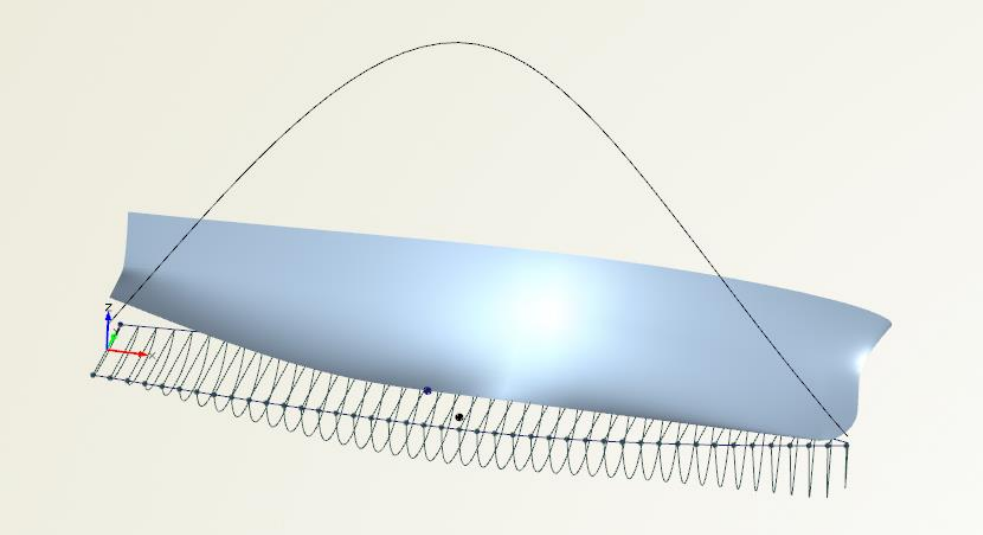
Hydrostatic Calculation

This introductory tutorial guides you through setting up a hydrostatic calculation. The numerical computation is based on planar sections that are created from the ship hull surface.



CAESES Project

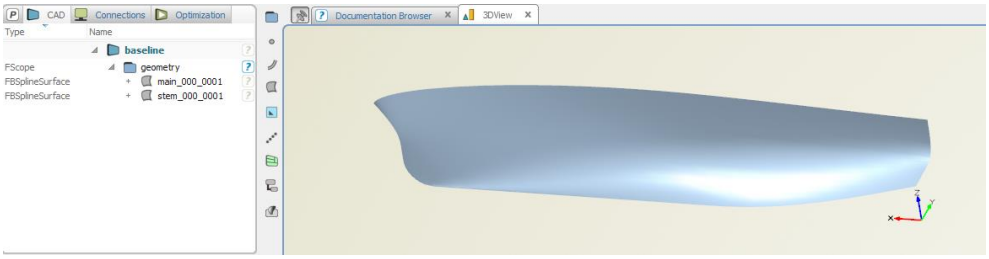
The resulting setup can also be found in the section *samples > tutorials* of the documentation browser.

✓ Note that, alternatively, there is the command *getDisplacement()* of the *surface group* (*CAD > surfaces > surface group*) that also quickly returns hydrostatic key values for a given set of surfaces. The calculation is based on the surface u- and v-resolutions i.e. rendering visualization. The higher the resolutions are set, the more accurate the calculation will be. There is also a feature available that encapsulates the surface group command and gives you the resulting values. It is also contained in the final tutorial project for demonstration purposes. A restriction of this command is that it does not provide the sectional area curve which is required for e.g. a *Lackenby shift*.

1 Initial Geometry

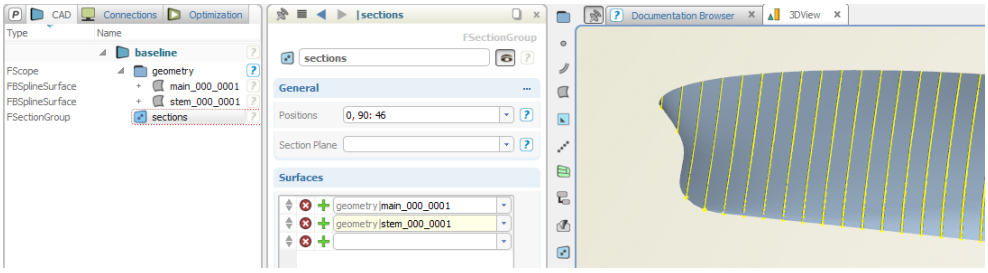
The hull needs to be imported first. Instead of using an existing IGES geometry, the subsequent steps do also apply for any parametric model that has been created in CAESES.

- Choose *file > import > IGES*.
- In the installation path of CAESES, open the geometry file *tutorials > 06_hull_design > basicosv.igs.*
- Rename the IGES scope to “geometry” in order to start with a structured object tree.



2 Section Group

The hydrostatic calculation will be based on discretized planar sections that need to be generated in longitudinal (x-) direction.



