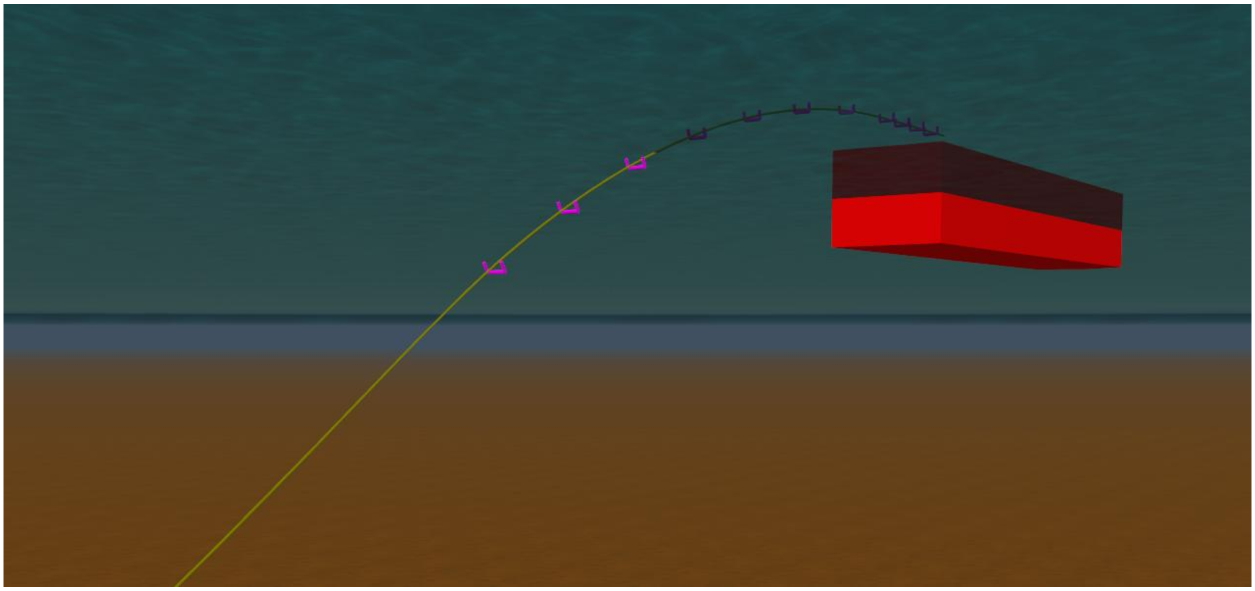
E01 Simple rigid stinger

Introduction

In these examples, two different methods of modelling a simple rigid stinger are presented. Both use the pipelay supports method, with one model using a *simple* supports geometry, and the other using an *explicit* method. The explicit method provides some additional features which are described in the relevant section of this document.

The two models have been created using an identical pipeline and vessel, and the rollers have been positioned in the same places to allow direct comparison of the results. Which method you choose will depend on which you find easiest to build, on the complexity of the desired stinger profile, and on whether you need the additional features made available by the explicit method.



Simple geometry stinger

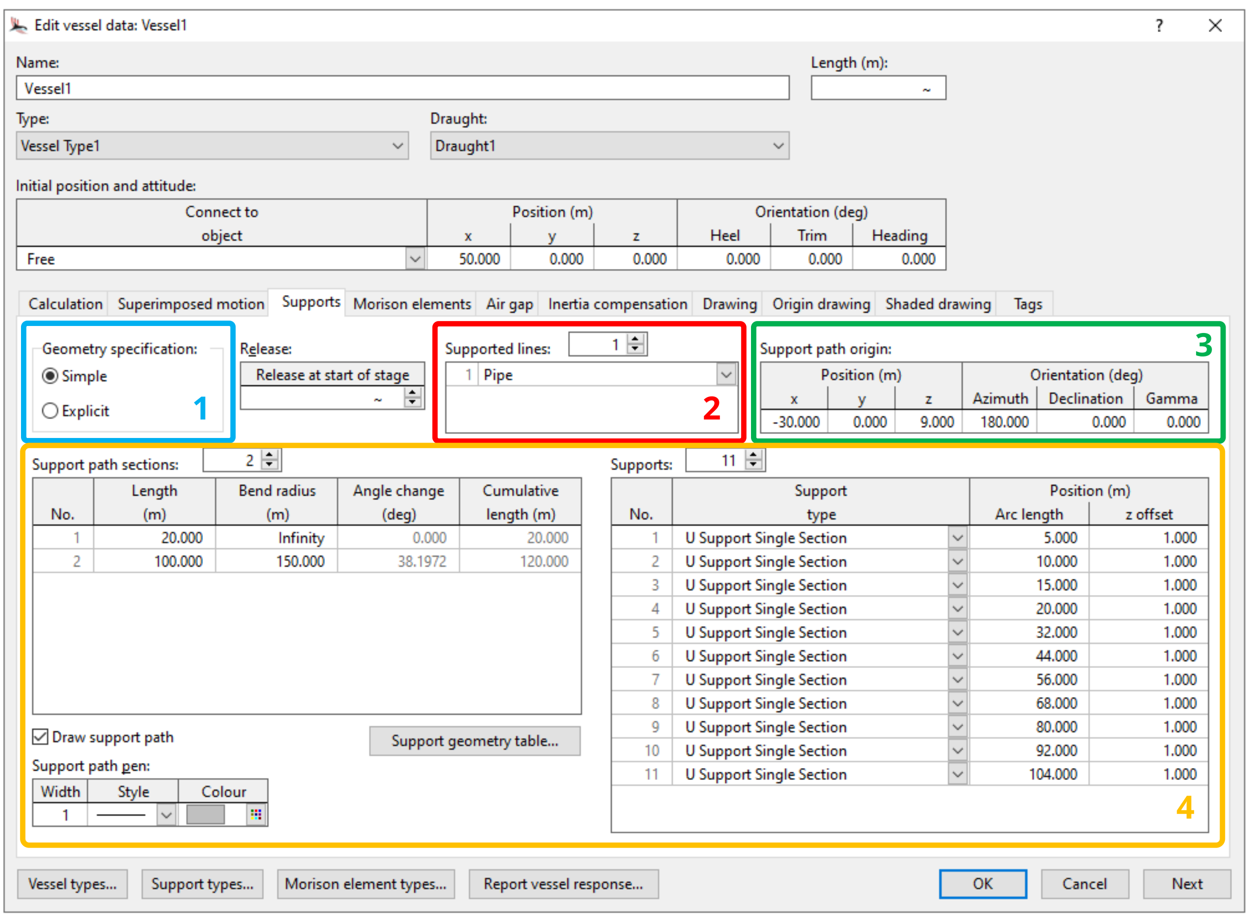
Building the model

The model named *E01 Simple geometry stinger.sim* consists of a lay vessel with stinger, and the pipe being laid.

The supports feature creates cylinders (rollers) of infinite length, and which have a designated contact side with which a supported line can make contact. The supported line is therefore pushed towards the contact surface during statics, until it sits on top of the supports, regardless of where it lies when the model is in the *reset* state.

The lay pipe is built-in at its connection to the vessel (end A) with infinite stiffness and a declination of 90°. The *target segment length* is set to 2 m over the upper section of the line (the roller region) and in the touchdown region. Longer segments are used elsewhere.

The vessel is the OrcaFlex default vessel, and the lay pipe and its supports are placed on the vessel's axis of symmetry. The stinger is modelled using the *simple geometry specification* for generating the supports, which requires an arc length and radius of curvature (useful if you know the radius of the stinger). Since pipelay supports are often attached to the installation vessel, the *supports* feature is accessed through the *vessel* data form, with a dedicated tab as shown below.



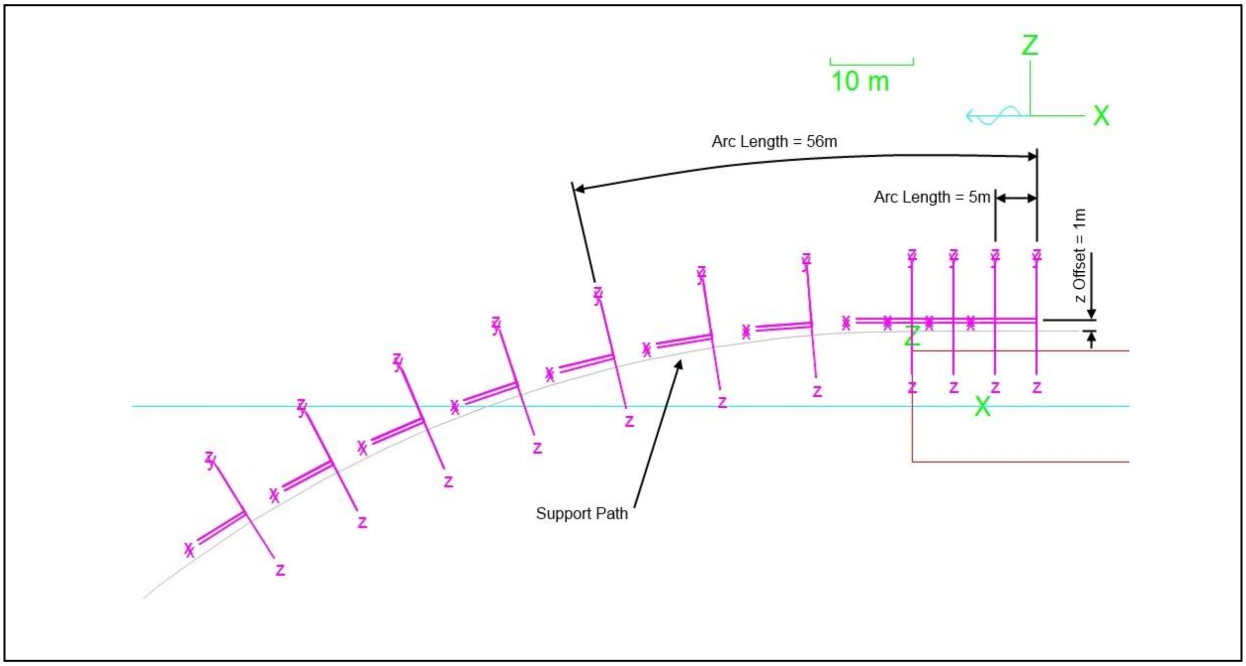
The *supports* page is split into 4 sections, as highlighted on the screen shot above. The *geometry specification* box (numbered as box 1) lets you specify how you position the supports on the stinger. The *supported lines* box (box 2) identifies which lines will be supported.

