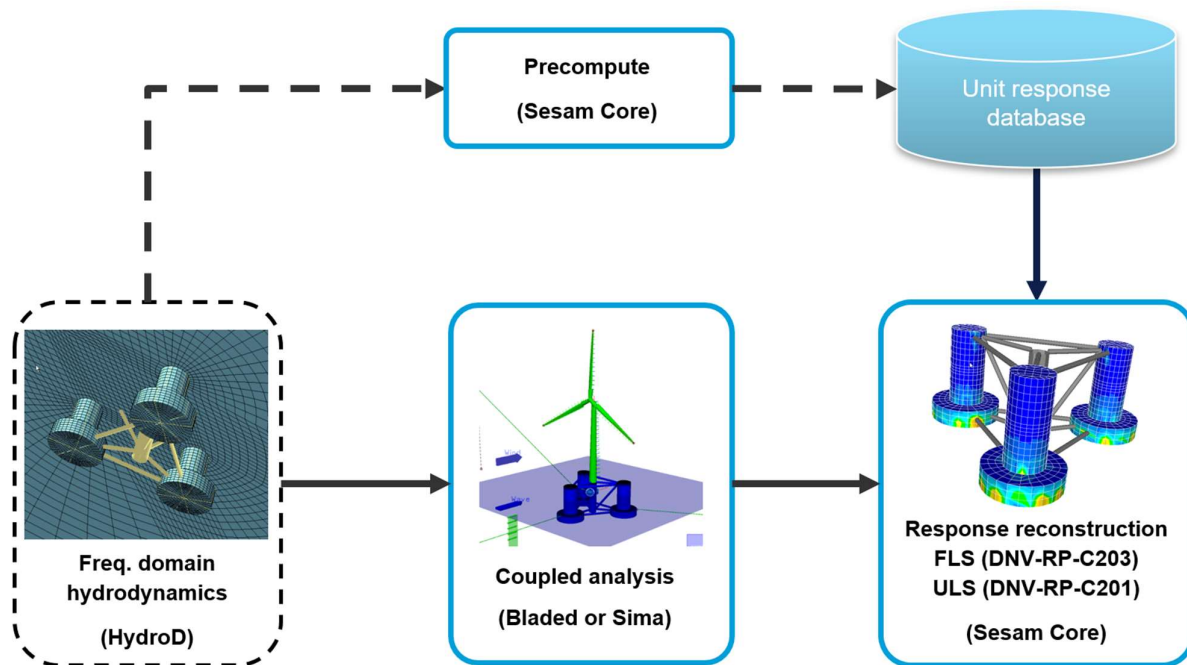The Response Reconstruction workflow has two distinct steps:

- **Precomputation** is performed once for each hydrodynamic loading condition. In this step, a **unit-response database** is generated. The database comprises the local response of the structure due to unit loads, unit motions and unit waves.
- **Reconstruction** is performed for every DLC. In this step, the time-domain detailed response of the structure is reconstructed by combining the precomputed unit responses and the structural kinematics, wave elevation and auxiliary forces computed in a **coupled analysis**.

The Reconstruction step is performed using **SesamCore**, which also calculates the ultimate limit state according to **DNV-RP-C201**. The structural kinematics, wave elevation, and auxiliary forces are computed in a coupled analysis using either **Bladed** or **Sima**. It is assumed that the coupled analysis for the given DLC has been run in advance of the Reconstruction step.