- Select the two surfaces either in the object tree or in the 3D view.
- While the surfaces are selected, choose *CAD > offsets > section group*.
- Positions for the sections have to be clarified with a *series* such as "0,90:46".

The last step creates 46 sections from x-position “0” to x-position “90”.

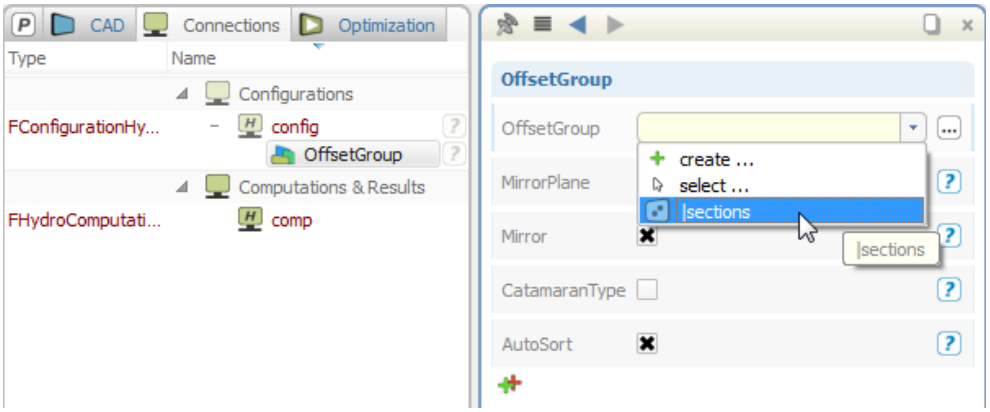
✓ For more information about *series* i.e. how to define series in various ways, please see the type documentation of *FSeries*. Common definitions are “a,b:N” or “a,a+delta..b”. For instance, the expression “0,2..90” generates a section at x-position “0”, at position “2” and continuously with this distance up to “90”. So this also leads to 46 sections between 0 and 46 meter. Combinations and sequences are possible as well. Try sections with "0,0.5..10,70:13,71..90" for training purposes.

✓ The attribute *section plane* of the section group is required for sections in different normal directions. The default section plane is the yz-plane. Planes can be created via *CAD > offsets > plane*.

3 Creation of Hydrostatic Setup

CAESES provides a configuration and a computation for hydrostatic calculation. Both are connected to each other when a setup is created:

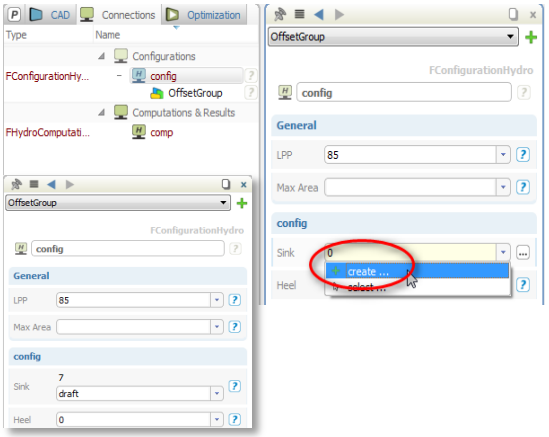
- Choose *connections > hydrostatics* which creates a configuration and a computation in the tab *connections*.
- Expand the object "config" in the tree and select *offsetgroup*.



- Choose "sections" from the pull-down menu of the *offsetgroup* entry.

✓ Hydrostatic configurations can also handle offset groups of type *FOffsetGroup*. This type is given in the menu *CAD > offset > offset group*. There is also a simple ASCII import for offset groups (*file > import > offsets*) which is particularly of interest for *SHIPFLOW* users.

- Select "config" in the tree and set *LPP* to "85".
- Create a parameter for *sink* by selecting *"create ..." > parameters > parameter* from the *sink* pull-down option
- Name it "draft" and set it to "7".

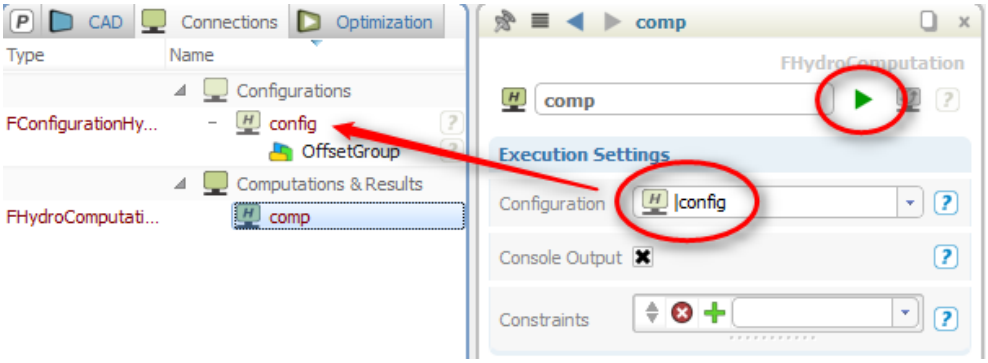


4 Hydrostatic Computation

The setup is ready for hydrostatic calculation:

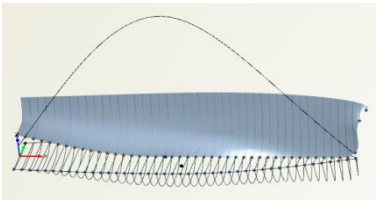
- Check that the configuration is chosen in the computation.
- Select “comp” and start the calculation by clicking at the green *run* button in the upper right corner of “comp” in the object editor.

