SESAM TUTORIAL

Sesam

Time History Buckling Analysis of EMULF Delta Floater

Sesam Tutorial

Date: 07 June 2024

Prepared by: Digital Solutions at DNV

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

E-mail sales: digital@dnv.com

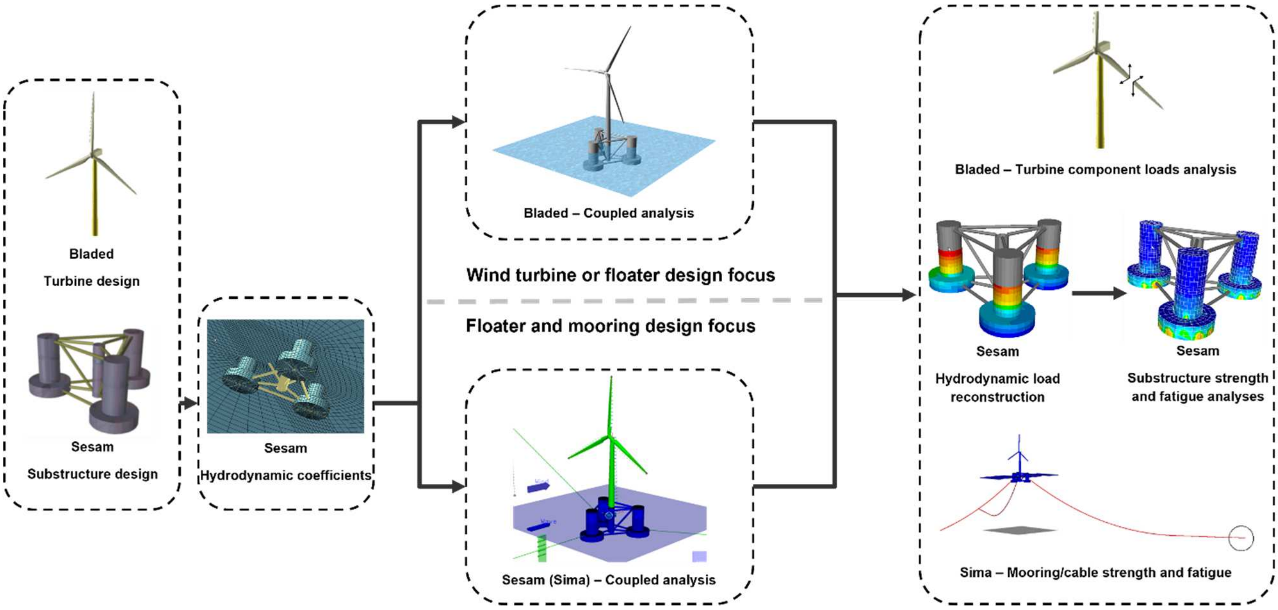
Table of contents

1 INTRODUCTION

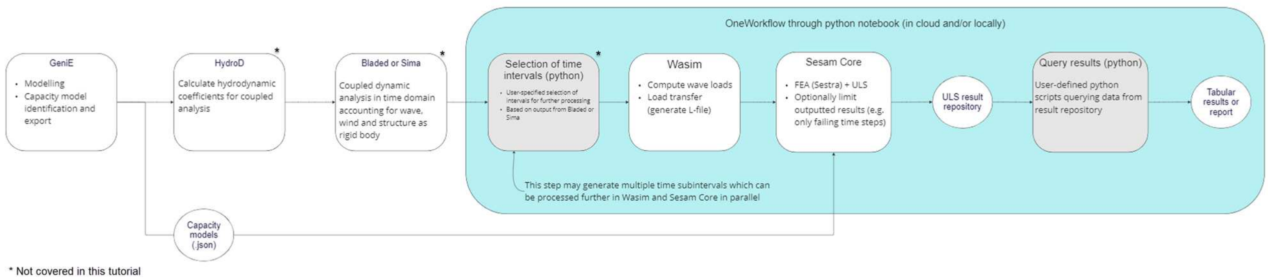
This is a tutorial covering a floating OWT structure time history buckling analysis. The plate verification is done according to the DNV RP-C201 using the stiffener capacity model therein referred to as SCM2.

The following versions or higher are required: GeniE 8.10, Sesam Core V3.0, Wasim 6.5.8.

