Revision 15, 2015-04-16

SHIPFLOW DESIGN
Tutorials

BASIC

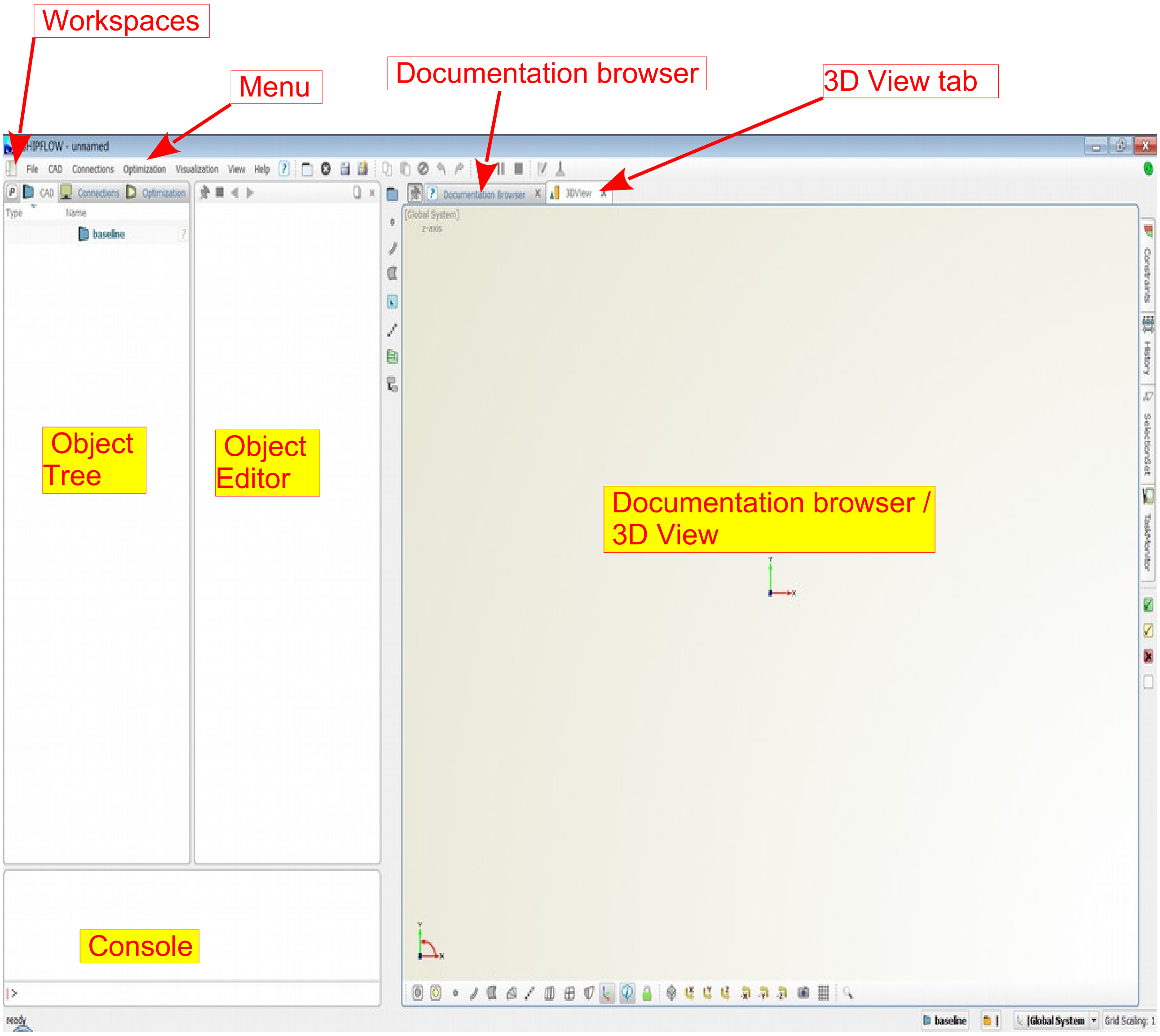
SHIPFLOW®

Table of Contents

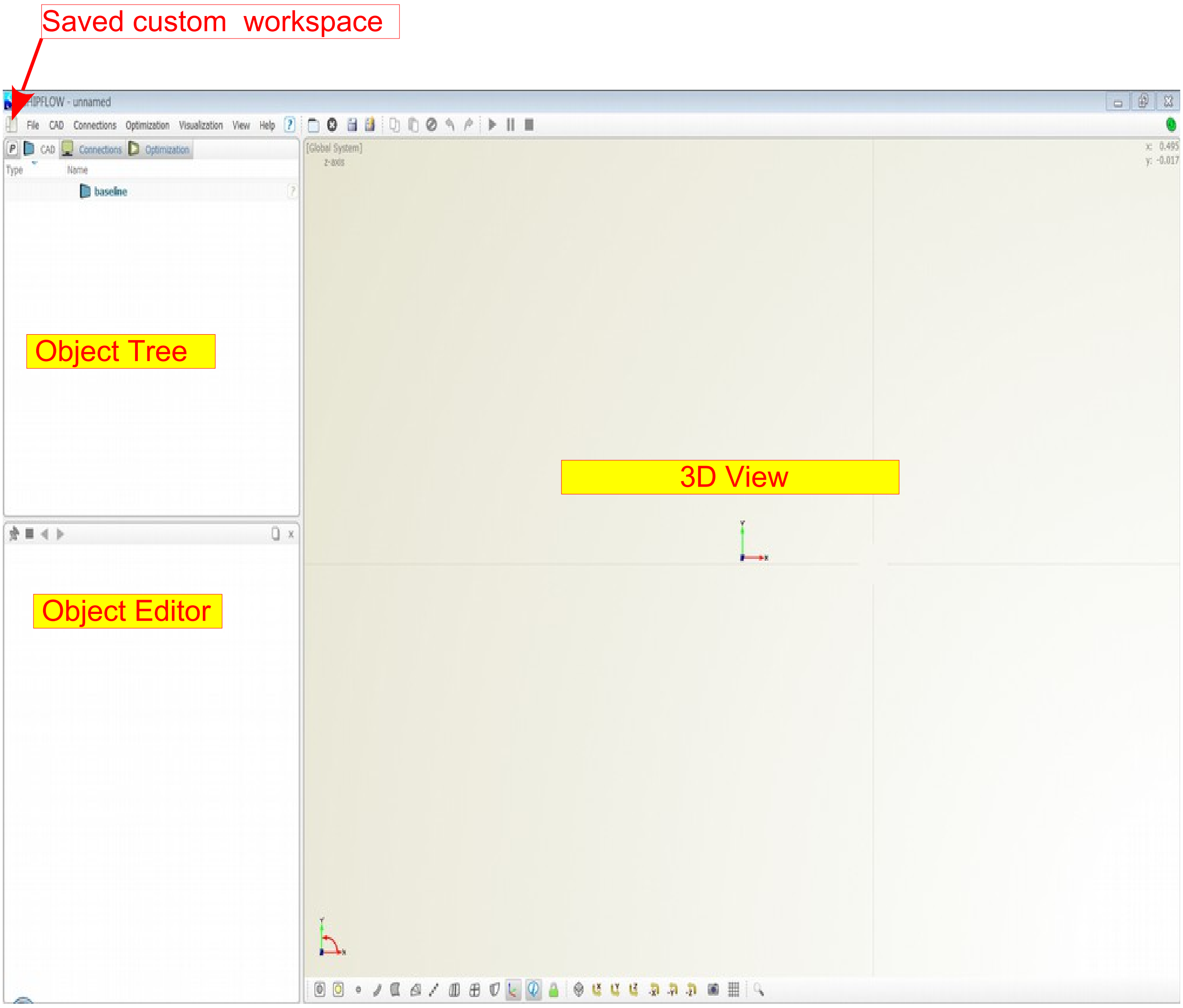
Introduction to SHIPFLOW DESIGN environment

- First start of the SHIPFLOW Design application (Windows OS)

START > All Programs > FLOWTECH > SHIPFLOW x.x.xx > Shipflow Design x.x.xx



- Customized workspace



Tutorial 1 part 1 – Setting up new case directly from IGES file

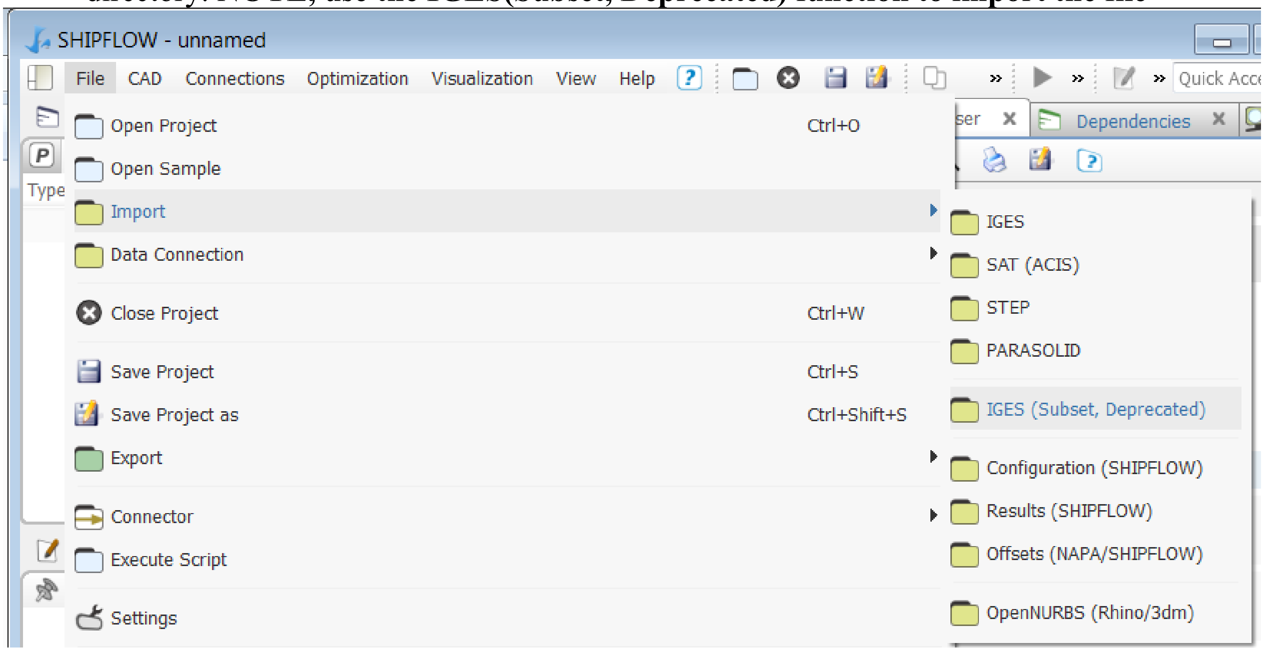
This tutorial takes the user through necessary steps to compute wave pattern and resistance for a container ship. Part 1 of the tutorial shows how to set up a new SHIPFLOW project starting from an existing IGES file. The XPAN module is used to obtain the results and the default values are used where possible.

Ship data
- KCS (Korean Container Ship), IGES file: **kcs_g2010.igs**
- Lpp 230m
- Draft 10.8m
- Design speed Fn = 0.25, Rn=1e6

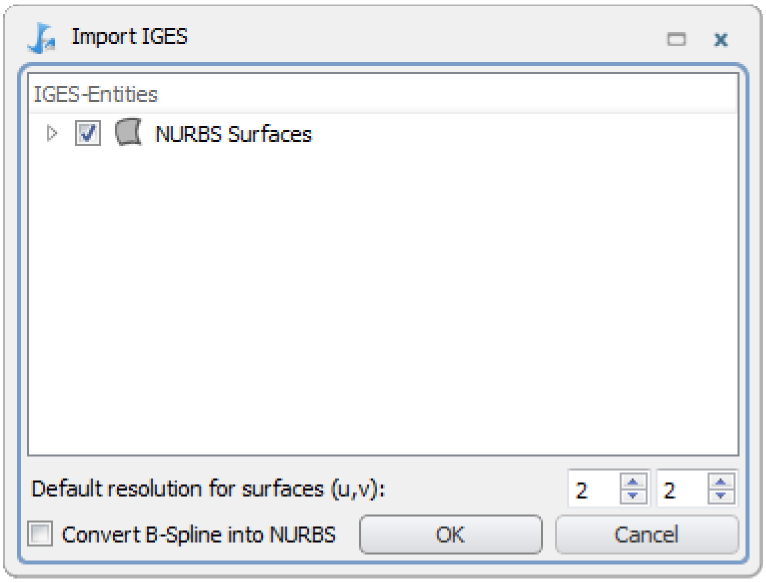
Computations:
- Free surface with free sinkage and trim

---

1. Open New Project
2. Import kcs_g2010.igs IGES file from C:\FLOWTECH\SHIPFLOW6.x.x-x\examples directory. **NOTE, use the IGES(Subset, Deprecated) function to import the file**

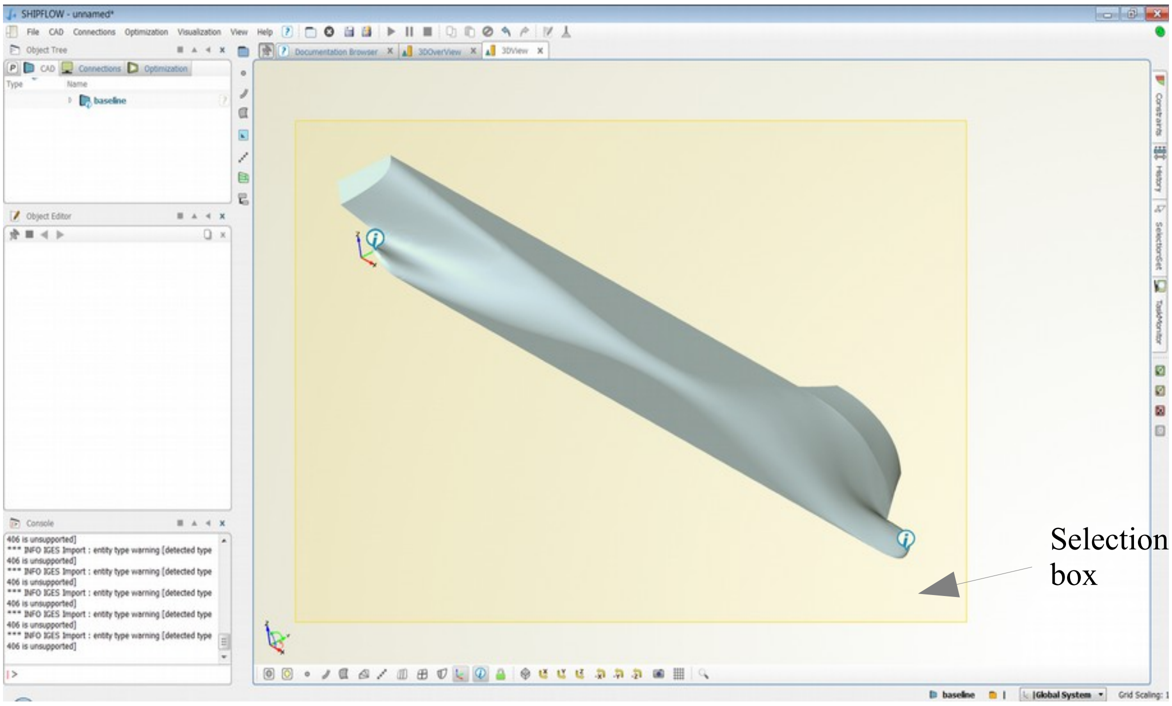


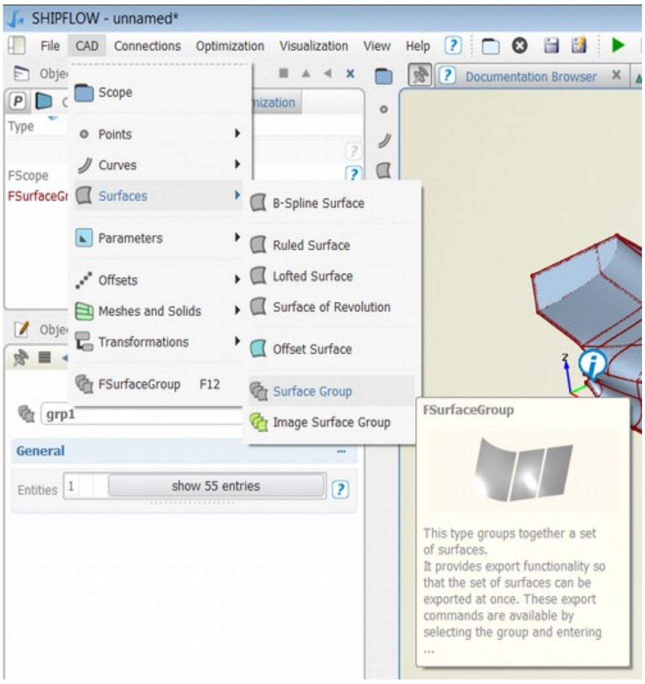
3. A file selection window will then appear in which the user can navigate to and select the IGES file. Thereafter a window for selecting the IGES entities pops up. In this case accept all entities that are pre-selected by pushing the OK button. The warnings about unsupported entities that appears in the Console Widget can be ignored in this example.



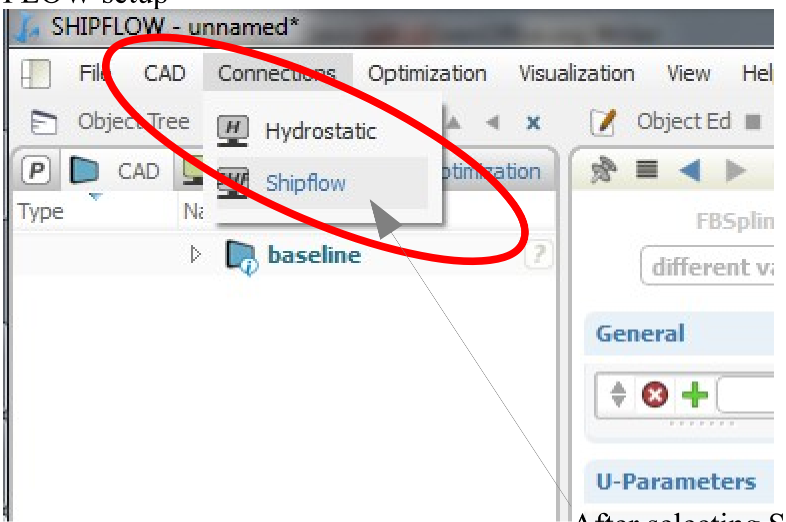
4. Change the main view to 3Dview by clicking on the tab 3DView. Select the Zoom extent button in the lower left corner of the 3DView window, or use the short cut F10, to adjust the view of the hull surface.
5. Thereafter try the short commands Control+X, Control+Y and Control+Z or click on the corresponding buttons in the lower part of the 3Dview window to change the view direction. Try the isometric view options as well.
6. Save the project in your working directory from the File menu or using the short cut Ctrl+s

7. **Select imported surfaces** to be used in computations and **create Surface Group**

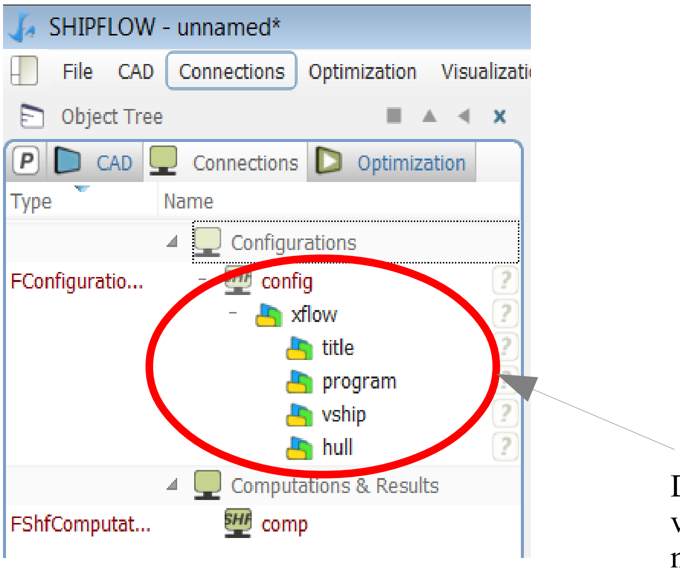




8. Create SHIPFLOW setup

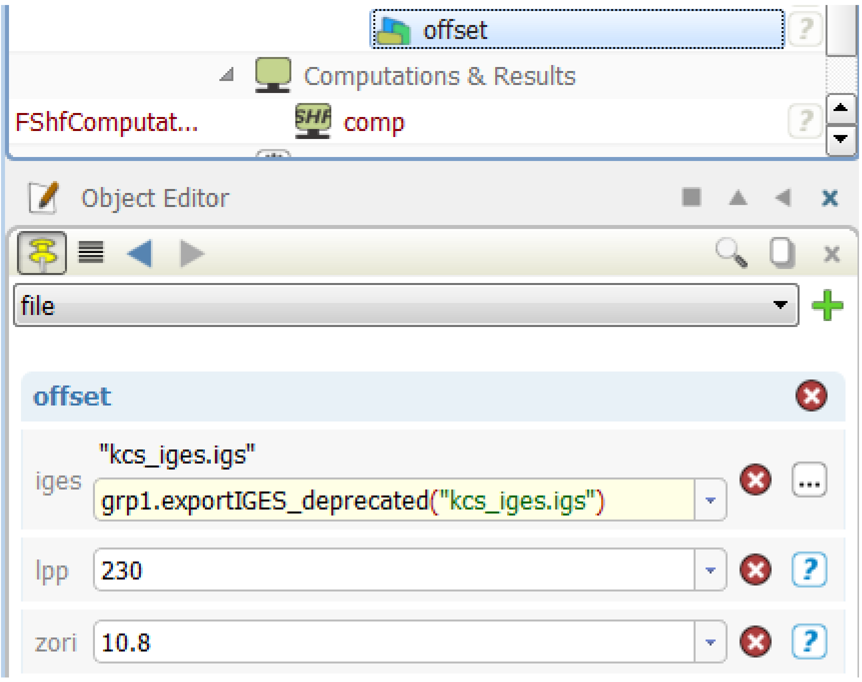


After selecting SHIPFLOW from Connections > Setups menu. Configuration and Computation will be created.



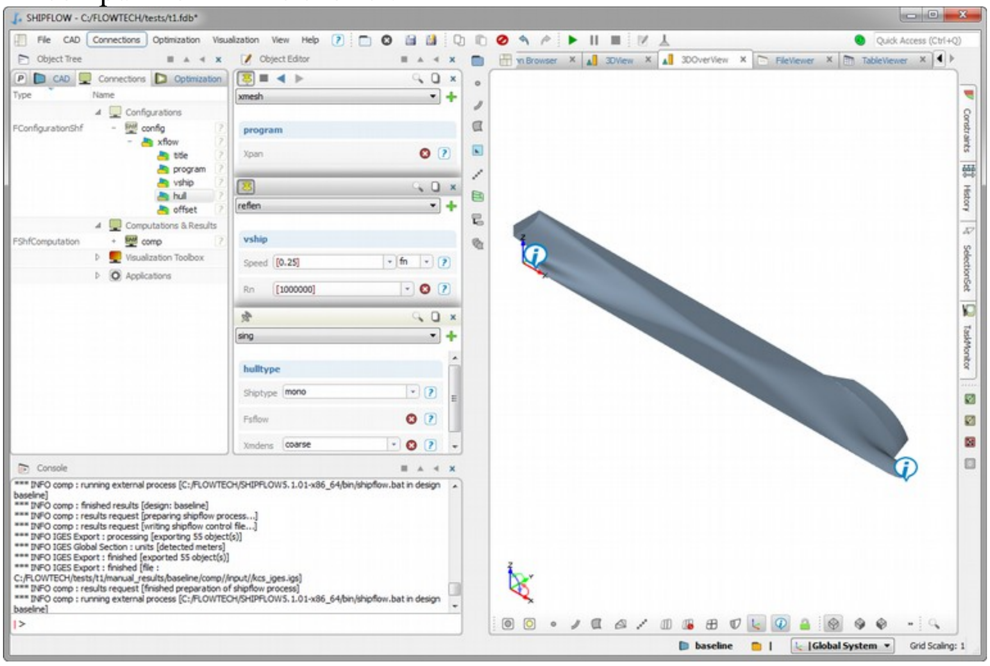
Default set of program parametres will be visible in the object tree under "config" node.

9. Configure the case
   - Add **offset** configuration to **xflow** and set it up according to given instructions:
     - Select **xflow** in the Object Tree
     - Select **offset** in the **xflow** Object Editor
     - Press the + button to add the **offset** configuration in the Object Tree
     - Similarly, select and add **iges**, **lpp** and **zori** in the offset editor
     - Choose the CAD tab in the Object Tree. Drag the surface group object **grp1** into the Iges box. Complete the command by filling in the remaining part of the command: **grp1.exportIGES_deprecated("kcs_iges.igs").** Tip: Type Cntl+Space to auto complete a command.
     - Fill in the remaining data for **Lpp** and the draft **Zori**.



   - Complete the configuration such that SHIPFLOW will run a free-surface potential flow simulation at Froude number 0.25.

- Select the **program** command and choose **xpan** in the roll down menu. Don't forget to push the add button (+).
- Select the **vship** command and set **Fn** to 0.25. Add also the **Rn** (Reynolds number) from the roll down menu and set it to 1e6.
- Finally, add the **fsflow** keyword in the **hulltype** command. This will instruct XPAN to include a free-surface. Add also **xmdens** and set it to **coarse** in order to make the computational time shorter.

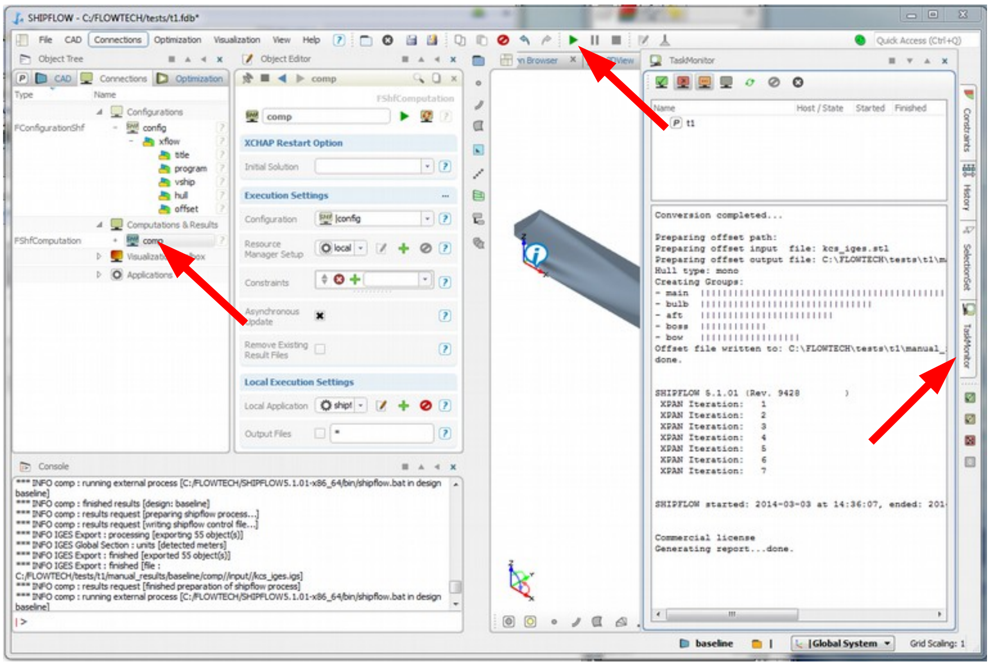


10. The configuration is now completed. Save the project and continue to the next section.

Tutorial 1 part 2 – Starting and monitoring

In order to start the calculations following the instructions below.

1. Select the Computation object **comp** in the Object Tree
2. Press the green arrow button to start the SHIPFLOW run
3. Check progress in the Task Monitor. It's found on the side bar.

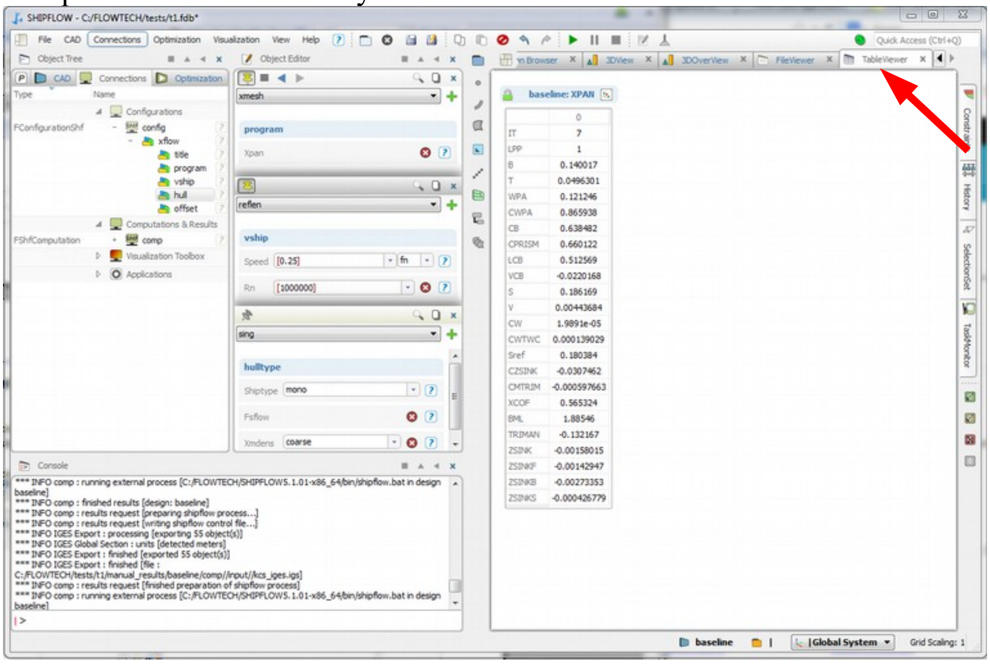


4. Save the project. This project is used as a starting point in later parts of the tutorial.

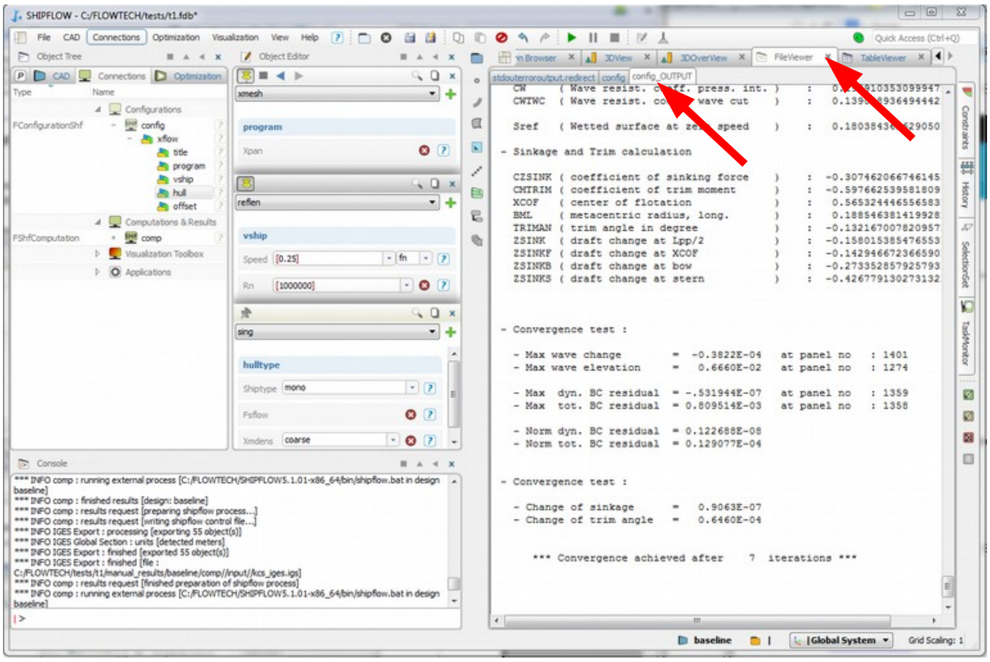
Tutorial 1 part 3 – Result post processing

1. The computations are finished when the "play" button turns green again and the Task Monitor shows the SHIPFLOW message with the start and end of computations times.

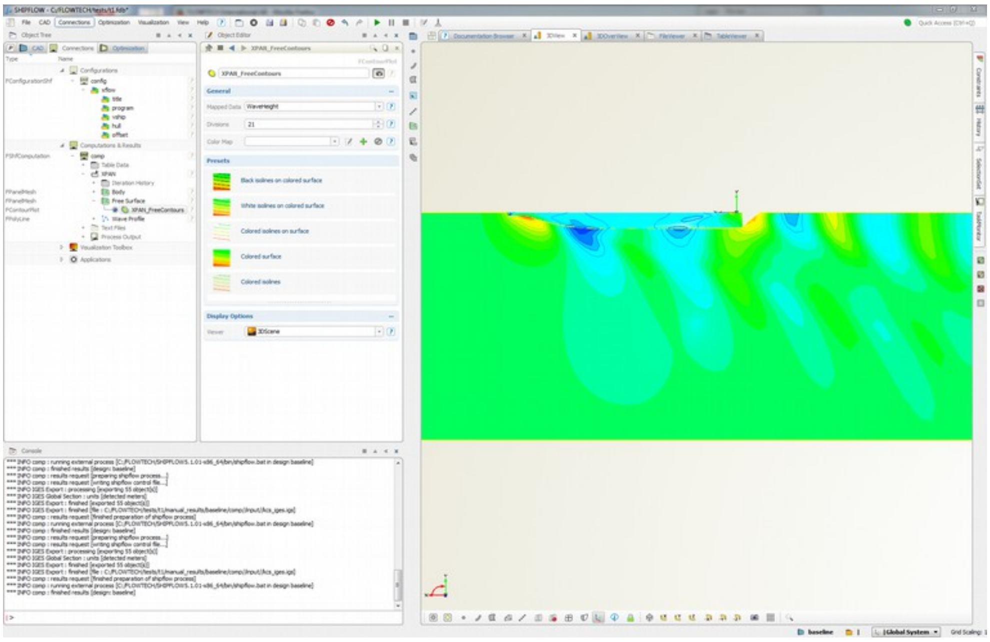
2. The computation result summary is contained in the table available in the **Table Viewer** tab.



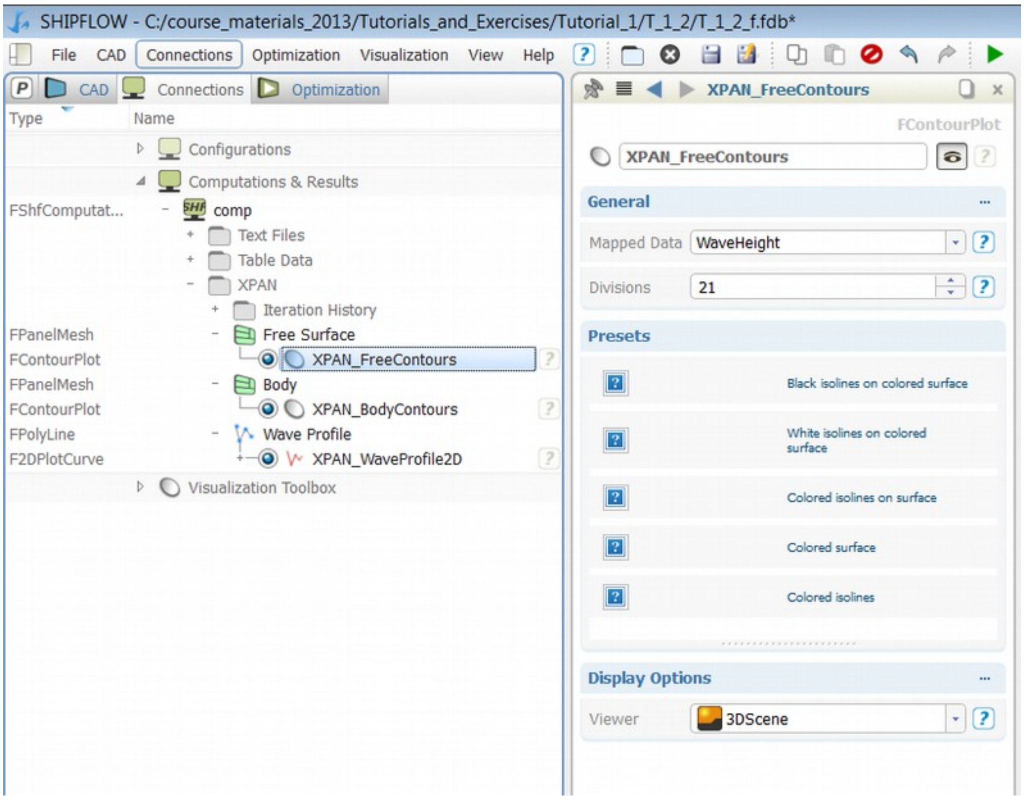
3. More informations about the computation settings and results can be found in **File Viewer** tab under case1_OUTPUT tab. Scroll down to see the complete data.



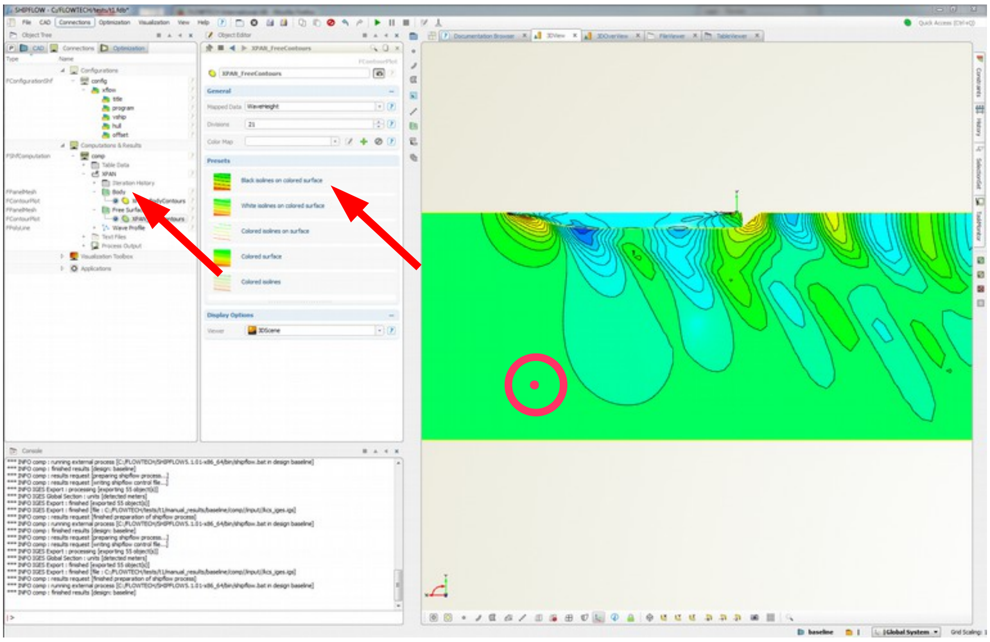
4. The wave pattern from XPAN is visualised by default in 3DView.



5. The post-processing tools are available in **Object Tree | Connections | Computations & Results | comp.**

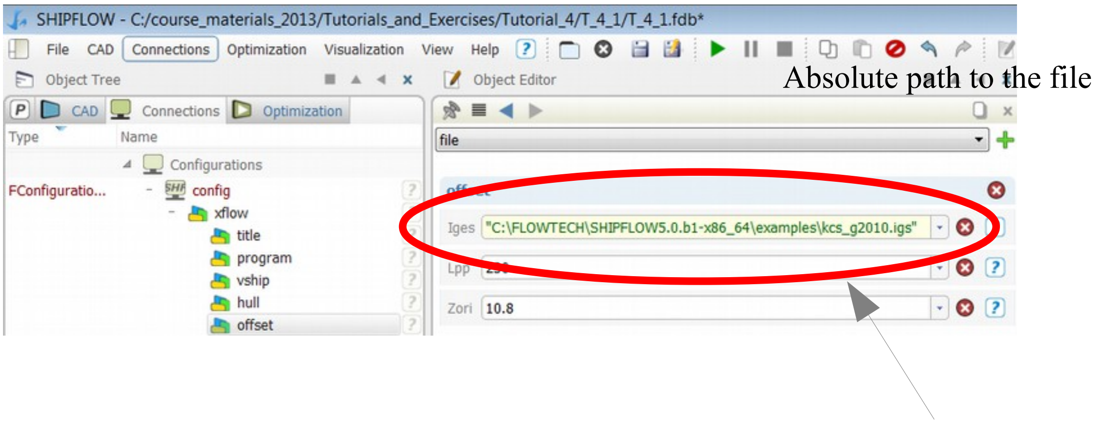


6. To change for example the wave pattern visualisation style go to **Object Tree | Connections | Computations & Results | comp | XPAN | Free Surface | XPAN** (or click on the free surface in the 3DView) and in the **Object Editor** change **Presets** to “Black isolines on coloured surface”.
   In a similar way the style can be changed on the hull panels



Tutorial 1 part 4 – Additional information for IGES import

1. In case of trimmed surfaces the IGES can be imported directly to the solver bypassing the GUI. In such case there is no need for importing hull surfaces into the user interface. Instead give a **full path** to the IGES file in the **offset | Iges** parameter.



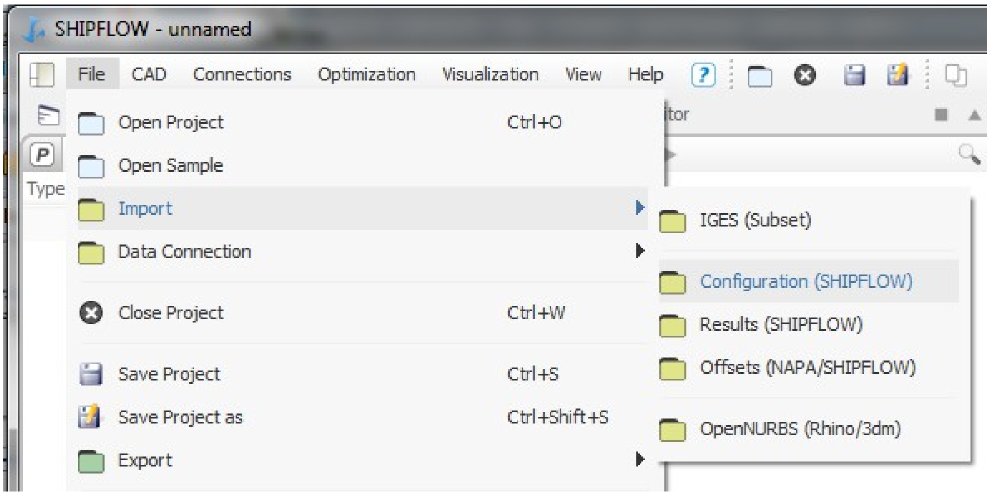
2. Create a new project and repeat the previous exercise with this technique.
3. The IGES surfaces will not be visible in the GUI, but configurations and results are displayed as usual.
4. The offset file created during the IGES import can be used for visualizing the hull. How to import the offsets is explained in another part of the tutorial.

Tutorial 1 part 5 – Import of a configuration

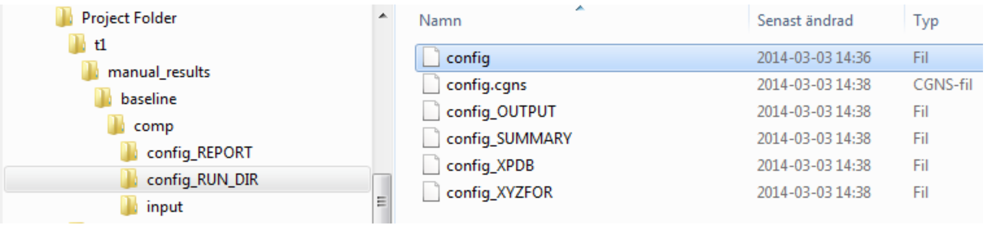
This tutorial requires that the previous parts 1 to 3 are completed.

The purpose with this tutorial is to show how to import an existing configuration. After completion the user will know how to start a project from a new IGES file, convert it to an offset file and finally import a configuration and the offset file for further computations.

1. Choose the Import Configuration (SHIPFLOW) option in the File menu



2. Search for the configuration file created in part 2. Select the configuration file in the SHIPFLOW working folder “config_RUN_DIR”. This configuration is using the automatically created offset file.

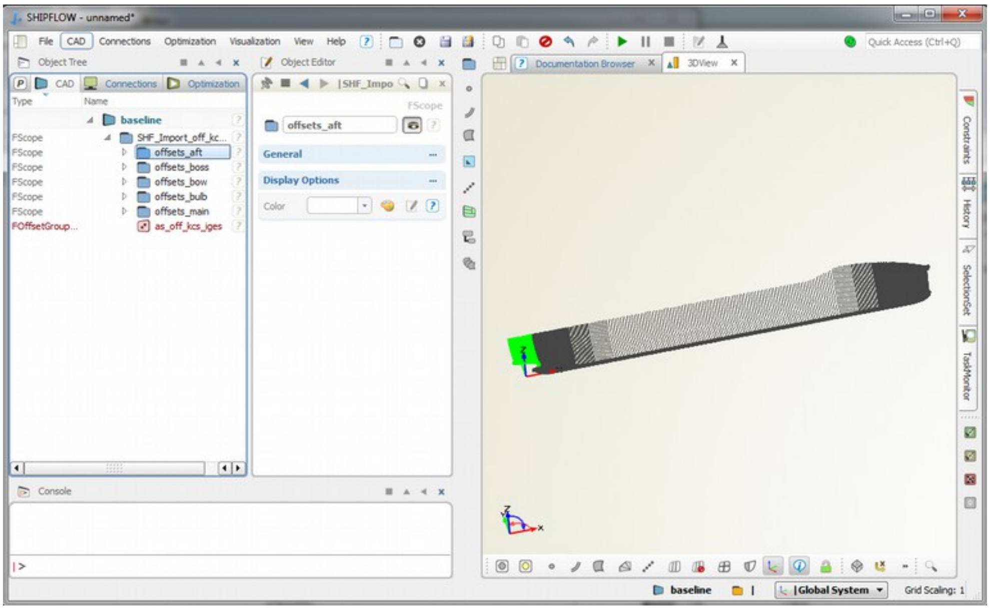


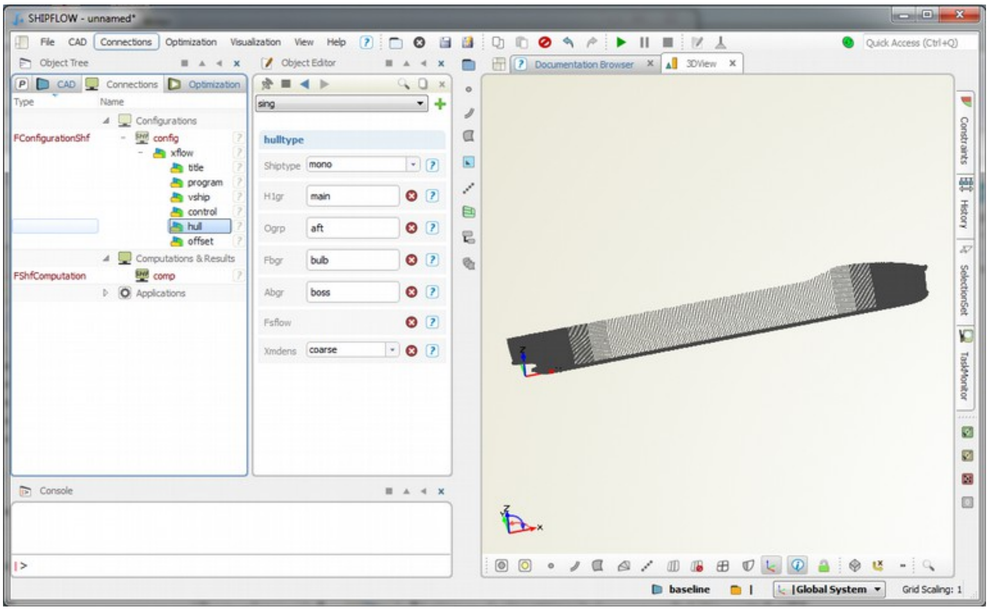
   The configuration in the “comp” folder is the original configuration with the IGES import.

3. The user may be required to select the offset file under some circumstances, but mostly the file can be found automatically.

4. SHIPFLOW have created an offset file and a configuration file for the offsets during the conversion process. The program has added additional information regarding offset groups in the “hull” command. You can see the different offset groups by selecting them in the CAD Object Tree tab. The names of these groups can be found in the entries of the “hull” command.

   The automatically generated configuration file in the RUN_DIR folder contains a command “control”. In this case it holds only information about the executable and the working directory. It's recommended to delete this command before any further usage.





5. Save the project.

Tutorial 1 part 6 – Setting up new case directly from offset file

This tutorial takes the user through necessary steps to compute wave pattern and resistance for a tanker ship. Part 1 of the tutorial shows how to set up a new SHIPFLOW project starting from an existing offset file. The XPAN module is used to obtain the results and the default values are used where possible.

Ship data

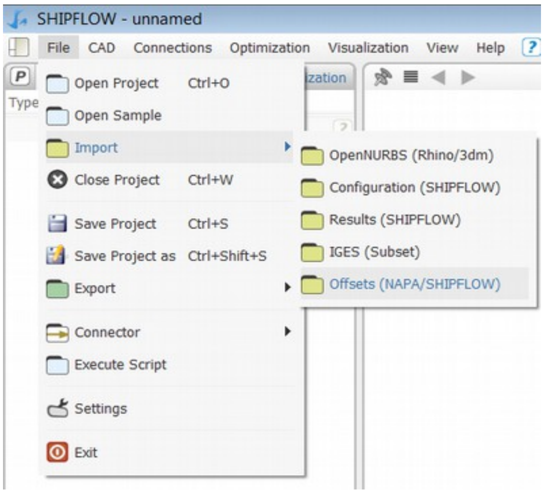
- Dyne tanker, offset file: **off_dyne3**
- Lpp 253m
- Draft 14.25m
- Design speed Fn = 0.165, Rn=1e6

Computations:

- Free surface with free sinkage and trim

---

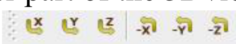
1. Import **off_dyne3** offset file from **/examples** directory found in SHIPFLOW installation folder.



2. A file selection window will then appear in which the user can navigate to and select the offset file. Thereafter a window for selecting the offset groups in the offset file pops up. In this case accept all offset groups that are pre-selected by pushing the OK button.

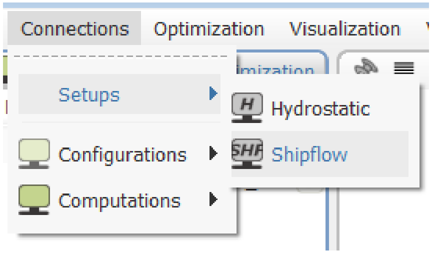
3. Change the main view to 3Dview by clicking on the tab  . Select the Zoom extent button  in the lower left corner of the 3DView window to adjust the view of the offsets.