A **Jupyter notebook** using **OneWorkflow** covers the **load generation** and **buckling** analysis steps represented by the floater images in the right picture below.
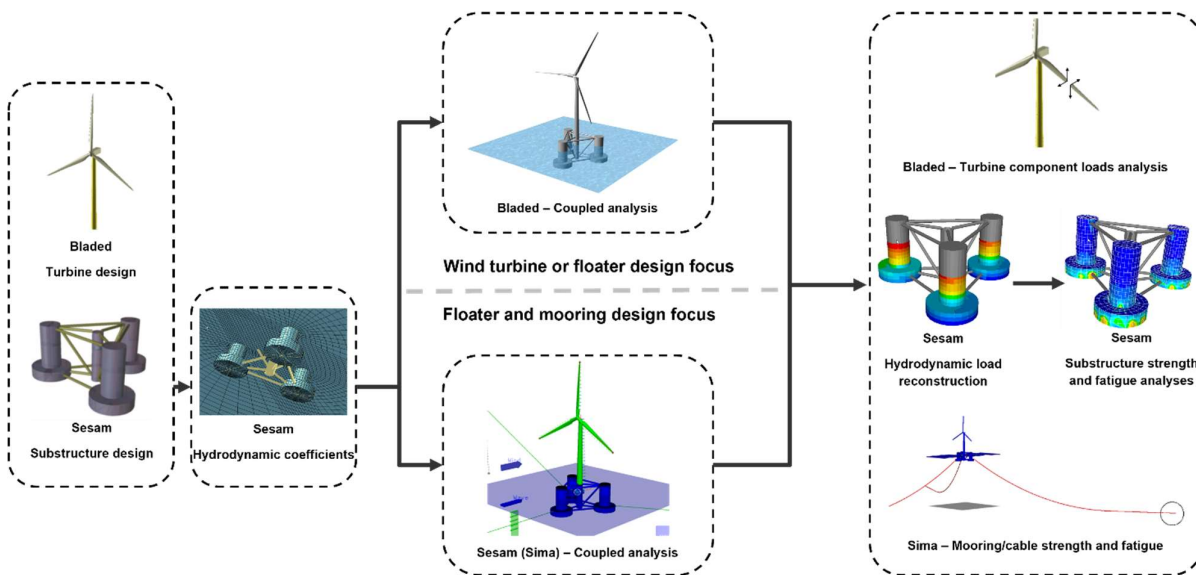
1.1 The Response Reconstruction Workflow

1.1.1 Precompute

Precomputation is performed once for each hydrodynamic loading condition. In this step, a unit-response database is generated. The database comprises the local response of the structure due to unit loads, unit motions and unit waves, as well as hydrostatic pressure and gravity. The responses due to unit motions and unit waves are computed in the frequency domain. The range of frequencies for which to compute this response must be selected by the user.

In the precompute step, **HydroD** is first used to set up the hydrodynamic model (structure, mass and panel model), loading condition (compartment filling), and decide on response frequencies. It is also necessary to create a Wadam template input file.
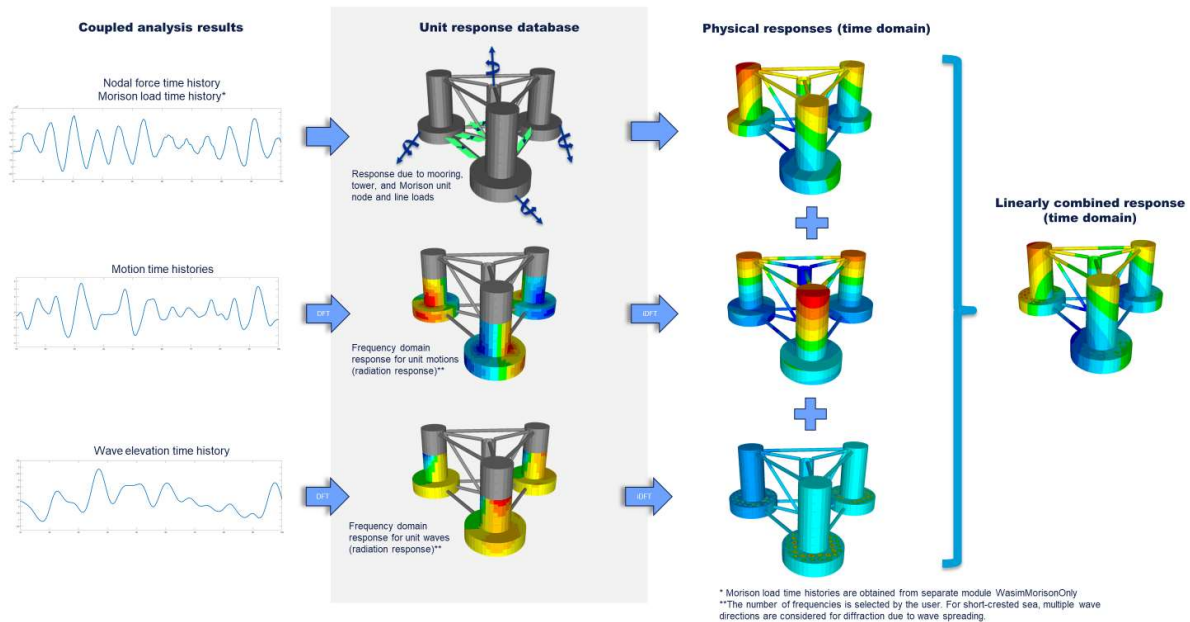
Subsequently, **SesamCore** is run to perform the generation of the unit response database. For the responses due to unit loads, **Sestra** is run in the background; for the responses due to unit motions and unit waves, **Wadam** and **Sestra** are run in the background.

1.1.2 Reconstruct

Response reconstruction is performed for every DLC. In this step, the structural kinematics, wave elevation and auxiliary forces computed in a coupled analysis is combined with the precomputed dataset to reconstruct the detailed local structure response. In this tutorial, it is assumed that the coupled analyses for all relevant design load cases have been run in advance using Sima.

The reconstruction step is performed using **SesamCore** as an integrated part of either an FLS or ULS analysis.

In the reconstruction step, the time-domain signals computed in the coupled analysis are combined with the precomputed data to reconstruct the detailed time-domain local quasi-static structure response. The time histories for nodal forces (mooring and tower interface) are directly combined with precomputed unit nodal-load responses. The six time histories for the structural kinematics and the time history for wave elevation are first transformed to the frequency domain; the discrete spectra are then weighted with the respective precomputed unit-motion and unit-wave responses before transformed back to the time domain. Combining all responses yields the total time-domain response. The diagram below depicts this procedure.

Coupled analysis results — Unit response database — Physical responses (time domain)

Nodal force time history
Morison load time history*

Response due to mooring, tower, and Morison unit node and line loads

Motion time histories

Frequency domain response for unit motions (radiation response)**

Wave elevation time history

Frequency domain response for unit waves (radiation response)**

Linearly combined response (time domain)

* Morison load time histories are obtained from separate module WasimMorisonOnly
**The number of frequencies is selected by the user. For short-crested sea, multiple wave directions are considered for diffraction due to wave spreading.
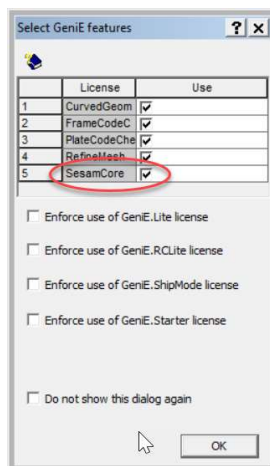
2 DEFINING THE BUCKLING CAPACITY MODEL IN GENIE

In this chapter, the structural model will be prepared and exported from GeniE for further use in OneWorkflow.
Modelling is not in the scope of this tutorial so the structural model is provided. For more information about how to the model in GeniE please check the GeniE tutorials in the help section.

The structural model for this tutorial is the EMULF Delta Floater Model which was designed as part of the COWI Fonden funded EMULF project.
Importing the model into GeniE
The required functionality in GeniE requires a
GENIE__SCORE license as shown in the
Edit > License / features dialog.

Select GeniE features

	License	Use
1	CurvedGeom	☑
2	FrameCodeC	☑
3	PlateCodeChe	☑
4	RefineMesh	☑
5	SesamCore	☑

☐ Enforce use of GeniE.Lite license

☐ Enforce use of GeniE.RCLite license

☐ Enforce use of GeniE.ShipMode license

☐ Enforce use of GeniE.Starter license

☐ Do not show this dialog again

OK

- Open GeniE and create a new workspace.
    - **File > New Workspace**
    - Input the name
      FOWT_ULS_model
    - Check the **Use Dual assembly**
      option
- **NOTE:** Use **SI units**.
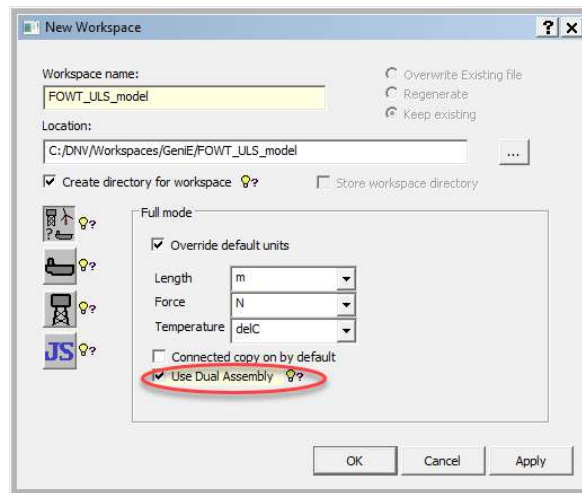
- Import the substructure model file *EMULF_Delta_Fine.gnx* from the workshop input files into GeniE.
    - Use *File > Import > Workspace (GNX file)* to import the model file.
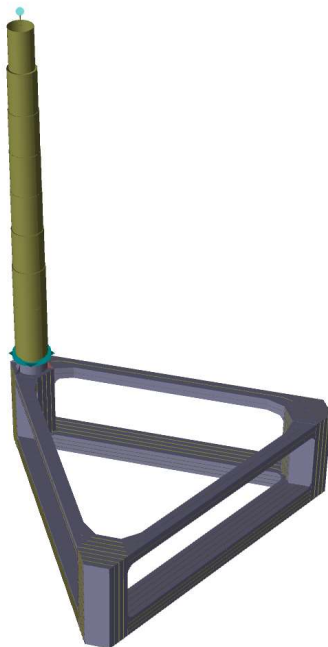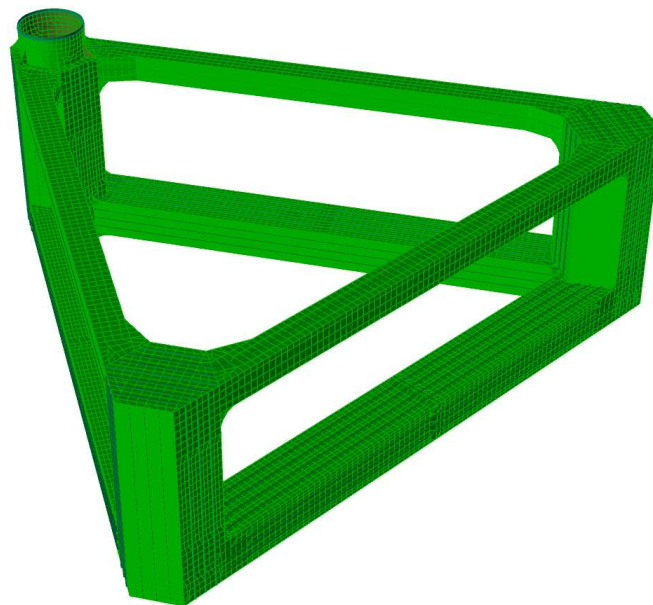
Figure 1 Global model of substructure   **Figure 2 Mesh of global model**

2.1   Exporting the mesh

Note that the mesh model must **be exported from GeniE and imported into HydroD to have Wadam compute the
hydrodynamic coefficients** (that are input to the Bladed or Sima coupled analysis runs).

To view the mesh in GeniE:
- Select the **Mesh – Transparent view** in the GeniE toolbar.

To export the mesh of the model for use in HydroD:

- Go to **File > Export > FEM file**. Save the mesh model as **T3.FEM**.

A .FEM file prepared in advance is provided with this tutorial. In the following, make sure to use the provided files as input to OneWorkflow. Note that if the GeniE model is remeshed, then the .json files with the capacity model may have to be exported again.

2.2 Defining the capacity manager

- Click **right mouse button (RMB)** on the **Capacity** folder to create a new capacity manager.

- Select the existing analysis, DNV-RP-C201 code check and no corrosion addition rule.
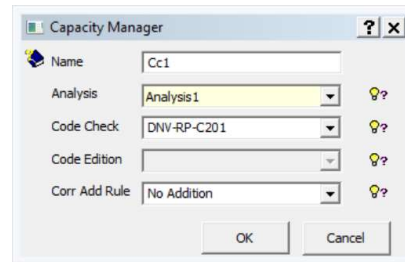
2.3 Creating the panels

- **Right mouse button (RMB)** on the newly created **capacity model** and select to create panels.

- On the new menu select the subset Col2_out and keep the other options as default, then press OK.
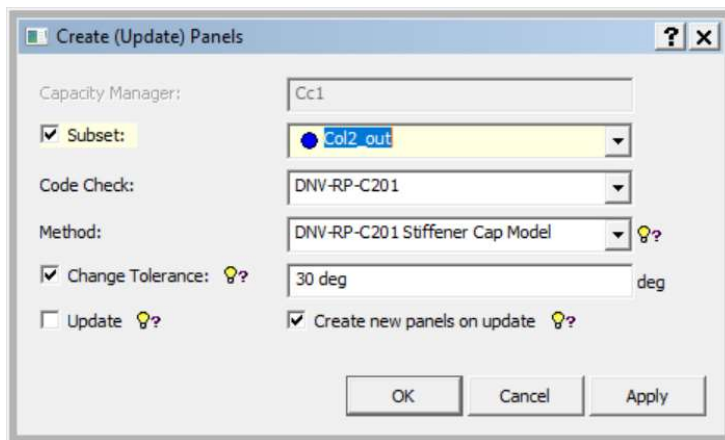
The panels will be created based on the selected subset using the stiffener capacity model (SCM2).

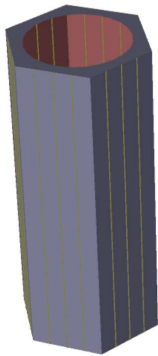- To view the capacity model change the view type to **Capacity Models**
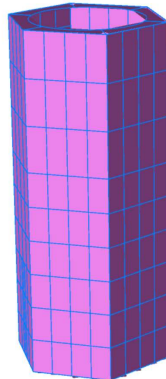
Figure 3 Model of COL2_OUT

Figure 4 Panels in the capacity model

2.4   Create the code check run

- Press **Right mouse button (RMB)** on the capacity model and select **Add run** to create the code check run

- Confirm that DNV-RP-C-201 is selected for the code check option.

- In the General tab the material factor can be adjusted for other limit states. For ULS the default value of 1.15 is adequate.

- In the Panel tab the end support for the stiffeners can be set to continuous or sniped. This is applied to both ends of the stiffeners. For this case it is ok to leave it as continuous
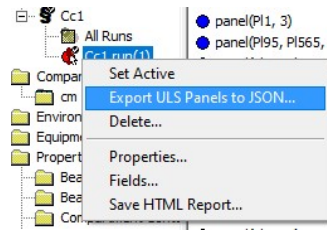
- Press OK to close the dialog.
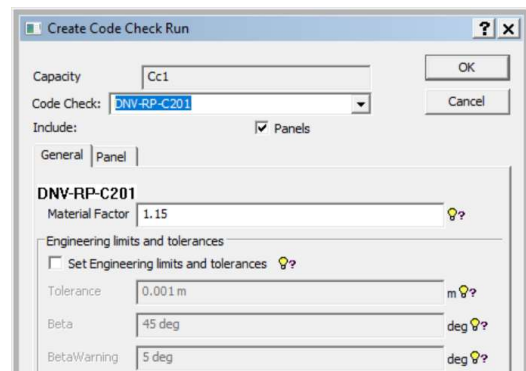
2.5   Export capacity model

Export the capacity model to a JSON file to be used with SesamCore in OneWorkflow.

- RMB click the code check run and and select **Export ULS Panels to JSON**


- Type **col2_ULS** as file name and press save.

The JSON file will be created in the GeniE workspace folder. This file and the structural model file will be used in OneWorkflow to run the ULS code check.

3 PRECOMPUTE

In this chapter, the precomputation step is set up using HydroD. Generation of the precomputed unit response database is covered in chapter 5.

HydroD must be used here to specify the hydrodynamic loading condition which will be common for all DLCs. No analysis is to be performed in HydroD, but a WADAM1.FEM template file must be generated. SesamCore will later use this file to perform precomputation.

- Open the HydroD workspace EMULF_HD_Workspace.hydx located inside the folder HydroD_precompute_setup.
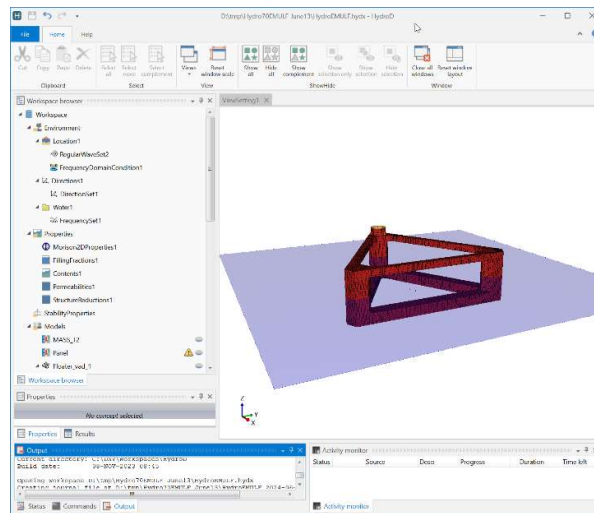
- In the **Workspace browser**, select **Water1** then **FrequencySet1**. In the **Properties** dialog, select the frequencies for which to later precompute diffracted and radiated pressure responses. The provided workspace has preselected 20 equidistant frequencies from 0.1 rad/s to 5 rad/s. This low fidelity was selected to speed up the calculations in the tutorial. For higher fidelity you can reduce the frequency step which should increase the accuracy of the precomputed response curves but which also implies that the precompute step may take up to several hours (recall that this step is performed only once for all relevant DLCs).

- In the **Workspace browser**, select **Analyses** then **ResponseReconstruction** and **WadamPrecomputeTemplate**.

  o In the **Properties** dialog, on the **General** tab, make sure that only **Generate Wadam input data** is selected.
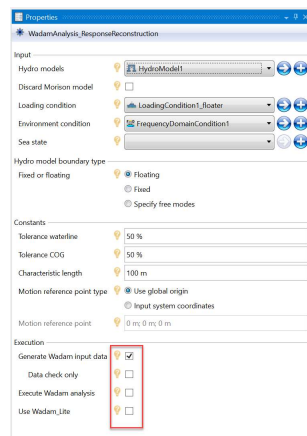
o   In the Properties dialog, on the Load tab, make sure that Load transfer is selected and that all static and dynamic load types are included.
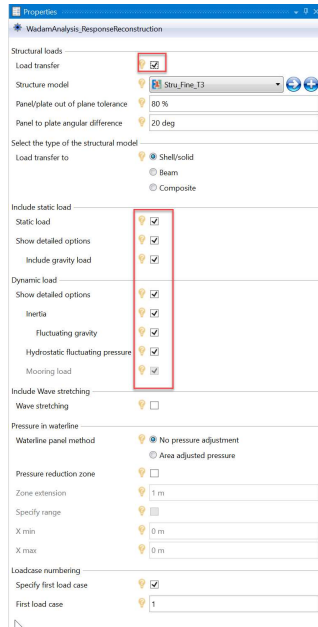
o   Execute the Wadam analysis (right-click the analysis to bring up the menu)
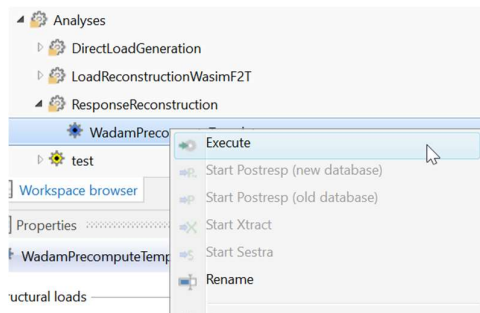
o   Open the workspace folder in Explorer (right-click the analysis to bring up the menu). Observe five files T3.FEM (structure model), T7320.FEM, T7301.FEM, T7302.FEM (panel model), and WADAM1.FEM (Wadam input file). These files are the necessary input to Sesam Core precompute.
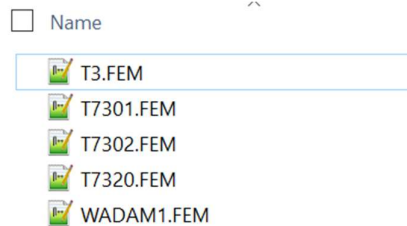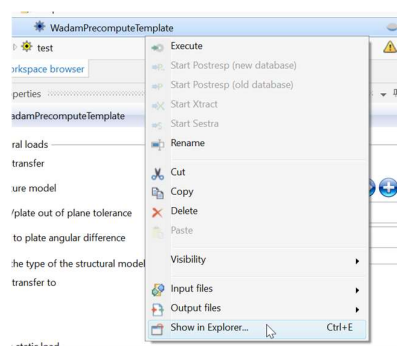
For a comprehensive example of how to set up a Wadam analysis in HydroD, please refer to several examples provided in the HydroD Help menu.

4 HYDRODYNAMIC COEFFICENTS AND COUPLED ANALYSIS

At this point in the workflow a HydroD/Wadam run is necessary to compute the hydrodynamic coefficients, and eventually adjust load cards with regards to compartment filling in the Structural model file. Subsequently, a coupled analysis using either Bladed or Sima must be run to establish the structural kinematics, wave conditions and loads for the structure. For brevity these steps are skipped in this tutorial and the resulting files from a Sima coupled analysis are provided as input to the rest of the workflow.

The skipped steps have been part of Sesam workflows for many years and are documented as follows:

- For information about how to calculate hydrodynamic coefficients and adjust the compartment fillings, please check the HydroD user's manual.

- For information about how to run the coupled analysis in Sima or Bladed please check their respective user's manual.

For access to more comprehensive training material related to these steps please contact Software Support.

5 PERFORMING BUCKLING ANALYSIS IN TIME DOMAIN FOR FLOATING OWT STRUCTURE USING THE RESPONSE RECONSTRUCTION METHOD

This chapter explains how to perform **buckling** analysis in Sesam for a selection of **design load cases (DLCs)** using the response reconstruction method.

Prior to the reconstruction step, a **coupled analysis** using **Sima** has been run. Using the computed structural kinematics (motions), wave elevation and auxiliary forces for mooring and tower interface nodes, the detailed structure response is reconstructed from the precomputed unit response database. **SesamCore** performs the reconstruction in an integrated ULS analysis according to **DNV-RP-C201**.

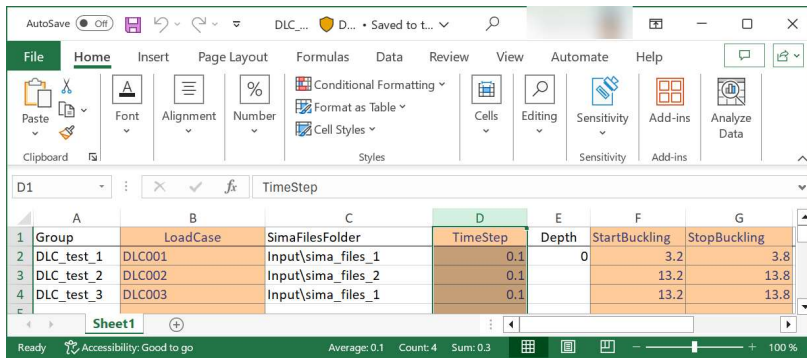The precomputation step is also set up as part of the workflow.

5.1 File structure for the workflow

The Jupyter notebook is provided as FOWT_ULS.ipynb.

In the folder two levels above the Jupyter notebook a folder called *Pythonmodules* includes some functions which facilitate the workflow definition. For example, one module check SesamCore output files to confirm the successful completion of the analyses.

All the necessary files for running the remaining of the workflow are provided, copy them (as well as the Pythonmodule folder) into a suitable folder of your choice, but keep the relative paths intact as those paths are used in the Python script.

Inside the **Workspace** folder there is a spreadsheet DLC_Input.xlsx which contains the description of each DLC. The code in the notebook will be able to automatically read this sheet and generate the tasks accordingly.

The **CommonFiles** folder contains files common to all DLCs such as a template input file for SesamCore, and the exported capacity model .json-file from GeniE.

The **Input** folder contains files that are relevant only to some of the DLCs. In this case, the Sima files inside the folder **sima_files_1** will be used for DLC001 and DLC003, while the Sima files inside the folder **sima_files_2** will be used for DLC002 (as defined in the spreadsheet).

The **Input** folder also contains a subfolder **precompute**, which contains the input files for precompute (among these are the five .FEM files generated in Section 4.1).

5.2   Running the notebook

A compatible reader for the Jupyter notebook is necessary to open it properly. In the illustrations we use Visual Studio Code but there are several others available online. The notebook serves as a documented Python code separated into cells alternating between describing the workflow and presenting the Python code.

Most code cells will begin by importing the necessary libraries for the code in that cell to run. After that it might be followed by a section called USER INPUT where the most common variables are declared. For running this tutorial none of those variables need to be changed. However, for running the same workflow on different models, it will be inside the USER INPUT sections that all relevant variables can be changed accordingly.

In most, if not all, Jupyter notebook readers it will be possible to select whether to run the code cell by cell or run all the code of the notebook at once. For the first run we suggest reading the descriptions and running the code cell by cell from the top to the bottom of the notebook.

About DNV

We are the independent expert in risk management and quality assurance. Driven by our purpose, to safeguard life, property, and the environment, we empower our customers and their stakeholders with facts and reliable insights so that critical decisions can be made with confidence. As a trusted voice for many of the world's most successful organizations, we use our knowledge to advance safety and performance, set industry benchmarks, and inspire and invent solutions to tackle global transformations.

Digital Solutions

DNV is a world-leading provider of digital solutions and software applications with focus on the energy, maritime and healthcare markets. Our solutions are used worldwide to manage risk and performance for wind turbines, electric grids, pipelines, processing plants, offshore structures, ships, and more. Supported by our domain knowledge and Veracity assurance platform, we enable companies to digitize and manage business critical activities in a sustainable, cost-efficient, safe, and secure way.