The appearance of the table(s) shown in boxes 3 & 4 will change, depending on your selection for geometry input. In this case, box 3 identifies the *support path origin*, which can be positioned anywhere on the vessel, and is itself specified relative to the vessel origin. It can also be rotated as required.

In this example model – and shown on the left hand side of box 4 in the figure above – the stinger comprises a 20m straight section (it has a *bend radius* of 'Infinity') and a 100m section with a *bend radius* of 150m. We could add further sections of different radii if required. You can choose to preview the support path by ticking the *draw support path* box, which displays the path in the model, and you have the usual options for pen style, colour, etc.

Once you have specified a path along which to place the supports, you then need to specify their positions (in the table to the right) by means of an *arc length* and *z offset*. The *arc length* is the length along the support path from the *support path origin*, and the *z offset* is the offset from the path in the radial direction. This is shown by the figure below. Looking at the preview in the model with the local-axes visible (*Ctrl + Y*) will help position the supports correctly.

Note that the specified positions refer to the centre point of the roller.



Support types

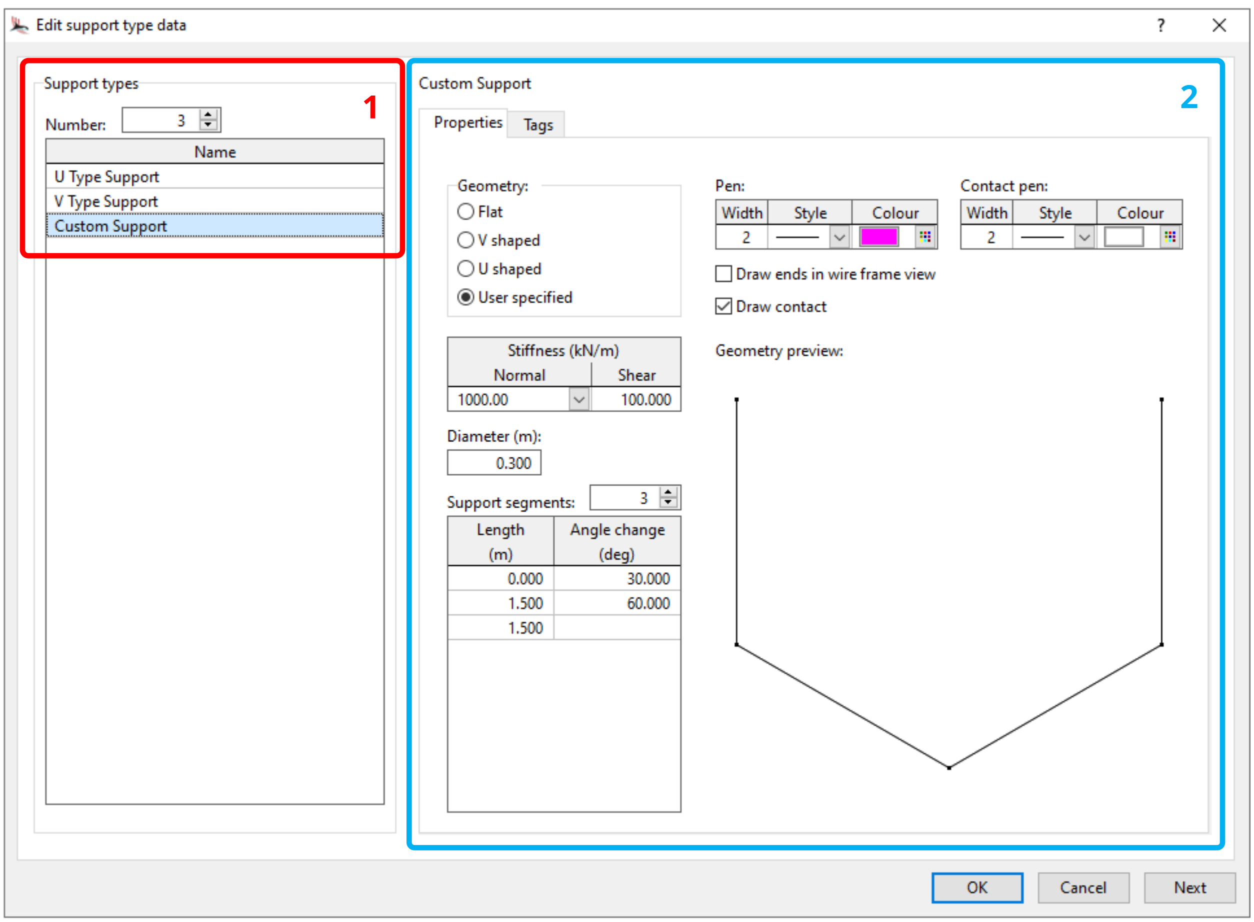
In addition to the support positions, we also need to specify at least one support type. This is done on the *support type* data form, which can be accessed using the button on the bottom of the *supports* page.

You can specify multiple support types to include in the model. This is done in the usual OrcaFlex way, i.e. by first setting the *number* of support types and naming them (see box 1 below). Each support type is then defined by its geometry, as shown by box 2.

The support type *geometry* options are flat, V shaped, U shaped and user specified. The user specified option lets you configure your own set of support sides, for example the v-bottomed, vertical side arrangement shown below.

The support must be given a contact stiffness and some dimensions (including diameter). The *geometry preview* area shows the profile the resulting support will have. The *diameter* defines both the diameter used in the subsequent calculations and the diameter shown in the shaded graphics view.

Friction effects can be included between the pipe and the supports, if required. This is defined on the *friction coefficients* data form, which is accessed from the *model browser*.



Initial set up

In general, a pipelay configuration will be set up to achieve a specific level of top tension. In OrcaFlex this can be done using the *line setup wizard* which is accessed through the *calculation* menu on the OrcaFlex main window. This allows either the line's length or its anchor position to be altered to achieve a target condition. In this model the wizard was used to alter the anchor position to achieve a top tension of 150 kN in still water.

Open the *environment* data form. On the *waves* page, the environmental conditions in the example are set to consider the JONSWAP waves, with Hs 1m, Tz 4s, 45° off the bow of the vessel. On the *current* page, a slab current is applied, with a speed of 0.5 m/s, 20° off the bow of the vessel.

The *waves preview* facility was used to identify the largest event in the first three hours of the simulation and the *simulation time origin* on the *waves* page has been set accordingly, such that a 150 s simulation models the largest wave in the middle of the simulation. The applied wave elevation can be seen by clicking on the *view profile* button on the *waves preview* page of the *environment* data form (note that the *position* used in the waves preview is the point where the pipe connects to the vessel, which is the position where we want to detect the largest wave).

Results

Loading the simulation file also loads the default workspace so results summaries are automatically generated.

The range graph of *effective tension* (top left) shows that the maximum tension at the tensioner is about 200kN, which is close to 1.5 times the static tensioner load. It also shows that the entire pipe remains in tension throughout the simulation.

The time history of *effective tension* at end A (top right) shows how the tensioner load varies. The range graph of *maximum von Mises stress* (bottom left) shows that peak stresses are below 300 MPa at all times.

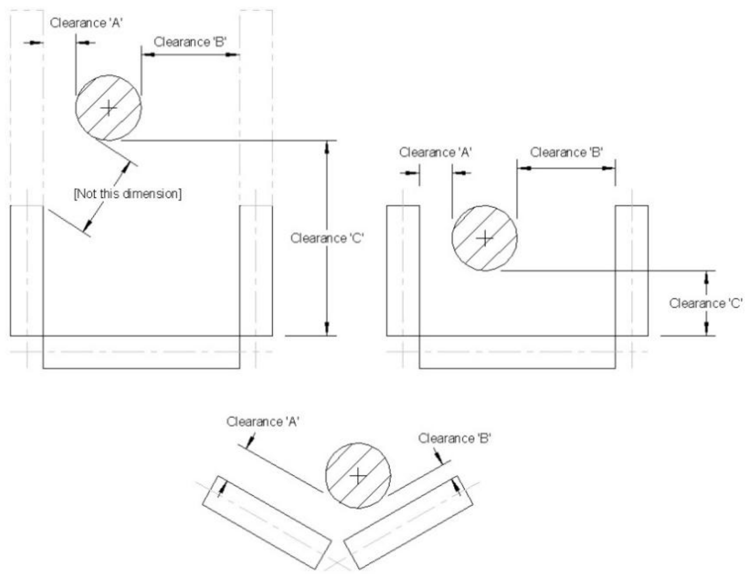
Note that the analysis uses a constant normal drag coefficient of 1.2 for the pipeline. This simple approach is likely to be conservative and the use of a variable drag coefficient based on the local instantaneous Reynolds number would probably reduce stress levels.

For further details about modelling variation of line drag with Reynolds number, see the example A01 Catenary and wave systems.

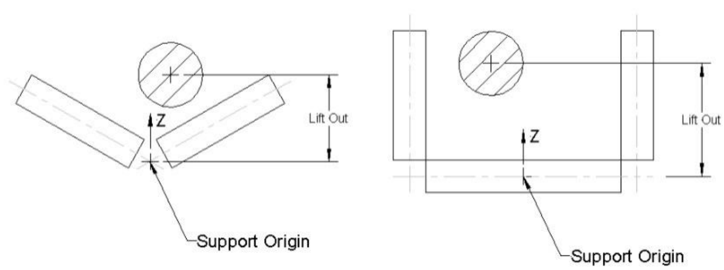
In addition to the normal OrcaFlex results, there are several results available relating to supports. If you run the dynamic simulation, you will find that time history results for the *vessel* object include:

- *Support reaction force* and its components (*Support reaction x*, *y*, and *z force*)
- *Support contact clearance* – This is the minimum clearance between the supported line and all of the support surfaces of a particular support e.g. for a U-shaped support it will report the minimum clearance to whichever support surface (base or sides) is closest. Note that OrcaFlex projects the cylinders of a support to infinity, so if a line is sitting in free space above a support, it will still report the closest approach to the support as if the support extends up and beyond the line.

In the examples below, OrcaFlex will calculate clearances A, B & C (or just A & B in the case of a V-shaped support) and report the contact clearance as the smallest of these values i.e. the distance from the line outer surface to the closest support surface. A negative value indicates contact between the surface of the line and the support surface.



- *Support lift out* – This helps to assess if a line is at risk of lifting out of the roller box. It is defined in simple terms as the distance, in the support's z-direction, from the support's position to the centre point of the line passing over the support. Note, the distance is calculated from axis-to-axis, not surface-to-surface. Note that the *support origin* is not shown in the GUI.



- *Support contact arc length* – The arc length of the point on the line closest to the specified support.
- *Support off end contact distance* – This is useful for checking if a line has escaped the confines of the specified support.

You can also get the results reported for the force and moment exerted on to the vessel from all the supports assigned to it. These results are:

- *Supports force*, and its components.
- *Supports moment*, and its components.
- *Connections + supports force* and its components.
- *Connections + supports moment* and its components.

Explicit geometry stinger

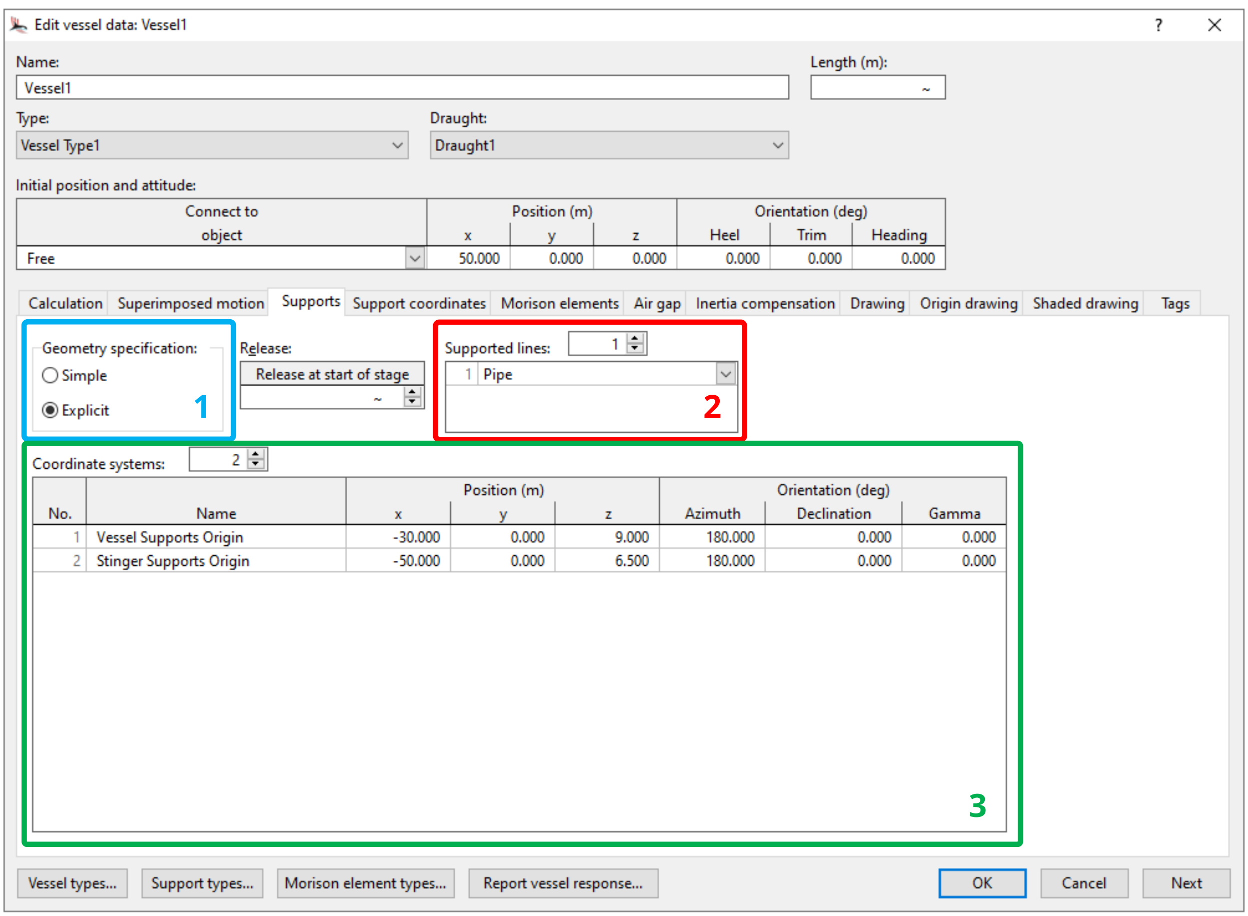
Building the model

The model named *E01 Explicit geometry stinger.sim* uses an identical arrangement to the previous model. The difference here is the way the supports are positioned.

The *simple* method, used in the example described above, is useful for determining an initial approximate arrangement of rollers on a stinger. However, such simplicity is not normally available on a real stinger, where rollers can only be fixed in a number of discrete positions. Also, these roller positions are often defined relative to an axis system that rotates with the stinger, rather than being fixed to the vessel; the latter is the case for the *simple* option.

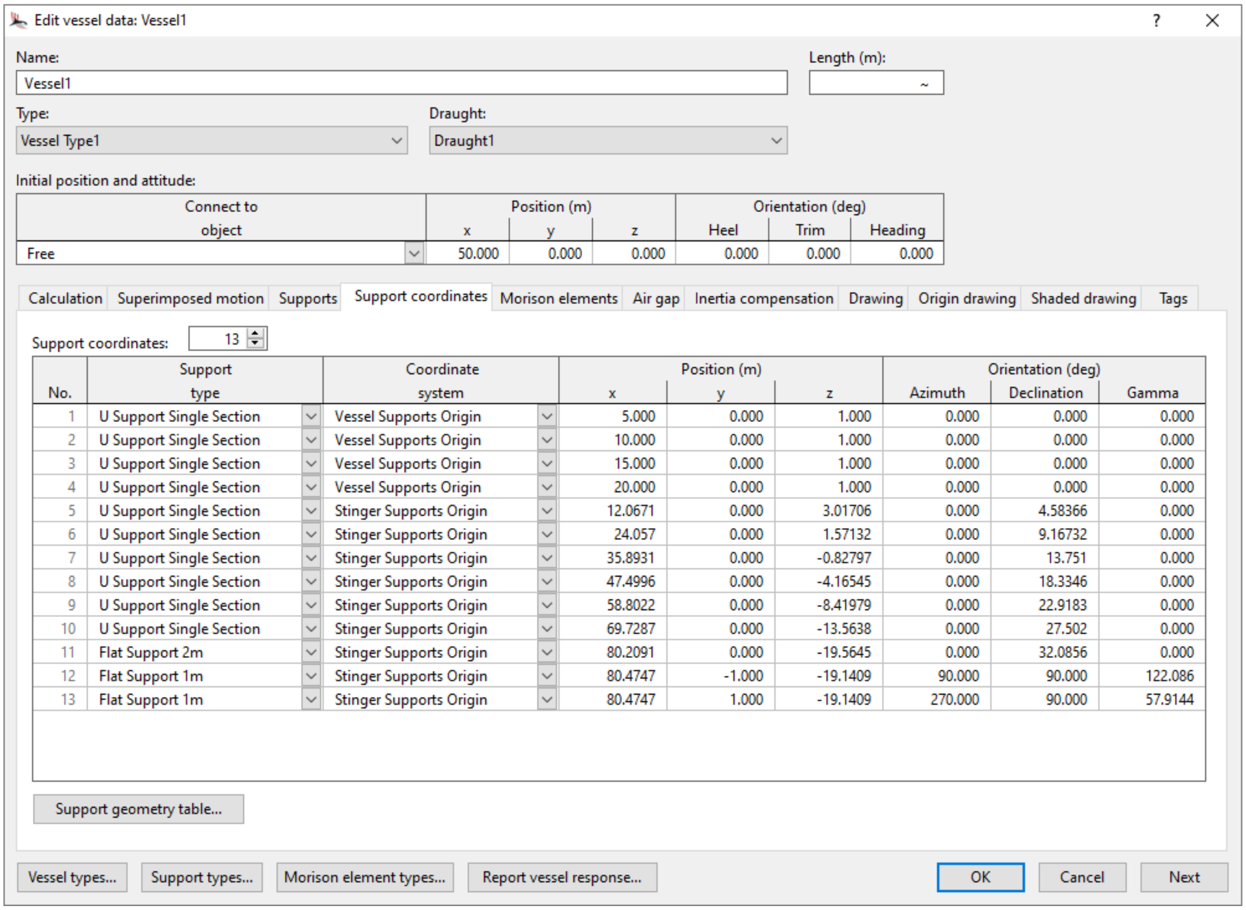
The *explicit geometry specification*, used in this model, addresses both these issues. This option requires you to know the positions of your supports in Cartesian (x, y, z) coordinates. You can then position each support relative to a coordinate system of your choice.