4. Thereafter try the short commands Control+X, Control+Y and Control+Z or click on the corresponding buttons in the lower part of the 3Dview window to change the view direction:

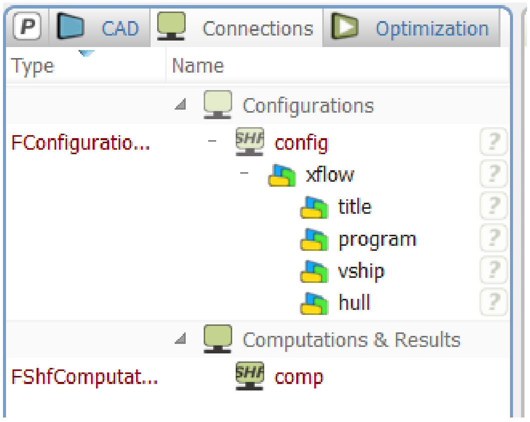


5. Save the project in your working directory

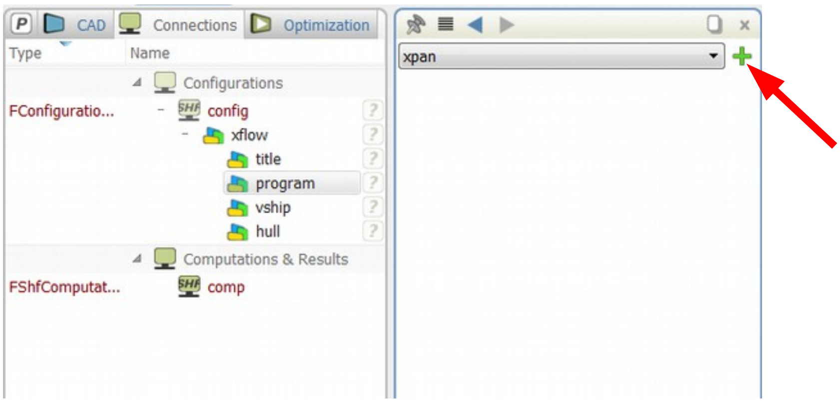
6. Create SHIPFLOW setup from menu **Connections > Shipflow.**



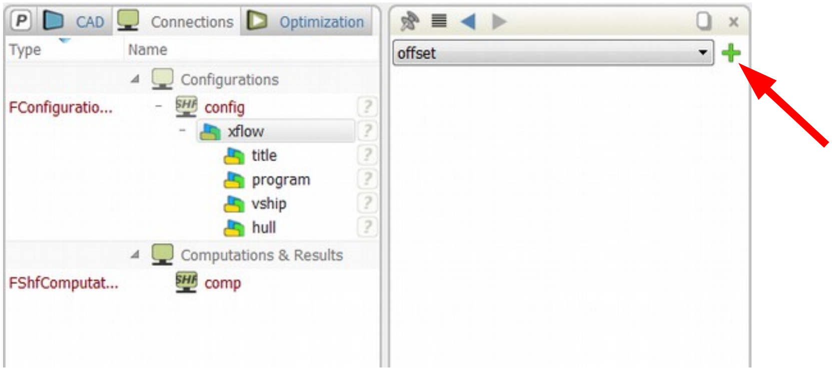
7. SHIPFLOW configuration is in the **Connections** tab of the **Object tree**



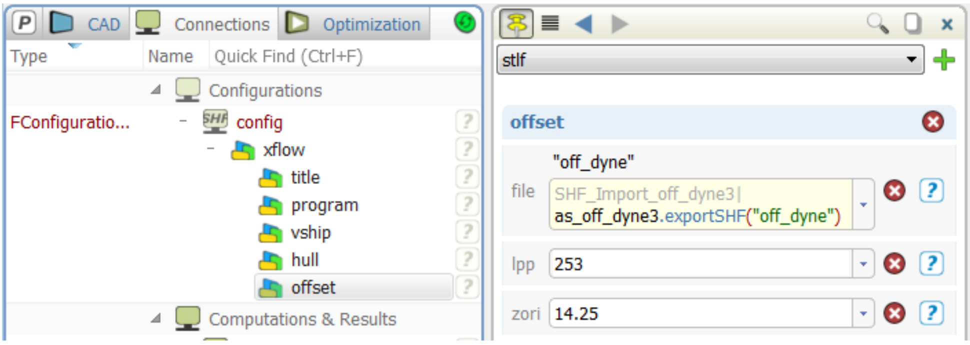
8. Add *xpan* to **Object Tree | Configurations | config | xflow | program**



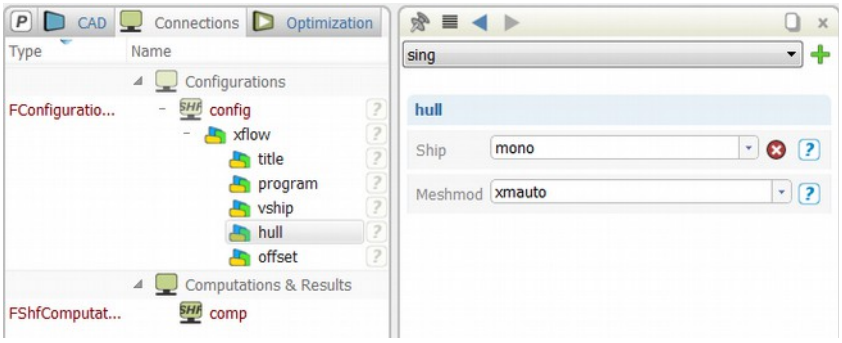
9. Add *offset* command to **Object Tree | Configurations | config | case1 | xflow**



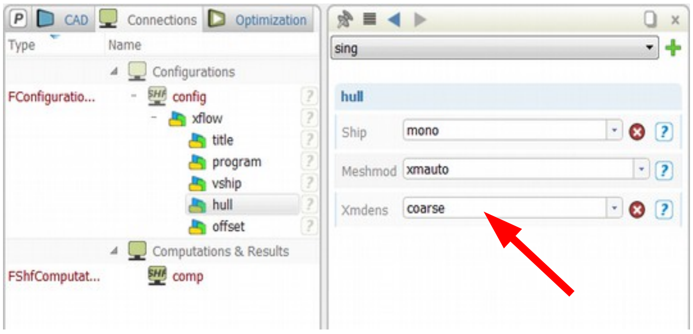
10. Edit data in **Object Tree | Configurations | config | xflow | offset**, specify Lpp and zori according to ship data
11. Associate the offset file with the configuration using following command:
    SHF_Import_off_dyne3|as_off_dyne3.exportSHF("off_dyne")



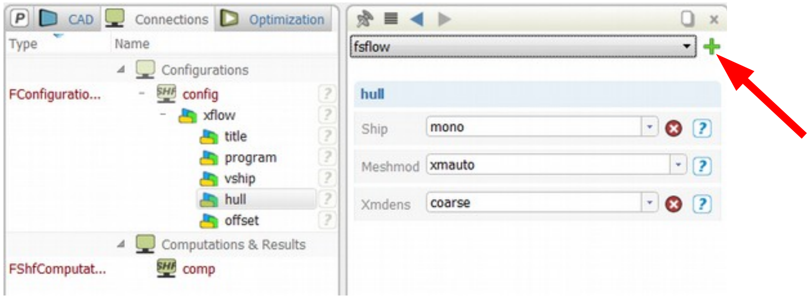
12. Add *Shiptype* to **Object Tree | Configurations | config | xflow | hull** and set it to *mono.*



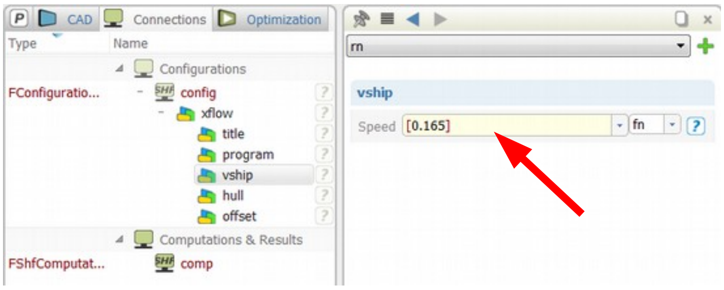
13. Add *xmdens* to **Object Tree | Configurations | config | xflow | hull** and set density parameter to *coarse.*



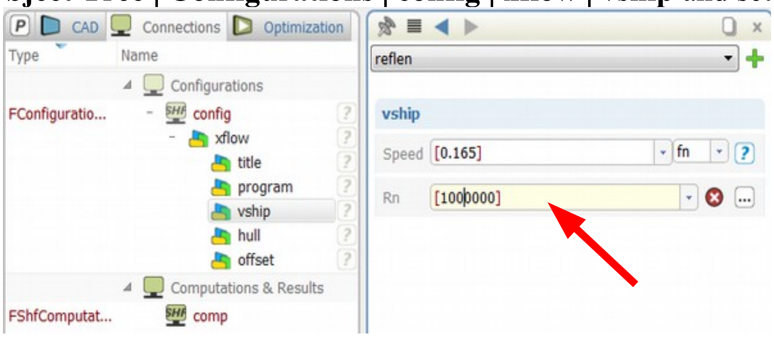
14. Add *fsflow* to **Object Tree | Configurations | config | xflow | hull**



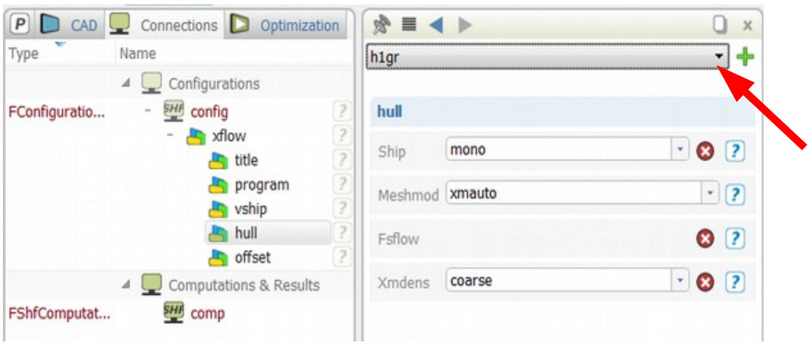
15. Set *speed to Fn = 0.165* in **Object Tree | Configurations | config | xflow | vship**

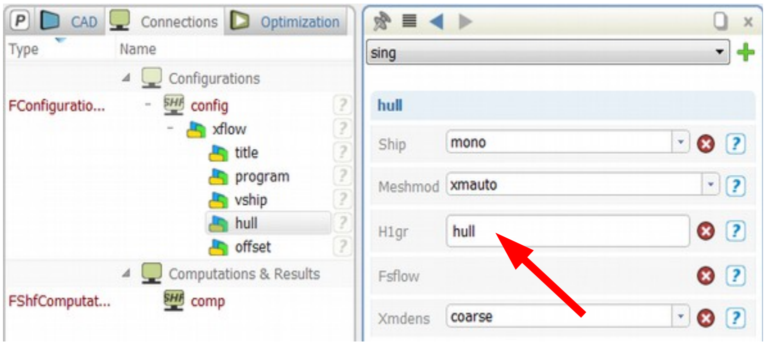


16. Add *Rn* in **Object Tree | Configurations | config | xflow | vship** and set it to 1e6.

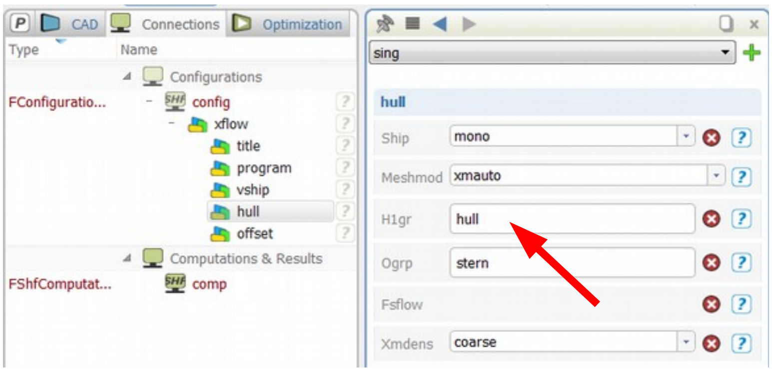


17. Add *h1gr* entry to **Object Tree | Configurations | config | xflow | hull** and set it to hull

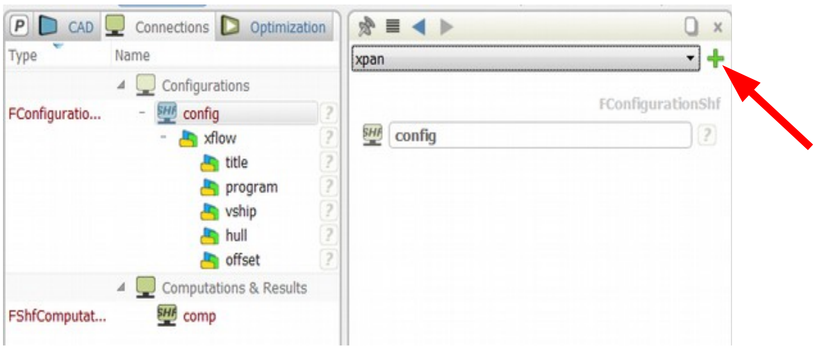




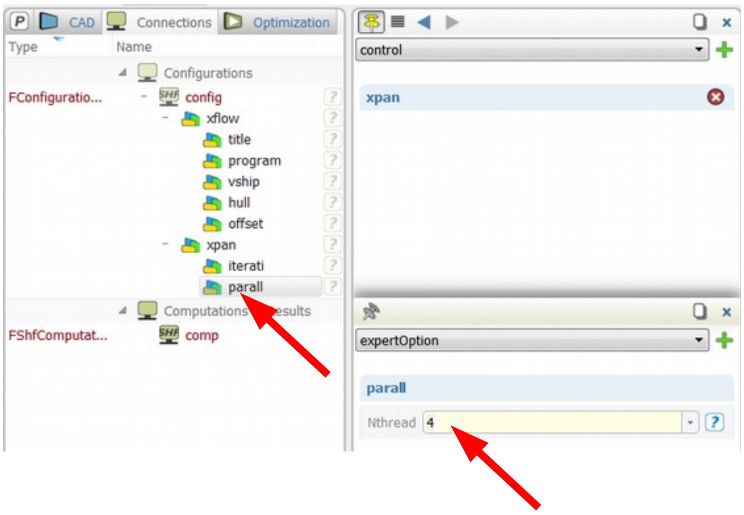
18. Similarily add *ogrp* entry and set it to **stern**



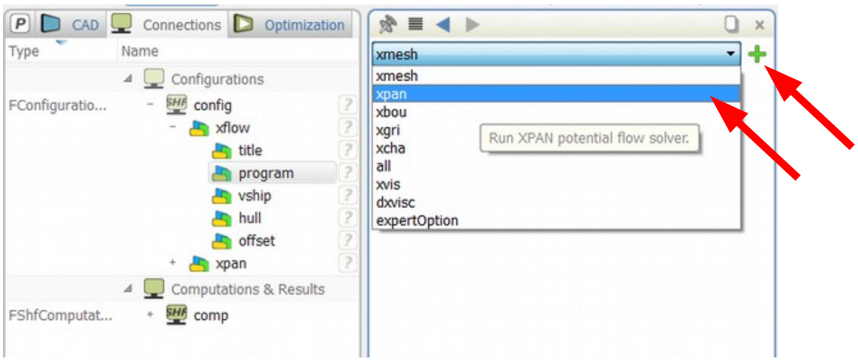
19. Add **xpan** to **Object Tree | Configurations | config**



20. Set *parall* in **Object Tree | Configurations | config | xpan** to number of available cores in your computer CPU



21. Finally select program module to be run. Add *xpan* to **Object Tree | Configurations | config | xflow | program**

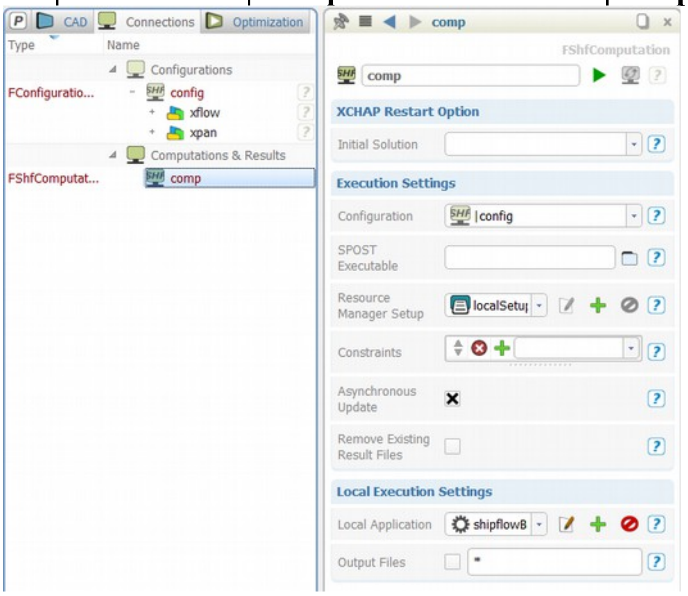


22. Save the project

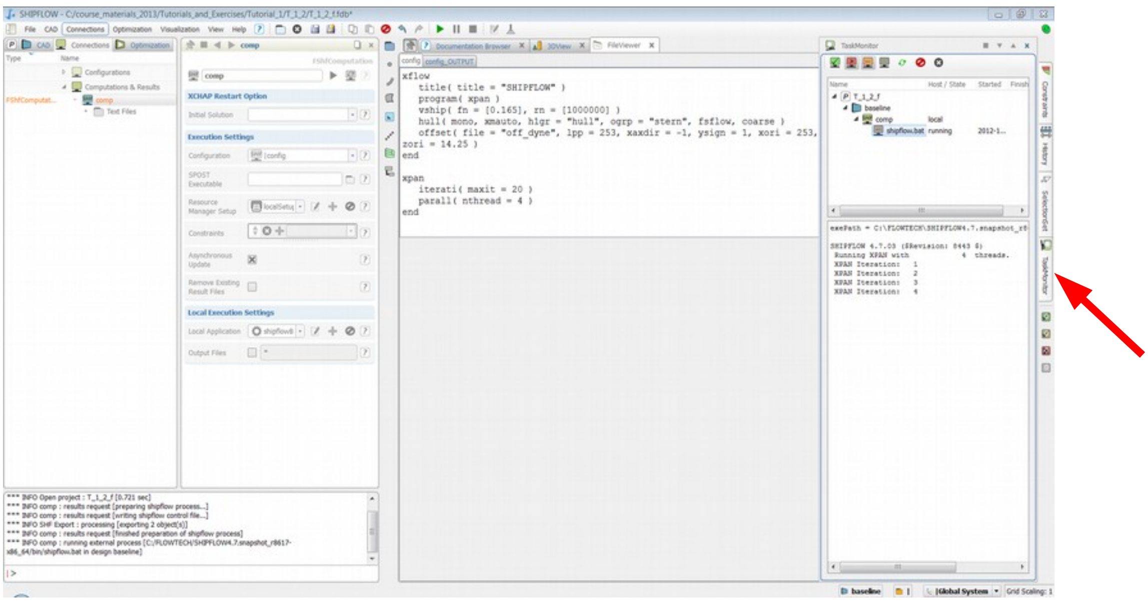
Tutorial 1 part 7 – Starting and monitoring

In order to start the calculations follow the instruction below.

1. Select **Object Tree | Connections | Computations & Results | comp** and click "play" icon.



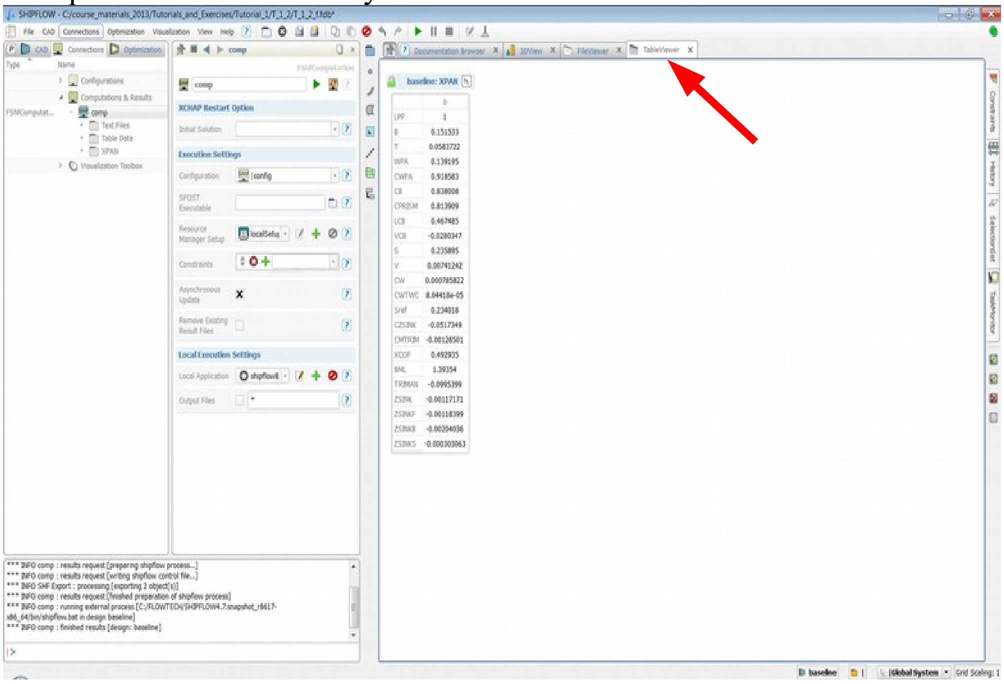
2. When the computations are running it is possible to follow their progress. For this purpose a tool called Task Monitor is used which is located on the side bar.



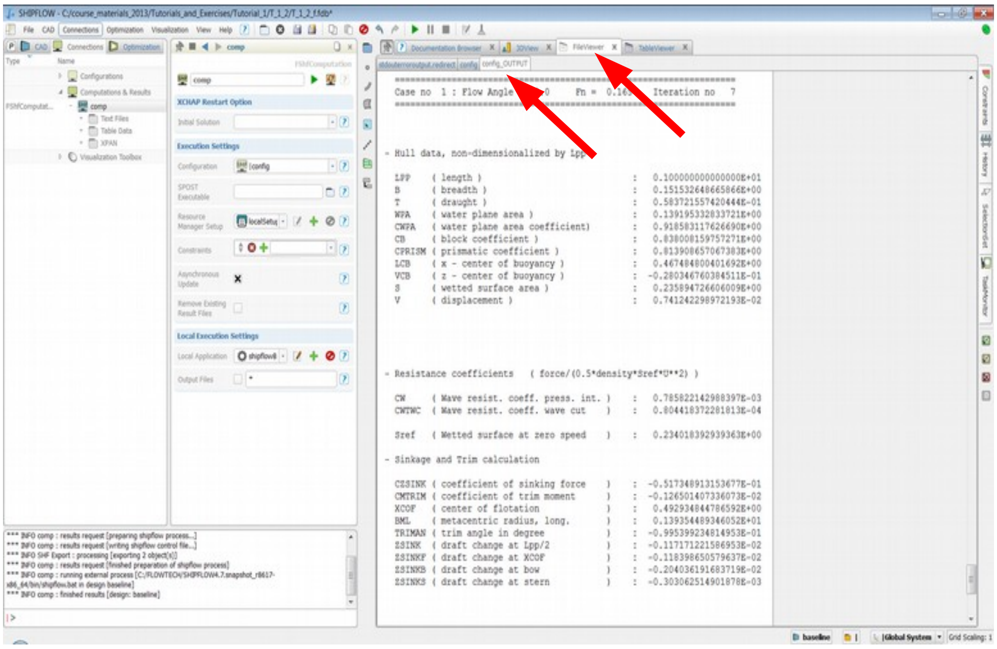
Tutorial 1 part 8 – Result post processing

1. The computations are finished when the "play" button turns green again and the Task Monitor shows the SHIPFLOW message with the start and end of computations times.

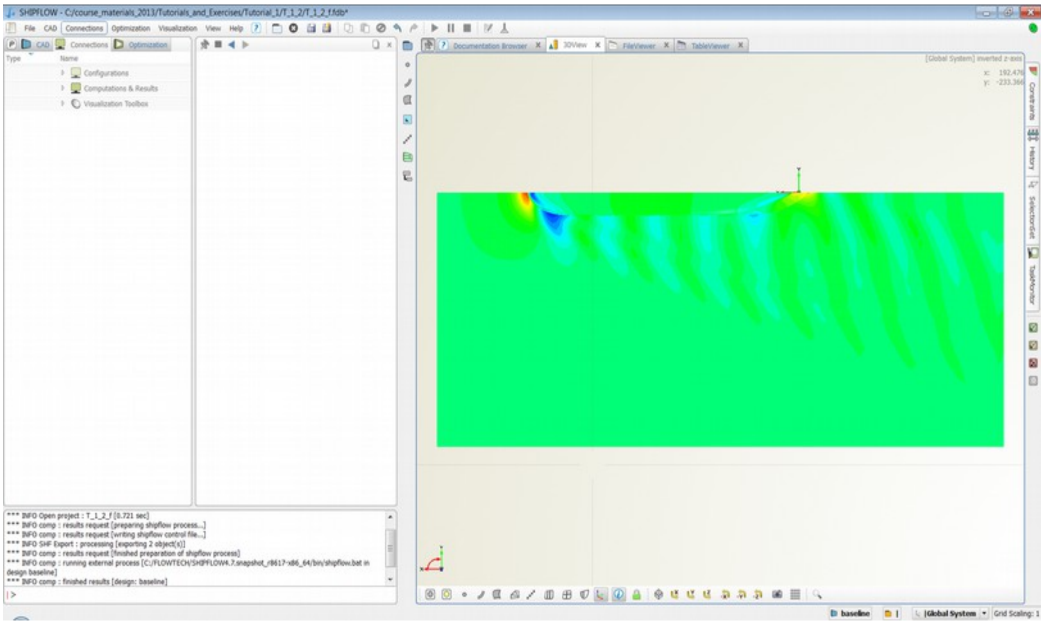
2. The computation result summary is contained in the table available in the **Table Viewer** tab.



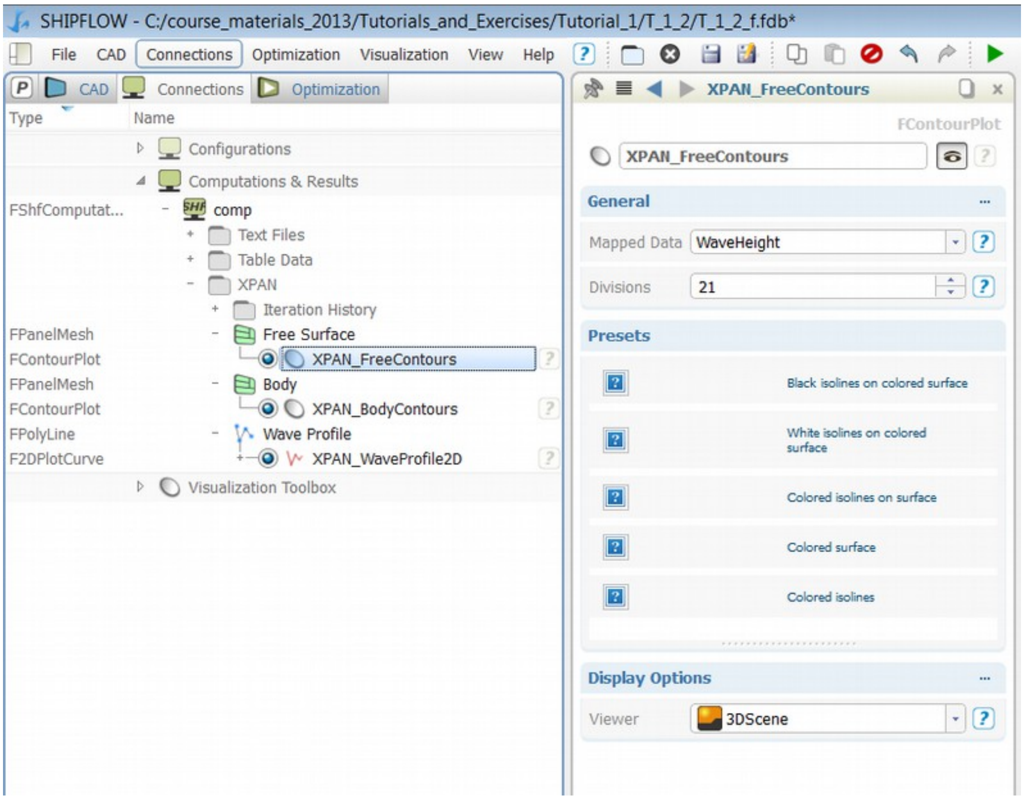
3. More informations about the computation settings and results can be found in **File Viewer** tab under case1_OUTPUT tab. Scroll down to see the complete data.



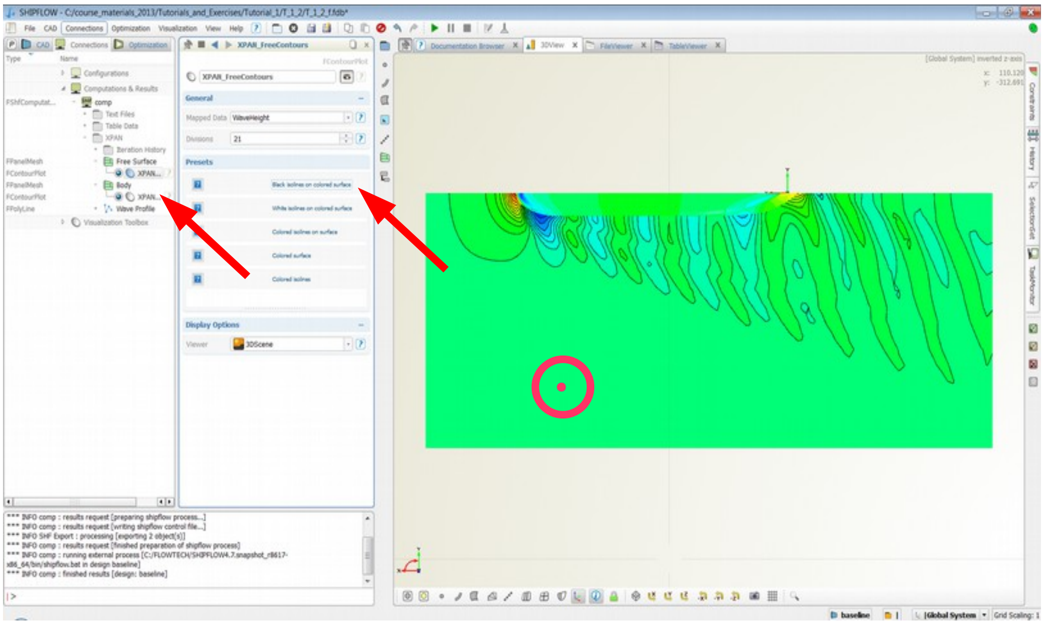
4. The wave pattern from XPAN is visualised by default in 3DView.



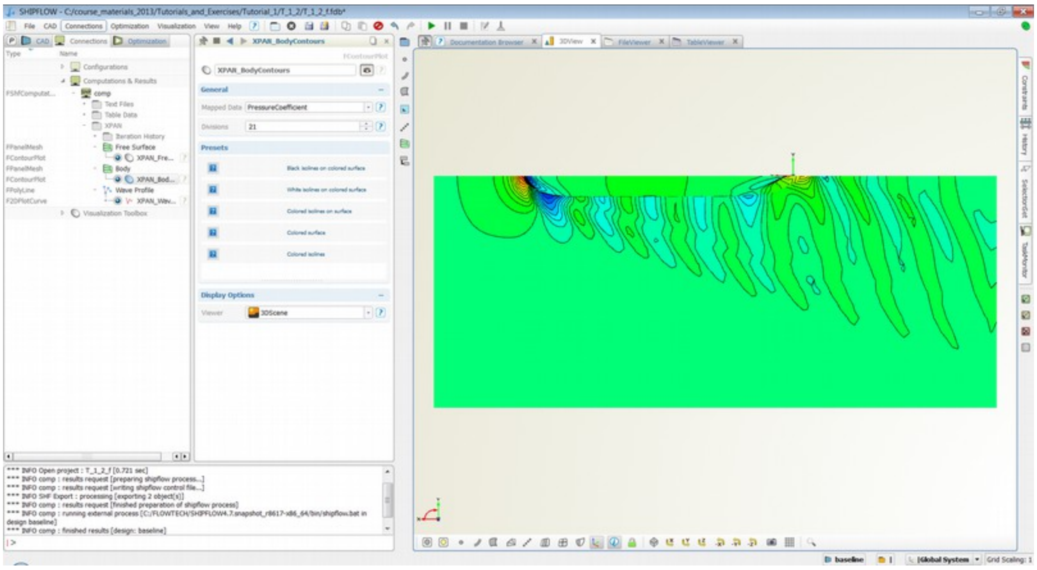
5. The post-processing tools are available in **Object Tree | Connections | Computations & Results | comp.**



6. To change for example the wave pattern visualisation style go to **Object Tree | Connections | Computations & Results | comp | XPAN | Free Surface | XPAN** (or click on the free surface in the 3DView) and in the **Object Editor** change **Presets** to “Black isolines on coloured surface”.



7. In a similar way the style can be changed on the hull panels



Tutorial 2 part 1 – Creating Design Variants

The purpose with this part of the tutorial is to repeat the case setup with and show how to create design variants as well as speed variation.

Ship data

- Athena ship
- Lpp 1
- Draft set by geometry
- Design speed Fn = 0.632, Rn=1e7

Computations:

- Free surface with free sinkage and trim
- Variants: with and without dry transom option
- Investigations: speed variation

Before the variants are created prepare the base configurations in the way similar to the previous tutorial or go through the steps summarised below.

- Import file off_ath offset file from /examples directory found in SHIPFLOW installation folder and save the project e.g. Tutorial_2_1

- Create SHIPFLOW setup from menu **Connections > Setups > Shipflow.**
- Add *xpan* to **Object Tree | Configurations | config | xflow | program**
- Add *offset* command to **Object Tree | Configurations | config | xflow**
- Edit data in **Object Tree | Configurations | config | xflow | offset**, specify:
  - Lpp=1

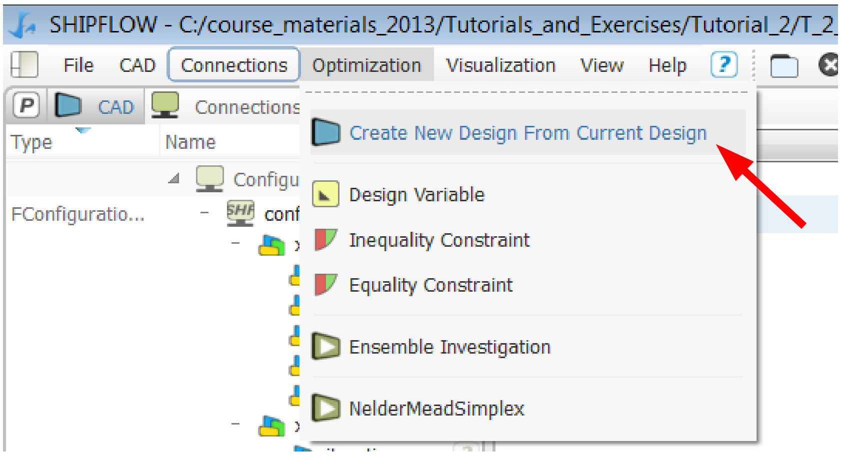
- Associate the offset file with the configuration using following command:
  - SHF_Import_off_ath|as_off_ath.exportSHF("off_ath")

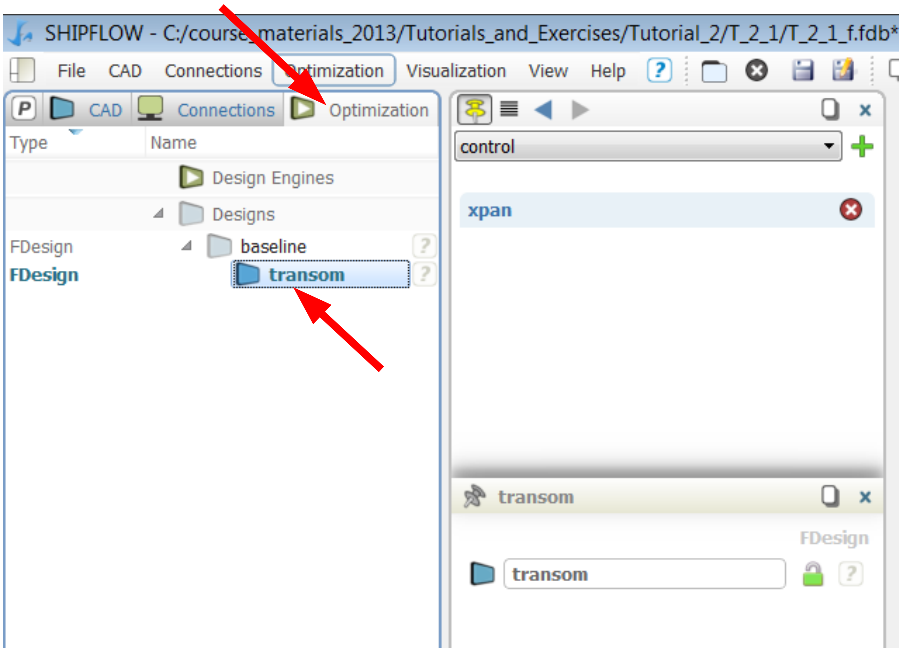
- Add *xmdens* to **Object Tree | Configurations | config | xflow | hull** and set density parameter to *coarse.*
- Add *fsflow* to **Object Tree | Configurations | config | xflow | hull**
- Set *speed to Fn = 0.632* in **Object Tree | Configurations | config | xflow | vship**
- Add *Rn* in **Object Tree | Configurations | config | xflow | vship** and set it to 1e7.
- Add *h1gr* entry to **Object Tree | Configurations | config | xflow | hull** and set it to *athena.*
- Add **xpan** to **Object Tree | Configurations | config**
- Add *parall* to **Object Tree | Configurations | config | xpan** and set number of threads to available cores in your computer CPU.
- Run the computations.

- Save the project.

Now we will create a design variant where we will add a transom group to the free surface.

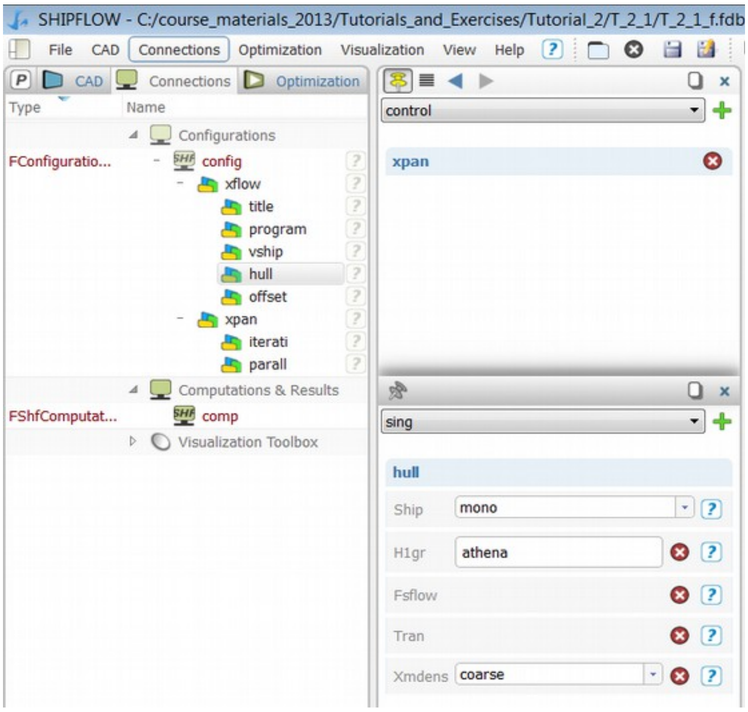
- First create a design variant from menu **Optimization > Create New Design From Current Design** and name it e.g. *transom*



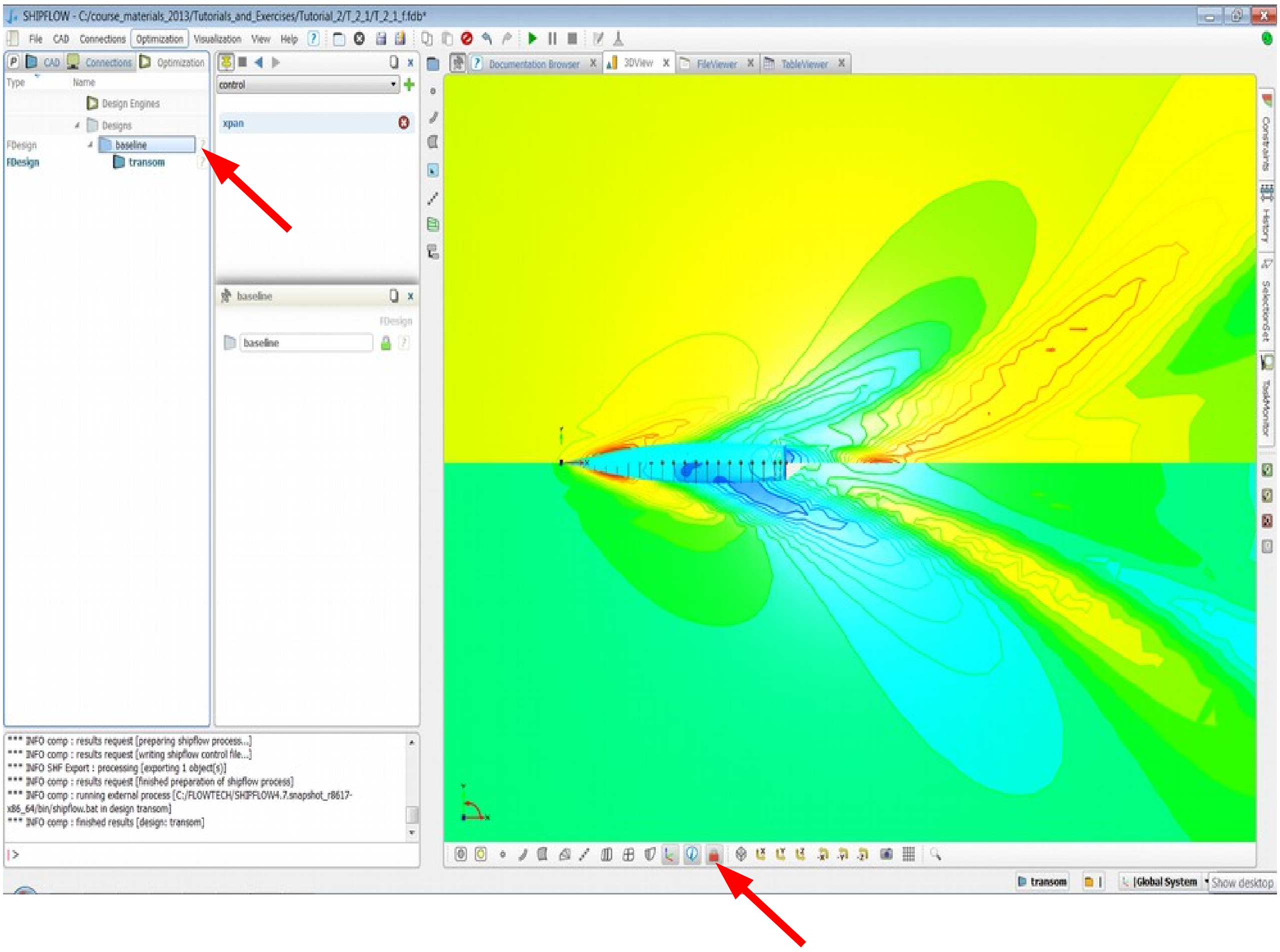


- This new design will become the current one automatically, you can see its name on the status bar at the bottom of the application window. All changes you make to the configuration will be contained in this variant and the baseline variant will remain unchanged.

- Now, add a *transom* option to **Object Tree | Configurations | config | xflow | hull** and run the computations again.
- Make sure that the **Ship** is set to **"mono".**

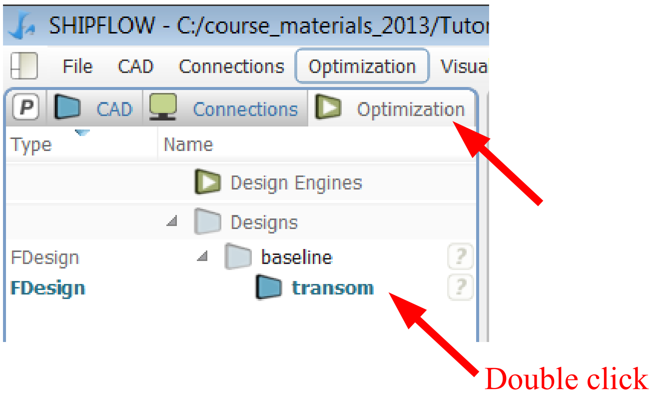


- Start the computation
- To compare the result with the baseline lock the current one by clicking the "lock" icon at the bottom of the 3DView and thereafter select **Object Tree | Optimization | Designs | baseline.**

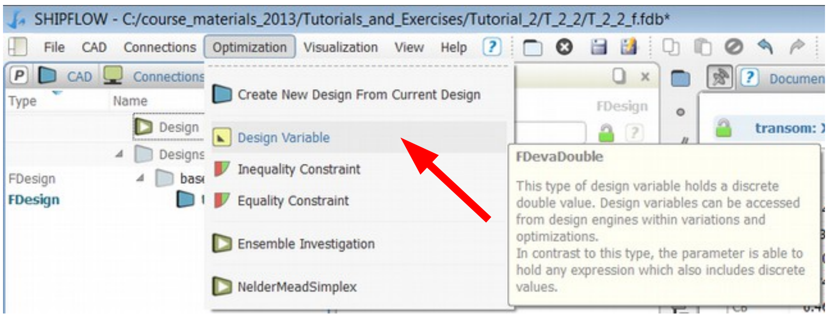


Tutorial 2 part 2 – Speed variation using Ensemble investigation

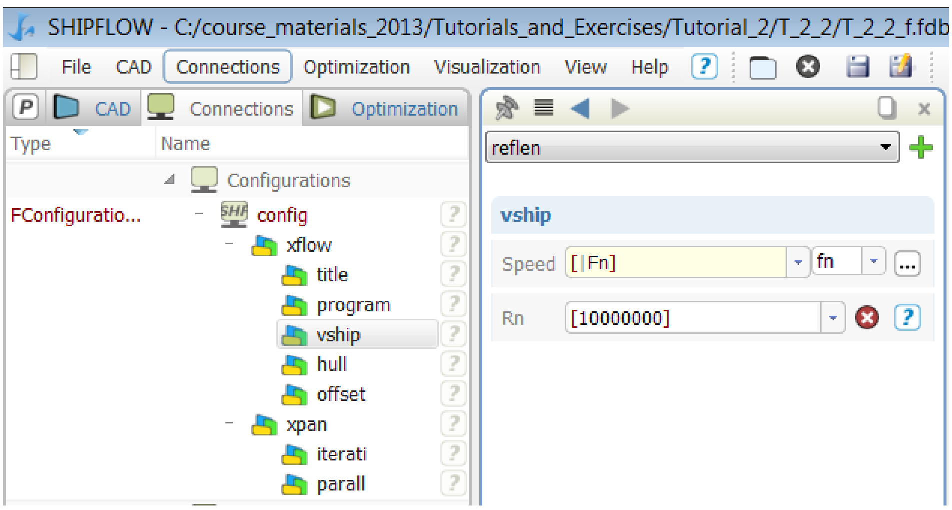
- Continue the previous work. Save the project as Tutorial_2_2.fdb.
- Make sure your current design is **transom**



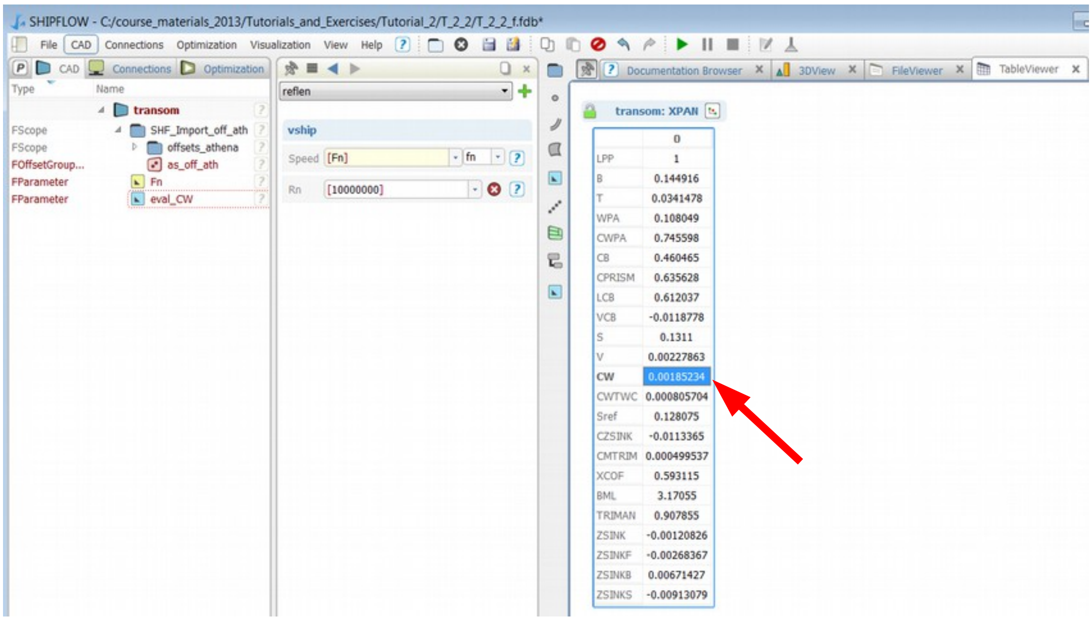
- Create a Design Variable from the menu **Optimization > Design Variable**. Name it **Fn** and set it to 0.5.



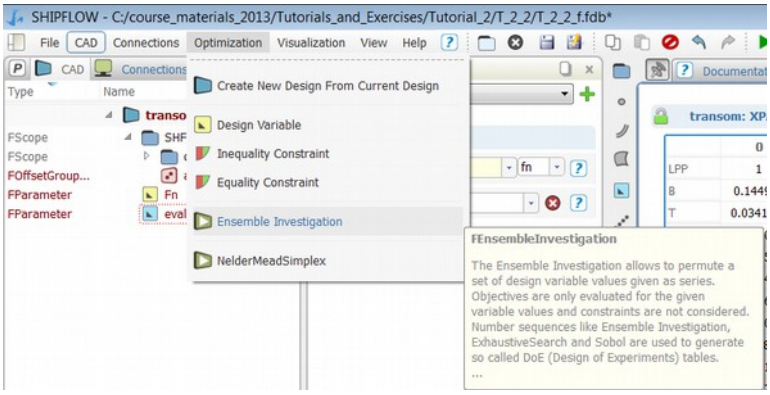
- Add the Fn variable in the vship command instead of the speed value in **Object Tree | Configurations | config | xflow | vship** section.



- Create a Parameter for the pressure resistance cw by double clicking **CW** value in the result table in the TableViewer



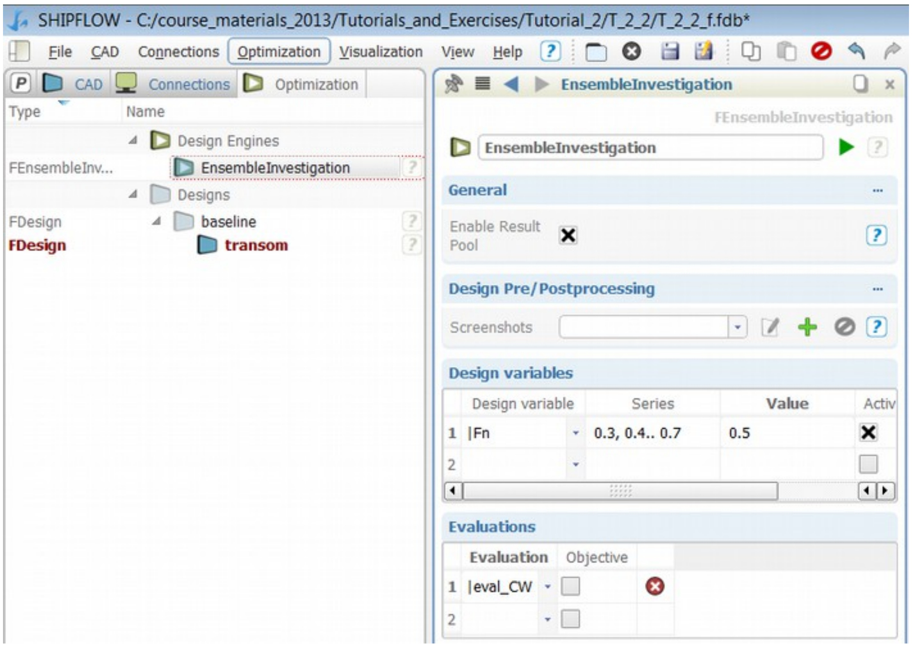
- Create an Ensemble Investigation from the menu **Optimization > Ensemble Investigation**.



- In the ObjectEditor for the Ensemble Investigation chose **Fn** as the Design variable. Set the

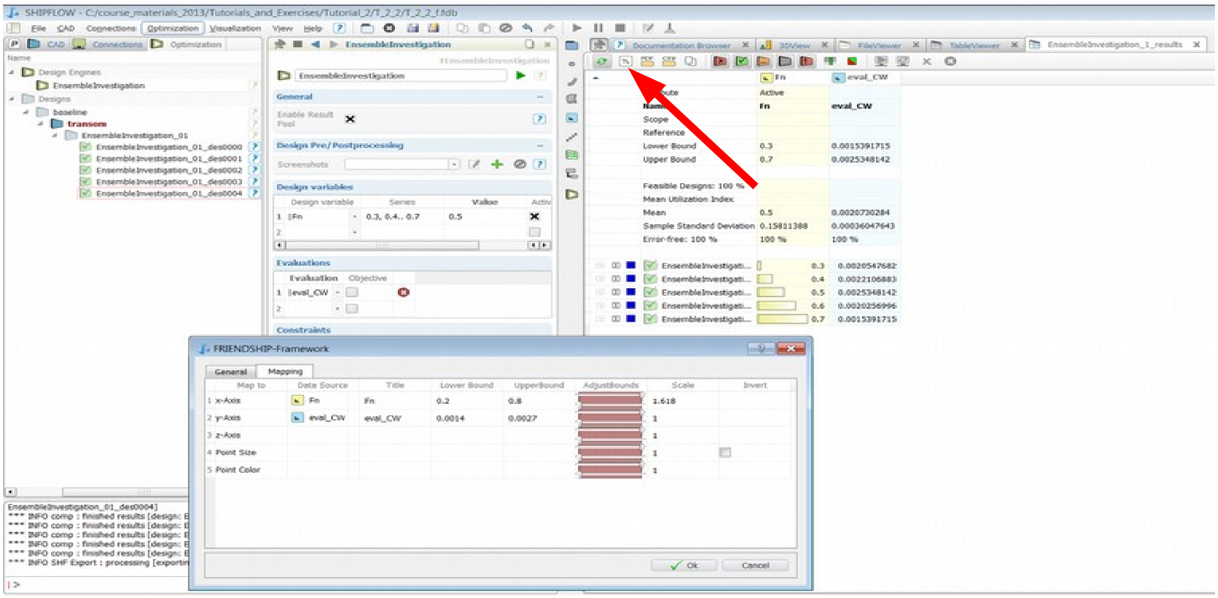
Series to **0.3,0.4..0.7**.

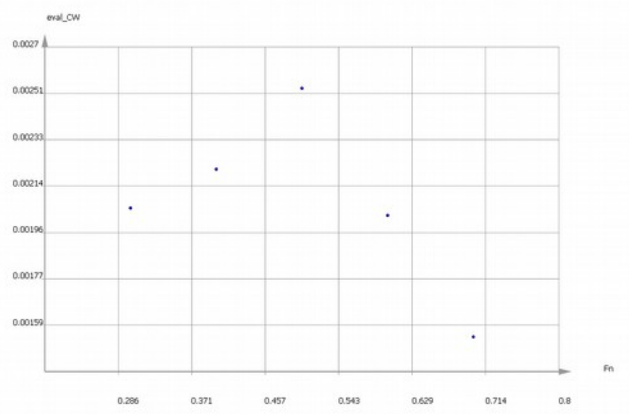
- In the ObjectEditor for the Ensemble Investigation choose **eval_CW** as the Evaluation parameter.



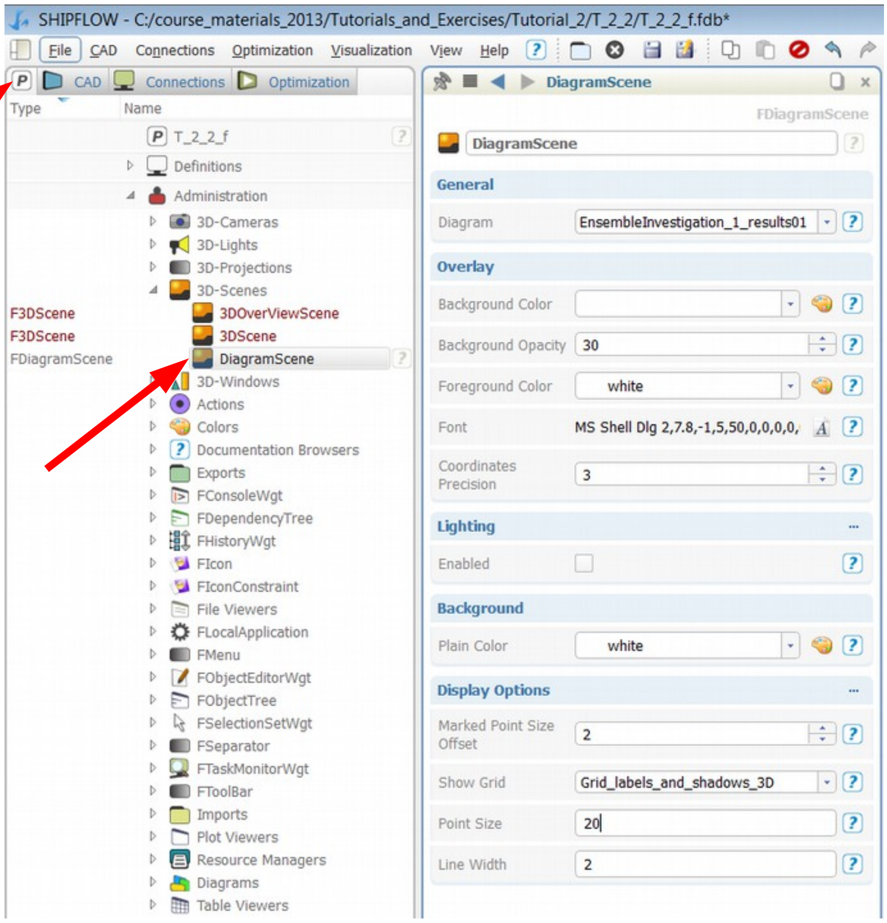
- Select the **Object Tree | Optimization | Design Engines | Ensemble Investigation** in and click Run (play icon).

- When it is finished you will get a table with Fn values and obtained CW values. These values can be displayed graphically by clicking the "create new diagram" button. When the diagram editor is open switch to Mapping tab and select the Fn for x-axis data and the CW for the y-axis. Finally click OK.

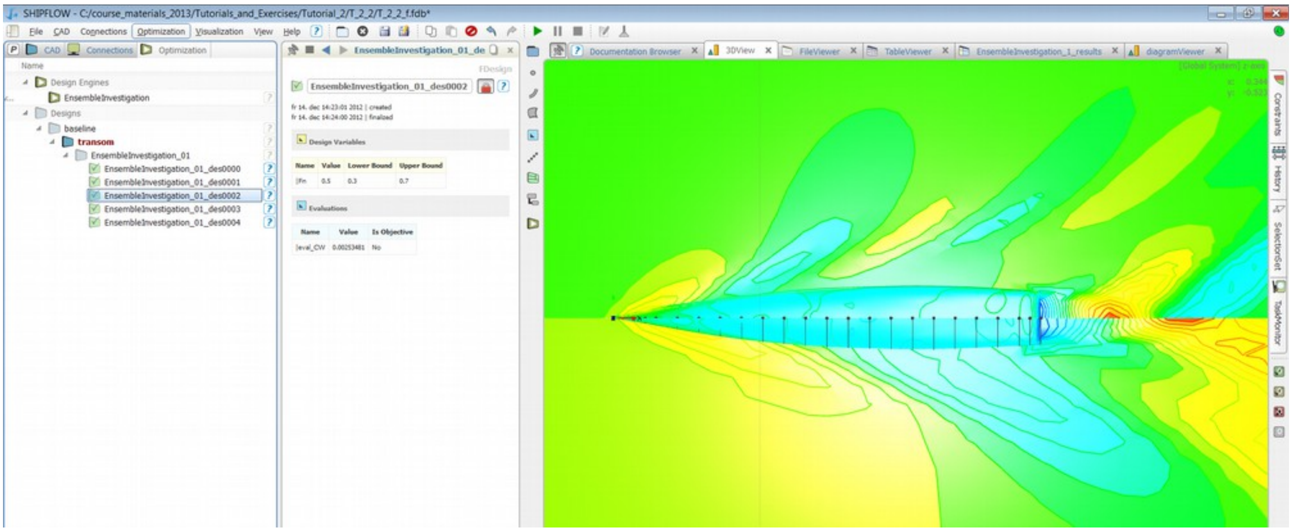




- The Diagram settings such as point size may be modified in the Adminstration tree in the DiagramScene. The Administration tab is located to the left in the Object tree and marked with letter P, see the picture below.



- The CFD Results can also be visualized and compared. You can find each computation under **Object Tree | Optimization | Designs | baseline | transom | EnsembleInvestigation.**

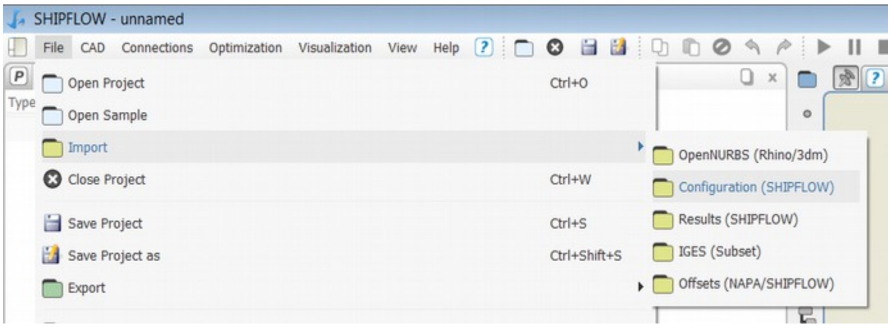


- Save and close the Project.

