

SESAM TUTORIAL

Spar-type Floating Offshore Wind

Parametric Modelling of Spar for GeniE and HydroD Workflow





Sesam Tutorial

Spar-type Floating Offshore Wind: Parametric Modelling of Spar for GeniE and HydroD Workflow

Date: April 2025

Valid from GeniE version 8.12, HydroD 7.2, Postresp 7.2.3, and Sesam Manager 6.7.

Prepared by: Digital Solutions at DNV

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

E-mail sales: digital@dnv.com

© DNV AS. All rights reserved.

This publication or parts thereof may not be reproduced or transmitted in any form or by any means, including copying or recording, without the prior written consent of DNV AS.



Table of contents

INTROE	DUCTION	1
Regardi	ng JS code blocks	2
1	PANEL MODEL CREATION IN GENIE	
1.1	Panel Model – Manual Modelling	3
1.1.1	Creating a new workspace	3
1.1.2	Creating properties	4
1.1.3	Creating guiding points	5
1.1.4	Creating the outer hull (side wall)	6
1.1.5	Creating the outer hull (bottom plate)	7
1.1.6	Applying mesh property	8
1.1.7	Applying wet surface property	8
1.1.8	Adding the plates to a named set	9
1.1.9	Creating analysis to generate panel model mesh	9
1 1 11	Generating and exporting the panel model mesh	10
4.0	Denel Medel Autometed Medelling	10
1.2	Pariel Model – Automateu Modelling Design parameters	13
1.2.1	Defining user defined parameters	13
1.2.3	Pre-calculating variables	14
1.2.4	Modifying the manual modelling scripts	14
1.2.5	Running the parametric script and checking the panel model	15
2	HYDRODYNAMIC ANALYSIS IN HYDROD	
2.1	Hydrodynamic Analysis – Manual Setup	17
2.1.1	Creating a new workspace	17
2.1.2	Creating a Wadam wizard	17
2.1.3	Defining environmental data	18
2.1.4	Defining a hydro model	20
2.1.5	Creating and executing Wadam analysis	22
2.1.6	Checking the results with Postresp	24
2.2	Hydrodynamic Analysis – Automated Setup	27
2.2.1	Design parameters	27
2.2.2	Denning user denneu parameters Pro coloulating variables	27
2.2.3	Modifying the manual setup scripts	27
2.2.5	Running the parametric script and checking the results	28
3	AUTOMATING GENIE AND HYDROD WITH SESAM MANAGER	
3.1	Preparing the Parametric Scripts	30
3.1.1	Preparing the GeniE script	30
3.1.2	Preparing the HydroD script	30
3.1.3	Preparing the Postresp script	31
3.1.4	Preparing Sesam Manager Post Execute Script for Postresp	31
3.2	Setting Up Sesam Manager	32
3.2.1	Creating a new Job	32
3.2.2	Copying the parametric modelling scripts	32
3.2.3	Creating a Sequence folder	32
3.2.4	Creating a GeniE activity	33
3.2.5	Creating a HydroD activity	34
3.2.6	Creating a Postresp activity	35
১.∠./ ২০৪	Creating the rest of the Job Running all analyses	30 27
0.2.0	ivuining all analyses	57



INTRODUCTION

This tutorial demonstrates how to create a spar-type floating offshore wind (FOW) floater and automate the process using GeniE's and HydroD's parametric modelling capabilities. The goal is to introduce a straightforward approach for converting a model into parametric scripts, enabling parametric studies for hydrodynamic analysis. The resulting data can then be used for motion analysis in Sima or Bladed.

Before starting this tutorial, you should already be comfortable working with GeniE and HydroD.

The approach presented here is intentionally simple: we will modify portions of the scripts that are automatically generated during manual modelling. However, note that there is no single, definitive way to perform parametric modelling. Many alternative approaches may also be valid—and in some cases, more efficient—than the one described here.

For this tutorial, the design is based on the spar-type floater shown below:



The tutorial is organized into three chapters:

- 1. Panel Model Creation in GeniE Building the floater hull's panel model and exporting it for use in HydroD.
- 2. **Hydrodynamic Analysis in HydroD** Importing the panel model from GeniE, setting up relevant parameters, and performing frequency-domain hydrodynamic analysis.
- 3. Automation with Sesam Manager Combining GeniE and HydroD operations into a single automated workflow using Sesam Manager.

Each of the first two chapters is further divided into two sub-chapters:

- 1. Manual Modelling Step-by-step instructions to manually set up the model and workspace.
- 2. Automated Modelling Using JS scripts, recorded during the manual steps, to create parametric modelling scripts.



Disclaimer:

This tutorial is intended solely for training purposes to demonstrate how to use Sesam for automating the modelling of a FOW floater. It is not intended for use in real-world projects or decision-making. Users are responsible for validating all parameters and ensuring compliance with project-specific requirements and standards.

Regarding JS code blocks

Each manual modelling step is accompanied by its corresponding JS script. For example:



These JS scripts are automatically logged in the Command Line window of GeniE and HydroD when you perform modelling operations as shown below. Pay close attention to them, as they will be used in the Automated Modelling subchapter to build your parametric scripts.



GeniE



You can also find the complete JS scripts after your manual modelling in the GeniE or HydroD workspace folders:

- If the workspace has been saved, the file name will be named **<workspace_name>.js**.
- If the workspace has not been saved, the file name will be <workspace_name>_<datetime>.js.

Note: The logged JS commands may vary depending on the order of operations or if any steps are skipped. Rather than copying them exactly, focus on understanding the core logic—this tutorial is about learning to build your own automation functions.



1 PANEL MODEL CREATION IN GENIE

1.1 Panel Model – Manual Modelling

In this first chapter, we will create the outer hull of the floater in GeniE. From the outer hull, we will export the panel model file (Panel_T1.FEM) to be used in the frequency-domain hydrodynamic analysis in HydroD.

1.1.1 Creating a new workspace

Create a new GeniE workspace with Full Mode and Use Dual Assembly selected.

New Worksp	bace				?	×
Workspace nan ManualModellir Location:	ne:			Overwrite Existi Regenerate Keep existing	ng file	
C:/DNV/Works	paces/GeniE/Manu	alModelling				
🗹 Create direc	ctory for workspace	89	Store	workspace directory		
日本しの	Full mode					
?= *?	Override d	efault units				
<u>بہ</u> ص	Length	m	~			
	Force	N	~			
R	Temperature	delC	~			
JS 83	Connected	l copy on by o ssembly 8	default ?			
			ОК	Cancel	Appl	y

The following JS commands will be logged in the **Command Line** window of GeniE (by default, located at the bottom of the screen):

1	<pre>GenieRules.Compatibility.version = "V8.12-03";</pre>
2	<pre>GenieRules.Tolerances.useTolerantModelling = true;</pre>
3	<pre>GenieRules.Tolerances.angleTolerance = 2 deg;</pre>
4	<pre>GenieRules.Meshing.autoSimplifyTopology = true;</pre>
5	<pre>GenieRules.Meshing.eliminateInternalEdges = true;</pre>
6	<pre>GenieRules.BeamCreation.DefaultCurveOffset = ReparameterizedBeamCurveOffset();</pre>
7	<pre>GenieRules.Geometry.AssemblyType = DualAssembly;</pre>
8	<pre>GenieRules.Transformation.DefaultConnectedCopy = false;</pre>

Note: The dual assembly format is recommended when modelling a floater substructure that combines shell and beam elements. Although this tutorial does not require beam elements for the panel model, they may be needed when developing a structural model later.

If you mistakenly create the workspace without selecting Use Dual Assembly, you can change the workspace type afterward using the following JS command:

9 GenieRules.Geometry.AssemblyType = DualAssembly;

Note: If you are unable to copy text from this PDF, a copy of the JS script can be found in the input_files folder under: Chapter 1\Ch1-1_PanelManualModelling.js.



1.1.2 Creating properties

Create the following properties:

1. Create material property

Linear Isotropic Material			? ×
t <u>-</u> \$355	•		
	Yield	355000000 Pa	Pa
σ	Density	7850 kg/m^3	kg/m^3
yield-	Young	2.1e+11 Pa	Pa
/soung	Poisson	0.3	
l →g	Thermal	1.2e-05 delC^-1	delC^-1
	Damping	0.03 N*s/m	N*s/m
ſ	Tensile		Pa
	0	Close	Apply

2. Create mesh density property



3. Create **thickness** property

Thickne	ss		? ×
Name	• Th25		▼ 8?
Thickness	0.025 m		m % ?
		ОК	Close Apply

```
12 Th25 = Thickness(0.025 m);
```



4. Create wet surface property

	New Wet Surface	<u>? x</u>
	Name WS1	▼ 8?
	OK Ca	ncel Apply
13	WS1 = WetSurface();	

Hint: Right-click the appropriate folder in the browser and select New <PropertyType>. For example, to create a new material property, right-click the **Properties > Materials** folder and select **New Material**.

Set default material to S355 and default thickness to Th25.

<no sect<="" th=""><th>ion> _ [ℓ– \$\$355</th><th>▼ ● Th25</th><th> Customizable</th><th>✓ <no active="" set=""></no></th><th>▼ <no <no="" asymmetric="" slc="" slot="" symmetric="" th="" ▼="" ▼<=""></no></th></no>	ion> _ [ℓ– \$\$355	▼ ● Th25	Customizable	✓ <no active="" set=""></no>	▼ <no <no="" asymmetric="" slc="" slot="" symmetric="" th="" ▼="" ▼<=""></no>
14	S355.setDefault	();			
15	Th25.setDefault	();			

1.1.3 Creating guiding points

Create the following guide points with **Guiding Geometry** > **Guide Point Dialog**:

- 00-120 m
- 00-12 m
- 00-4 m
- 0010m

Ž

_(0 m,0 m,-120 m)

(0 m,0 m,10 m)

(0 m,0 m,−4 m) (0 m,0 m,−12 m)

16 Point1 = Point(0, 0, -120); 17 Point2 = Point(0, 0, -12); 18 Point3 = Point(0, 0, -4); 19 Point4 = Point(0, 0, 10);



1.1.4 Creating the outer hull (side wall)

Activate **Reference Point Modelling** by clicking the 🖻 icon on the toolbar. This enables command logging to refer to the object names rather than their specific values.

Create outer hull shell using **Structure > Shells with Circular Sections > Circular Cone/Cylinder**. When prompted, specify the following inputs in order:

- 1. Starting point
- 2. Radius at the starting point
- 3. Finish point
- 4. Radius at the finish point

Structure Loads Mesh & Analy	sis	Results Help
Beams and Piles	×	🖓 ଡ଼ ବା ଦେ ମ 🌢 ଛୁ ବ 🖬 🄇
Flat Plates	►	
Shells with Circular Sections	►	🖉 Circular Cone/Cylinder
Free Form Shells	≁	a Elliptic Cone/Cylinder
Shells from Point-Nets	×	Sphere
Hull Design Tools	×	📾 Revolve a Profile around an Axis
Features	•	🌈 Pipe



20	<pre>Pl1 = CreateShellCircularConeCylinder(Point1, 4.7, Point2, 4.7, 0, 360);</pre>	
21	<pre>Pl2 = CreateShellCircularConeCylinder(Point2, 4.7, Point3, 3.25, 0, 360);</pre>	
22	<pre>Pl3 = CreateShellCircularConeCylinder(Point3, 3.25, Point4, 3.25, 0, 360);</pre>	;

Note: If you did not activate the **Reference Point Modelling**, the logged commands will use explicit values like Point(0 m,0 m,-120 m) instead of referencing the object name, such as Point1.



1.1.5 Creating the outer hull (bottom plate)

Create a circle guiding curve with Guiding Geometry > Conic Sections > Circle from Center, Normal and Radius.

Click the **bottom guide point**, then the **top guide point** to define the normal vector.

Finally, enter the radius: 4.7 m.



Select the circular guide curve, right-click it, and choose **Cover Curves** to create the bottom plate. Note that the *bottom* surface (displayed in red) is facing outward.



Select the bottom plate, right-click it, and choose **Flip Normal**. This sets the *front* surface (shown in grey) as the wetted surface.



```
25 Pl4.flipNormal();
```



1.1.6 Applying mesh property

Select all the outer hull plates and apply the mesh property **PanelMeshDensity** as follows: Navigate to **Properties > Mesh**, right-click **PanelMeshDensity**, and choose **Apply Mesh Properties to Selection** while the plates are selected.



26	<pre>Pl1.meshDensity = PanelMeshDensity;</pre>
27	<pre>Pl2.meshDensity = PanelMeshDensity;</pre>
28	<pre>Pl3.meshDensity = PanelMeshDensity;</pre>
29	<pre>Pl4.meshDensity = PanelMeshDensity;</pre>

1.1.7 Applying wet surface property

Apply the wet surface property **WS1** to the front side of all plates: Navigate to **Properties > Wet Surface**, right-click **WS1**, and select **Apply Wet Surface Properties to Selection** while the plates are selected. In the dialog, choose **Front**, then click **OK**.



30	<pre>Pl1.front.WetSurface = WS1;</pre>	
31	<pre>Pl2.front.WetSurface = WS1;</pre>	
32	<pre>Pl3.front.WetSurface = WS1;</pre>	
33	<pre>Pl4.front.WetSurface = WS1;</pre>	

| Sesam Tutorial - Parametric Modelling of Spar for GeniE - HydroD Workflow |



1.1.8 Adding the plates to a named set

Select *PI1* to *PI4*, right-click one of them and select **Named set**. Add them to a new regular set named **PanelModelSet**. This named set will be used to define the subset of the model for panel model meshing.



34	<pre>PanelModelSet = Set();</pre>
35	<pre>PanelModelSet.add(Pl1);</pre>
36	<pre>PanelModelSet.add(Pl2);</pre>
37	<pre>PanelModelSet.add(Pl3);</pre>
38	<pre>PanelModelSet.add(Pl4);</pre>

1.1.9 Creating analysis to generate panel model mesh

Create a new analysis by right-clicking **Analysis > Activities** and selecting **New Analysis**.

ManualModelling1		Name	Description
Capaci Compa Compa	New Analysis Field Save HTML Rep	oort	



Name the analysis **GeneratePanelModel** and remove all activities, as we only need to mesh the concept model.

Create Ac	tivity				? ×	
Name	GeneratePanel	Model				
Lod Lod Aut Available Mes Hull Locc Wa Line Pile Ten Ind Loc	k concepts after comatically impor e activities shing Girder Load Adj al Analysis ve Load Activity ear Structural An Soil Analysis usion/Compressie ep. Tank Coupir d Results	meshing t global loadc uster alysis on Analysis ig Analysis	99 ases			
Ear	thquake	00	Use Sestra IU	9.5		
Length	n analysis units	- ¥3				
Force	N	~				
Tempera	ture delC	~				
				ОК	Cancel	

- 39 GeneratePanelModel = Analysis(true);
- 40 GeneratePanelModel.add(MeshActivity());
- 41 GeneratePanelModel.add(LoadResultsActivity());
- 42 GeneratePanelModel.setActive();

1.1.10 Creating a dummy hydro load case

Next, create a dummy hydro load case to allow HydroD to recognize the wetted part of the model. Right-click **Analysis > Load Cases** and select **New Loadcase**.



Select Dummy Hydro Pressure option and set Wet Surface to WS1.

Insert Load Case	?	×
Name: LC1		
Oummy Hydro Pressure		-
Wet Surface WS1 ~		
OK Cancel		ply

43 LC1 = DummyHydroLoadCase(WS1);



1.1.11 Generating and exporting the panel model mesh

Open activity monitor (Alt + D), right-click Meshing activity, and select Edit Mesh Activity.

Activity Monitor				?	×
Þ			Start	Can	cel
Dournal activity executions					
	Duration	Chabus	Concrato	Inout	
	Duration	Status	Generate .	Input	
Mo 1 - GeneratePanelModel - An	a Os	Not Started			
V 1.1 - Meshing (Conditional	Ed Mesh /	Activity ted			
✓ 1.1.1 - Delete loads	TIN	ted			
I.1.2 - Generate loads [™]	US	NUL Started			
1.1.3 - Delete mesh	0s	Not Started			
 1.1.4 - Update mesh 	0s	Not Started			
R 1.2 - Load Results	0s	Not Started			

Set the **Regenerate mesh option** to **Always Regenerate Mesh**, and select **Mesh Subset** to **Include** with the **PanelModelSet** selected as the subset.

Mesh activity	? ×
Meshing Rules Regenerate mesh option Always Regen	erate Mesh v 8?
Export beams as members Smart load combinations &? Include loads on mesh &? Write FEM file &?	 □ First level combinations as BSELL ☑ Parallelize load application ☑ Parallelize mesh generation ☑ ♀
Override Global Superelement Data Top Superelement Type 1 Superelement Type 1	
Mesh Priority <none> Mesh Subset Include Exclude Subset Forget all existing keep mesh flags</none>	Kero mesh of previous subset
Pile boundary condition Pile Soil Interaction Subset <pre></pre>	♥?
Element numbers from beam names	OK Cancel Apply

44	<pre>GeneratePanelModel.step(1).subset = PanelModelSet;</pre>
45	<pre>GeneratePanelModel.step(1).regenerateMeshOption = anAlwaysRegenerateMesh;</pre>



Tick the Journal activity executions checkbox and click Start to run the analysis to perform the meshing.



Check if the dummy hydro loads (displayed as orange vectors) are correctly applied to the front side of the outer hull plates. Also, inspect the mesh quality for any issues such as sharp edges, sliver elements, etc.



Finally, select **File > Export > FEM File**, tick **Journal export operation** option, and export the FEM file as **Panel_T1.FEM**.

The journal export operation logs the JS commands associated with the export process.



With this, the panel model file (Panel_T1.FEM) is created. Verify the location of this file and keep it for later use.



1.2 Panel Model – Automated Modelling

In this sub-chapter, we will modify the JS commands from the manual modelling sub-chapter to create a simple parametric script. This approach is one of the simplest ways to generate a parametric script.

You can find the complete JS script from your manual modelling session in its corresponding workspace folder. The script file is named after the workspace, for example, if you used the "ManualModelling" as the workspace name, the file will be located in the ManualModelling folder as **ManualModelling.js** or **ManualModelling_<datetime>.js**.

Note: If you are unable to locate it, you may use the Ch1-1_PanelManualModelling.js located in the input_files\Chapter 1 folder.

1.2.1 Design parameters

The design parameters are as follows:



In addition, we will define the mesh density (**MESH_DENSITY**) and the panel model file name prefix (**PANEL_MODEL_PREFIX**) as additional parameters. The prefix helps differentiate between files and ensures that the file names do not conflict with each other.

1.2.2 Defining user defined parameters

First, we need to define the parameters as variables. Let's define them at the start of the script:

```
1
       // USER DEFINED PARAMETERS
2
       UPPER_LENGTH = 14;
3
       MIDDLE LENGTH = 8;
       BOTTOM_LENGTH = 108;
4
5
       UPPER_DIAMETER = 6.5;
       BOTTOM_DIAMETER = 9.4;
6
7
       BOTTOM_Z = -120;
8
       MESH_DENSITY = 0.75;
9
       PANEL_MODEL_PREFIX = "Panel1_";
```

Sesam Tutorial - Parametric Modelling of Spar for GeniE – HydroD Workflow |



1.2.3 Pre-calculating variables

To avoid recalculating the same values multiple times, we will pre-calculate the following two variables:

```
11 // PRE-CALCULATION OF DERIVED PARAMETERS
12 upperRadius = UPPER_DIAMETER/2;
13 bottomRadius = BOTTOM_DIAMETER/2;
```

1.2.4 Modifying the manual modelling scripts

Finally, take the scripts generated in the previous sub-chapter and make slight modifications. The modifications are highlighted in red, with the design parameters **bolded**.

```
15
       // MAIN FUNCTIONS
16
       GenieRules.Compatibility.version = "V8.12-03";
       GenieRules.Tolerances.useTolerantModelling = true;
17
18
       GenieRules.Tolerances.angleTolerance = 2 deg;
       GenieRules.Meshing.autoSimplifyTopology = true;
19
       GenieRules.Meshing.eliminateInternalEdges = true;
20
       GenieRules.BeamCreation.DefaultCurveOffset = ReparameterizedBeamCurveOffset();
21
22
       GenieRules.Geometry.AssemblyType = DualAssembly;
       GenieRules.Transformation.DefaultConnectedCopy = false;
23
       GenieRules.Geometry.AssemblyType = DualAssembly;
24
       S355 = MaterialLinear(355000000 Pa, 7850 kg/m^3, 2.1e+11 Pa, 0.3, 1.2e-05 delC^-1, 0.03
25
       N*s/m);
       PanelMeshDensity = MeshDensity(MESH_DENSITY);
26
       Th25 = Thickness(0.025 m);
27
       WS1 = WetSurface();
28
29
       S355.setDefault();
       Th25.setDefault();
30
31
       Point1 = Point(0,0,BOTTOM Z);
       Point2 = Point1 + Vector3d(0,0,BOTTOM LENGTH);
32
33
       Point3 = Point2 + Vector3d(0,0,MIDDLE_LENGTH);
34
       Point4 = Point3 + Vector3d(0,0,UPPER_LENGTH);
       Pl1 = CreateShellCircularConeCylinder(Point1, bottomRadius, Point2, bottomRadius, 0,
35
       360);
36
       Pl2 = CreateShellCircularConeCylinder(Point2, bottomRadius, Point3, upperRadius, 0,
       360):
37
       Pl3 = CreateShellCircularConeCylinder(Point3, upperRadius, Point4, upperRadius, 0,
       360);
       Curve1 = CreateCircleFromPlaneAndRadius(Point1, Vector3d(0,0,1), bottomRadius);
38
39
       P14 = CoverCurves(Curve1);
       Pl4.flipNormal();
40
41
       Pl1.meshDensity = PanelMeshDensity;
42
       Pl2.meshDensity = PanelMeshDensity;
43
       Pl3.meshDensity = PanelMeshDensity;
44
       Pl4.meshDensity = PanelMeshDensity;
45
       Pl1.front.WetSurface = WS1;
       Pl2.front.WetSurface = WS1;
46
       Pl3.front.WetSurface = WS1;
47
```

Sesam Tutorial - Parametric Modelling of Spar for GeniE - HydroD Workflow |



48	Pl4.front.WetSurface = WS1;
49	<pre>PanelModelSet = Set();</pre>
50	PanelModelSet.add(Pl1);
51	PanelModelSet.add(Pl2);
52	PanelModelSet.add(Pl3);
53	PanelModelSet.add(P14);
54	GeneratePanelModel = Analysis(true);
55	<pre>GeneratePanelModel.add(MeshActivity());</pre>
56	<pre>GeneratePanelModel.add(LoadResultsActivity());</pre>
57	GeneratePanelModel.setActive();
58	<pre>LC1 = DummyHydroLoadCase(WS1);</pre>
59	<pre>GeneratePanelModel.step(1).subset = PanelModelSet;</pre>
60	<pre>GeneratePanelModel.step(1).regenerateMeshOption = anAlwaysRegenerateMesh;</pre>
61	<pre>SimplifyTopology();</pre>
62	GeneratePanelModel.execute();
63	<pre>ExportMeshFem1 = ExportMeshFem();</pre>
64	<pre>ExportMeshFem1.DoExport(PANEL_MODEL_PREFIX + "T1.FEM");</pre>

Save the modified parametric script file.

Note: A completed JS file, Ch1-2_PanelAutomatedModelling.js, is provided in the input_files\Chapter 1 folder.

1.2.5 Running the parametric script and checking the panel model

Create a new GeniE workspace and run the script by selecting File > Read Command File. In the next window, browse for the parametric script (or the Ch1-2_PanelAutomatedModelling.js) and click Open.



The model and the mesh will be created in sequence. After the operations are complete, verify the following:

- The dimensions
- The dummy hydro loads
- The mesh quality

Additionally, ensure that **Panel1_T1.FEM** has been outputted correctly.





We can experiment by changing some design parameters and observing the results.

Table 1 Some design cases for experiment

Design Parameter	Case 1	Case 2	Case 3
UPPER_LENGTH	14	14	14
MIDDLE_LENGTH	8	8	8
BOTTOM_LENGTH	108	64.92667	46.81687
UPPER_DIAMETER	6.5	6.5	6.5
BOTTOM_DIAMETER	9.4	12	14
BOTTOM_Z	-120	-76.9267	-58.8169

Case 1













2 HYDRODYNAMIC ANALYSIS IN HYDROD

2.1 Hydrodynamic Analysis – Manual Setup

In this chapter, we will set up and execute the frequency-domain hydrodynamic analysis using the **Panel_T1.FEM** file generated in the previous chapter.

2.1.1 Creating a new workspace

Start HydroD and create a new workspace. To open the New workspace dialog, select File > New.

-		×
HydroManualModelling		
C:\DNV\Workspaces\HydroD	Brow	vse
C:\DNV\Workspaces\HydroD\HydroManualModelling\		
Default* ~	Set de	efault
Metric (m, kg)*	Set de	efault
	Brow	vse
ОК	Cance	el
	HydroManualModelling C:\DNV\Workspaces\HydroD C:\DNV\Workspaces\HydroD\HydroManualModelling\ Default*	- C HydroManualModelling C:\DNV\Workspaces\HydroD Brow C:\DNV\Workspaces\HydroD\HydroManualModelling\ Default*

The following JS commands will be logged in the **Commands** window of HydroD (by default, located at the bottom of the screen, grouped with Output and Status tabs):



2.1.2 Creating a Wadam wizard

Create a **Wadam wizard** from browser with settings as shown below.



DNV	,		
💙 WadamWizard1		Roll damping	፼ □
Condition type	Frequency domain condition Deterministic condition	Stochastic roll damping Roll GZ curve Wave spectrum	 ♀ ♀ ♀ ● Bretschneider wave spectru
Load transfer			Jonswap wave spectrum Torsethaugen wave spectru
Panel model		Second order free surface mod	iel 💡 🗌
Morison model		Damping sheets	💡 🗖
Morison 2D properties	💡 🔲	Fixed lids	💡 🗆
Anchor properties	@	Load Cross Sections	💡 🗆
Morison 3D properties	0	Wadam offbody points	💡 🗆
		Damping matrix	💡 🗆
ILP properties	4	Critical damping matrix	💡 🗆
Drag linearization	Ŷ	Quadratic damping matrix	💡 🗆
Linearization option	💡 💿 Stochastic drag	Restoring matrix	💡 🗆
	 Drag by wave height 	Compartments	

184	WadamWizard1 = new WadamWizard(Wizards, "WadamWizard1");
185	WadamWizard1.PanelModel = true;

Select the wizard in the browser and select **Execute** from the right-click menu. Click the **Step** button to create the relevant concept.



2.1.3 Defining environmental data

Step 1-2: Create a Directions folder and a DirectionSet. Give directions from 0 deg to 360 deg in steps of 30 deg.

は、DirectionSet1						
	From	То	Step			
1	0 deg	360 deg	30 deg			
+						

186 Directions1 = new Directions(Environment, "Directions1"); 187 DirectionSet1 = new DirectionSet(Directions1, "DirectionSet1"); 188 DirectionSet1.Items.Add(new AngleInterval()); 189 DirectionSet1.Items[0].To = 360 deg; 190 DirectionSet1.Items[0].Step = 45 deg; 191 DirectionSet1.Items[0].Step = 15 deg; 192 DirectionSet1.Items[0].Step = 30 deg;

| Sesam Tutorial - Parametric Modelling of Spar for GeniE - HydroD Workflow |



Step 3-4: Create a Water folder and a FrequencySet.

Select Frequency and specify from 0.2 rad/s to 2 rad/s in steps of 0.05 rad/s.

∑ FrequencySet1		
Type Frequency set type 💡 🔵 Pe 🔵 W ● Fre	eriod ave length equency	
pecify value		
From	То	Step
1 0.2 rad/s	2 rad/s	5E-02 rad/s
+		

193	Water1 = new Water(Environment, "Water1");
194	<pre>FrequencySet1 = new FrequencySet(Water1, "FrequencySet1");</pre>
195	<pre>FrequencySet1.WavePeriodItems.Add(new TimeInterval());</pre>
196	<pre>FrequencySet1.WavePeriodItems[0].To = 30 s;</pre>
197	<pre>FrequencySet1.WavePeriodItems[0].From = 5 s;</pre>
198	<pre>FrequencySet1.WavePeriodItems[0].Step = 5 s;</pre>
199	<pre>FrequencySet1.Type = FrequencySetType.Frequency;</pre>
200	<pre>FrequencySet1.WaveFrequencyItems.Add(new FrequencyInterval());</pre>
201	<pre>FrequencySet1.WaveFrequencyItems[0].From = 0.2 rad/s;</pre>
202	<pre>FrequencySet1.WaveFrequencyItems[0].To = 0.2 rad/s;</pre>
203	<pre>FrequencySet1.WaveFrequencyItems[0].To = 2 rad/s;</pre>
204	<pre>FrequencySet1.WaveFrequencyItems[0].Step = 5E-02 rad/s;</pre>

Step 5: In Location, set the water depth to 320 m. Keep the default settings for the other items.

💼 Location1			
۲			
Location			
Gravity	9	9.8066 m/s^2	
Water depth	Ŷ	320 m	
Water			
Density	Ŷ	1025 kg/m^3	
Kinematic Viscosity	Ŷ	1.19E-06 m^2	/s
Air			
Density	Ŷ	1.222 kg/m^3	
Kinematic Viscosity	Ŷ	1.462E-05 m^2/s	

205	<pre>Location1 = new Location(Environment,</pre>	"Location1");
206	Location1.WaterDepth = 320 m;	



Step 6: Combine the *DirectionSet1* and the *FrequencySet1* into a Frequency domain condition.

207	<pre>FrequencyDomainCondition1 = new FrequencyDomainCondition(Location1,</pre>
	<pre>"FrequencyDomainCondition1");</pre>
208	<pre>FrequencyDomainCondition1.DirectionSet = DirectionSet1;</pre>
209	<pre>FrequencyDomainCondition1.FrequencySet = FrequencySet1;</pre>

2.1.4 Defining a hydro model

Step 7-8: Create a HydroModel. Note that the values for AP and FP will not be used in this analysis.

In the ElementModel1, browse to the Panel_T1.FEM file outputted in Chapter 1.

Properties			~	ą.	\times
ElementModel1					
Source File name Copy to workspace	ଡ଼ ଡ଼	es\Workshop_Spar_repository\Panel_T1.FEN	1		
Create from section model	9				

210	HydroModel1 = new HydroModel(HydroModels, "HydroModel1");
211	<pre>ElementModel1 = new ElementModel(Models, "ElementModel1");</pre>
212	<pre>ElementModel1.FileName =</pre>
	<pre>"C:\\DNV\\Workspaces\\Workshop_Spar_repository\\Panel_T1.FEM";</pre>



Step 9: Select ElementModel1 as the panel model.

PanelModel1	
Source	
Element model 💡 🛃 ElementModel1 🗸 🗸 🔂	
Symmetry in XZ 💡 🗌	
Symmetry in YZ 💡 🗌	
Translation 💡	
<pre>213 PanelModel1 = new PanelModel(HydroModel1, "PanelModel1");</pre>	

214 PanelModel1.ElementModel = ElementModel1;



In the **Style** tab, check the boxes for **Show edges** and **Show hydro pressure arrows**. Verify that the hydro pressure arrows are pointing inward, indicating the wetted surface is defined properly.

PanelModel1	
Symmetry Show symmetric parts	💡 🗹
Plates Show plates	💡 🖌
Shrink	
Show labels	💡 🗆
Show edges	💡 🔽
Solids	
Show solids	· ·
Shrink	💡 🗆
Show labels	💡 🗆
Show edges	💡 🗖
Hydro pressure Show hydro pressure arrow	ws 💡 🗸
Pressure panels Model Pressure Panels Di	pole Elements Style

215 PanelModel1.ShowPlateEdges = true; 216 PanelModel1.ShowHydroPressureArrows = true;

Step 10-11: Create a **LoadingConditions** folder and a **Loading condition**. Specify 0 m as waterline and zero trim and heel angles.

📥 LoadingCondition1				
Environment Location	Ŷ	📤 Location1 🗸 🌍 🚭		
Specify By —				
Туре	ę	Waterline & Trim		
		O Drafts		
Waterline & Ti	rim			
Waterline Z	Ŷ	0 m		
Trim angle	9	0 deg		
Drafts				
Draft AP	Ŷ	0 m		
Draft FP	Ŷ	0 m		
Heel				
Heel angle	Ŷ	0 deg		

217	LoadingConditions1 = new LoadingConditions(HydroModel1, "LoadingConditions1");
218	<pre>LoadingCondition1 = new LoadingCondition(LoadingConditions1, "LoadingCondition1");</pre>
219	LoadingCondition1.Location = Location1;



Step 12: Create a Mass Model. Select User specified and enter the following parameters.

Properties	 Coordinate sy 	stem for inertia: RX/RY/RZ and RXY/RXZ/RYZ
👗 MassModel1	Inertia	💡 🔵 Input
		Global
Mass		COG centered
Mass 💡 7466330 kg	 Radii of gyrati 	ion
Coordinate system for center of gravity	RX	💡 23.8 m
COG 🢡 🖲 Input	RY	💡 23.8 m
Global	RZ	💡 4.69 m
Center of gravity	Specific prod	uct of inertia radii
X 💡 0 m	RXY	💡 0 m
Y 🢡 0 m	RXZ	💡 0 m
Z 💡 -89.916 m	RYZ	💡 0 m

220	<pre>MassModel1 = new MassModel(LoadingCondition1, "MassModel1");</pre>
221	<pre>MassModel1.UserSpecifiedMass = 7.46633E+06 kg;</pre>
222	<pre>MassModel1.UserSpecifiedCenterOfGravityX = 0 m;</pre>
223	<pre>MassModel1.UserSpecifiedCenterOfGravityY = 0 m;</pre>
224	<pre>MassModel1.UserSpecifiedCenterOfGravityZ = -89.9155 m;</pre>
225	<pre>MassModel1.UserSpecifiedRadiusOfGyrationX = 23.8 m;</pre>
226	<pre>MassModel1.UserSpecifiedRadiusOfGyrationY = 23.8 m;</pre>
227	<pre>MassModel1.UserSpecifiedRadiusOfGyrationZ = 4.69 m;</pre>

Note: This mass model is not yet balanced with the floater's displacement. For a first-order wave hydrodynamic analysis, an unbalanced condition is acceptable, as it does not affect the first-order wave excitation forces, the added mass coefficients, or the wave damping coefficients. However, achieving a balanced condition is crucial for accurately computing the second-order wave loads.

2.1.5 Creating and executing Wadam analysis

Step 13: Create a Wadam analysis. Please note the following commands are truncated to save space.

```
228 WadamAnalysis1 = new WadamAnalysis(Analyses, "WadamAnalysis1");
229 WadamAnalysis1.DataPreparation = true;
... ...
284 WadamAnalysis1.EnvironmentCondition = FrequencyDomainCondition1;
285 WadamAnalysis1.UseStochasticLinearization = false;
286 WadamAnalysis1.UseRollGZCurve = false;
```



Modify the following options:

In **General** tab, change the **tolerance waterline** to **100%**. This adjustment is necessary to allow Wadam to calculate an unbalanced condition without stopping the analysis.

Constants		
Tolerance waterline	Ŷ	100 %
Tolerance COG	9	5 %
Characteristic length	Ŷ	100 m
Motion reference point type	Ŷ	Use global origin
		 Input system coordinates
Motion reference point	Ŷ	0 m, 0 m, 0 m

287 WadamAnalysis1.ToleranceWaterline = new Fraction(1);

In the **Output** tab, check the box for **Global Response Files (G*.SIF/SIN/SIU)**. Set the **Prefix** to "**FirstOrder_**", and select **Move** as the **Output File Operation**. This will save the resulting **G1.SIF** file as **FirstOrder_G1.SIF**.



In the Advanced tab, uncheck the Calculate drift forces.

Drift forces	
Calculate drift forces	
Include bidirectional waves	
Pressure or control surface integration	💡 🔲
Use control surface integration also for vertical components	💡 🔲
Far field integration	💡 🗹
Wave drift damping	💡 🔲
Forward speed	
Include forward speed	💡 🗆
Vx	💡 0 m/s
Vy	💡 0 m/s
General Output Viscous Load Advanced Solver Style	

291 WadamAnalysis1.CalculateDriftForces = false;



In the Solver tab, check Include limiting frequencies.

Settings	
Logarithm singularity	💡 💿 Analytical
	O Numerical
Numerical integration	💡 💿 One node Gauss
	O Four node Gauss
Panel dimension	💡 💿 Maximum diagonal
	Area
Remove irregular frequencies	
Use damping sheets	
Include limiting frequencies	💡 🔽
292 WadamAnalys	is1.IncludeLimitingFrequenc
-	C .

Step 14: Execute the Wadam analysis and wait for it to finish.

<pre>293 WadamAnalysis1.Execute();</pre>	
--	--

2.1.6 Checking the results with Postresp

Right-click WadamAnalysis1 and select Start Postresp (new database).



294 WadamAnalysis1.StartPostrespNewDatabase();

The FirstOrder_G1.SIF file created by Wadam will be read automatically.



Display the wave transfer functions, for example FORCE5 (first-order wave moment in pitch direction).

1. Click **Display > Response Variable**.

Sesam Tutorial - Parametric Modelling of Spar for GeniE - HydroD Workflow |



- 2. Select FORCE5 as Variable Name and 0.0 90.0 as Wave Direction.
- 3. Click OK.



Let's print one of the response variables as a CSV file.

- 1. Click Set > Print.
- 2. Set the print options as shown below:



- 3. Click **Print > Response Variables**.
- 4. Select First Order, FORCE5 as Variable Name, and 0.0 as Wave Direction.



Exit Postresp by clicking File > Exit.



The corresponding Postresp JNL commands for the above operations can be found in the analysis folder of the Wadam analysis as the **POSTRESP.JNL** file. Below are the commands logged in the file:

```
FILE READ SIF-FORMATTED 'FirstOrder_' G1
1
       DISPLAY RESPONSE-VARIABLE ( ONLY FORCE5 ) ( ONLY 0.0 30.0 60.0 90.0 )
2
3
       SET PRINT DESTINATION CSV-FILE
4
       SET PRINT PAGE-HEIGHT 6000
5
       SET PRINT SCREEN-HEIGHT 24
       SET PRINT FILE FORCE5 POSTRESP
6
7
       SET PRINT PAGE-ORIENTATION LANDSCAPE
8
       PRINT RESPONSE-VARIABLE ( ONLY FORCE5 ) ( ONLY 0.0 )
```

We can use this JNL file to operate Postresp.



2.2 Hydrodynamic Analysis – Automated Setup

We will use the same approach here, modifying part of the manual setup scripts generated in the previous sub-chapter.

You can find the complete JS script from your manual modelling session in its corresponding workspace folder. The script file is named after the workspace, for example, if you used the "HydroManualModelling" as the workspace name, the file will be located in the HydroManualModelling folder as **HydroManualModelling.js** or **HydroManualModelling** https://doi.org/10.1016/j.js or

Note: If you are unable to locate it, you may use the Ch2-1_HydroManualSetup.js located in the input_files\Chapter 2 folder.

2.2.1 Design parameters

We will define the design parameters as follows:

- COG_Z: The substructure's center of gravity Z position
- PANEL_MODEL_FILEPATH: The panel model file path
- G1_PREFIX: The prefix for the G1.SIF file

You may also add other parameters, such as the frequency range or the location's depth, but for simplicity, we will focus on these parameters for now.

2.2.2 Defining user defined parameters

The user-defined parameters are as follows:

```
1 // USER DEFINED PARAMETERS
2 COG_Z = -89.9155 m;
3 PANEL_MODEL_FILEPATH = "Panel1_T1.FEM";
4 G1_PREFIX = "FirstOrder1_";
```

2.2.3 Pre-calculating variables

We do not need to pre-calculate any variable this time.

2.2.4 Modifying the manual setup scripts

Take the scripts generated in the previous sub-chapter and add slight modifications. The modifications are highlighted in red, with the design parameters **bolded**. Some parts of the code that remain unmodified are removed to save space. Note that comments from the manual modelling scripts have been removed for brevity.

```
6
       // MAIN FUNCTION
7
       Workspace.AccelerationUnit = "m/s^2";
       Workspace.AngleUnit = "deg";
8
9
       Workspace.AngularFrequencyUnit = "deg/s";
10
       Workspace.AreaUnit = "m^2";
11
       Workspace.DensityUnit = "kg/m^3";
12
       Workspace.Description = null;
199
       ElementModel1.FileName = PANEL_MODEL_FILEPATH;
       PanelModel1 = new PanelModel(HydroModel1, "PanelModel1");
200
```

| Sesam Tutorial - Parametric Modelling of Spar for GeniE - HydroD Workflow |



201	<pre>PanelModel1.ElementModel = ElementModel1;</pre>
202	PanelModel1.ShowPlateEdges = true;
203	PanelModel1.ShowHydroPressureArrows = true;
211	<pre>MassModel1.UserSpecifiedCenterOfGravityZ = COG_Z;</pre>
212	<pre>MassModel1.UserSpecifiedRadiusOfGyrationX = 23.8 m;</pre>
213	<pre>MassModel1.UserSpecifiedRadiusOfGyrationY = 23.8 m;</pre>
214	<pre>MassModel1.UserSpecifiedRadiusOfGyrationZ = 4.69 m;</pre>
276	WadamAnalysis1.Prefix = G1_PREFIX;
277	WadamAnalysis1.OutputFileOperationType = OutputFileOperationType.Move;
278	WadamAnalysis1.CalculateDriftForces = false;
279	WadamAnalysis1.IncludeLimitingFrequencies = true;
280	WadamAnalysis1.Execute();
281	WadamAnalysis1.StartPostrespNewDatabase(); //deleted because it is not needed

Note: A completed JS file, Ch2-2_HydroAutomatedSetup.js, is provided in the input_files\Chapter 2 folder.

2.2.5 Running the parametric script and checking the results

Create a new HydroD workspace and run the script by going to File > Read script.

In the next window, browse for the parametric script and click **Open**.

H Open				×
\leftarrow \rightarrow \checkmark \uparrow	<u> </u>	ienieMo > input_files \lor \bigcirc	Search input_files	م
Organize 🔻 🛛 Ne	ew folder		≣ ▼	
		Name	Date modified	Туре
		Ch1-1_PanelManualModelling.js	2025-04-10 23:52	JS File
		Ch1-2_PanelAutomatedModelling.js	2025-04-11 11:03	JS File
		Ch2-1_HydroManualSetup.js	2025-04-11 14:10	JS File
		Ch2-2_HydroAutomatedSetup.js	2025-04-11 15:13	JS File
		Ch3_GeniE_Input1.js	2025-04-11 15:17	JS File
		Ch3_GeniE_Input2.js	2025-04-11 15:17	JS File
	1	Ch3_GeniE_Input3.js	2025-04-11 11:03	JS File
	File name:	Ch2-2_HydroAutomatedSetup.js	Command files (*.js)	~
			Open	Cancel

The workspace will be populated, and the Wadam analysis will be run. After the operations are finished, verify the model and the hydro arrows are displayed correctly.

Also, check if the FirstOrder1_G1.SIF is outputted successfully.

In addition, you may open Postresp afterward to review and verify the analysis results.



3 AUTOMATING GENIE AND HYDROD WITH SESAM MANAGER

We will use **Sesam Manager** to manage multiple cases of our analyses. **Sesam Manager** is a free module within **Sesam** that helps in managing analyses. In this chapter, we will perform a simple comparative study of the following spar-type FOW floaters, all with the same displaced volume.

Note: We will focus on how to set up the analyses rather than comparing the results themselves. Additionally, in an actual project, the user must ensure that the analyses have successfully converged before moving on to the next steps in the design workflow.

Design Parameter	Case 1	Case 2	Case 3
UPPER_LENGTH	14	14	14
MIDDLE_LENGTH	8	8	8
BOTTOM_LENGTH	108	64.92667	46.81687
UPPER_DIAMETER	6.5	6.5	6.5
BOTTOM_DIAMETER	9.4	12	14
BOTTOM_Z	-120	-76.9267	-58.8169
COG_Z	-89.9155	-56.0	-40.0
Prefix	Case1_	Case2_	Case3_

Table 2 Design cases to be inspected





3.1 Preparing the Parametric Scripts

3.1.1 Preparing the GeniE script

Prepare three separate **GeniE** parametric modelling scripts, one for each case. The only difference between the files is the value of the user-defined parameters. Copy the JS script you used in Chapter 1.2 (or use the provided **Ch1-2_PanelAutomatedModelling.js**) and name the copied files as follows:

- Case 1: Ch3_GeniE_Input1.js
- Case 2: Ch3_GeniE_Input2.js
- Case 3: Ch3_GeniE_Input3.js

Populate each file's user defined parameters according to Table 2. For example, for Case 3:

```
1
       // USER DEFINED PARAMETERS
2
       UPPER LENGTH = 14;
       MIDDLE LENGTH = 8;
3
4
       BOTTOM LENGTH = 46.81687075;
5
       UPPER DIAMETER = 6.5;
6
       BOTTOM DIAMETER = 14;
7
       BOTTOM Z = -58.81687075;
8
       MESH DENSITY = 0.75;
9
       PANEL MODEL PREFIX = "Case3 ";
```

Note: For each case, the panel model will be saved as Case#_T1.FEM file. We need to reflect this in the HydroD scripts.

3.1.2 Preparing the HydroD script

Similarly, prepare three separate **HydroD** scripts using the JS script you used in Chapter 2.2 (or use the provided **Ch2-2_HydroAutomatedSetup.js**). Name the copied files as follows:

- Case 1: Ch3_HydroD_Input1.js
- Case 2: Ch3_HydroD_Input2.js
- Case 3: Ch3_HydroD_Input3.js

Populate each file's user defined parameters according to Table 2. For example, for Case 3:

```
1 // USER DEFINED PARAMETERS
2 COG_Z = -40.0 m;
3 PANEL_MODEL_FILEPATH = "Case3_T1.FEM";
4 G1_PREFIX = "Case3_";
```



3.1.3 Preparing the Postresp script

Prepare Postresp scripts by creating them manually for each case:

- Case 1: Ch3_POSTRESP1.JNL
- Case 2: Ch3_POSTRESP2.JNL
- Case 3: Ch3_POSTRESP3.JNL

Enter the following commands based on the operations performed in Chapter 2.1.6. Be sure to update the FILE READ section to match the prefix of the G1.SIF file generated by HydroD (highlighted red). For example, for **Case 3**:

1	FILE READ	SIF-FORMATTED 'Case3_' G1
2	SET PRINT	DESTINATION CSV-FILE
3	SET PRINT	PAGE-HEIGHT 6000
4	SET PRINT	SCREEN-HEIGHT 24
5	SET PRINT	FILE CASE1_F5_ POSTRESP
6	SET PRINT	PAGE-ORIENTATION LANDSCAPE
7	PRINT RES	PONSE-VARIABLE (ONLY FORCE5) (ONLY 0.0)
8	FILE EXIT	

Note: The DISPLAY command has been omitted intentionally, as we do not require any graph output when running the script in the background. Additionally, make sure to include a FILE EXIT command at the end of the script. This is necessary to properly close Postresp and ensure that the analysis completes successfully.

3.1.4 Preparing Sesam Manager Post Execute Script for Postresp

The final file, **PostrespActivity_post.js**, is used to copy all the CSV files in the **Postresp** analysis folder to the **_repository** folder in the **Sesam Manager** workspace. It only contains one line:

1 Copy(CurrentActivity.Workspace + "*.csv", "_repository", CurrentActivity.StartExecutionTime, FileSearchOption.AllDirectories);

You may refer to Sesam Manager's User Manual for more information about the Post Execute Script functionality.

Note: Completed scripts are provided in the input_files\Chapter 3\Scripts folder.



3.2 Setting Up Sesam Manager

3.2.1 Creating a new Job

Open Sesam Manager 6.7 and click New Job icon.

₩ ≂	Create Job X
File Home View Tool	Name and folder of new job: Job name: SparComparativeStudy Create job in folder: C:\DNV\Workspaces\ Select Template: None v Sesam examples
	OK Cancel

3.2.2 Copying the parametric modelling scripts

Copy the prepared scripts into the _repository folder inside the Sesam Manager workspace folder.

□ > ··· Workspaces > Sp	arComparativeStudy >	repository	Search _
î	ort $\stackrel{_{\scriptstyle\scriptstyle\vee}}{=}$ \equiv View $\stackrel{_{\scriptstyle\scriptstyle\vee}}{-}$		
Name	Date modified	Туре	Size
Ch3_GeniE_Input1.js	2025-04-11 16:21	JS File	3 KB
Ch3_GeniE_Input2.js	2025-04-11 15:17	JS File	3 KB
Ch3_GeniE_Input3.js	2025-04-11 15:17	JS File	3 KB
Ch3_HydroD_Input1.js	2025-04-11 17:03	JS File	14 KB
Ch3_HydroD_Input2.js	2025-04-11 17:04	JS File	14 KB
Ch3_HydroD_Input3.js	2025-04-11 17:04	JS File	14 KB
Ch3_POSTRESP1.JNL	2025-04-11 17:03	JNL File	1 KB
Ch3_POSTRESP2.JNL	2025-04-11 17:04	JNL File	1 KB
📓 Ch3_POSTRESP3.JNL	2025-04-11 17:04	JNL File	1 KB
PostrespActivity_post.js	2025-04-11 17:01	JS File	1 KB

3.2.3 Creating a Sequence folder

Drag and drop a **Sequence** from the **Applications** pane (on the left) to the **Flow Chart** pane (at the centre of the window).



X =	SparComparativeStudy" - Sesam Manager V6.7-00		- 0 ×
file Memo View Teel			
New	Image: Second		
New Open Save	Import/Export Process		^
Applications • 0 ×	SparComparativeStudy* × ○ Start □ Quick Start < >	Properties	- 0 X
	Flow Chart Tree View	CompositeActivity Se	iquence1
T Sesam applications		Search	×
Cutres		 General 	
FatigueManager		Name	Sequence1
Framework	Description of the second	Description	CADNAWadaman) SearCommentingStudi (Searcent)
G GeniE	Drag and drop	workspace	C.(DWVWOrkspaces/spaceomparatives/duby/sequencer/
S Gensod			
HydroD			
S Installjac	E SparComparativeStudy 😤		
Mimosa	V		
Pilgen			
Platework	E Sequence1 😤		
Postresp			
Postresp_time			
Preframe			
S Prepost			
Presel			
Sestra			
Sima			
Splice			
Stofat			
Submod			
💽 Usfos			
🛐 Wajac			
Waloco			
Maveship	Messane List		
Xtract	C DErrors A OWArrings O Messages		
© Custom			
Almost	ype beschpion Source		
User defined			
PreselNoScript			
X ShellDesign		•	
🛠 Wamod			
File Overview Applications	Command Line Message List Job Comments Activity Run Manager	Properties Attachmer	nts

3.2.4 Creating a GeniE activity

Drag and drop **GeniE** from the **Applications** pane (on the left) to **Sequence1**.

M =	SparComparativeStudy* - Sesam Manager V6.7-00		- a ×
File Home View Tool			<u>~</u> 0
New New New New Open Jobs See Open See	Image: Topo Image Image Image Image Image: Topo Image Topo Image Image Image Image Image Image Image		^
Applications v 0 >	SourComparativeStudy* X O Start D Ouick Start	Properties	- 0 X
Control flow	Blow Chart Tete Kiny	GeniEActivity GeniEActivity	1
E Sequence		Search	×
E Sesam applications		Juneir	^
S Cutres		4 General	
FatigueManager		Description	GenitActivity
Erameuntk		Deteksethere	Constantine d
G GeniE	F SparComparativeStudy	Databaservame	New Y
Gensod	Drag and drop	Worksmace	Workspaces\SparComparativeStudy\Sequence1\GeniEActivity1\
HydroD		CmdInputFile	default
🔢 Installjac		UseLocalCmdinputFile	7
Mimosa	E Sequence1 🛪	PreviousCmdInputFile	Clear
💽 Pilgen	V	ImportFile	Cir Edit
S Platework		SuperElementNumber	1 \$
S Postresp	🕼 GeniEActivity1 🔗	InputMode	Interactive ~
Postresp_time		ProgramVersion	8.12.3 (64) ~
9 Prefem		SelectedLicenses	CurvedGeometry,FrameCodeCheck,PlateCodeCheck,RefineMesl *
Preframe	×	AdditionalArguments	
Prepost		CommandLine	C:\Program Files\DNV\GeniE V8.12-03\Program\GenieR.exe "GeniEA
Presel		 Script DirablePreEveruteScript 	
Sectra		PreExecuteScript	defaultEdit Execute
Sima Sima		DisablePostExecuteScri	
Solice		PostExecuteScript	default Edit Execute
Confect		4 Units	
Colored Colored		EnableTolerantModelli	✓
		ForceUnit	N ~
Weize		LengthUnit	m ~
Wajac .		TemperatureUnit	delC ~
Waloco			
wavesnip	Message List		
Maract 20	O UErrors A OWarnings 0 Messages		
© Custom			
R Idenset	lype Description Source		
EUser defined			
PreselNoScript			
ShellDesign			
X Wamod			
File Overview Applications	Command Line Message List Job Comments Activity Run Manager	Properties Attachments	



Click on the **GeniEActivity1** and look at the **Properties** pane (on the right).

Change the **CmdInputFile** by clicking the Browse button .

Set it to Ch3_GeniE_Input1.js.

Change the **InputMode** to **Background**.

Properties accordences		2000000000000000 👻 🛱 🗙			
GeniEActivity GeniEActivity	l				
Search		×			
 General 					
Name	GeniEActivity1				
Description		•			
DatabaseName	GeniEActivity1				
DatabaseStatus	New	~			
Workspace	ComparativeStudy\Seq	uence1\GeniEActivity1\			
CmdInputFile	<pre>/eStudy_repository\Cl</pre>	n3_GeniE_Input1.js <mark>E</mark> dit			
UseLocalCmdInputFile	v				
PreviousCmdInputFile		Clear			
ImportFile	Clr Edit				
SuperElementNumber	1	 ▼			
InputMode	Background	Ŷ			
ProgramVersion	8.12.2 (64)	v			
SelectedLicenses	CurvedGeometry,Fran	neCodeCheck,PlateCod			
AdditionalArguments					
CommandLine	C:\Program Files\DNV	\GeniE V8.12-02\Program\@			
 Script 	_				
DisablePreExecuteScript					
PreExecuteScript	default	Edit Execute			
DisablePostExecuteScri					
PostExecuteScript	default	Edit Execute			
 Units 	_				
EnableTolerantModelli	~				
ForceUnit	N	~			
LengthUnit	m	~			
TemperatureUnit	delC	~			

3.2.5 Creating a HydroD activity

Drag and drop HydroD from the Applications pane to Sequence1, under the GeniEActivity1.

M 🗢		SparComparativeStudy* - Sesam Manager V6.7-00			– a ×
File Home View Tool					 Ø
New	Image: Section of the sectio	ected ity			
wew Open Save	Import/Export Protess				^
Applications * 0 ×	SparComparativeStudy" X 😹 Jobs 🕠 Start 🔛	Quick Start	Properties		• 0 X
Control flow	Flow Chart Tree View		- 📋 HydroDAc	ctivity HydroDActivi	ty1
E sequence			Search		×
Cutres			4 Genera	al	
Estimute fan seer		E SparComparativeStudy	Name	2 F	łydroDActivity1
Constant Sector			Descri	iption	•
			Databa	aseName H	lydroDActivity1
		E Sequence1	Databa	aseStatus I	New ~
Consod		T refrance.	Works	space 'o	rkspaces\SparComparativeStudy\Sequence1\HydroDActivity1\
HydroD		\bigtriangledown	Cmdir	nputFile d	lefault Edit
Installjac D	rad and drop		UseLo	calCmdinputFile	v
Mimosa		GeniEActivity1	Previo	susCmdInputFile	Clear
9 Pilgen			Import	tFile	Clr Ede
Platework			Input	Mode	Interactive ~
F Postresp		∇	Progra	amVersion	7.3.1 (64)
Postresp_time		÷	Selecti	edLicenses	Wadam, Wasim, Stability
Prefem		The second secon	Additi	ionalArguments	
F Preframe	_	HydroDActivity1 A	Comm 4. Easter	nandLine C	
Prepost			- Script	In the secure Script	
Presel			PreExe	ecuteScript	lefaultEditExecute
Sectra		\bigtriangledown	Disable	ePostExecuteScri	
Sima Sima			PostEx	xecuteScript	lefault Edit Execute
Colice			4 Units		
Spice Spice		-	Forcel	Unit /	N ~
Stotat			Lengt	hUnit	m ~
Submod			Tempe	eratureUnit	delC ~
Usfos					
5 Wajac					
Maloco					
Sea Waveship	Marranelist				
3 Xtract			- + ~		
E Custom	U U Warnings U Messages				
🛠 General	Type Description	Source	e		
JsImport					
E User defined					
PreselNoScript					
🔆 ShellDesign					
🛠 Wamod 🗸	11				
File Overview Applications	Command Line Message List Job Comments Activity Run Mar	nager	Properties	Attachricents	



Select the newly created HydroDActivity1.

Change the **CmdInputFile** by clicking the Browse button .

Set it to Ch3_HydroD_Input1.js.

Change the **InputMode** to **Background**.

HydroDActivity HydroDActivity1 Search General Name HydroDActivity1	×
Search 4 General Name HydroDActivity1	×
General Name HydroDActivity1	
Name HydroDActivity1	_
Description	•
DatabaseName HydroDActivity1	
DatabaseStatus New	~
Workspace mparativeStudy\Sequence1\HydroDActivity	1\
CmdInputFile itudy_repository\Ch3_HydroD_Input1.js	dit
UseLocalCmdInputFile	
PreviousCmdInputFile	Clear
ImportFile CIr	Edit
InputMode Background	~
ProgramVersion 7.2.1 (64)	~
SelectedLicenses Wadam,Wasim,Stability	*
AdditionalArguments	
CommandLine C:\Program Files\DNV\HydroD V7.2-01\Bin	Hydro
4 Script	
DisablePreExecuteScript	
PreExecuteScript default Edit Ex	ecute
DisablePostExecuteScri	
PostExecuteScript default Edit Ex	ecute
4 Units	
ForceUnit N	\sim
LengthUnit m	~
TemperatureUnit delC	~

3.2.6 Creating a Postresp activity

Similarly, drag and drop **Postresp** from the **Applications** pane to **Sequence1**, under the **HydroDActivity1**.

🕅 🗢		SparComparativeStudy* - Sesam Manager	V6.7-00			- 0 ×
File Home View Tool	not import import Export Export Expert	Selected				^ 0
New Onen Save	Imnart/Funart Drord					
uncon Open Save		D o L o L				
Applications + U ×	SparComparativeStudy* × Jag Jobs 💭 Start	Quick Start	() ()	Properties		,
E Control now	Flow Chart Tree View		~ H	Postrespactivity Postrespa	ctivityi	
- Second applications				Search		X
Cutres		E SparComparativeStudy 😤		 General 		
FatigueManager				Name	PostrespActivity1	
Framework				Description		•
Ganif		E Sequence1		DatabaseName	PostrespActivity1	
Connel				DatabaseStatus	New	~
Gensod				Workspace	orkspaces\SparComparativeStudy\Sequence1\Po	istrespActivity1\
HydroD		G GeniEActivity1		CmdInputFile	default	Edit
instalijac 👔				UseLocalCmdInputFile	v	
Mimosa				PreviousCmdInputFile		Clear
Bilgen		~		ResultsFile		Clr
Platework				GFileNumber	1	
S Postresp	Drag and drop	HydroDActivity1 🖉		InputFilePath	C:\DNV\Workspaces\SparComparativeStudy_re	pository
Vostresp_time	Drag and drop			InputhilePretox	000000	
🐔 Prefem				ReactionForcesCentre	1.1	
I Preframe				Brownode	T 2 2 (64)	
Prepost				AdditionalAssesses	7-2-3 (04)	
T Presel		PostrespActivity1 😤	NT .	AdditionalArguments	510 EL 101010 - 1070 000110	
Castra				CommandLine		
E Gana				DisablePreExecuteScript		
Sina Caller				PreExecuteScript	default	Edit Execute
spice				DisablePostExecuteScri.		
Stofat				PostExecuteScript	default	Edit Execute
Submod						
🔠 Usfos						
🚳 Wajac						
Waloco						
S Waveship	and the second s		- 1 - 1			
S Xtract	Message List		······································			
- Custom	0 Warnings 0 Messages					
📯 General	Type Description		Source			
JsImport						
⊡User defined						
PreselNoScript						
🛠 ShellDesign						
🛠 Wamod	é					
File Overview Applications	Command Line Message List Job Comments Activity Run	Manager		Properties Attachments		



Select the newly created	Properties 🔹 👎								
Postrosn Activity1		PostrespActivity PostrespActivity1							
Fostiespacifyity 1.	Search X								
Change the CmdInputFile by clicking	4	General							
the Browne button		Name	PostrespActivity1						
		Description		•					
Sat it to Ch2 BOSTBESD1 INI		DatabaseName	PostrespActivity1						
Set It to Ch3_POSTRESPT.JNL.		DatabaseStatus	New	~					
Observe the InvestMester to		Workspace	nparativeStudy\Sequence	e1\PostrespActivitv1\					
Change the inputmode to		CmdInputFile	eStudy_repository\Ch3_	POSTRESP1.JNL Edit					
Background.		UseLocalCmdInputFile	v						
		PreviousCmdInputFile		Clear					
Then, change the PostExecuteScript		ResultsFile		Clr					
by diaking the Browce button		GFileNumber	1	÷					
by clicking the browse button .		InputFilePath	orkspaces\SparCompara	tiveStudy_repository					
		InputFilePrefix							
Set it to PostrespActivity_post.js.		ReactionForcesCentre	0.0 0.0 0.0						
		InputMode	Background	~					
		ProgramVersion	7.2.3 (64)	Ŷ					
		AdditionalArguments							
		CommandLine	C:\Program Files\DNV\P	ostresp V7.2-03\Bin\Post					
	-	Script							
		DisablePreExecuteScript							
		PreExecuteScript	default	Edit Execute					
		DisablePostExecuteScri.							
		PostExecuteScript	ository\PostrespActivity	post.js Edit Execute					

3.2.7 Creating the rest of the Job

Repeat the process to create **two more Sequences** for **Case 2** and **Case 3**. Make sure each Sequence contains the correct:

- GeniE script (Ch3_GeniE_Input2.js, Ch3_GeniE_Input3.js)
- **HydroD script** (Ch3_HydroD_Input2.js, Ch3_HydroD_Input3.js)
- Postresp script (Ch3_POSTRESP2.JNL, Ch3_POSTRESP3.JNL)

Once completed, your Job should contain three Sequences, each with the full set of activities.



Tip: Switch to the Tree View tab to review the full structure clearly, as shown above.

Note: You can click Import ZIP and browse the Ch3_SesamManagerJob.zip provided in the input files folder to import the finished Sesam Manager Workspace.

Μ	~												
File	Home	View	Tool										
*	G	D	B	🚽 🚅		9	JS	XML	Zip		15		Zip
New Job	New GeniE Job	New DeepC Job	New HydroD Job	Open Jobs Job	Save	Save All	Import JS	Import XML	Import ZIP	Import JNL	Export JS	Export XML	Export ZIP +
		New		Open	Save				Imp	ort/Exp	ort		



3.2.8 Running all analyses

To run the analyses, right-click on the **SparComparativeStudy** container and select **Run**. When prompted, choose **Always overwrite files** and click **Run**.



The activities will run sequentially. After all analyses are finished, the results will be saved into the _repository folder:

Name	Date modified	Туре	Size	
CASE1_F5_POSTRESP.csv	2025-04-11 18:01	Microsoft Excel C	9 KB	
Case1_G1.SIF	2025-04-11 18:00	SIF File	431 KB	
💕 Case1_T1.FEM	2025-04-11 17:55	FEM File	4,732 KB	
CASE2_F5_POSTRESP.csv	2025-04-11 18:06	Microsoft Excel C	9 KB	
Case2_G1.SIF	2025-04-11 18:05	SIF File	431 KB	
🛃 Case2_T1.FEM	2025-04-11 18:01	FEM File	4,024 KB	
CASE3_F5_POSTRESP.csv	2025-04-11 18:09	Microsoft Excel C	9 KB	
Case3_G1.SIF	2025-04-11 18:09	SIF File	431 KB	
💕 Case3_T1.FEM	2025-04-11 18:06	FEM File	3,609 KB	

You can check each activity to ensure the analyses were set up correctly by running them individually, or review the CSV results containing the first-order wave moment in the pitch direction for each floater.

This concludes the Sesam tutorial. We hope it has inspired you to create more sophisticated parametric modeling methods tailored to your design workflow.



About DNV

DNV is an independent assurance and risk management provider, operating in more than 100 countries. Through its broad experience and deep expertise DNV advances safety and sustainable performance, sets industry standards, and inspires and invents solutions.

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.