Multiple coordinate systems can be defined using the area highlighted by box 3 on the figure below. These coordinate systems are defined relative to the vessel coordinate system (as the supports are attached to the vessel object) and can be positioned and rotated anywhere on the vessel as required.



From the image above, note that two *coordinate systems* have been defined; the first is used to position supports on the vessel, the second is used to position the supports on the stinger (which coincides with the location of the stinger's pivot in this case). This means that the declination angle corresponds to the ramp angle. Moving or rotating one of these coordinate systems means that the supports associated with it will move also.

You will note that selecting the *explicit geometry specification* has enabled an additional page on the data form, called *support coordinates*. This page is where the coordinate *position* and *orientation* are defined for each support, along with the coordinate system it references.



No.	Support type	Coordinate system	Position (m) x	y	z	Orientation (deg) Azimuth	Declination	Gamma
1	U Support Single Section	Vessel Supports Origin	5.000	0.000	1.000	0.000	0.000	0.000
2	U Support Single Section	Vessel Supports Origin	10.000	0.000	1.000	0.000	0.000	0.000
3	U Support Single Section	Vessel Supports Origin	15.000	0.000	1.000	0.000	0.000	0.000
4	U Support Single Section	Vessel Supports Origin	20.000	0.000	1.000	0.000	0.000	0.000
5	U Support Single Section	Stinger Supports Origin	12.0671	0.000	3.01706	0.000	4.58366	0.000
6	U Support Single Section	Stinger Supports Origin	24.057	0.000	1.57132	0.000	9.16732	0.000
7	U Support Single Section	Stinger Supports Origin	35.8931	0.000	-0.82797	0.000	13.751	0.000
8	U Support Single Section	Stinger Supports Origin	47.4996	0.000	-4.16545	0.000	18.3346	0.000
9	U Support Single Section	Stinger Supports Origin	58.8022	0.000	-8.41979	0.000	22.9183	0.000
10	U Support Single Section	Stinger Supports Origin	69.7287	0.000	-13.5638	0.000	27.502	0.000
11	Flat Support 2m	Stinger Supports Origin	80.2091	0.000	-19.5645	0.000	32.0856	0.000
12	Flat Support 1m	Stinger Supports Origin	80.4747	-1.000	-19.1409	90.000	90.000	122.086
13	Flat Support 1m	Stinger Supports Origin	80.4747	1.000	-19.1409	270.000	90.000	57.9144