✓ Alternatively, choose *run* from the context menu of the computation “comp” or click at the green run button in the main toolbar while “comp” is selected.

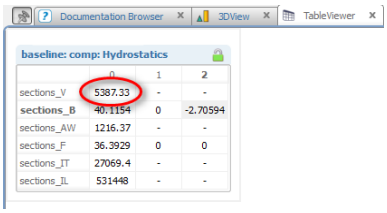


5 Results

Results are provided in the *3D view* as well as in a table. Values from the table can be extracted.



The *table viewer* widget provides result values from the analysis. If you want to further work with one of these values, just put them into parameters:



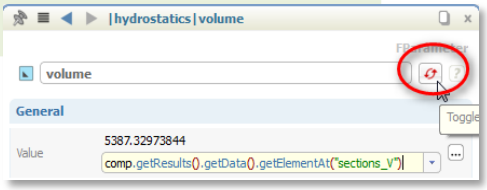
	0	1	2
sections_V	5387.33	-	-
sections_B	40.1154	0	-2.70594
sections_AW	1216.37	-	-
sections_F	36.3929	0	0
sections_IT	27069.4	-	-
sections_IL	531448	-	-

- Double-click at the cells with the values of interest, e.g. at column "0" of *sections_V* and *sections_IT*.

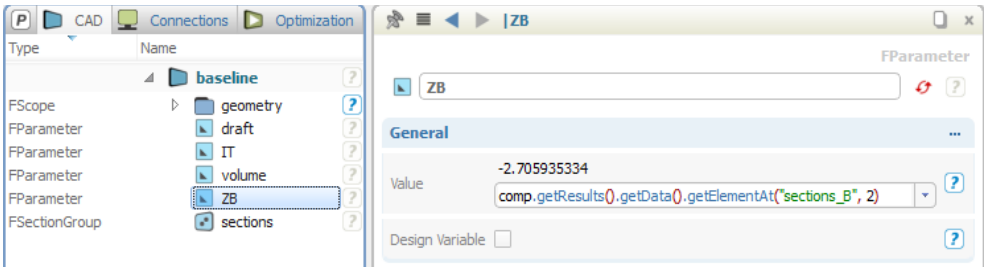
This automatically creates parameters with the values of volume and moment of inertia in the *CAD* tree.

- Rename the two parameters to "volume" and "IT", respectively.

✓ These parameters monitor the hydrostatic value. If geometry is modified and the calculation is run again, the parameters will show updated values. Parameters do not automatically update their values. They need to be updated manually e.g. by using the *run* button or they are indirectly updated via triggering the hydrostatic calculation again. If you need an automatic i.e. instant update for a parameter, then activate the corresponding button. In our case, this means that the calculation would automatically start each time the model is changed!

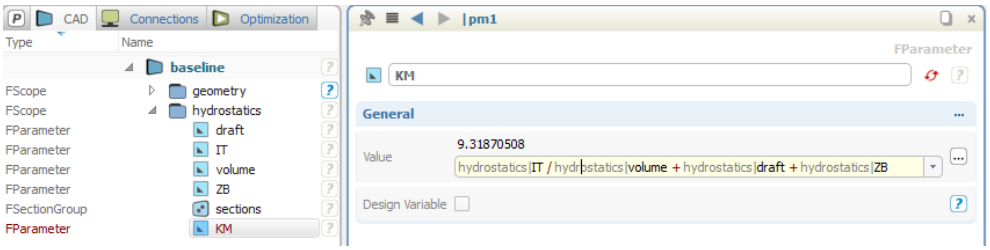


- Repeat this procedure for the z-value of the *centre of buoyancy* (*sections_B*, column "2") and name the new parameter "ZB".
  Note that this value is negative because it is related to the sink value (z=0 level).



6 Derived Parameters

Let's create a derived parameter based on the hydrostatic output. As an example, a parameter "KM" is calculated, being the distance value between keel and meta centre.



- Create a new parameter via *CAD > parameters > parameter* and name it "KM".
- Enter the following expression into the editor of "KM": IT / volume + draft + ZB

✓ Remember: Do not manually type this expression but rather use drag & drop, the editor's pull down menu or the ALT-key (see the tutorial GUI introduction). This is much quicker.

Finally, quickly tidy up the project:

- Select the parameters "KM", "draft", "IT", "volume", "ZB" as well as the section group "sections" and create a scope via *CAD > scope*.
- Set the name of the scope to "hydrostatics".

✓ If you need additional views, just create them via e.g. *view > new > viewer > table viewer*. Viewers are set and can also be deactivated at the result items of the computation: