

# **SHIPFLOW DESIGN Tutorials**

## **ADVANCED**



## Table of Contents

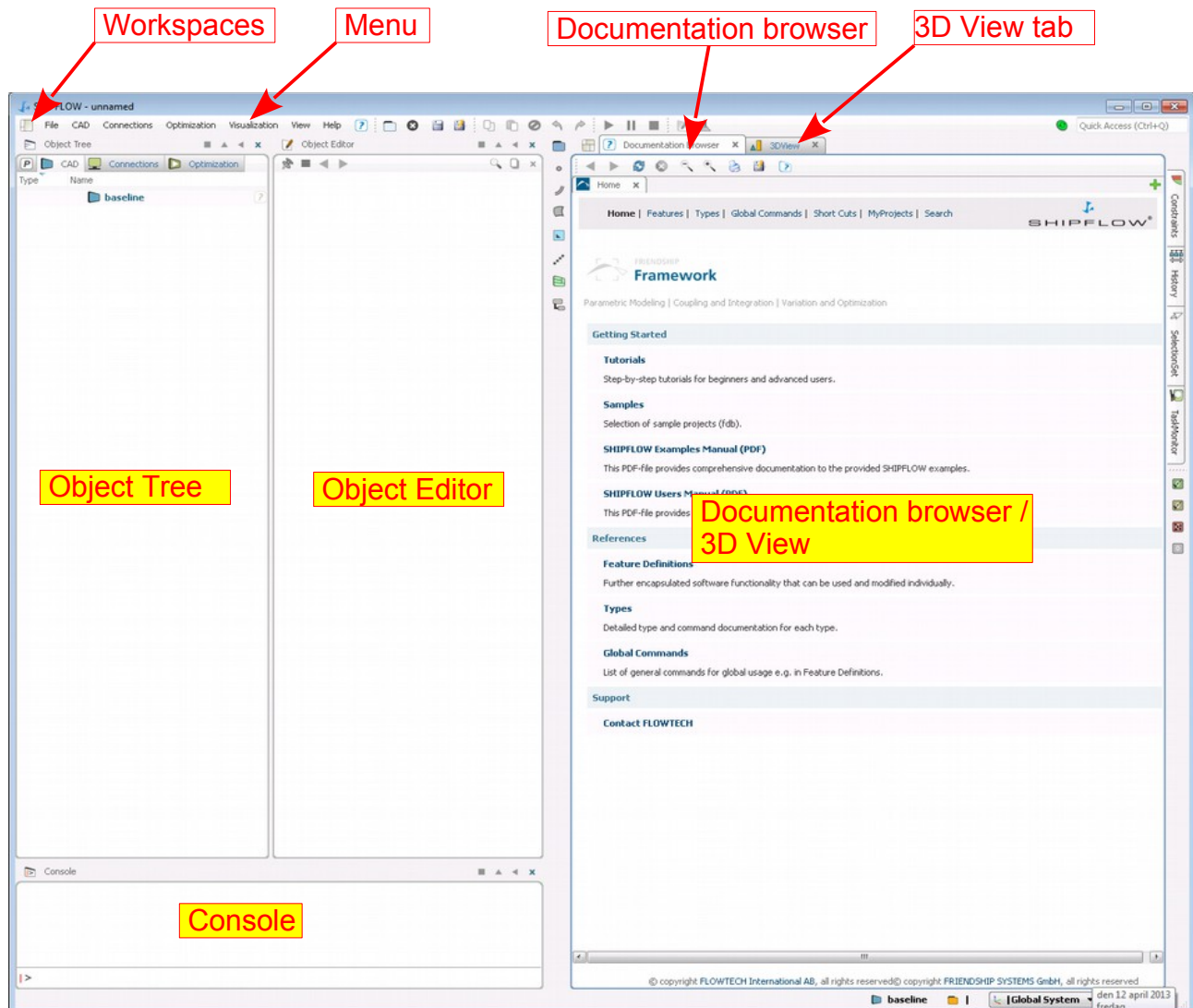
Introduction to SHIPFLOW-FRIENDSHIP DESIGN environment.....	5
Tutorial 1 part 1 – Self propulsion simulations.....	7
Tutorial 2 part 1 – Forebody optimization – bulb – Delta Shift.....	13
Tutorial 2 part 2 – Ensemble Investigation .....	24
Tutorial 3 part 1 – Forebody optimization – shoulder – Surface Delta Shift.....	26
Tutorial 3 part 2 – NSGA-II Optimization.....	34
Tutorial 4 part 1 – Appendage modelling – overlapping grids.....	36
Tutorial 5 part 1 – Complex Appendage modelling .....	43
Tutorial 5 part 2 – Appendage optimization.....	48



## Introduction to SHIPFLOW-FRIENDSHIP DESIGN environment

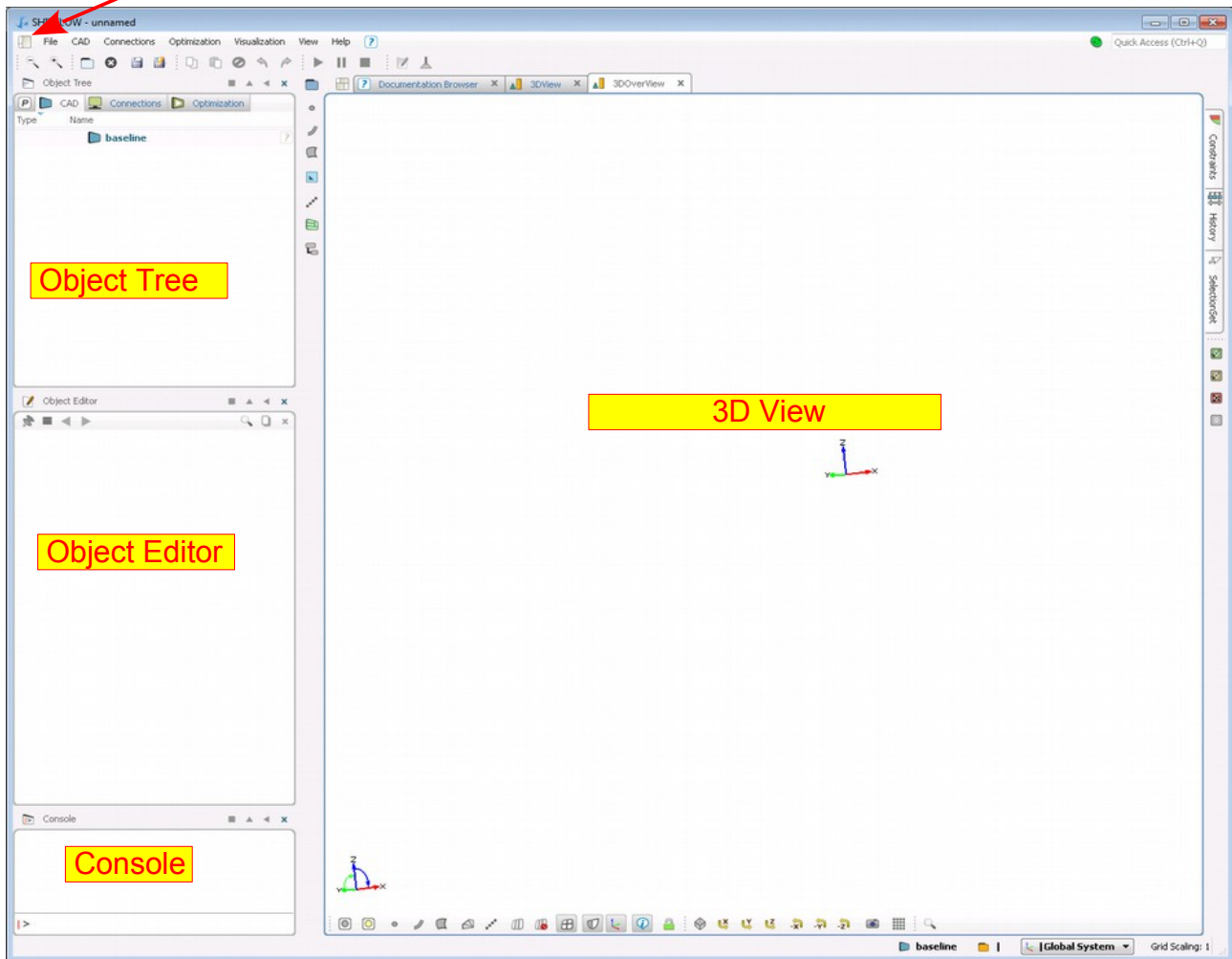
- First start of the SHIPFLOW Design application (Windows OS)

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



- Customized workspace

Saved custom workspace



## ***Tutorial 1 part 1 – Self propulsion simulations***

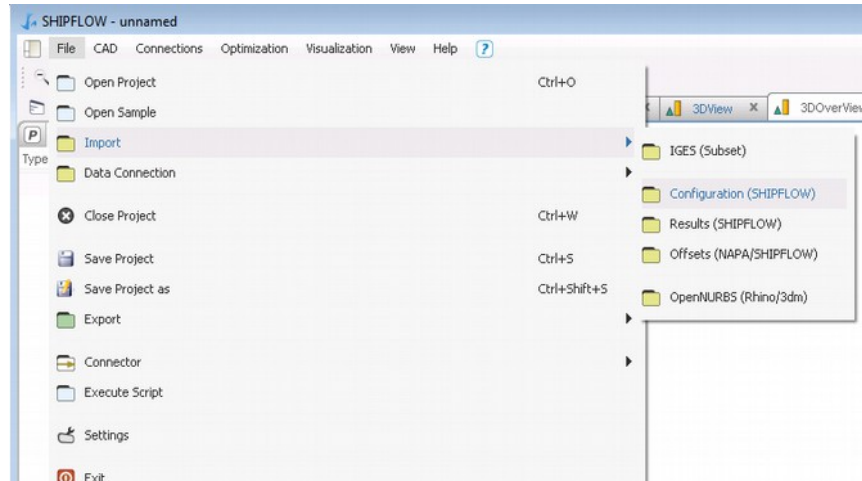
This exercise is a practical introduction to self propulsion simulations with SHIPFLOW. It requires knowledge about XPAN, XBOUND and XCHAP. It also requires basic knowledge about propeller modelling with SHIPFLOW, for example the tutorial in ho\_propellers.

The examples are set up using the zonal approach with very coarse grids to reduce computational times during the exercise. Real cases must be performed with a much finer grid resolution.

There are three simulations required to evaluate the propulsive factors, a towing (resistance), an open water (POW), and self propulsion simulation. All three simulations will be set up during the exercise.

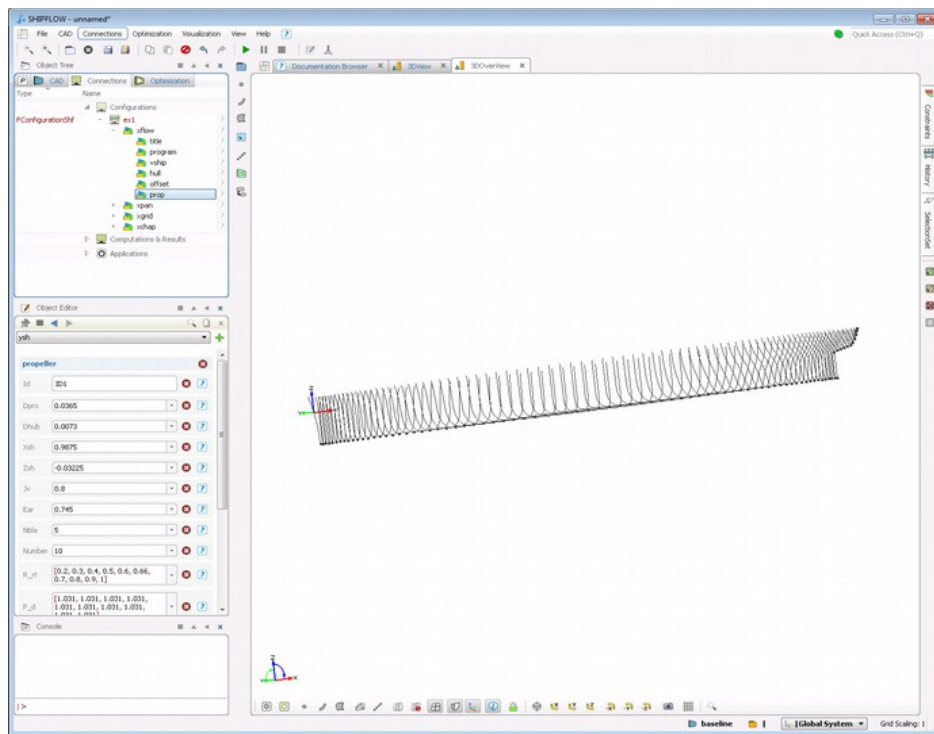
There are two ways to perform the self propulsion simulations, the first is fully automatic where the program executes resistance, pow and selfpropulsion in a series and the second way is to run the three parts separately. The manual variant is described in handout ho-selfpropulsion.pdf that can be found on the course DVD.

1. Start with importing the configuration “ex1” from  
**DVD:\AdvancedTraining\Tutorial\_Advanced\_1\source**

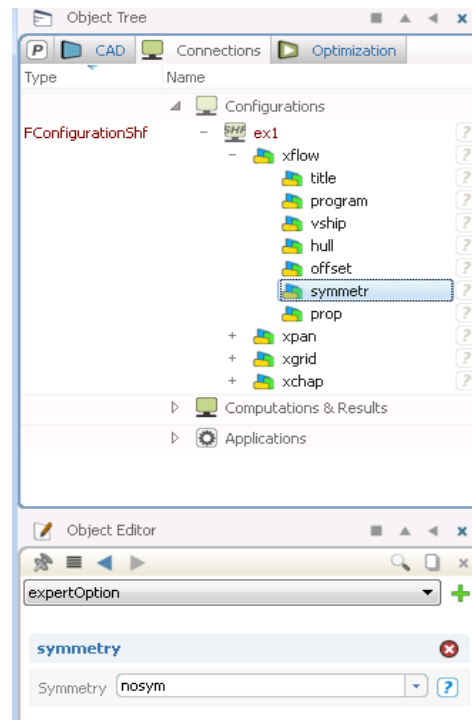


2. Save the project in your working directory

- Now check the configuration, see that there is already a propeller geometry input but no propeller model defined.

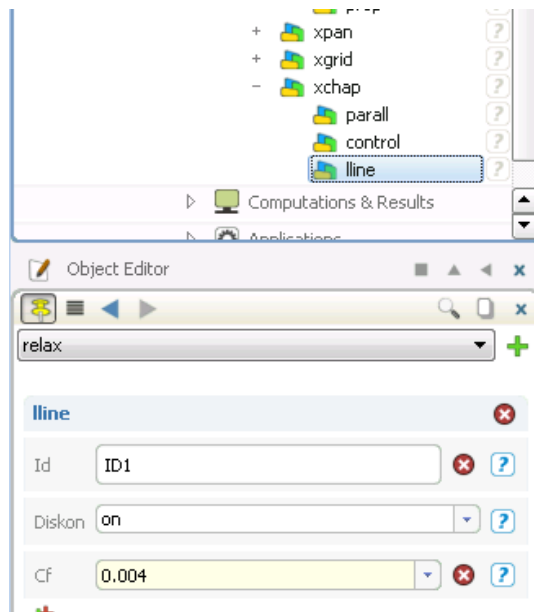


- Since the propeller adds a non symmetrical flow in the domain it has to cover both sides, therefore one should add `symm (nosym)` command in the `xflow` section as below

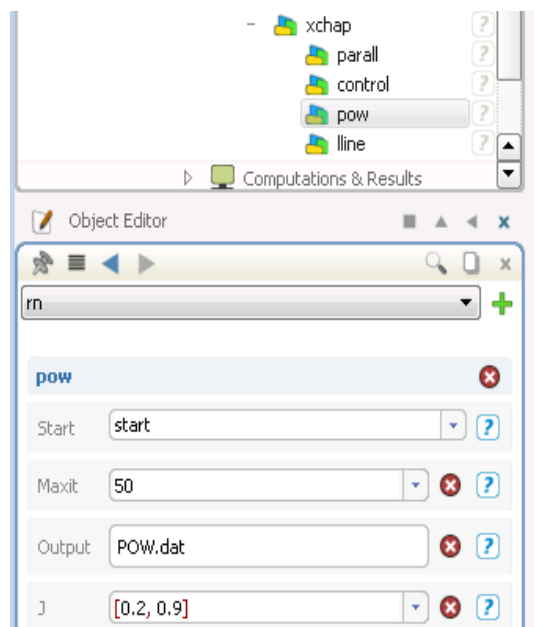




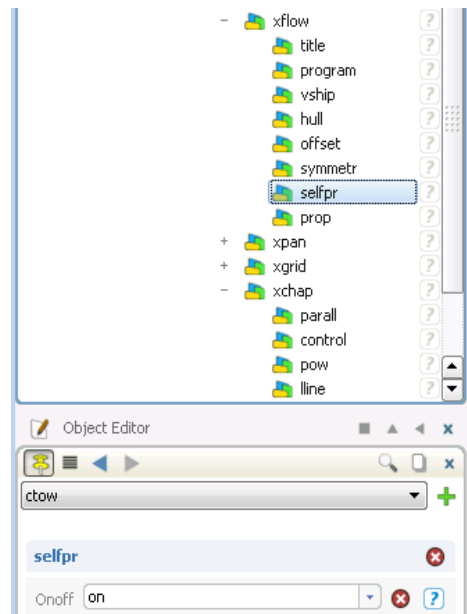
5. The next step is to add the propeller lifting line model in the xchap section, make sure that the id matches the propeller id. The cf parameter sets the frictional coefficient for the propeller blades at its local Reynolds number.



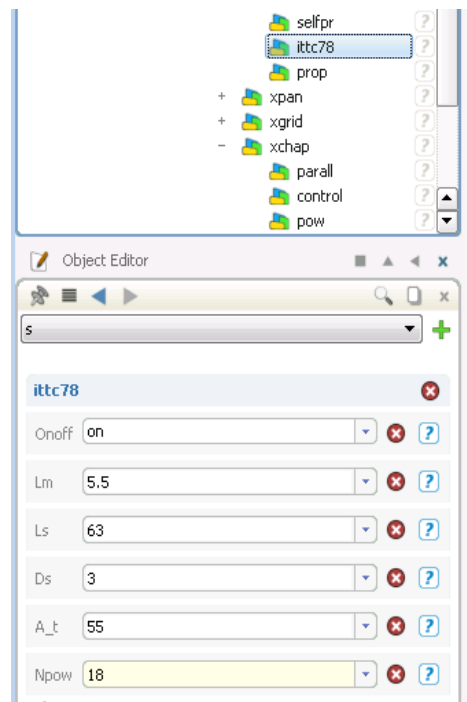
6. Add the POW command and select JV=[0.2,0.9] (recommended is to cover the whole working range of the propeller with intervals not larger than 0.1). Set START to start. Limit the number of iterations to 50 with the keyword MAXIT in the POW command (recommended is 500). Set the name of the POW output file with the keyword OUTPUT to POW.dat



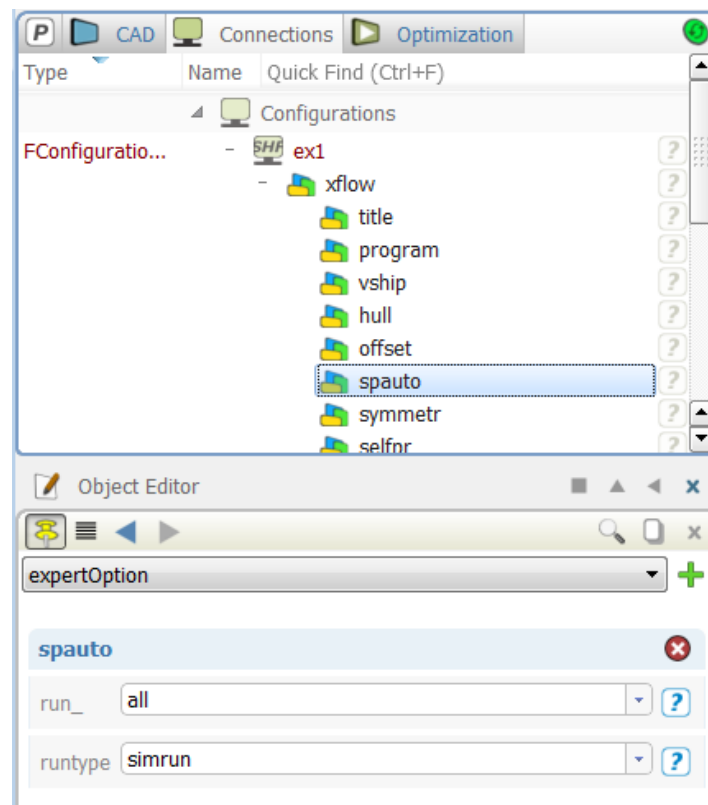
7. Add the SELFPROPULSION command and set it to ON. The CTOW value will not have to be set since it will be automatically calculated by the ITTC78 command that we will set further down.