This document explains how to perform a **buckling** analysis in **Sesam Core** though **OneWorkflow** for a selection of **design load cases (DLC)**. A **coupled analysis** using either **Bladed** or **Sima** has been run in advance to establish the structural kinematics, wave conditions and loads for the structure. Based on that output and mass input from **HydroD**/**Wadam**, **Wasim** is run to do a load mapping onto the FE structural model, then **Sestra** runs the linear structural analysis before **SesamCore** calculates the ultimate limit state according to **DNV-RP-C201**.



A **Jupyter notebook** using **OneWorkflow** covers the **load generation** and **buckling** analysis steps represented by the floater images in the right picture above. This implies running **Wasim**, **Sestra** and **SesamCore** for multiple **DLCs**.

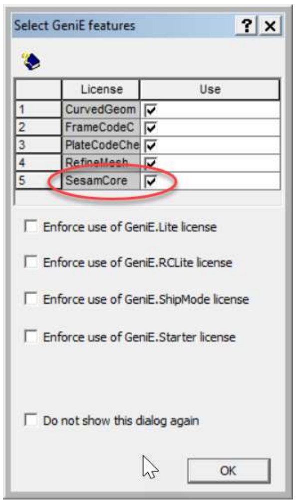


The diagram above shows an example of time domain buckling analysis workflow and the software involved.

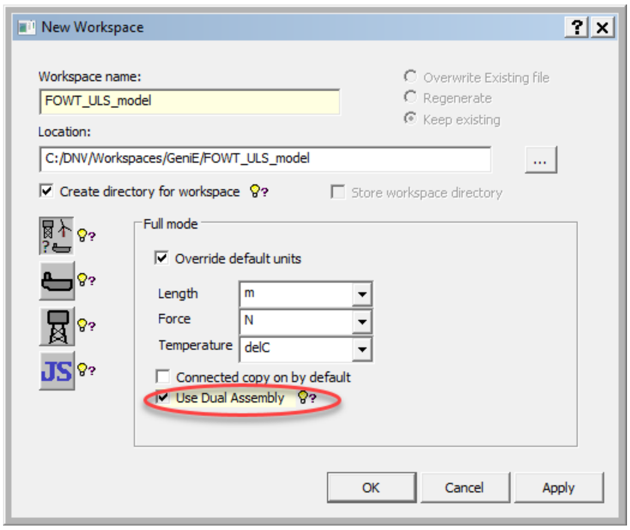
2 DEFINING THE BUCKLING CAPACITY MODEL IN GENIE

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.

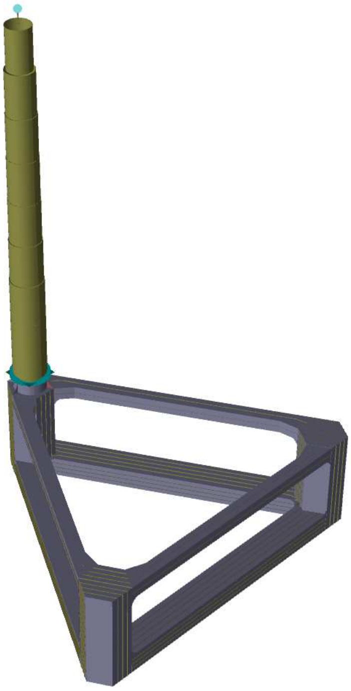
The structural model for this tutorial is the EMULF Delta Floater Model which was designed as part of a JIP.
Importing the model into GeniE
The required fatigue functionality in GeniE requires a GENIE__SCORE license as shown in the Edit > License / features dialog.



- 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 ***FOWT_ULS_model_start.gnx*** from the workshop input files into GeniE.
  - Use *File > Import > Workspace (GNX file)* to import the model file.



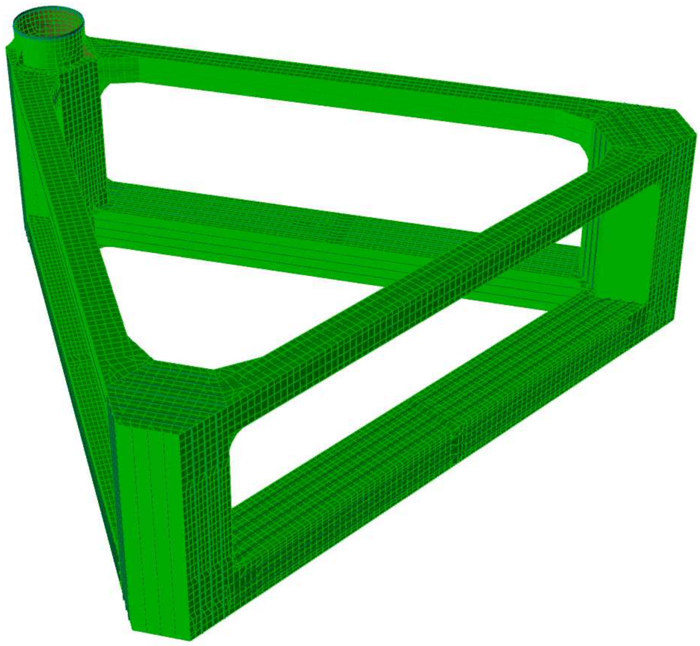


Figure 1 Global model of substructure **Figure 2 Mesh of global model**

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:

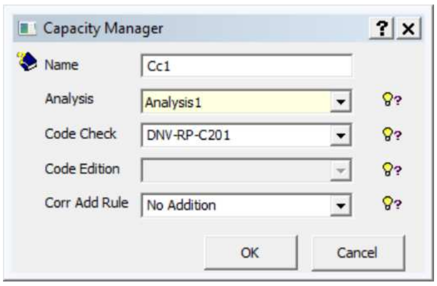
- Go to **File > Export > FEM file**. Save the mesh model as ***FOWT_ULS_T2.FEM***.

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 fatigue input may have to be re-exported too.

2.2 Defining the capacity manager

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

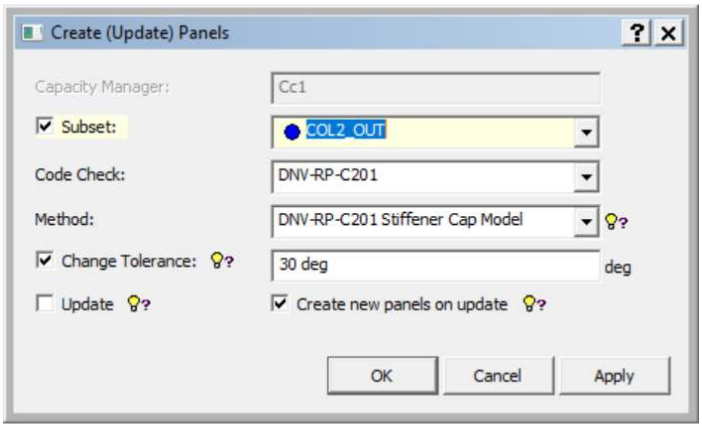
- Select the existing analysis, DNV-RP-C201 code check and no corrosion addition rule.



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**

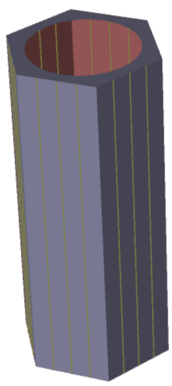


Figure 3 Model of COL2_OUT

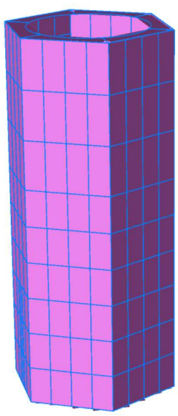
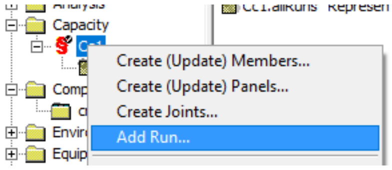


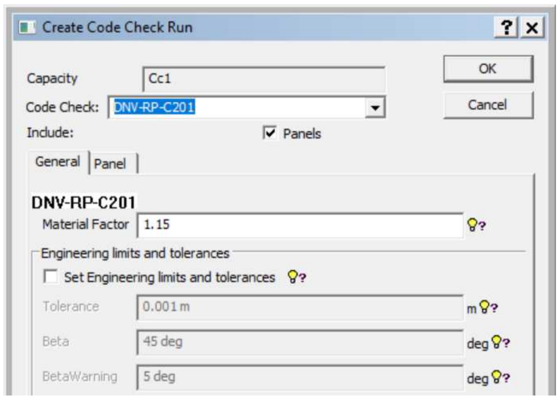
Figure 4 Panels in the capacity model

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



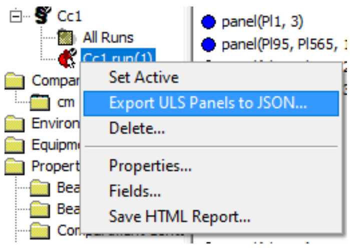
- Confirm that DNV-RP-C-201 is selected for the code check option.
- In the General tab the material factor can be adjusted for other limit states. For ULS the default value of 1.15 is adequate.
- In the Panel tab the end support for the stiffeners can be set to continuous or snipped. This is applied 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 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:

- 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.

4 PERFORMING BUCKLING ANALYSIS IN TIME DOMAIN FOR FLOATING OWT STRUCTURE

This chapter explains how to perform **buckling** analysis in Sesam for a selection of **design load cases (DLCs)**. A **coupled analysis** using **Sima** has been run in advance. Based on that output and mass input from **HydroD**/**Wadam**, **Wasim** is run to do a load mapping onto the FE structural model, then **Sestra** runs the linear structural analysis before **SesamCore** calculates the ultimate limit state according to **DNV-RP-C201**.

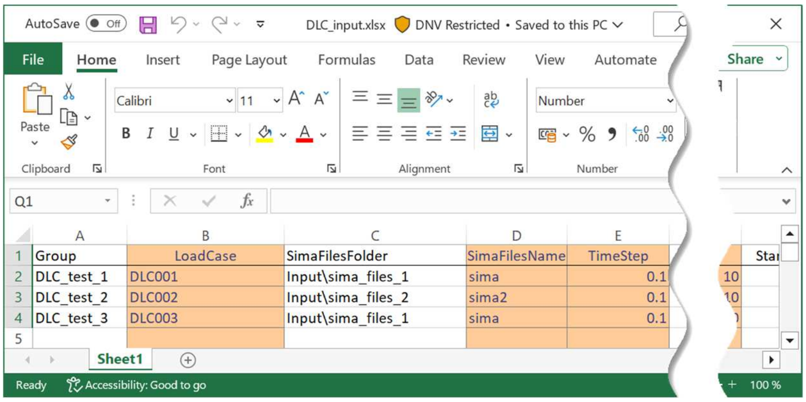
4.1 File structure for the workflow

The Jupyter notebook is provided as FOWT_ULS.ipynb.

In the folder one level above the Jupyter notebook a folder called *Pythonmodules* includes some functions which facilitate the workflow definition. For example, one module will simplify the task creation for Wasim and another module will check Wasim and 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.

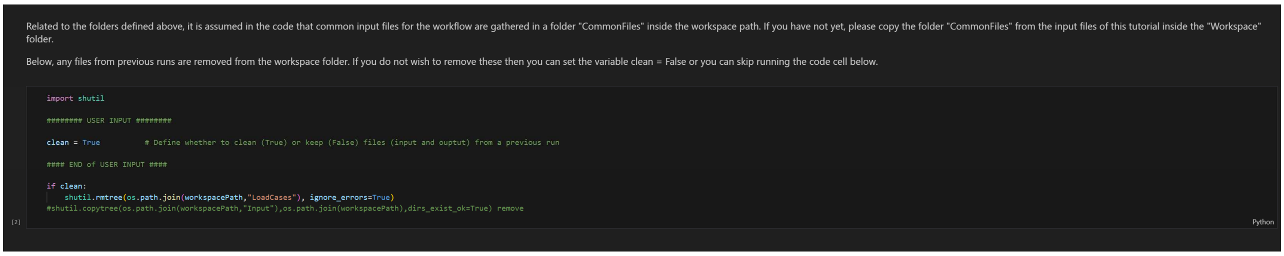


Inside the CommonFiles folder will be the files that are common to all DLCs such as template files from Wasim, Sestra, SesamCore and other files.

Inside the Input folder will be files that are only relevant 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).

4.2 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 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.