Tutorial 2 part 3 – Importing a command file and modifying the panellization

The purpose with this part of the tutorial is to show how to import existing SHIPFLOW command and offset files and modify the hull panellization. The command file is a simple hull configuration without a free-surface.

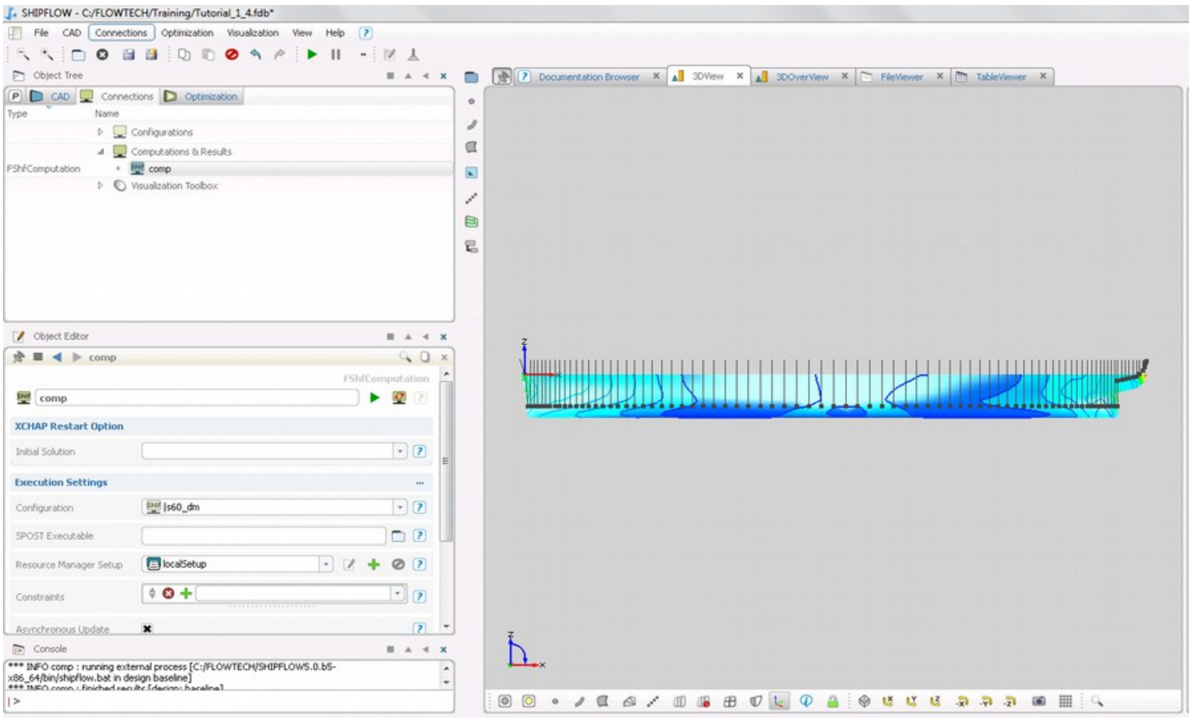
- Import file s60_*dm* from the directory **..\BasicTraining\Tutorial_2\T2_3\source** by selecting **File > Import > Configuration(SHIPFLOW)**



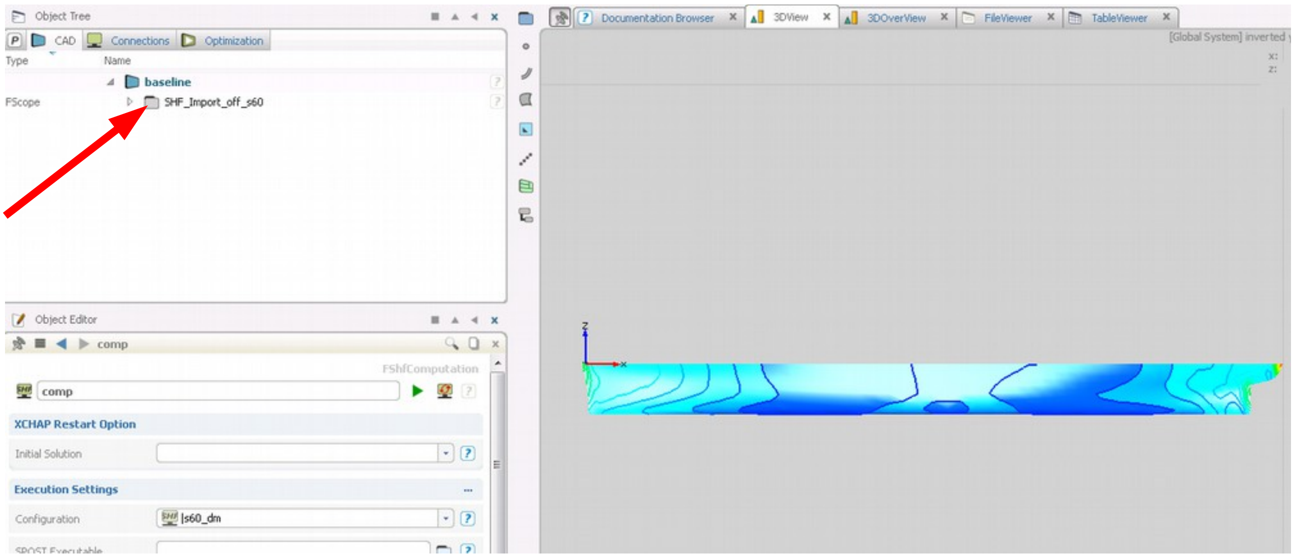
- Save the project e.g. Tutorial_2_3 by selecting File > Save as or click on the icon 

- Select SHIPFLOW computation in the object tree and Run the calculations by clicking on the icon 

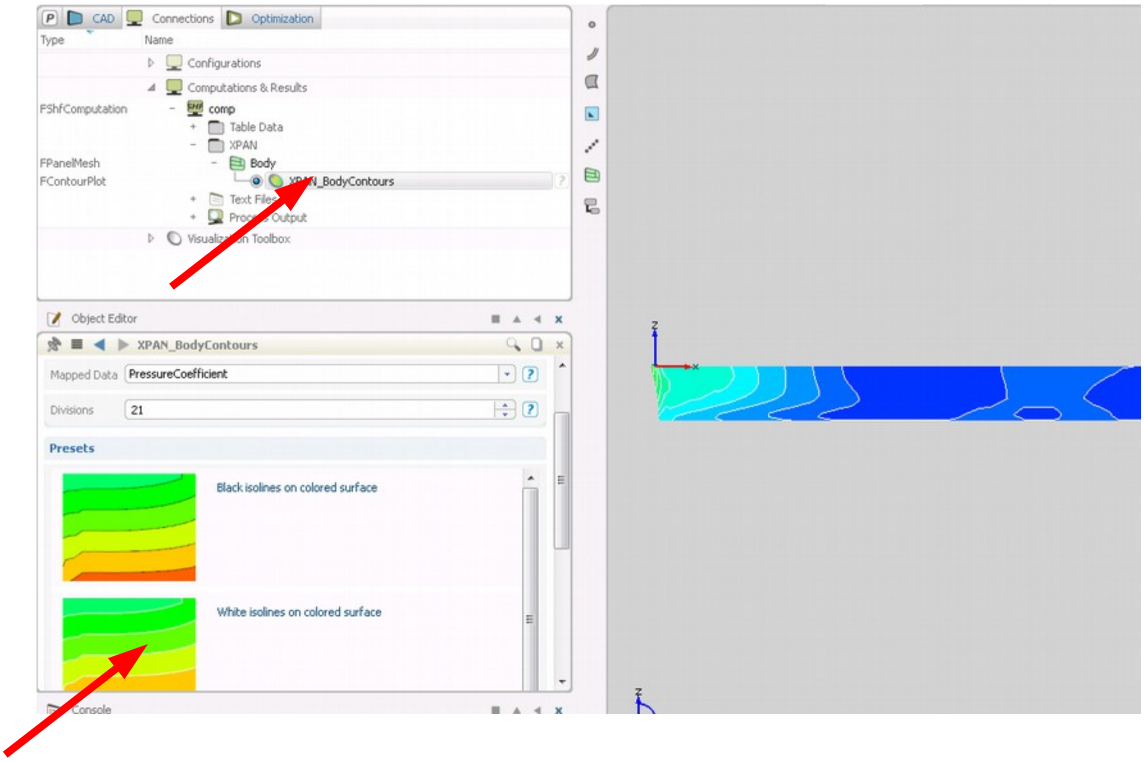
- Click 3DView tab to see the dynamic pressure on the hull.



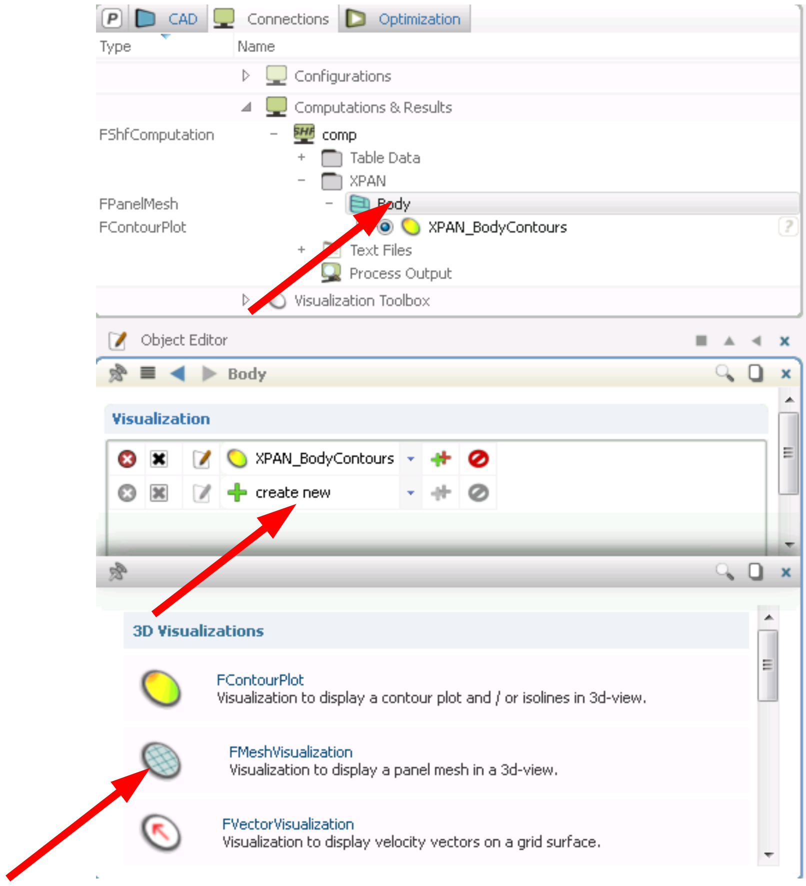
- In **Object Tree > CAD** click on the scope containing the offsets (SHF_Import_off_s60) to hide them from the 3DView.

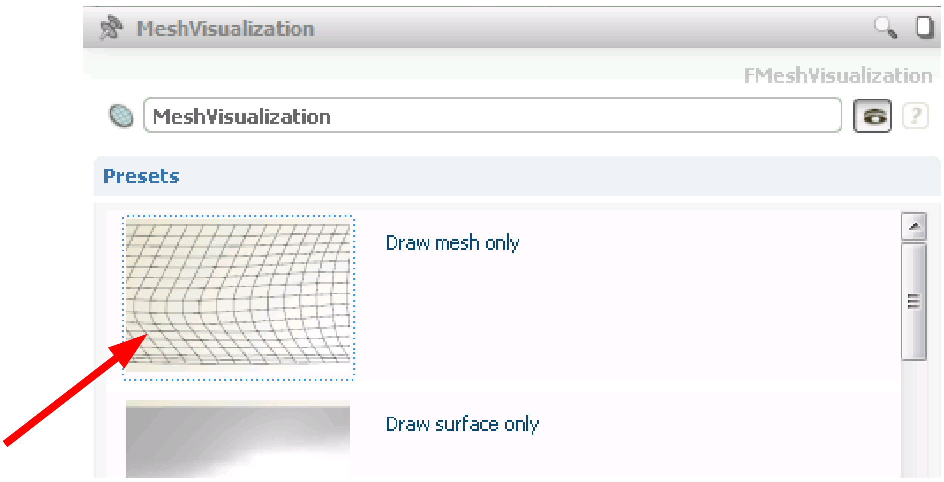


- In order to modify Visualisations go to **Object Tree > Connections > Computations & Results > comp > XPAN**.
  - Select **XPAN_BodyContours** and in the Object Editor click "**White isolines on colored surface**".

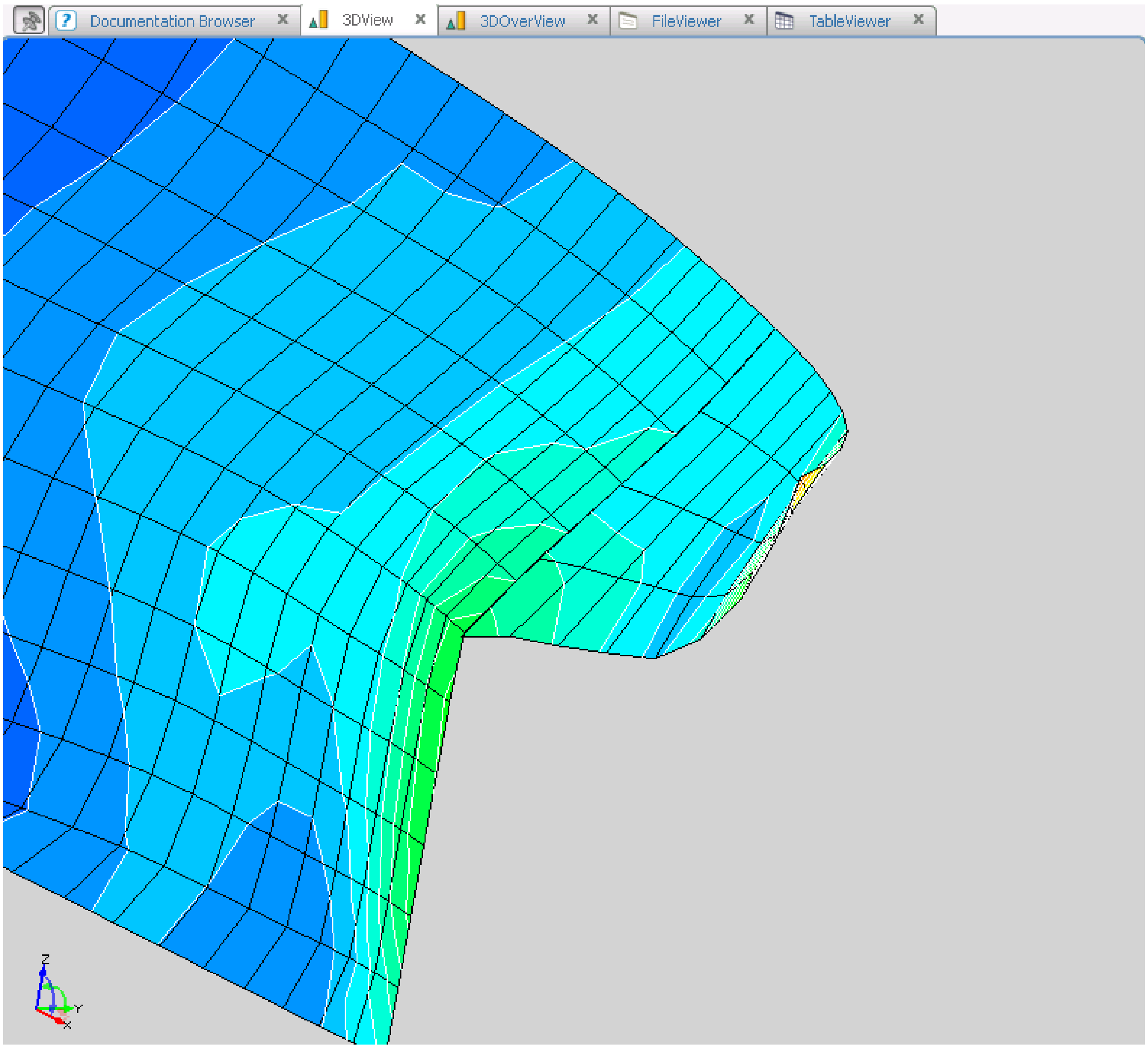


- Select **Body** and in the Object Editor first select “**create new**” and the click **FMeshVisualization** to show the panellization on the hull. Finally click “**Draw mesh only**”.





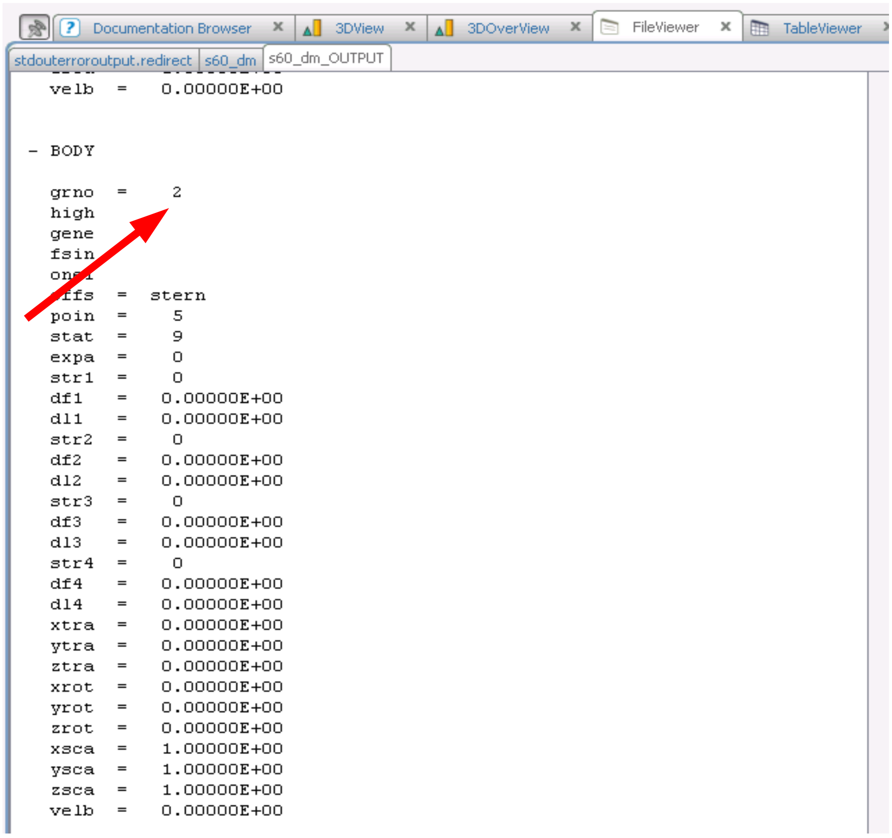
- Zoom in on the stern and you can see for this specific hull that the panels do not match between the stern and main groups. The stern has 5 points and the main group has 7 points for the same region. We will now fix this!



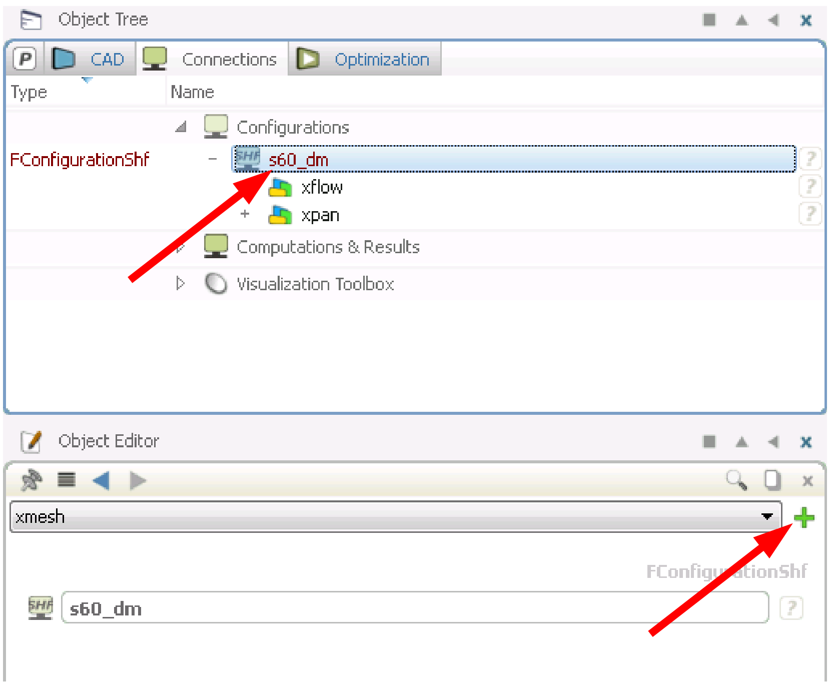
- In order to keep the result we have we will work on a different design. Select "**Create New Design From Current Design**" in the Optimization menu. The newly created design will be called des1 and will be the active one in the CAD and Connections tabs in the Object Tree.

- Run XPAN for the new design by select SHIPFLOW computation in the object tree and Run the calculations by clicking on the icon 

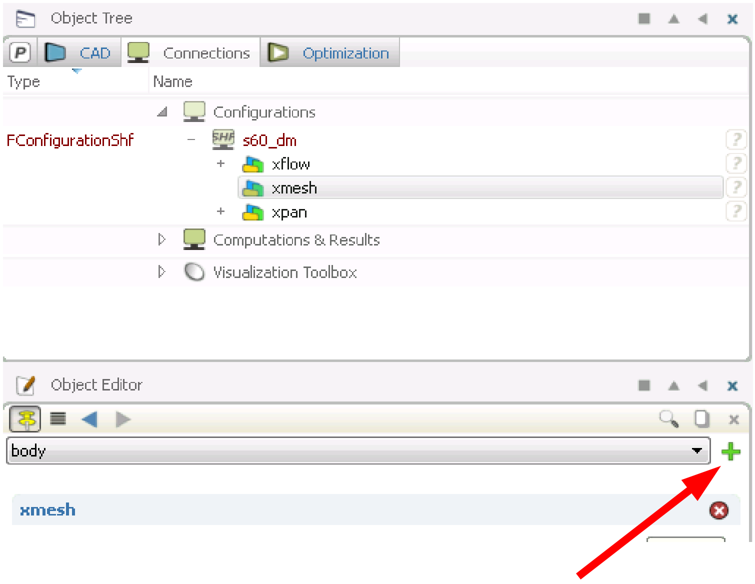
- Check which group number the stern group has in the SHIPFLOW OUTPUT file by selecting FileViewer>s60_dm_OUTPUT and scrolling down to the stern group.

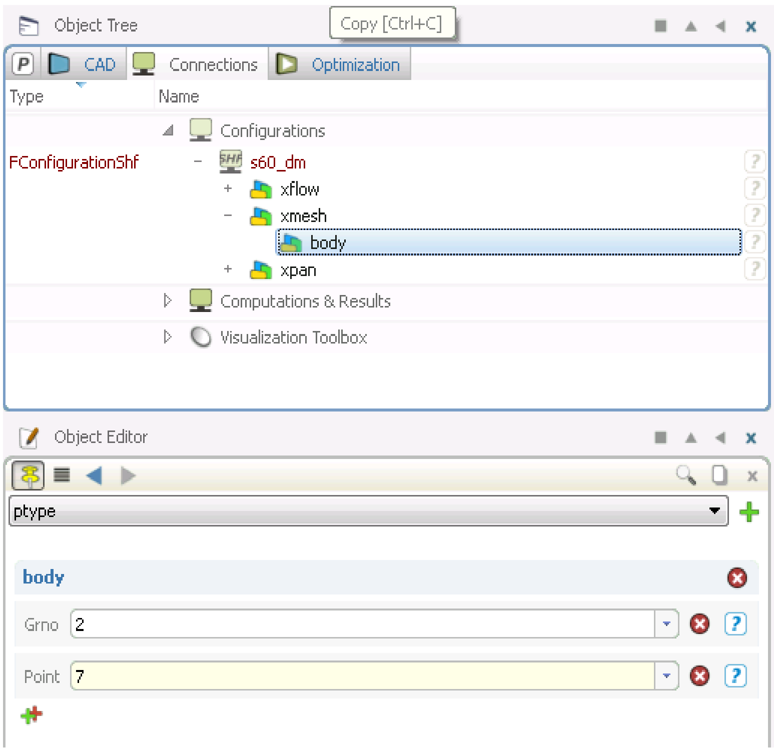


- Select Object Tree > Connections > Configurations > s60_dm and create the XMESH module in the Object Editor.



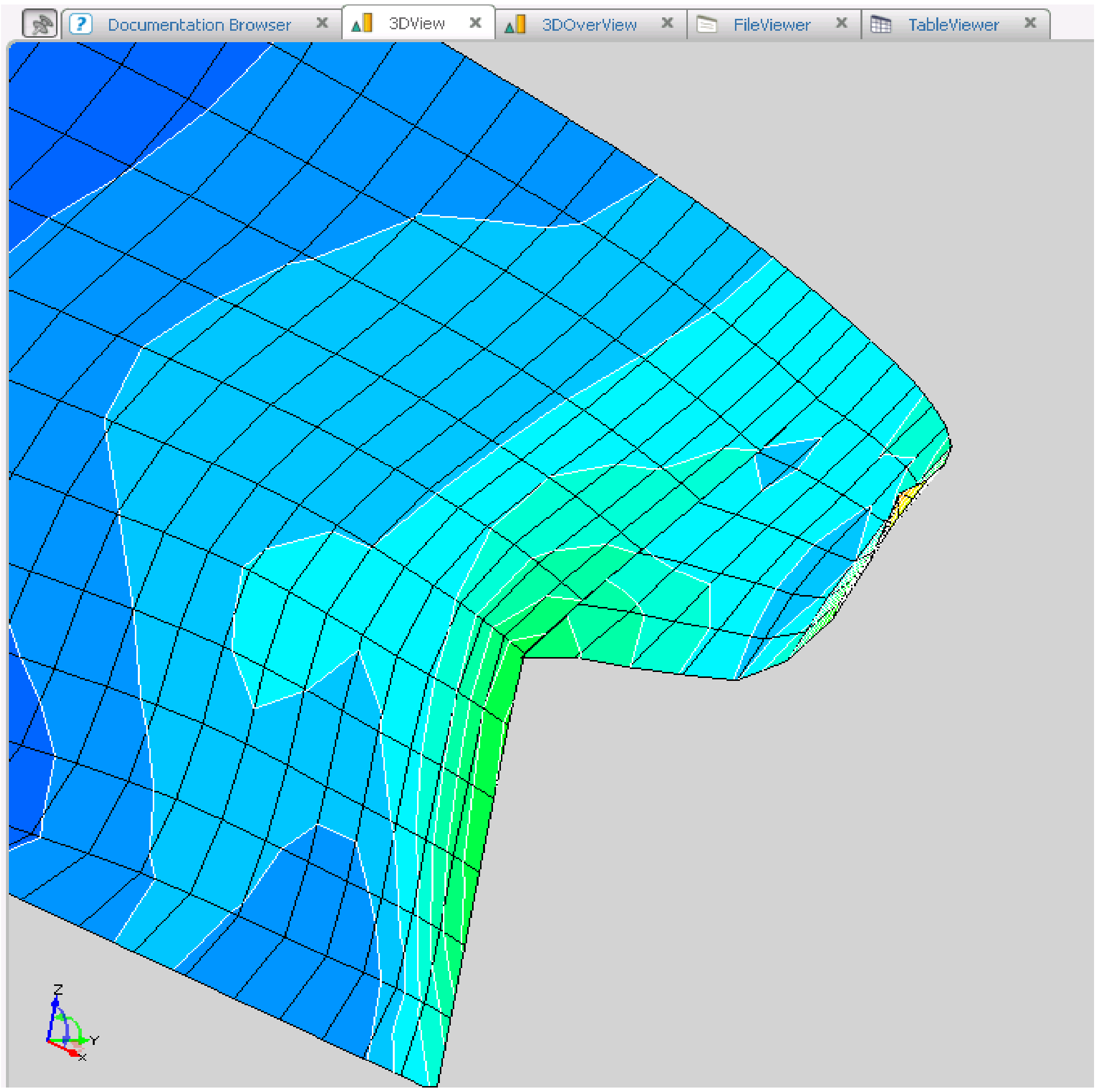
- In the XMESH module create a BODY command with the keywords GRNO and POINT. Set GRNO to the stern group number found in the OUTPUT file previously and POINT to 7.





- Select SHIPFLOW computation in the object tree and Run the calculations by clicking on the icon 

- In the 3DView you can now see that the panellization looks better around the stern.



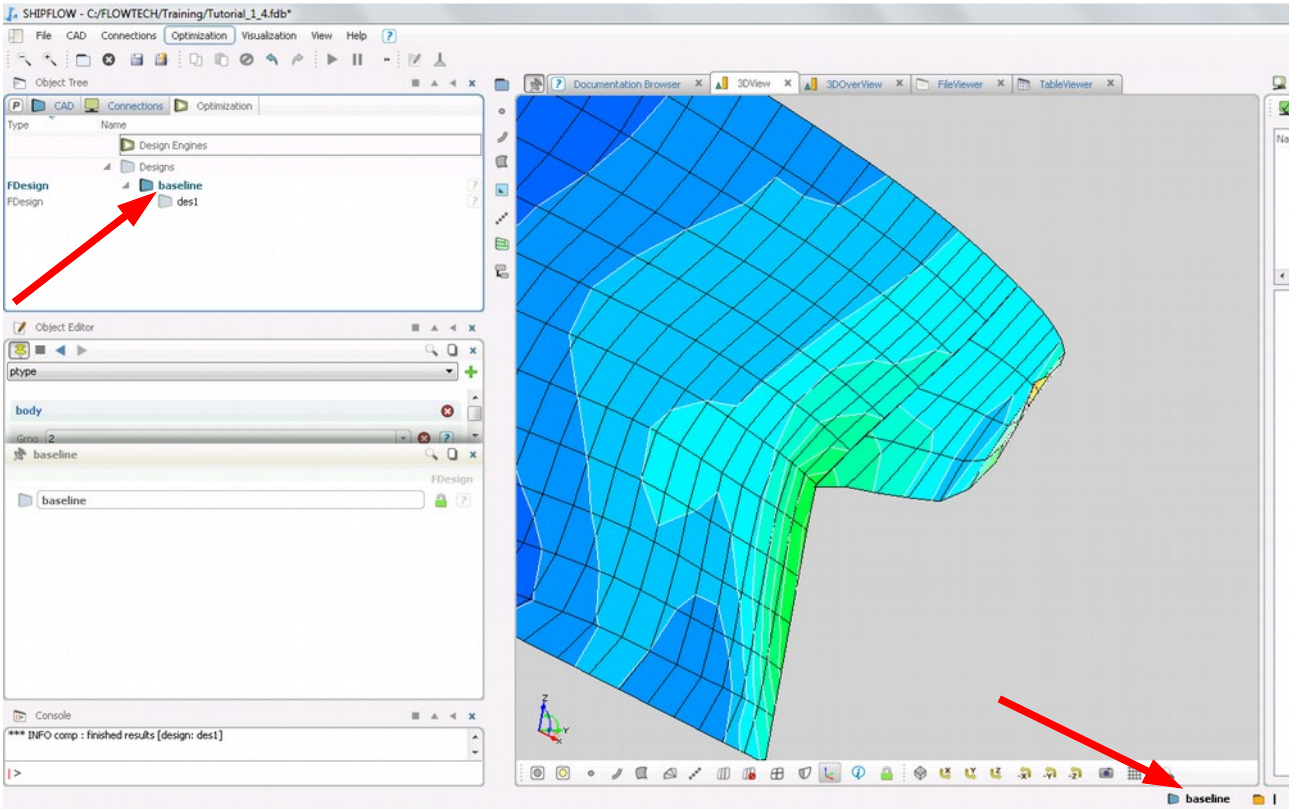
- Save the project.

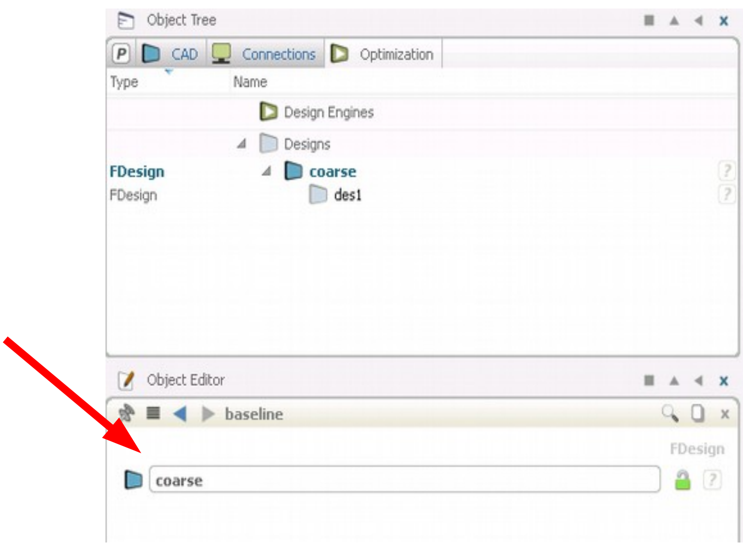
Tutorial 2 part 4 – A mesh refinement study

This is a continuation of the previous tutorial. The purpose with this tutorial is to learn how to refine the mesh and understanding why it is important.

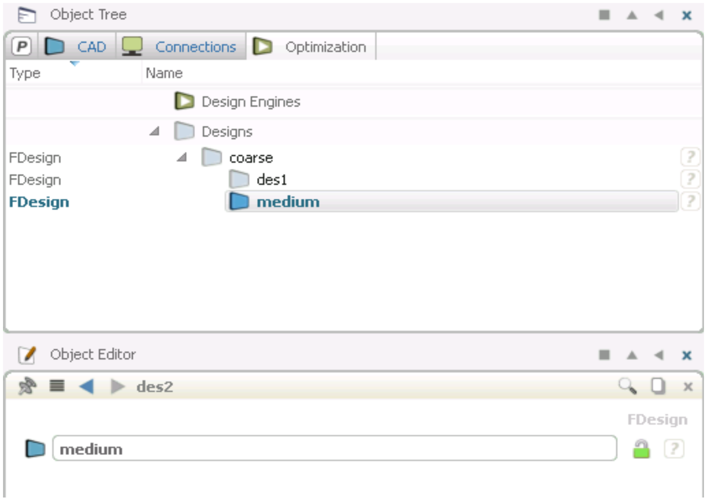
- Continue from the project in the previous tutorial.

- In **Object Tree > Optimization** double click on **baseline** to make it the current design. Check the lower right corner to see that baseline is set as the current design. Rename the design “coarse”.

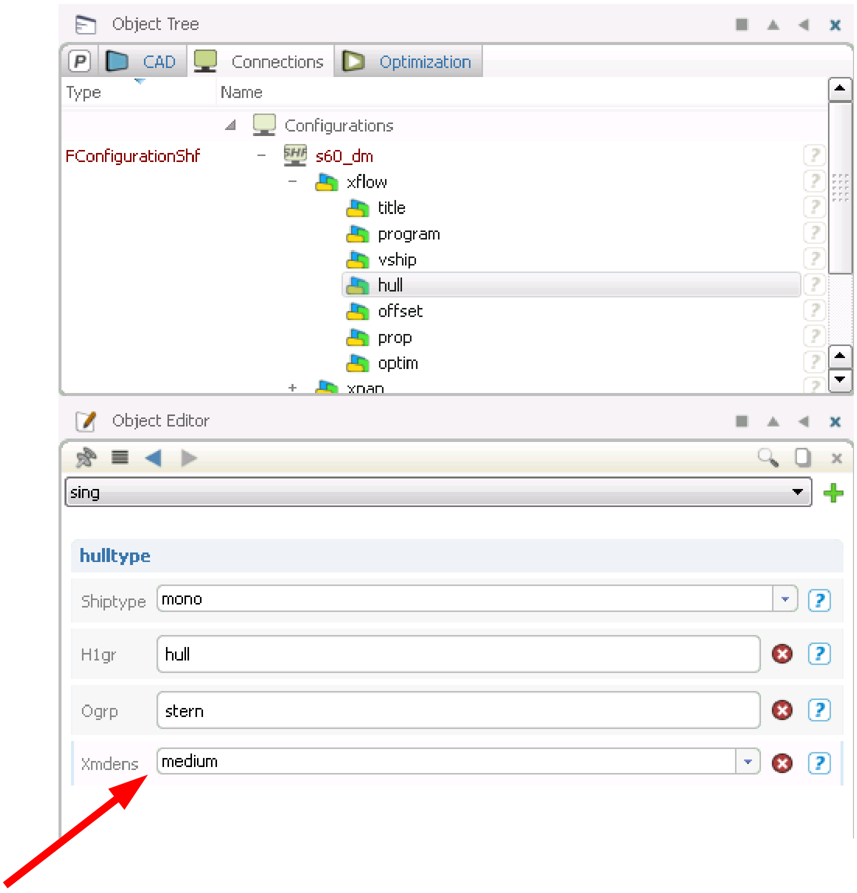




- Select “Create New Design from Current Design” from the Optimization menu. Rename it “medium”.

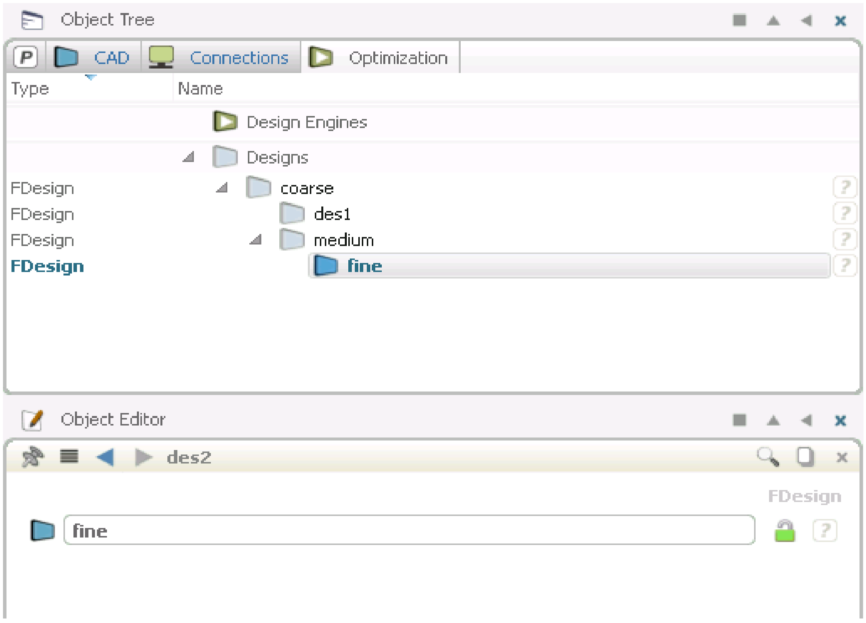


- In Object Tree > Connections > Configurations > s60_dm > xflow > hull change keyword Xmdens from coarse to medium.



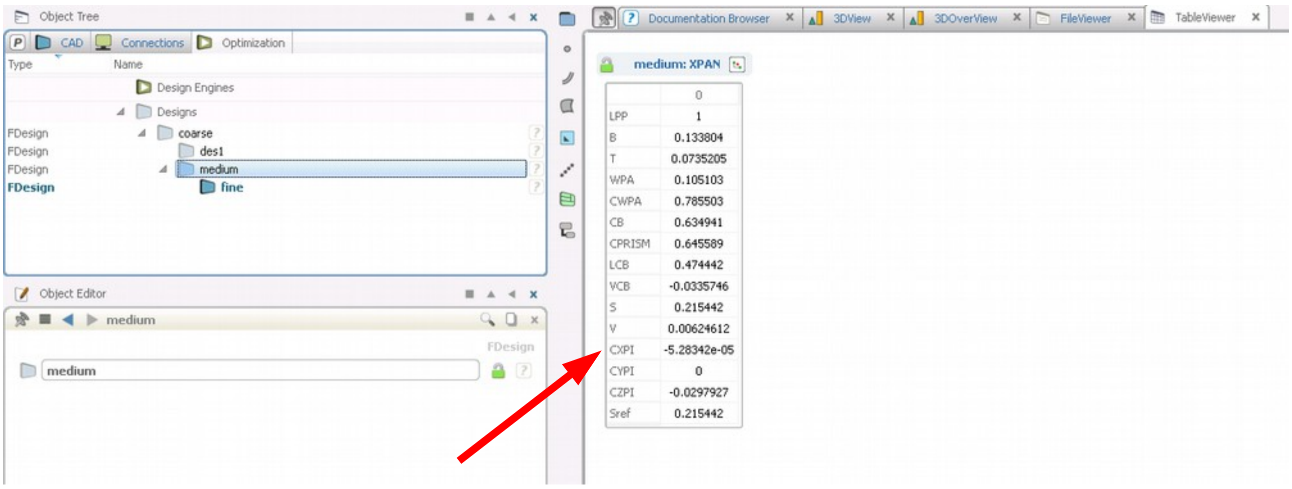
- Select SHIPFLOW computation in the object tree and Run the calculations by clicking on the icon 

- Create another design from the medium design and call it fine. For this design set Xmdens to fine and run SHIPFLOW.



For double model calculations (no free surface) the integrated pressure force on the hull is zero according to potential flow theory. Since we are using numerical methods to calculate the pressure on the hull, the pressure force from XPAN will not be zero, however it is a good measure of how good your hull panellization is. CXPI is the integrated pressure force from XPAN. It is important that CXPI is much smaller than the wave resistance CW from XPAN with a free surface, since CW is also calculated from pressure integration.

- Check how CXPI varies for different panellizations. CXPI is found in TableViewer tab. You can see results for different designs by clicking on them in Object Tree > Optimization.



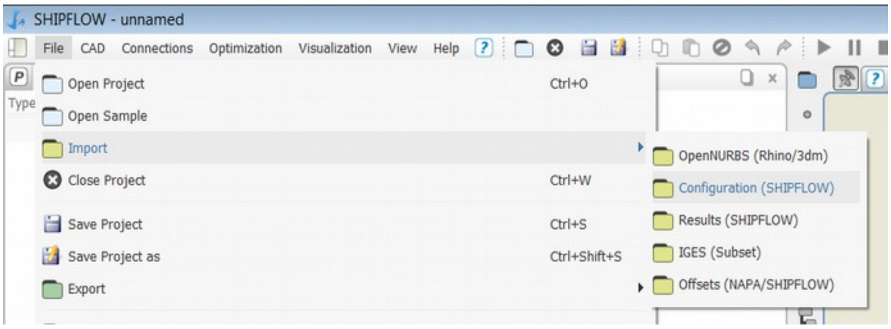
- Save the project.

- Extra exercise: Create more designs and add a free surface. Run and compare CW to CXPI.

Tutorial 3 part 1 – Importing a command file and running XPAN

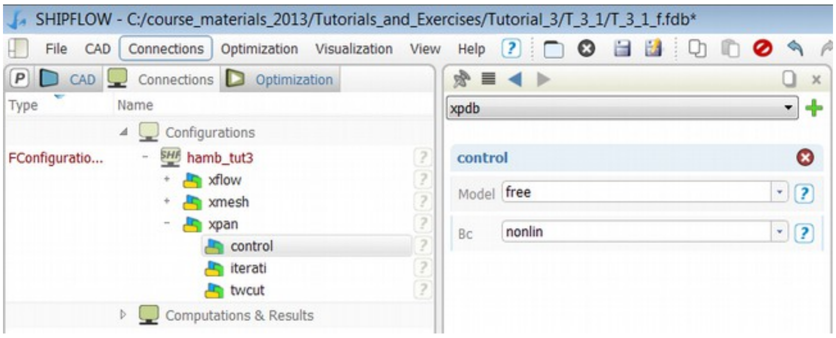
The purpose with this part of the tutorial is to show how to import existing SHIPFLOW command and offset files to create a new project. The command file is a simple configuration for a linear free-surface computation. It will serve as a starting point for the following tutorial on optimisation.

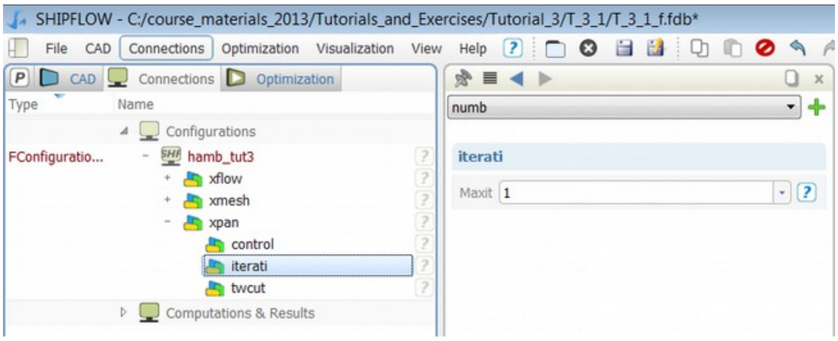
- Import file *hamb_tut3* from the directory **..\BasicTraining\Tutorial_3\source** by selecting **File > Import > Configuration(SHIPFLOW)**



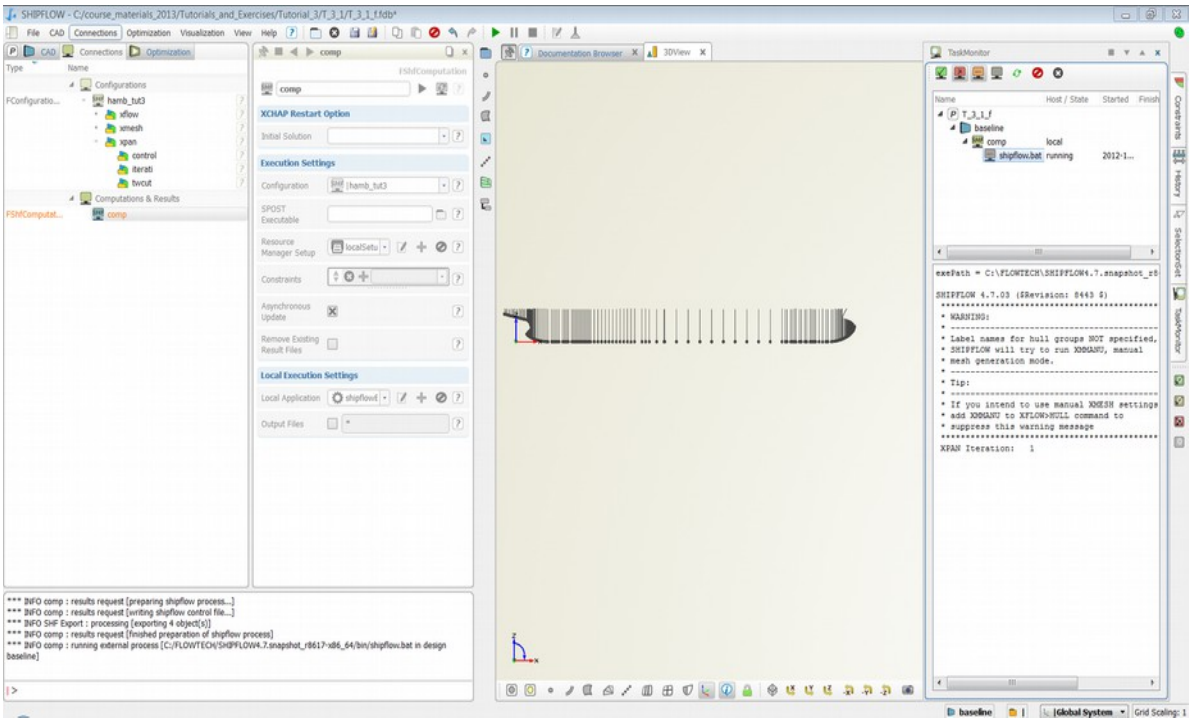
- Save the project e.g. Tutorial_3_1 by selecting File > Save as or click on the icon 

- Check that the XPAN configuration is set to **nonlinear** and **free** condition and that the number of iterations is restricted to **1**.

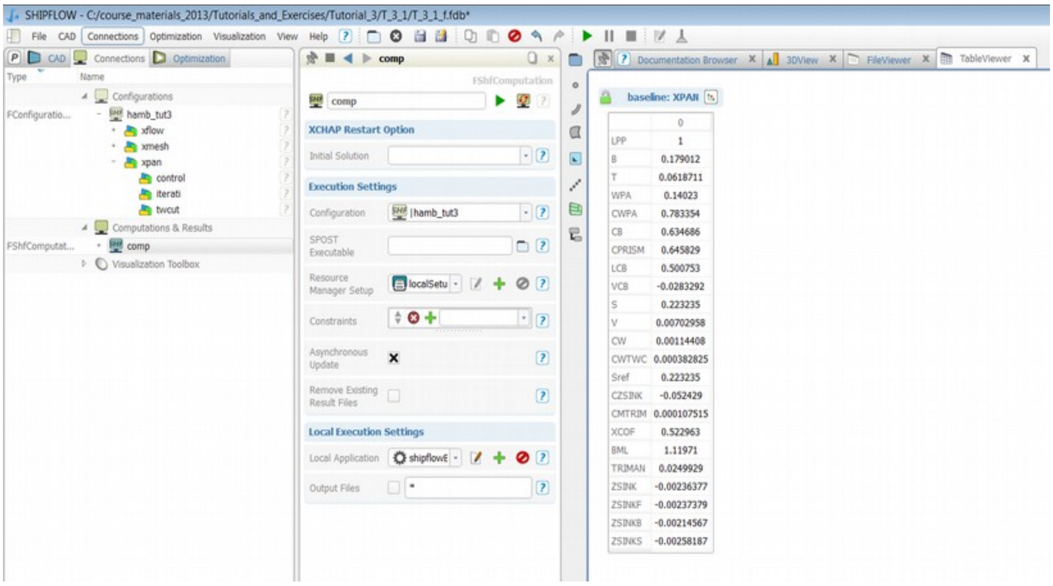




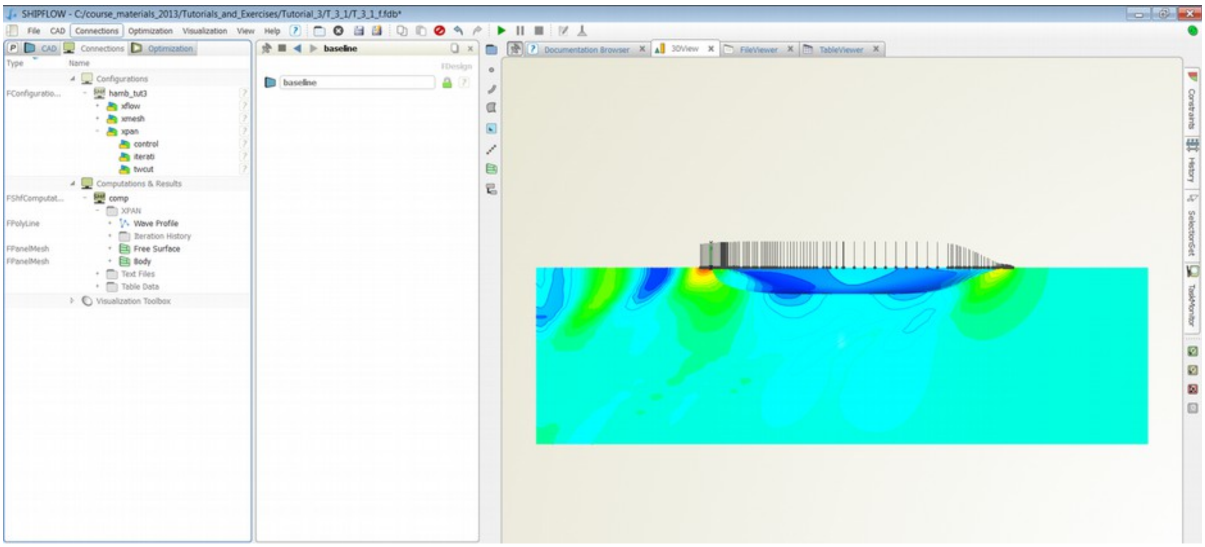
- Select SHIPFLOW computation in the object tree and Run the calculations by clicking on the icon 



- When the calculations are finished check the results in the FileViewer. A copy of the command file and the OUTPUT file are available in the tabs and the results summarised in a table available in the TableViewer tab.



- To display the XPAN results select 3DView window tab. In order to modify Visualisations go to **Object Tree > Connections > Computations & Results > comp > XPAN.**

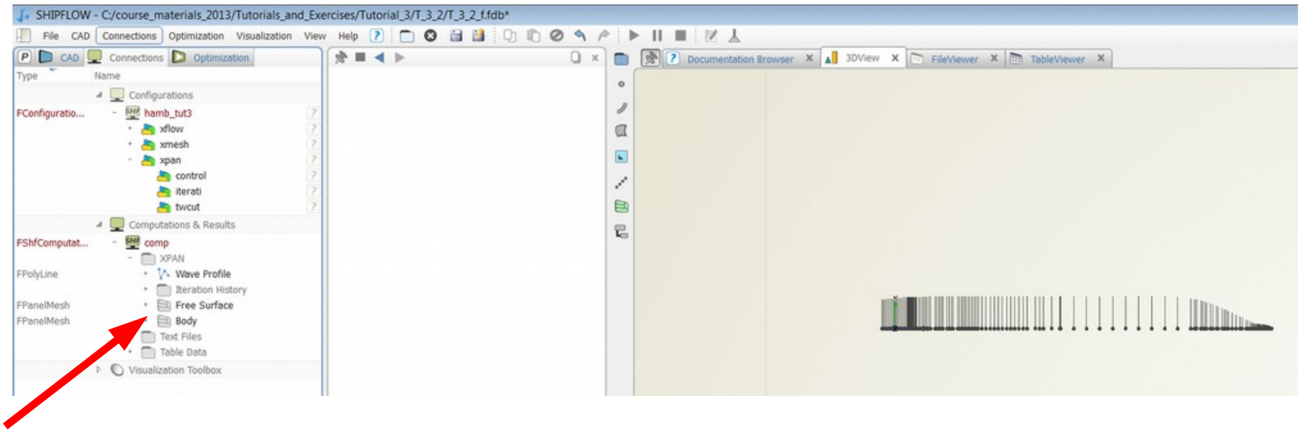


- Select the top view by clicking ”Z” axis and zoom to extent. For clearer view you can switch off the visibility of the scope with the offset points. The program window should look as above.

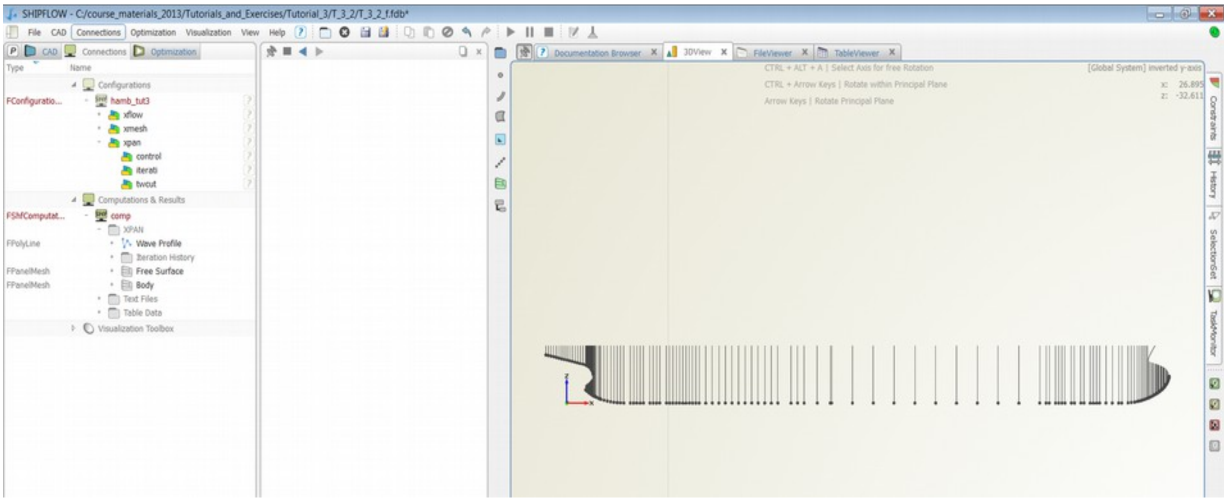
- Save the project.

Tutorial 3 part 2 – Hydrostatics calculations

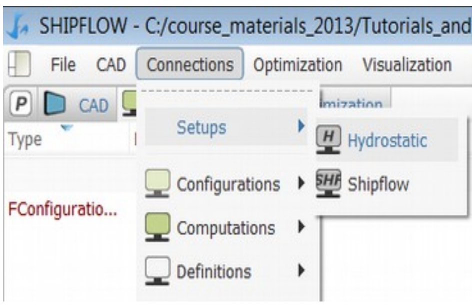
- Continue the previous work or open the T_3_1_f.fdb file from **..\BasicTraining\Tutorial_3\T_3_1\.** Save the project as Tutorial_3_2.fdb.
- For convenience switch off the visibility of the CFD results by clicking on the icon of Free surface and Body in the object tree.



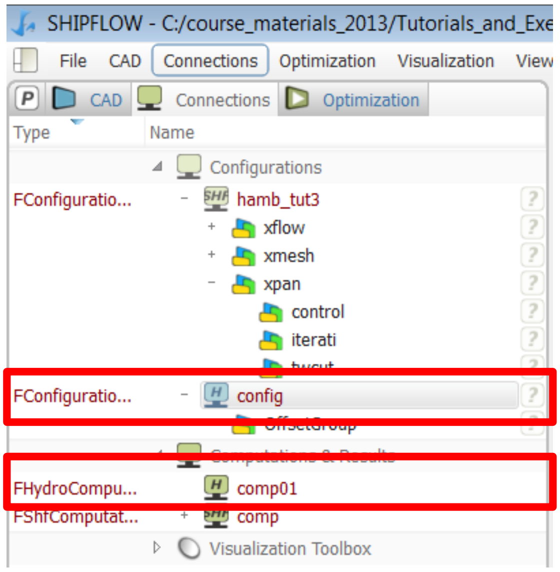
- Switch on the offset points visibility.
- Set view to "Y" and zoom extent. The screen should resemble the one below.



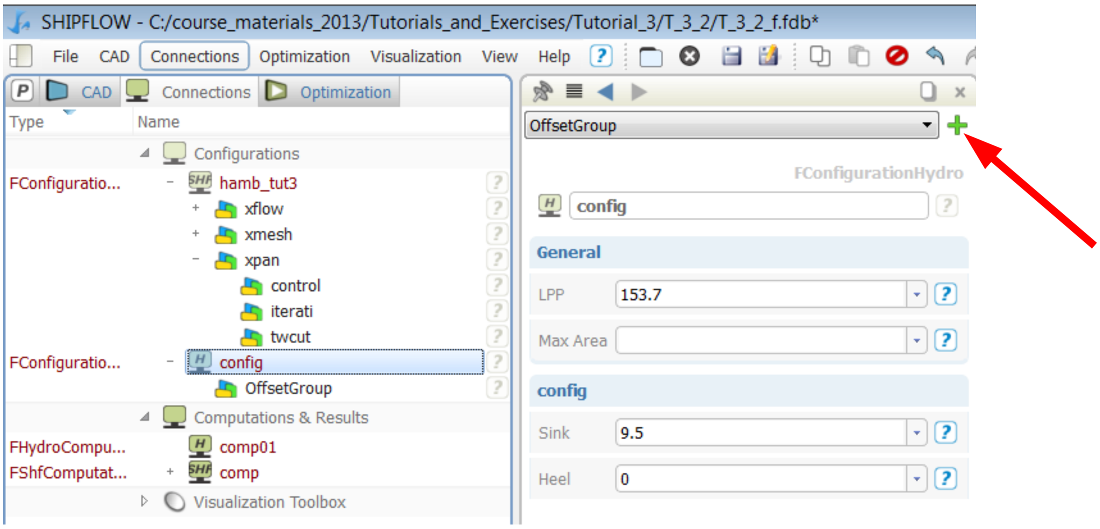
- Create the hydrostatic configuration and computations by selecting from the menu **Connections > Setups > Hydrostatic.**



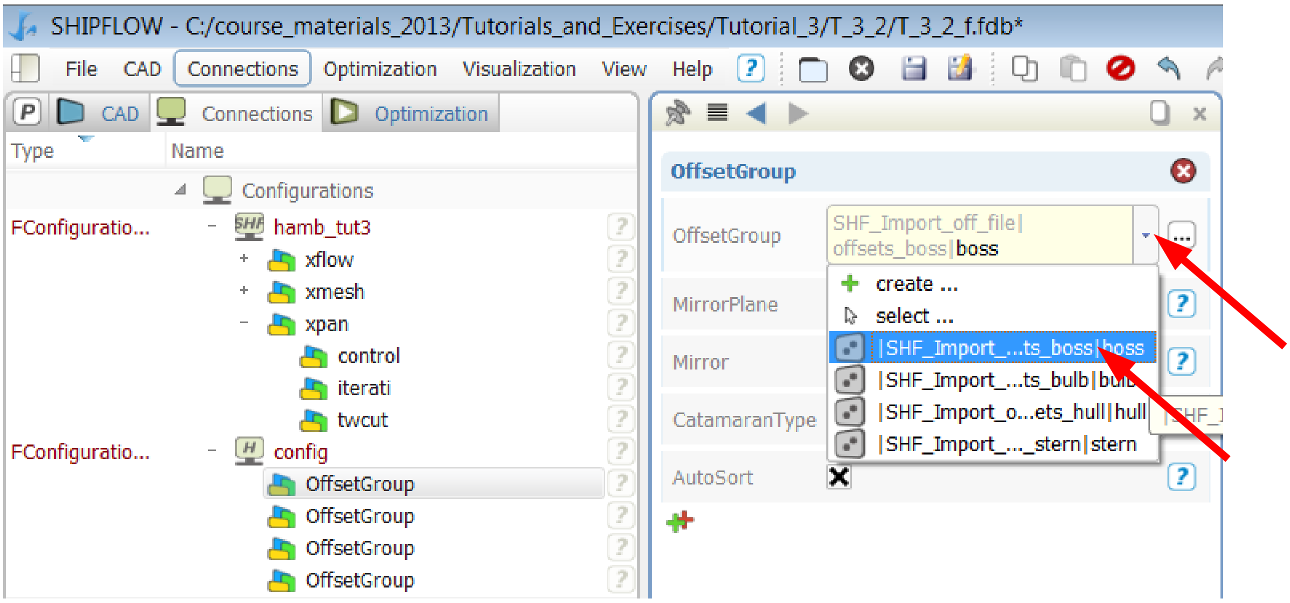
- Now in the Object Tree you should find the FconfigurationHydro (see Type of the object) with a default name (case1) in the baseline scope and also FhydroComputation with default name (caseRun2) in the Computations scope.



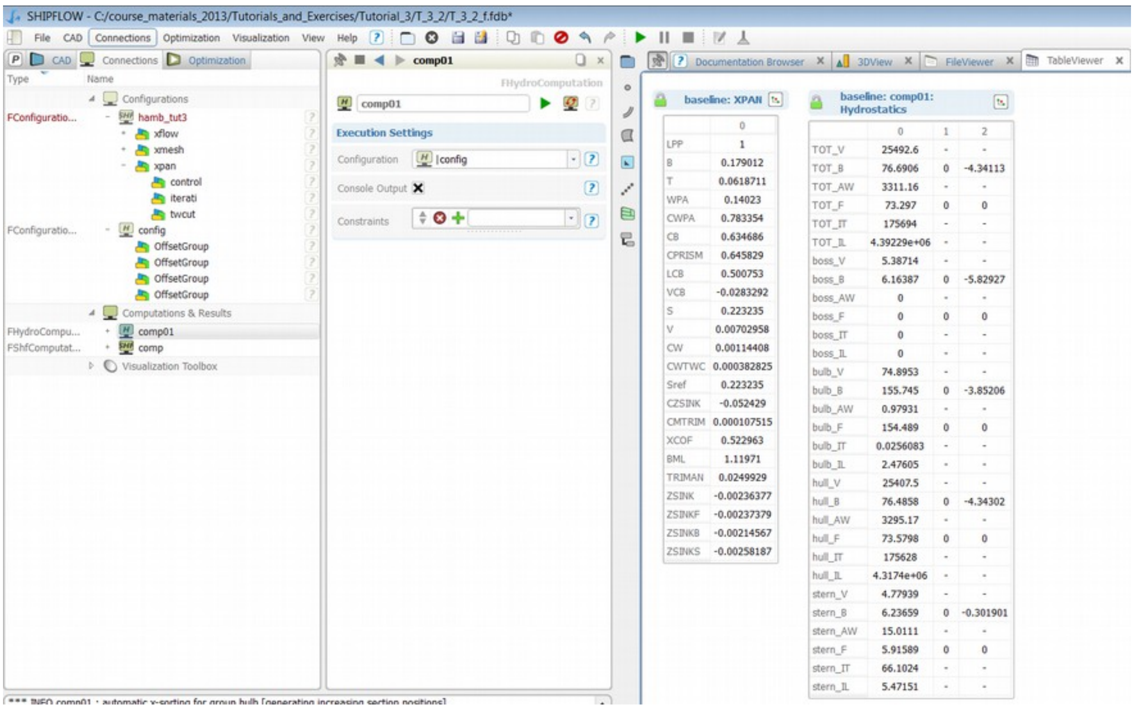
- Select hydrostatic configuration in the Object Tree and in the Editor give the Lpp = 153.7 and set the Sink to 9.5.
- Add three more Offsetgroups in the hydrostatic configuration by clicking the ”+” icon.



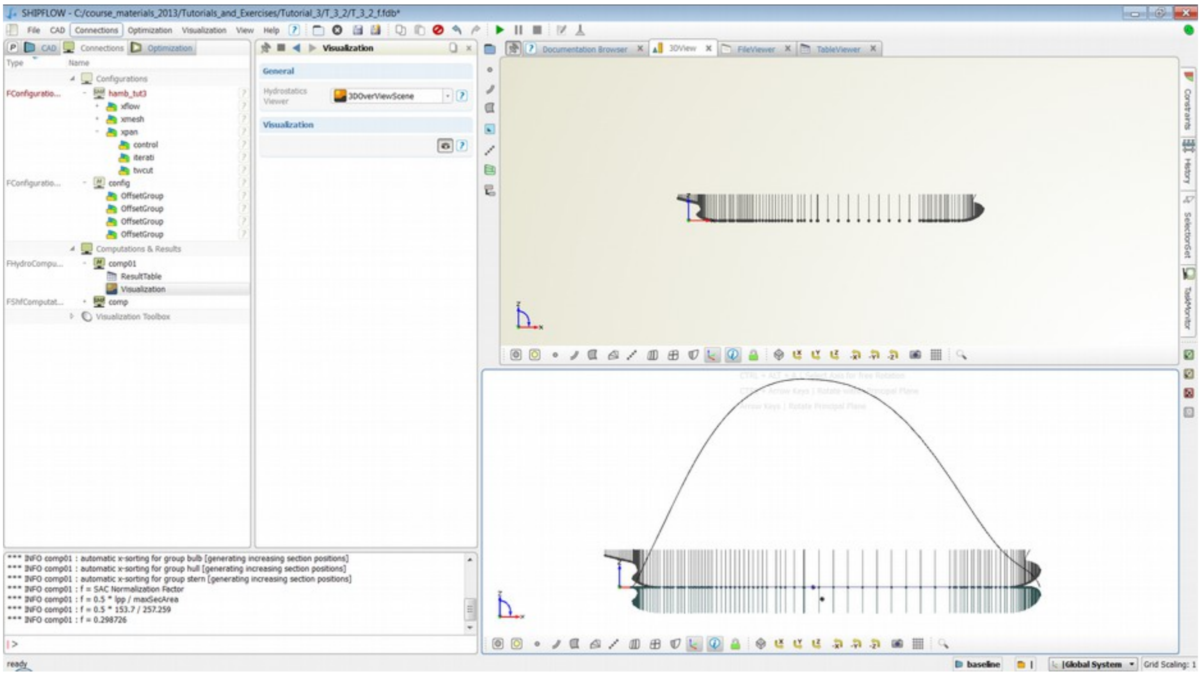
- For each offset group in the hydro configuration set the appropriate offset group from the SHF_Import_off_file scope as shown in the picture below. To select you can use the drop-down menu in the OffsetGroup entry.



- Select the Hydrostatic Computations in the Object Tree and run the hydrostatic calculations by clicking the ”play” icon. A table with hydrostatic data should be now available in the TableViewer.



- Display the 3DOverView window from menu **View > Windows > 3DOverView**, rotate the view to look from a side. Now you should see the Sectional Area Curve as below.

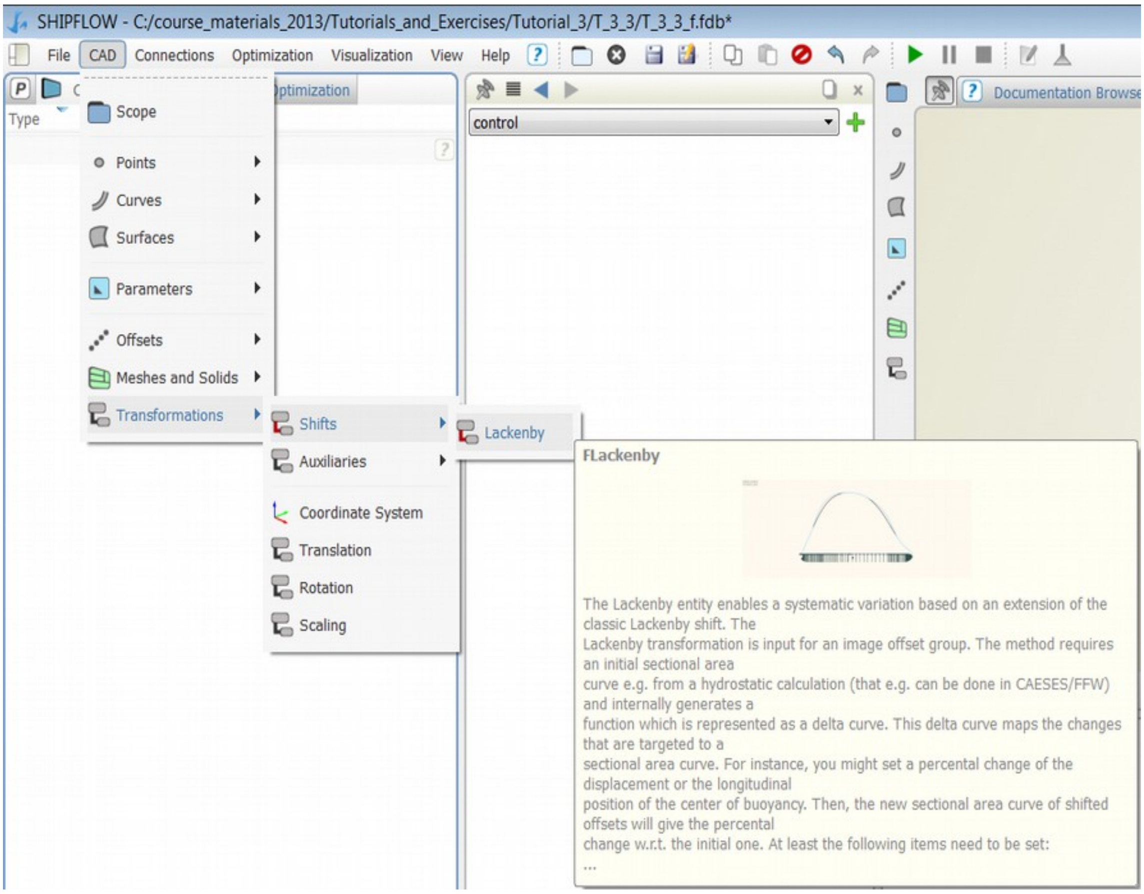


- Save the project.

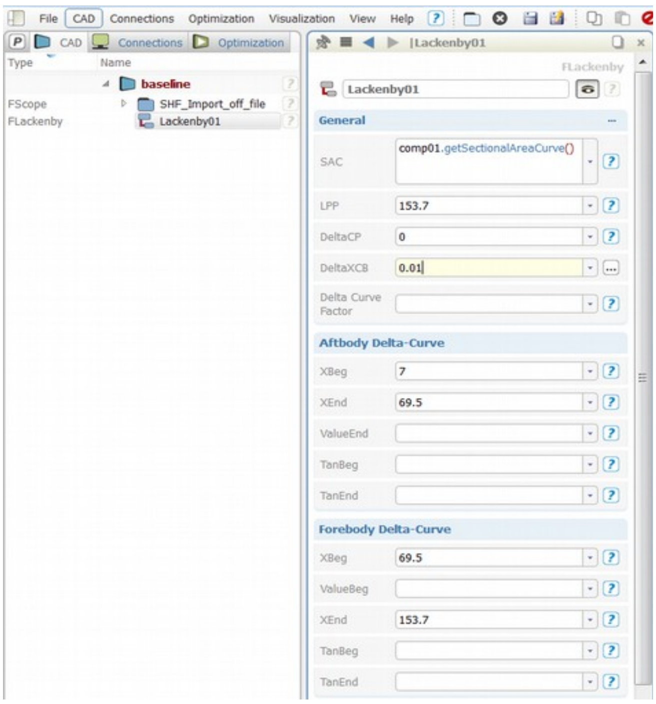
Tutorial 3 part 3 – Hull form modification using Lackenby shift

- In this part of the Tutorial a hull modification using the Lackenby shift will be applied to the main part of the hull.
- Continue the previous work or open the T_3_2_f.fdb file from **..\BasicTraining\Tutorial_3\T_3_2**. Save the project as Tutorial_3_3.fdb.

- Create a Lackenby entity by selecting from the menu **CAD** > **Transformations** > **Shifts** > **Lackenby.**

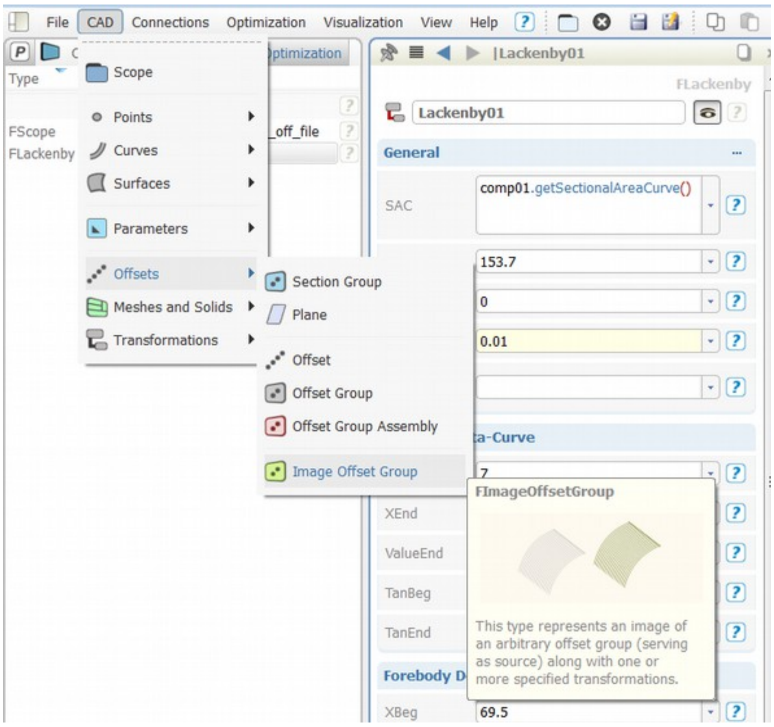


- In the Object Editor specify the parametres for the Lackenby as follows:
  - Lpp = 153.7
  - DeltaXCB = 0.01 (this value corresponds to a longitudinal shift of the centre of bouancy by 1% of Lpp forward)
  - Xbeg = 7.0 for the Aftbody
  - Xend = 69.5 for the Aftbody
  - Xbeg = 69.5 for the Forebody
  - Xend = 153.7 for the Forebody
  - For the SAC the input should be the Sectional Area Curve computed by the Hydro Computations. This can be extracted by giving the following command:
    - **comp01.getSectionalAreaCurve()** , where the first part is the name of the Hydro Computation and the second is the command necessary to extract the SAC from it, see also picture below.

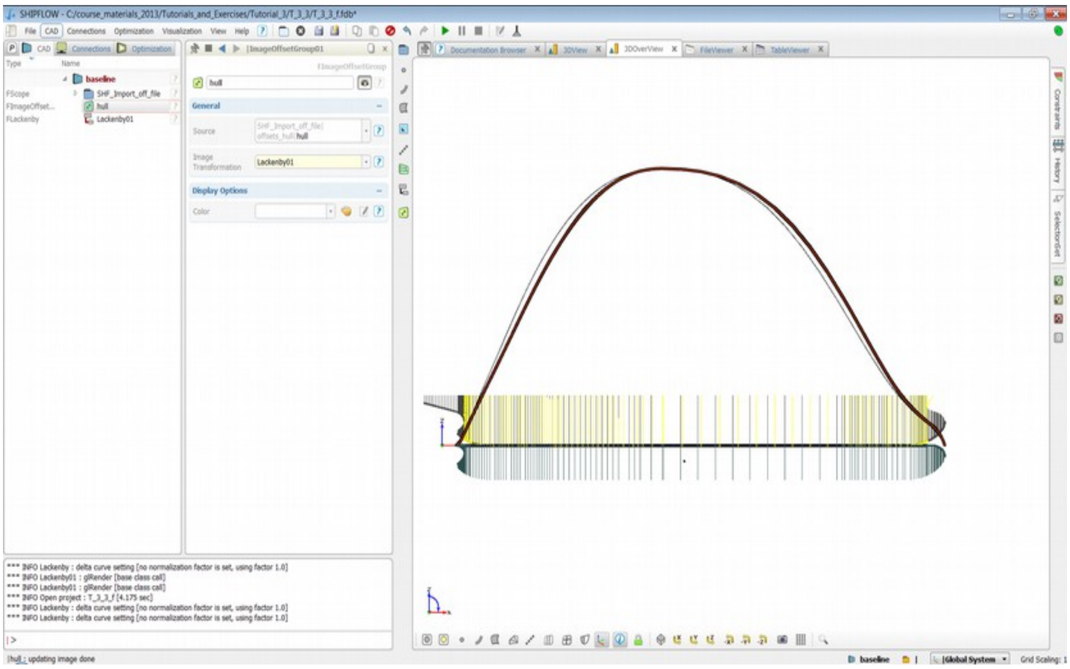


- Now the modification will be applied to the offset points and stored in a new offset group called Image Offset Group.

- Create the Image Offset Group selecting from the menu **CAD > Offsets > Image Offset Group.**



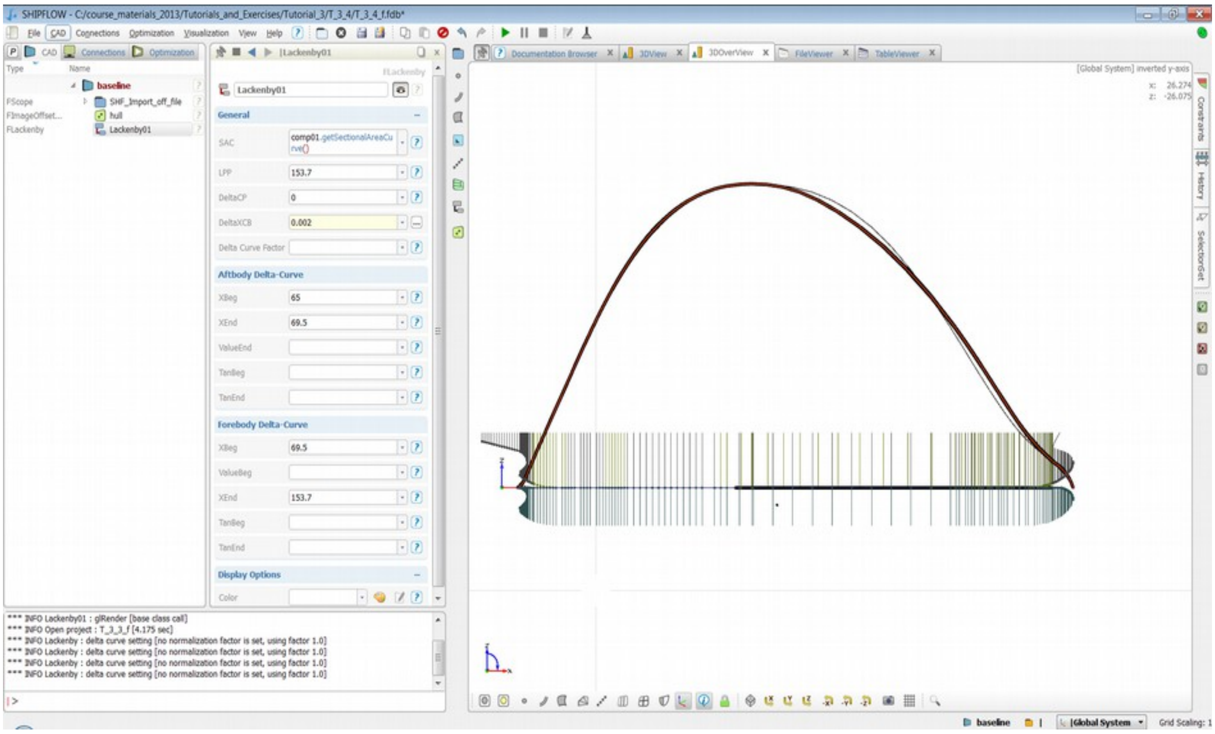
- Rename the newly created ImageOffsetGroup to "hull" (this name should correspond to the group in the original offset file that we will modify) and set the parameters as follows:
  - As the source take the "hull" offset group from the original offset file.
  - ImageTransformation set to the Lackenby01. (The Lackenby object created above.)
- Now you can observe in the 3D Overview that there is a new set of offset points on the main part of the hull that will react on the value of DeltaXCB in the Lackenby object.



Tutorial 3 part 4 – Optimization of the forebody

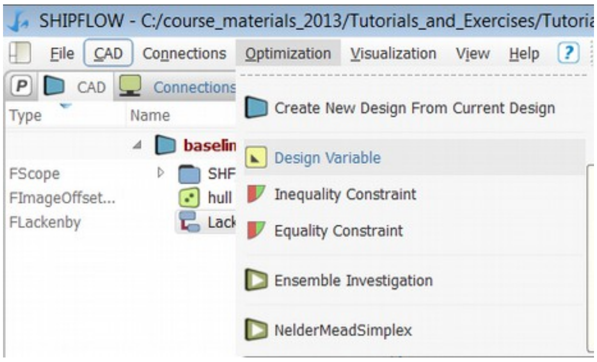
In this part of the tutorial an optimization of the forebody with the Lackenby shift will be set up

- Continue the previous work or open the T_3_3_f.fdb file from ..**BasicTraining\Tutorial_3\T_3_3**\. Save the project as Tutorial_3_4.fdb.
- Since the forebody only will be taken into consideration during the optimization the Lackenby shift will be restricted to that part. The following changes have to be made compared to the setup from the previous task:
  - DeltaXCB = 0.002
  - Xbeg = 65.0 for the Aftbody
  - Xend = 69.5 for the Aftbody
  - TanBeg and TanEnd set to 0 for the Aftbody

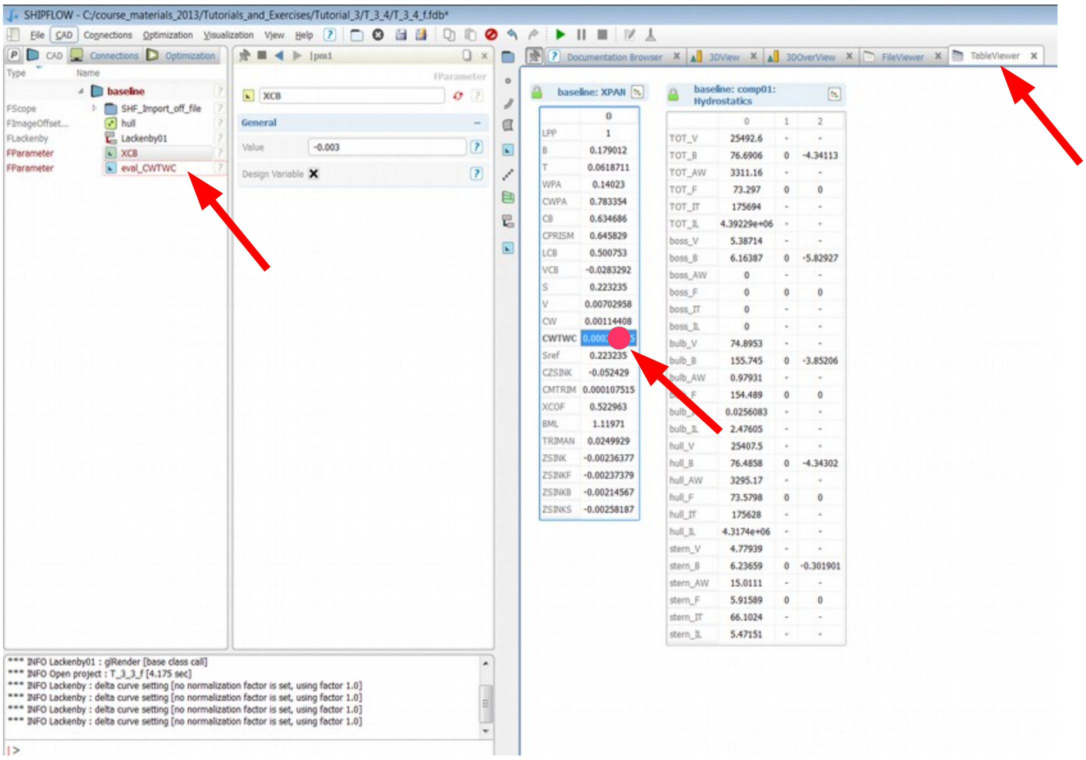


- For the optimization process a design variable and parameter will be necessary to control the process.

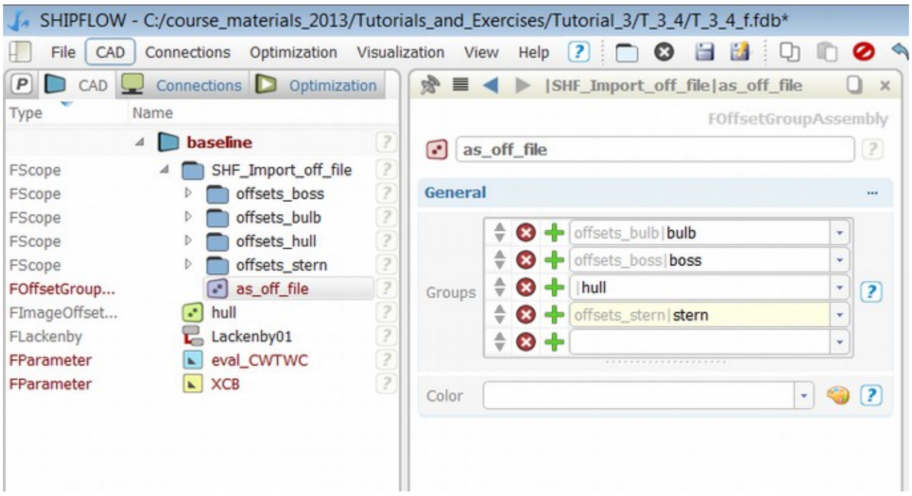
- To create a design variable select from the menu **Optimization > Design Variable**. Rename it to **XCB and set it to -0.003.**



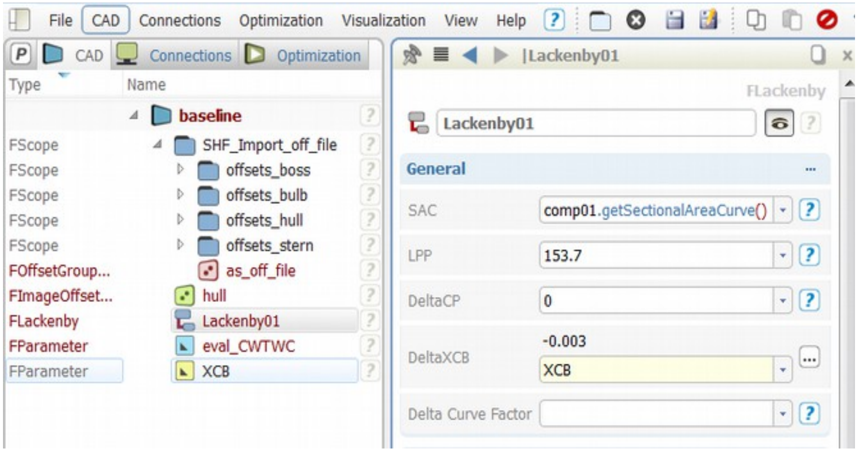
- Create a parameter by double clicking the CWTWC value in the Table viewer (If there is no the XPAN result table in Table Viewer the SHIPFLOW computations have to be performed).



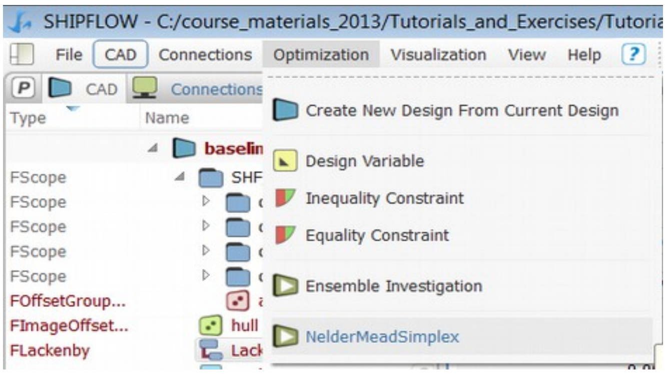
- In the offset group assembly the "hull" offset group has to be replaced with the image offset group created in the previous part of the tutorial.



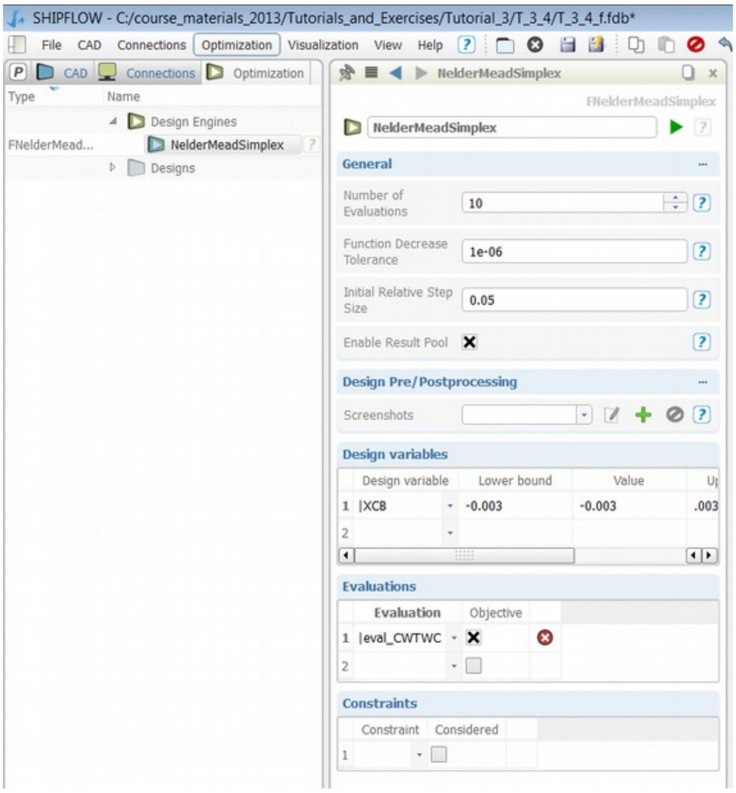
- The design variable now has to be inserted in the Lackenby as an input variable. Go to the Lackenby object in the Object Tree and edit the DeltaXCB. Set it to XCB variable.



- To optimize the hull Simplex method will be used. To create engine for this optimization select from the menu **Optimization > NelderMeadSimplex**

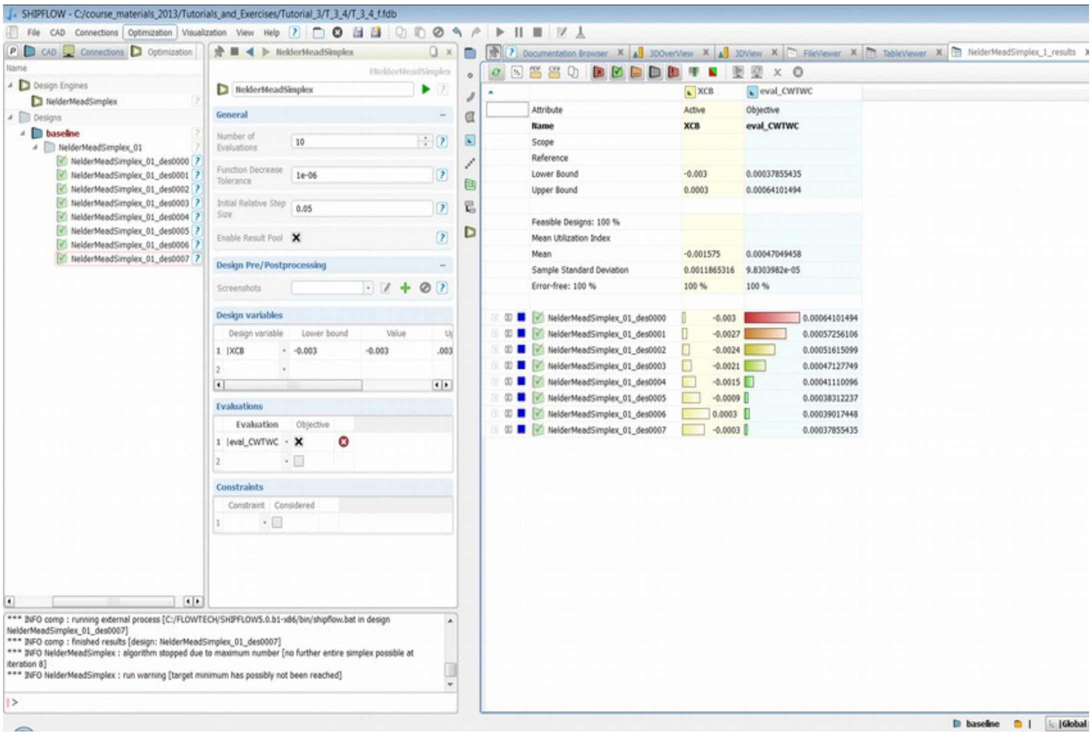


- The Design Engine now has to configured in the following way:
  - Set number of evaluations to 10
  - Design Variable: choose XCB and set lower bound to -0.003 and the upper bound to 0.003
  - Evaluations: chose CWTWC parameter.
  - The rest of parameters is left as default for this exercise but more advanced configurations are encouraged to e.g. speed up the optimization process.

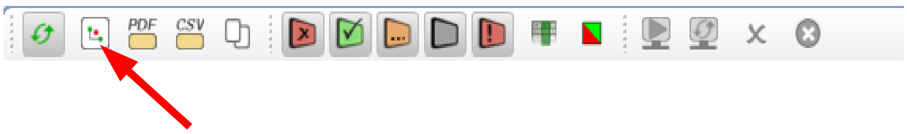


- Now you can run the optimization by clicking on the ”play” icon

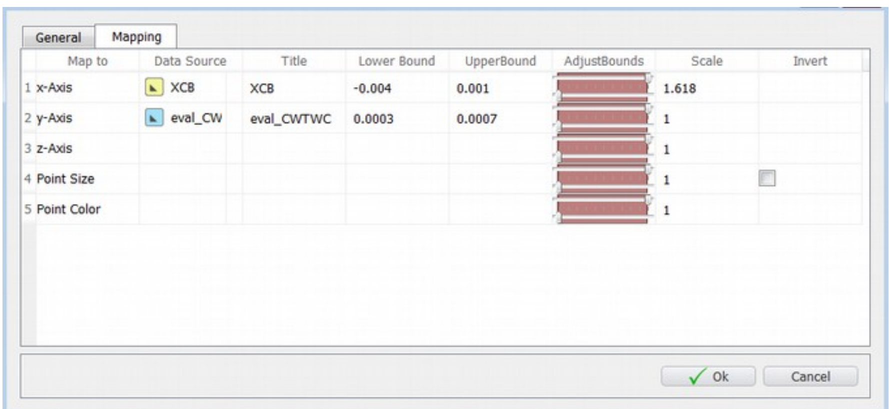
- When the optimization is finished a tab with the results of the NelderMeadSimplex process is available.

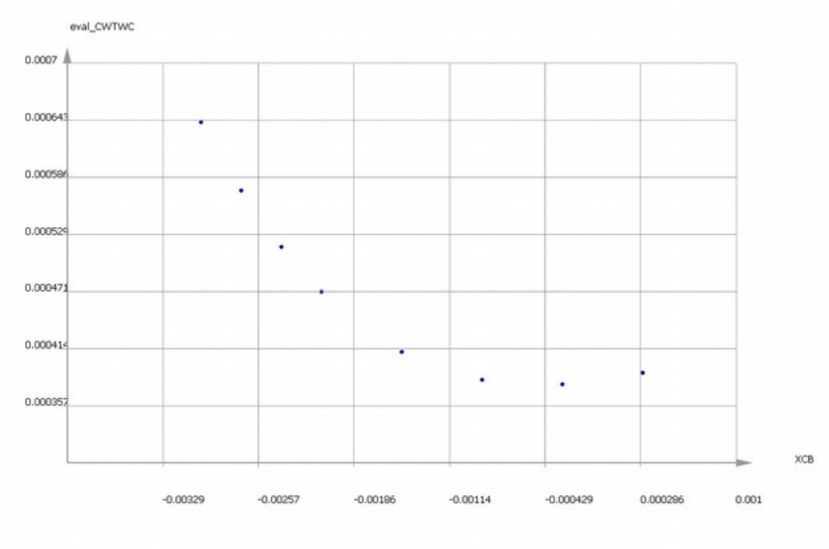


- You can see the Design Variable and also evaluated parameter values, click on the name at the top of the table will sort the values.
- To create a diagram showing the results click on the new diagram icon – second from the left in the tab with the results table

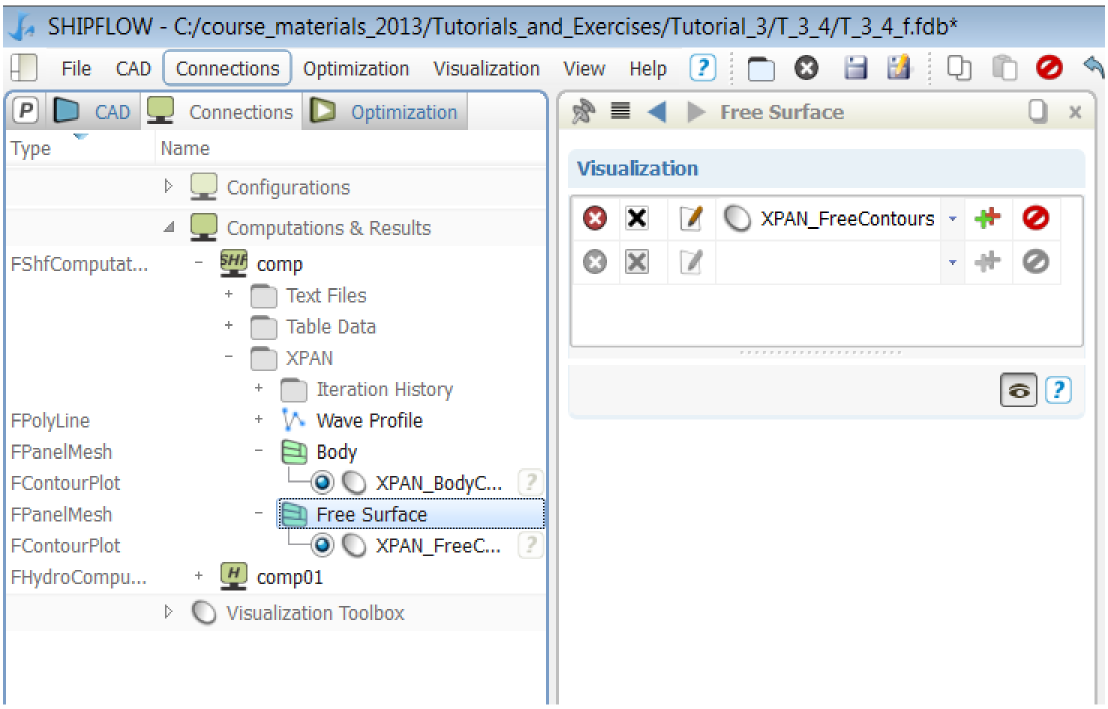


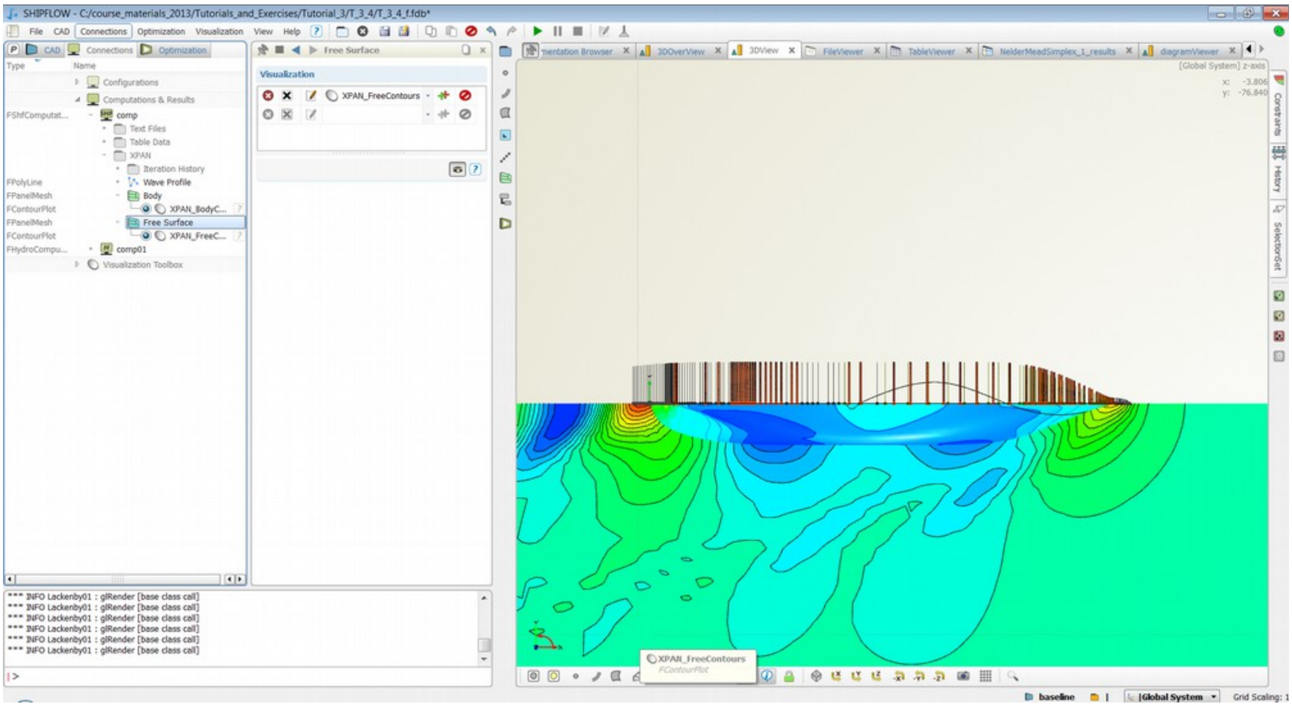
- and then in the mapping tab select the XCB and CWTWC for x and y axes respectively. After confirming with OK a new tab with the diagram will appear.



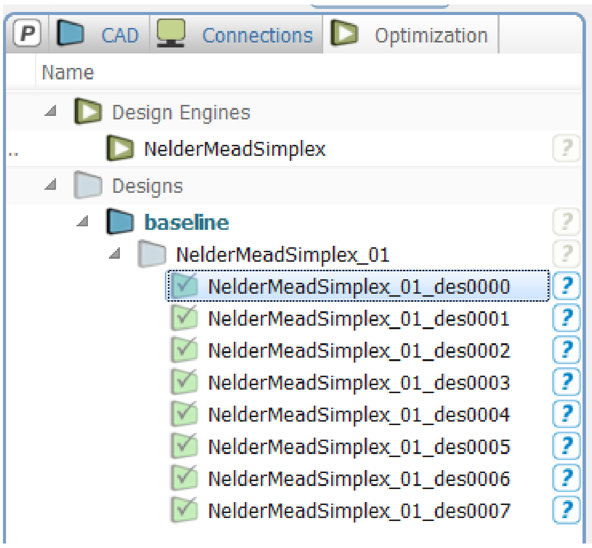


- If there is no wave pattern visible in either 3DView or 3DOverview select from the **Object tree Tab Connections | Computation & Results | comp | XPAN** and switch visibility option for Body and Free Surface objects by clicking on their icons in the tree. Then go to 3DView window and rotate the view to see the pattern from above ”Z”.

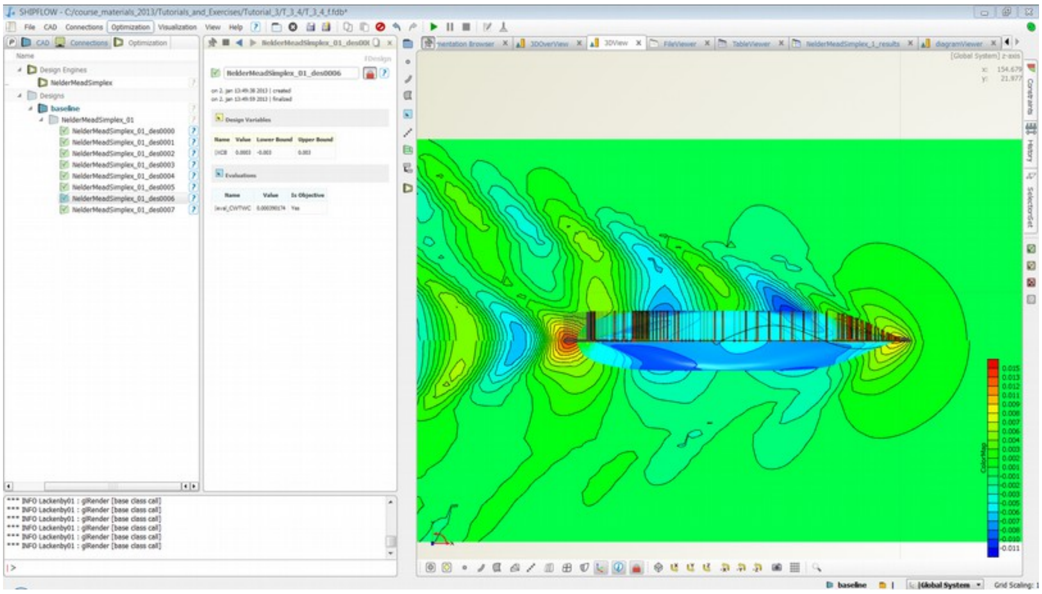




- To check the wave pattern for different design variants go to the **Object Tree Tab Optimization** and select first variant of the **Designs | baseline | NelderMeadSimplex** optimization object. Set the CFD Result to be displayed in 3DView

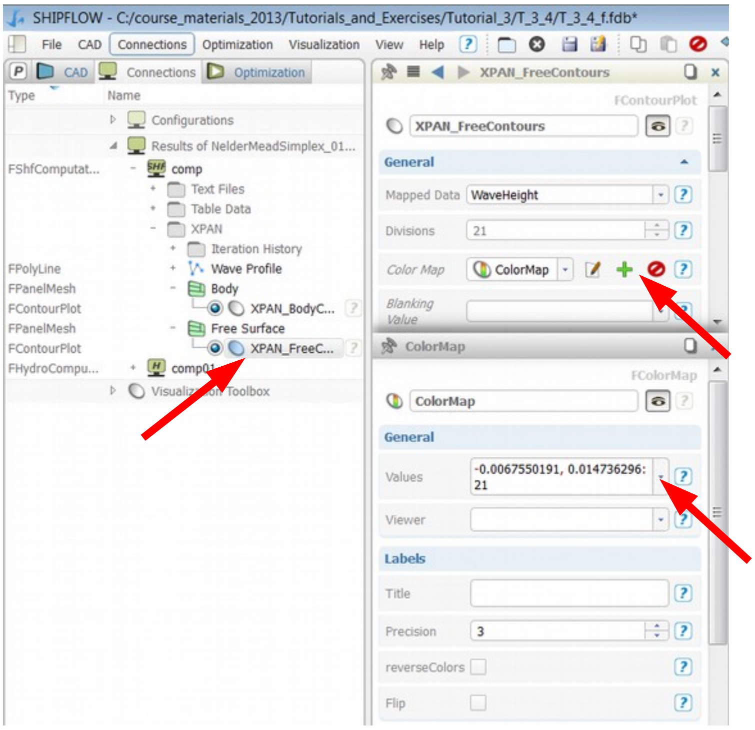


- Now lock the view clicking on the small lock icon at the bottom of the screen, now the mirror image appeared and was locked
- To compare different variants simply select other variant in the Designs tree.



Important note:

By default the contour plot range is set dynamically for all variants independently and may lead to difficulties in interpretation of the results. In order to use the same contour plot range a custom color map has to be created manually. For example in Connections | Results (…) | comp | XPAN | Free Surface | XPAN_FreeContours find a Color Map and click “+” icon to add a new color map, then the range and number of countours can be easily adjusted.



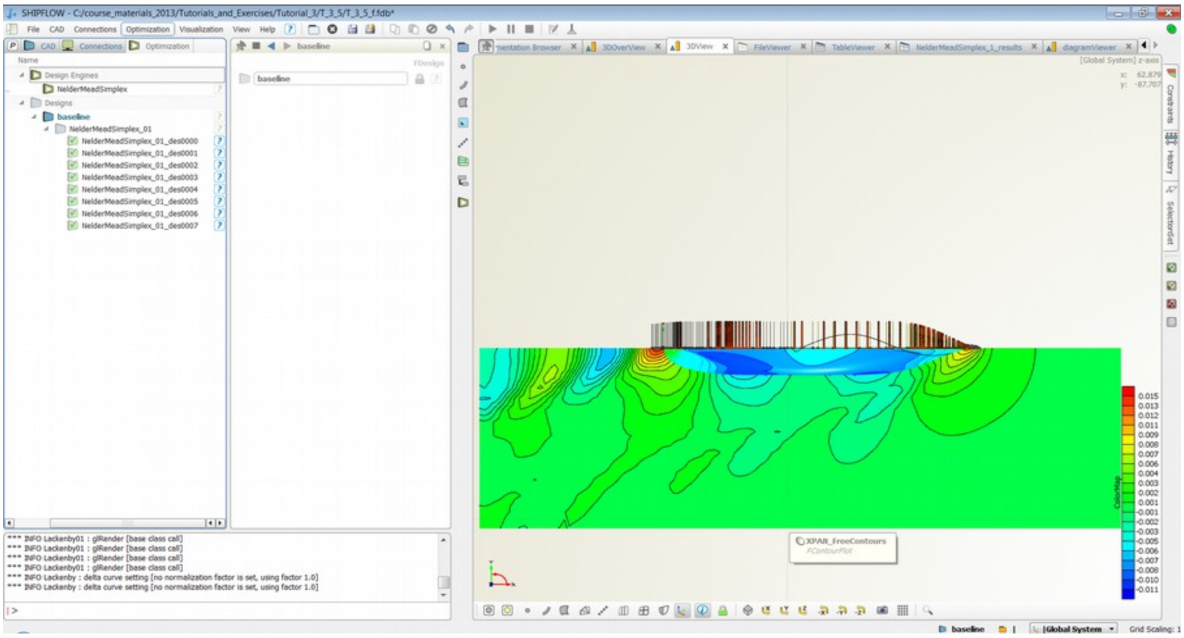
- Save the project

Tutorial 3 part 5 – Creating manual variants of the design

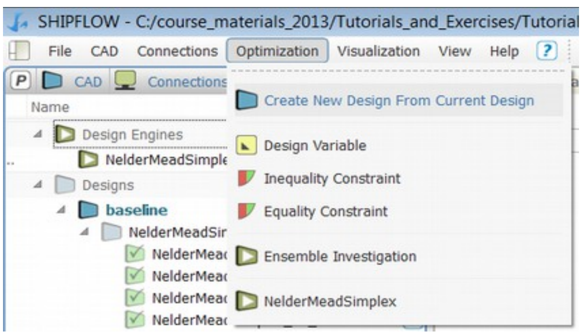
In this part of the tutorial a new variant will be created and an optimization of the forebody with the Lackenby shift will be carried out for a different ship speed

- Continue the previous work or open the T_3_4_f.fdb file from **..\BasicTraining\Tutorial_3\T_3_4**. Save the project as Tutorial_3_5.fdb.

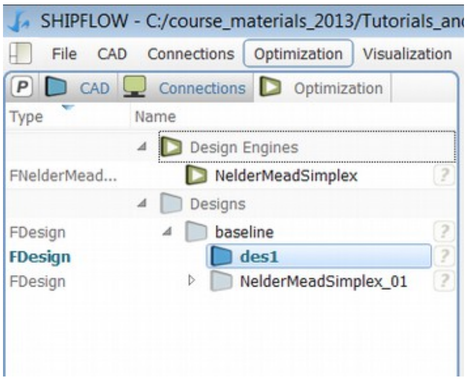
- First make sure you are working with the baseline variant by double clicking on the baseline in the Designs in the Object Tree.
- To verify that you are in the baseline variant check the status line at the very bottom of the program window. Current design should be baseline.



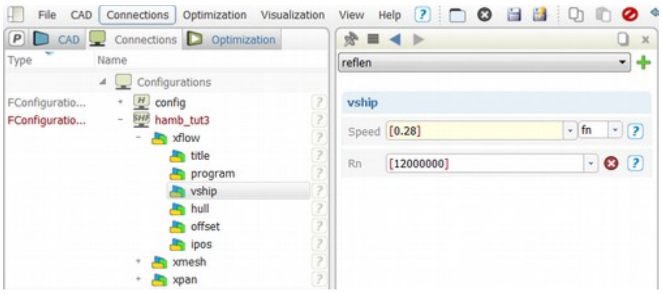
- Create a new variant from the baseline by selecting from the menu **Optimization > Create New Design From Current Design**

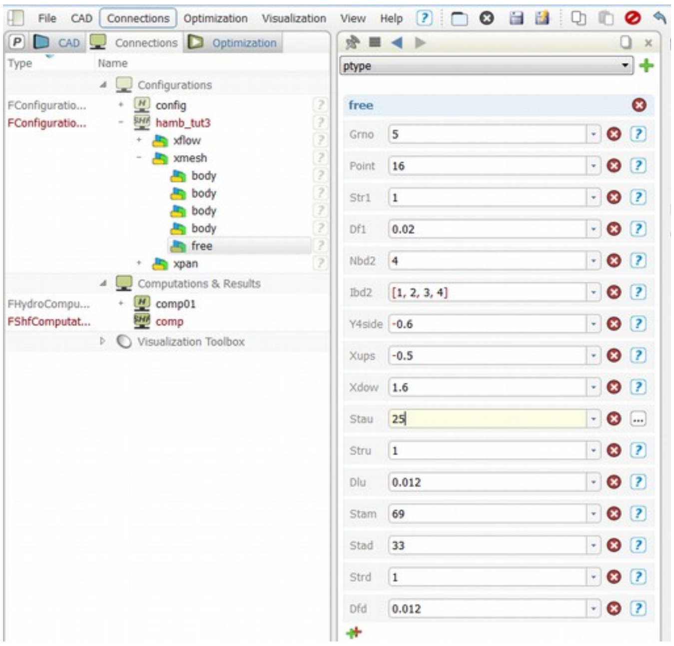


- A new variant should appear in the Object Tree, here with a default name des1
- This newly created variant became a current one automatically. Now we can edit the configuration without affecting other variants.

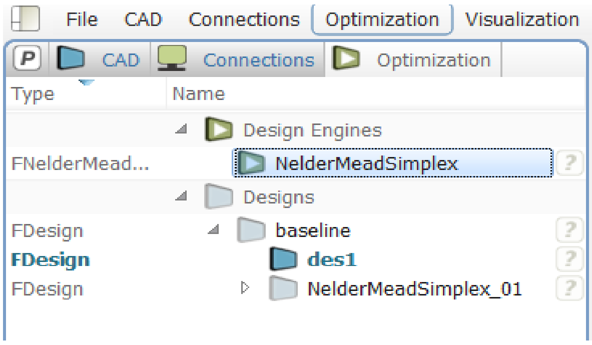


- As an example we will modify the speed and adjust the free surface panellization.
- In the Object Editor set the Fn to 0.28 and decrease number of panels on the free surface in the longitudinal direction as follows: Stad=33, Stam=69, Stau=25.
- Save the project

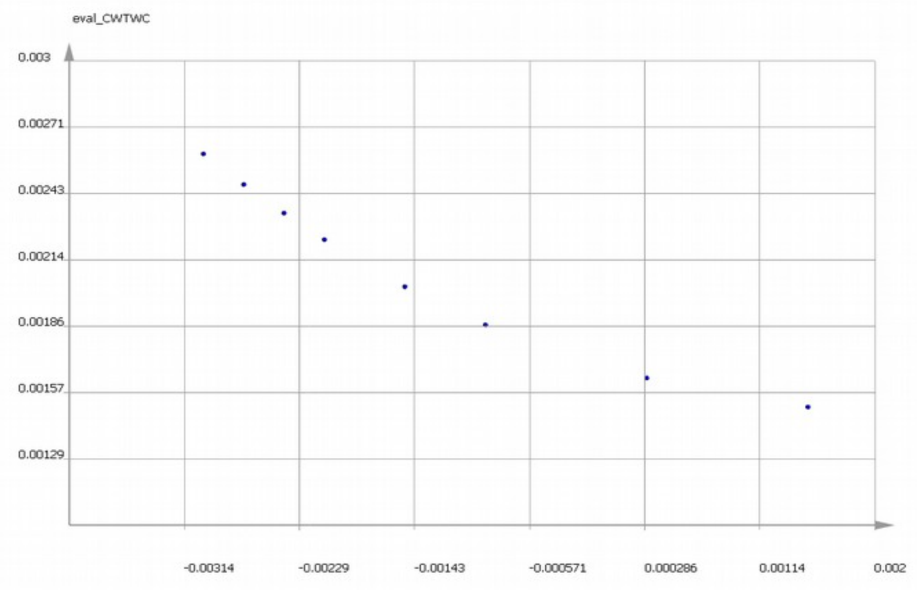




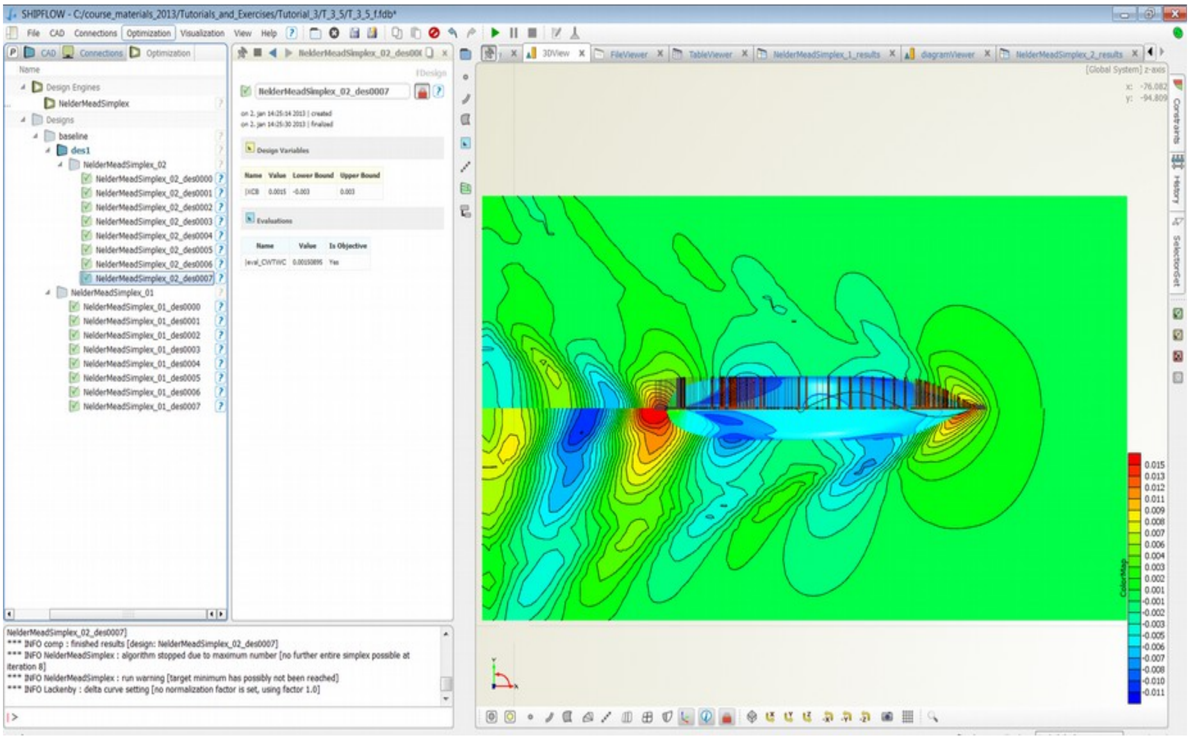
- Go back to the Design engine in the Object tree and run the optimization process again



- When the optimization is finished display the diagram with the results and compare it with the results of the previous optimization, notice that the optimum now is different due to the Fn difference and some designs were out of the bounds (red markers)



- You can also find the best designs for both speed and compare the resulting free surface pattern in the 3D View.

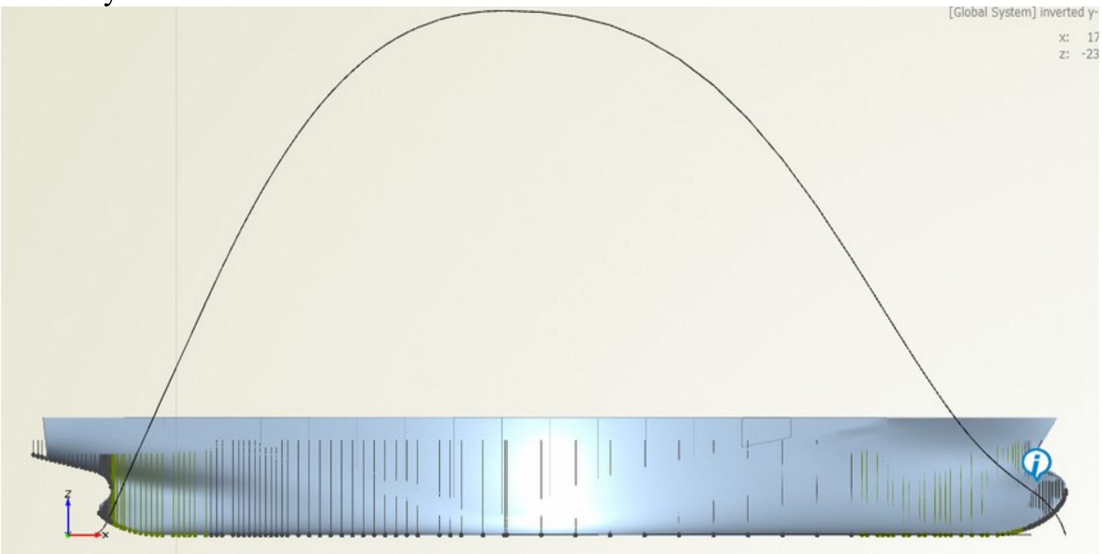


- Save the project and
- Good luck with your own designs!

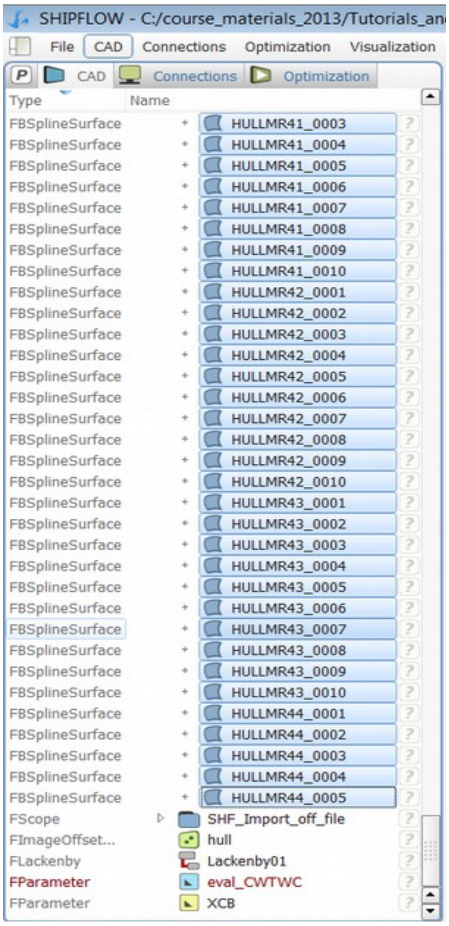
Tutorial 3 part 6 – Export of optimized geometry to IGES

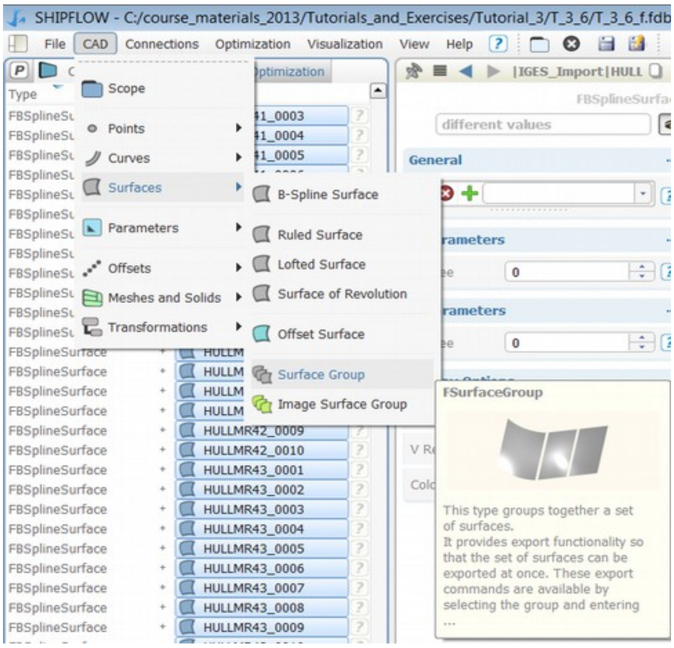
- Continue from the Tutorial 3 part 4 or part 5.

- Import the IGES file *t3_6_surface_model.igs* from the **..\BasicTraining\Tutorial_3\T_3_6** directory.

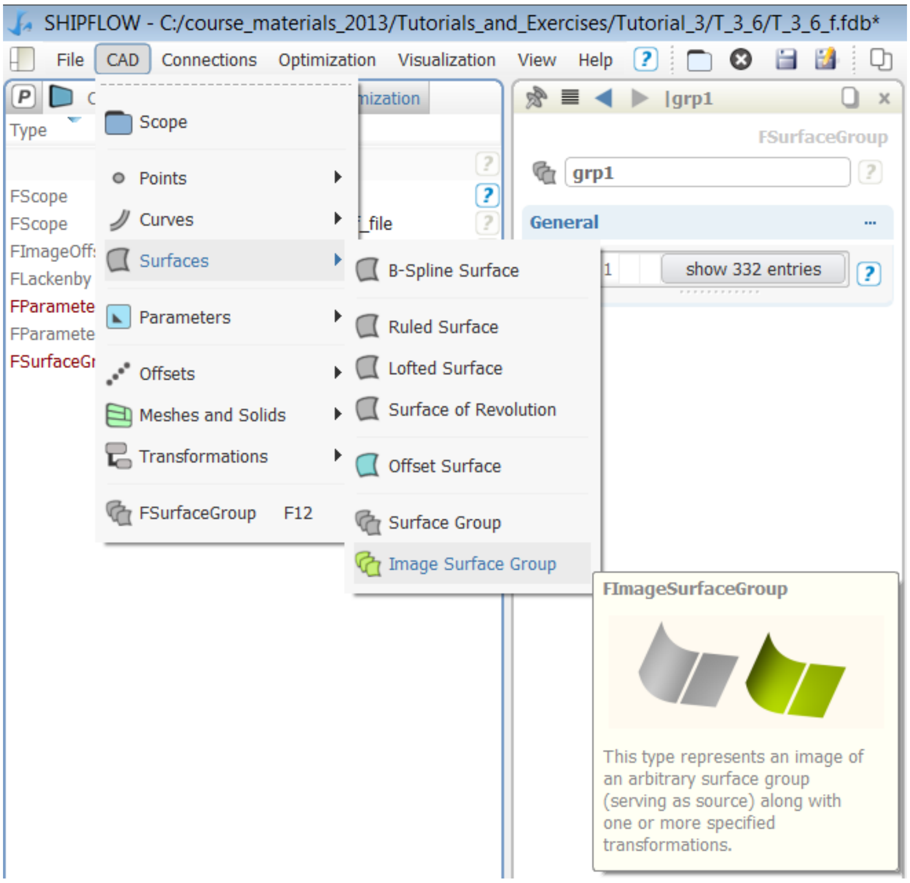


- Select all the imported surfaces and thereafter create a surface group

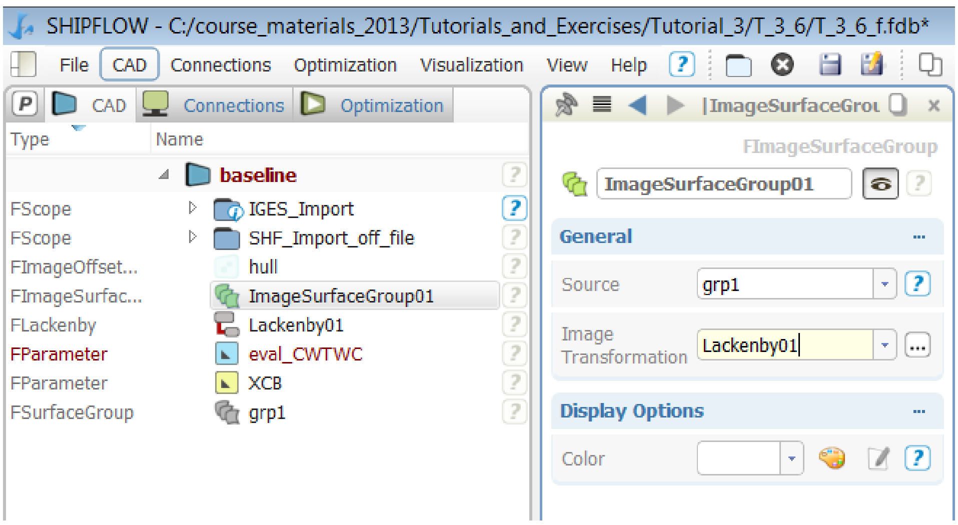




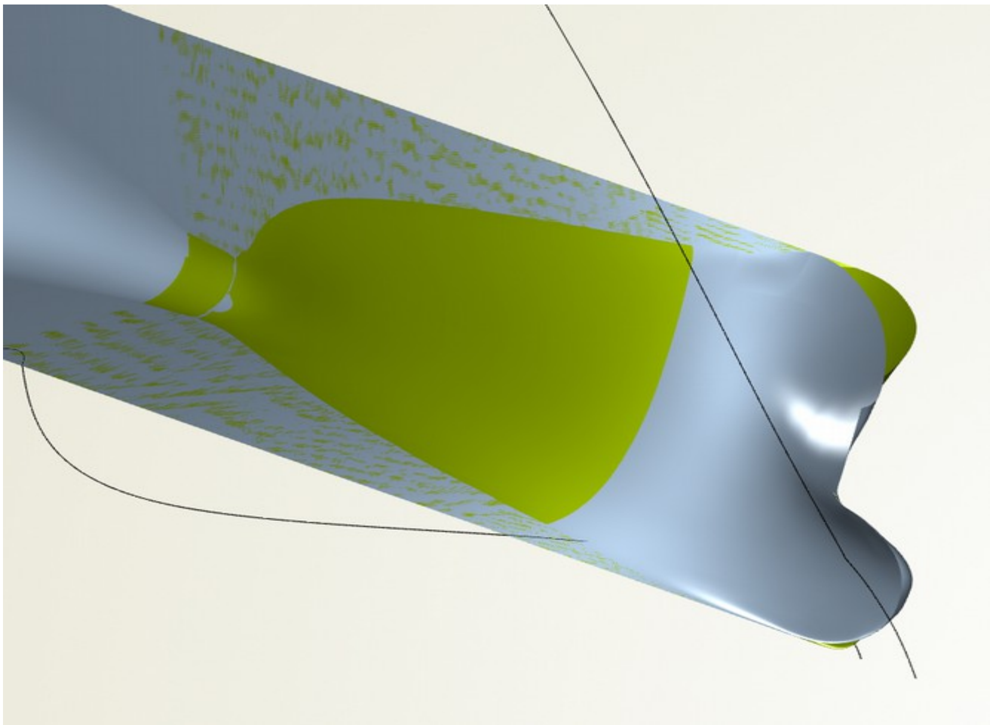
- Select the newly created surface group and create a Image Surface Group



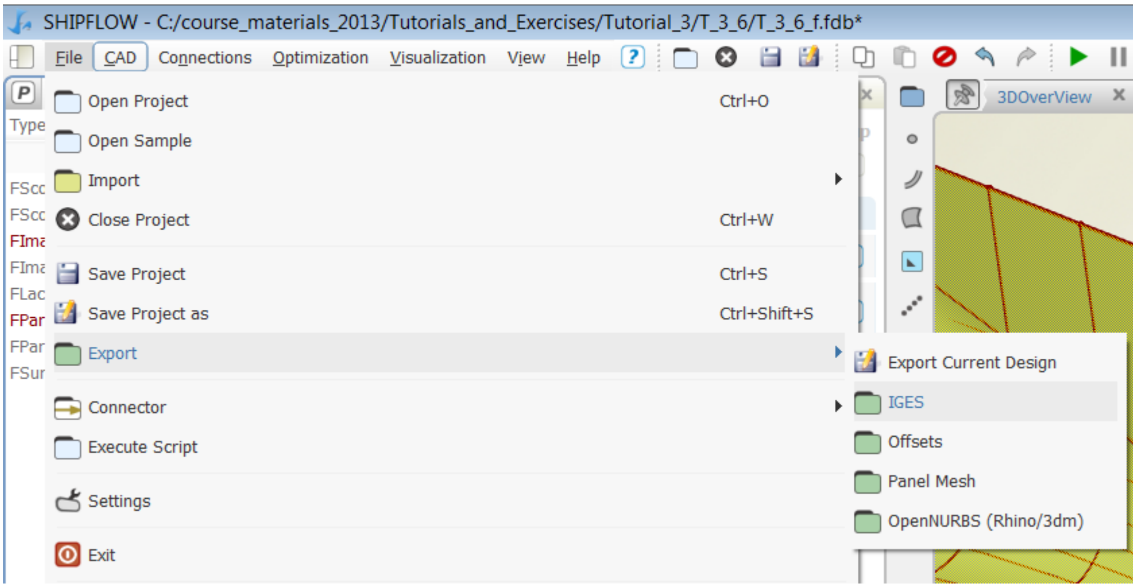
- Edit the Image Surface Group object such that the Image transformation is set to your Lackenby shift



• Now if the XCB parameter is modified the hull surface will change according to the Lackenby transformation. So set it to the value which was the best in the optimization task and observe the new hull surface.

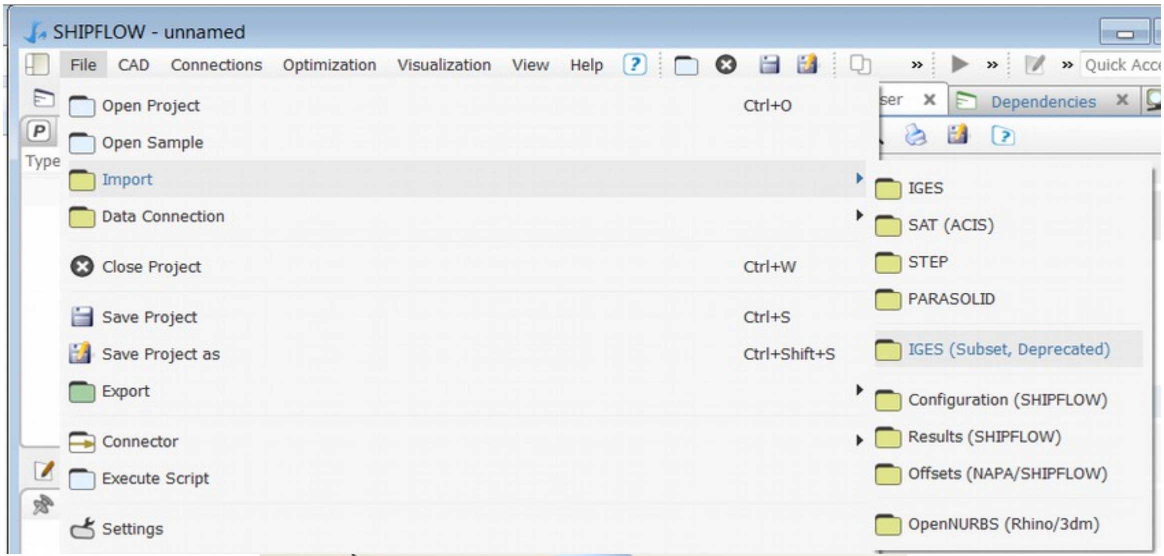


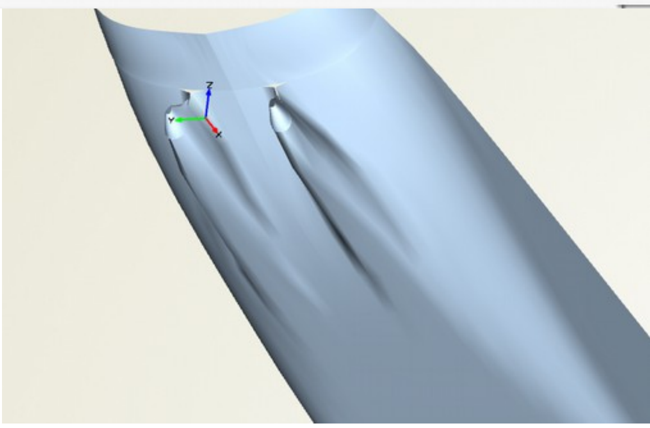
• In order to export the new hull select the Image Surface Group object and in the drop down menu select File | Export | IGES.



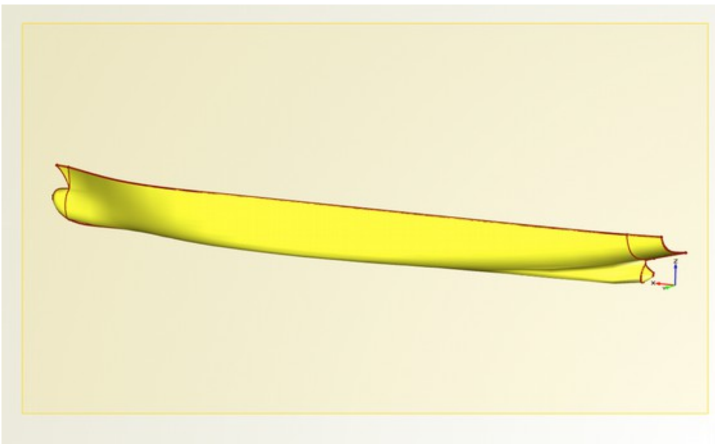
Tutorial 4 part 1 – Twin-skeg example – IGES

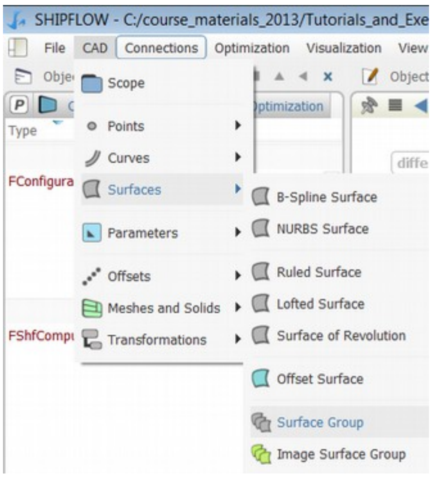
- Import twinskeg.igs file from Tutorial_4 folder and save the project



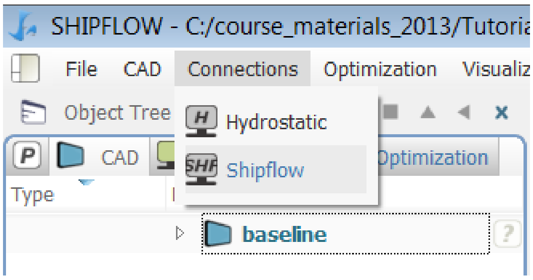


- Select all surfaces and create surface group

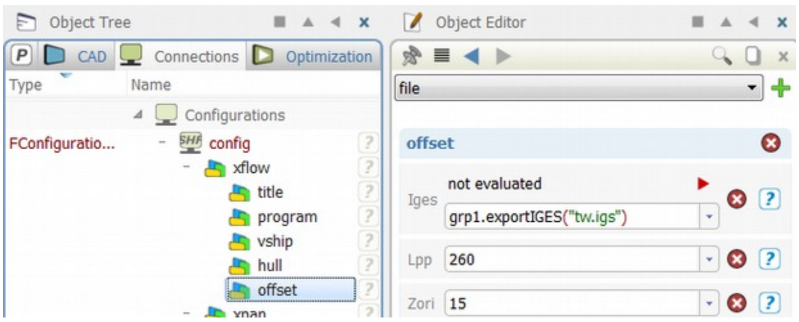


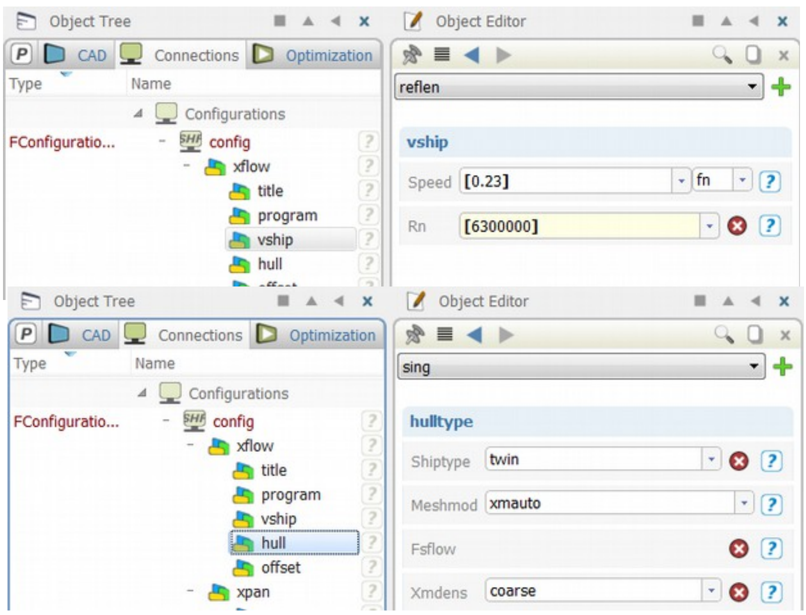


- Create SHIPFLOW configuration from MENU > Connections > Shipflow.

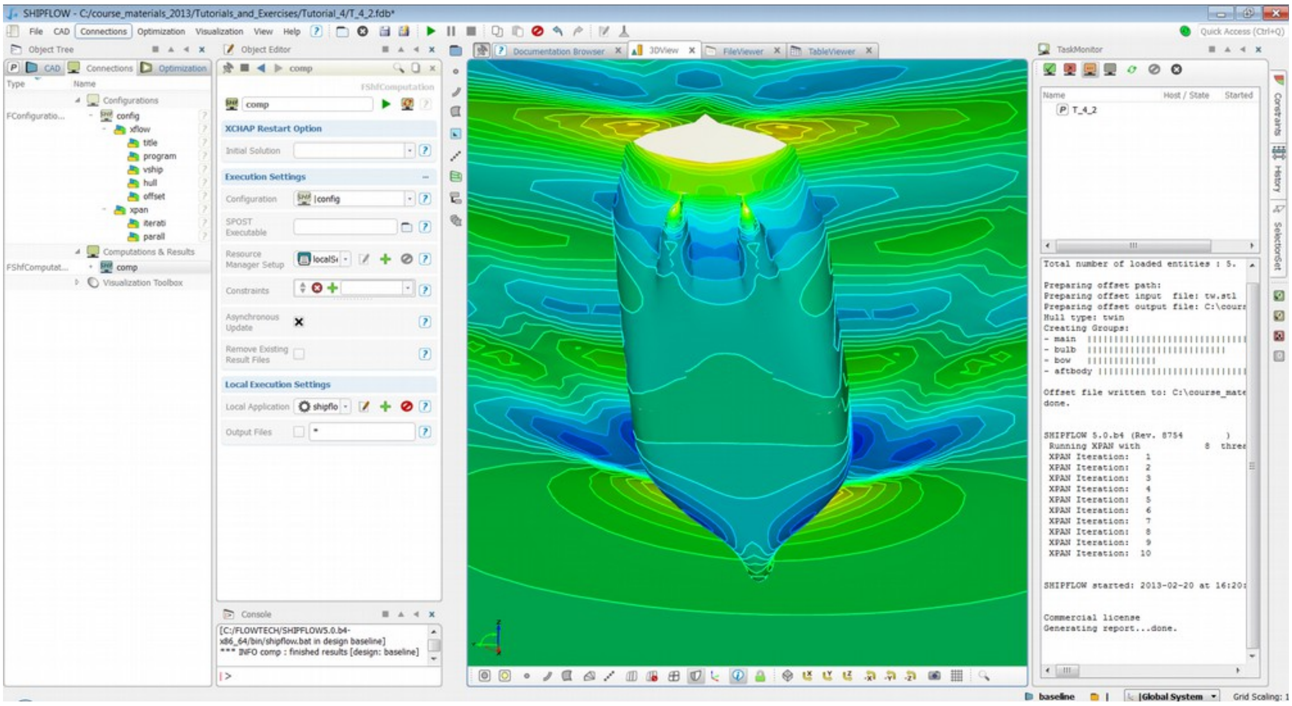


- Next configure the SHIPFLOW setup using the following data:
  - Lpp = 260
  - T = 15
  - iges file as a geometry input (use the created surface group)
  - Fn = 0.23
  - Rn = 6.3e6
  - coarse mesh
  - xpan calculations only

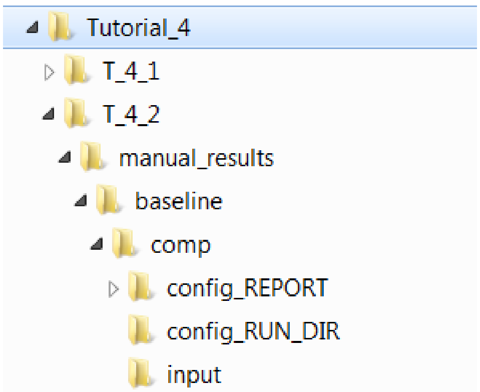




- Run SHIPFLOW. The program should start the IGES to offset converter first, then compute the flow around the hull.



- During this run SHIPFLOW first translated the IGES file to an STL file and thereafter created an offset file that was submitted together with the configuration to the solver. The process is automatic but it is good to be familiar with the process.
- The directory structure is as follows:



  - in this example the project file called T_4_2.fdb is located in Tutorial_4 folder
  - if default names were used for the variant and configuration in the project the configuration file together with the offset file will be located in folder called comp.
  - There SHIPFLOW solver is executed and another subfolder named config_RUN_DIR is created where temporary solutions are stored.
  - In the config_RUN_DIR one can find a config file that contains some additional informations. The most important for the user is the information about groups that are in the offset file.

- The picture below shows the initial configuration as prepared by the GUI:

```
xflow
   title( title = "SHIPFLOW" )
   program( xpan )
   vship( fn = [0.23], rn = [6300000] )
   hull( twin, xmauto, fsflow, coarse )
   offset( iges = "tw.igs", lpp = 260, zori = 15 )
end

xpan
   iterati( maxit = 20 )
   parall( nthread = 8 )
end
```

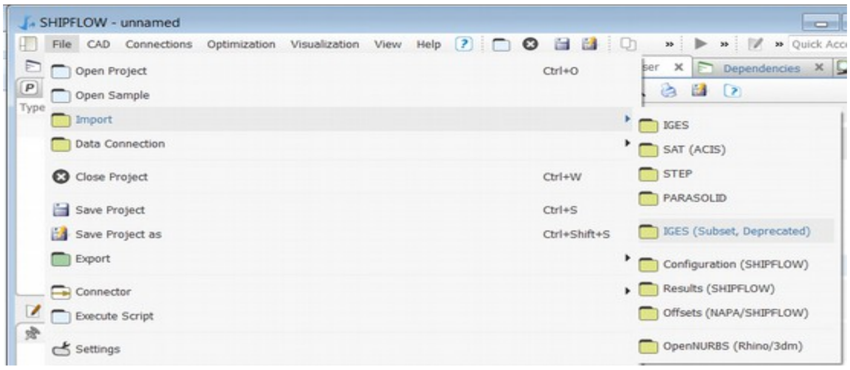
- The next picture shows the final configuration used in the calculations that includes the offset group information.

```
xflo
  titl(titl="SHIPFLOW")
  prog(xpan)
  vshi(fn=[0.23],rn=[6.3e+006])
  hull(twin,xmau,fsfl,coar,
       h1go="f_main",h2go="o_main",h2gi="i_main",fbgo="bulb",
       abgo="o_boss",abgi="i_boss",ogro="aft")
  offs(lpp=260,zori=15,ysig=1, xaxd=-1,xori=260,file="../off_tw")
  control(exepath="C:\FLOWTECH\SHIPFLOW5.0.b4-x86_64\bin\..\\bin/",
          runid="config_RUN_DIR")
end
xpan
  iter(maxi=20)
  para(nthr=8)
end
xgri
  offs(h1go="o_main",h1gi="i_main",h1gr="f_g_main",fbgr="bulb",
       abgo="o_boss",abgi="i_boss",ogro="o_aft",ogri="i_aft")
end
```

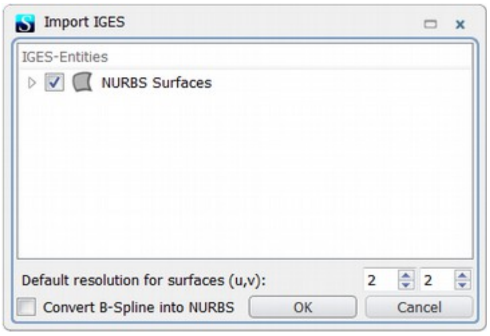
- Note that the offset groups are different in xflow and for xgrid modules, this is due to the fact that the xmesh and xgrid have different requirements regarding the offset station arrangements.
- After the initial run from an IGES file one can import the final configuration together with the offset file into the GUI to save time. This method will be used later in connection with xchap calculations.

Tutorial 5 part 1 – Manual offset generation

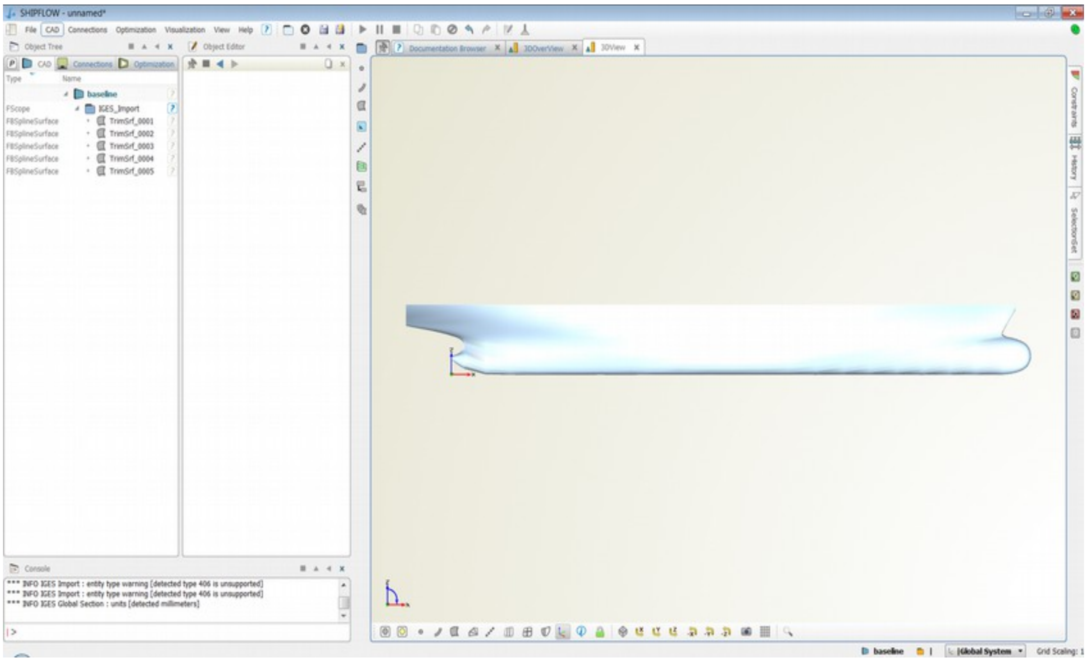
- Start new project and save it as **Tutorial_5_1**.
- Import (menu **File > Import > IGES (Subset)**)the **iges_version_A.igs** file from the directory: **..\BasicTraining\Tutorial_5\source**



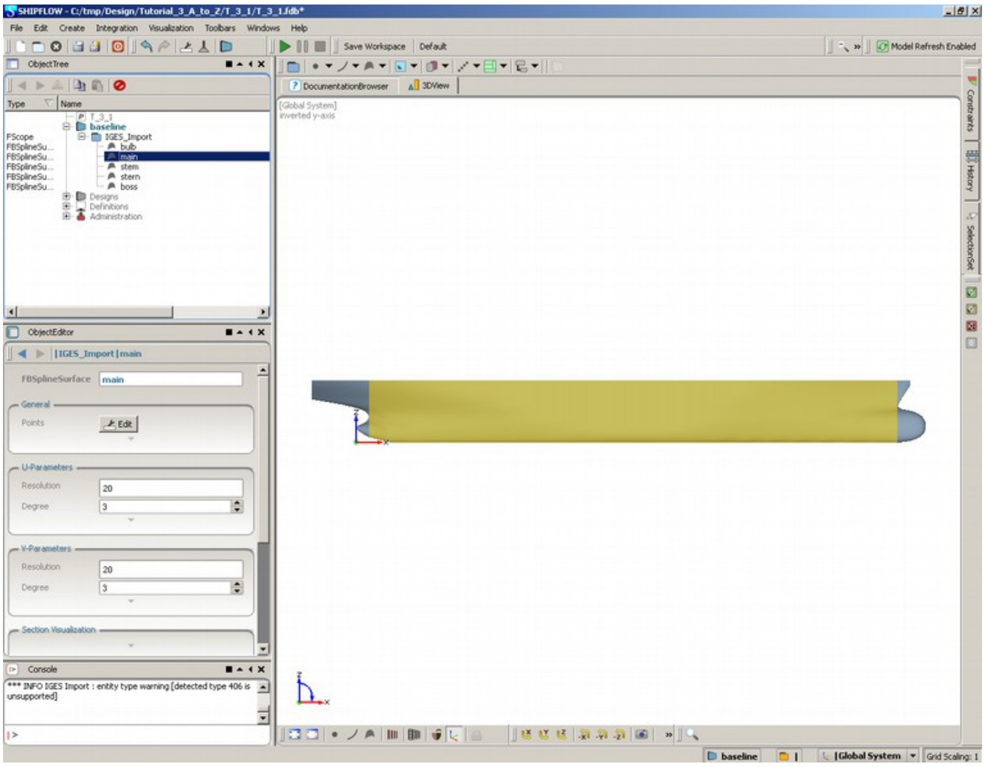
- Confirm the default settings of the imported iges.



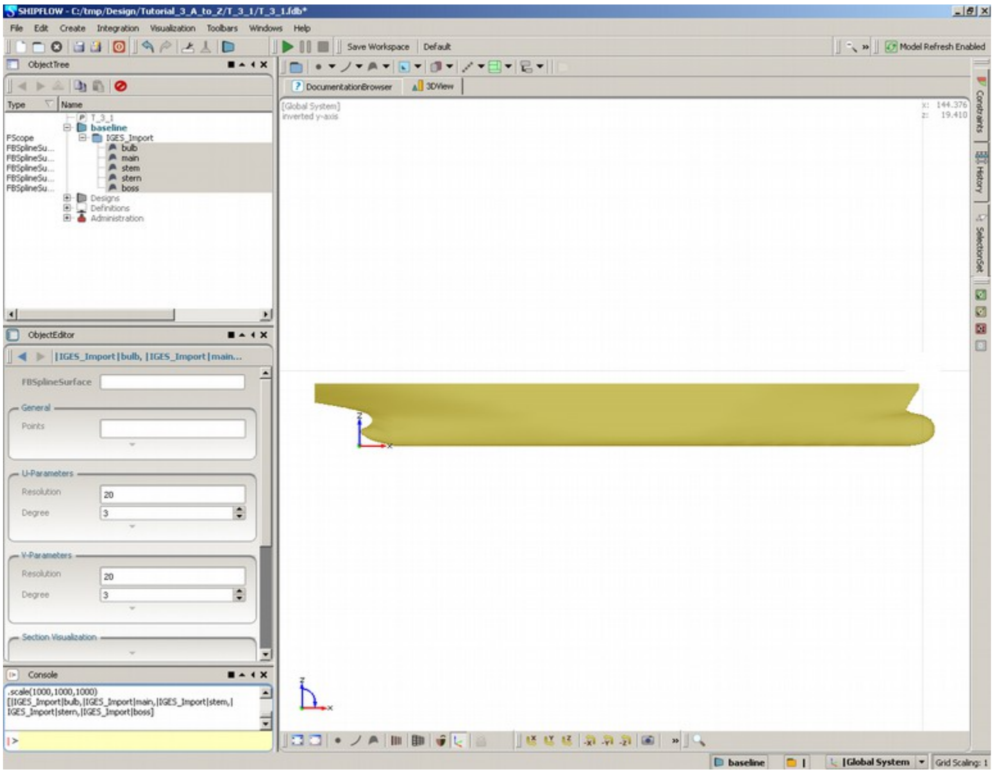
- Rotate the view to see the hull from the side ("Y") and extend the view, you should see the picture similar to that below.



- In the Object Tree select the BsplineSurface objects that are in the IGES_Import scope one by one and observe that different surface patches are being highlighted.
- Rename the patches using Object Editor using the following names:
    - bulb for forward bulb
    - stem for the bow overhang
    - main for the main part of the hull
    - boss for the boss
    - stern for stern overhang



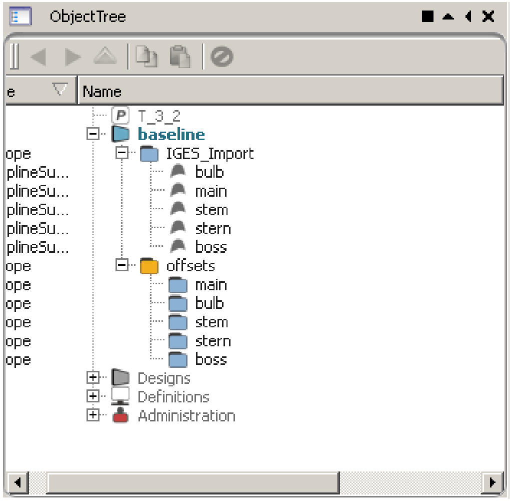
- Check the length of the hull with the coordinates display in the top right corner of the 3D View (available only in the orthogonal views), notice that due to the unit conversion the hull is 1000 times too small.
- Select all the surfaces in the tree or on the screen and scale them using the following command: **.scale(1000,1000,1000)**
- extend the view to see the whole ship



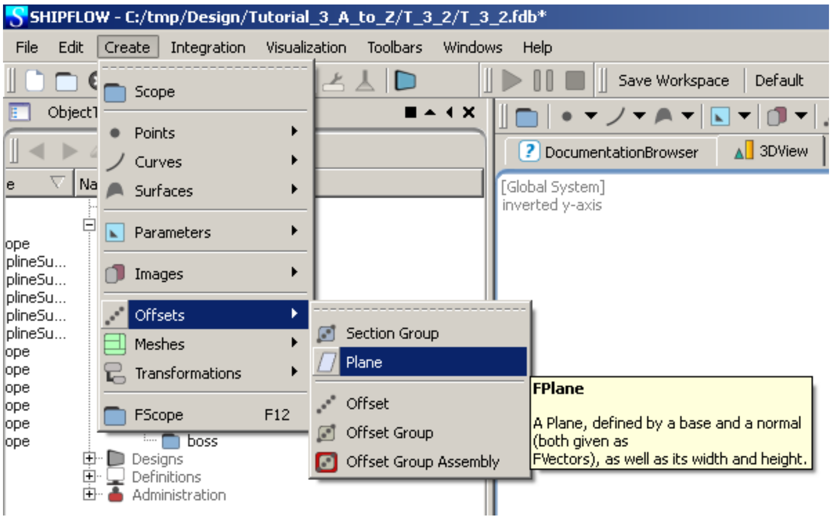
- Save your project

Tutorial 5 part 2 – Creating and exporting offset data

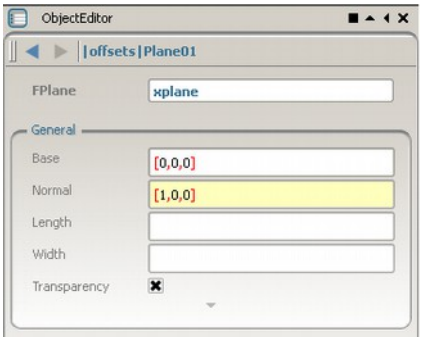
- Continue the previous work or open the **T_4_1_f.fdb** file from **..\BasicTraining\Tutorial_4\T_4_1**. Save the project as **Tutorial_4_2.fdb**.
- Create a scope (selecting from the menu **Create > Scope**) and rename it to **offsets**, then create scopes **bulb**, **boss**, **main**, **stem**, **stern** inside the offsets scope.



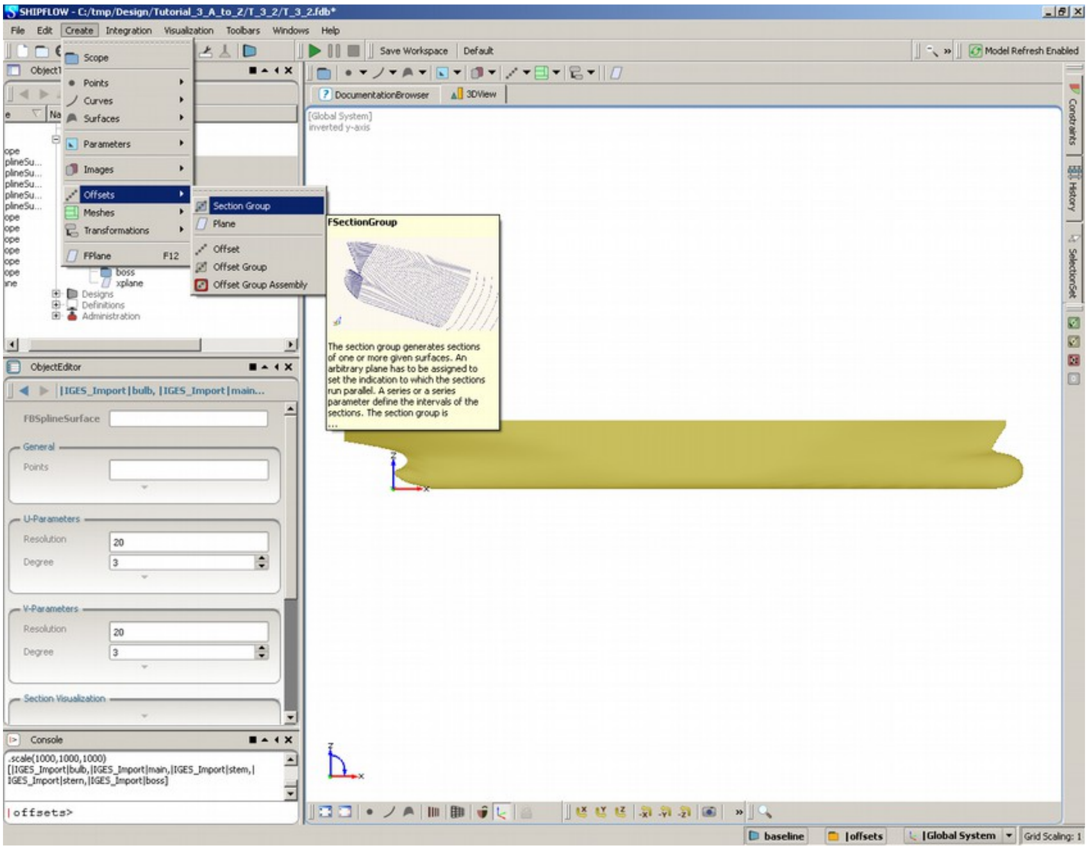
- Create a plane selecting from the menu **Create > Offsets > Plane**



- Give it a name **xplane** and check if the normal is pointing in x direction [1,0,0]. We will use it later as the cutting plane to create sections.

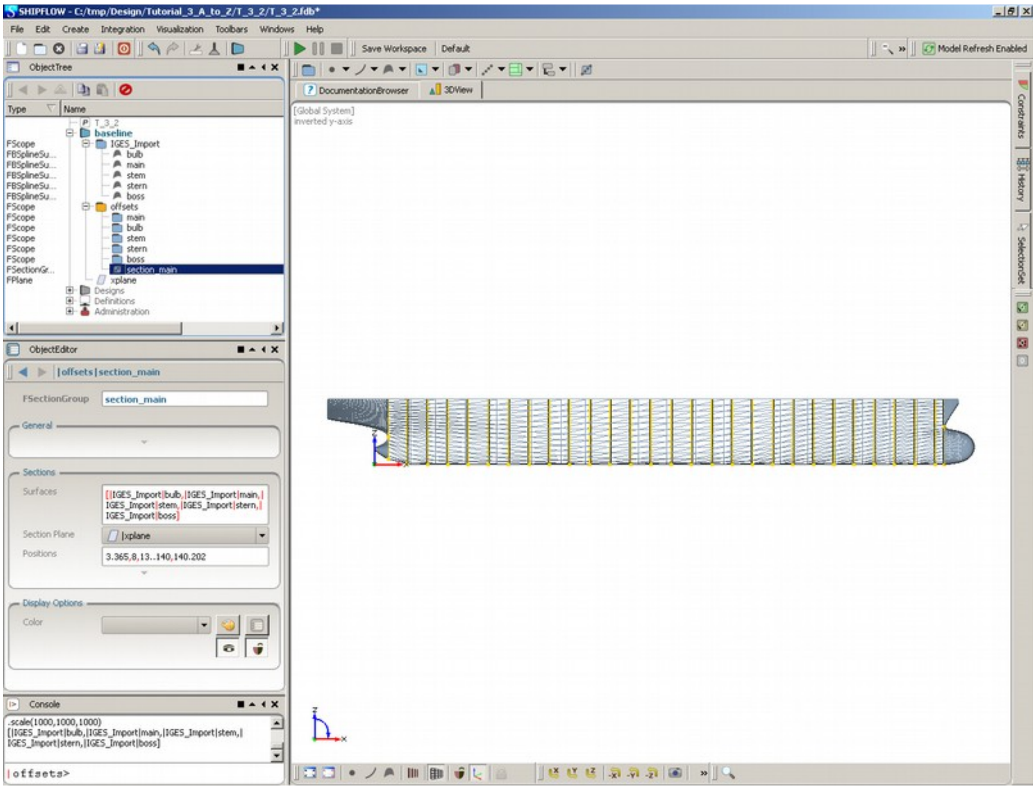


- Now we will create sections on the main part of the hull. Select all surfaces and create a Section group from the menu **Create > Offsets > Section Group**.

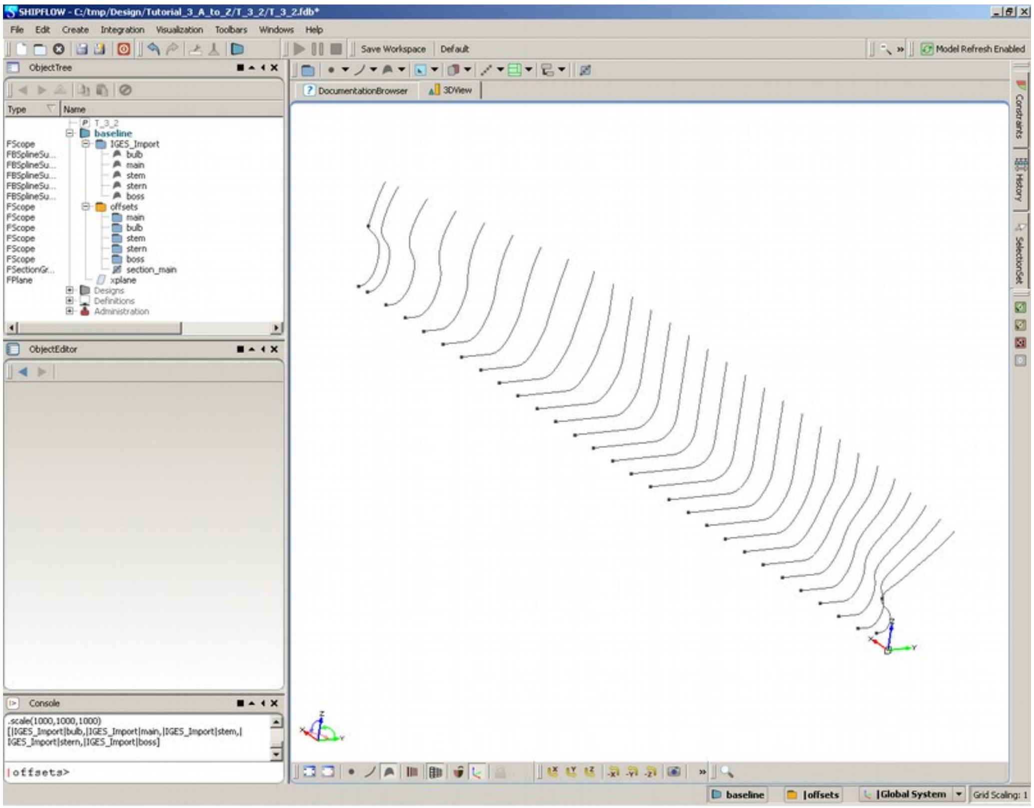


- Select newly created section group and using the Object Editor do the following steps:
  - Rename it to section_main.
  - Put it in the offsets scope.
  - Make sure that in the window with Surfaces you have all surfaces that belong to the hull.
  - Set the Section Plane to xplane.
  - Specify the position giving the x coordinates 3.365,8,13..140,140.202 ,the first and the last input lies just outside the main part and cuts the overhangs, boss and bulb, the second, third and fourth input is necessary to create a set of sections from 8 to 140 with step of 5. As you can see it is possible to give a specific positions separated by commas and also specify a range by giving first position comma second position double dot last position.

- For the time being leave the rest as default.

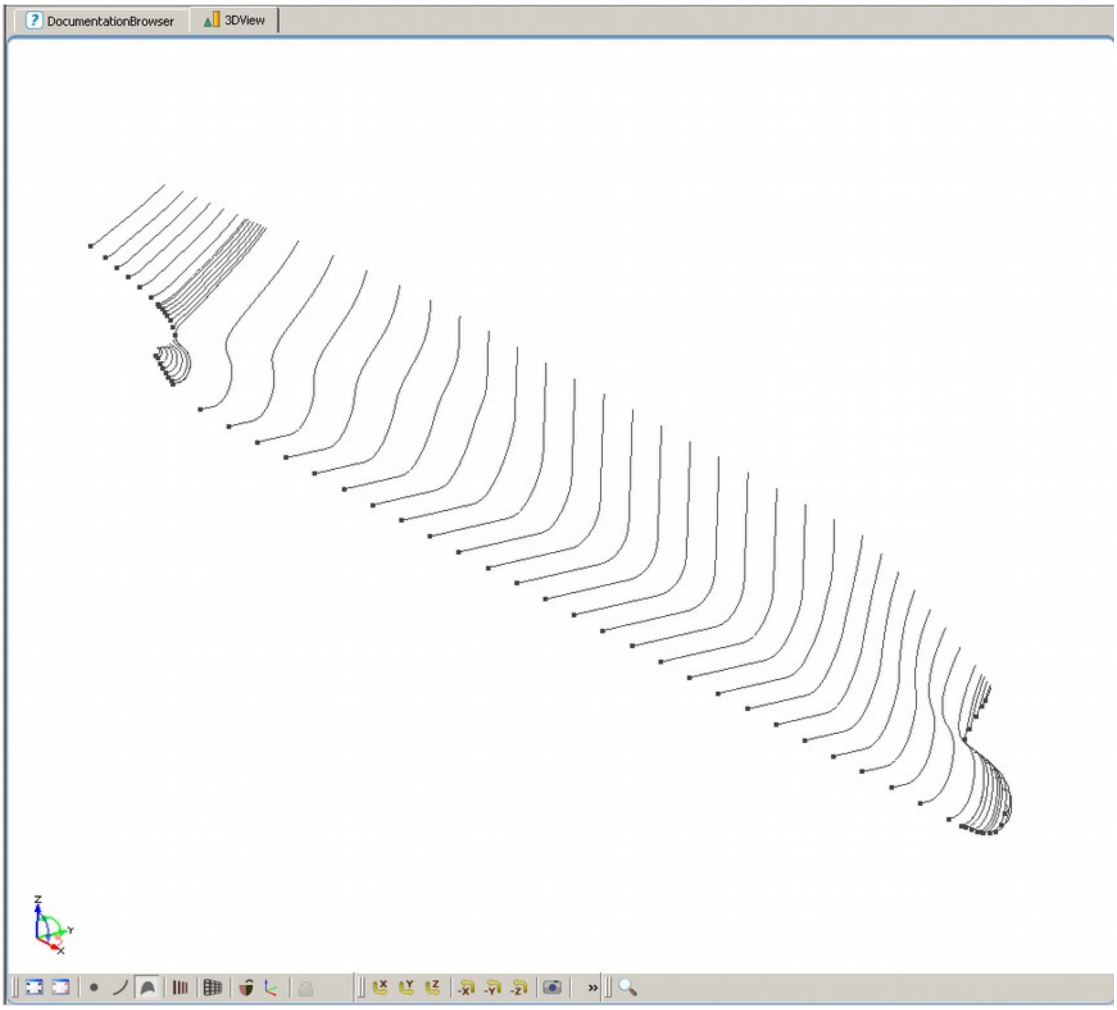


- After all the input is given the sections are created, switch of the surface visibility, rotate the view to look at the sections.

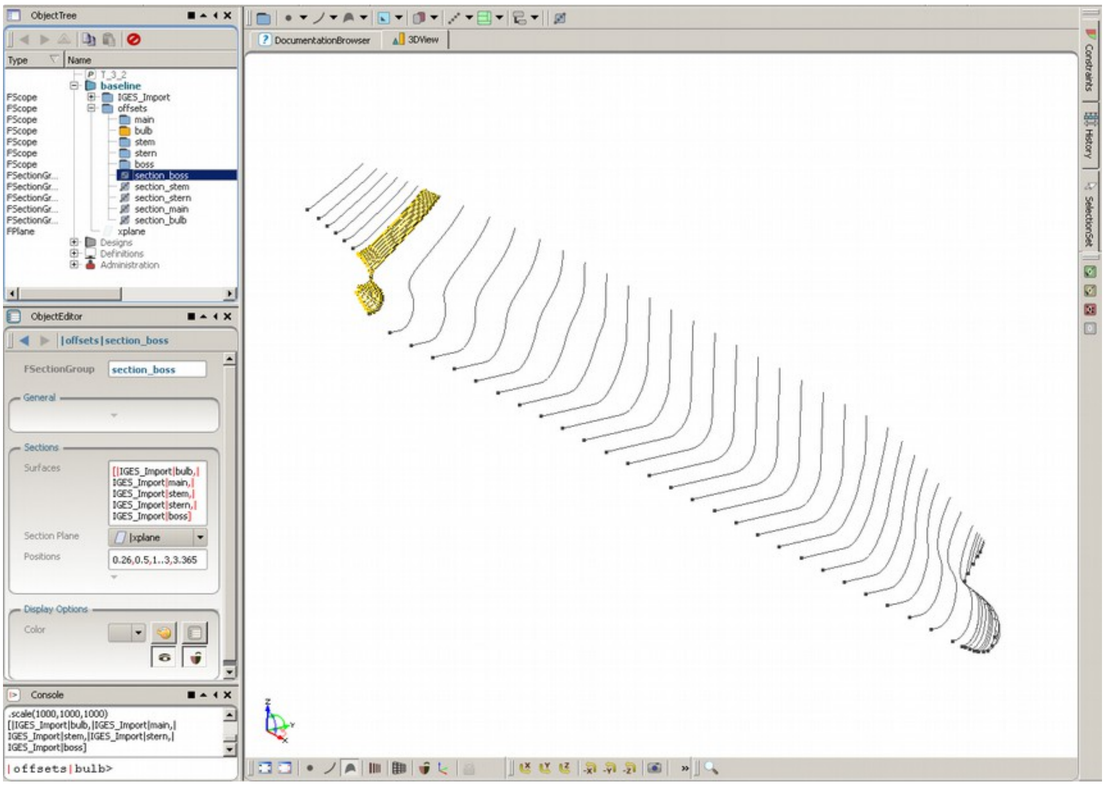


- In a similar way create the section groups for the rest of hull parts, use the positions:
  - bulb: 140.202,141,142..147,147.6
  - stem: 140.202,141,142,143,143.5
  - stern: -11.5,-9,-7..3,3.365
  - boss: 0.26,0.5,1..3,3.365

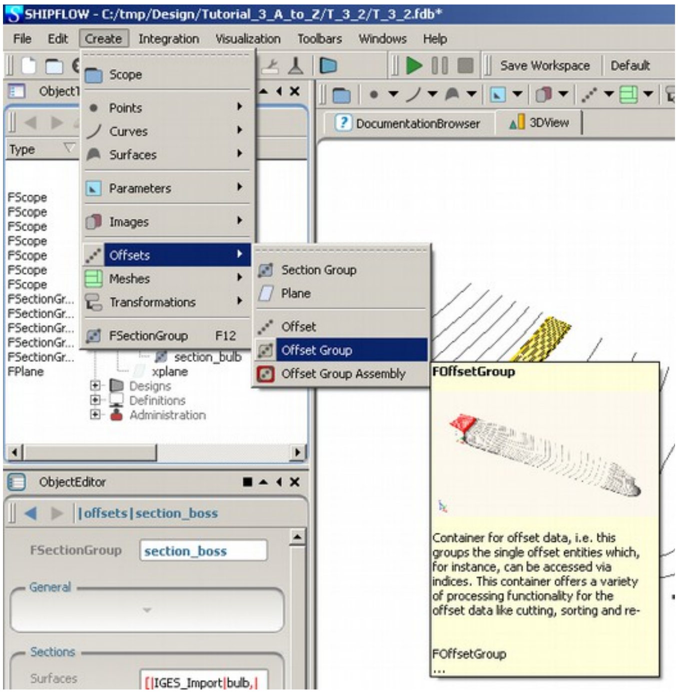
- All sections are prepared now, your 3D View may look like the one below. Notice that the distribution is rather coarse. This is however only for this exercise, some more informations and guidelines about sections will be given at the end of this tutorial.



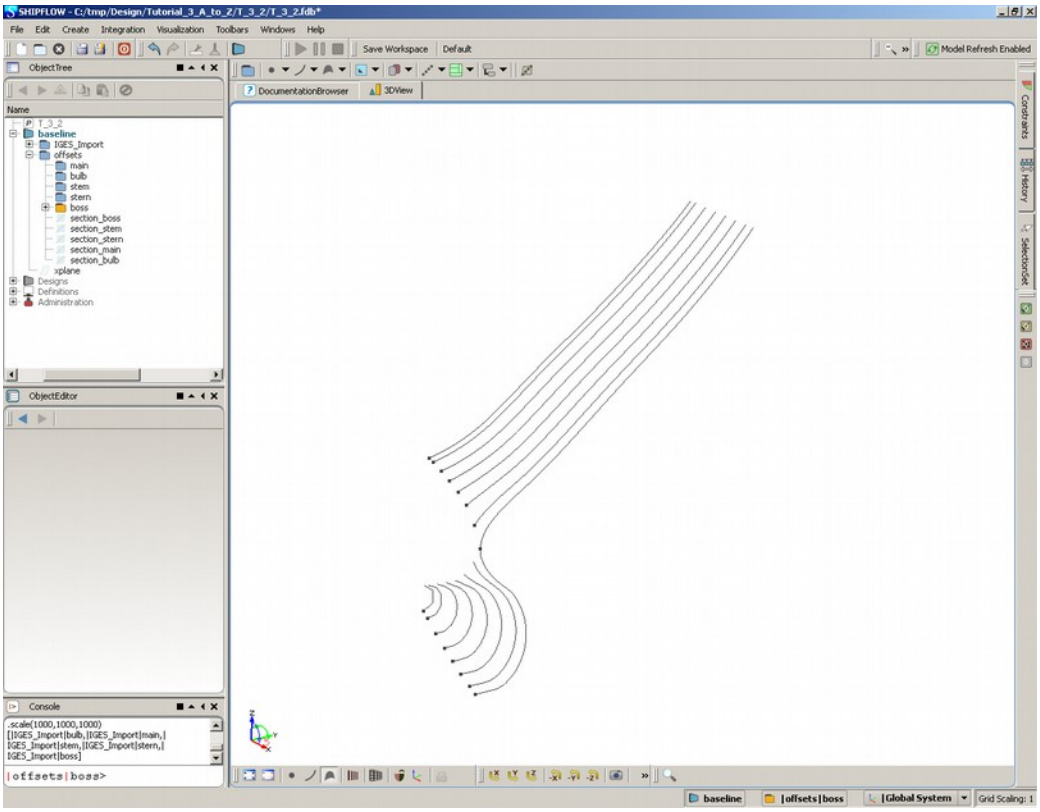
- ******* Now create the offsets from the sections, first we start with the boss *******
- Make boss scope current by clicking on it with the middle mouse button.
- Select the section_boss in the Object Tree as shown below, the boss section should become highlighted in the 3D View



- Create the offsets now by choosing from the menu **Create > Offsets > Offset Group**



- The Offsets for each section on the boss and the Offset Group consisting those offsets were created.
- The offset group was put in the current scope (|offsets|boss), rename it to **boss**, this will be the name in your offset file you will create later**.**
- Switch off visibility of all section groups, your screen should look like the one below:

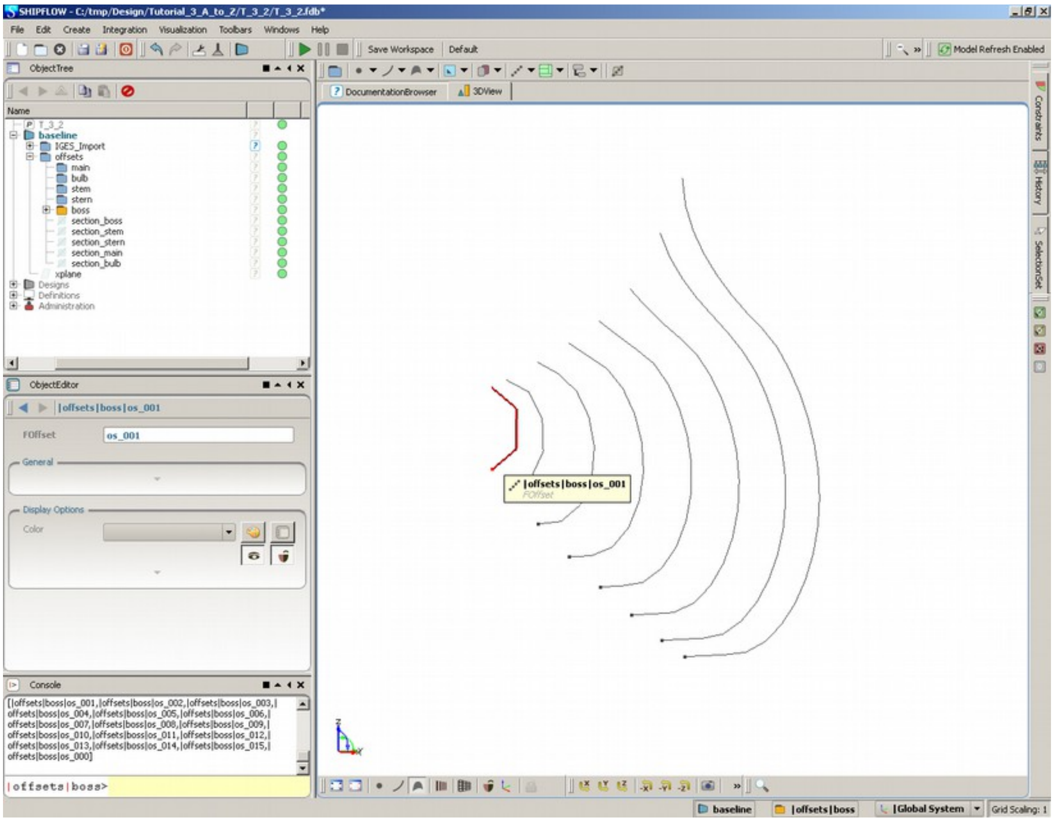


- Notice that the boss offset group contains also a part of the hull that is above it, this has to be removed.
- Select the offsets and from console window use command **.cutMinMax(2,-1,6.618)** where 2 means direction along z-axis and -1,6.618 is the interval of offsets to be remained

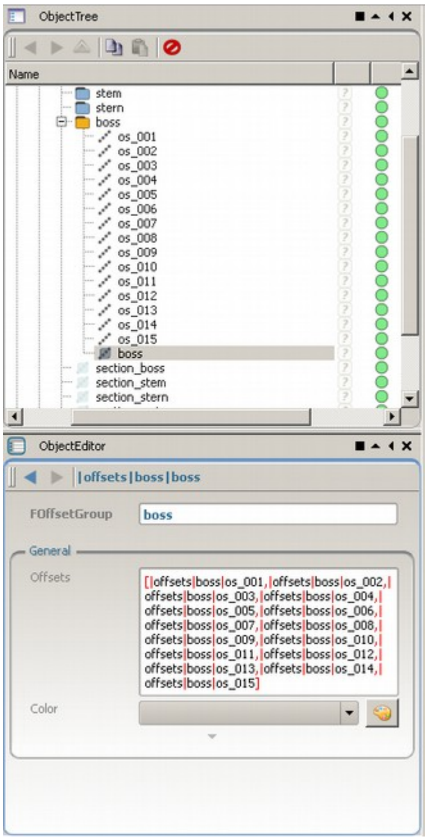
- SHIPFLOW is build in such a way that it accepts only the offsets where the first point (marked with larger point marker) is at the keel line and therefore one should make sure that the sections have the right direction.

- If the point order has to be reversed: Select the Offsets that need to be reordered and type in the Console the following command: **.reverse()** and hit return (enter) key. You should see on the screen that the sections start at the other end of the curve.

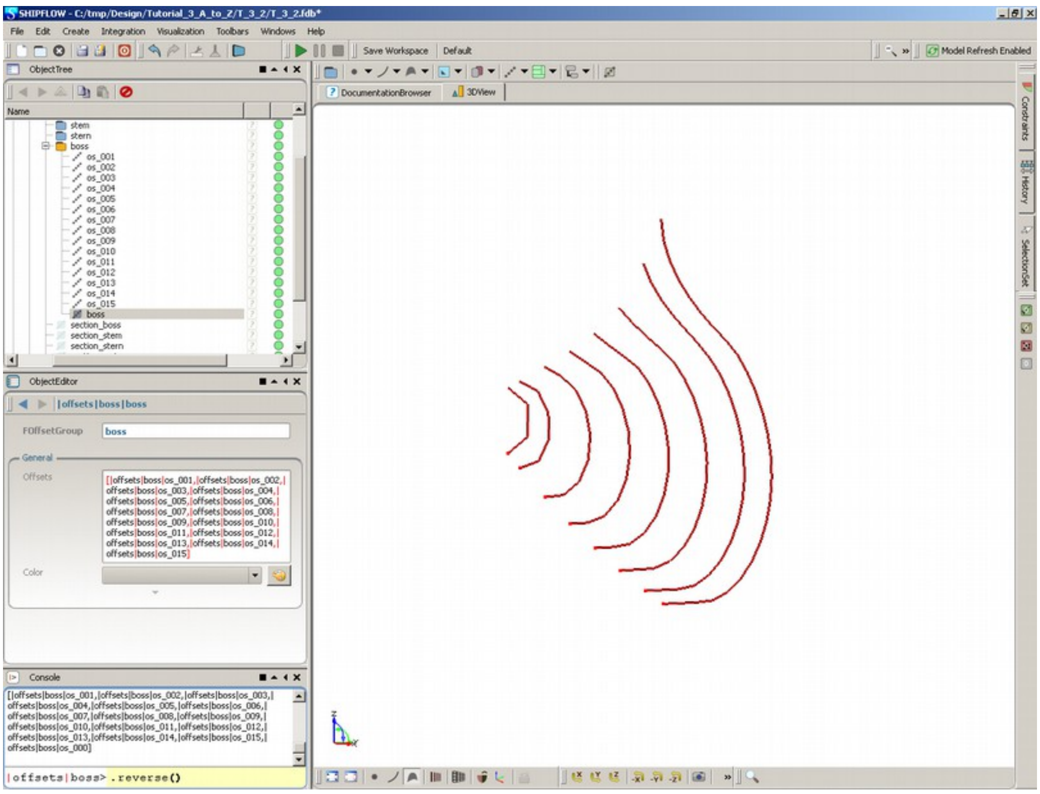
- **+++++ Next important step is to check the offset stations order.** Select the first section (section001) and observe in the 3D View that it is the tip of the boss



- Now in the Object tree select the boss OffsetGroup, in the Object Editor check the Entities window (click on Edit there if necessary). Look at the list and check the first section name. In this case it is the section001 which was the tip of the boss. In SHIPFLOW we need to have the offset sections ordered from the bow to the stern, this is not the case here. We will need to reverse the order.

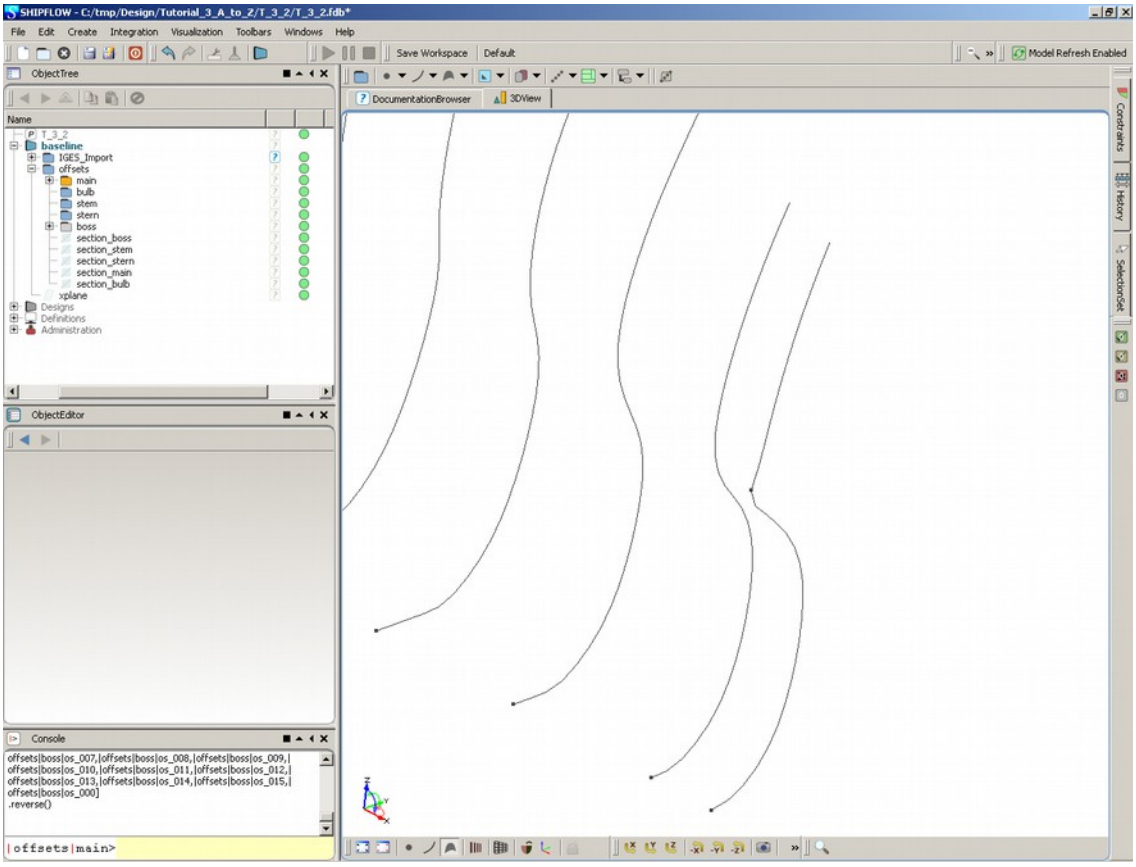


- Make sure that the boss offset group is selected in the Object Tree and in the Console type **.reverse()** and hit return key. Check again the Entities in the offset group, notice that the section000 is at the end of the list. Now everything is in the right order for the boss.



- Repeat the steps for other groups (stern, stem, bulb, when you begin with the main part stop just after you reversed the offset point order and read few additional informations that are below) starting from the point that starts with ***** when we started creating offsets.

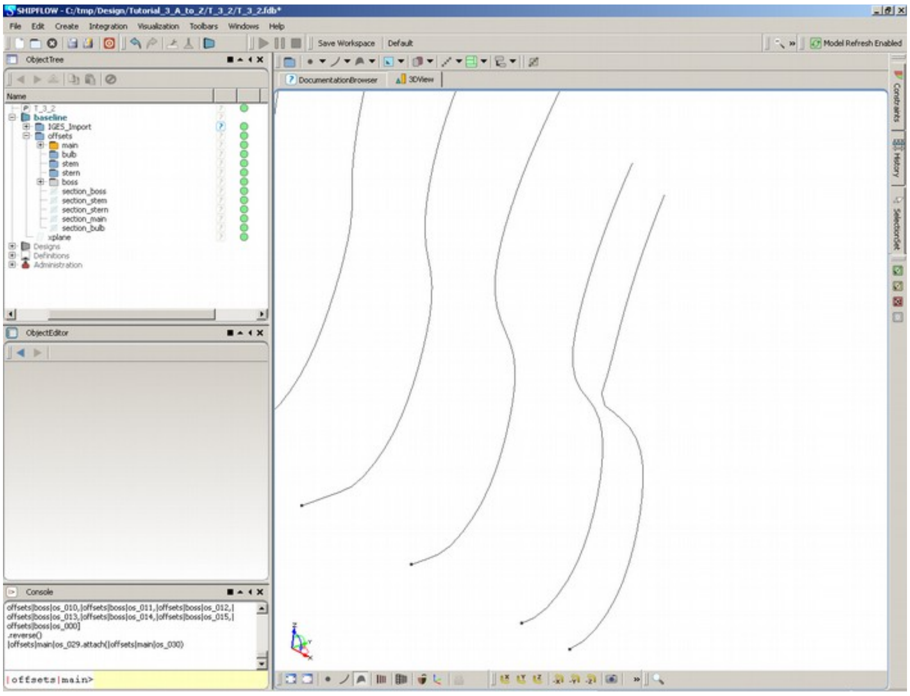
- So now you are at the point where the offset lines start at the keel line and the sections are ordered in longitudinal direction. Zoom in to see closer the first section at the bow.
- Notice that it looks as if it had two beginnings, it is because there are two lines in fact, we will now join them together.
- Select the lower part and check the name in the Object Editor (here it is section029) , select the lower part and check the name in the Object Editor (here it is section030)



- Now to join them type in the Console the following command: **|offsets|main|section029.attach(|offsets|main|section030)**

  this means: first_line**.attach(**second_line**)**

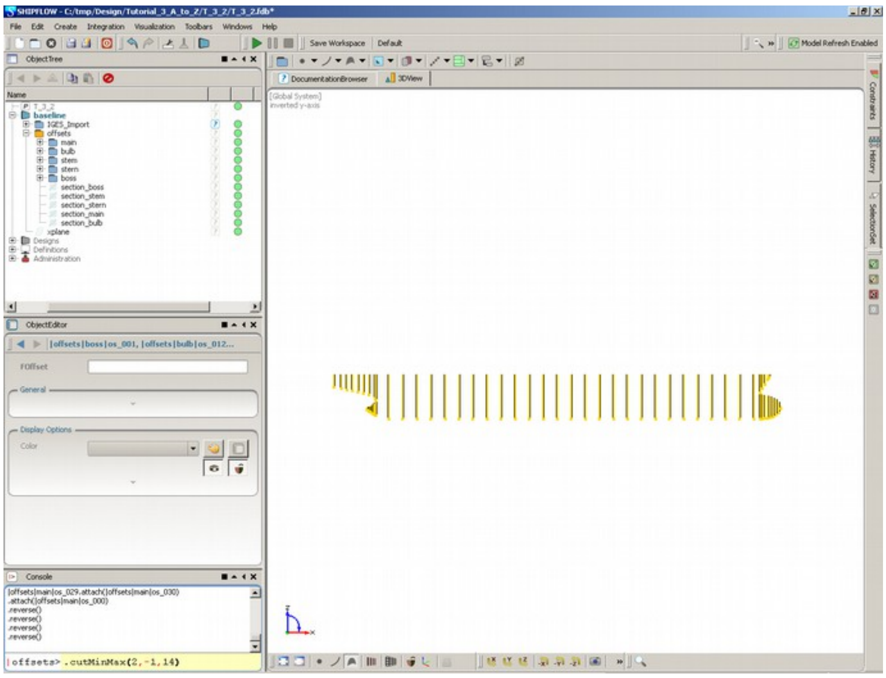
- The lines are now joined but still the upper part remains, delete it.



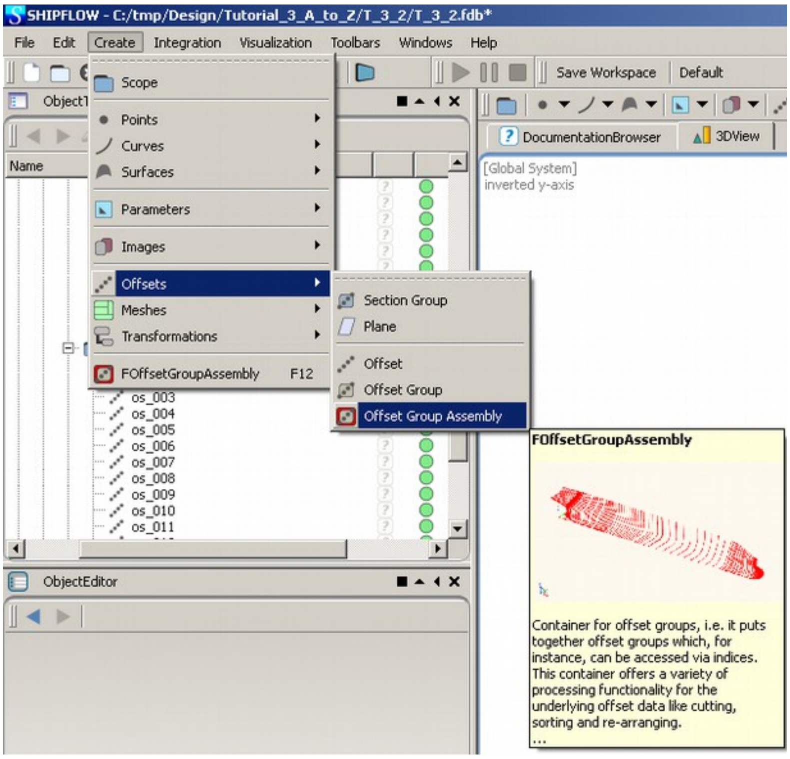
- Repeat similar steps if you find other “broken” sections.
- Now we will even out the sheer line, it is not necessary here but the commands can be useful in some situations and it is good to know them. Switch on all the offset groups if you switched them before for some reason.
- Set view to “Y” and using the coordinates displayed in the top right corner of the 3D View check where you like to cut the sheer line, say it is 14.0.
- Select all offset groups and use the following command to trim them off:

  .cutMinMax(2,-1,14)

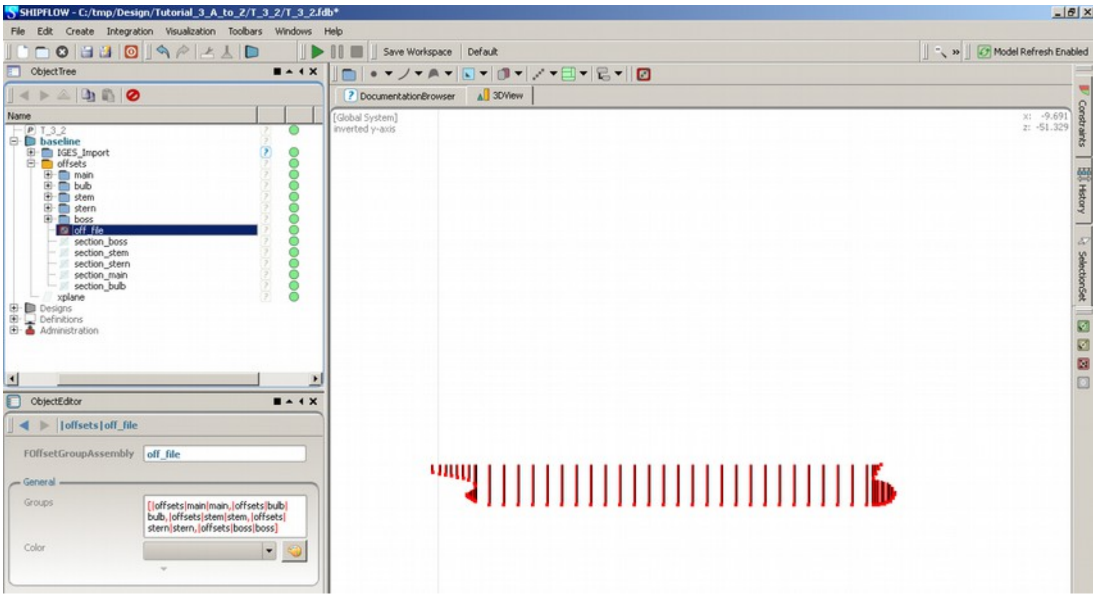
  this means we cut perpendicular to the “Z” direction (direction 2) and we cut everything below -1 and above 14



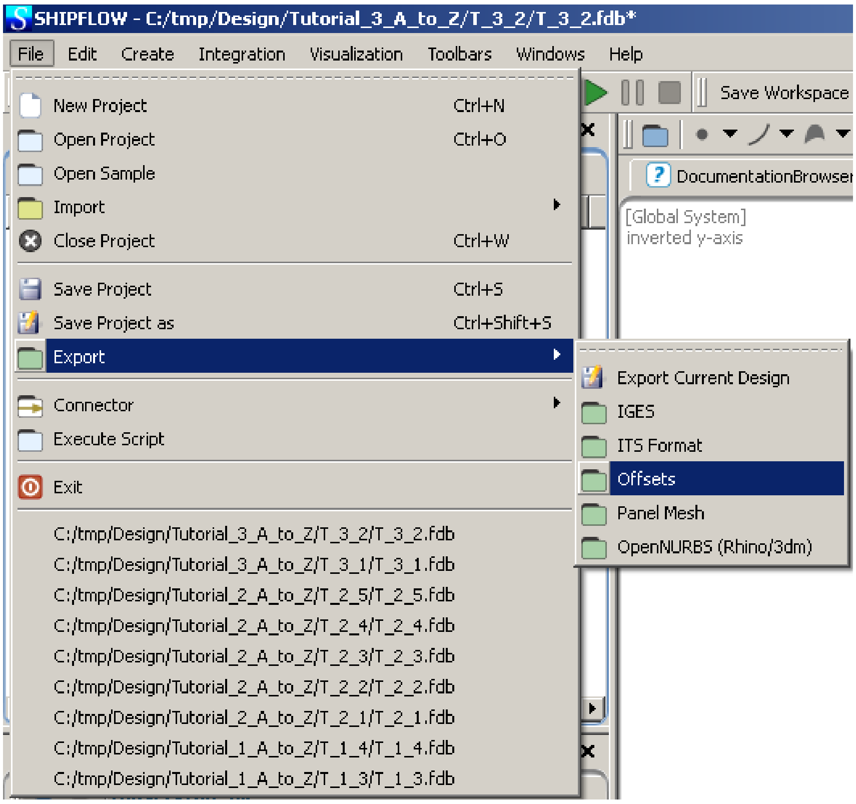
- To create an assembly of the offset groups make the offsets scope current (clicking with the middle mouse button) and then select all the offset groups (you can hold the Ctrl key and select them from the Object Tree) and choose from the menu **Create > Offsets > Offset Group Assembly**



- The offset assembly is now created. Rename it to e.g. off_file and check in the Object Editor that it consists all of the offsets groups (stem, bulb, main, boss, stern) in the Entities.
- This is your offset group assembly that can be used in the further calculations directly or it can be exported to the offset file.



- To export the offset file select the offset group assembly and choose from the menu **File > Export > Offsets.**



- Save the project and take a look at the offset file you just created with your favourite text editor

Tutorial 5 part 3 – Method for quick offset file generation

To make the offset file quicker you don't have to do it from the beginning for each new hull you work on. Here an example will be given how to replace the hull lines and keep the same or similar settings for new offset file.

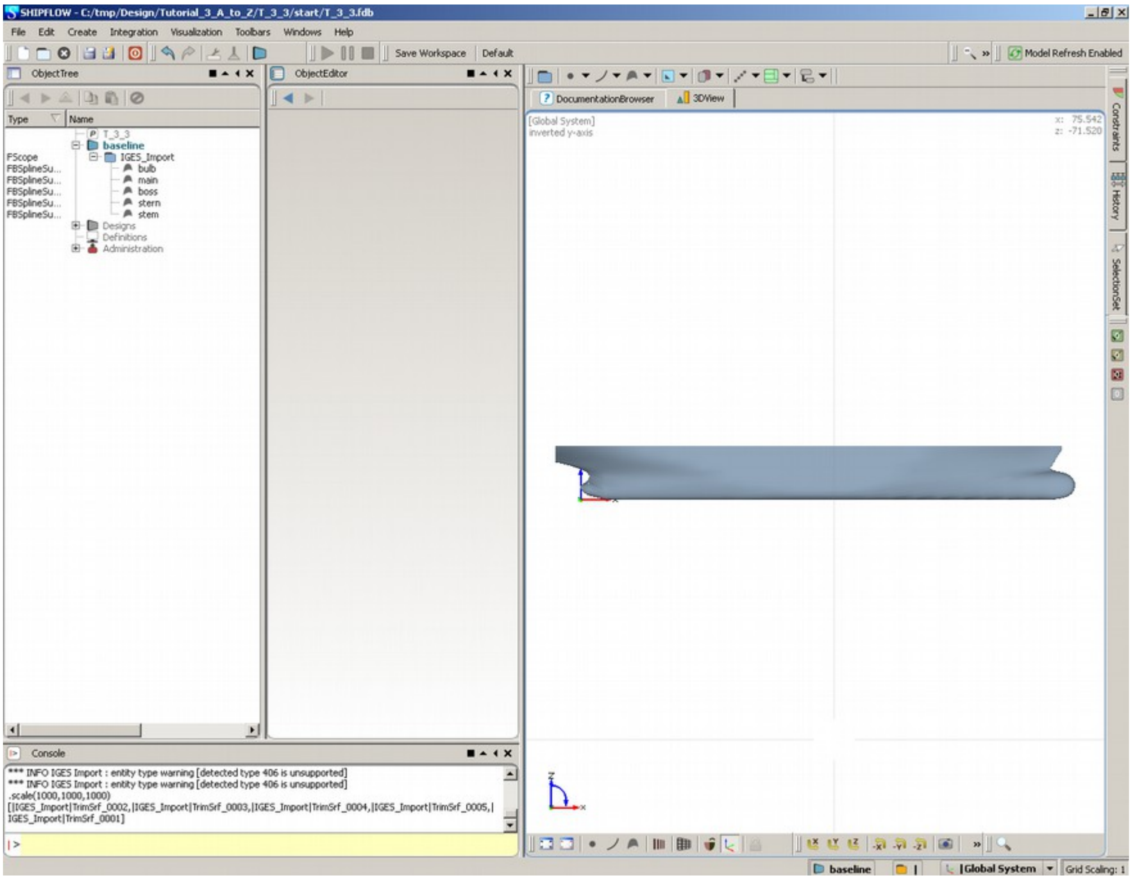
- Open a new project and import an IGES file from

  ..**\BasicTraining\Tutorial_5\source**

- Rename the surface patches as in the part 1 of this tutorial, remember about the scaling.

  (you can also find a project file named T_5_3_B in the same directory, it is the project with already imported, scaled and renamed patches)

- Save the file as Tutorial_5_3

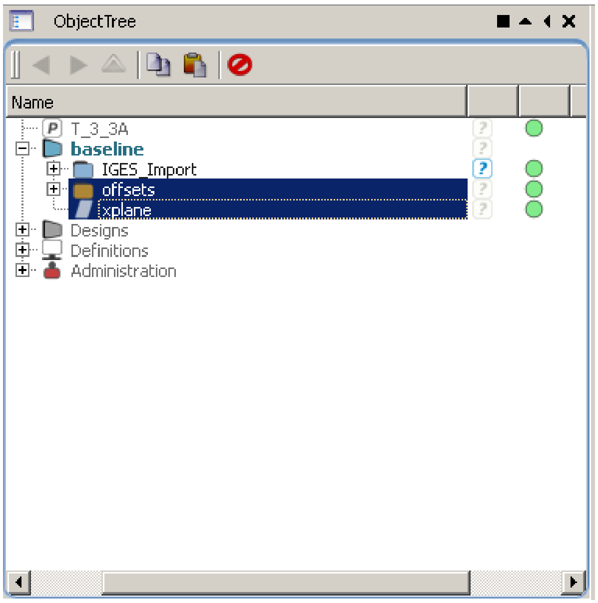


- Open another SHIPFLOW 4.0 application and load the result of the part 2 of the tutorial or the file named **T_5_2_f** from

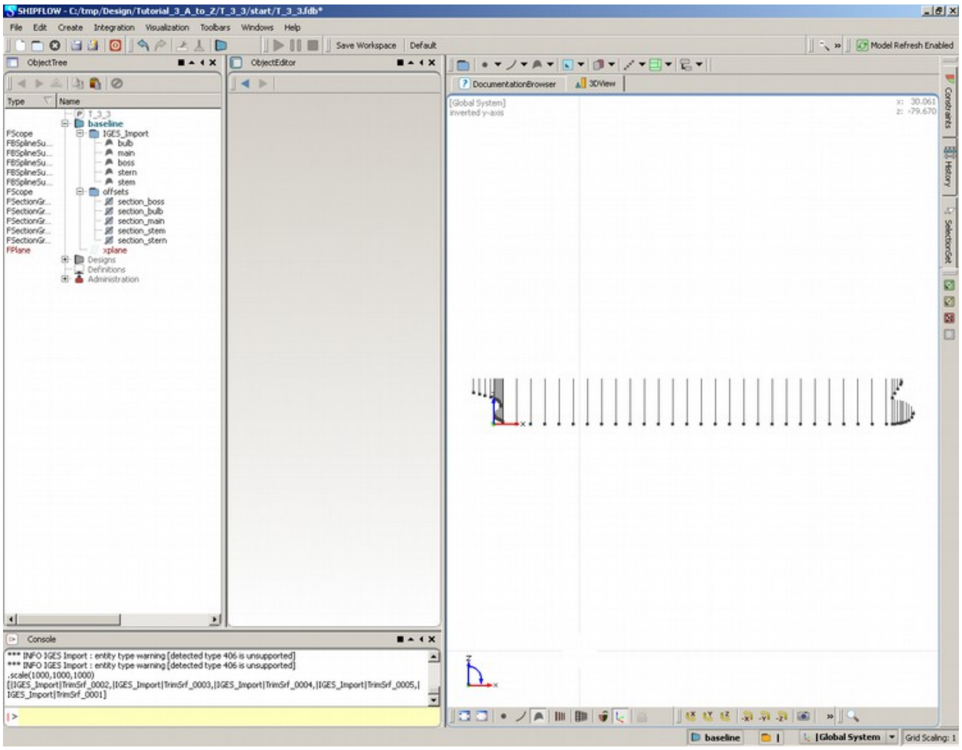
  ..**\BasicTraining\Tutorial_5\T_5_2**

- Remove the offsets from all groups, just select and delete.

- Select in the Object Tree the offsets scope as well as the xplane object



- Press Ctrl+C or select the Copy icon in the Object Tree to copy the selected objects
- Go to your second SHIPFLOW 4.0 application with the imported Version B of the hull and paste the objects. Switch on the visibility of the section groups.
- The sections with all the settings from the part 1 and 2 of the tutorial are now applied to the second hull.

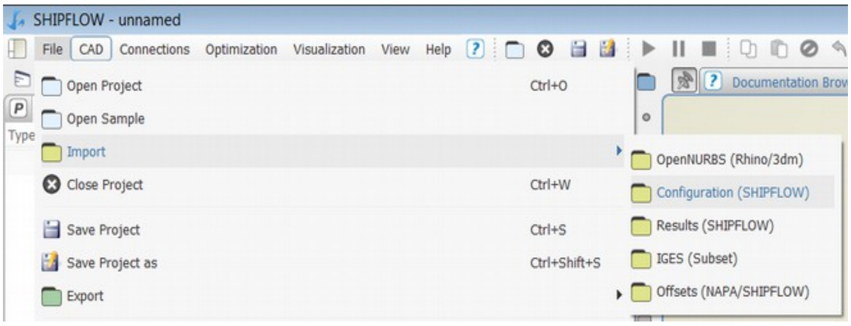


- This new hull will probably require some adjustments of the positions and creation and editing of the offsets but now you know how to do it so go back to the part 2 of the tutorial if you need some help and create new offset file for this hull.
- Good luck

Tutorial 6 part 1 – XBOUND

The purpose with this part of the tutorial is to show how to run XBOUND and make some basic changes of the XBOUND configuration. The command file is a simple configuration for potential and boundary layer flow.

- Import configuration file *config* from the directory **..\BasicTraining\Tutorial_6\source** by selecting **File > Import > Configuration(SHIPFLOW)**

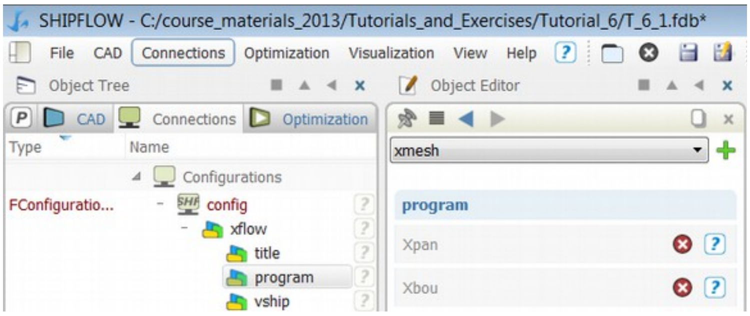


- Save the project e.g. Tutorial_6_1 by selecting File > Save as or click on the icon 

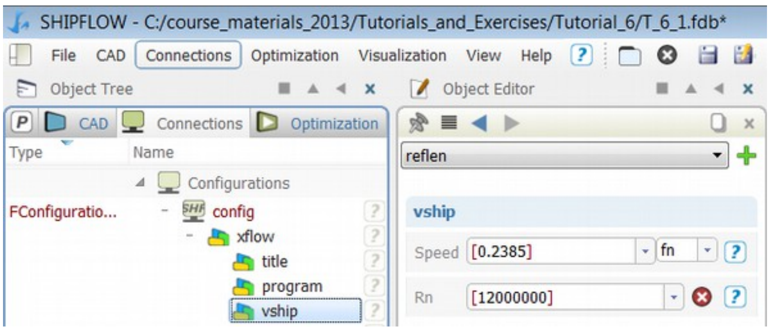
The configuration is set up for a "basic model" solution with XPAN, i.e. it has a horizontal symmetry plane instead of a free surface. This is sufficient for the purpose of this tutorial.

The configuration is set to run XMESH and XPAN and a command to run XBOUND must be added. The speed of the ship must also be added. XBOUND requires that the Reynolds number Rn is set.

- Check that *xpan* and *xbound* is selected in **Object Tree | Connections | Configurations | config | xflow | program**



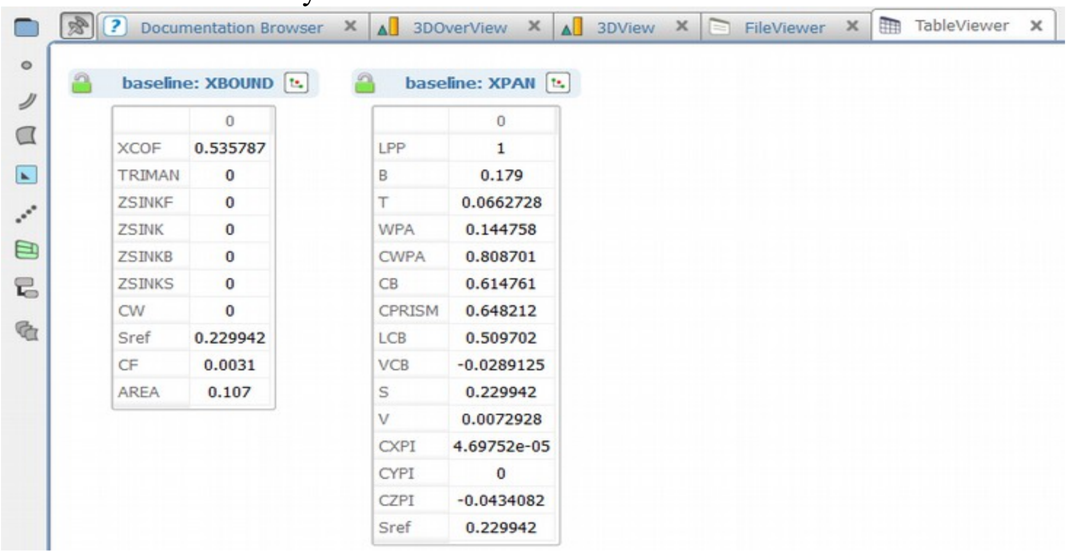
- Check that the Reynolds number *Rn* is set to 1.2e7 in **Object Tree | Connections | Configurations | config | xflow | vship**



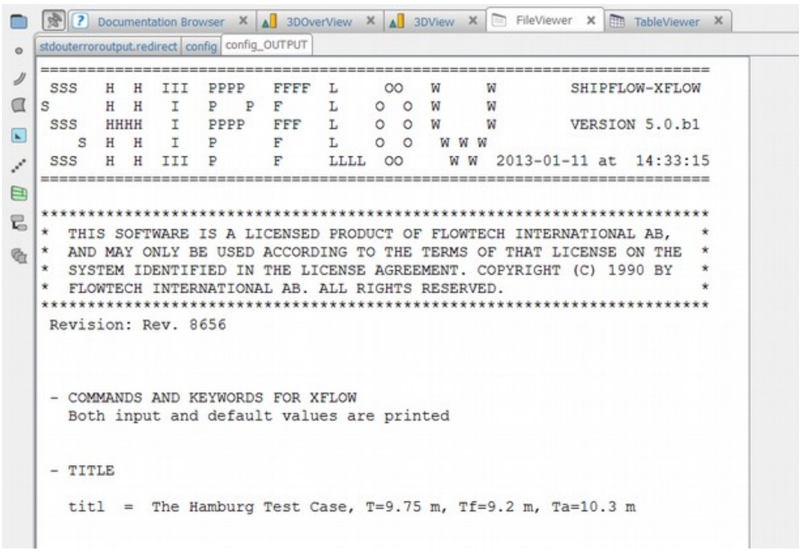
- Run the computations.
- Save the project.

The keywords xmesh and xpan can be removed from the program command once the XPAN solution is computed. The next time XBOUND is run it will read the potential flow solution from the database files.

- A summary of the results can be found in the TableViewer. Note that the wave resistance is zero in this case since XPAN was computed without the free surface. The estimated friction coefficient CF from the boundary layer computation is printed. The reference area Sref from XPAN should be used for the rescaling of the coefficient, the AREA value is the the part of the hull that is covered by the streamlines.

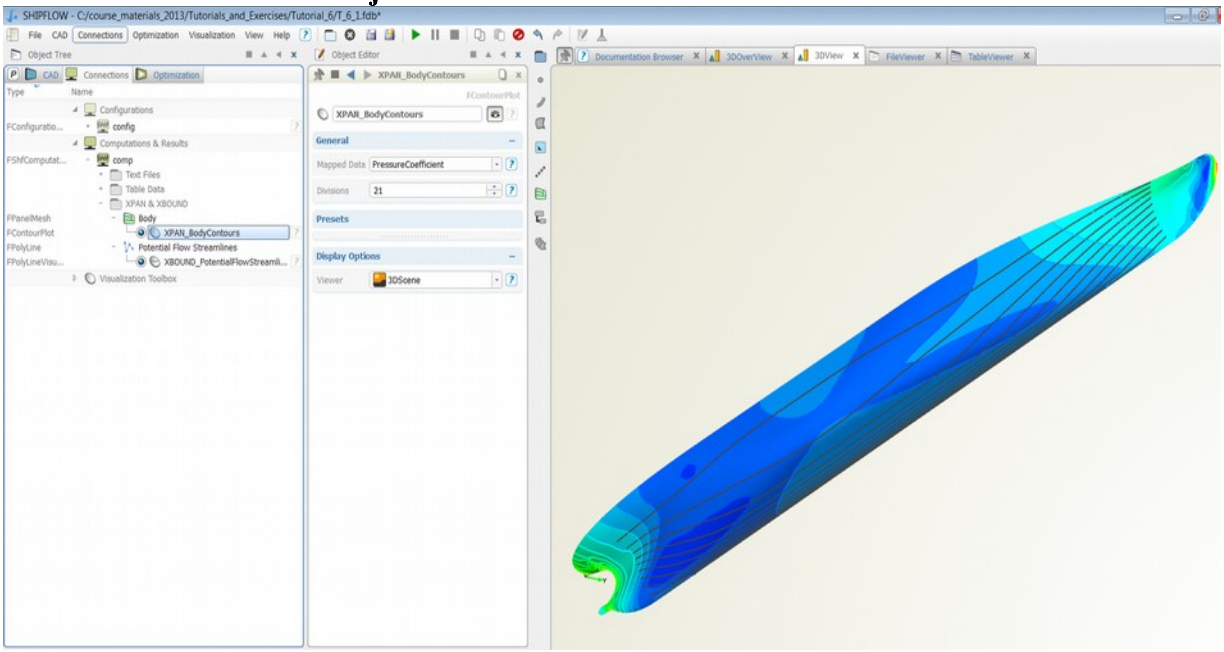


- The complete configuration for XBOUND and some results can be read in the output file in the FileViewer



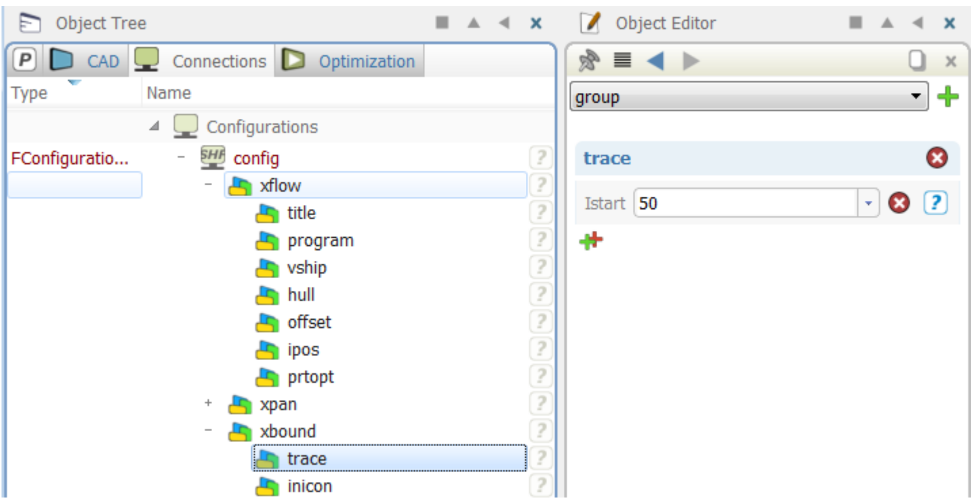
- Hide the offset sections and go to **Connections | Configurations & Results | comp | XPAN**

& XBOUND in the **ObjectTree** to control the visualization of the results.

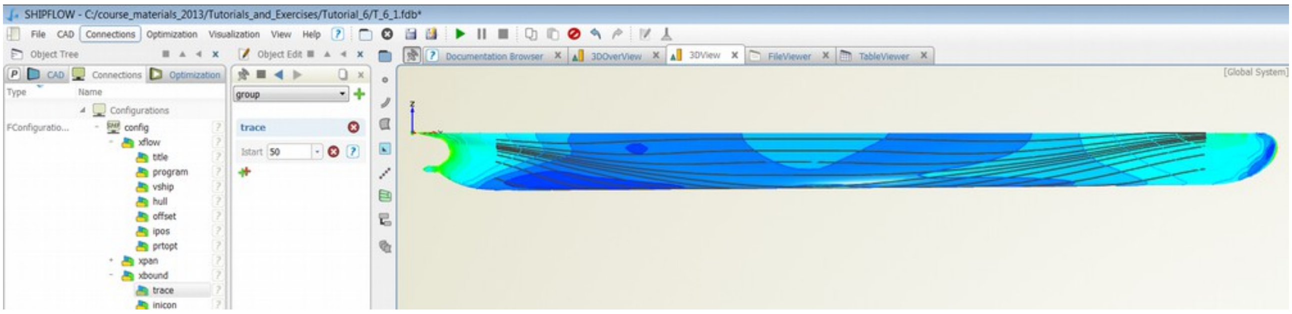


The next part of the tutorial will show some possibilities for the user to control the distribution of the potential flow streamlines. In our example we will try to get some additional streamlines on the side of the ship at midships. This can be done for example by specifying another value of the ISTART parameter that controls the location of points from which the streamlines are traced (up- and downstream).

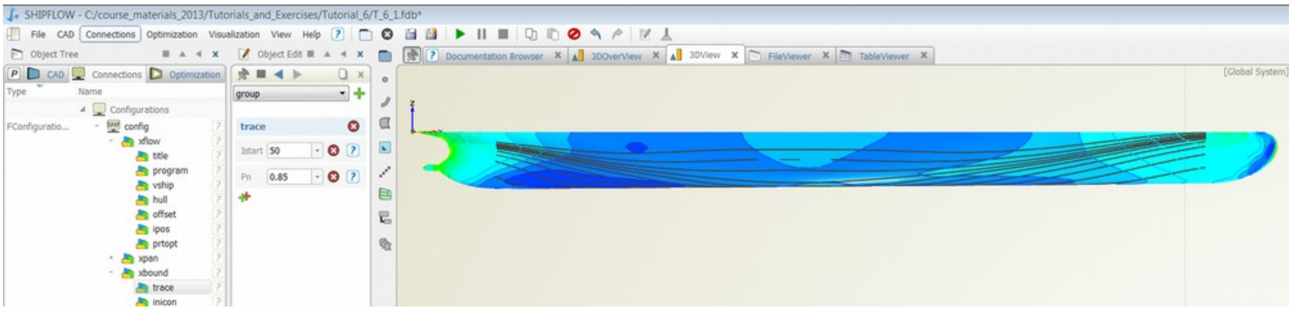
- Create an **xbound** section in the configuration tree and add a **trace** command. The default values for the parameters can be found in the output file from the previous computation. Change the value of *Istart* to 50 by adding this change to the **trace** command.



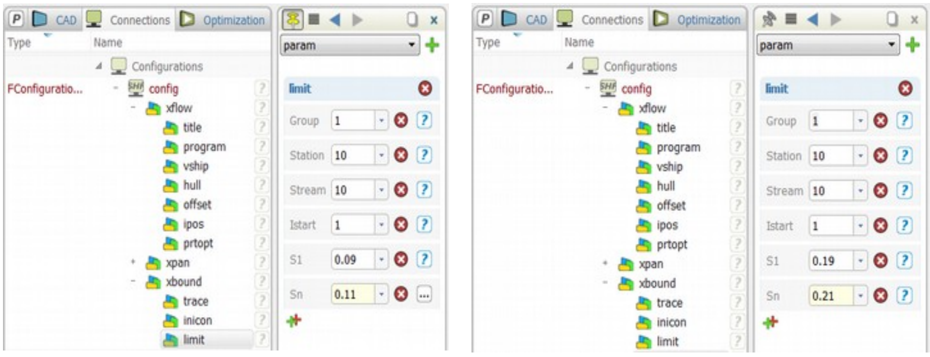
- Delete *xmesh* and *xpan* from the **program** command and re-run XBOUND

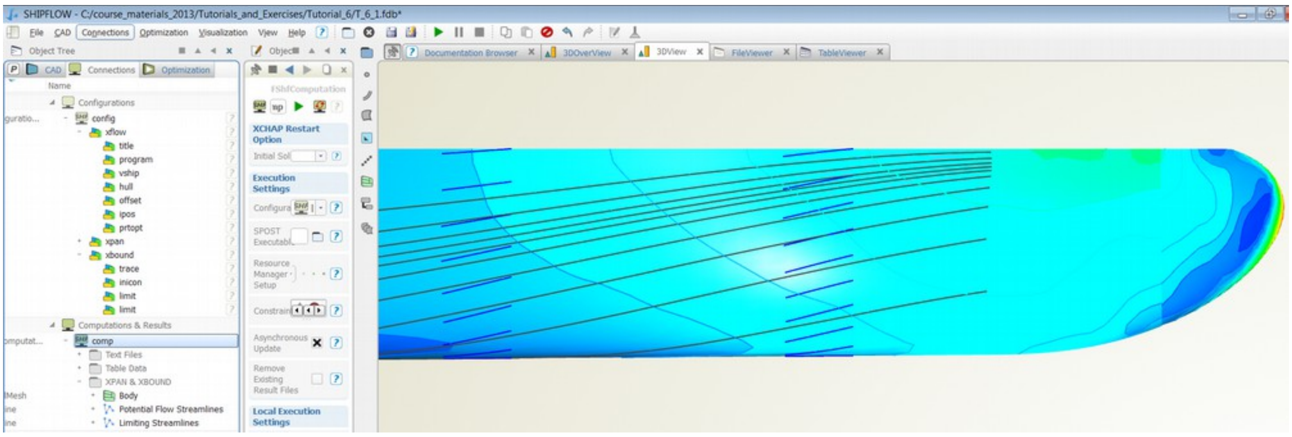


- Modify the transverse distribution of starting points at *Istart* by increasing the distance between the uppermost streamline and the waterline by changing the parameter *Pn* from 0.95 to 0.85.



- Trace limiting streamlines on the forebody by adding two **limit** commands to the **xbound** section. Set up the parameters required to trace 10 streamlines of 10 stations starting the tracing at the first station. The first limiting streamline group should start at x/L=0.09 and end at 0.11 and the second group from 0.19 to 0.21.



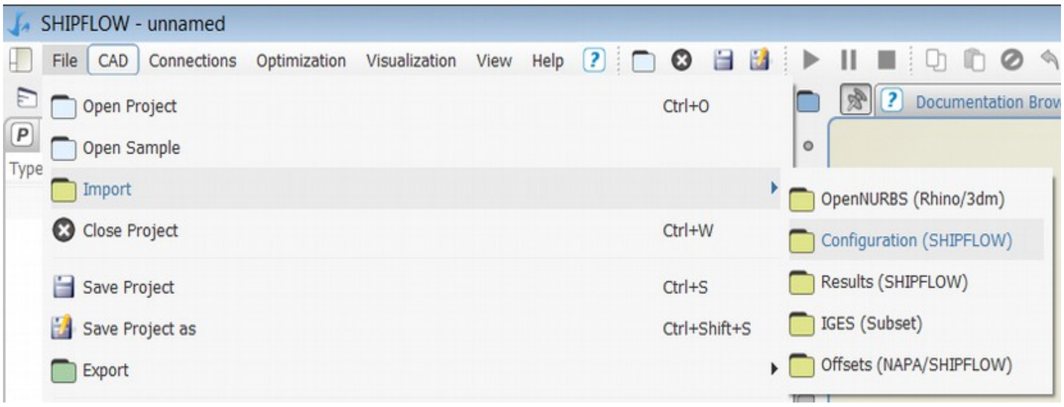


- Additional task:
  The configuration contains the command **prtopt** with the keyword *strlres* to create an additional output file with detailed results from XBOUND. Find this file in the directory structure of your project, in this case **Tutorial_6_1\manual_results\baseline\comp\config_RUN_DIR\config_STRLRES**. The first half of the file contains the coordinates, arc-length, speed at the outer edge of the boundary layer and curvature for each streamline. The last half contains the boundary layer results along the streamlines.

Tutorial 7 part 1 – Automatic grid generation with XGRID

The purpose of this tutorial is to demonstrate automatic grid generation with XGRID. Different grid refinements and simulation requirements will be applied for a container ship. Some basic post-processing techniques for visualizing the grid and boundary conditions will be demonstrated.

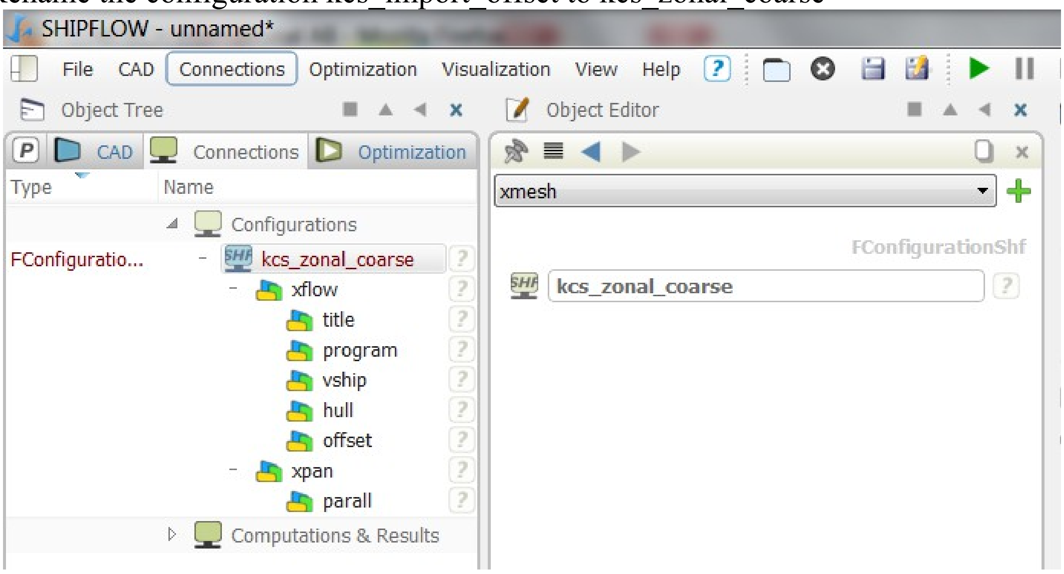
- Import configuration file *kcs_import_offset* from the shipflow examples directory by selecting **File > Import > Configuration(SHIPFLOW)**



- Save the project e.g. Tutorial_7_1 by selecting File > Save as or click on the icon 

The configuration is set up to run free surface computation with XPAN on a coarse mesh. Change the configuration to run all modules with a coarse XGRID grid and zero XCHAP iterations.

- Rename the configuration kcs_import_offset to kcs_zonal_coarse

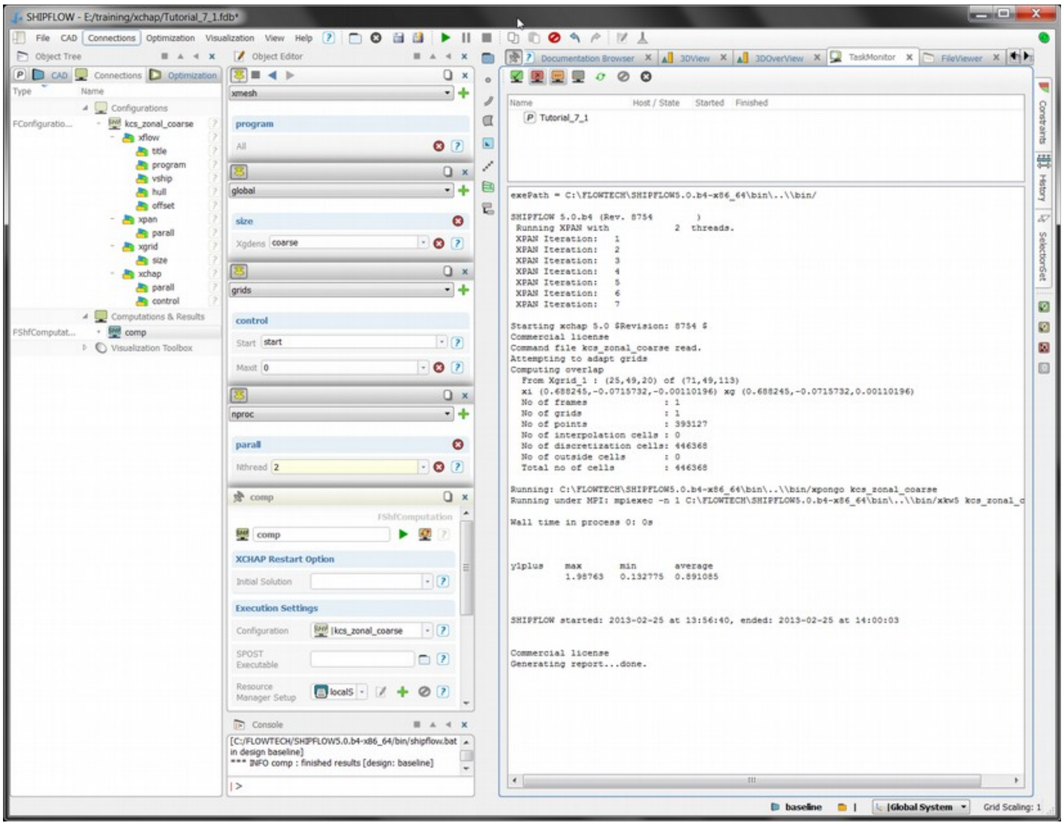


- Select *all* in **Object Tree | Connections | Configurations | kcs_zonal_offset | xflow | program**. This will make all modules XMESH, XPAN, XBOUND, XGRID, XCHAP run
- Add *xgrid* in **Object Tree | Connections | Configurations | kcs_zonal_offset**
  - Add the *size* command
  - Add *Xgdens* in the **size** command and select *coarse.* XGRID will generate a coarse grid for the zonal approach

- Add *xchap* in **Object Tree | Connections | Configurations | kcs_zonal_offset**
  - Add *maxit* in the **control** command and make sure it is set to 0
  - Add the *parallel* command
  - Add nproc in the parallel command and set it to a suitable number of processes for your computer, for example 2 on a dual core processor

The configuration is now completed. All SHIPFLOW modules will be run in a sequence in the zonal approach. The grids densities for XPAN and XGRID are set to coarse to make the computation faster. XCHAP will only set boundary conditions and create an initial solution since the number of iterations are set to zero.

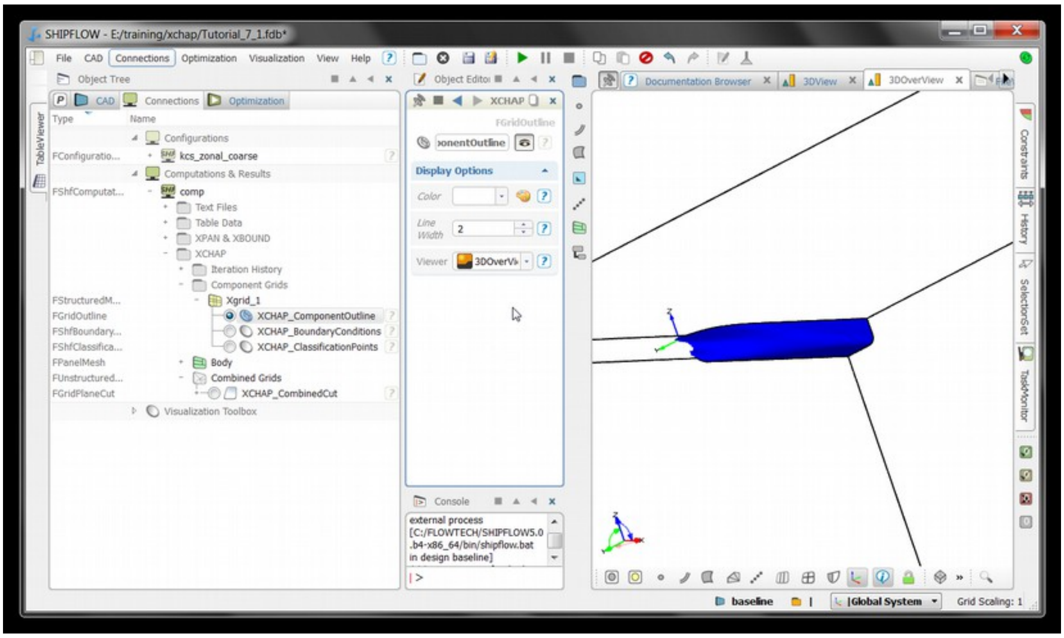
- Save the project
- Select the Computation object *comp* in **Object Tree | Connections | Computations & Results**
- Press the green start button
- View the execution in the TaskMonitor. It can be opened from the menu **View | Windows**



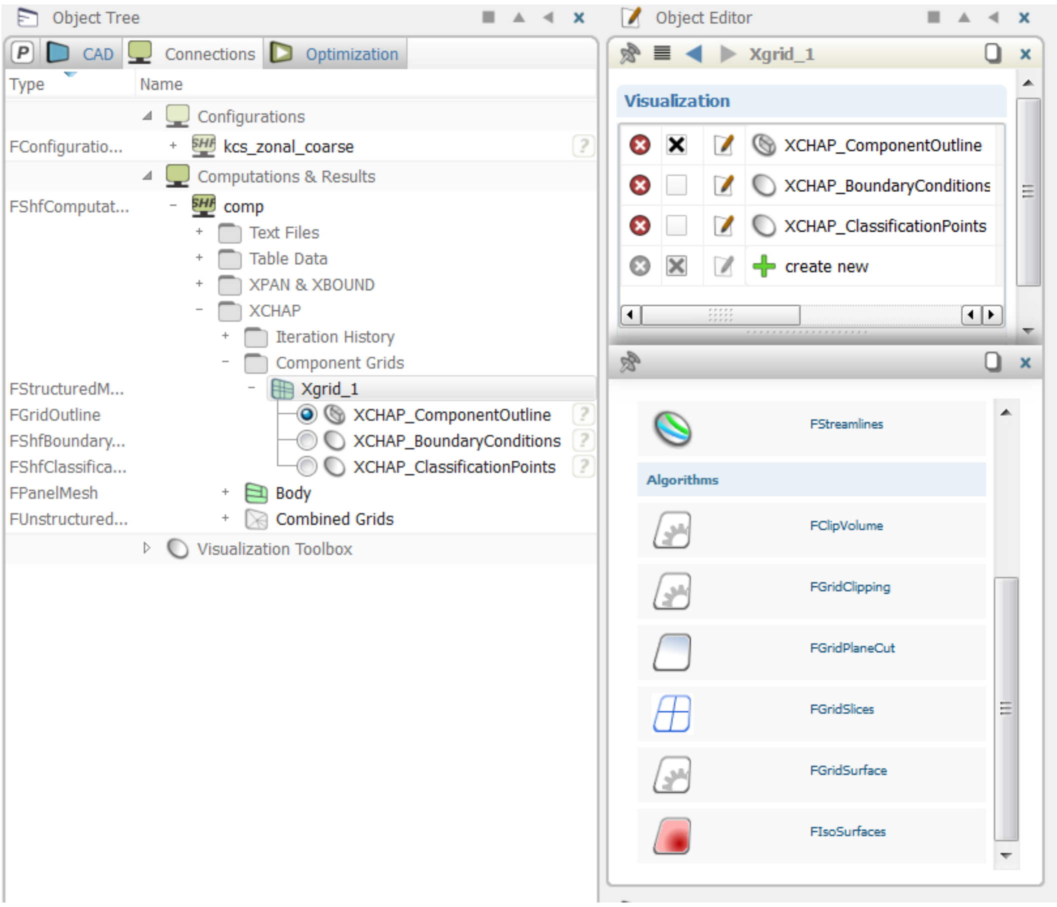
It is only the number of iterations for XCHAP that needs to be increased to about 4000 before a XCHAP computation can be started. However, the grids should be changed to *fine* to achieve accurate solutions. This will be left for the reader to exploit. The remaining part of the tutorial will explain how to visualize the grid and the boundary conditions.

The results are found under the **Object Tree | Connections | Computations & Results | comp**. Press (the update icon and) the "+". The program shows the pressure coefficient on the hull in the 3DOverView window by default. In our case the pressure is zero so the hull is rendered in one colour. The visualization of the offsets has been turned off in the picture below. The outline of the XGRID grid has be turned on by activating the **Object Tree | Connections | Computations &**

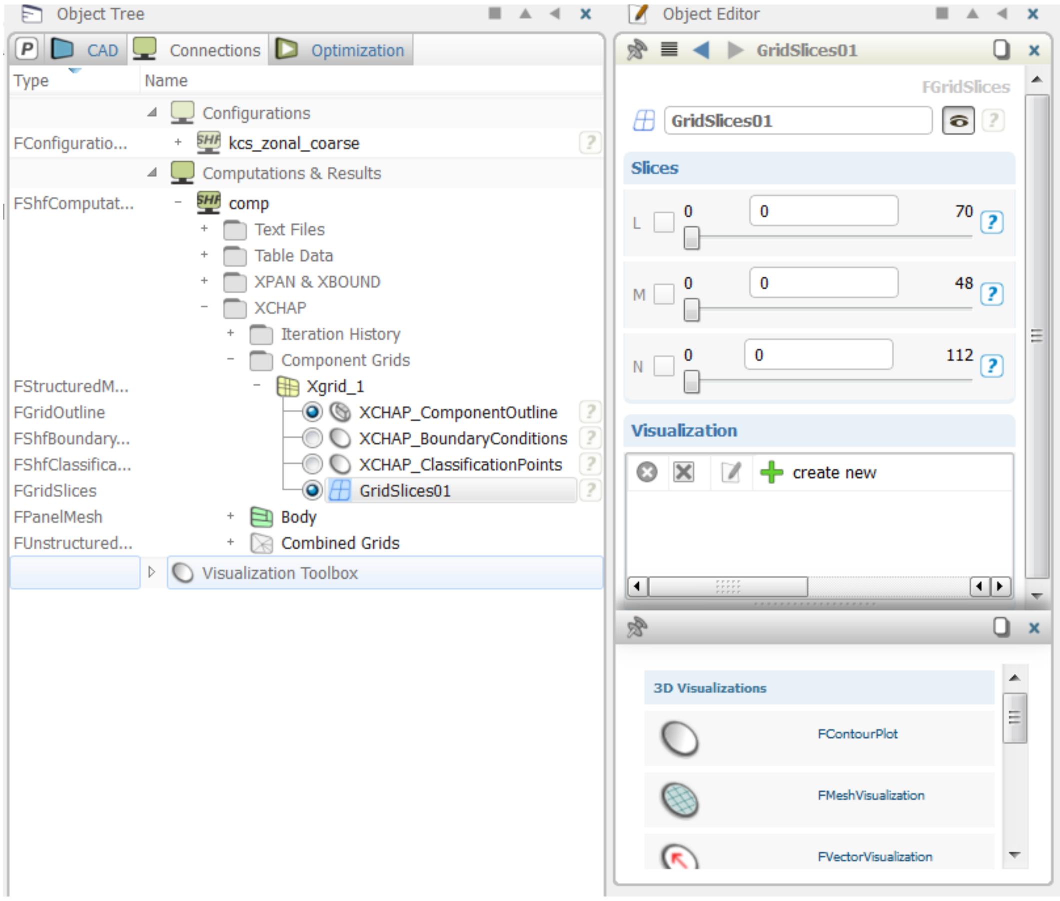
Results | comp | XCHAP | Component Grids | Xgrid_1 | XCHAP_ComponentOutline tool.



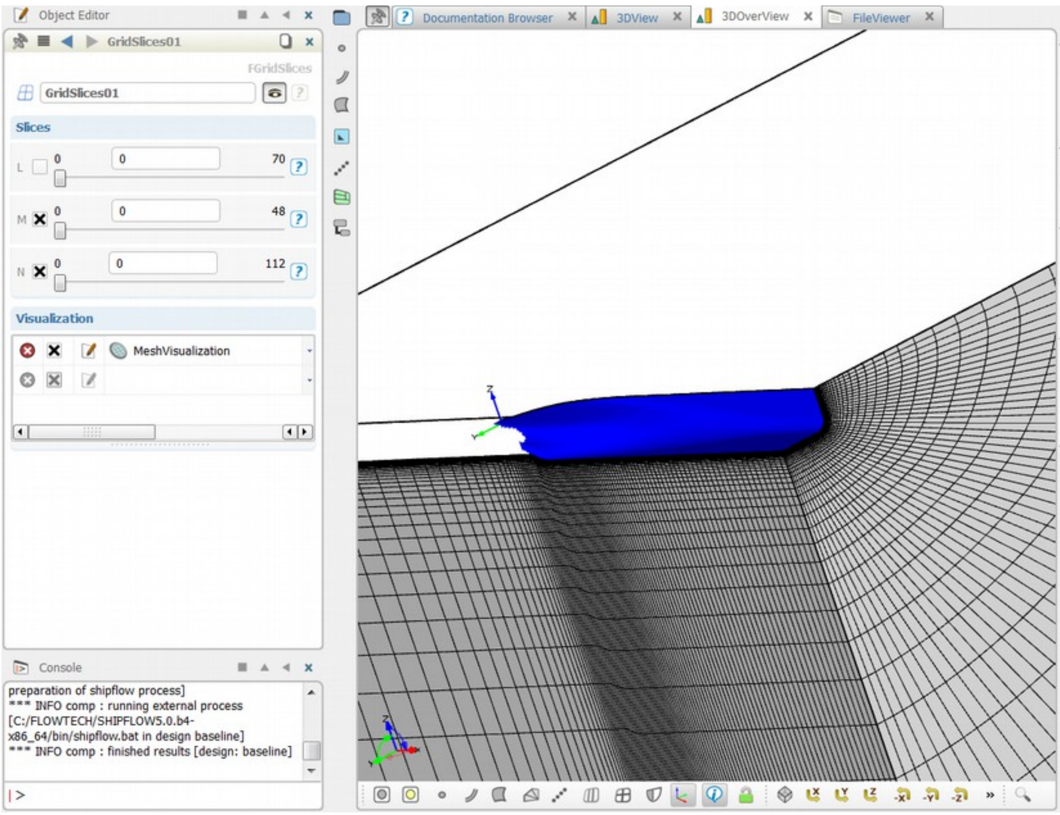
The GridSlice tool can be used for inspecting the grid planes in a 3d grid. The tool is created by selecting the Xgrid_1 object and then selecting Create New in the empty row of the Visualization widget as shown in the picture below.



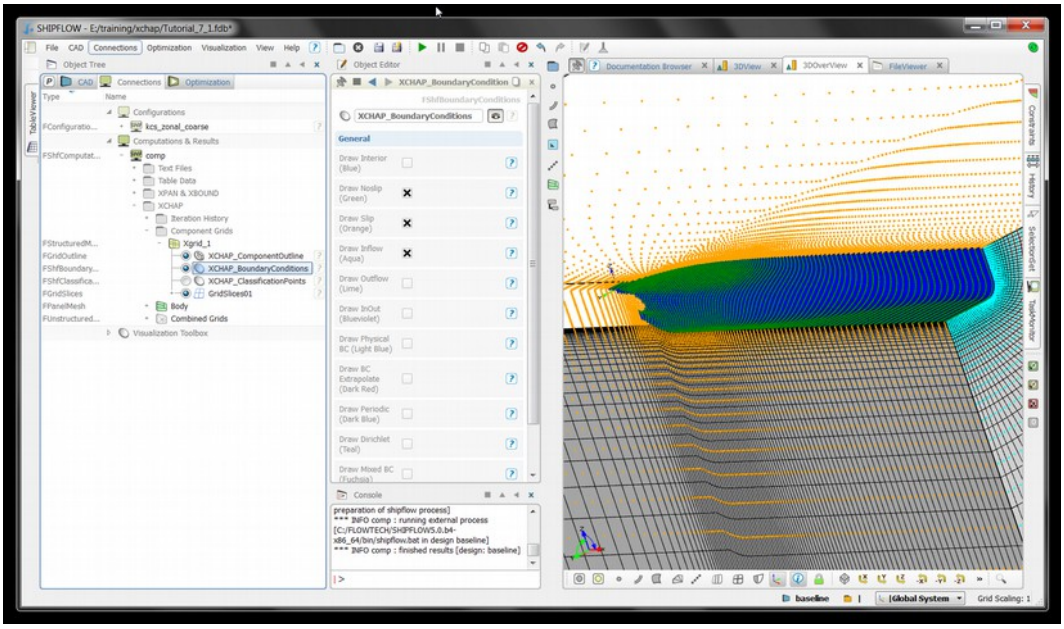
The next step is to select the newly created GridSlices object and “create new” in the empty row in the Visualization widget. Choose the FmeshVisualization alternative.



Grid planes in the three parametric directions L, M, N can be activated in the editor. The planes can be shifted in the corresponding parametric direction by moving the slider or entering the value. Try to change the location of the grid plane and zoom in to inspect the grid.



The boundary conditions can be visualized with the XCHAP_BoundaryConditions tool which can be found in **XCHAP | Component Grids | Xgrid_1** in this case. Activate the tool with the round button in the Object Tree. Press the text part to show the editor. Select for example the Noslip, Slip and Inflow boundary conditions. A coloured sphere is then displayed at the centre of each boundary cell face.

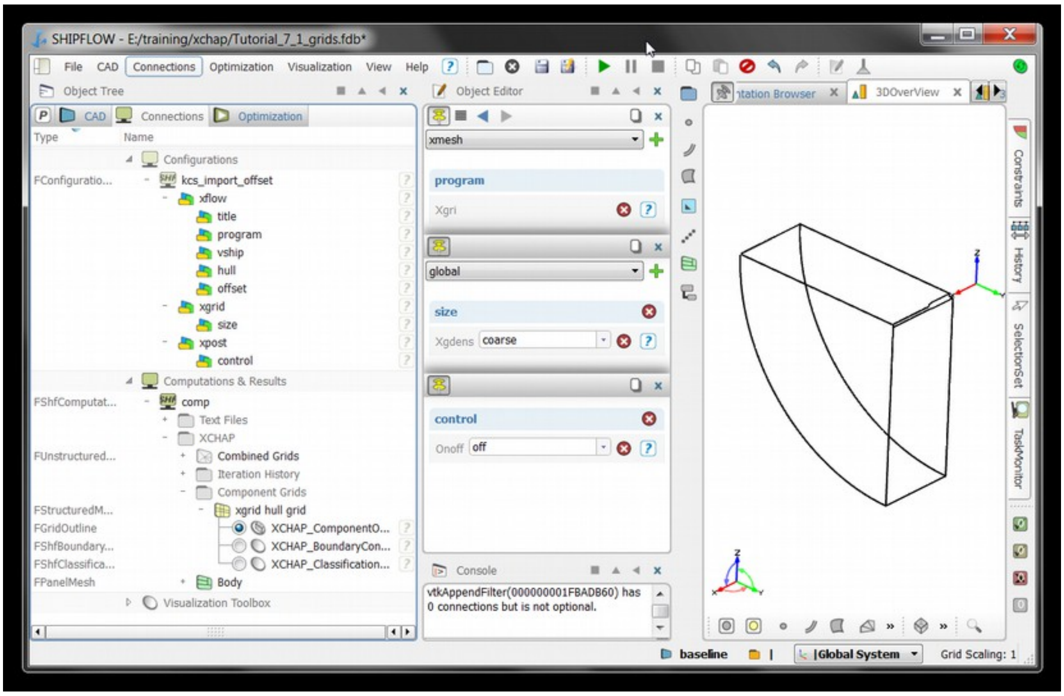


Automatic grid generation – FINE, MEDIUM and COARSE grids

A coarse grid was used in the example above. This type of grids yields a too low grid resolution for normal purposes. The grid density can easily be increase by using the FINE or MEDIUM options in the XGRID command SIZE.

In the following we will compute several grids of varying grid density and types. To make the exercises quicker to execute the configurations are limited to only run XGRID. The results will not contain any flow solution or boundary conditions in this case and thus only the visualization tools for displaying grids can be used.

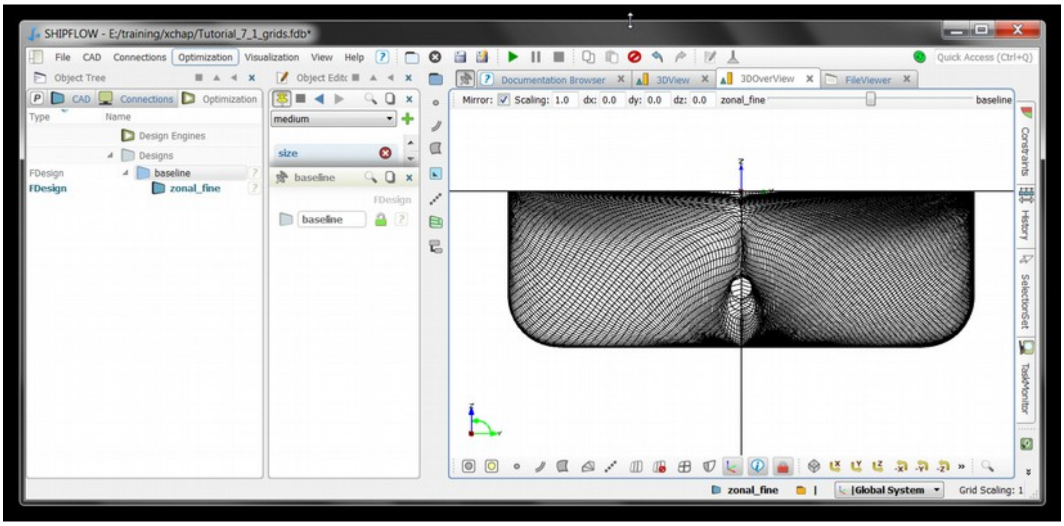
- Close any projects in the GUI.
- Import configuration file *kcs_import_offset* from the shipflow examples directory
- Change the configuration to just run XGRID with the selected grid density
  - Select only *xgrid* in **xflow | program**
  - Remove the *xpan* configuration section
  - Add a configuration for XGRID
  - Add the *size* command and set *xgdens* to *coarse*
  - Add a configuration for XPOST
  - Add the *control* command and set *onoff* to *off*
- Save the project and run SHIPFLOW
- Toggle the **Object Tree | Connections | Computations & Results | comp | XCHAP | Component Grids | Xgrid_1 | XCHAP_ComponentOutline** tool active



- Create also a GridSlice visualizer as describe in the previous section

Next we will compute a finer grid for comparison. The easiest way to do this is to first create a new Design and then change the grid density. The grids can be compared afterwards using for example the GridSlice tools.

- First create a design variant from menu **Optimization > Create New Design From Current Design** and name it e.g. *zonal_fine*
- Next change to *fine* grid on the *size* command for XGRID
- Select the comp object and run SHIPFLOW
- Afterwards you should be able to compare the grid surfaces on the hull side by side with the GridSlice tool

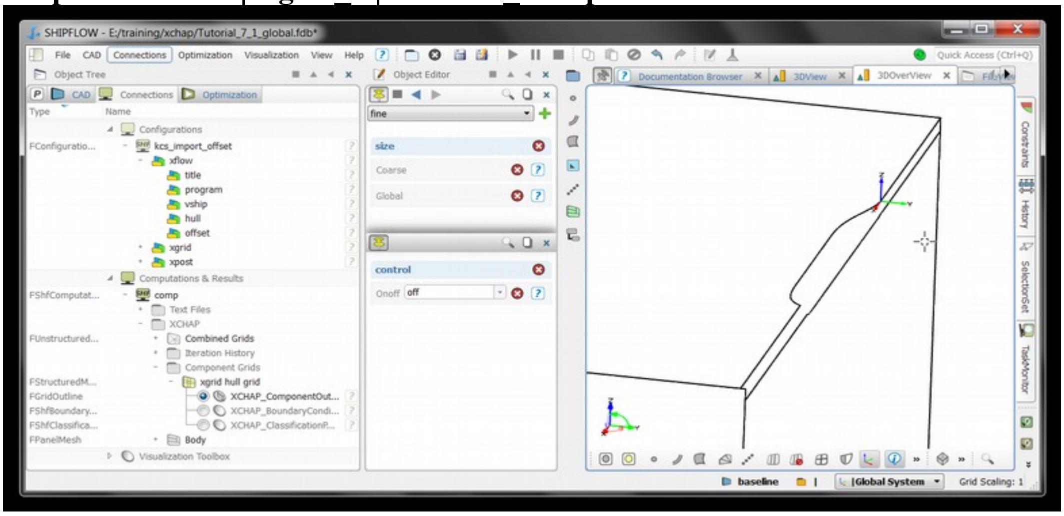


Automatic grid generation – Global approach

XGRID can generate a grid starting upstream and ending downstream of the ship. This is simply done by adding the keyword GLOBAL in the XGRID command SIZE.

We start out by importing the same configuration file as in the previous example. Changes are then applied to run only XGRID with necessary modifications.

- Close any projects in the GUI.
- Import configuration file *kcs_import_offset* from the shipflow examples directory
- Change the configuration to just run XGRID with the selected grid density
  - Select only *xgrid* in **xflow | program**
  - Remove the *xpan* configuration section
  - Add a configuration for XGRID
  - Add the *size* command and set *xgdens* to *coarse*
  - Add the keyword *global* to the *size* command
  - Add a configuration for XPOST
  - Add the *control* command and set *onoff* to *off*
- Save the project and run SHIPFLOW
- Toggle the **Object Tree | Connections | Computations & Results | comp | XCHAP | Component Grids | Xgrid_1 | XCHAP_ComponentOutline** tool active



- Create also a GridSlice visualizer as describe in the previous section

Automatic grid generation – One and two sides

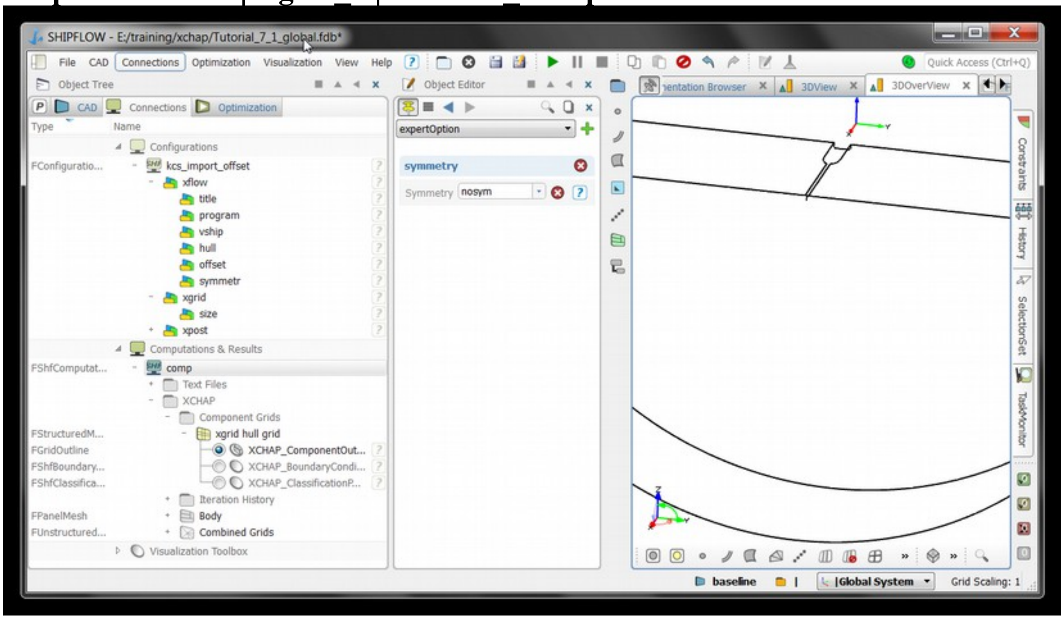
Both side of the ship must be gridded when the flow is asymmetric. This is the case in for example single screw propeller or manoeuvring simulations. XGRID will mirror the grid when asymmetry is specified in the XFLOW section of the configuration.

We start out by importing the same configuration file as in the previous example. Changes are then applied to run only XGRID with necessary modifications. Alternatively, you can continue working with the previous example and apply the modification to the current configuration.

- Close any projects in the GUI.
- Import configuration file *kcs_import_offset* from the shipflow examples directory
- Change the configuration to just run XGRID with the selected grid density
  - Select *all* in **xflow | program**
  - Add the *symmetry* command in the *xflow* section and add the *symmetry* list from which the *nosym* option can be selected
  - Add a configuration for XGRID

- Add the *size* command and set *xgdens* to *coarse*
- Add a configuration for XCHAP and set **control | maxit**=*0*
- Add a configuration for XPOST
- Add the *control* command and set *onoff* to *off*

- Save the project and run SHIPFLOW
- Toggle the **Object Tree | Connections | Computations & Results | comp | XCHAP | Component Grids | Xgrid_1 | XCHAP_ComponentOutline** tool active



- Create also a GridSlice visualizer as describe in the previous section

Optional exercise: Boundary conditions on the folded surface.

The purpose with this optional exercise is to demonstrate how the boundary conditions are set when the grid is mirrored. The part below the singularity line in the y=0 plane will become an internal grid surface and needs no boundary conditions. However, the part above the singularity line is a folded boundary surface. Two neighbouring cells on each side of the y=0 plane have a common cell face where the flow should pass through as if it was an internal cell face. XCHAP will mark these faces as internal.

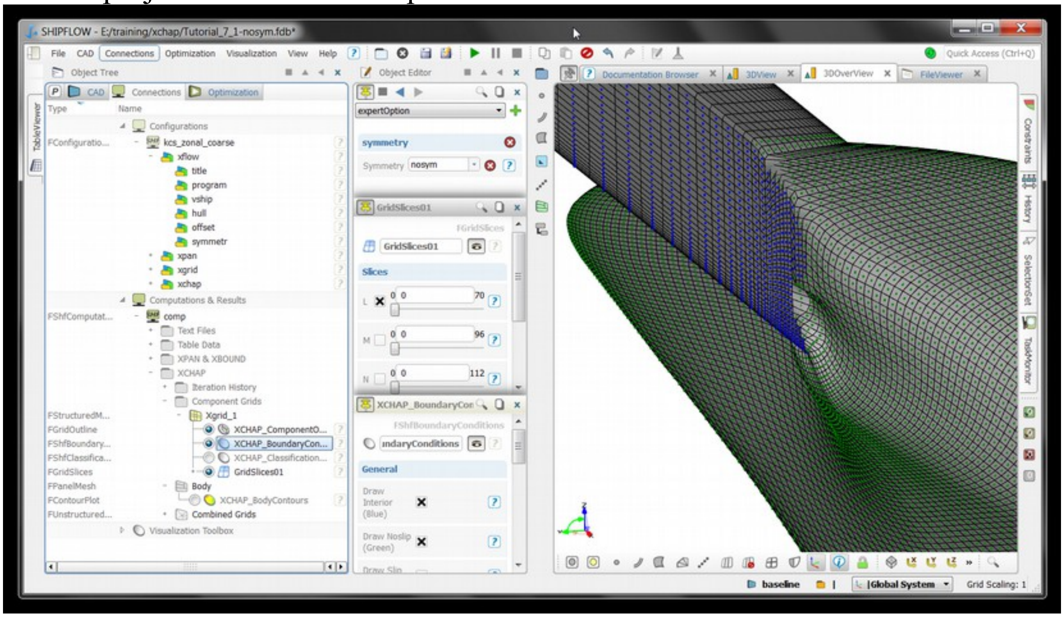
XCHAP computes the boundary conditions and needs to be run for at least zero iterations before it can be visualized. XPAN and XBOUND must also be run when we use the zonal approach. We will import the same configuration as above and modify it so all modules will be run with coarse grids.

- Import configuration file *kcs_import_offset* from the shipflow examples directory. (Alternatively you can continue from the project Tutorial_7_1.)
- Save the project e.g. Tutorial_7_1_zonal_2sides by selecting File > Save as or  click on the icon

The configuration is set up to run free surface computation with XPAN on a coarse mesh. Change the configuration to run all modules with a coarse XGRID grid and zero XCHAP iterations.

- Rename the configuration kcs_import_offset to kcs_zonal_2sides
- Select *all* in **Object Tree | Connections | Configurations | kcs_zonal_offset | xflow | program**. This will make all modules XMESH, XPAN, XBOUND, XGRID, XCHAP run

- Add the *symmetry* command in the *xflow* section and add the *symmetry* list from which the *nosym* option can be selected
- Add *xgrid* in **Object Tree | Connections | Configurations | kcs_zonal_offset**
  - Add the *size* command
  - Add *Xgdens* in the **size** command and select *coarse.* XGRID will generate a coarse grid for the zonal approach
- Add *xchap* in **Object Tree | Connections | Configurations | kcs_zonal_offset**
  - Add *maxit* in the **control** command and make sure it is set to 0
  - Add the *parallel* command
  - Add nthreads in the parallel command and set it to a suitable number of threads for your computer, for example 2 on a dual core processor
- Save the project and run the computation



Use for example a GridSlice tool to display the surface grid and the folded surface. Show noslip and interior boundary conditions with the XCHAP_BoundaryCondition tool.

Note that XPAN and XBOUND ignores the nosym command in Automatic Mode. This is indicated by the warning message:

*** XFLOW WARNING: XZ-SYMMETRY IS ENFORCED FOR STANDARD CASES

Unsymmetric cases must be set up in Manual Mode for XPAN and XBOUND. The flow at the inlet that is needed by XCHAP in the zonal approached is mirrored.

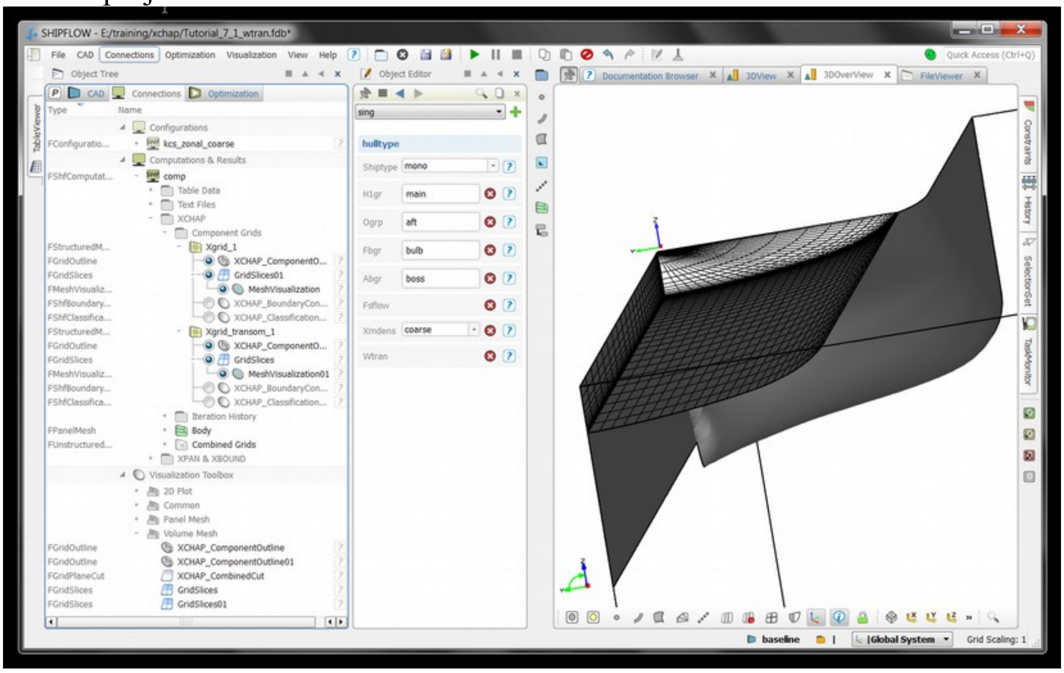
Automatic grid generation – Wet transom grid

Hulls with a large submerged transom needs special treatment to avoid poor grid quality. The normal XGRID grid would make a sudden large jump from the hull to the free surface which leads to a poor quality of the grid. A special component grid can be made that fills the domain from the transom and downstream of the ship. The hull grid will leave the hull surface and follow the outer boundary of the transom grid.

To illustrate this we will create a new configuration of the container ship with the draft is increased to 14.0 meters to create a large submerged transom.

- Import configuration file *kcs_import_offset* from the shipflow examples directory

- Change the configuration to just run XGRID with the selected grid density
  - Select *all* in **xflow | program**
  - Add the keyword *wtransom* in the **xflow** *hull* command
  - Add a configuration for XGRID
  - Add the *size* command and set *xgdens* to *coarse*
  - Add a configuration for XCHAP and set **control | maxit**=*0*
  - Add a configuration for XPOST
  - Add the *control* command and set *onoff* to *off*
- Save the project and run SHIPFLOW



Note that two grids were created in XGRID, the Xgrid_1 and the Xgrid_transom_1. You can see this under **XCHAP | Components Grids** in the Object tree. Two GridSlice tools have created in the picture above to visualize the grids.

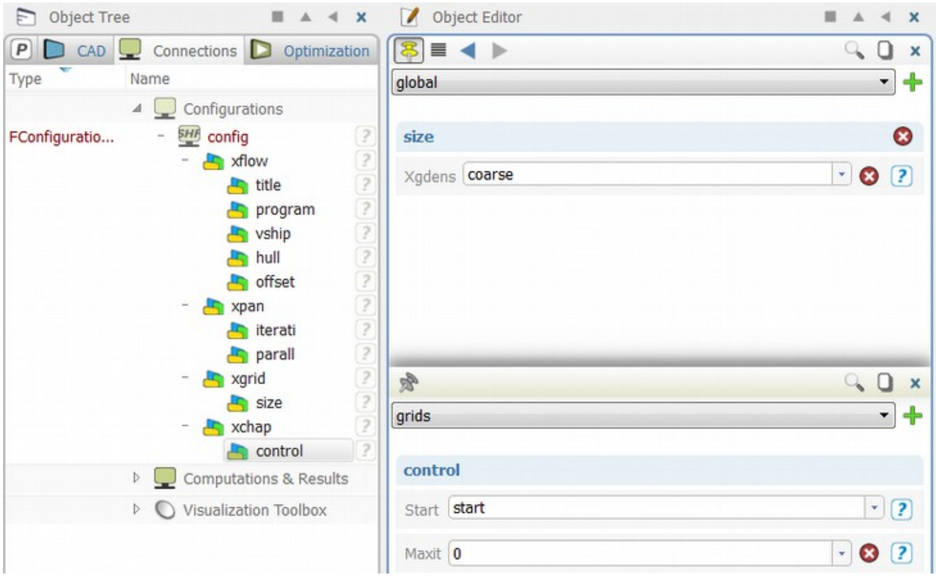
Tutorial 7 part 2 – Automatic grid generation for Twin-skeg hulls

This tutorial requires basic knowledge of the xchap and xgrid as well as post-processing features of SHIPFLOW.

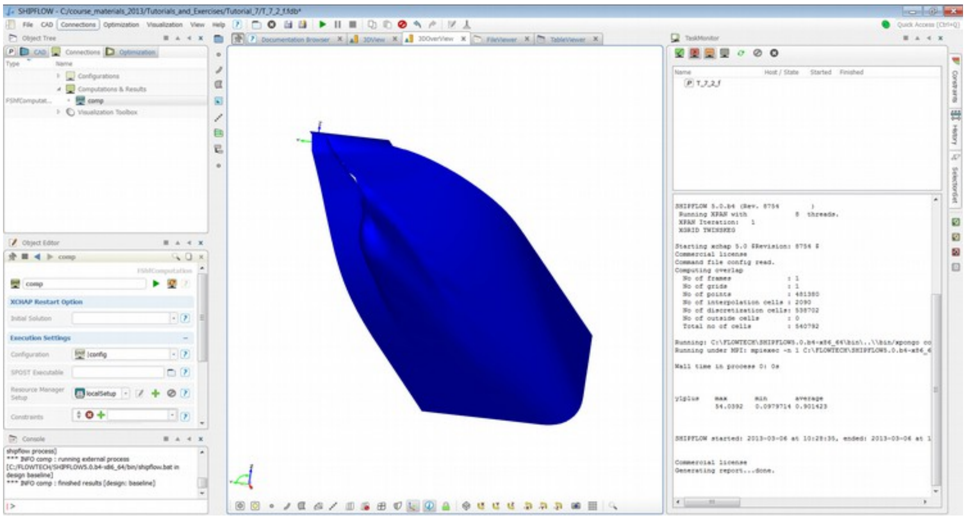
The first task is to add default coarse grid setup for xgrid and start the case with zero iterations.

ZONAL

- Import the IGES file twinskeg_igs.igs from the example folder. Use "IGES(Subset, Deprecated).
- Make a surface group
- Import the configuration file twinskeg_import_IGES from the example folder.
- Delete the filename xflow | offset | iges and drag the surface group there.
- Export an IGES(Subset, Deprecated) from the group
- Change the program to all in xflow
- Change number of XPAN iterations to 1.
- Add XGRID module configuration and set the grid size to coarse and delete global
- Add XCHAP module configuration and set number of iterations to zero in control command



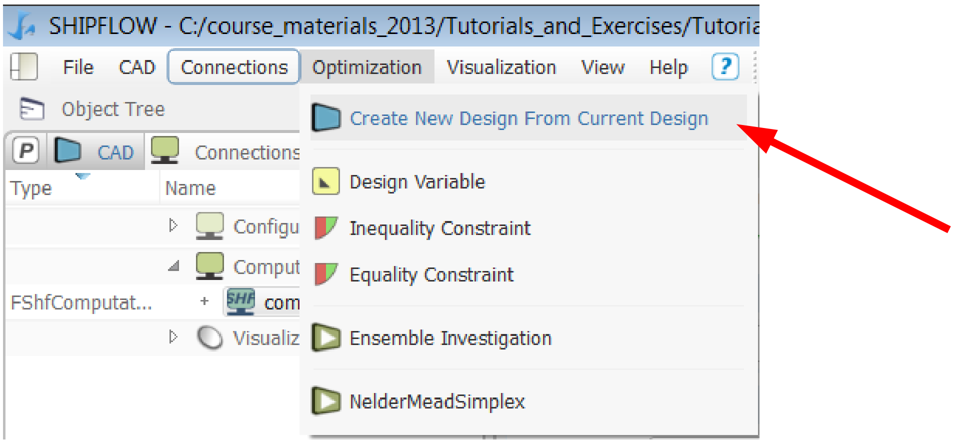
- Start the computations and display TaskMonitor window.



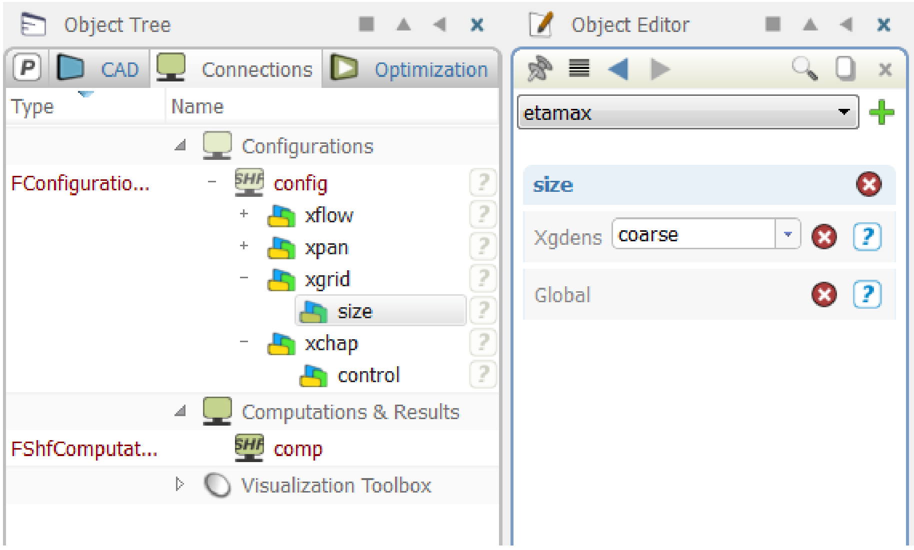
- The XPAN results can be visualised in 3DView window and the XCHAP results can be visualised in 3DOverView window by default.

GLOBAL

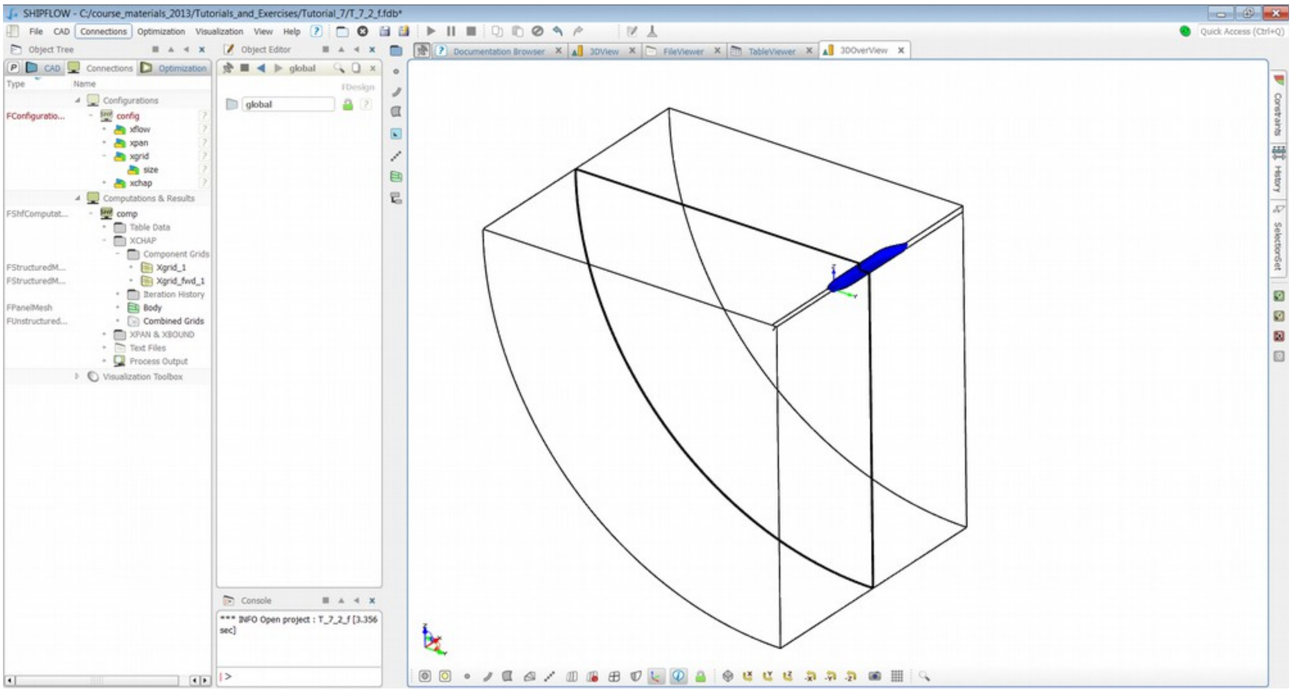
- Create a new variant in your project from menu **Optimization > Create New Design from Current Design** and name it “global”



- The global calculations of twinskeg hulls can be executed by adding the “global” keyword to the size command in XGRID module.