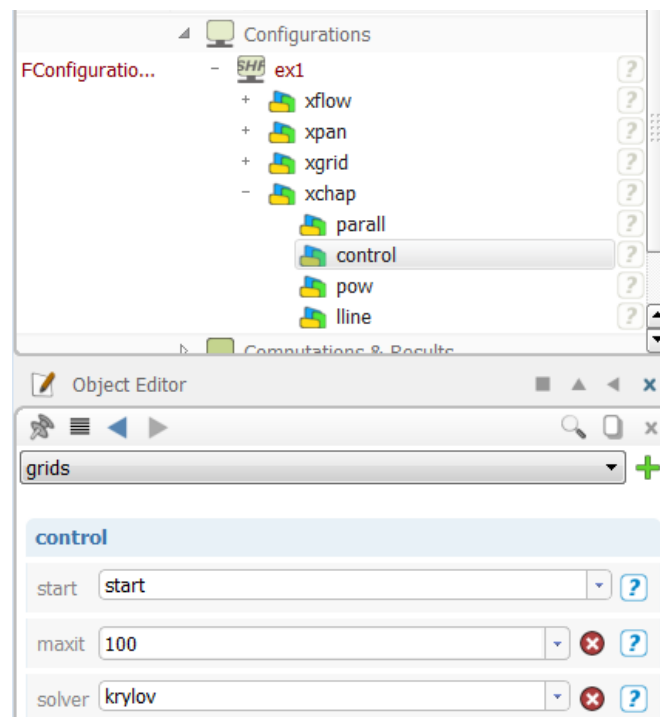
8. Add the ITTC78 command and set it to ON. In order to extrapolate data to full scale some data has to be set here:
1. LWL in model scale  $LM=5.5$
  2. LWL in full scale  $LS=63.0$
  3. Ship propeller diameter  $DS=3.0$
  4. Maximum transverse area above the waterline  $A_T=55$
  5. Model propeller open water RPS  $NPOW=18$



9. In order to run the self propulsion automatically add spauto( all, simrun ) in xflow section.



10. Optionally a new, faster solver can be used in SHIPFLOW versions 6.0 and newer. In xchap| control add solver ( krylov ) like in the picture below.



11. Now the configuration is ready to run, check that the number of iterations is set to 200, save the project and start the computations. (The computations should take approximately 15 minutes on a 2.0Ghz laptop with 2 processes.)
12. After the computations are finalized all the results are written to id\_OUTPUT file and also a report is prepared which can be visualised in the GUI under Documentation Browser tab or in any web browser as it is an html file.

Documentation Browser x 3DView x 3DOverview x TaskMonitor x FileViewer x TableViewer x

Home x Results report

Self Propulsion example 1

**FLOWTECH**  
INTERNATIONAL AB

Ship particulars	
Length $L_{WL}$ [m]	63.00
Length $L_{WL}$ (model) [m]	5.500
Hull surface roughness $k_s$ [ $\mu$ m]	150
Wetted surface $S_{Hull}$ [ $m^2$ ]	678
Bilge keel area $S_{BK}$ [ $m^2$ ]	0.000
Proj. area above water line $A_T$ [ $m^2$ ]	55.000

Self Propulsion example 1

**FLOWTECH**  
INTERNATIONAL AB

### Self Propulsion Simulation Results

Simulation results

Ship speed $V_S$ [kn]	Model speed $V_m$ [m/s]	Froude no. $F_n$ [-]	Reynolds no. $Rn_m$ [-]	Res. coeff. total $C_{Tm} \cdot 1000$ [-]	Res. coeff. friction $C_{Fm} \cdot 1000$ [-]	Res. coeff. pressure $C_{Pm} \cdot 1000$ [-]	Res. coeff. wave $C_{Wm} \cdot 1000$ [-]	Form factor $k$ [-]	Thrust ded. $t_m$ [-]	Mean wake $w_{Tm}$ [-]	Rel. rot. effic. $\eta_{Rm}$ [-]	Prop. effic. $\eta_{0m}$ [-]	Nom. wake $w_{Nm}$ [-]
15.27	2.321	0.316	1.217e+007	5.843	3.048	0.498	2.296	0.182	-0.370	0.220	1.048	0.689	0.586

ITTC78 performance prediction

Ship speed $V_S$ [kn]	Ship res. $R_S$ [kN]	Eff. power $P_E$ [MW]	Deliv. power $P_D$ [MW]	Shaft rate $n_S$ [1/s]	Thrust $T_S$ [kN]	Torque $Q_S$ [kNm]	Tot. eff. $\eta_D$ [-]	Prop. eff. $\eta_0$ [-]	Hull eff. $\eta_H$ [-]	Mean wake $w_{TS}$ [-]	Advance ratio $J_{TS}$ [-]
15.27	125.555	0.986	0.425	2.679	91.67	25	2.319	1.477	1.498	0.086	0.894

Self Propulsion example 1

**FLOWTECH**  
INTERNATIONAL AB

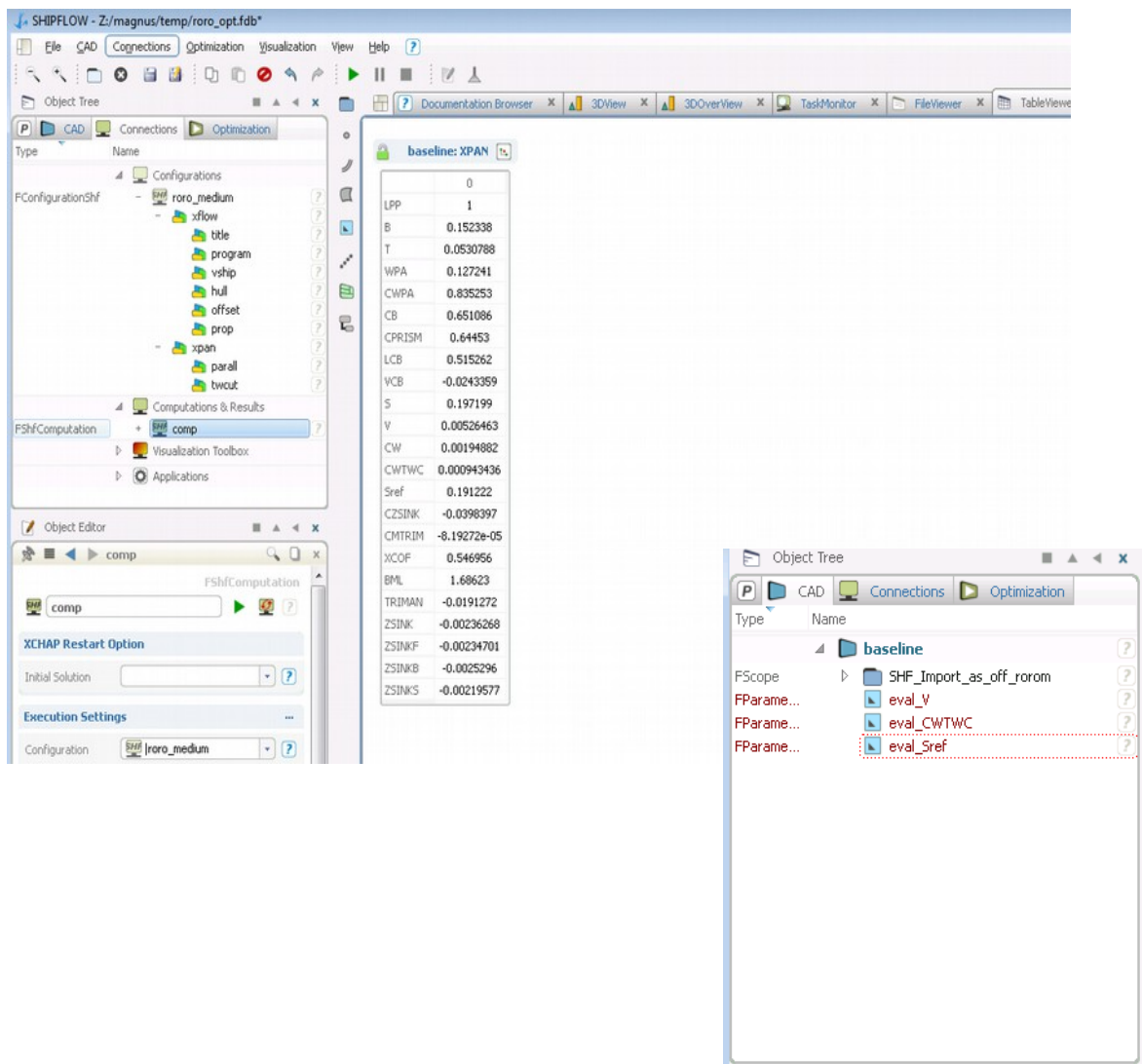
Model propulsor open water characteristics		
Advance ratio $J$ [-]	Thrust coeff. $10^4 K_{Tm}$ [-]	Torque coeff. $100^4 K_{Qm}$ [-]
0.2000	3.6459	4.4278
0.9000	0.8608	1.7030

Ship propulsor open water characteristics

## Tutorial 2 part 1 – Forebody optimization – bulb – Delta Shift

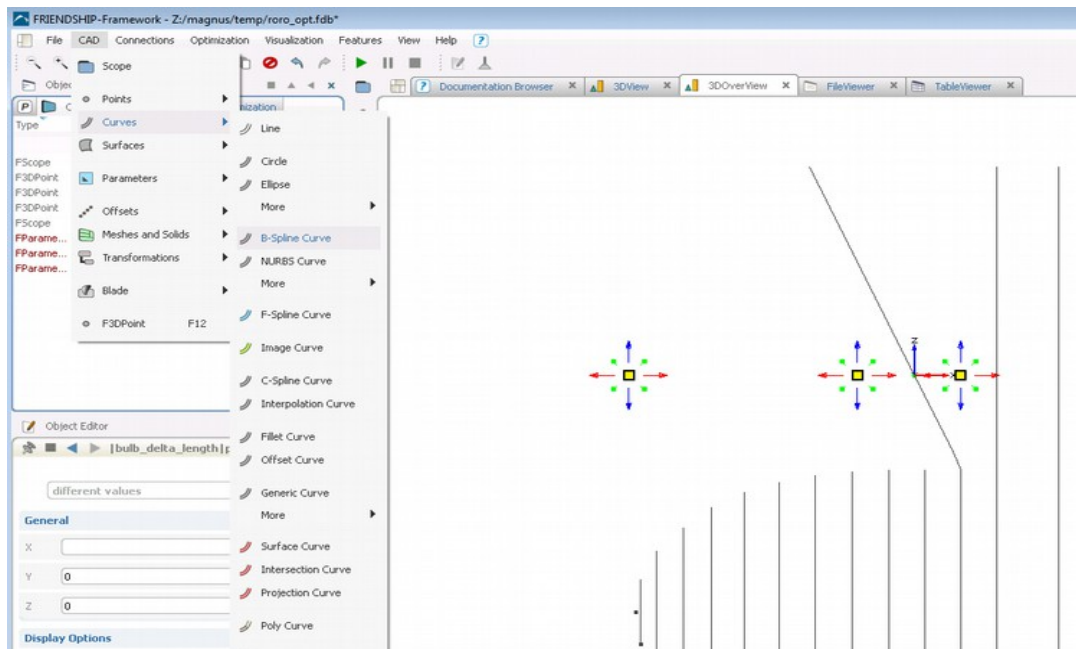
For this tutorial you need to use Friendship FRAMEWORK together with SHIPFLOW.

1. Import roro\_medium SHIPFLOW configuration from **DVD:\AdvancedTraining\Tutorial\_Advanced\_2\source**
2. Save the project as roro\_opt.fdb and run calculations without any changes to the configuration.
3. Go to the Table Viewer and double click on results for V, CWTWC and Sref to create parametres for evaluation.

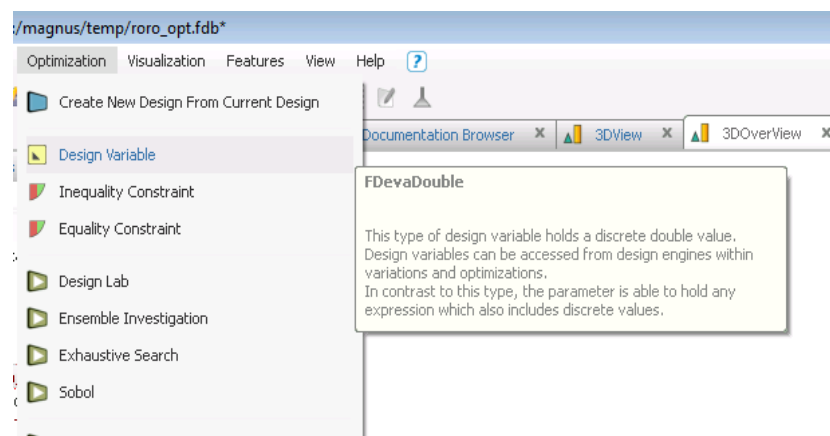


4. Go to 3DView and extend the view to see the entire hull.
5. Set the view to Y to and zoom in to see the bulb
6. Create a Scope and name it bulb\_delta\_length, make it default by clicking on it with middle mouse button.

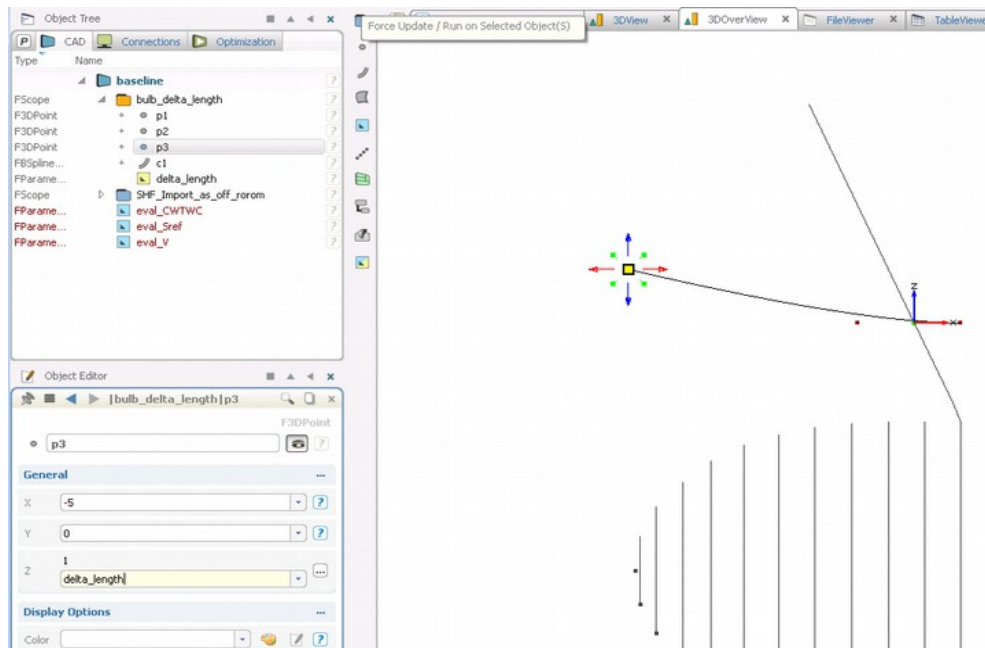
7. Create 3 points and locate them as specified below:
  1.  $p1(0.81,0,0)$
  2.  $p2(-1,0,0)$
  3.  $p3(-5,0,0)$
8. Select these points and create a B-Spline curve, make sure it is 2<sup>nd</sup> order.



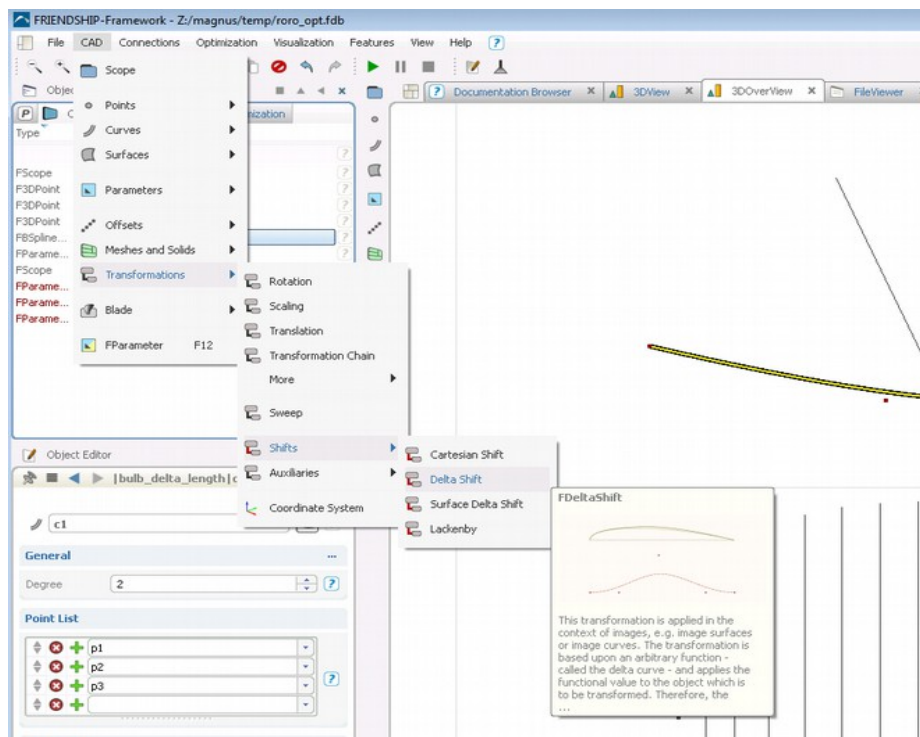
9. Create a Design Variable, name it delta\_length, set the value to 1.



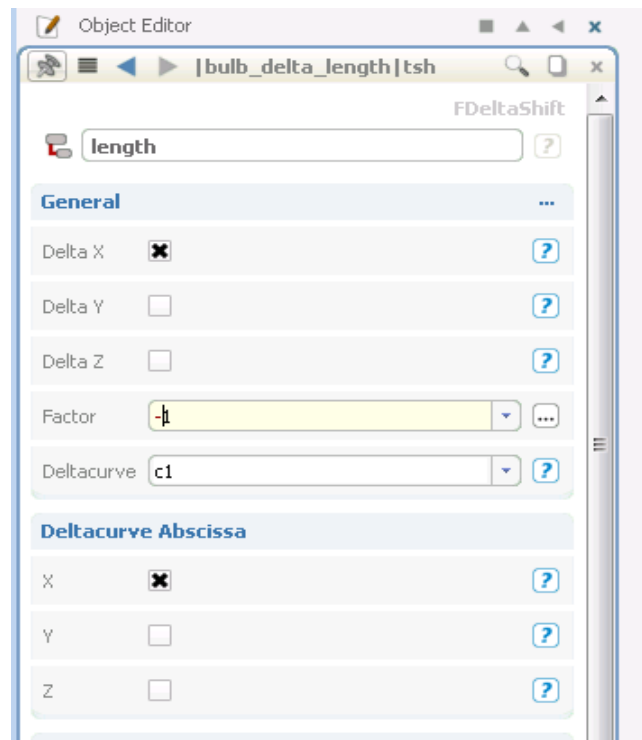
10. Now replace the Z coordinate of the p3 with this design variable



11. Select the curve, create Delta Shift and name it length



12. Edit the length Delta shift, mark Delta X in the General section and set factor to -1

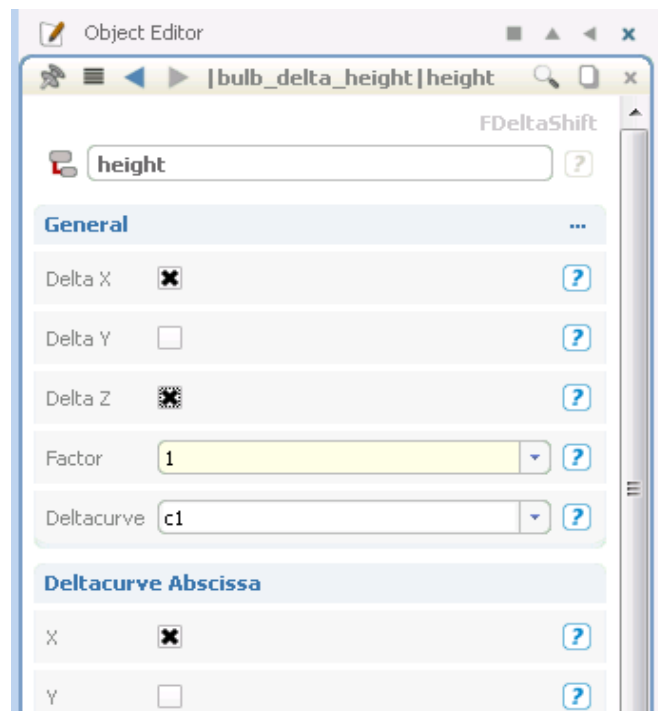


13. Make the baseline the default scope

14. Select bulb\_delta\_length scope and make a copy of it, rename it to bulb\_delta\_height

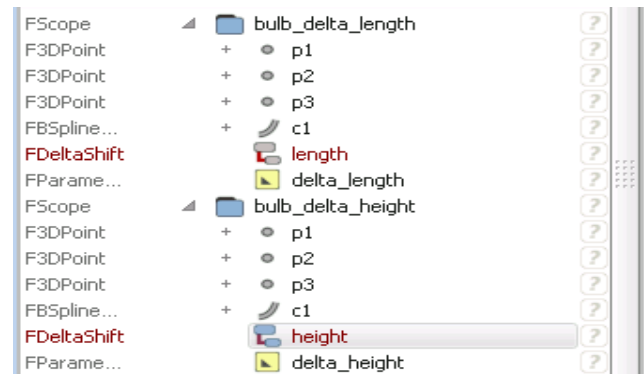
15. Go to that scope and rename length with height and delta\_length with delta\_height

16. Edit the height Delta shift, mark Delta Z in the General section and set factor to 1

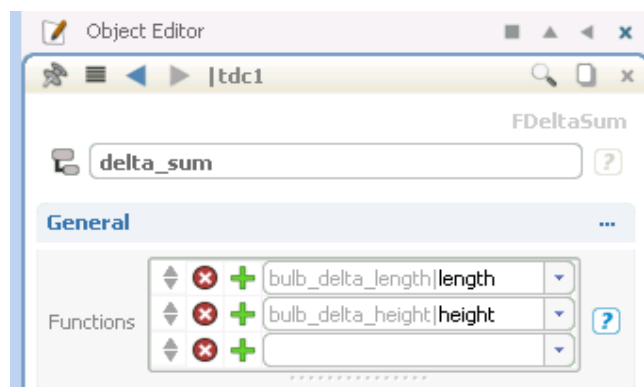
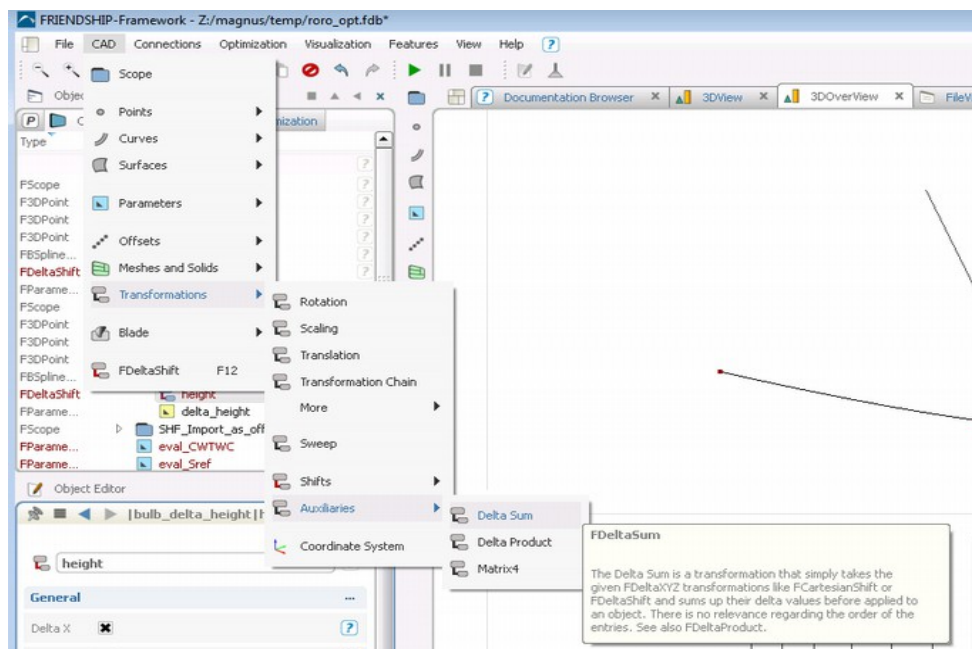




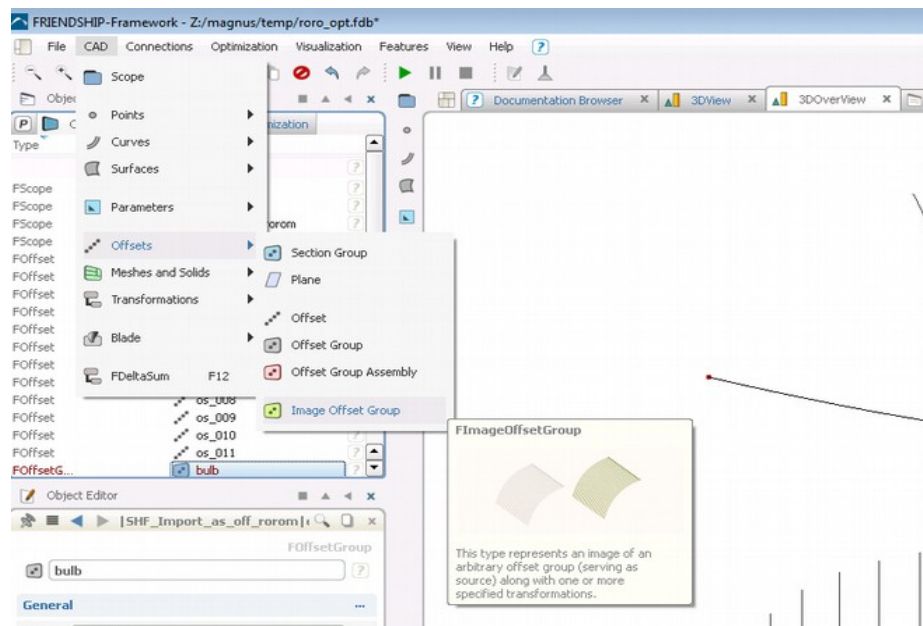
17. Now you should have two scopes as in the picture below:



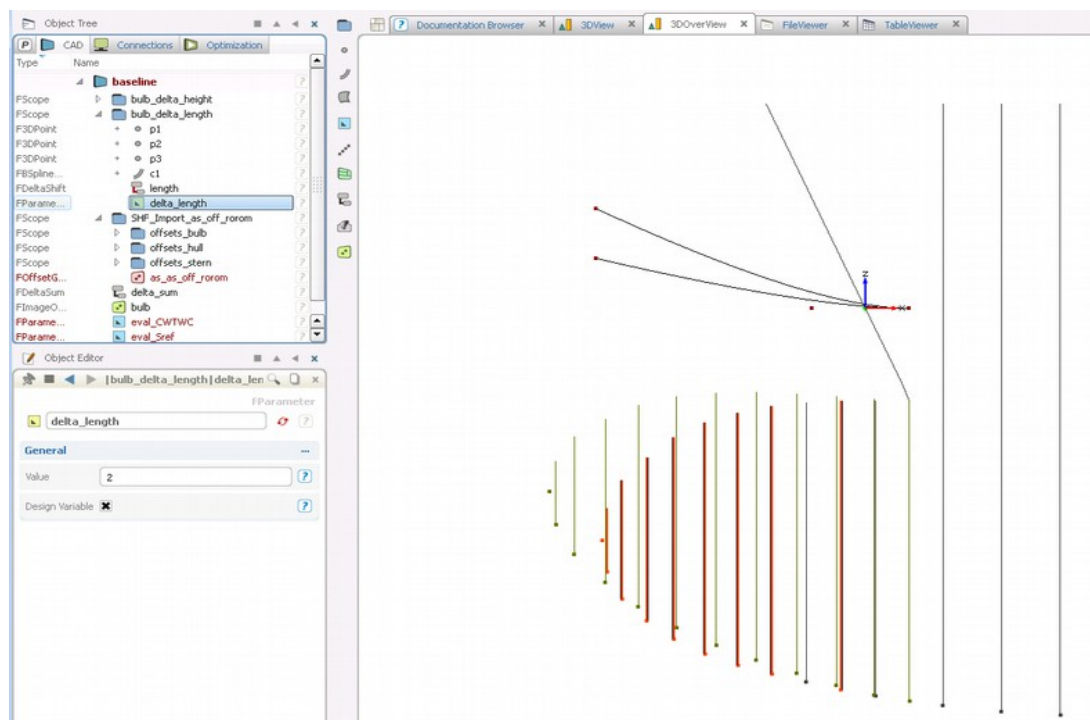
18. Create Delta Sum name it delta\_bulb and use delta\_length and delta\_height as input.



19. Select the bulb offset group and create and Image Offset Group and name it bulb

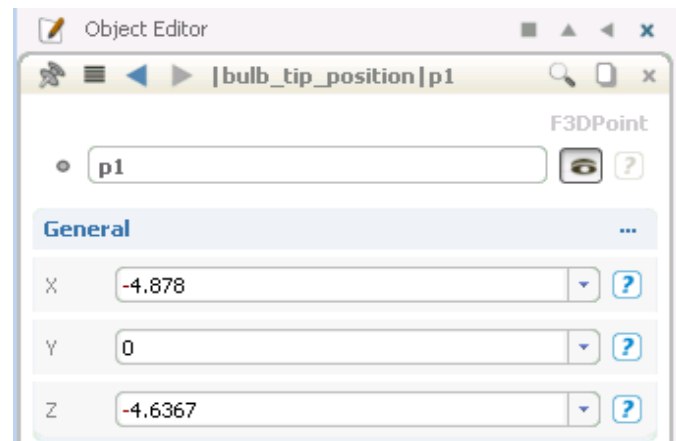


20. Use the delta\_bulb Delta Sum transformation as an Image transformation.
21. Now go to the offset group assembly and replace the existing bulb group with the image that was just created
22. You should be able to see that the Image Offset changes its shape when you edit delta\_length and delta\_height design variables.

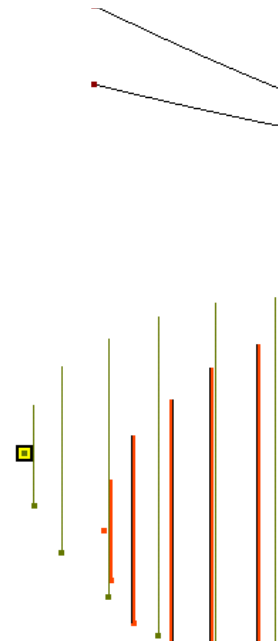
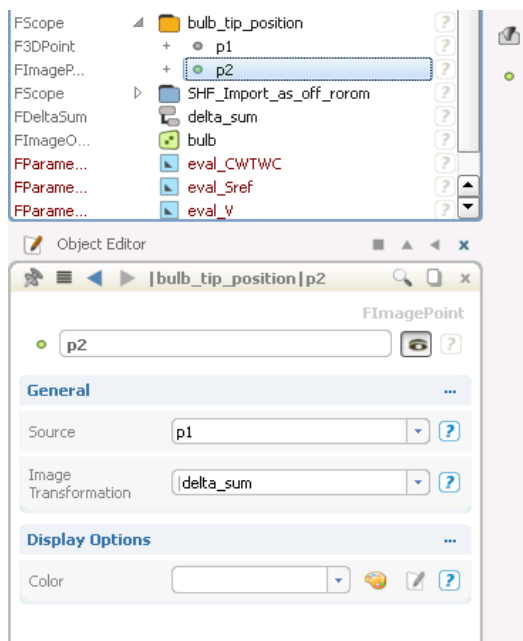


23. Create a scope and name it bulb\_tip\_position and make it default

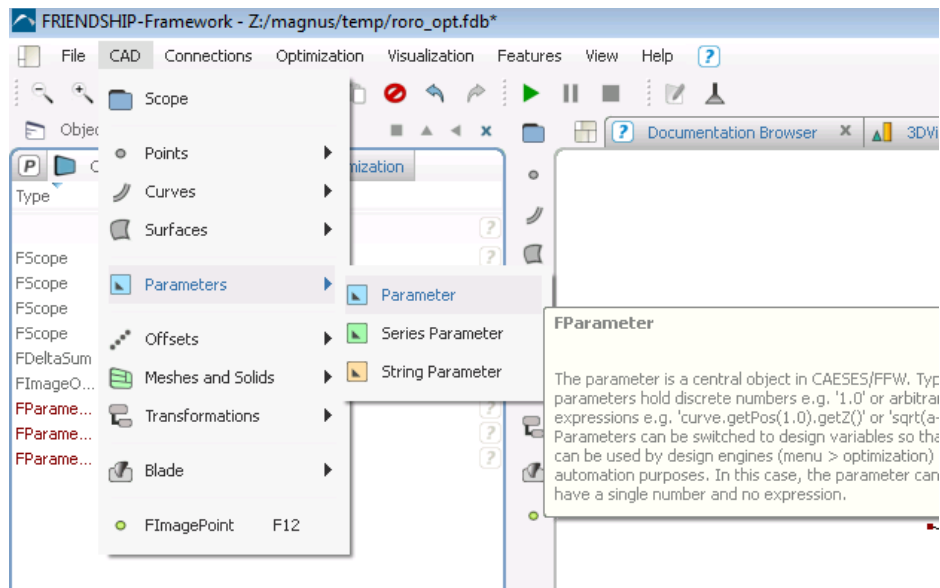
24. Create point at the tip of the bulb



25. Select this point and create Image Point and use the delta\_bulb Delta Sum transformation as an Image transformation.



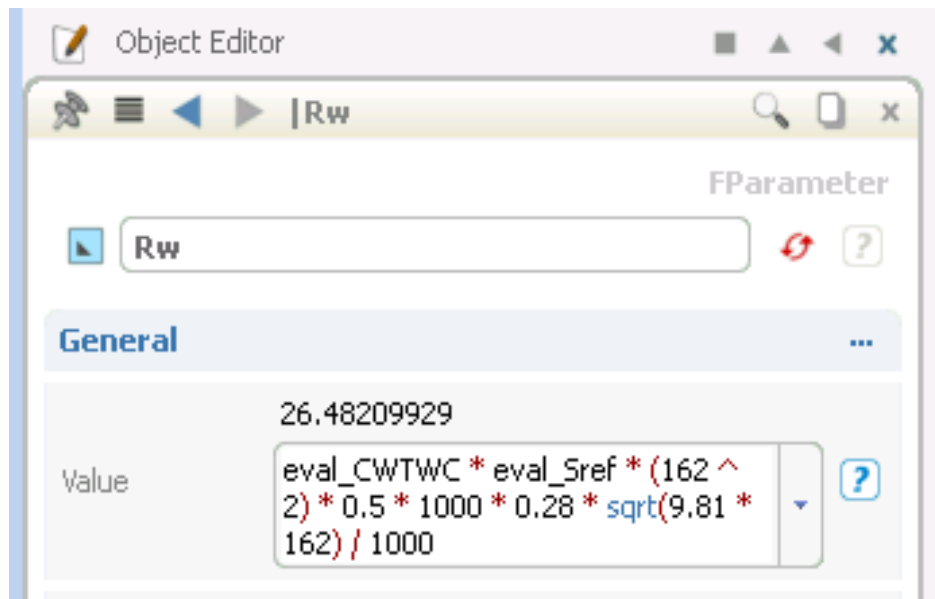
26. Create a new Parameter and name it LOA



27. By using 3DPoint:x and getMax(0) functions of the Image bulb tip point and transom offset section create a formula to measure the Length Over All. This will be used as a constraint during the optimization.



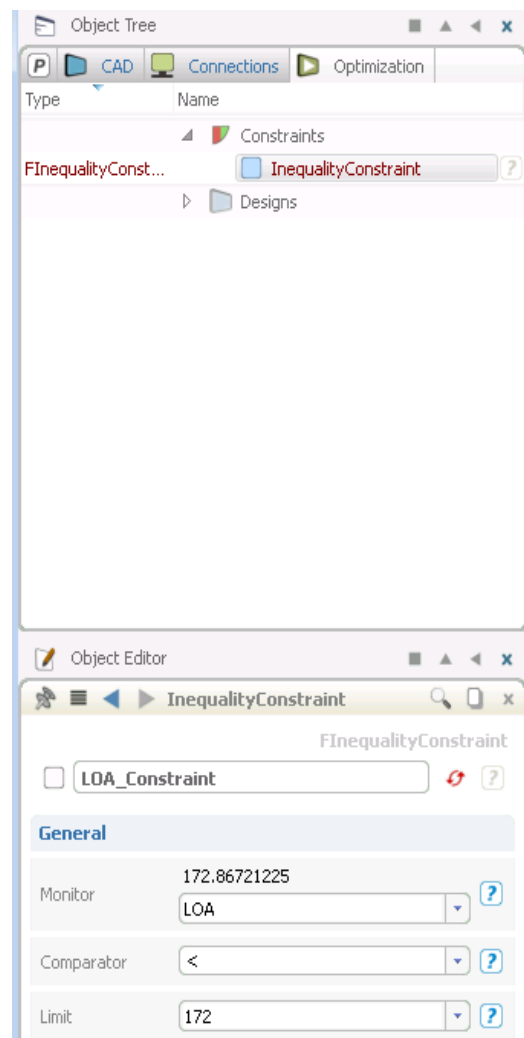
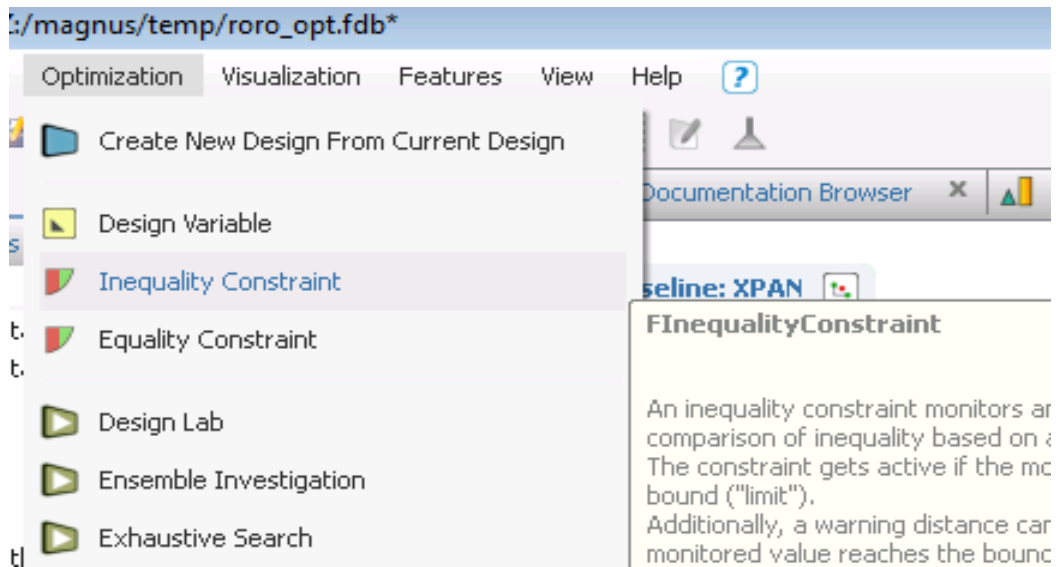
28. Add another parameter which will compute wave resistance in kN, name it Rw



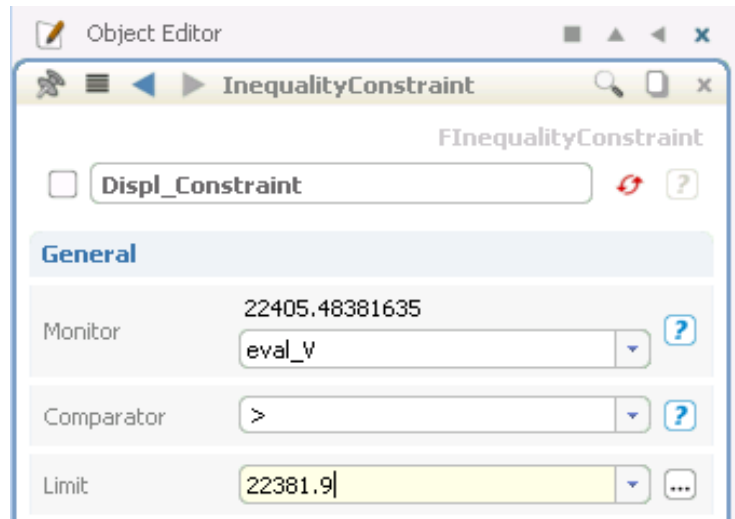
29. Rescale eval\_V to get the value in m<sup>3</sup>



30. Add a Constraint, name it LOA\_constraint. Monitor LOA parameter and set the max value to 172

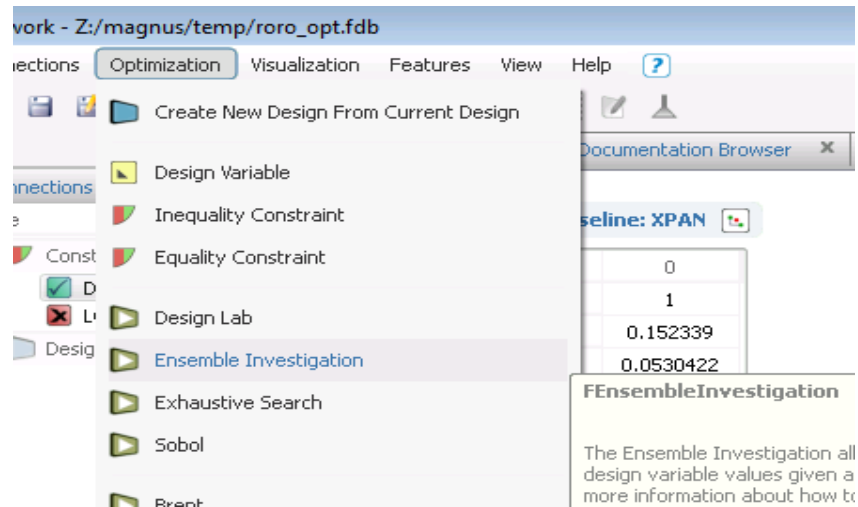


31. Add a Constraint, name it Displ\_constraint. Monitor eval\_V parameter and set the min value to 22381.9

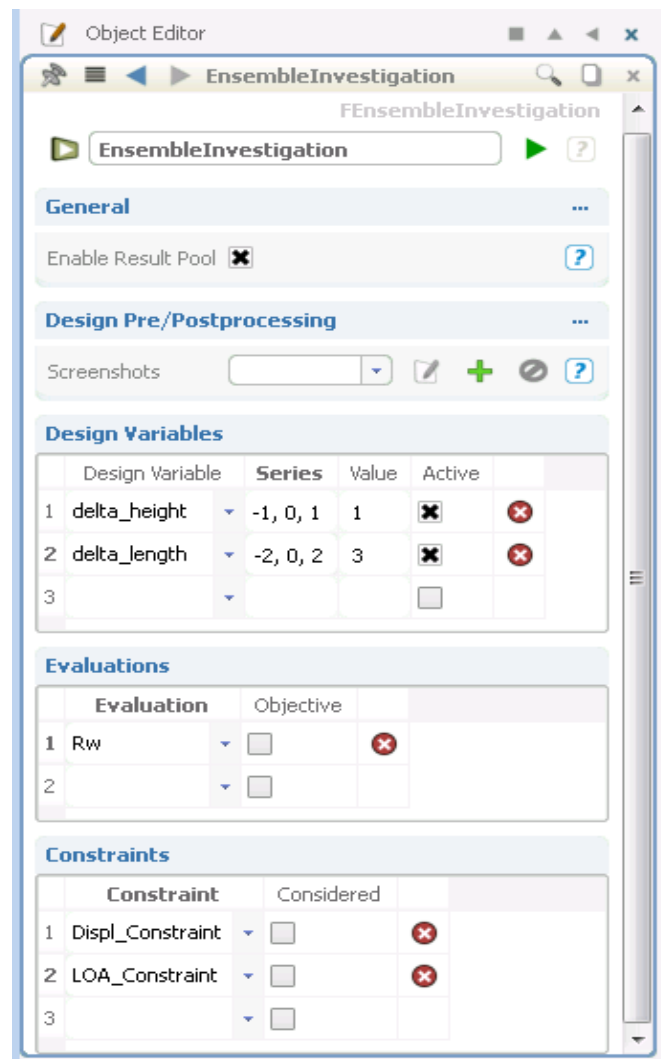


## Tutorial 2 part 2 – Ensemble Investigation

### 1. Create Ensemble Investigations



### 2. Set the parameters according to the image below:





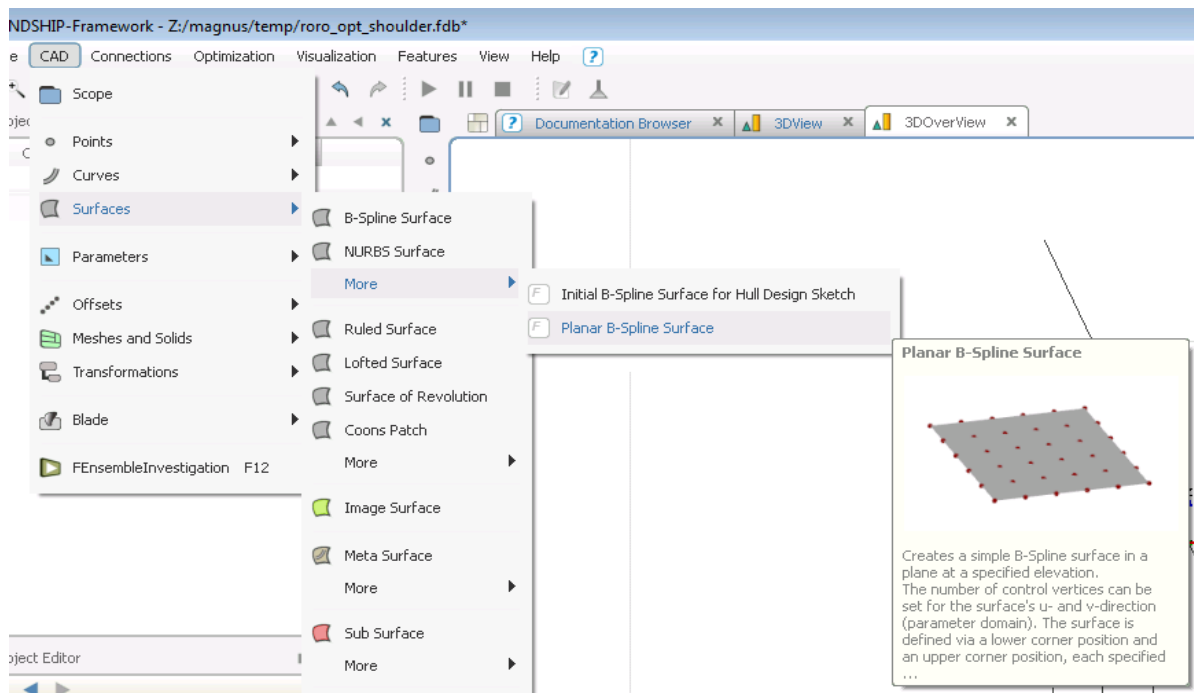
3. Save the Project.
4. Run Ensemble Investigation.
5. When the calculations are finished the following result table should be available. Notice That some of the Design Variants violate the constraints.

EnsembleInvestigat						
	delta_height	delta_length	Rw	Displ_Constr	LOA_Constrai	
Attribute	Active	Active				
Name	delta_height	delta_length	Rw	Displ_Constraint	LOA_Constraint	
Scope	bulb_delta_height	bulb_delta_length				
Reference						
Lower Bound	-1	-2	26.319114	> 22381.9	< 172	
Upper Bound	1	2	26.54825			
Feasible Designs: 33.333333 %				55.555556 %	77.777778 %	
Mean Utilization Index				96.713386 %	48.395715 %	
Mean	0	0	26.428745	22382.688	170.928	
Sample Standard Deviation	0.8660254	1.7320508	0.082876061	23.960222	1.8776342	
Error-free: 100 %	100 %	100 %	100 %	100 %	100 %	
EnsembleInvestigati...	-1	-2	26.361546	22371.307	169.95839	
EnsembleInvestigati...	-1	0	26.319114	22396.404	171.89761	
EnsembleInvestigati...	-1	2	26.320528	22420.154	173.83682	
EnsembleInvestigati...	0	-2	26.484882	22357.448	168.98879	
EnsembleInvestigati...	0	0	26.42412	22382.706	170.928	
EnsembleInvestigati...	0	2	26.411024	22406.748	172.86721	
EnsembleInvestigati...	1	-2	26.54825	22345.724	168.01918	
EnsembleInvestigati...	1	0	26.5059	22369.856	169.95839	
EnsembleInvestigati...	1	2	26.483344	22393.844	171.89761	

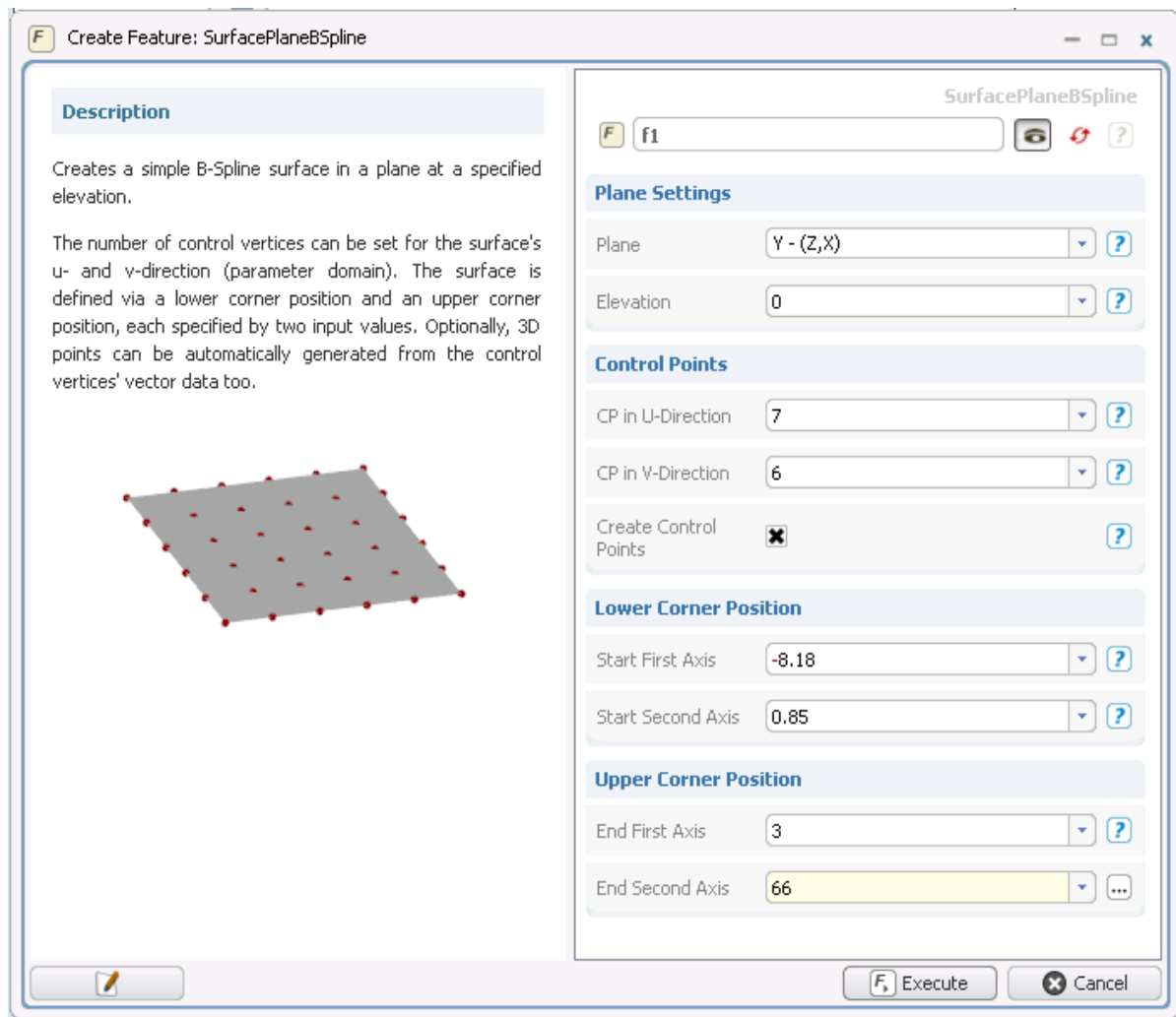
## ***Tutorial 3 part 1 – Forebody optimization – shoulder – Surface Delta Shift***

For this tutorial you need to use Friendship FRAMEWORK together with SHIPFLOW.

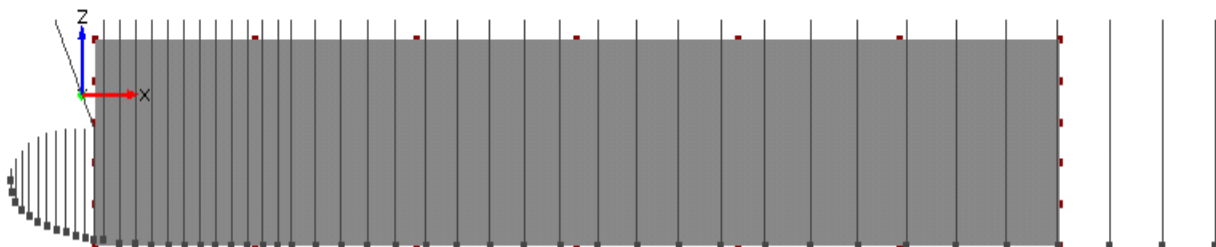
1. Import roro\_medium SHIPFLOW configuration from **DVD:\AdvancedTraining\Tutorial\_Advanced\_3\source**
2. Save the project as roro\_opt\_shoulder.fdb
3. Go to 3DView end extend the view to see the entire hull.
4. From the CAD menu select Surfaces | Planar B-Spline Surface



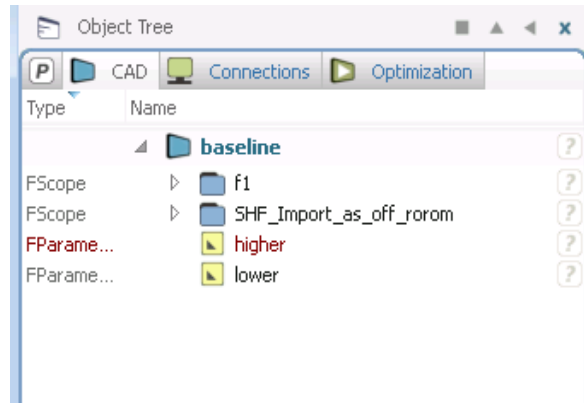
5. Use the settings as below to cover the forward part of the hull



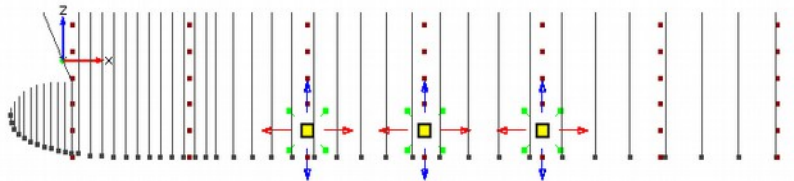
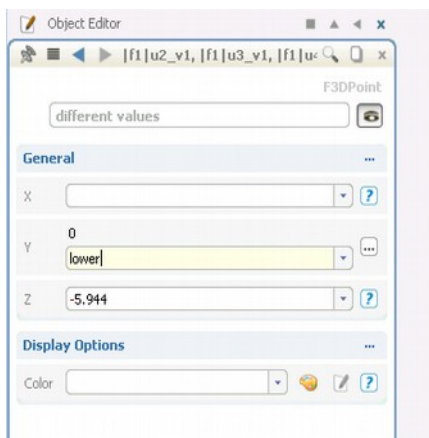
6. The resulting patch should resemble the one shown below



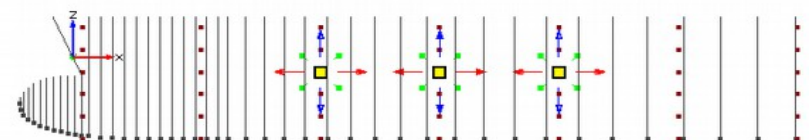
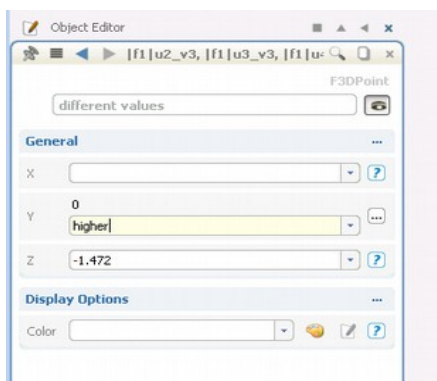
7. Create two Design Variables, name them lower and higher



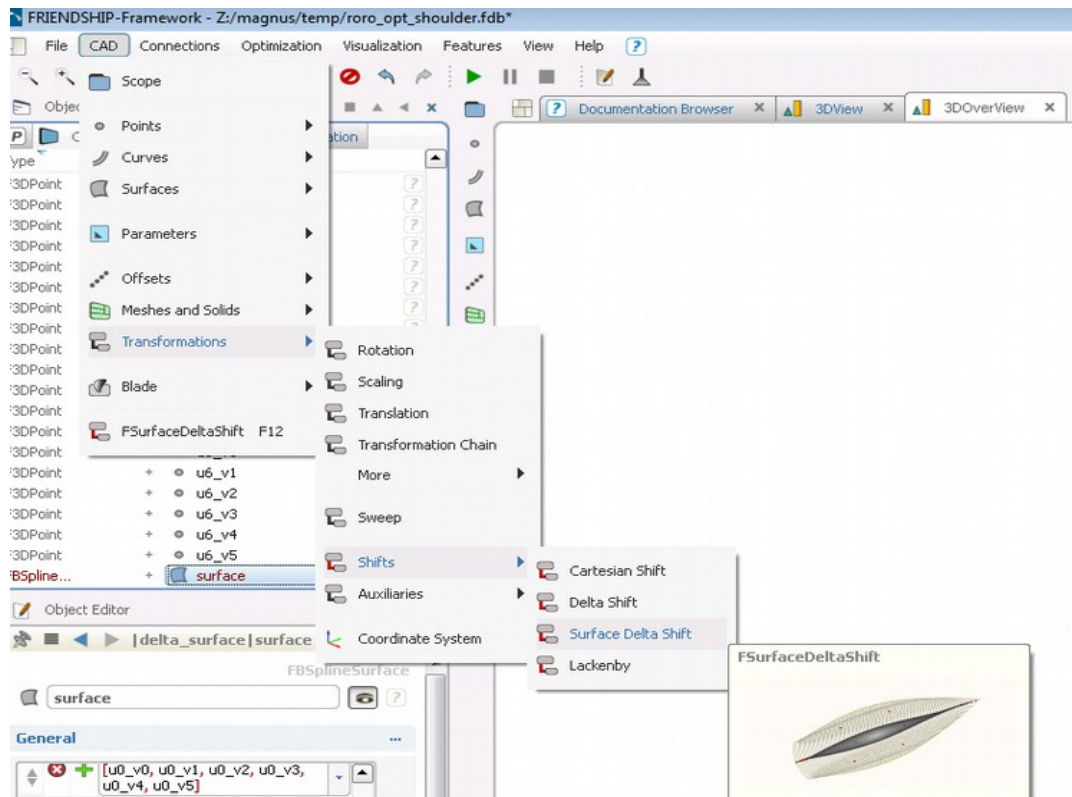
8. Select points of the delta surface as in the picture (2<sup>nd</sup> row from the bottom, columns 3,4 and 5) and replace their y-coordinate with lower Design Variable



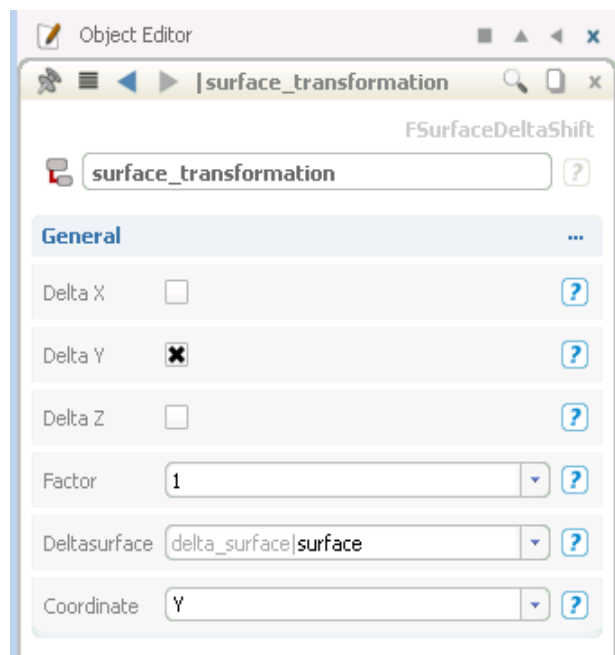
9. Select points of the delta surface as in the picture (4<sup>th</sup> row from the bottom, columns 3,4 and 5) and replace their y-coordinate with higher Design Variable



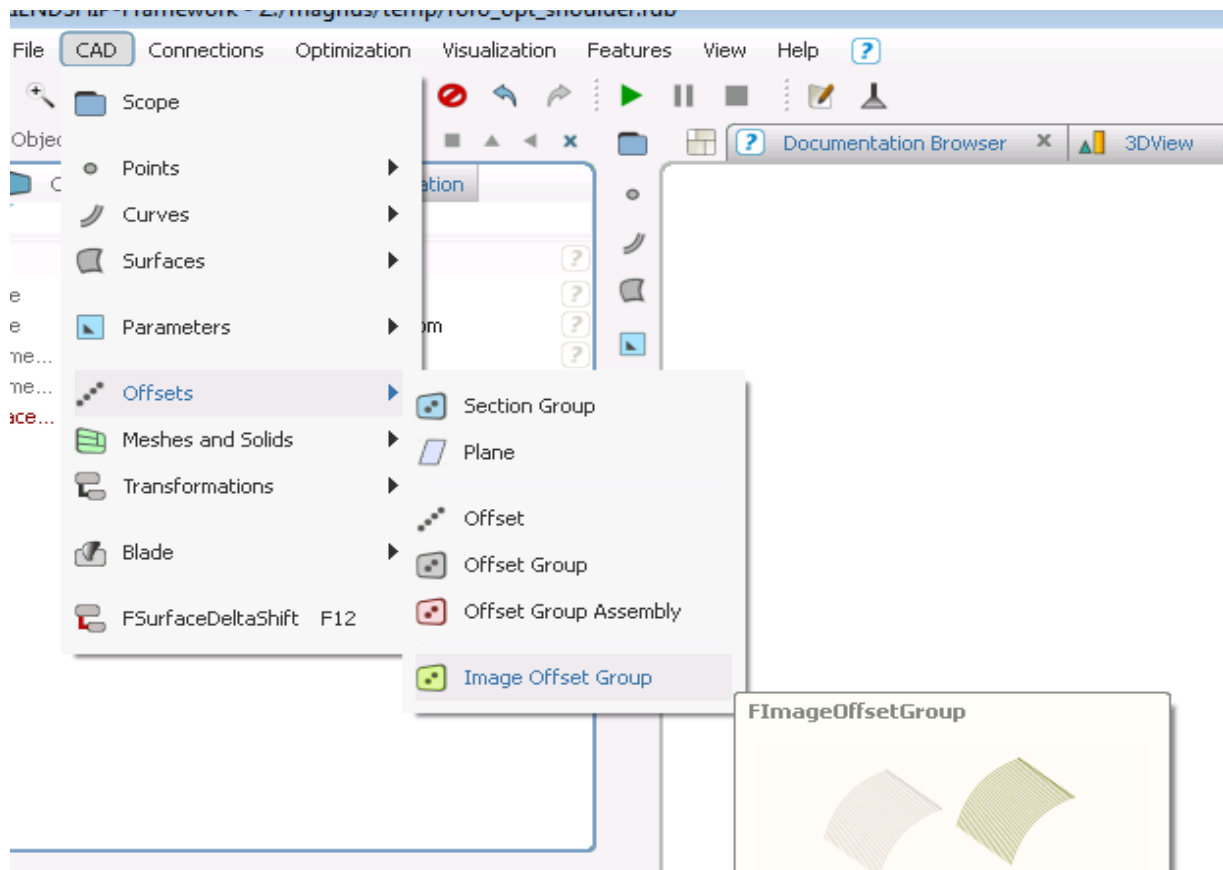
10. Create Surface Delta Shift transformation based on the surface patch that was added earlier. Name it `surface_transformation`.



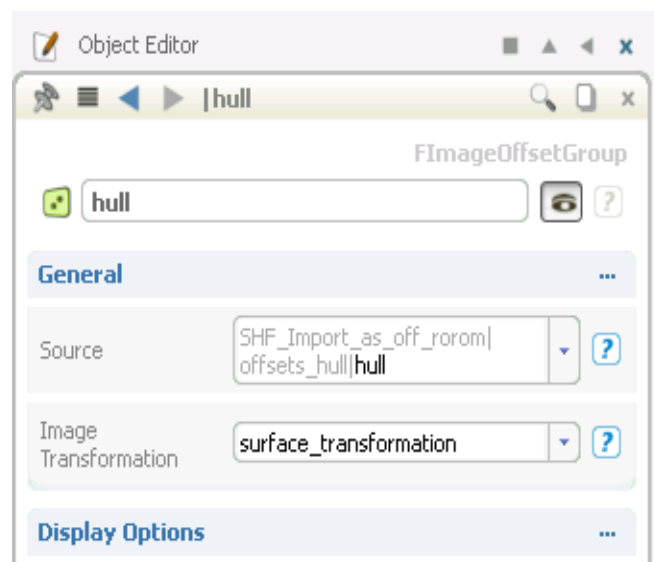
11. Set it up to use surface patch and act in Y direction.



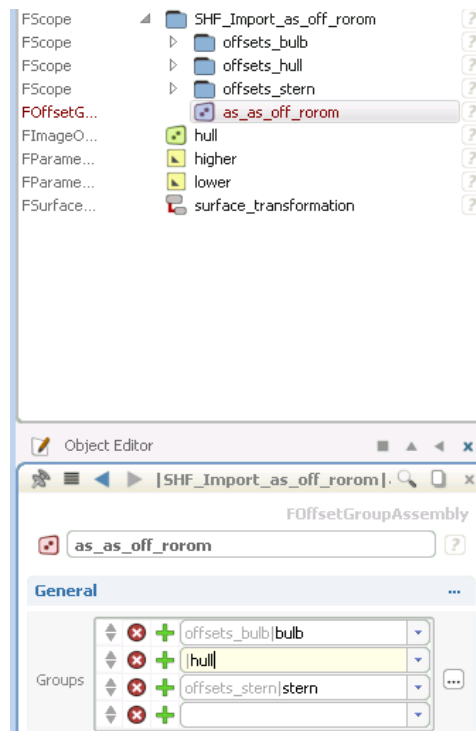
12. Create image of the hull offset group, name it hull.



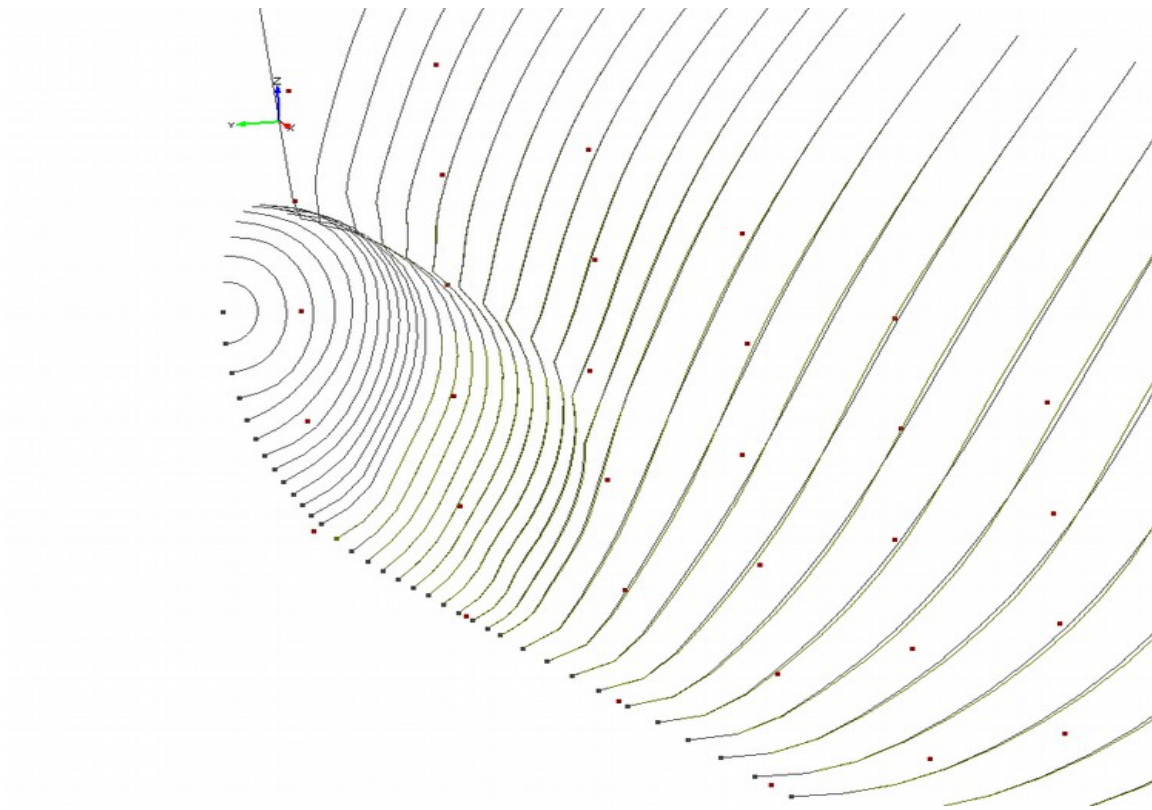
13. Use surface\_transformation as the Image transformation



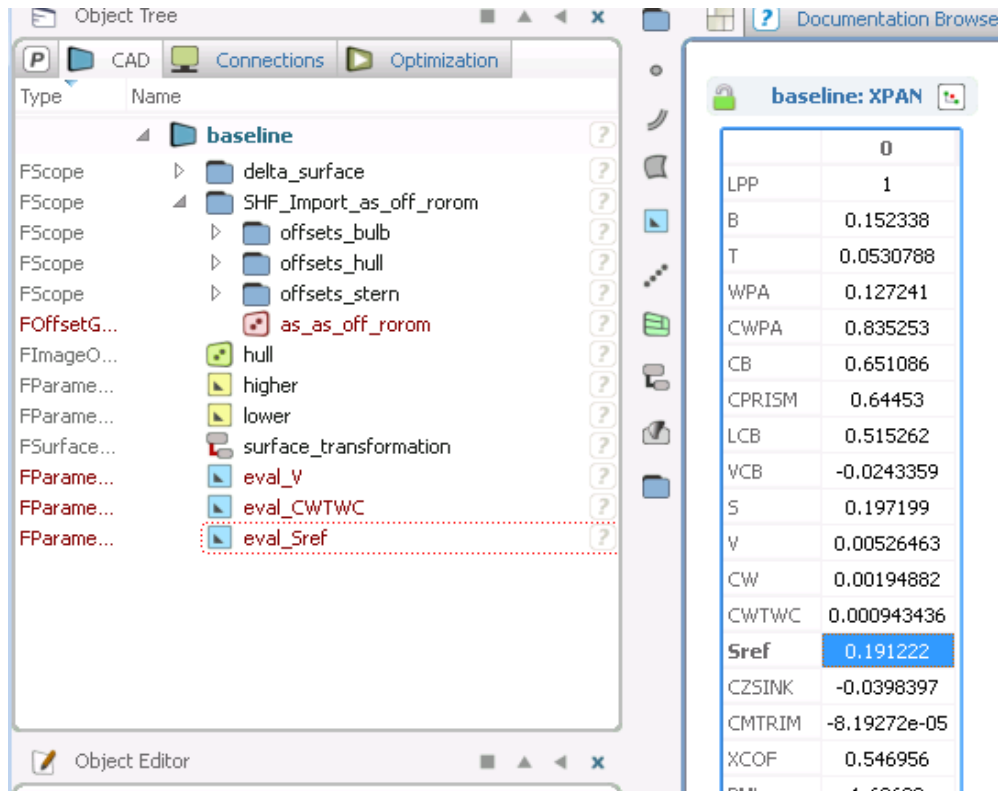
14. In the offset group assembly replace the original hull group with the image.



15. Test if the set up works by modifying lower and higher design variables and see if the hull offset changes shape.



16. Set both lower and higher design variables to 0 and run the calculations.
17. When the calculations are finished create parameters for evaluations by double clicking on values of CWTWC, Sref and V.

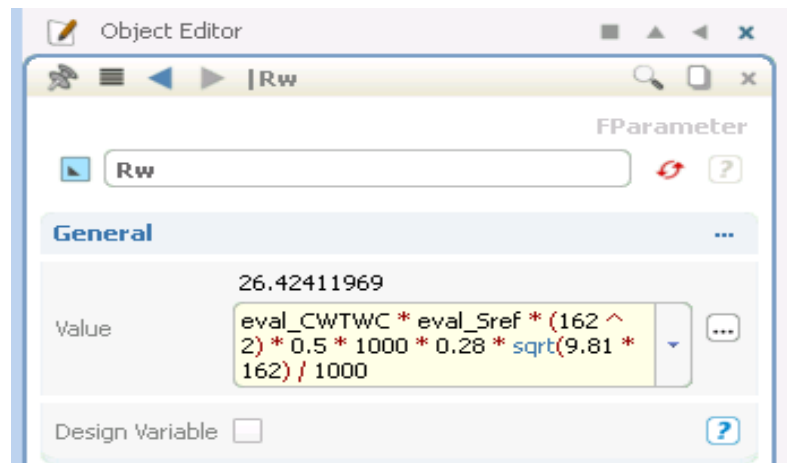


18. Rescale eval\_V to get the value in m<sup>3</sup>

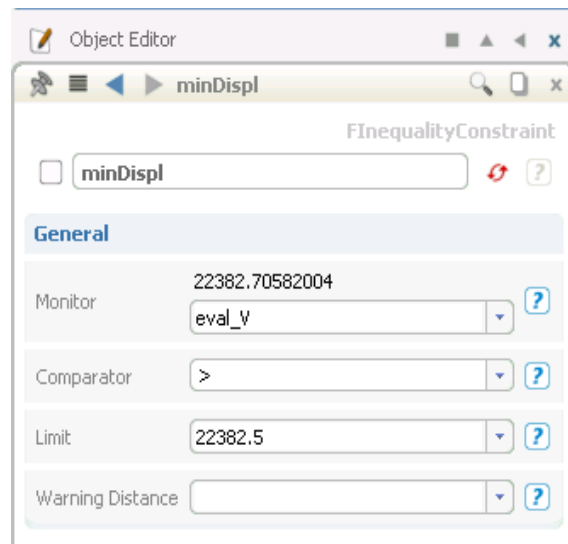




19. Add another parameter which will compute wave resistance in kN, name it Rw

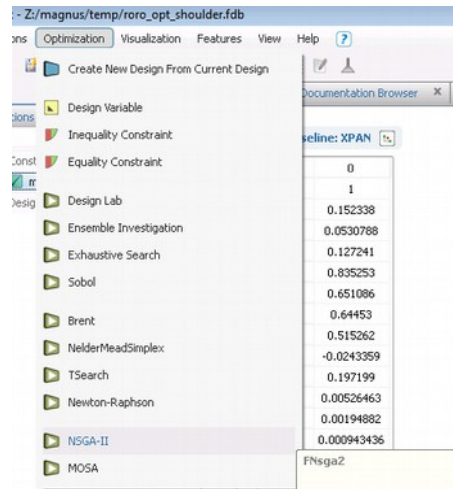


20. Add a constraint minDispl for minimum Displacement and set it to 22382.5

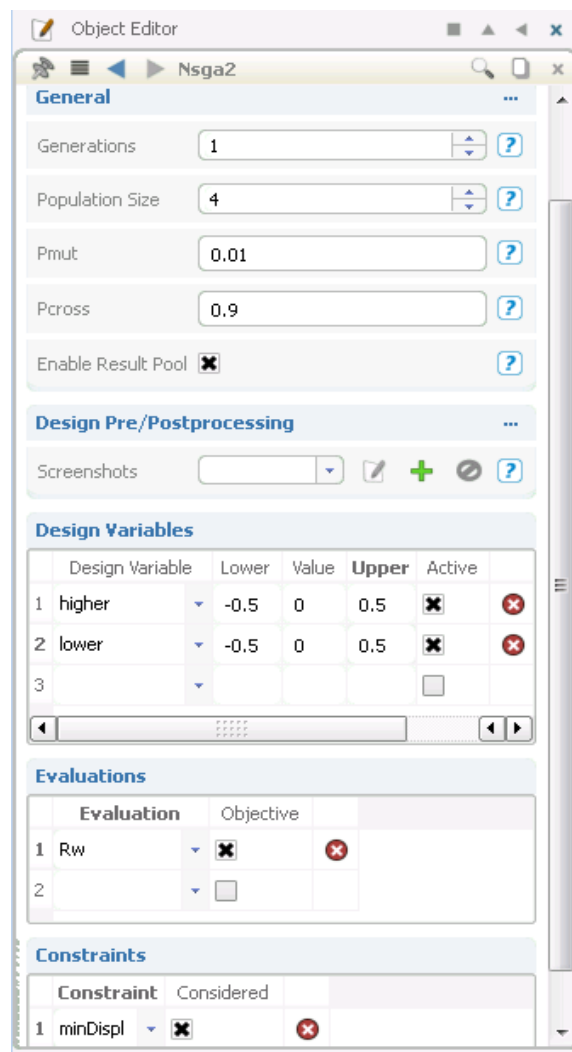


## Tutorial 3 part 2 – NSGA-II Optimization

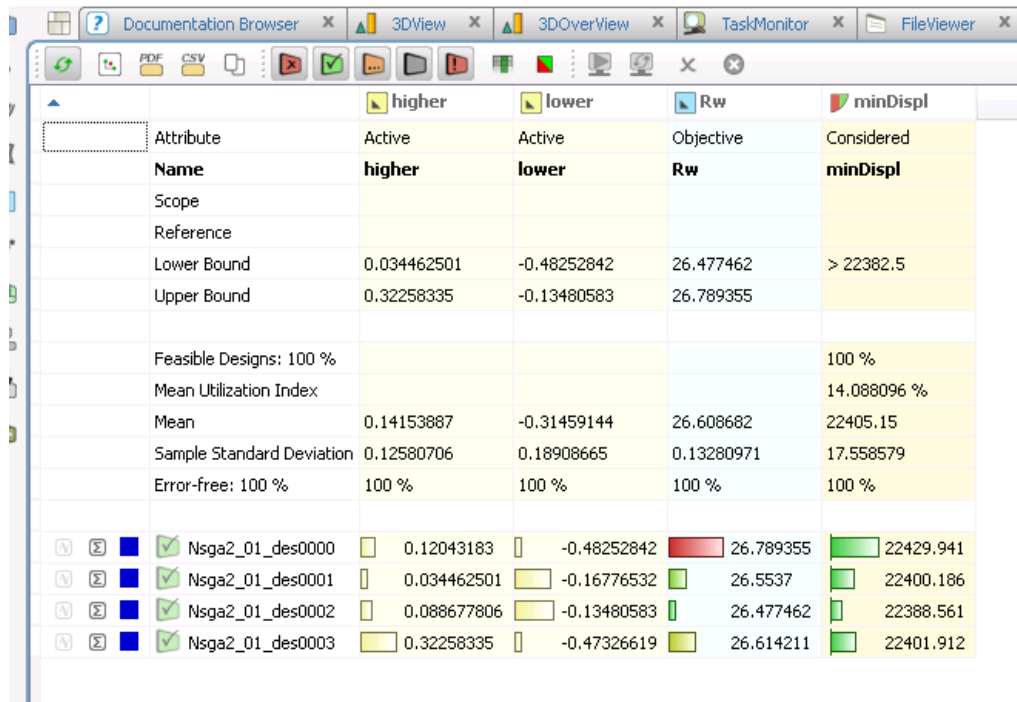
1. Create NSGA-II Design Engine (genetic algorithm optimization)



2. Set it up according to the image below, lower bound -0.5, upper bound 0.5, evaluation set to Rw and minDispl as a constraint.



3. Save the project and run the optimization.
4. It should take less than 10 minutes on a laptop and the result should resemble the one below

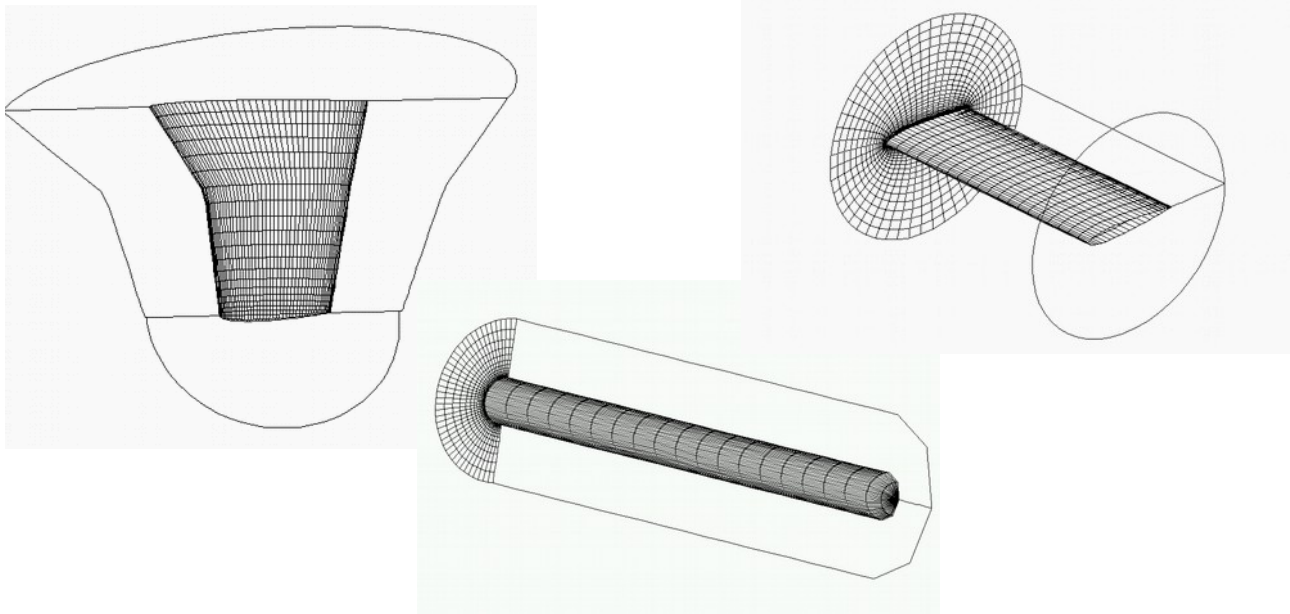


	higher	lower	Rw	minDispl
Attribute	Active	Active	Objective	Considered
Name	higher	lower	Rw	minDispl
Scope				
Reference				
Lower Bound	0.034462501	-0.48252842	26.477462	> 22382.5
Upper Bound	0.32258335	-0.13480583	26.789355	
Feasible Designs: 100 %				100 %
Mean Utilization Index				14.088096 %
Mean	0.14153887	-0.31459144	26.608682	22405.15
Sample Standard Deviation	0.12580706	0.18908665	0.13280971	17.558579
Error-free: 100 %	100 %	100 %	100 %	100 %
Nsga2_01_des0000	0.12043183	-0.48252842	26.789355	22429.941
Nsga2_01_des0001	0.034462501	-0.16776532	26.5537	22400.186
Nsga2_01_des0002	0.088677806	-0.13480583	26.477462	22388.561
Nsga2_01_des0003	0.32258335	-0.47326619	26.614211	22401.912

5. This is just an example setup and the optimization parameters should be refined as well as more generations in the NSGA-II should be used.

## ***Tutorial 4 part 1 – Appendage modelling – overlapping grids***

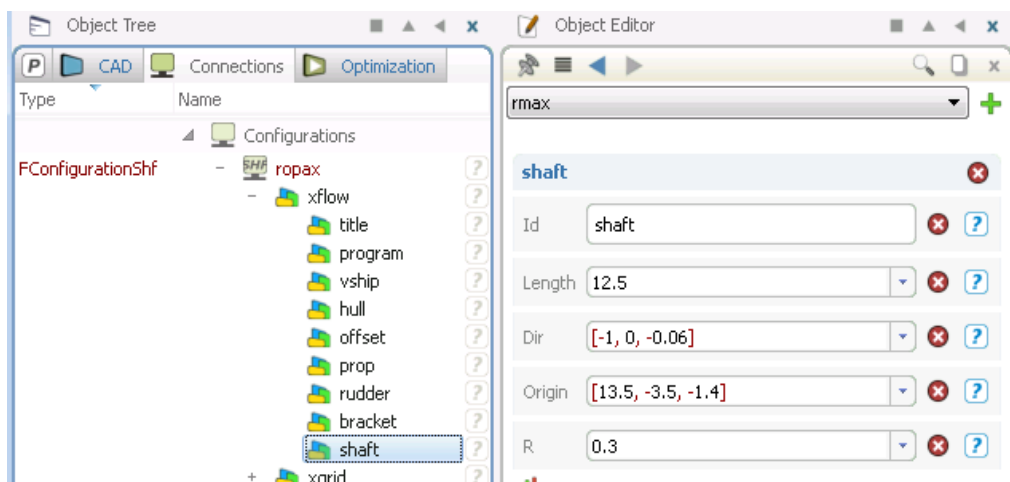
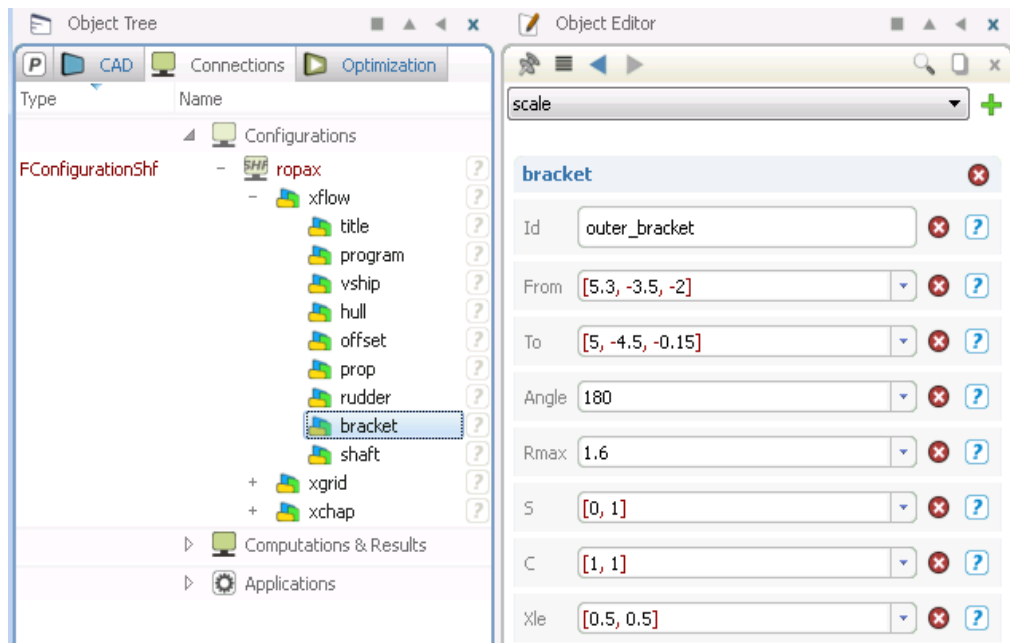
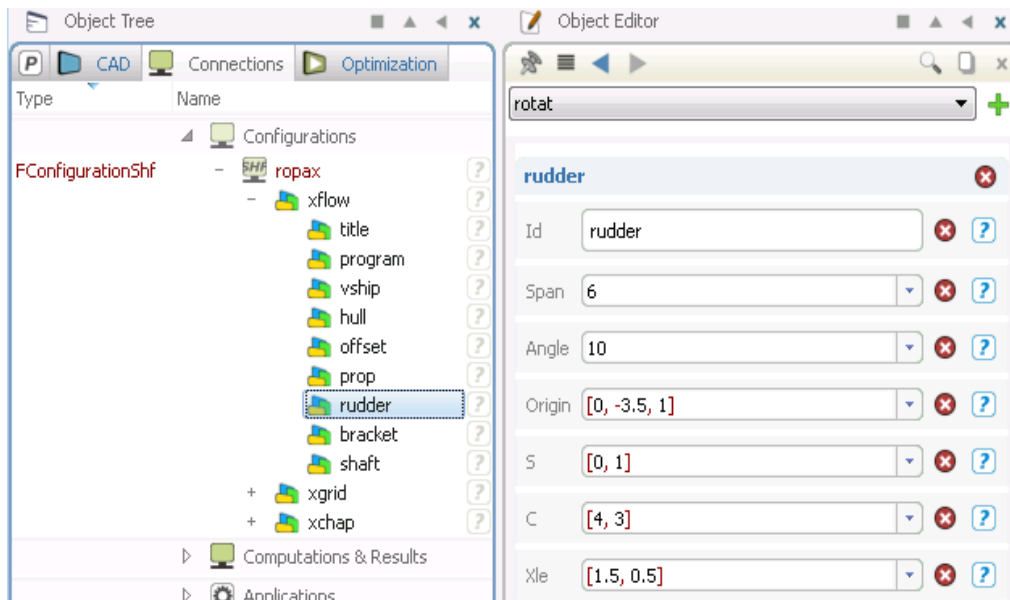
There are currently several appendage types in SHIPFLOW. In this tutorial, three of them will be used: rudder, propeller shaft and shaft bracket. All parameters for the respective appendage are given in the description of the RUDDER, SHAFT and BRACKET commands in the Users Manual. Like other commands that specify the geometry, the appendage commands are given in the XFLOW section of the command file. The commands may be repeated to specify several appendages of the same type.



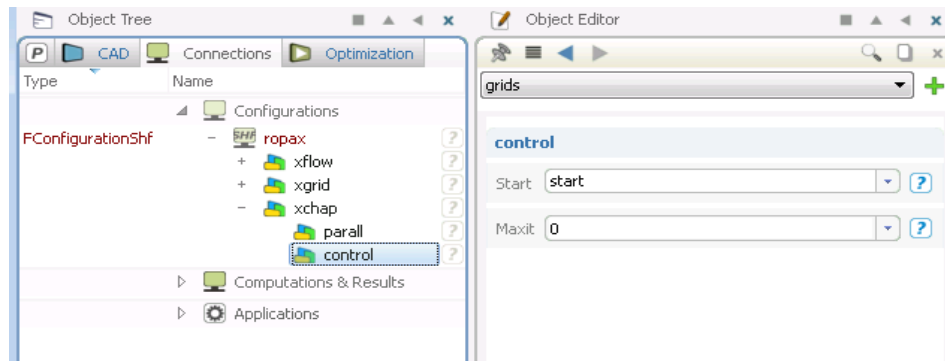
*Figure 1: Rudder, shaft and bracket grids.*

The appendage grids are not intended to be used independently, they are normally embedded in a grid that covers the computational domain of the un-appended configuration. This is usually the grid generated by XGRID, but there is also the possibility of generating a rectilinear grid with the BOX command. In the example command file below the Hamburg Test Case is appended with a rudder.

1. Create a directory which you will use for storing the files used during the assignment
2. Import “ropax” configuration from  
**DVD:\AdvancedTraining\Tutorial\_Advanced\_4\source**
3. Examine the rudder, bracket and shaft parameters.



- Run the computation with 0 iterations. This will generate the grids and the overlap information.



- During the computations important information will be printed in the Task Manager

```
[Captured output from application C:\FLOWTECH\SHIPFLOW5.0.00-x86_64\bin\shipflow.bat]

exePath = C:\FLOWTECH\SHIPFLOW5.0.00-x86_64\bin\..\bin/

SHIPFLOW 5.0.00 (Rev. 8859 )
xch2 location changed to: 1.0250000000000000

Starting xchap 5.0 $Revision: 8859 $
Commercial license
Command file ropax read.
Computing overlap
  No of frames      : 5
  No of grids       : 5
  No of points      : 452146
  No of interpolation cells : 47591
  No of discretization cells: 457565
  No of outside cells : 39777
  Total no of cells  : 544933

*****
WARNING: Less then 60% of the cells in grid "outer_bracket_1" is discretization cells. This is probably an OK setup that will run, but
If you have any questions, please contact FLOWTECH (support@flowtech.se) with the command file and offset file. If you know that your s
Total number of cells 49300
Number of discretization cells 28086
Number of non-fluid (outside) cells 14727
*****

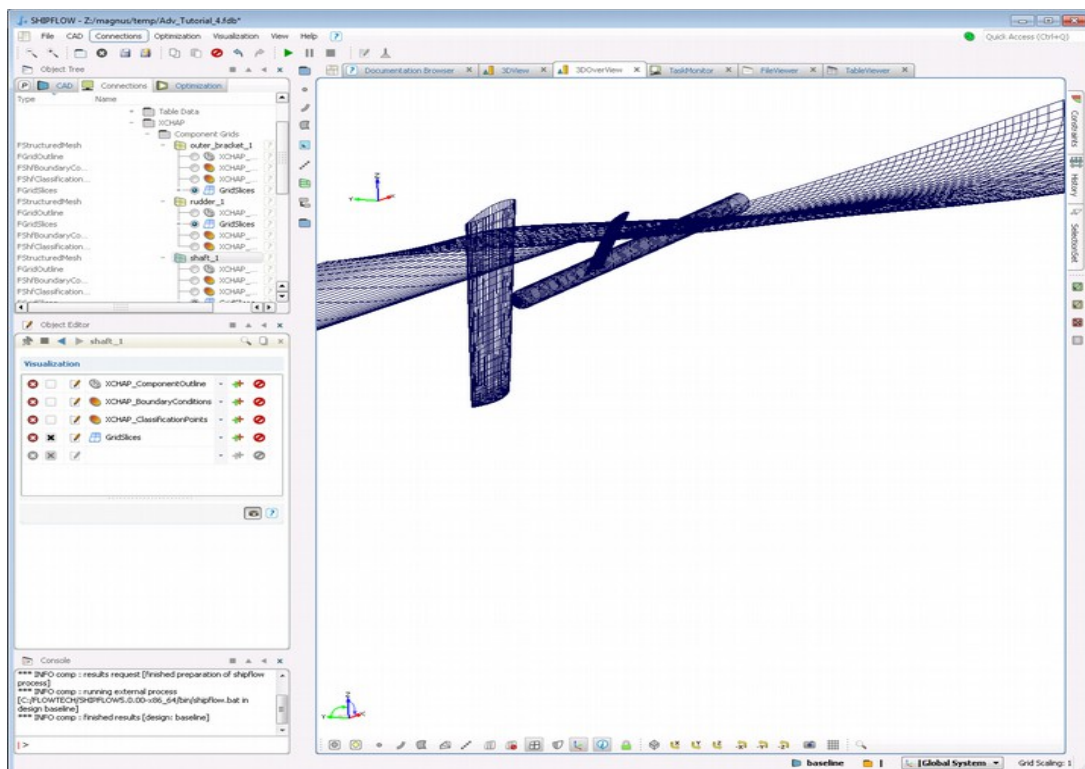
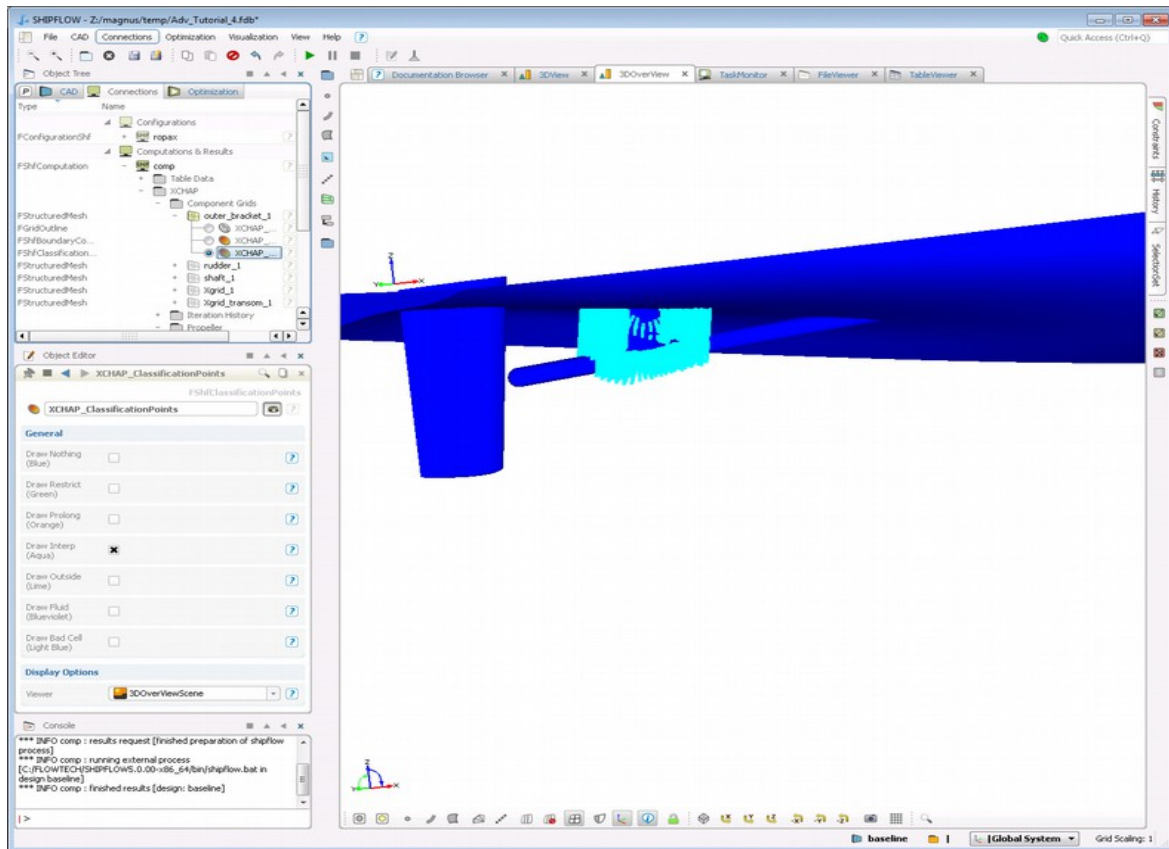
Running: C:\FLOWTECH\SHIPFLOW5.0.00-x86_64\bin\..\bin\xpongo ropax
XCHAP: CGNS : create base
Running under MPI: mpiexec -n 1 C:\FLOWTECH\SHIPFLOW5.0.00-x86_64\bin\..\bin\xkv5 ropax

Wall time in process 0: 0s

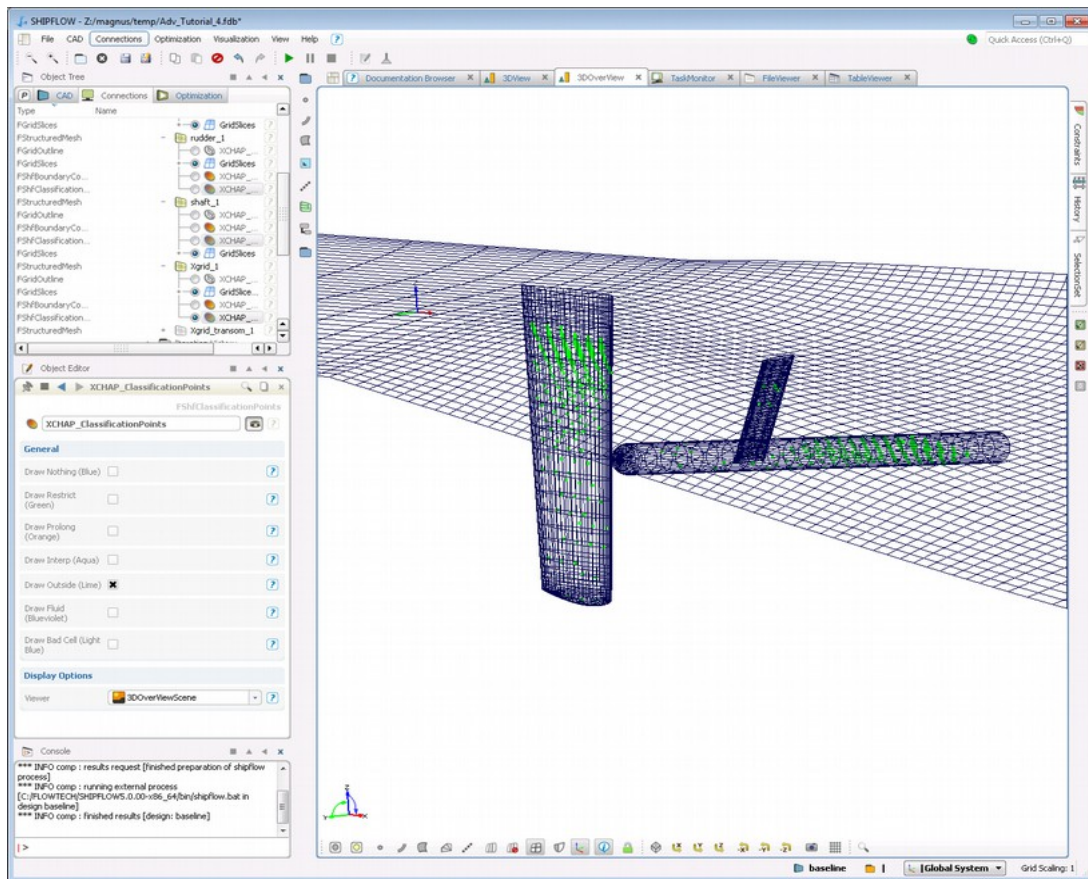
ylplus      max      min      average
          4.48643      0      1.03784

SHIPFLOW Task-1: 2013-04-15 15:13:03-16 Task-2: 2013-04-15 15:03:10
```

- When the computation is finished examine the generated grid. Look at the component grids and the interpolation and outside classification points. Observe how the appendages intersects the hull and each other.

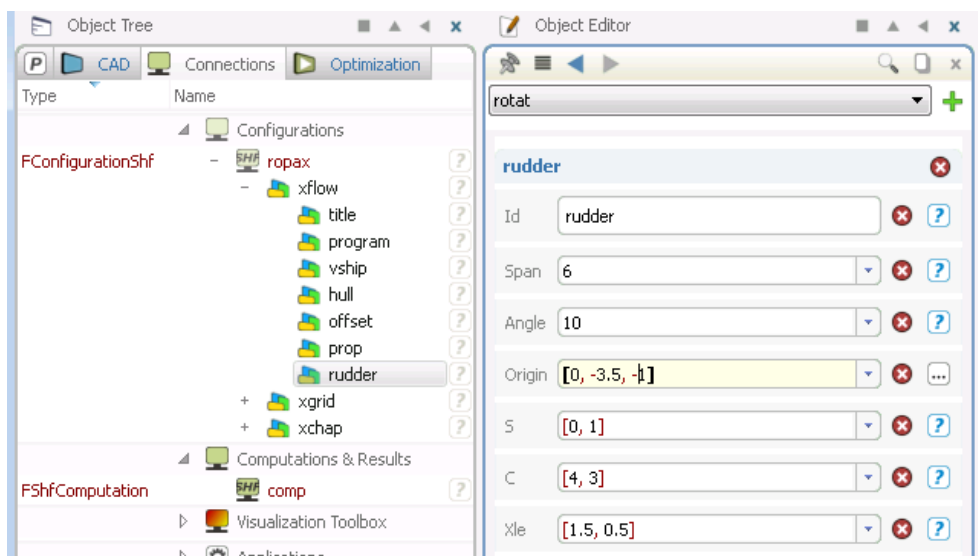






Now the geometry is changed to deliberately create a “leak”

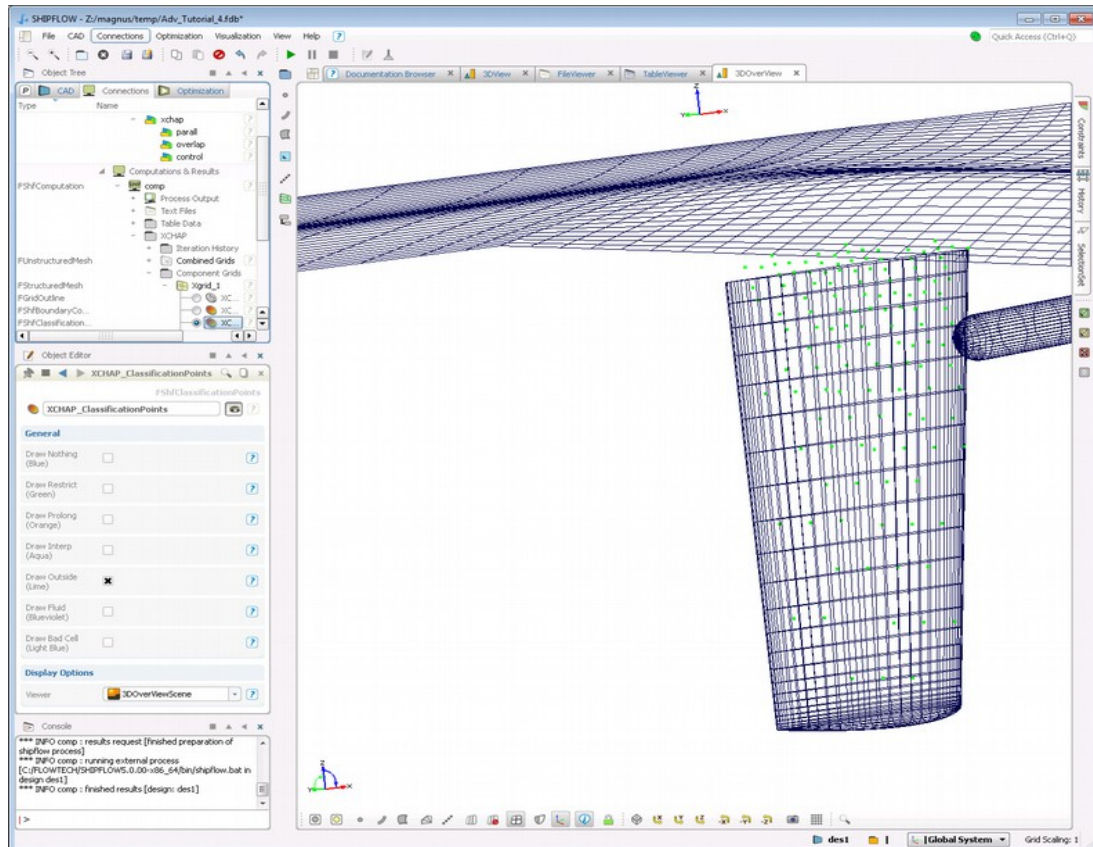
1. Create a Design Variant. Delete the shaft and the bracket from the variant, and change the z-component of the rudder origin to -1. Also add xchap | overlap | noleakcheck.



2. Run the variant.



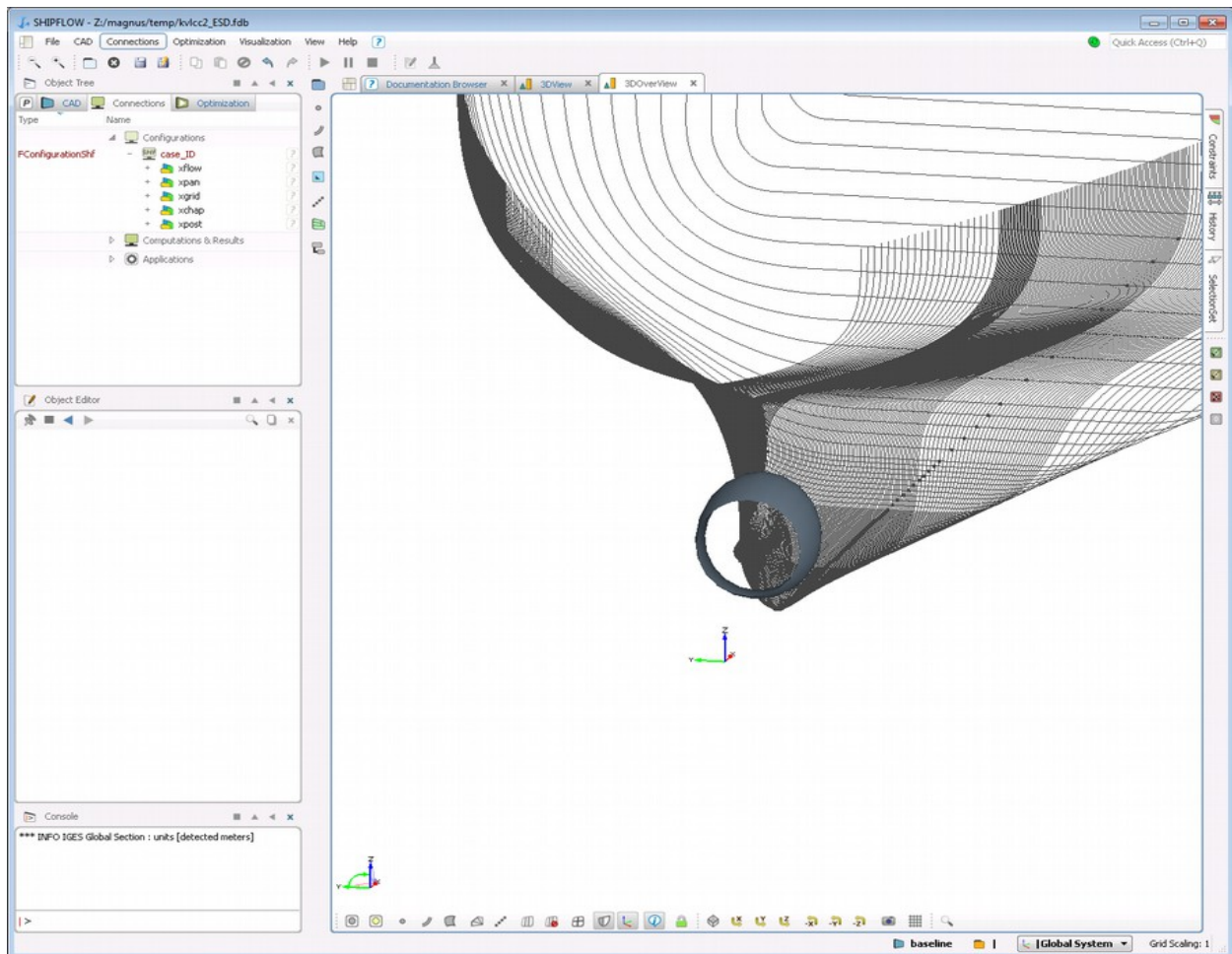
3. The program detects the leak and prints an error message “ERROR: Less than 20% of the cells...” and stops.
4. To find the leak, set the parameter `xchap>overlap>nfill=70` and re-run. Nfill limits the number of recursions so that the outside points don't contaminate the whole domain and that makes the leak easier to find.
5. Re-run the variant and examine the resulting classification. Notice that the Outside cells “spill out” at the top of the rudder since it is an open geometry.



## ***Tutorial 5 part 1 – Complex Appendage modelling***

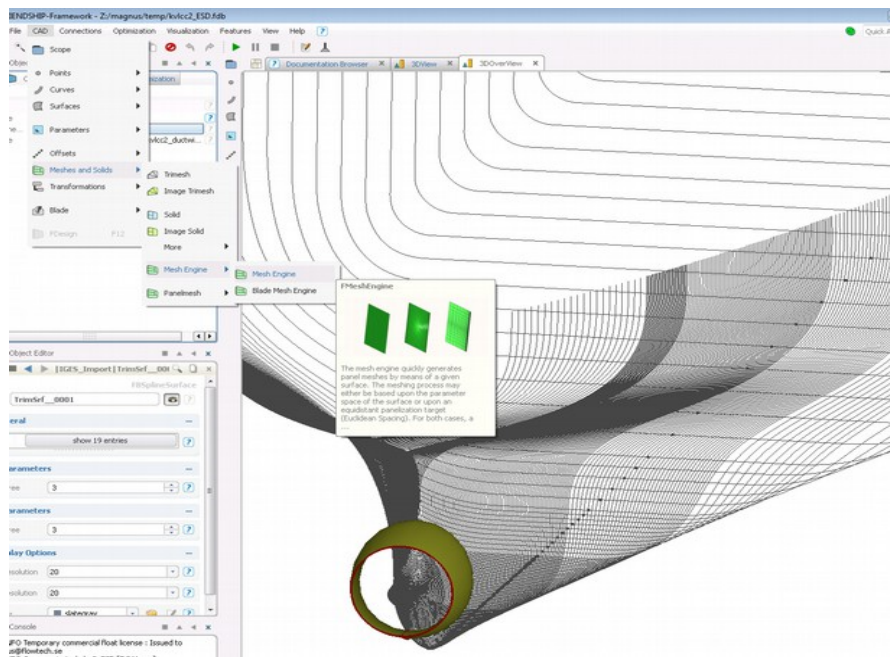
For this tutorial you need to use Friendship FRAMEWORK together with SHIPFLOW.

1. Import case\_ID SHIPFLOW configuration from **DVD:\AdvancedTraining\Tutorial\_Advanced\_5\source**
2. Import duct.igs IGES file.
3. Save the project as kvlcc2\_ESD.fdb
4. Check the setup, make sure you see the offset sections and duct surface on your screen.

