SESAM USER MANUAL

# HYDROMESH USER MANUAL

## User Controlled Free Surface Meshing for HydroD

Valid from HydroMesh version 4.0





Sesam User Manual

HydroMesh

Date: 2023-10-23

Valid from HydroMesh version 4.0

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


1 Introduction . . . 1
1.1 Overview . . . 1
1.2 HydroMesh in the Sesam System . . . 1
1.3 Terminology and Notation . . . 1
2 HydroMesh Application . . . 1
2.1 Introduction . . . 1
2.2 Command Line Arguments . . . 1
3 HydroMeshSetup Application . . . 2
3.1 Introduction . . . 2
3.2 Command Line Arguments . . . 2
3.3 User Interface . . . 3
3.3.1 Menu Items . . . 3
3.3.2 Tool Bar Buttons . . . 4
3.3.3 Control Panel . . . 4
3.3.4 Properties Editing Area . . . 6
3.3.5 Operating area . . . 12
3.3.6 Status Bar Area . . . 17
3.4 Integration with HydroD . . . 17
3.4.1 Generate ElementModel . . . 17
3.4.2 Section Mesh . . . 19
3.4.3 Loading condition . . . 21
3.5 Data Organization . . . 21
3.5.1 HydroMeshSetup input/output files . . . 21
4 Requirements to section model . . . 22
4.1 General requirement . . . 22
4.2 Hull meshing . . . 23
4.3 Parametric free surface meshing . . . 23

# 1 Introduction

## 1.1 Overview

HydroMesh is a meshing tool serving several purposes:

1. Create free surface meshes for Wadam.
2. Create a panel model from a Section model.
3. Create meshes for Wasim (replacing Wasim_Mesh).

The free surface mesh can be created interactively or automatically.
NB: HydroMesh and HydroMeshSetup require that the section model is properly formatted and ordered. Refer to section 4 for details.

## 1.2 HydroMesh in the Sesam System

HydroMesh is integrated into the HydroD/SESAM system, it can also be run as a stand along program using input files created by HydroD and optionally with an XML file (free surface settings file) generated by HydroMeshSetup.

## 1.3 Terminology and Notation

| | |
|---|---|
| LMB: | Left mouse button |
| RMB: | Right mouse button |
| PLN: | File format for sectional hull geometry |
| SSG: | File format for meshes |
| GEO: | File format for irregular meshes |
| FEM: | General SIF file format (irregular meshes) |
| XML: | File format for free surface mesh settings (waterline/split line coordinates, mesh- resolution etc.) |
| HydroMeshSetup: | User interface for HydroMesh for generating and editing free surface mesh configuration file. |

# 2 HydroMesh Application

This chapter provides an overview of the HydroMesh application.

## 2.1 Introduction

HydroMesh consist of two executable elements, HydroMesh.exe and HydroMeshSetup.exe. HydroMeshSetup.exe is an interactive tool for creation of the mesh on the free surface. HydroMesh is a batch program used to create the meshes. This can be based on the settings defined in HydroMeshSetup or just on global parameters given in the defaults file.
As a fist step towards merging this into one single application HydroMeshSetup has been extended with the HydroMesh functionality and can be run in silent mode. It is expected that in the next release there will be only one executable element.

## 2.2 Command Line Arguments

HydroMesh execution:

1. *HydroMesh.exe*
   This should generate the mesh above the waterline in *_dry.ssg file, based on the *.pln file and "*defaults*" in the working directory. *_dry.ssg can be taken as input to HydroMeshSetup for free surface mesh editing.
2. *HydroMesh.exe fem*
   This should generate the mesh on the ship hull only in T7301.FEM, T7302.FEM and T7320.FEM, based on the *.pln file and "*defaults*" file in the working directory. The FEM files are used as panel model in a Wadam analysis.

3. *HydroMesh.exe* "*filename*"*_fs.xml XXX*
   This should generate mesh files on the free surface based on the settings in the "filename"_fs.xml file and "*defaults*" file in the working directory. If a pln file is specified on the "*defaults*" file the mesh is generated also on the hull (SSG, GEO, FEM) and these files can be taken as input to a Wasim analysis. If a FEM file is specified, the user already has a mesh of the hull and only the free surface mesh will be generated on T7373.FEM, T7303.FEM and possible T7374.FEM. *XXX* is an optional argument that can be used to override the default prefix of 73 for the FEM files produced. Any number between 1 and 999 are accepted, i.e. T99973.FEM.

4. *HydroMesh.exe auto*
   This should generate mesh files on the free surface based on the *.pln and "*defaults*" file in the working directory. The mesh is generated also on the hull (SSG, GEO, FEM) and these files can be taken as input to a Wasim analysis.

# 3 HydroMeshSetup Application

## 3.1 Introduction

HydroMeshSetup is used to read in waterline coordinates, divide the free surface into user defined patches and finally store the result in an XML file. This XML file could be read into HydroMeshSetup again for further editing.

## 3.2 Command Line Arguments

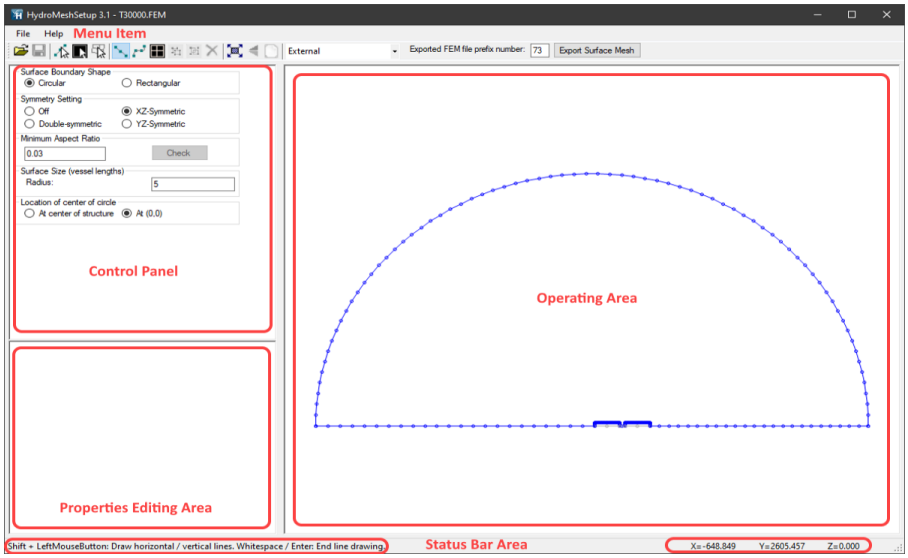
HydroMeshSetup execution:

1. *HydroMeshSetup.exe*
   This will start HydroMeshSetup. it searches and opens the default file and related files.
   (a) If there is not default file in the current folder, open GUI without open any file.
   (b) If default is presented but has error (filnam or prefil has invalid character etc), show error message, open GUI but do not open any file.
   (c) If default is presented and has no error, if filpre+_fs.xml exist in current folder, open it, otherwise open and read filnam.
   (d) filnam in default file can be any of the file type:
      i. _fs.xml
      ii. _dry.ssg
      iii. FEM
      iv. pln

2. *HydroMeshSetup.exe args*
   HydroMeshSetup accepts any amount of args, the args are used as:
   (a) Find the first _fs.xml file which exist in the current folder, and open it. Ignore other args.
   (b) If no file satisfy previous step, then find the first _dry.ssg file which exist in the current folder and open it. Ignore other args.
   (c) If no file satisfy previous step, then find the first .fem file which exist in the current folder and open it. Ignore other args.
   (d) If no file satisfy previous step, then find the first .pln file which exist in the current folder and open it. Ignore other args.
   (e) If no file satisfy previous step. then open GUI without open any file.
   (f) please note that when opens _dry.ssg, .fem and .pln, the default file must present in the folder.

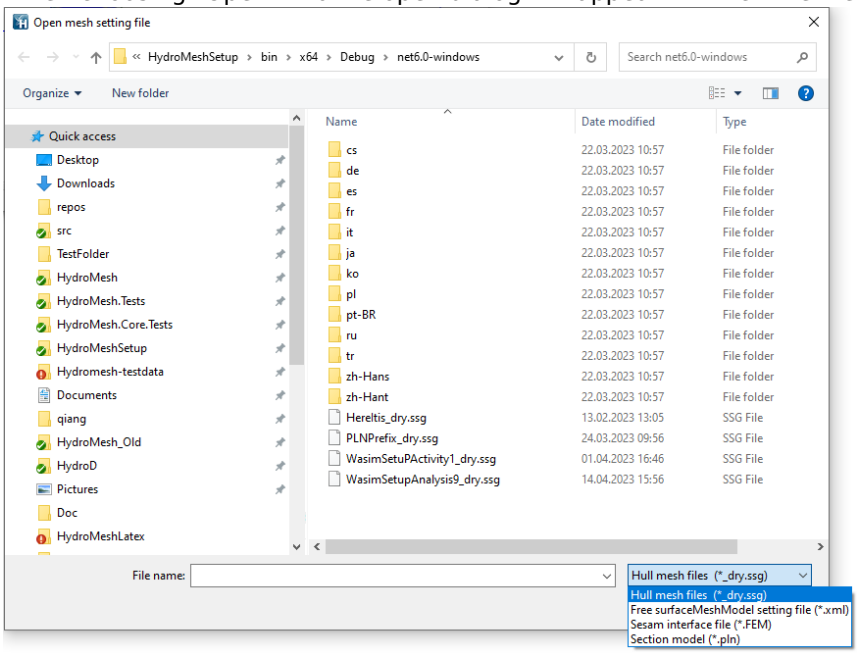
3. *HydroMeshSetup.exe -s*
   HydroMeshSetup supports silent mode, which is enabled by call "HydroMeshSetup.exe -s" or "HydroMeshSetup.exe /silent". HydroMeshSetup.exe silent mode equals to HydroMesh.exe. For example, "HydroMeshSetup.exe -s auto" equals "HydroMesh.exe auto".

## 3.3 User Interface




### 3.3.1 Menu Items

1. "Open ..." menu item
   When choosing "Open ..." a file open dialog will appear in which the file type is fixed.




2. "Save" menu item
   Save the free surface setting into xml configuration file, the file name is ended with *_fs.xml.

3. "Exit" menu item
   Exit the HydroMesh application.

### 3.3.2 Tool Bar Buttons




 Open file button. Shortcut: Ctrl + O

 Save file button, available only when changes has been made. Shortcut: Ctrl + S

 Reload button, clear everything, reload waterline. Shortcut: L

 Select split line mode, click to select split line to edit its properties. Shortcut: S

 Switch the active areas to be meshed. Shortcut: A

 Select patch mode, click to select a patch to edit its properties. Shortcut: P

 Draw lines mode button, click in operating area to draw a split line. Shortcut: D

 Draw elliptical arc button, button, click in operating area to draw an elliptical split line. Shortcut: V

 Create patch mode button, click in closed region to create a mesh patch. Shortcut: C

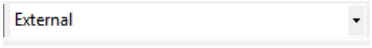
 Merge two selected patches into single one, Shortcut: G

 Unmerge a selected patch. Shortcut: U

 Delete mode button. Delete selected patch or split line. Shortcut: E

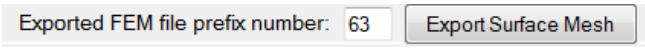
 Scale content to window button, click to zoom the free surface to fit the window. Shortcut: F

 Mirror the model button, available only when the model is symmetric. Shortcut: M




Switch to tank internal free surface view when applicable.

#### Export Surface Mesh Button

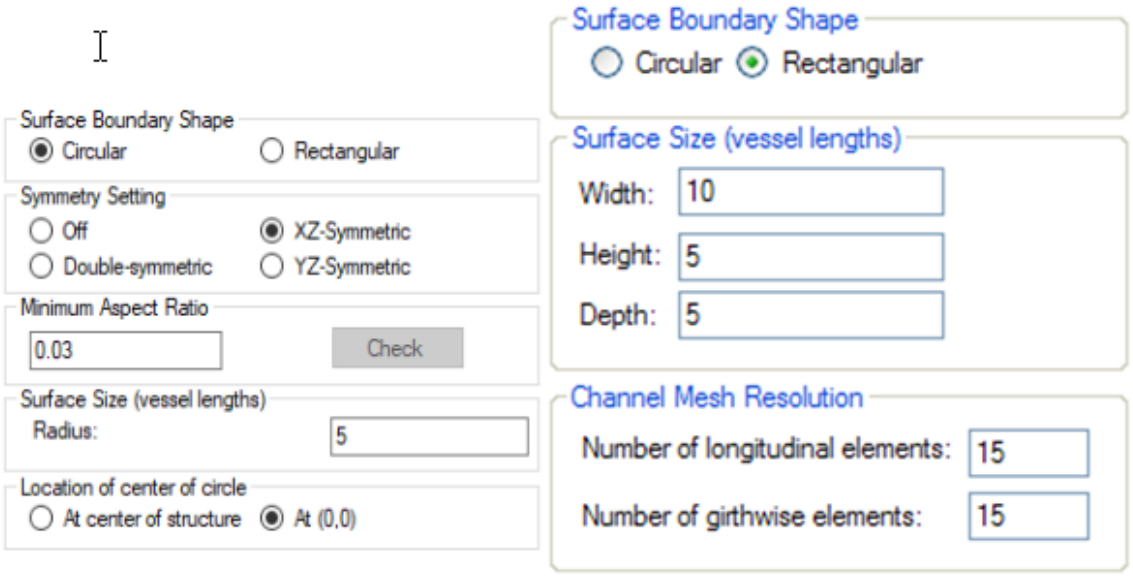



Clicking Export Surface Mesh will export the free surface mesh on FEM files with the given FEM file prefix XXX. TXXX03.FEM, TXXX73.FEM and possibly TXXX74.FEM, in the case of symmetry, will be generated in the folder where the program is started.

### 3.3.3 Control Panel

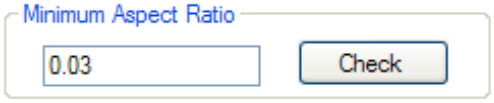
#### Surface boundary shape

Select the shape of the free surface boundary condition shape, you could choose either circular or rectangular boundary shape.




## Aspect ratio check

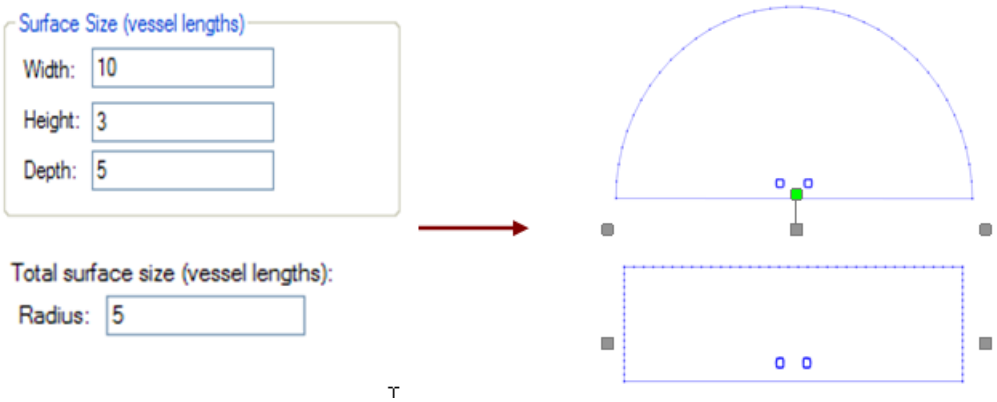
You could check the aspect ratio of the free surface patch as follows:




When you press the "Check" button, any invalid patch whose aspect ratio is less than the value specified in the above TextBox will be highlighted in red color.

## Total surface size (vessel lengths)

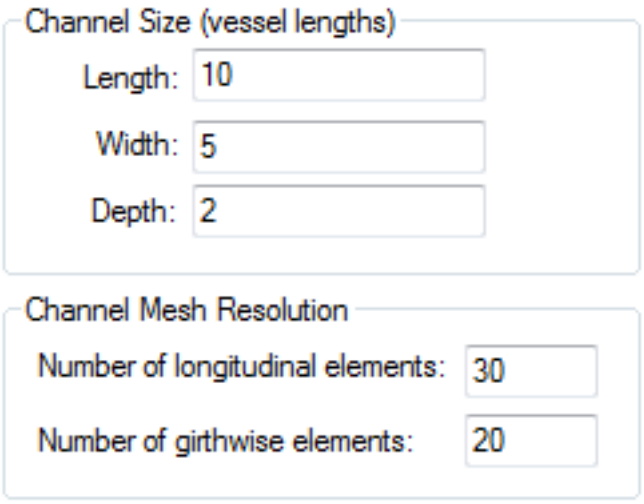
The surface size value decides the size of the free surface, by default the radius is 5 vessel lengths, the width is 10, and the height is 5, the depth is 5.



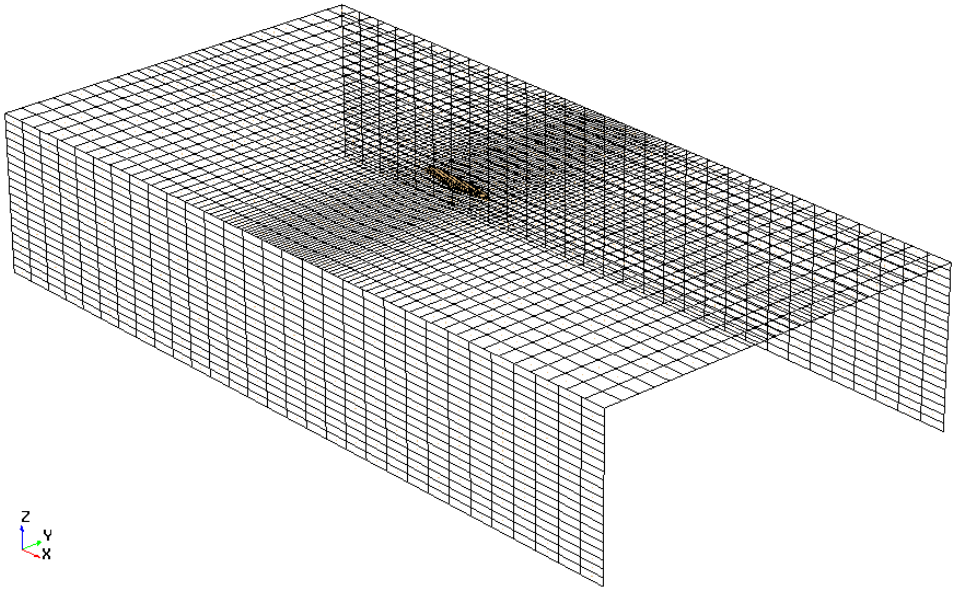

## Channel mesh resolution

This could change the mesh resolution for the both sides of channel at the girth wise and longitudinal direction, you could only examine the mesh resolution visually from HydroD:

1. Mesh Resolution Setting from HydroMesh




2. 3D Mesh Visualization in HydroD

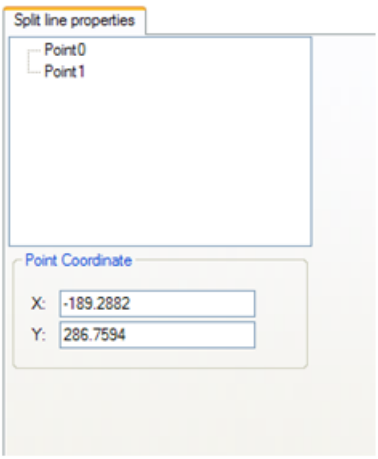


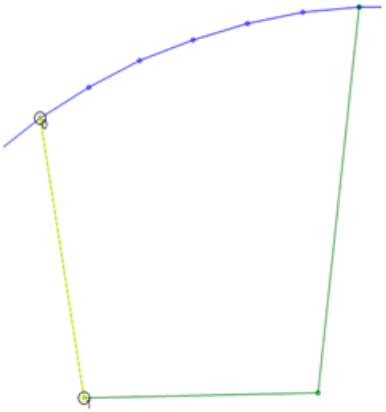

### 3.3.4 Properties Editing Area

The properties editing area contains two editors: Split line properties editor and Patch properties editor.

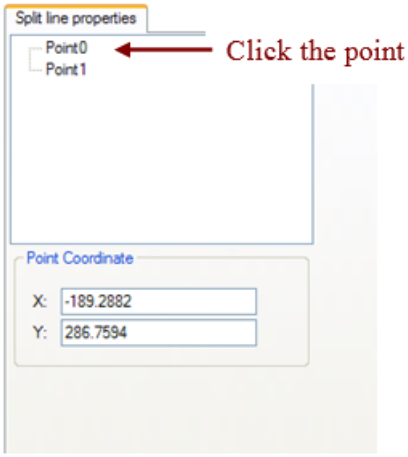
## Split line properties editor

Select a split line (highlighted in yellow), the editor will shows in the properties editing area.



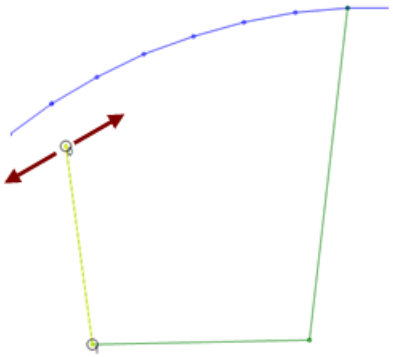



The coordinates of any point could be changed manually, simply by clicking the point on the list then enter the new coordinates in the textboxes.



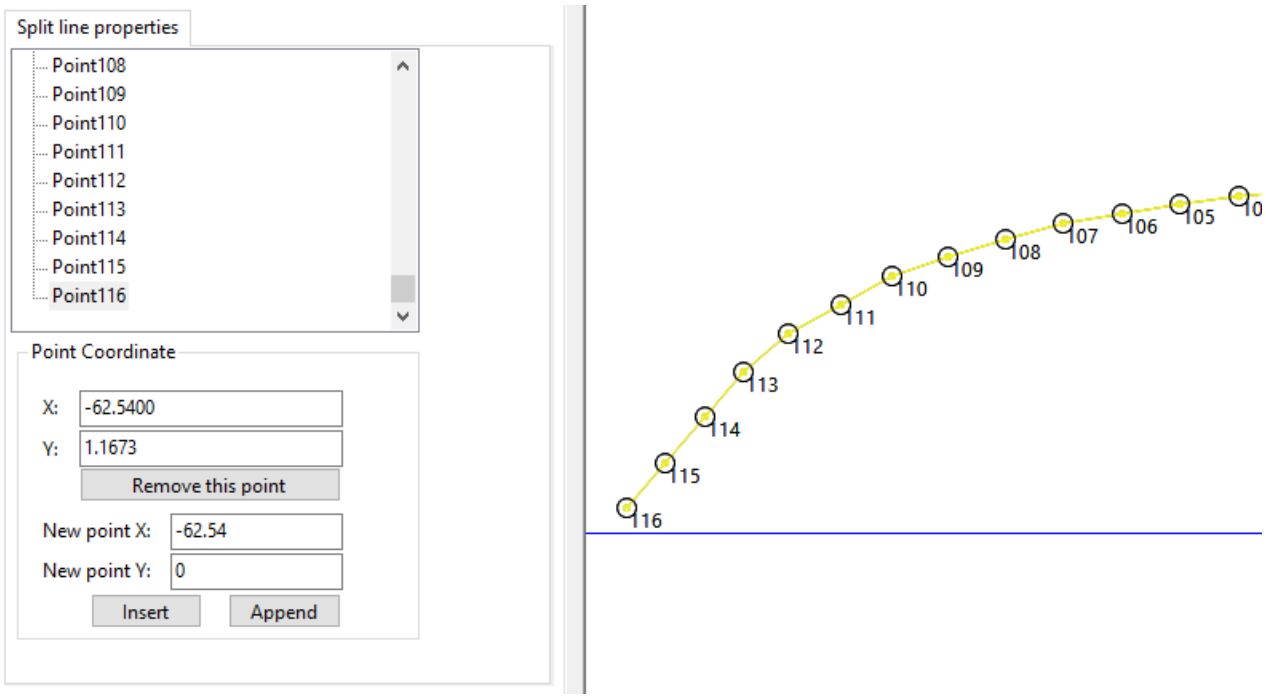

The point's location can be changed by dragging it as well.

Select the point and drag



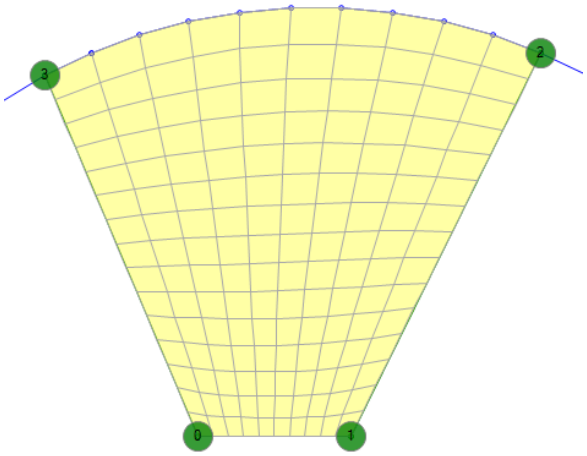
Users can edit waterline using the split line properties editor.

- Update the coordinate of the points of the selected waterline.
- Delete the selected waterline point.
- Insert a new point in front of the selected point of the waterline.
- Append a new point after the selected point of the waterline.