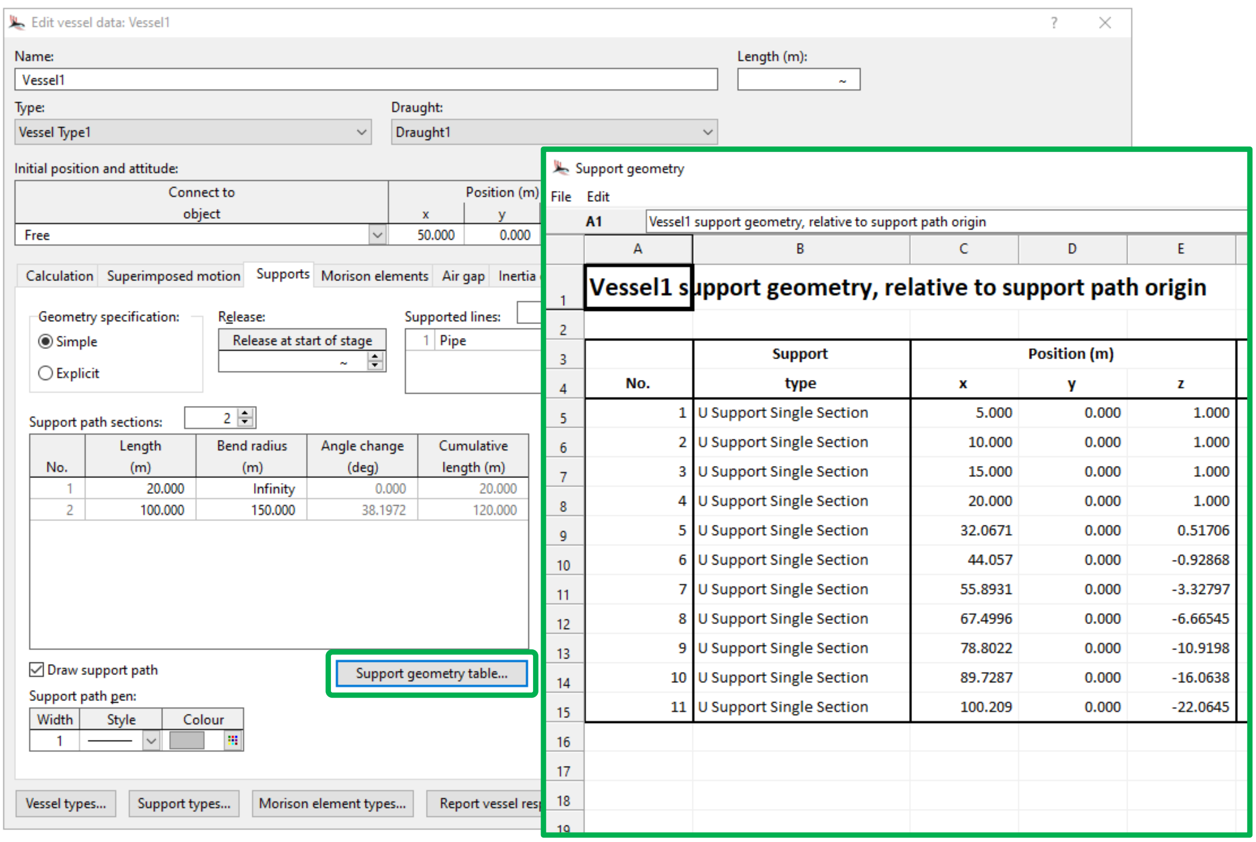
It is likely that in some situations you will want to get clearance or force results for each *individual* cylinder in a support. For example, you might need to know whether the supported line is in contact with the bottom cylinder or one of the side cylinders in a U-shaped support. To enable this, the final roller assembly on this stinger has been modelled as 3 separate flat supports (shown as numbers 11, 12 & 13 on the image below). *Flat Support 2m* is used for the bottom and *Flat Support 1m* for each side of the U configuration.

Note that the direction of the support local y-axes are important here. The y-axis defines which side of the cylinder is the contact side. These flat supports are oriented so that their y-axes point inwards (i.e. into the cylinder box). This helps OrcaFlex to guide the line towards the desired static solution on top of the roller assembly. Note, in the model view, you can see the local axes of each support cylinder by making the local axes visible (*Ctrl + Y*).

To allow direct comparison between both support geometry methods, the coordinates and orientations for this model have been established using exported values from the *simple geometry* model. This is done using the *support geometry table* option (only visible when *simple geometry specification* is selected) on the vessel data *supports* tab.

These values can be either be selected and inserted into the *explicit supports coordinates* data form using standard 'copy' and 'paste' commands (*Ctrl+C* and *Ctrl+V*) or can be saved into an Excel spreadsheet.

With reference to the image above, the first four supports are defined relative to the *vessel supports origin*, with all the others defined relative to *stinger supports origin*.



No.	Length (m)	Bend radius (m)	Angle change (deg)	Cumulative length (m)
1	20.000	Infinity	0.000	20.000
2	100.000	150.000	38.1972	120.000

Vessel1 support geometry, relative to support path origin

No.	Support type	Position (m) x	y	z
1	U Support Single Section	5.000	0.000	1.000
2	U Support Single Section	10.000	0.000	1.000
3	U Support Single Section	15.000	0.000	1.000
4	U Support Single Section	20.000	0.000	1.000
5	U Support Single Section	32.0671	0.000	0.51706
6	U Support Single Section	44.057	0.000	-0.92868
7	U Support Single Section	55.8931	0.000	-3.32797
8	U Support Single Section	67.4996	0.000	-6.66545
9	U Support Single Section	78.8022	0.000	-10.9198
10	U Support Single Section	89.7287	0.000	-16.0638
11	U Support Single Section	100.209	0.000	-22.0645

Results

Opening the *E01 explicit geometry stinger* simulation file will open it with the same workspace as the simple geometry example; the results are identical.

Open the workspace file *E01 Explicit geometry stinger_detailed support results.wrk*. This presents graphs of the clearance results between the supported line and the three sides of the U support located at the end of the stinger.

The top left-hand graph shows that the pipeline remains in contact with the base support throughout. The top right-hand (port roller) and bottom left hand (starboard roller) graphs show that the line stays predominantly close to the port support, and only occasionally moves towards the starboard support.

If the supported line was to lift off the bottom support, then the *support lift out* results would tell you how far up the side supports any contact with them would be.