- Run the case
- When the calculation is finished display the component grid outlines to see the flow domain. Note that there are aft-body and fore-body grids.

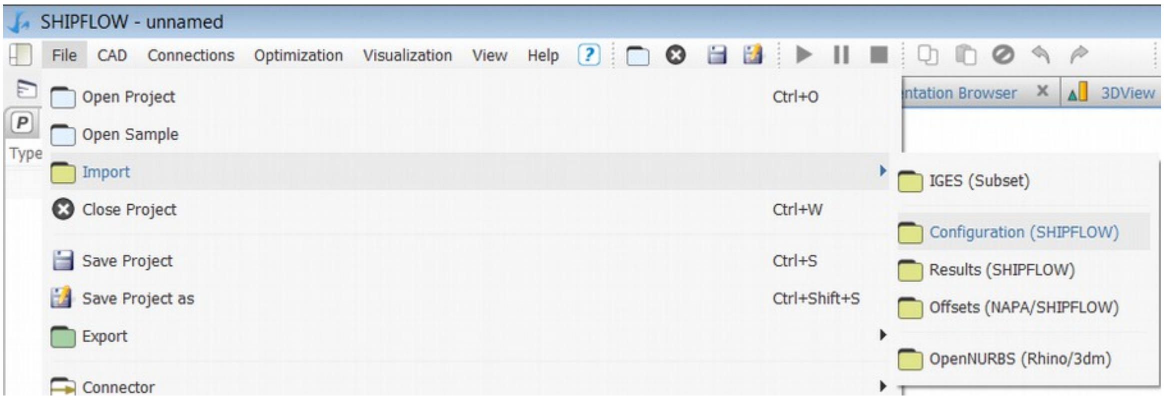


- Save the project.

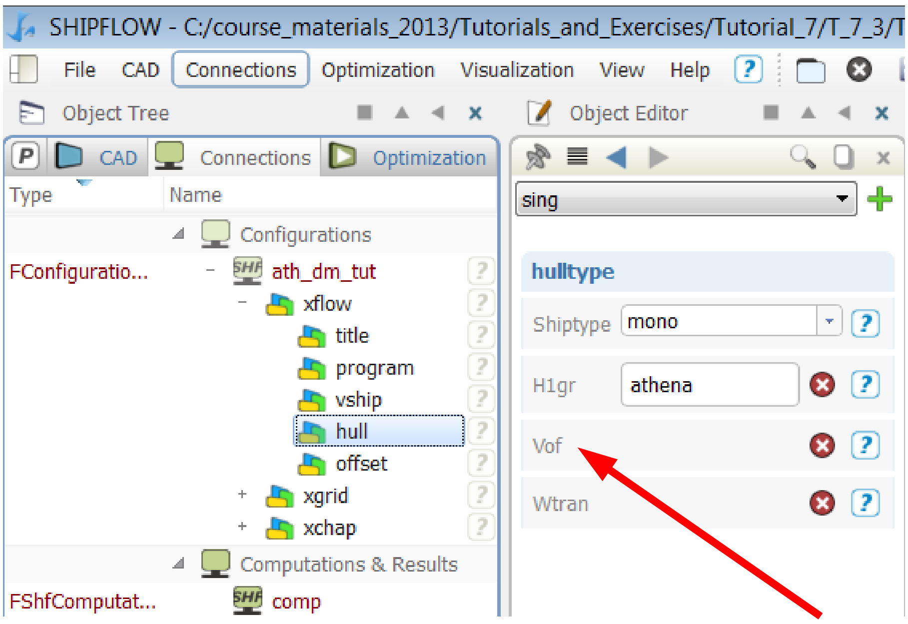
Tutorial 7 part 3 – Viscous free-surface

This tutorial requires basic knowledge of the xchap and xgrid as well as post-processing features of SHIPFLOW.

1. Import configuration file **ath_vof_tut** from the examples folder and save the project.

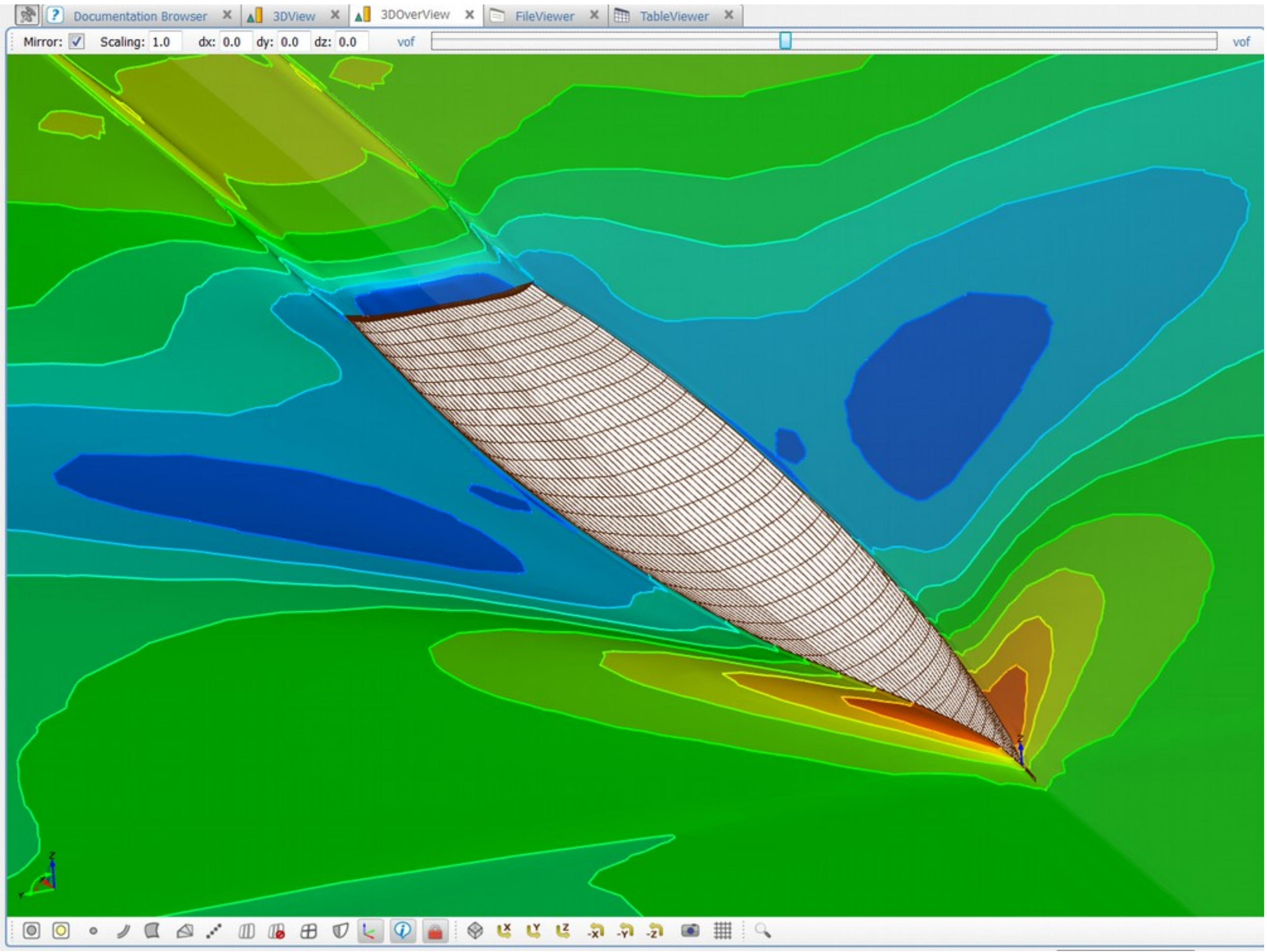


2. Note that this configuration is very similar to a regular double model case.
3. Add VOF keyword in the hull command in xflow module to trigger vof computations.



4. Set number of XCHAP iterations to 500.
5. Run this case. It will take about 10 minutes to finish this computation.

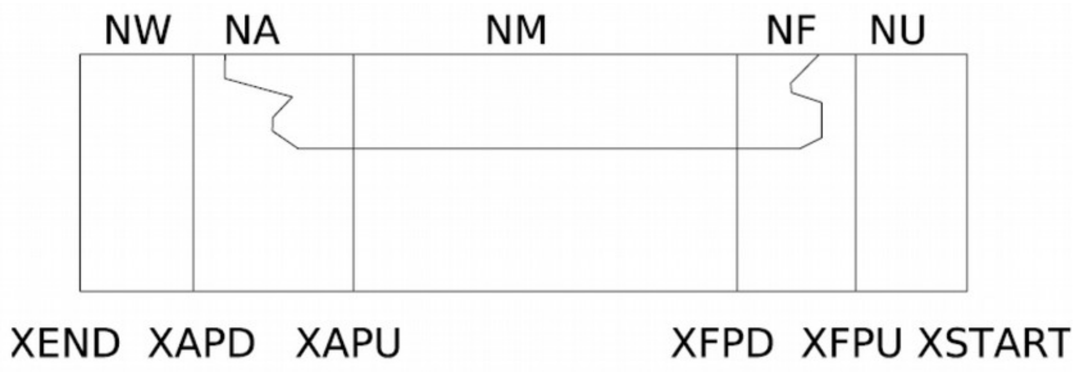
6. When the calculation is completed, display the free-surface and mirror the data set. The wave pattern should be now clearly visible. **Note that this is an extremely coarse grid and for commercial work it is advised to use the grid size set to at least medium.**



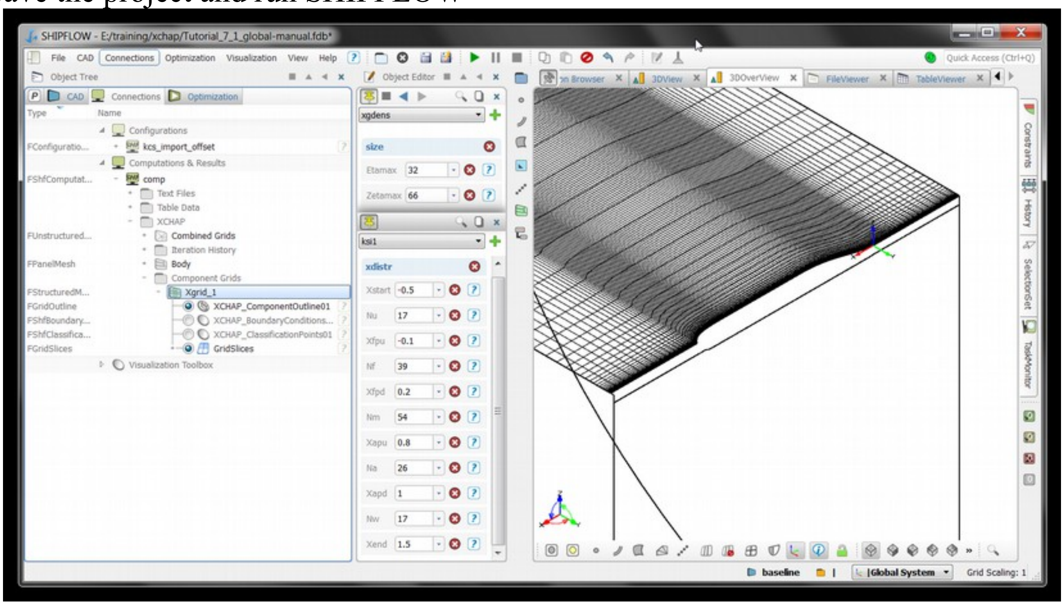
Tutorial 7 part 4 – XGRID manual control

The following part of the tutorial will show how the number of cells and the distribution can be controlled in XGRID. The KCS container ship will be used in this exercise. First we will create a global grid and after that a grid for the zonal approach.

Create a global grid with 153x31x65 cells in the longitudinal, circumferential and normal directions, respectively. Place the inlet at x=-0.5 and the outlet at x=1.5. Concentrate cells in the regions around the stem (-0.1 < x < 0.2) and in the stern (0.8 < x < 1.0). The exact distribution of cells in the x-direction is left to the user, but use approximately one third of the cells for the midship, one tenth upstream and one tenth downstream the ship. The remaining 50% of the cells are used to resolve the fore and aftbody.



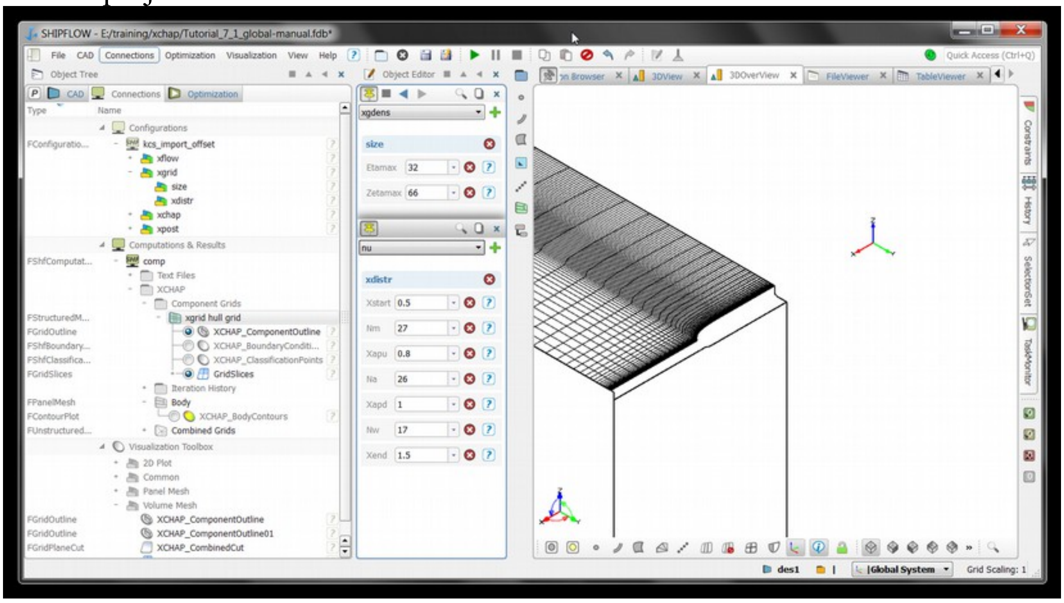
- Import configuration file *kcs_import_offset* from the shipflow examples directory
- Change the configuration to just run XGRID with the selected grid density
  - Select only *xgrid* in **xflow | program**
  - Remove the *xpan* configuration section
  - Add a configuration for XGRID
    - Add the *size* command and set *etamax=32* and *zetamax=66*
    - Add the *xdistribution* command and set the values for *xtart, xend, nu, nf, nm, na, nw, xfpu, xfpd, xapu* and *xapd* according to the recommendations above
  - Add a configuration for XPOST
  - Add the *control* command and set *onoff* to *off*
- Save the project and run SHIPFLOW



- Toggle the **Object Tree | Connections | Computations & Results | comp | XCHAP | Component Grids | Xgrid_1 | XCHAP_ComponentOutline** tool active
- Create also a GridSlice visualizer as describe in the previous section

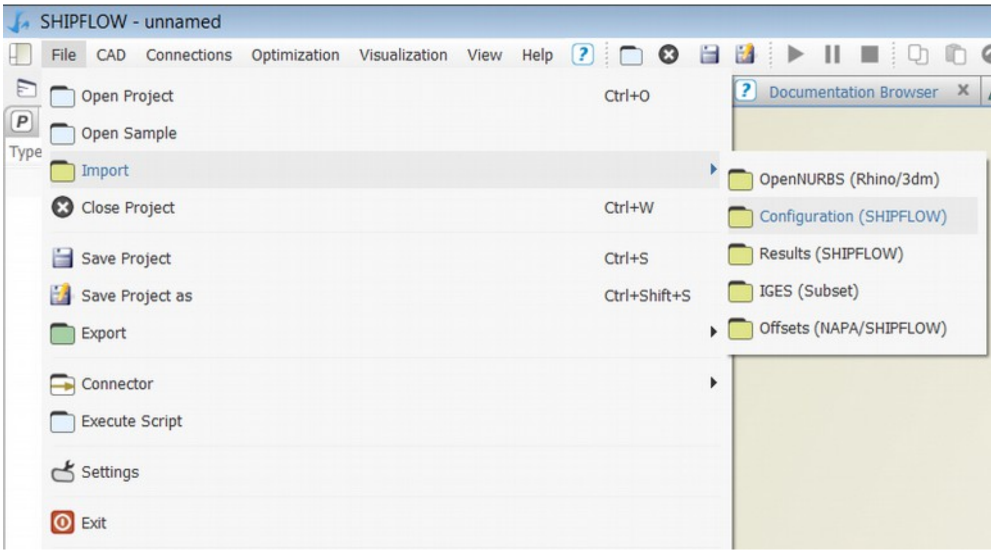
In the next exercise we create a grid of the same grid density but for the zonal approach. Let the grid start at midship.

- Create a new design variant in the current project
- Modify the *xdistribution* command to create the zonal grid. Move the *xstart* to 0.5 and remove the *nf, nf, xfpu* and *xfpd* parameters. Change *nm* about half of its previous value
- Save the project and run SHIPFLOW

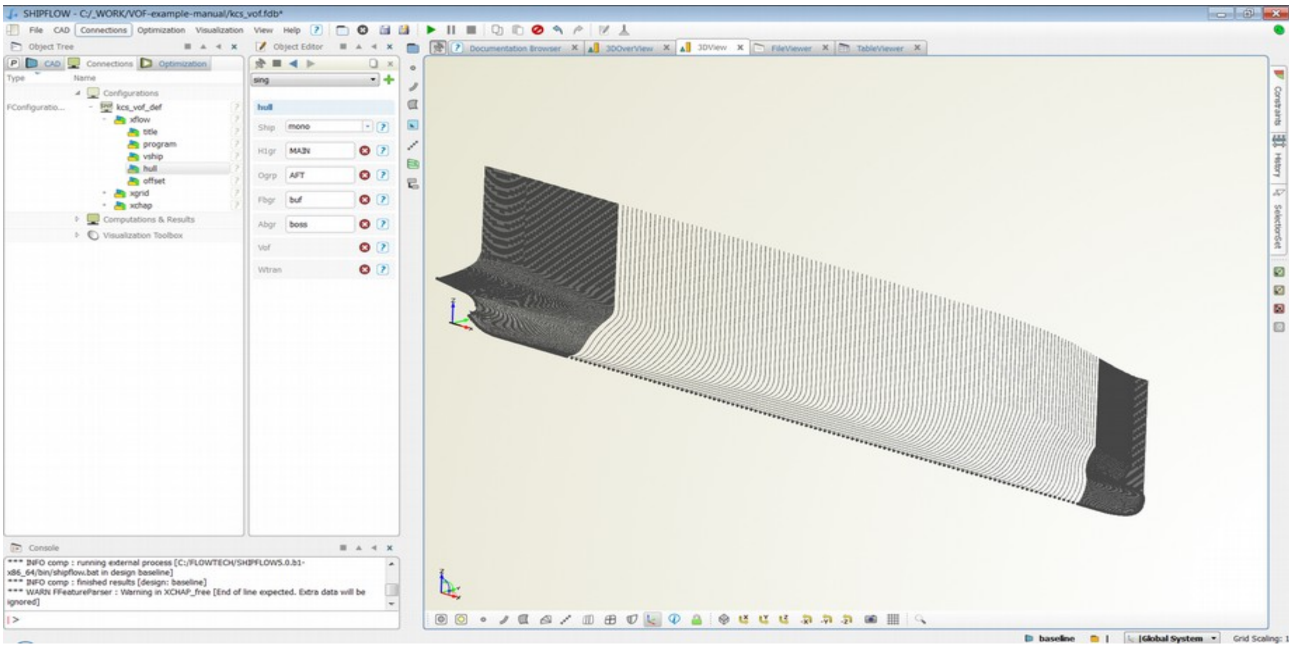


Tutorial 7 part 5 – XGRID manual control – VOF cases

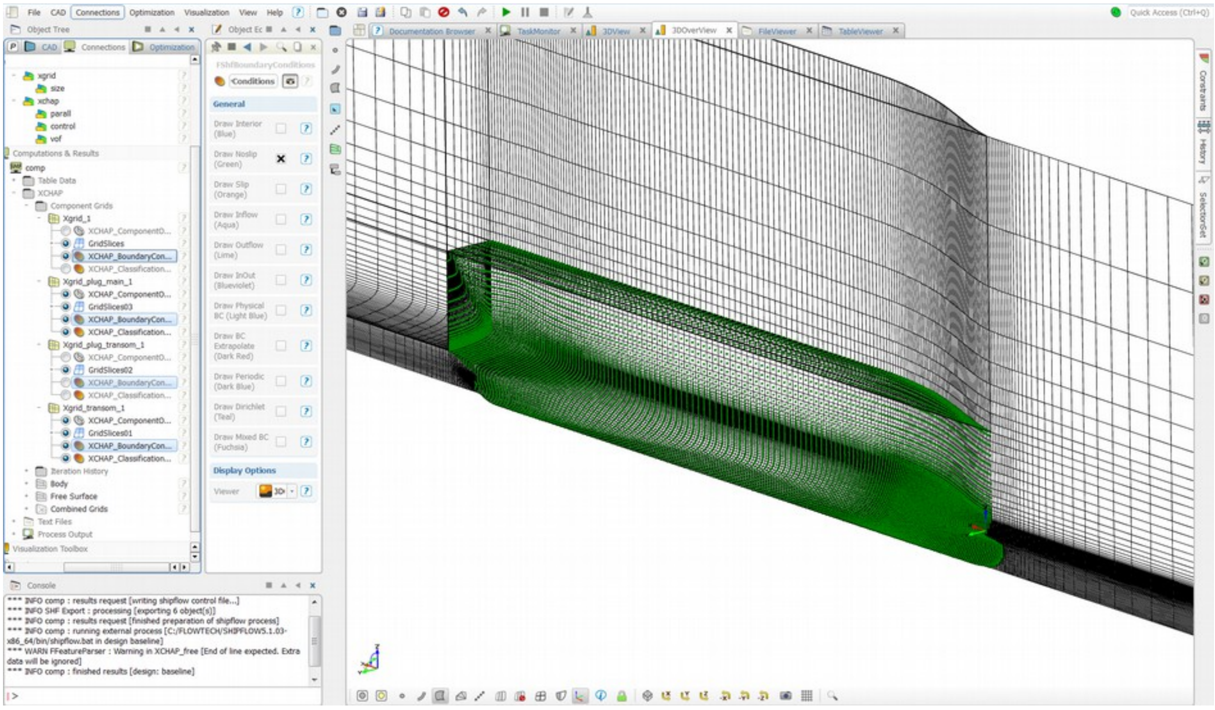
Import kcs_vof_def example SHIPFLOW configuration from /SHIPLFOW6.x.x/examples directory and thereafter save the project.



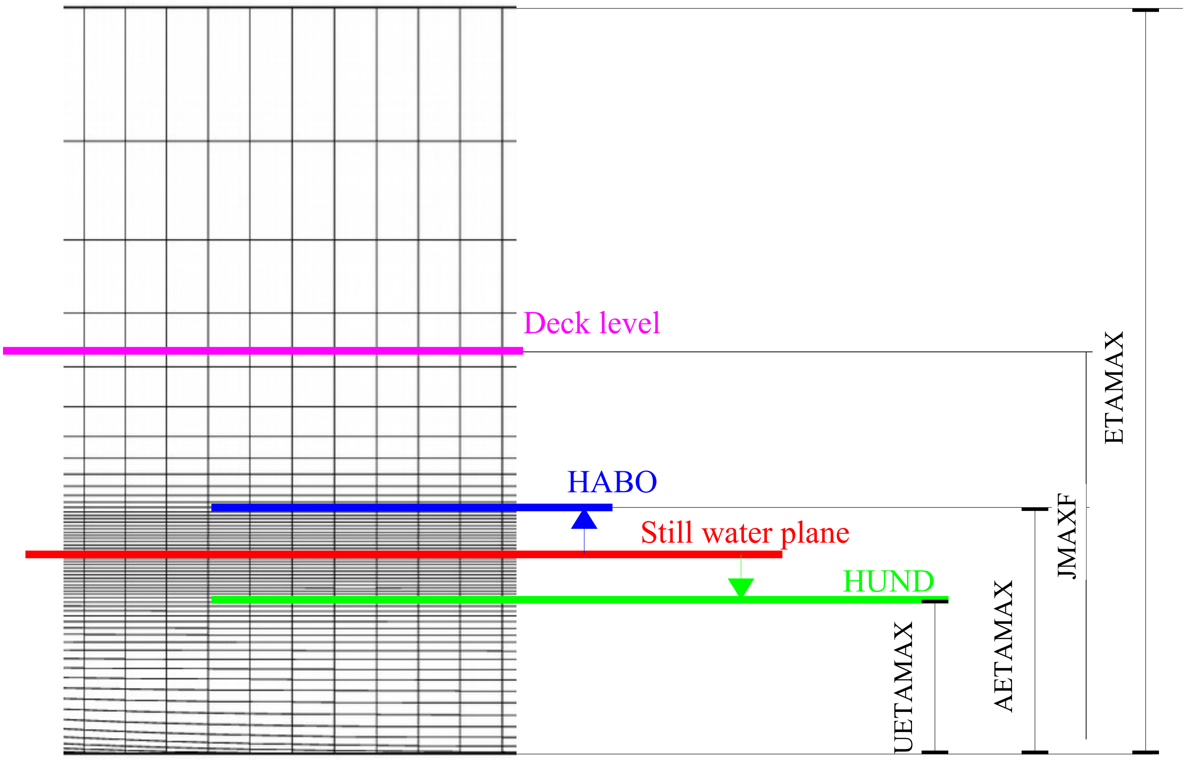
In the **Object Tree** in **Connections | Configurations | kcs_vof_def | xflow | hull** you can see the **vof** keyword which will trigger the vof computations. For many cases this is sufficient input for SHIPFLOW to run this type of cases. However, there is more input that should be known to the user in order to perform vof computations for various types of hulls.



Run it for 0 iterations and display surface mesh on the hull and transom

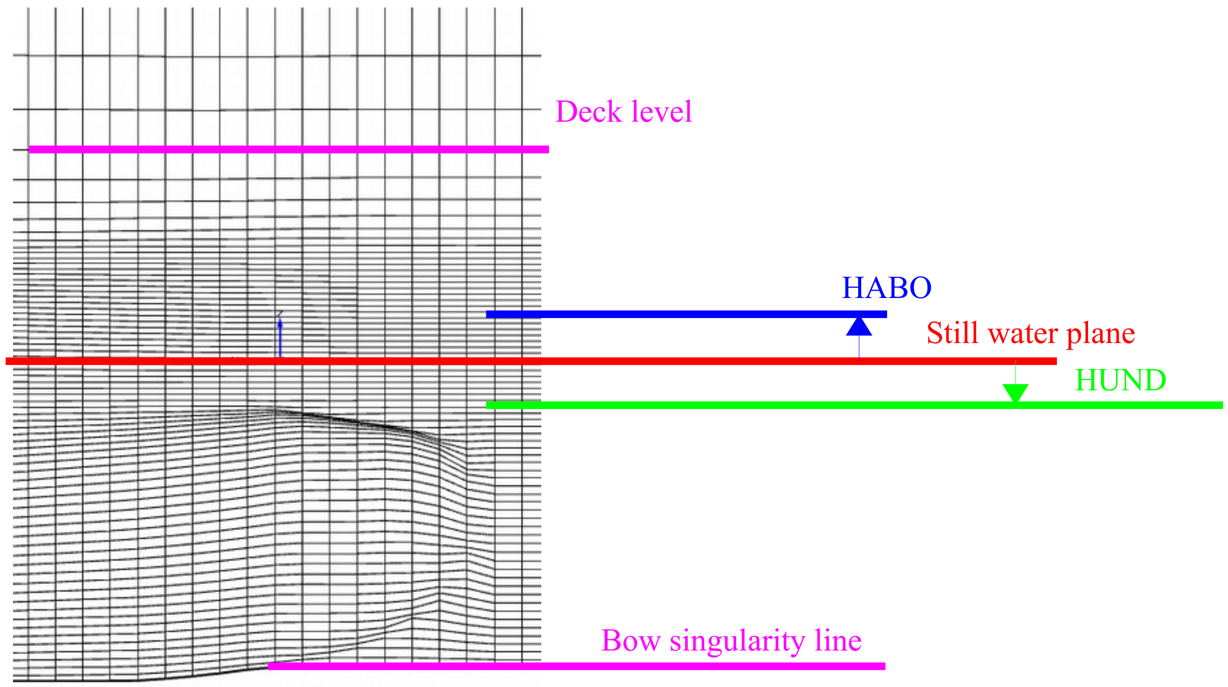


Looking on the surface mesh from a side one can see that there is a band of finer cells in vertical direction around the air-water interface. The number of cells in the girthwise direction can be set under the lower face of the refined region, in the refined region and above it using UETAMAX, AETAMAX, JMAXF and ETAMAX. The refinement dimension can be set using HUND and HABO to give non-dimensional distances from the still water plane in vertical direction.



HUND	The non-dimensional z-coordinate of the lower grid plane of the refined free surface interface. Default value is -0.005.
HABO	The non-dimensional z-coordinate of the higher grid plane of the refined free surface interface. Default value is 0.015.
UETAMAX	Number of planes in the circumferential direction up to HUND location. See section XDISTR for more information.
AETAMAX	Number of planes in the circumferential direction up to HABO location. See section XDISTR for more information.
JMAXF	Number of planes in the circumferential direction up to the deck location. See section XDISTR for more information.

The bow singularity has to be located below the HUND. For hulls without bulbous bow the singularity line may need to be specified manually e.g.: *singul(bow, xyzfwd = [0.025,0,-0.0263]),* see ath_vof_def example.



Now try to set the HABO and number of cells in girthwise direction to cover waves that are higher and keep similar cell size in the free-surface refinement band.

Tutorial 7 part 6: XCHAP post-processing

The purpose of this tutorial is to show some visualizations for XCHAP. Precomputed data sets are used throughout the tutorial. These can be found on the course DVD.

Cutting plane, contour and vector plots.

- Import CGNS file *kcs_import_IGES_zonal* from the course DVD. You find the import option in the menu **File | Import | Results (SHIPFLOW)**
- Save as a project when you are asked. (It's not mandatory for post-processing purposes)

Depending on the content in the CGNS file XCHAP results are displayed in the 3DOverView window and XPAN/XBOUND results in the 3dView window by default. The command file and output file are read when available. The force logs will also be imported when the log file is found.

We will start looking at the visualizations of the XCHAP results. Contours of Cp on the hull surface and wake fraction in the propeller disk are displayed by default when available.

Display a legend:

- Select Body | XCHAP_BodyContours
- Open General
- Add a color map, press the "+"

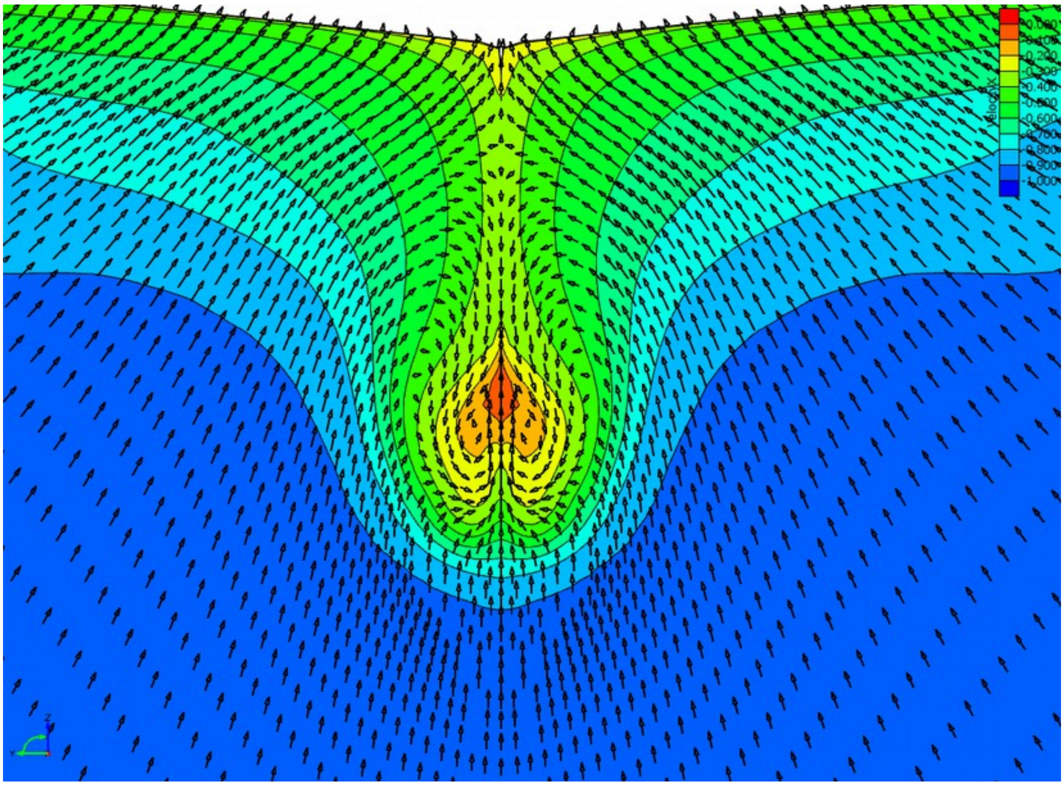
Cut the grid and show contours

- Open **Combined Grids | XCHAP_CombinedCut | XCHAP_CombinedCutContours**
- Activate visualization
- Select mapped data "VelocityX"
- (Add a legend, move it in the y-direction to Y=0)

- Move the cutting plane to the propeller location
- Add one cutting plane at 0.1*230
- Add one more plane at 0.2*230
- Switch to Colored Isolines
- Choose isolines from -1.0 to 0.0 with 0.1 increments

Display velocity vectors

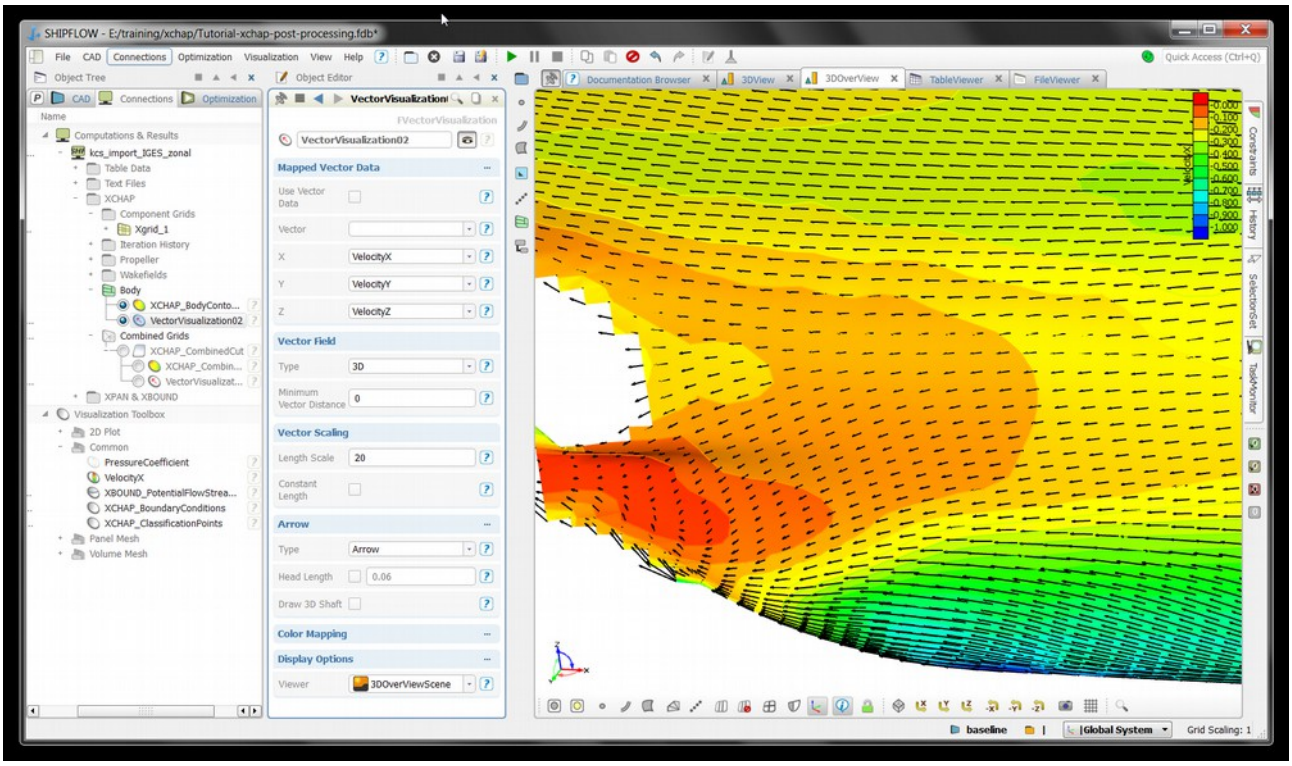
- Hide the propeller contours
- Keep the cut in the propeller plane and remove the other two
- Change view so the hull is shown from behind (-x)
- Choose "Black Isolines on Colored Surface" for the CombinedCutContours
- Select XCHAP_CombinedCut and add a VectorVisualization
- Change the following in the VectorVIsualization editor:
- Vector Field Type: Tangential to Surface"
- Minimum Vaector Distance to 0.2
- Length Scale to 2
- Arrow Type to Arrow
- Vector Color to Black
- Mirror the picture



- Open Visualization Toolbox and hide the PressureCoefficient legend

Display velocity vectors on the hull surface

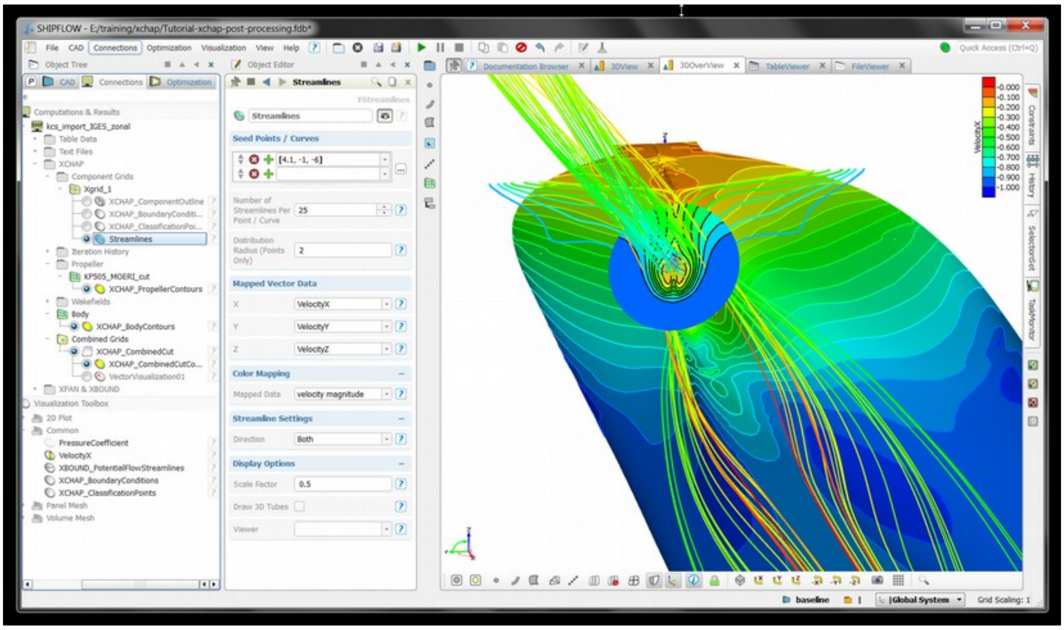
- Hide all visualizations but **Body | XCHAP_Body_Contours**
- Select Body and create a new VectorVisualization tool
- Change the Length Scale to 20, the Arrow Type to Arrow and set the Vector Color to Black
- Zoom in close to the stern



- Try also to use a Constant Length for the vectors and apply a suitable Length Scale

Trace and display streamlines

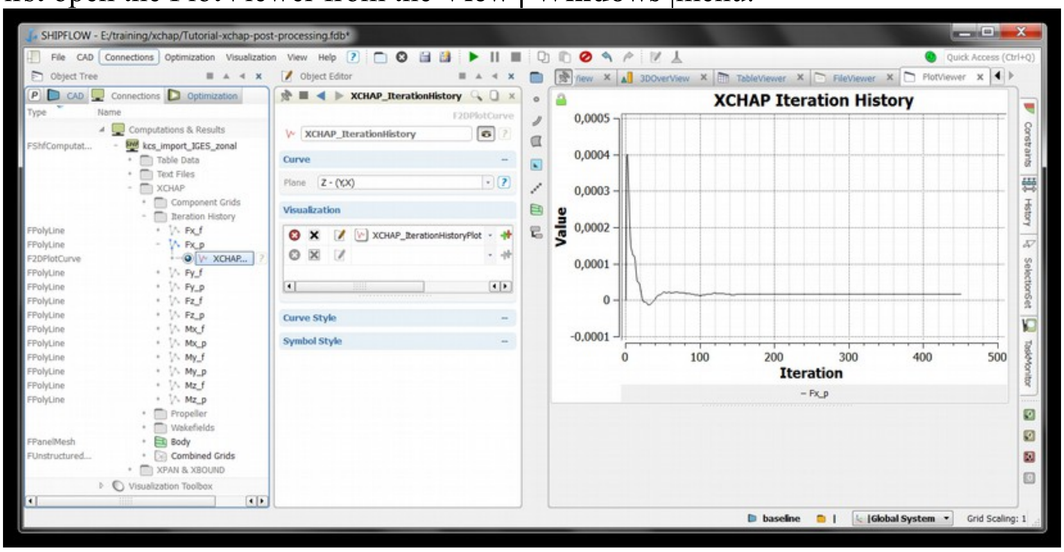
- Turn off XCHAP_VectorVisualization
- Activate XCHAP_PropellerContours and XCHAP_BodyContours
- Select Component Grids | XGRID_1 and create a Streamlines tool
- Set Seed Points to [4.1,-1,-6] and the Number of Streamlines Per Point to 25 and the Distribution Radius to 2



- Set the Scale Factor to 0.5 in the Display Options

Visualize the convergence history of XCHAP.

- First open the PlotViewer from the **View | Windows** |menu.



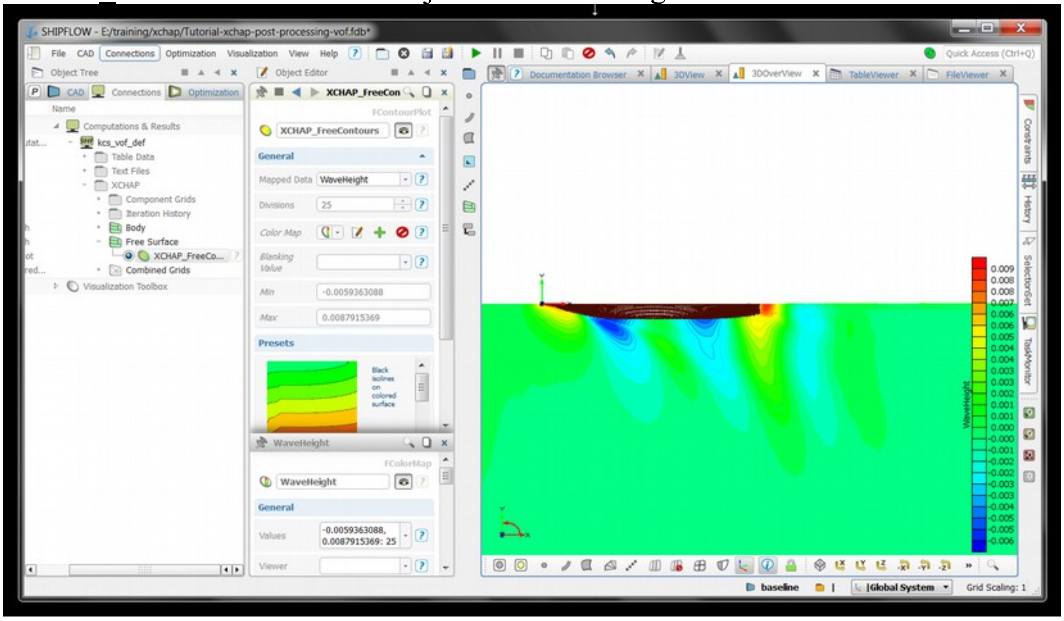
- Next open the **Computations & Results | kcs_import_IGES_zonal | XCHAP | Iteration History | Fx_p** and toggle it on. This quantity is the integrated pressure force in the x-direction and is the most suitable quantity to judge the convergence of the computation. The notations of the integrated forces and moments are F=force, M=moment, p=pressure, f=friction and x,y,z are the directions.

Visualization VOF

This tutorial will show how to visualize the free surface and wave cuts for a viscous free surface solution from XCHAP. The result files are call named kcs_vof_def and can be found on the course DVD. This computation is made with a very coarse grid (xchap|size(VCOARSE)).

Wave contours

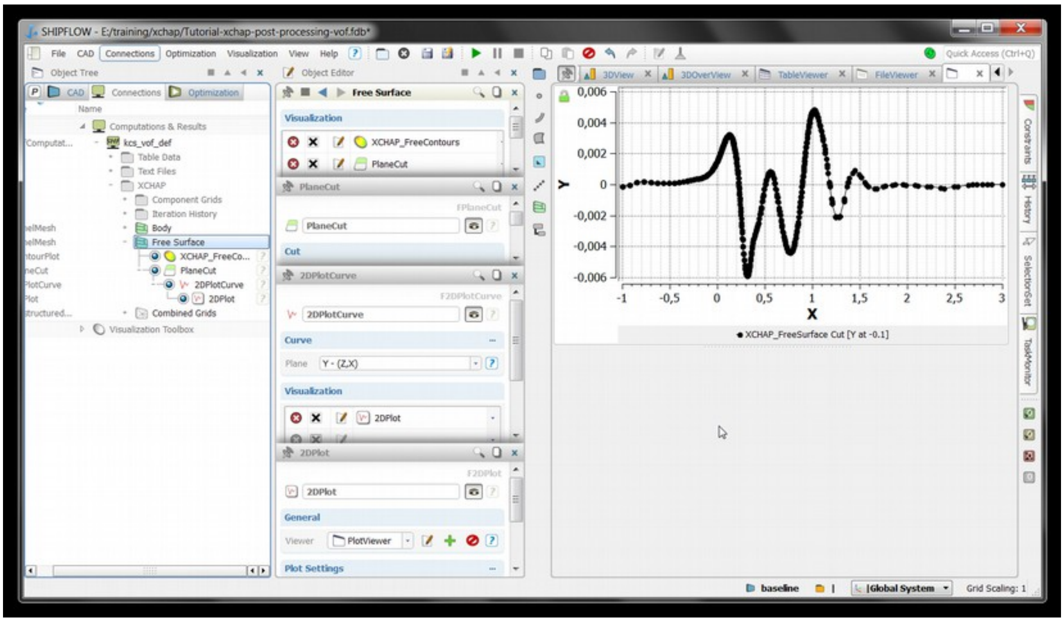
- Open a new session or close the current.
- Import the solution by selecting **File | Import | Results (SHIPFLOW)** in the menu. Open the kcs_vof_def from the course DVD.
- The wave pattern is displayed by default in the 3DOverview window. Select a top view
- Select the **Computations & Results | kcs_vof_def | XCHAP | Free Surface | XCHAP_FreeContours** in the object tree and change the number of Divisions to 25.



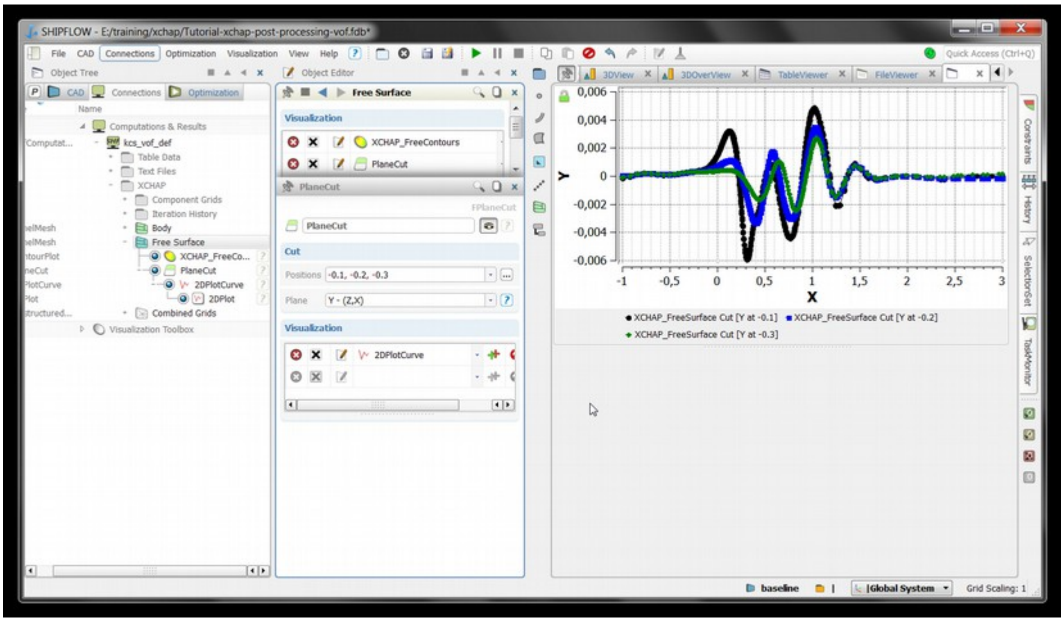
- Create also a legend by adding a Color Map in the General section

Wave cuts

- Create a PlaneCut in the Free Surface object and set the Y-value to -0.1
- Create a 2DPlotCurve in the new PlaneCut object
- Create a 2Dplot in the 2DPlotCurve object



- Open the the PlotViewer window from the Menu **View | Windows | PlotViewer** which shows the wave cut at Y=-0.1

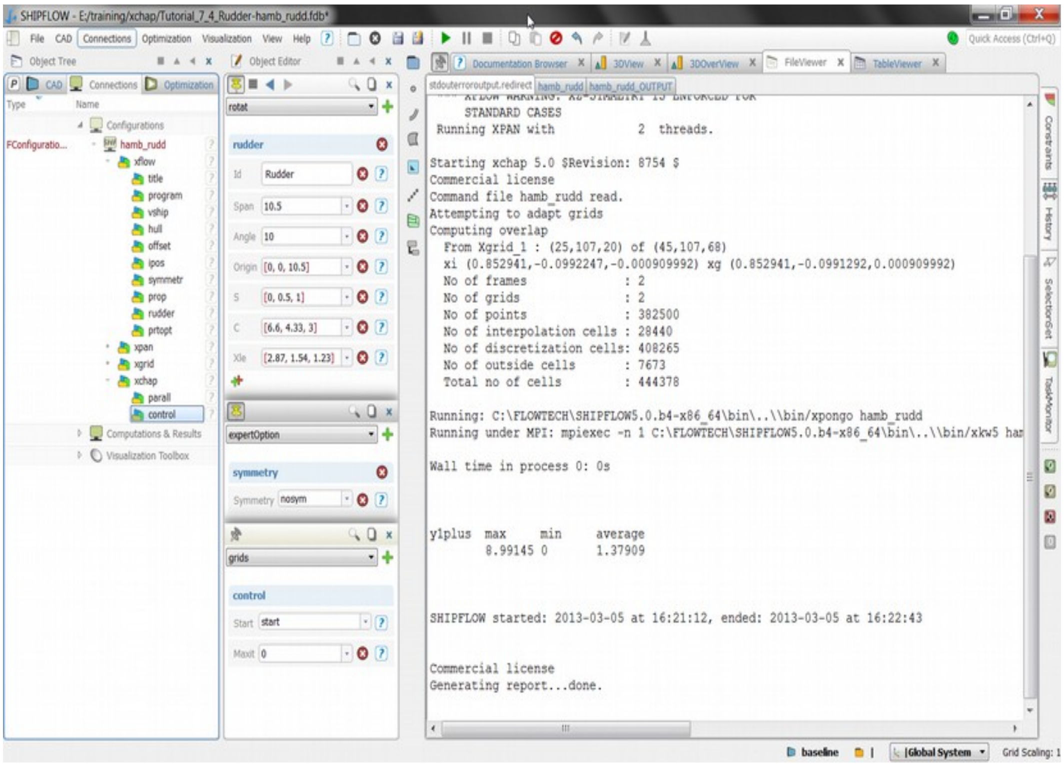


- Add a series of of Y-positions -0.1,-0.2,-0.3 in the PlaneCut tool

Tutorial 7 part 7 – XCHAP Rudder

The following part of the tutorial shows a simple example of a set up for a ship with a rudder. The hull grid is made very coarse to simplify the execution of the tutorial. A real case must be run with a finer grid.

- Import configuration file *hamb_rudd* from the shipflow examples directory
- Change the following in the XCHAP section
  - Remove the ACTUATOR command
  - Set the number of iterations to zero
- Change the following in the XFLOW section
  - Add an ID to the RUDDER command and assign it the name “Rudder”
- Study the parameters for the rudder
- Check that the SYMMETRY is set to NOSYM

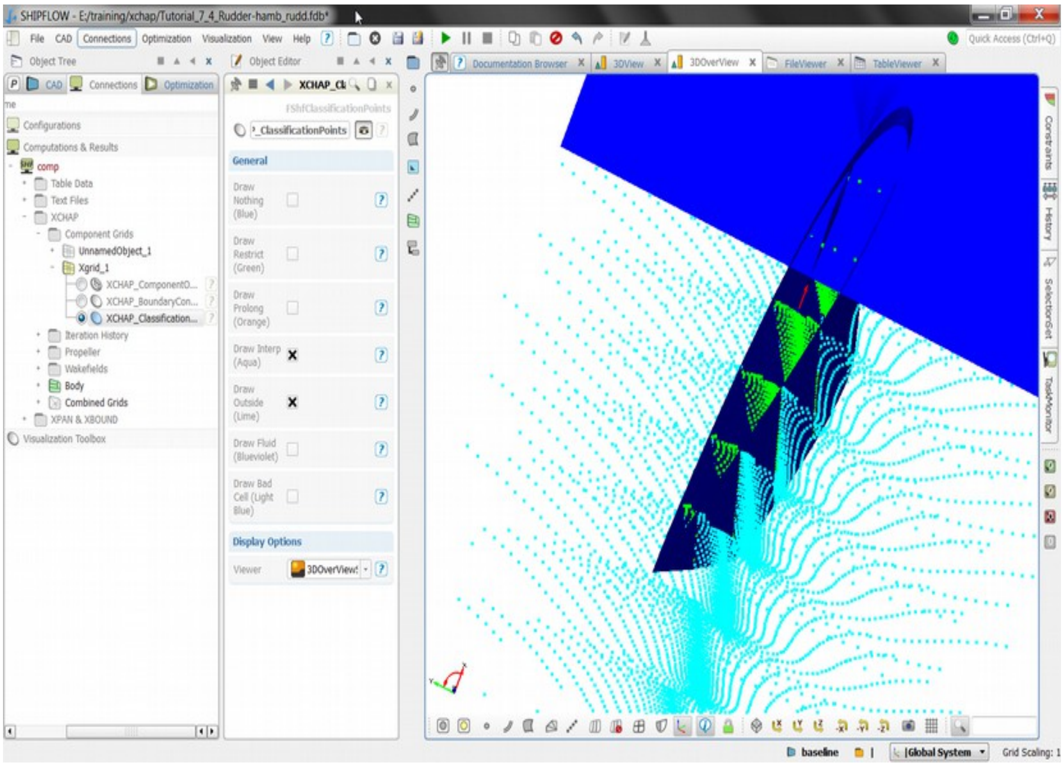


- Run XCHAP and observe the output

XCHAP was run with the maximum number of iterations set to zero. The program exited after the grids, the overlapping grid classification and the initial flow field were computed. The number of discretization, interpolation and outside cells are printed. We can see that in this case about 28000 cells were used for interpolation between the grids, 8000 cells are located inside the rudder or ship and the remaining 400000 cells were used for the discretization.

Next we will see how the classification can be visualized in the different component grids.

- Select **Object Tree | Connections | Computations & Results | comp | XCHAP | Component Grids | Xgrid_1 | XCHAP_ClassificationPoints** and activate first the interpolation points. Cells in the hull grid close to the rudder are all classified as interpolation points.
- Next activate the outside points and change the view angle so that you can look inside the rudder from above. Cells in the hull grid inside the rudder are marked as outside cells.



This tutorial gives a first impression on overlapping grids and how it is used to set up complexed geometries. More information on appendages and overlapping grids can be found in the SHIPFLOW Design Tutorials – Advanced.