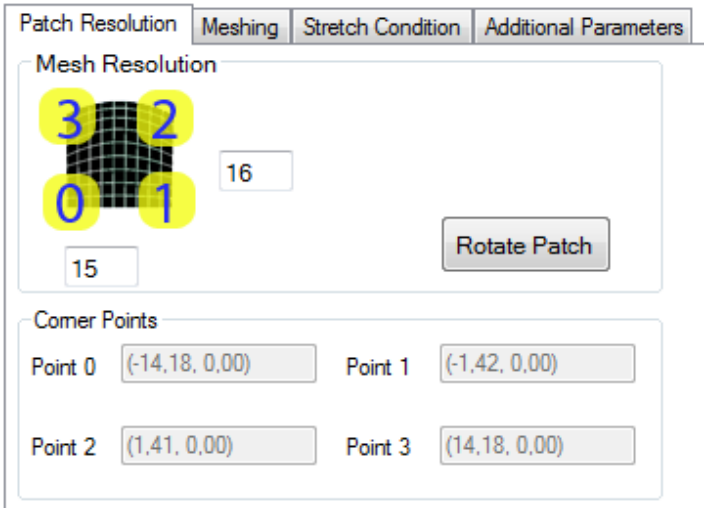



## Patch properties editor

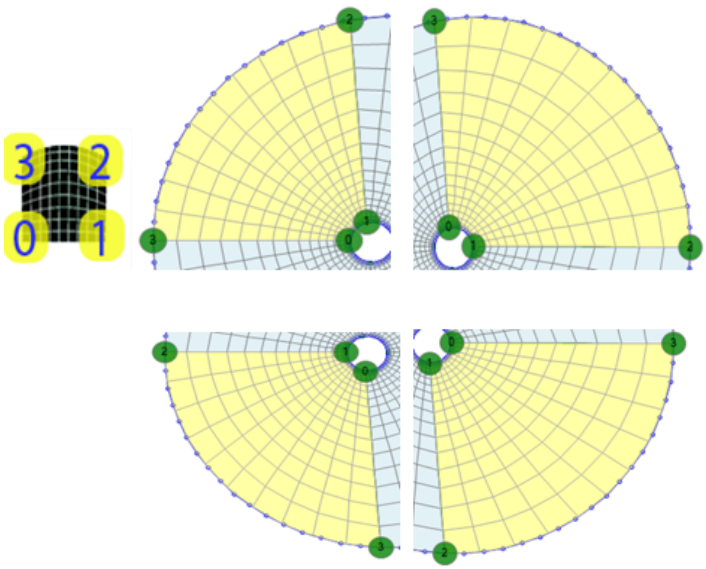
**Patch Resolution**



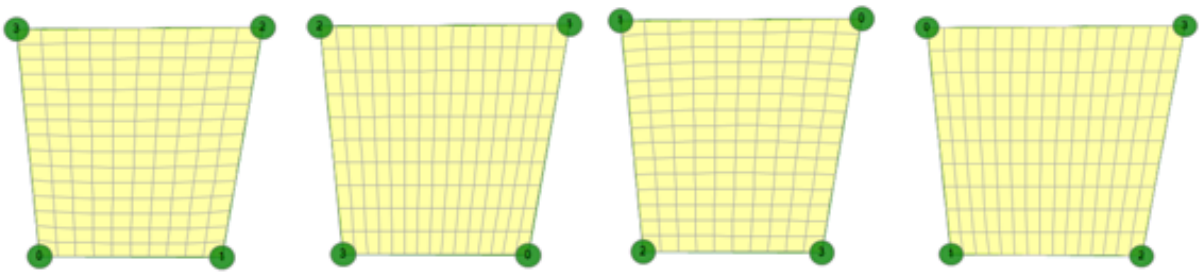

When a patch is selected in the operating area, the patch properties editor will shows up.




Mesh resolution can be changed by input new values in the textbox as shown in the picture above. The corner points' group box shows corner points coordinates (The coordinates are not editable, but the corner points can be moved around the edge of the patch. Sometimes this is necessary since HydroMeshSetup does not always recognize the correct corner points.).
It's highly recommended that the corner order of the currently edited patch in the operating area matches exactly as the corner order displayed in the Patch Resolution tab as follows:

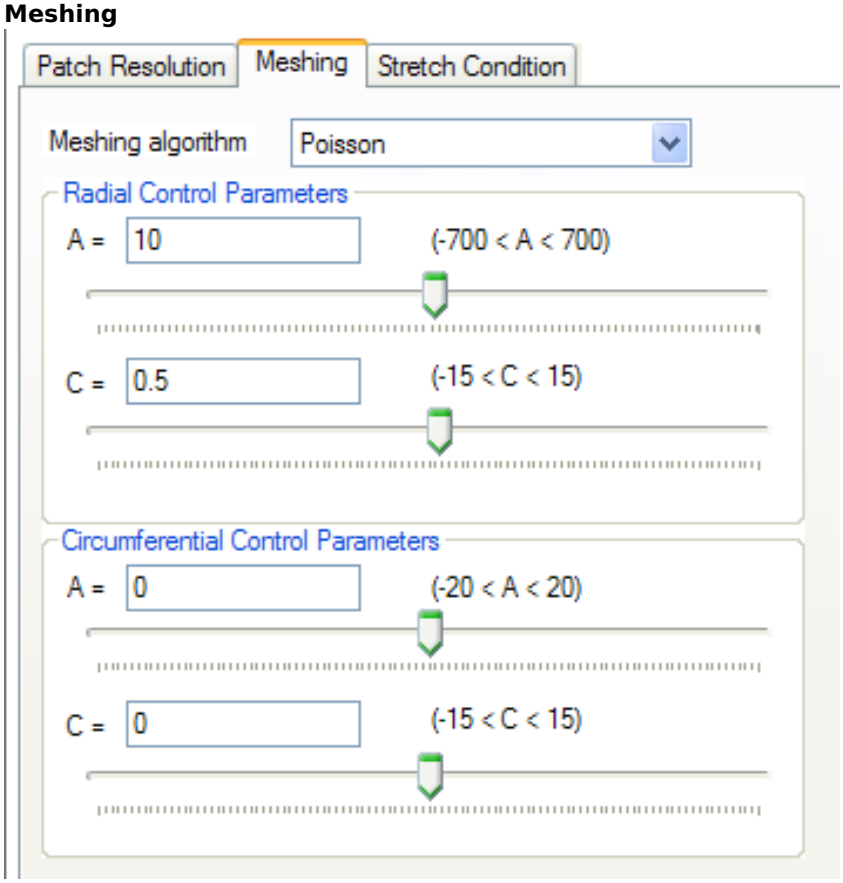



As above pictures shows, the edge marked from 0 - 1 for each four patches in four different quadrants are near the centre of the free surface physical domain.
In order to achieve this, you could click the "RotatePatch" button; it will rotate the corner 90 around the counter clockwise direction for each click as follows:



$0^0$ $90^0$ $180^0$ $270^0$

**Meshing**



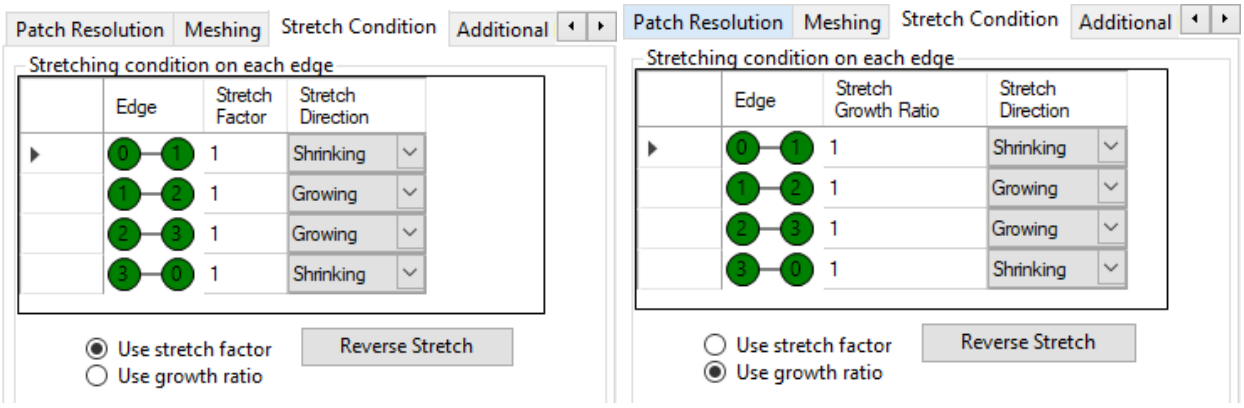

In "Meshing" tab page, you could control the interior mesh orientation both at radial and circumferential direction. You could also select the meshing algorithm between linear interpolation and Poisson meshing solver. HydroMesh uses the following Poisson control function for both radial and circumferential mesh control:

$$P = Q = -ae^{-c}$$

Note that the linear interpolation works correctly for patches with convex boundary shape, for complex concave boundary shape, you'd better use Poisson instead.

**Stretch Condition**

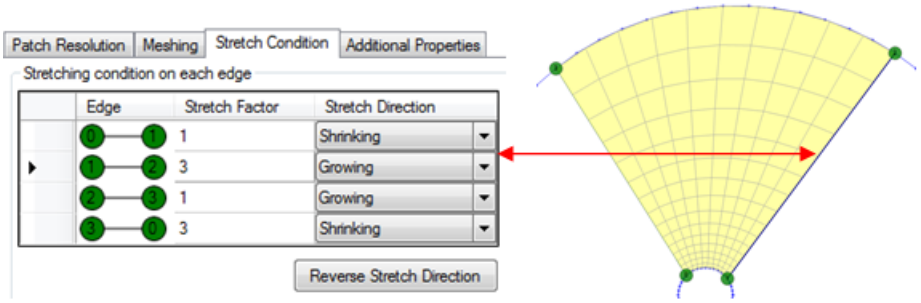
"Stretch Condition" tab page could define the point distribution (stretching) for a specified edge. Users can




use two methods to define the point distribution:

- Stretch factor: define the ratio between the shortest and the longest sections.
- Growth ratio: define the ratio between the neighbor sections.

The following is the definition for each column:
**Edge:** List the four edges which you could define stretch condition for.
**Stretch Direction:** This could change direction / orientation the point distribution should follow.
**Shrinking:** Element size will be decreasing from starting point to ending point.
**Growing:** Element size will be growing from point starting point to ending point.
**ShrinkingFromCenter:** Element size will be decreasing from the center point to both starting point and ending point respectively.
**GrowingFromCenter:** Element size will be growing from the center point to both starting point and ending point respectively.
When you select edge from the "Stretch Condition" tab page, the corresponding edge will be highlighted in the operating area:




**Reverse Stretch Direction:**The button is used to reverse the stretch direction for all 4 edges.

**Additional Properties**

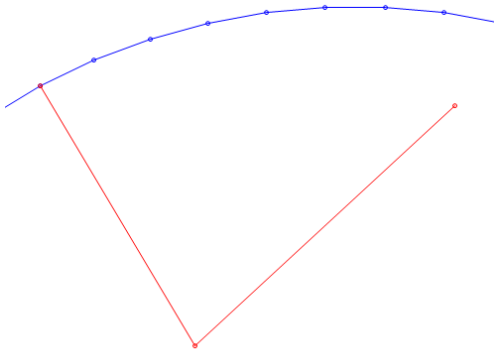
Free surface damping index can be assigned, so that WASIM will take the integer as the index to find the free surface damping coefficient associated to this patch. The free surface damping coefficient is defined in HydroD, associated with the free surface damping index. The default value is 0, which means no free surface damping will be assigned.

### 3.3.5 Operating area

#### Draw Line Mode

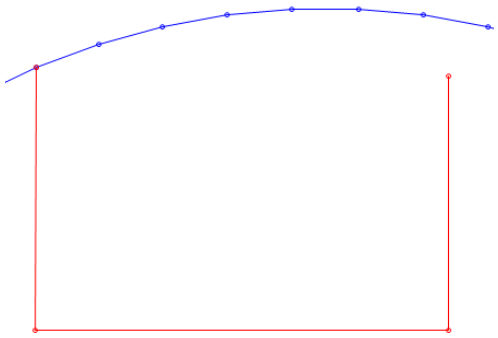
You can go to the "Draw Line Mode" by press the button in the toolbar.

- Draw split line (see 3.3.2 tool bar buttons)

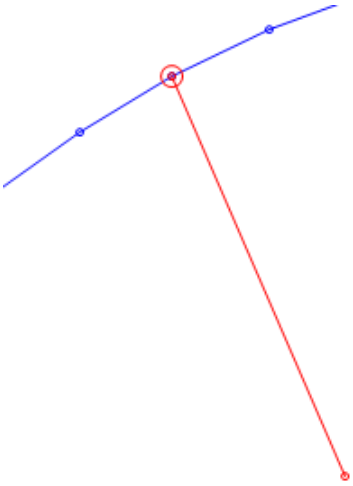


  * Split line shows in red when creating, and will not take effort before "Enter" or "whitespace" key is pressed.

- Draw orthogonal (horizontal or vertical) split lines by pressing shift key when moving the mouse to draw the lines as follows:



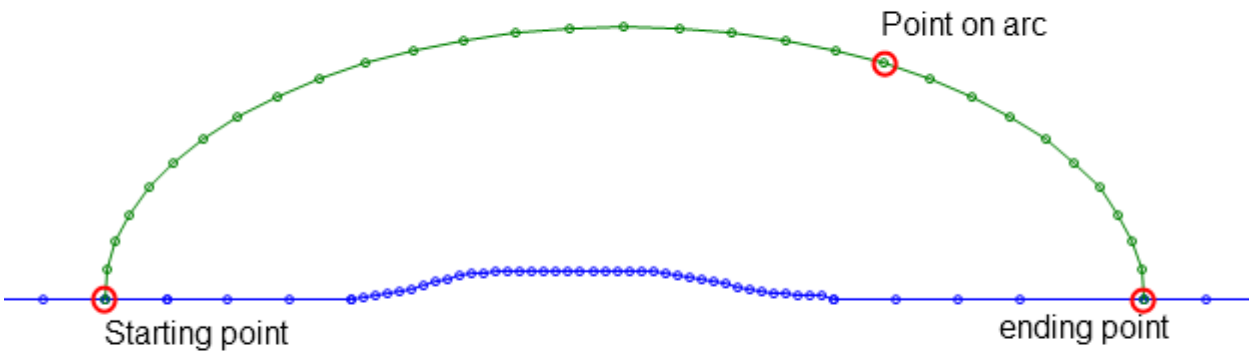
- The program will automatically snap to the intersecting points when you move the mouse near those points as follows:



* The red circle will be displayed when point are snapped.

## Draw Elliptical Arc

You can go to the "Draw elliptical arc" by press the button in the toolbar. To draw an elliptical arc, you need to use mouse to specify three points as follows:




## Select Split Line Mode

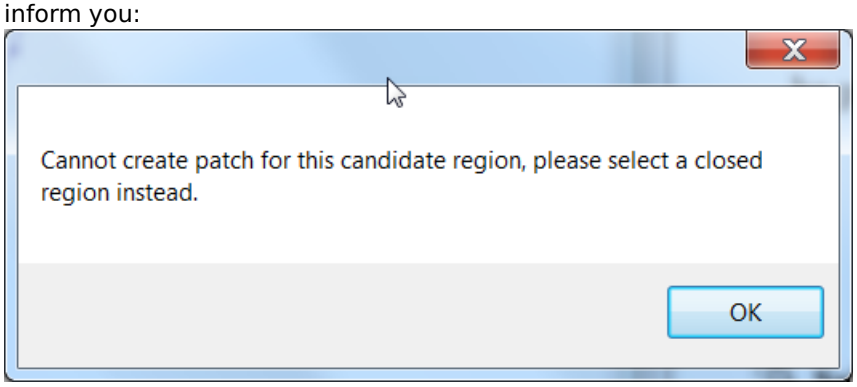
You can go to the "Select Split Line Mode" by pressing the button in the toolbar.

- Move split line nodes (see 3.3.4 Split line properties editor)
- Select split line using mouse
- Delete selected split line[s] by either pressing "delete" or "e" key or press button in the toolbar.
- Multi select split lines by pressing "shift" key, then subsequently use mouse to select lines.

## Create Patch Mode

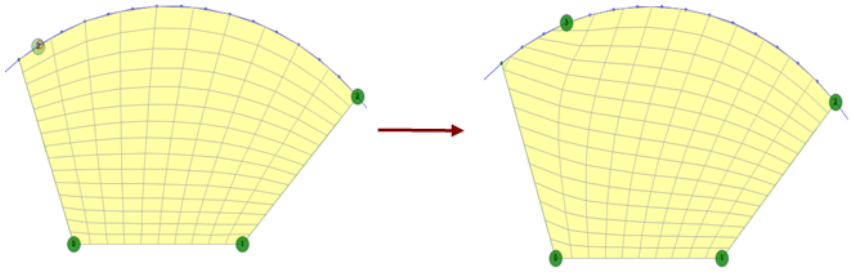
You could go to the "Create Patch Mode" by clicking the button in the toolbar. Then you could use mouse to select to closed regions to create patch.

* You could select a closed region; otherwise no patch will be created, and a dialog will be popped up to inform you:

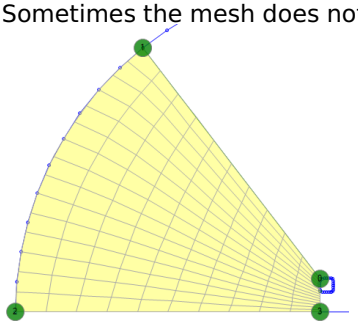



- Moves patch corner points
  You could drag one of the patch corner points to the desired intersecting points for the patch, when

you drag the corner near a specified point, a red circle will be displayed as following left hand side picture shows, when you could release the mouse, the program will automatically update the patch as the following right hand side picture shows:



* Sometimes the mesh does not look nice as follows:



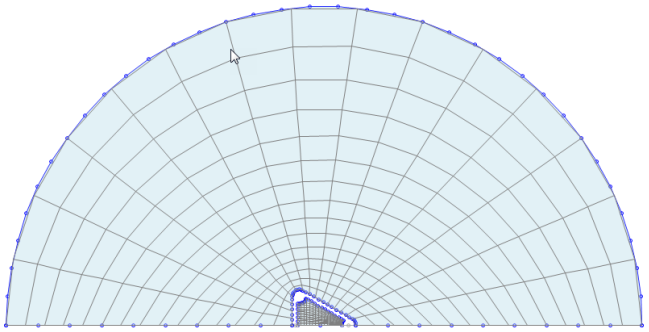
One simply way to fix this issue is going to the patch property editor, select “Mesh Resolution” tab page (see 3.3.4 "Patch Resolution" for further reference). Then you could try change the order of the corner points by continuously click the "RotatePatch" button until the edge marked as 0 – 1 is near the center of the free surface.
Another way to fix this issue is go to the "Poisson" tab page (see 3.3.4 "Poisson" for further reference) to control the circumferential and radial control parameters.

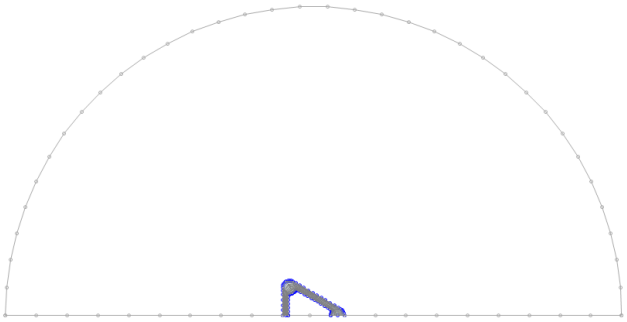
- Delete selected patch by either pressing "delete" or "e" key or press  button in the toolbar.
- Create patch. You could create patch in "Select Patch Mode" by pressing shift key, then use the mouse to pick up a closed region to create patch, after the patch is created, it will be in selected mode. Then you could do further operations on it such as redefining corner points, stretch condition and Poisson control parameters.
- Select patch edge for stretch condition definition.
  When a patch is selected, you could move the cursor near one of its patch edge to select the edge, then the "Stretch Condition" tab page will be displayed, the subsequent operation is the same as specified in 3.3.4 "Stretch Condition".
- Copy & Paste patch properties.
  If you want to share patch property settings across different patches, you could select a source patch first, then press CTRL + C to copy those patch properties into clipboard, then select another patch, press CTRL+ V to paste the patch properties to the destination patch, then the two patches will share the same patch property settings.
- Patch multiple selection.
  You could select more than one patch by pressing the CTRL key, and use the mouse to pick the individual patch you want to select.

 Switch Active Area Button

The free surface domain is divided into 2 parts: the exterior fluid domain and wet hull interior domain. By default, the exterior domain will be selected for free surface meshing and one cannot mesh the interior domain. By clicking this button a user can shift to the interior domain to be able to make free surface mesh

on the interior domain. Note that clicking this button will destroy existing mesh. A warning will be given. One can notice that the inactive boundary lines (on the outer circle and horizontal axis or vertical axis when applicable) are shown in gray whereas the active boundary lines are in blue. The below figures illustrates this:

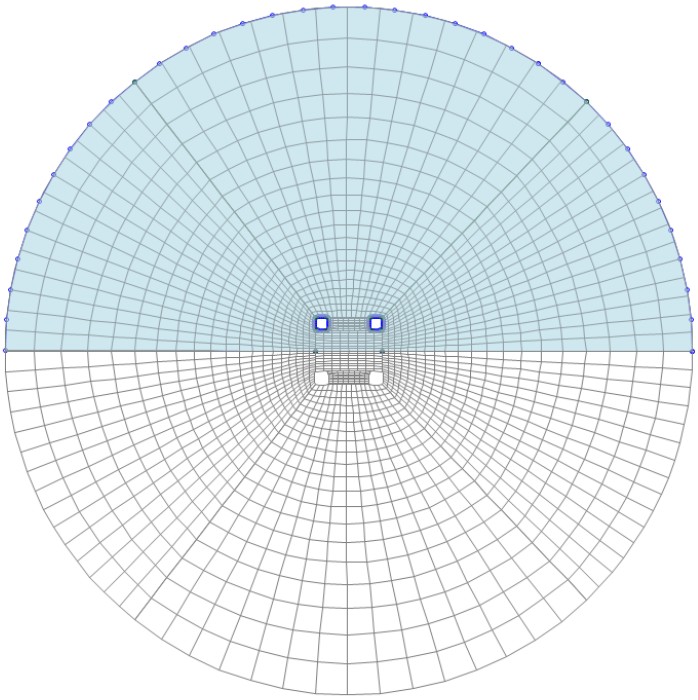




This feature is relevant when a user want to make internal free surface lid as input to Wadam for irregular frequencies removal.

### Show Mirrored Mesh

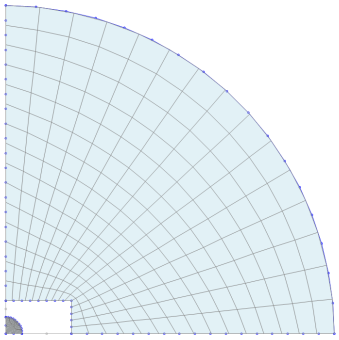
See 3.3.2 "Tool Bar Buttons". This option is available only when section model is symmetric.



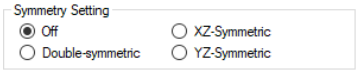
*The mirrored patches are displayed in white color.

### Symmetry

Input symmetry conditions is determined by the coordinates of the waterlines found. If all coordinates have $y > 0$ , xz-symmetry is assumed. If also $x > 0$ yz-symmetry is also assumed resulting in waterlines only shown in the first quadrant as shown below:



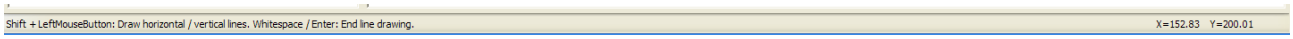
For SSG files symmetry will only be determined if ISYM=1 is specified on the SSG file. It is possible to override symmetry in the GUI setting below:




### Zooming

It is possible to zoom in and out with + and - keys and the mouse scroll wheel.

### 3.3.6 Status Bar Area



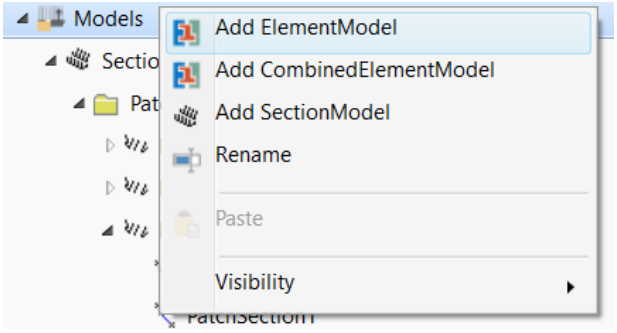

- The left hand side of the status bar shows the "helper message" telling you the different operations you could perform in each mode, "Draw Split Lines" mode, "Create Patch" mode etc.
- The right hand side of the status bar displays the coordinates of cursor position.

## 3.4 Integration with HydroD

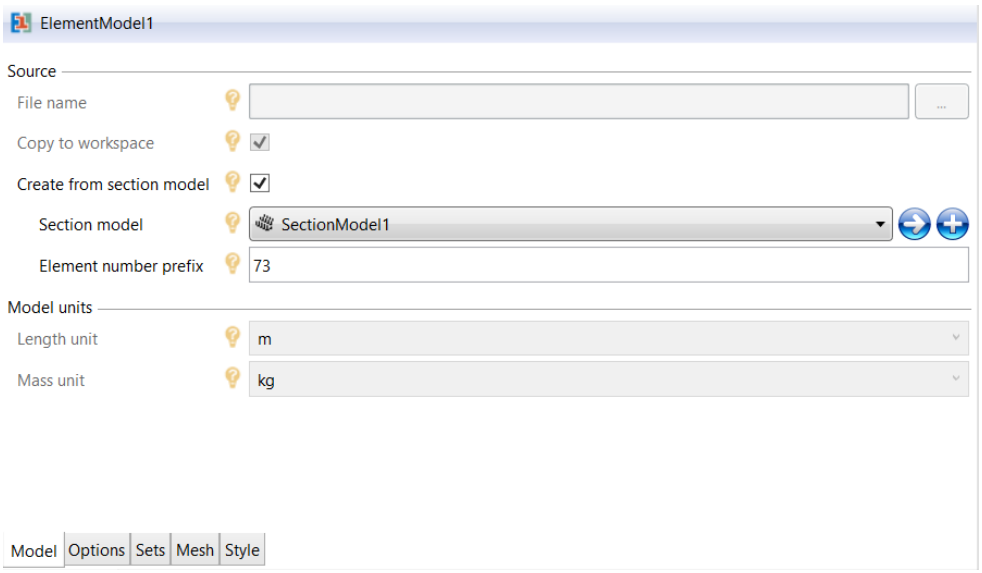
### 3.4.1 Generate ElementModel

ElementModel provides the mesh on the hull from a SectionModel. It is served as the basis for defining a loading condition and as the PanelModel for a WadamAnalysis or a WasimSetupAnalysis .
It is possible to use HydroMesh to generate the ElementModel based on the patches given in a Section-Model:

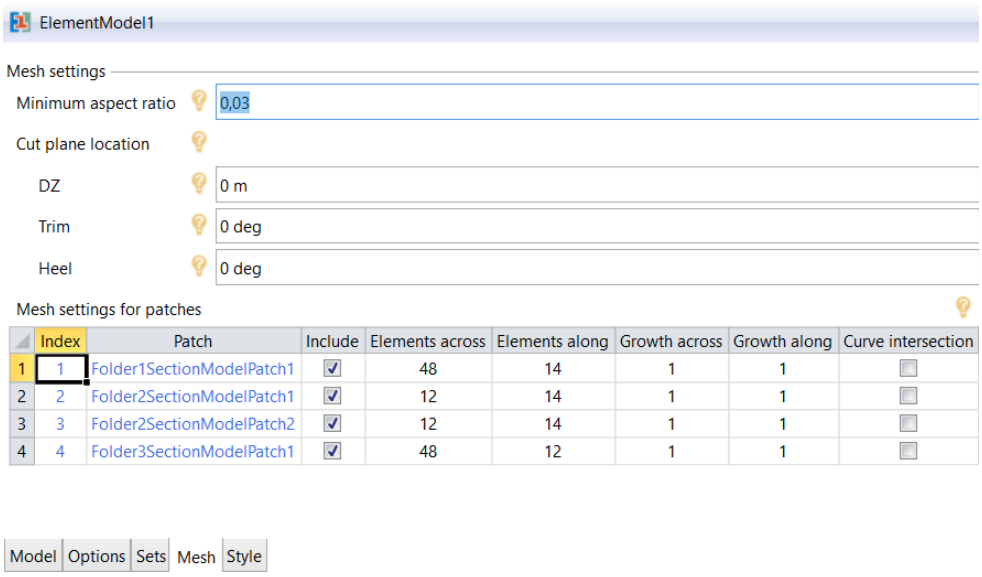
1. Add an ElementModel under Models



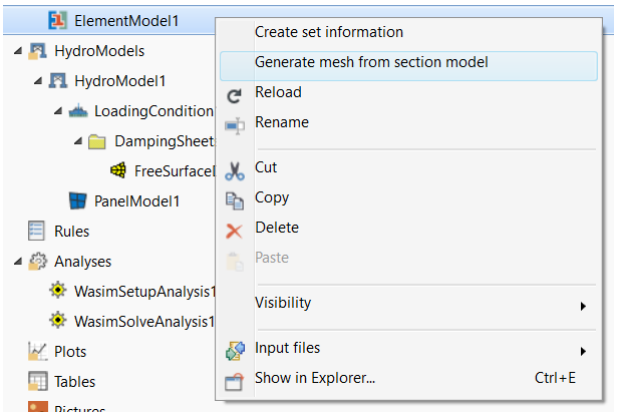

2. In the Model tab of the ElementModel's properties, check "Create from section model", and select the "Section model" from the list of available section models.




3. The mesh settings are available in tab "Mesh".



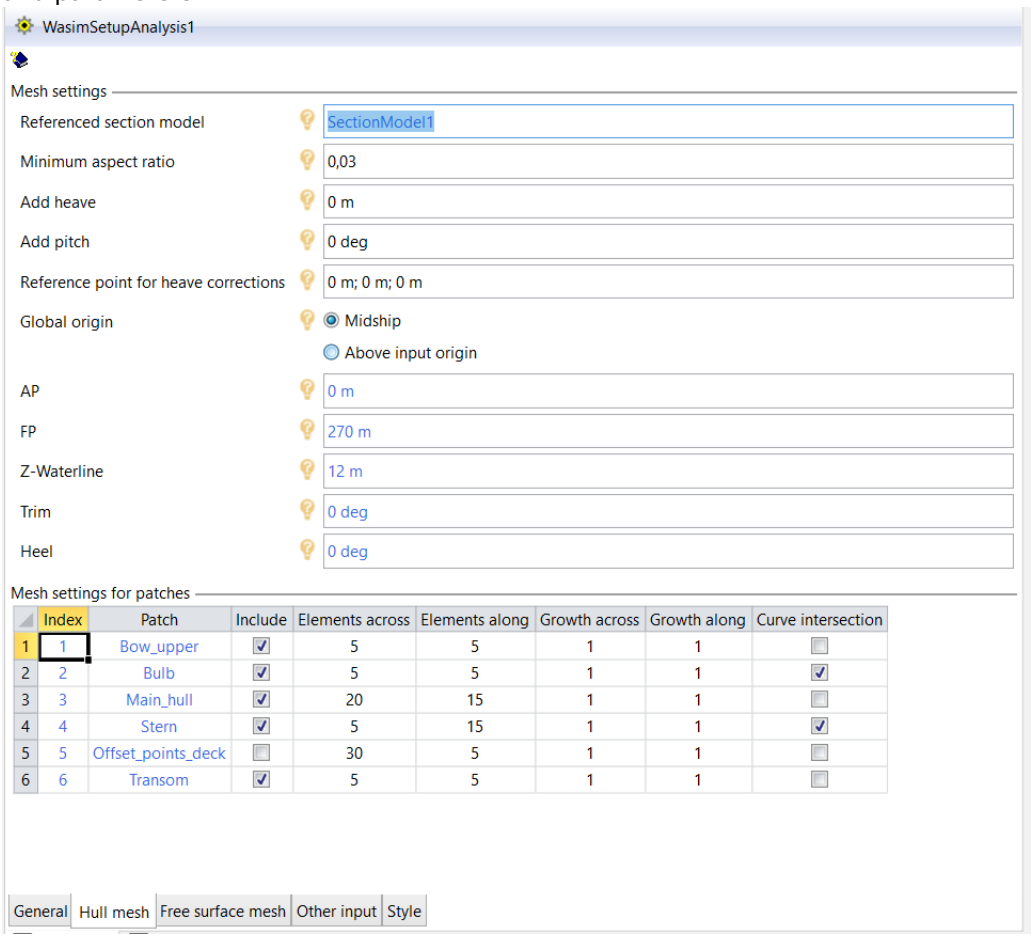

4. RMB on the newly created ElementModel, click "Generate mesh from section model". HydroMesh will be called by HydroD to generate the hull mesh.



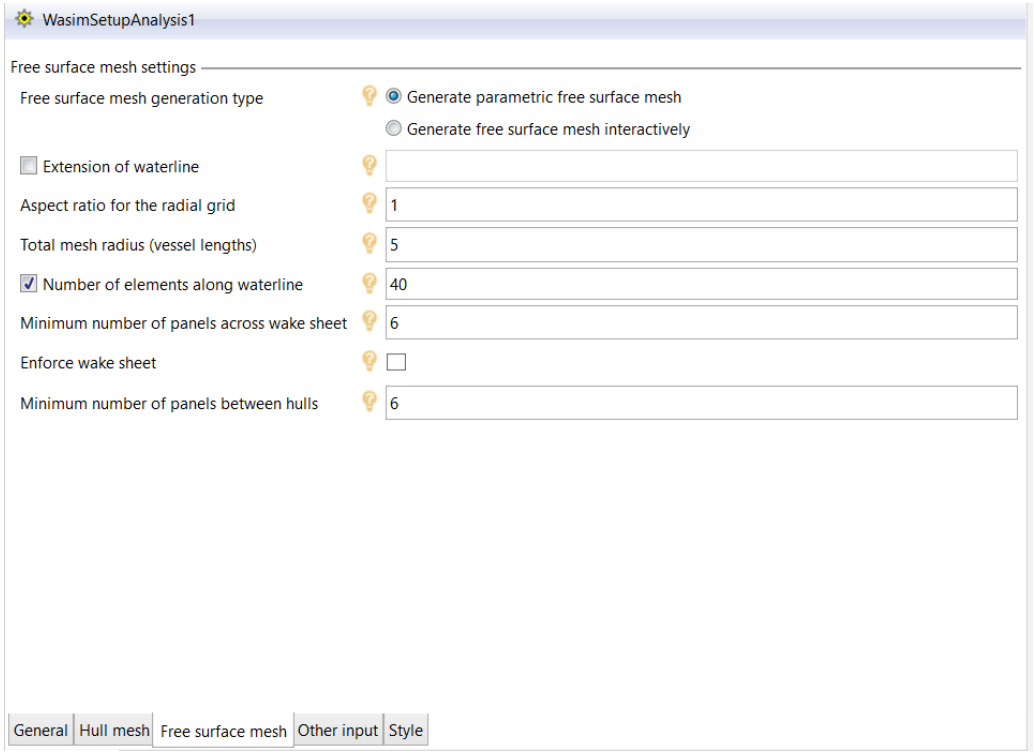

Be noted that the growth of mesh size in longitudinal direction is towards -x axis, and in girth-wise direction is from keel towards waterline. For dry patches above the waterline, or the dry part of the patches cutting waterline, the growth ratio will be always fixed to 1.0. This means that growth ratio will take effect only on the mean wet hull.

### 3.4.2 Section Mesh

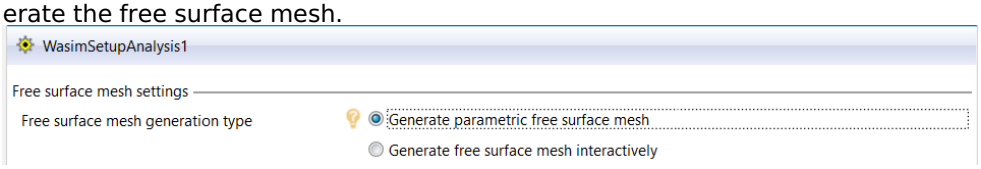
Section mesh provides the mesh for WasimSetupAnalysis, both on the hull and on the free surface. Comparing to the ElementModel mesh tab, the "Hull mesh" tab for WasimSetupAnalysis contains more information and parameters.



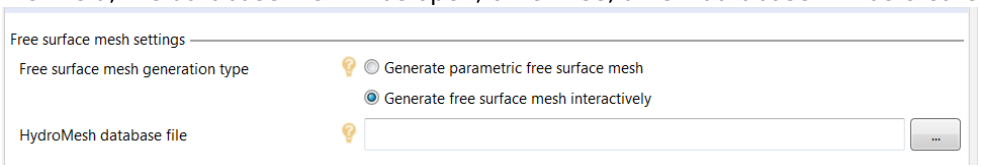

There is an additional tab for the meshing parameters for the free surface.



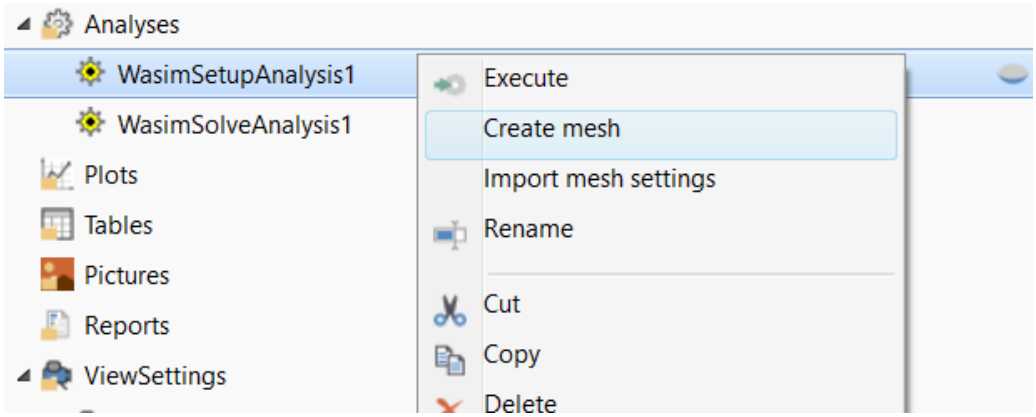

1. If "Generate parametric free surface mesh" is selected, HydroMesh will be called to automatically generate the free surface mesh.




2. If "Generate free surface mesh interactively" is selected, HydroMeshSetup will be called, which allows user to create free surface mesh interactively. If a valid database is given in the "HydroMesh database file" field, the database file will be open, otherwise, a new database will be created.



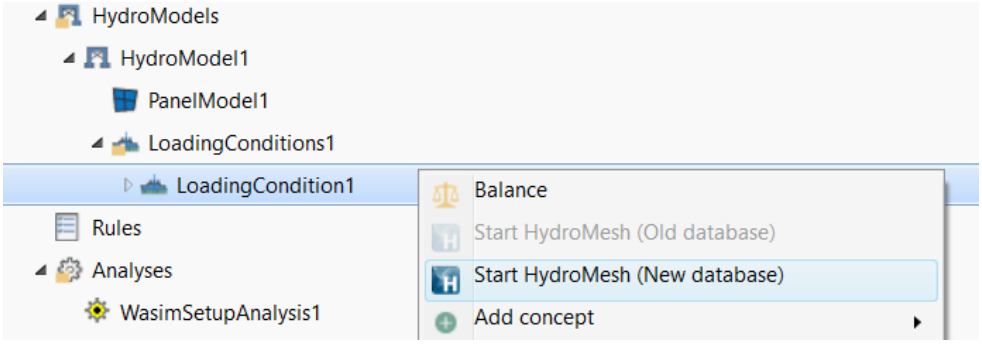

RMB on the WasimSetupAnalysis, and select "Create mesh", depending on the selection of "Free surface mesh generation type", HydroMesh or HydroMeshSetup will be called.



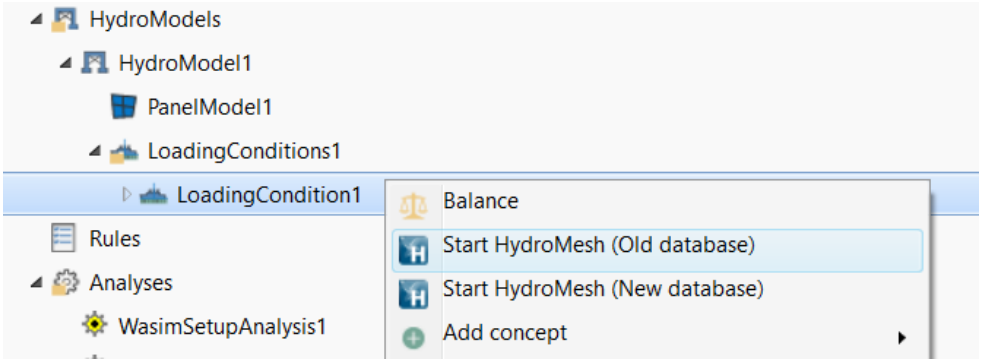

### 3.4.3 Loading condition

HydroMeshSetup can be started from the RMB menu "Start HydroMesh ...." when clicking on a loading condition in HydroD. The waterline will be automatically identified as a basis for free surface meshing.

1. If database file doesn't exist, a new database file will be created.




2. if database file exist, user can choose to open the existing database file, or create a new one.




## 3.5 Data Organization

### 3.5.1 HydroMeshSetup input/output files

#### Input

- "filename"_dry.ssg: dry hull mesh file.
- "filename"_fs.xml: user defined free surface mesh setting file.

- "defaults": configuration file produced by HydroD. Contains information about translations, rotations, input file name etc.

### Output

- Dry hull mesh file: "filename"_dry.ssg and "filename"_dry.geo (irregular mesh file)
- Wet hull mesh file: "filename"_wet:ssg and "filename"_wet.geo (irregular mesh file)
- Sesam interface file: "filename"TXXX.FEM containing both the dry and wet hull.
- Free surface mesh file: "filename"_fs.ssg and "filename"_fs.geo (irregular mesh file)
- Combined mesh file (free surface & wet hull) "filename".ssg and "filename".geo (irregular mesh file)

Files using SIF file format:

- T7301.FEM (Dry hull)
- T7302.FEM (Wet hull)
- T7303.FEM (Free surface)
- T7310.FEM (T7301 + T7302 + T7303) + mirror images in the case of symmetry
- T7320.FEM (T7301 + T7302)
- T7373.FEM (Mesh of free surface in global coordinates system)
- T7374.FEM (Symmetric free surface mesh, it will be generated only when the model is symmetric)

Be noted, free surface mesh in SIF (T7303.FEM, T7373.FEM, T7374.FEM) can take a user defined super element number, see Section 3.3.2.

# 4 Requirements to section model

## 4.1 General requirement

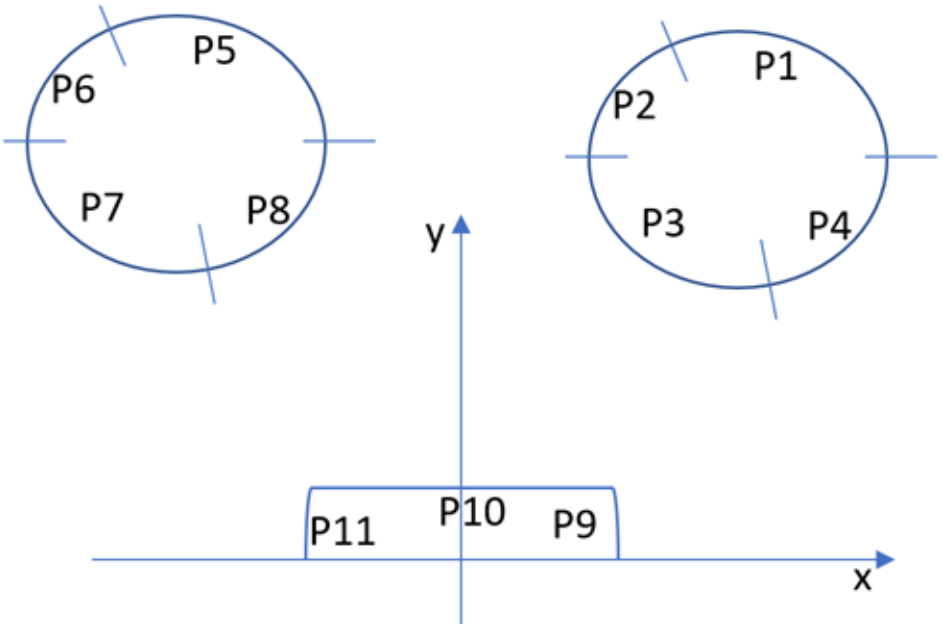
- For patches of type WetRight or WetLeft, i.e. patches that may potentially be intersected by the free surface:
  - If a section intersects the free surface it should intersect from below, i.e. the first point on the section is below the free surface and the last point is above.
  - When you look from the first to the last section:
    - If the wet side is to the right the type should be WetRight
    - If the wet side is to the left the type should be WetLeft
    - If you change the order of the sections you must also change type if you want to keep the same wet side
- Patches of type Submerged must be entirely below the mean free surface at all relevant loading conditions.
- Patches of type Dry must be entirely above the mean free surface at all relevant loading conditions.
- Point ordering for Submerged and Dry patches:
  - When you look from the first to the last section and move from the first point towards the last point the wet side is to the right.
    - Example: If the section defines a closed curve the wet side is the outside if the points are ordered counter-clockwise when viewing from first to last section

## 4.2 Hull meshing

When you are only meshing the hull, i.e. creating an ElementModel from a SectionModel, any section model satisfying the basic criteria above can be used since we just create the mesh patch by patch.

## 4.3 Parametric free surface meshing

HydroMesh offers the possibility to perform automated meshing of the free surface based on a set of parameters. This will put additional requirements to the section model. The waterline below will be used as example, both for this and the interactive meshing section.




The meshing is done in the following way: First a set of waterlines is created. This will always be an odd number if there is one plane of symmetry and an even number if there is no plane of symmetry. All waterlines are “aligned” with the x-axis. In this example we have one plane of symmetry and three waterlines will be created:

- Waterline 1 (w1) is created from patches P1, P2, P5 and P6. HydroMesh will join the patches P2 and P5 with a straight line.
- Waterline 2 (w2) is created from patches P4, P3, P8 and P7. HydroMesh will join the patches P3 and P8 with a straight line.
- Waterline 3 (w3) is created from patches P9, P10 and P11.

Then mesh patches will be created outside of w1 and between the waterline. In this case there will be two mesh patches on the free surface.
HydroMesh will know that it should start a new waterline when the type change from WetRight to WetLeft or vice versa.
In order to get correct ordering of the nodes on the free surface the following requirements must be fulfilled:

- Points along a waterline must be sorted towards negative x.
- The waterlines must be sorted towards negative y.

This puts the following requirements on the section model:

- All patches defining odd-numbered waterlines (w1 and w3 in the example) must be of type WetRight

and must be sorted towards negative x.

- All patches defining even-numbered waterlines (w2 in the example) must be of type WetLeft and must be sorted towards negative x.

This means that in the section model only one ordering of the patches is allowed:

| Index | Name | Type |
|---|---|---|
| 1 | P1 | WetRight |
| 2 | P2 | WetRight |
| 3 | P5 | WetRight |
| 4 | P6 | WetRight |
| 5 | P4 | WetLeft |
| 6 | P3 | WetLeft |
| 7 | P8 | WetLeft |
| 8 | P7 | WetLeft |
| 9 | P9 | WetRight |
| 10 | P10 | WetRight |
| 11 | P11 | WetRight |



## 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.

DNV | digital@dnv.com | dnv.com/digital