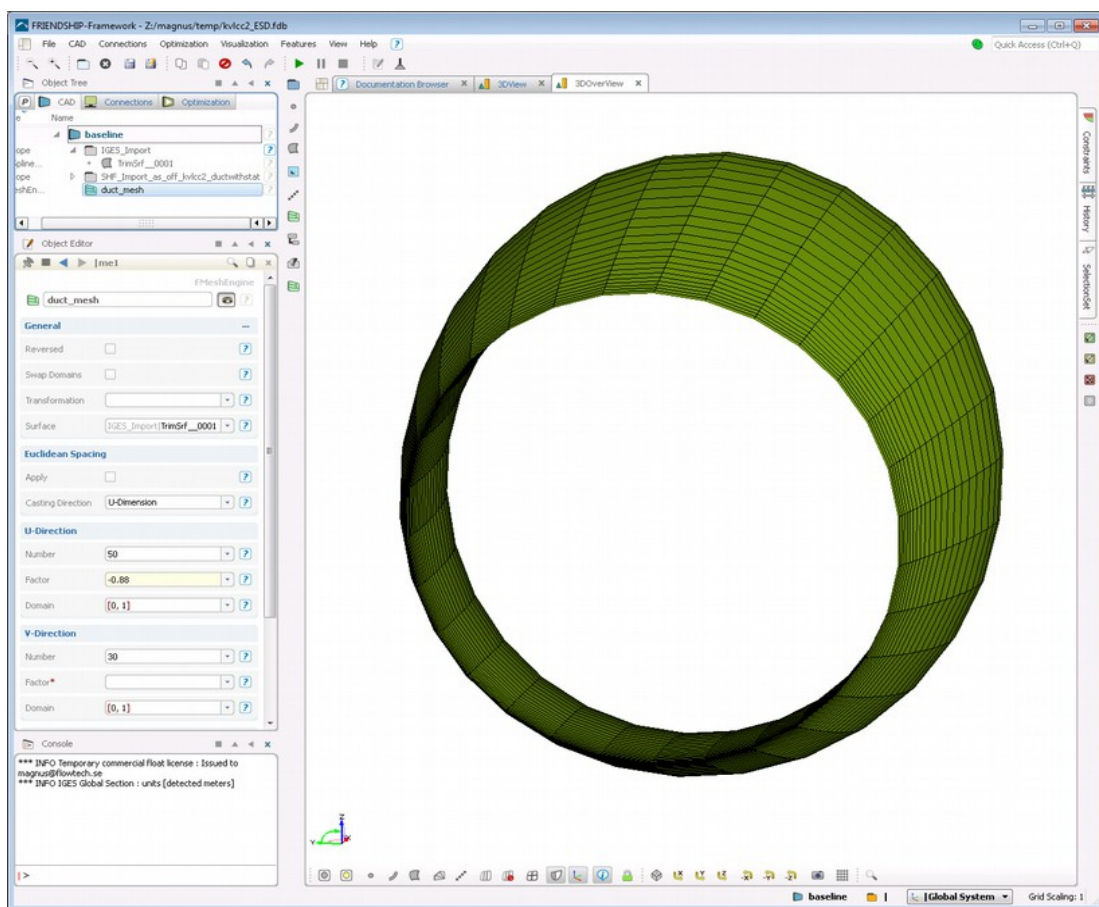


5. In order to create a volume grid for the imported duct grid we shall first make a surface mesh using the surface and thereafter use a hyperbolic grid generator to expand this mesh into 3D.

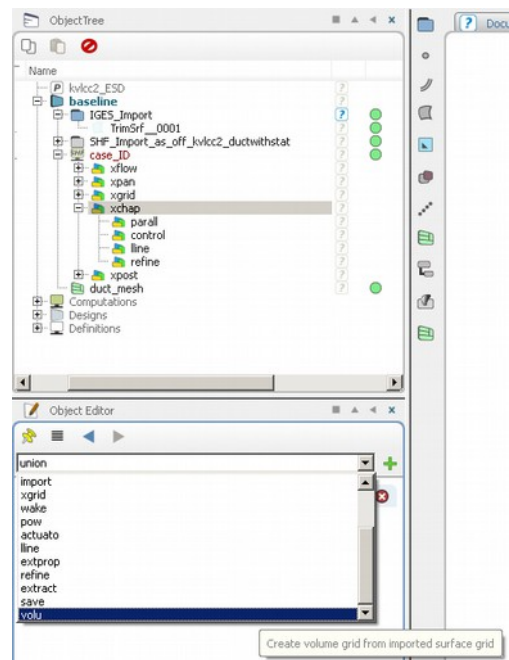
6. Select the duct surface and create a mesh using Mesh Engine, name it duct\_mesh



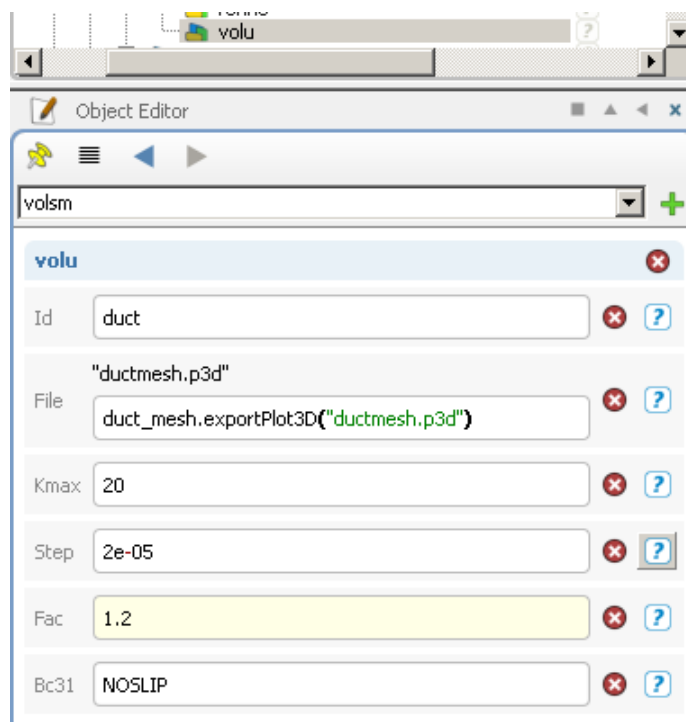
7. Switch off visibility of the offset sections and imported duct surface, notice that the mesh that was just created is extremely coarse and does not represent the object accurately.
8. Refine the mesh dimensions and use stretching factor according to the example below



9. To the xchap configuration add volume object



10. Now we will use the duct\_mesh to create volume grid, apply settings according to the illustration



11. The setup of the duct is ready but we should add support for this structure. We will use two wings created using rudder objects.

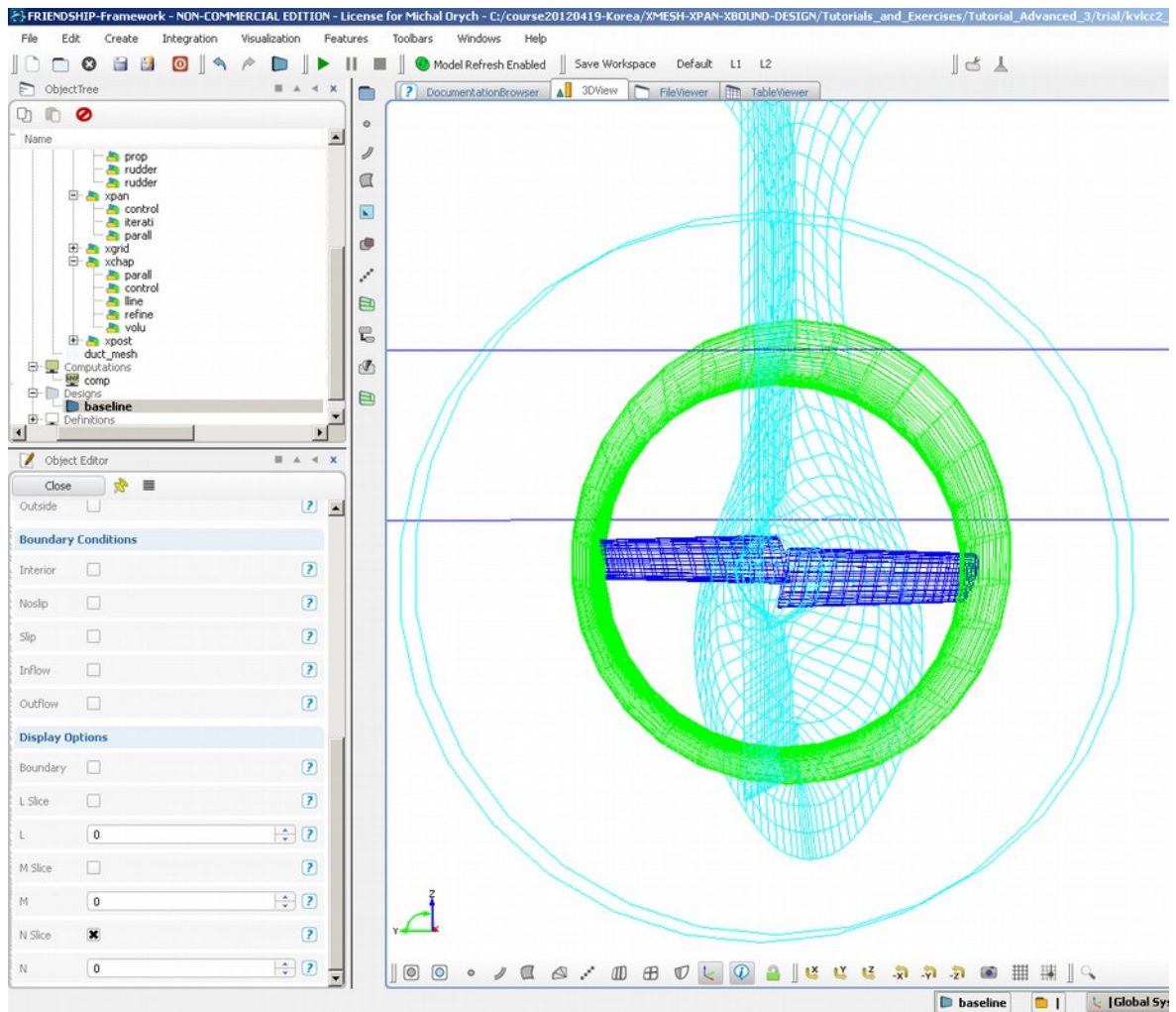
12. In the xflow configuration add two rudder objects with the following settings

rudder		✖	?
Id	rpc	✖	?
Span	3.2	✖	?
Angle	20	✖	?
Cant	90	✖	?
Origin	[7.6, 0, 5.8]	✖	?
S	[0, 1]	✖	?
C	[2.5, 2]	✖	?
Xle	[0.75, 0.5]	✖	?
Dime	[30, 20, 20]	✖	?

rudder		✖	?
Id	rsc	✖	?
Span	3.2	✖	?
Angle	20	✖	?
Cant	-90	✖	?
Origin	[7.6, 0, 5.8]	✖	?
S	[0, 1]	✖	?
C	[2.5, 2]	✖	?
Xle	[0.75, 0.5]	✖	?
Dime	[30, 20, 20]	✖	?

13. Make sure that the number of xchap iteration is set to 0 and start the computations.
14. While the computations are running there should be many warning messages appearing in the TaskMonitor, these usually would not appear. However, since this example case is for demonstration only and uses extremely coarse grids the solver may give various warnings.
15. When the computations are finished, display surface meshes on the duct and supporting it blades as well as on the refinement of the hull. The correct set up should resemble the one below.

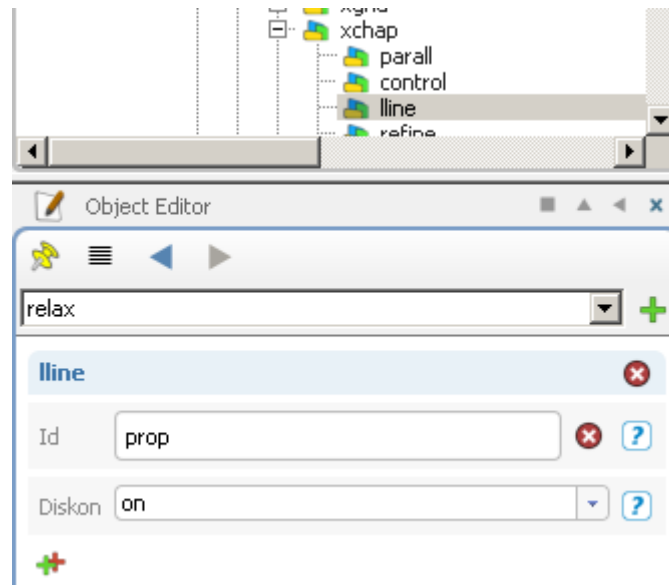




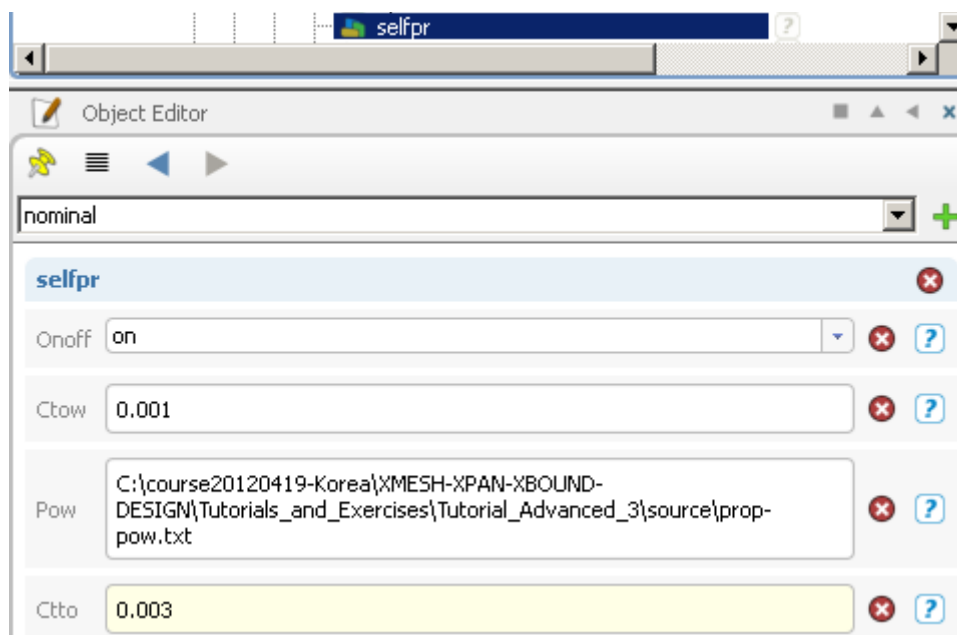
\* Note that the grids used in this tutorial is not fine enough for design applications.

## Tutorial 5 part 2 – Appendage optimization

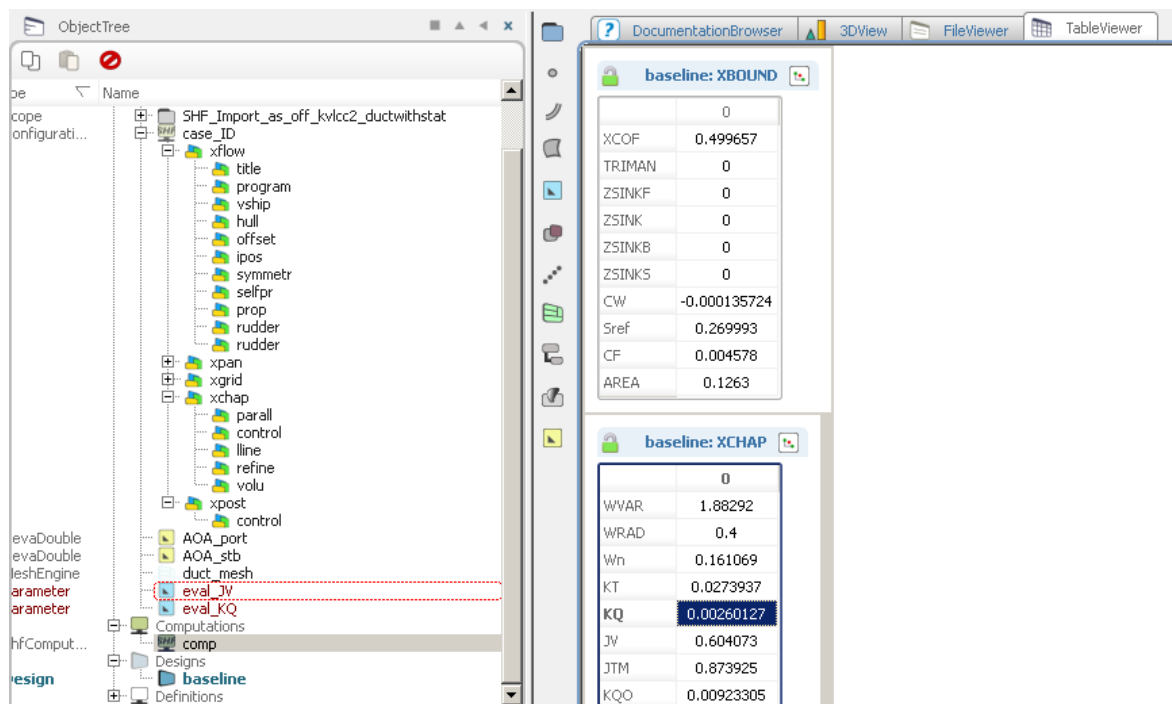
1. Continue the work from the previous part or open the file kvlcc2\_ESD.fdb located in **DVD:\AdvancedTraining\Tutorial\_Advanced\_5\intermediate**
2. We will optimize the supporting blade angle of attack using systematic variations with Ensemble Investigation.
3. First we will have to prepare a self propulsion setup in order to have objective function for the optimization.
4. Turn on the propeller.



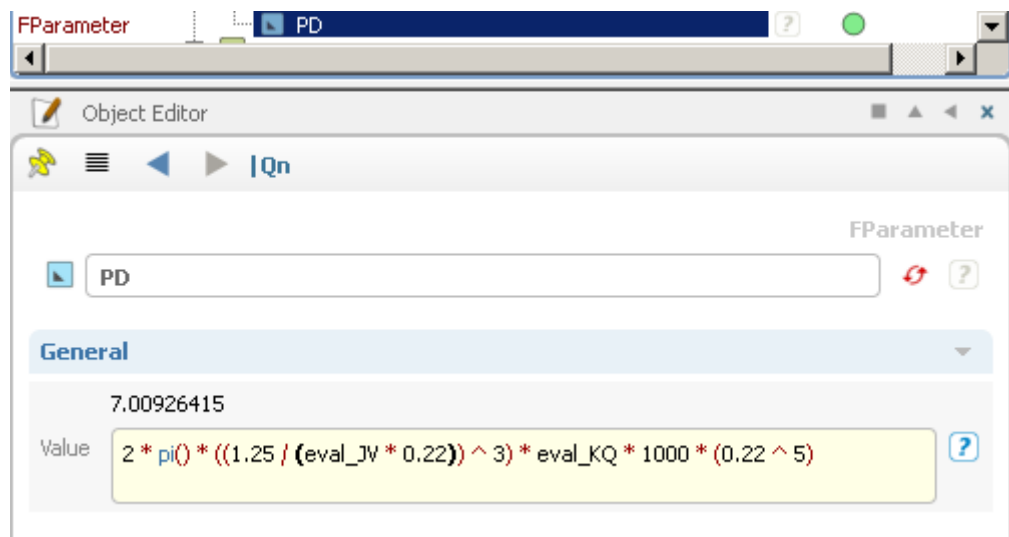
5. Add selfprop command to xflow configuration and use the settings as below, make sure that the Pow command points to the right file which was in source directory for this Tutorial.



6. Set number of xchap iterations to 10 and start the case, it should take about 10 minutes on a laptop
7. When the calculations are finished go to the TableViewer and by double clicking on the KQ and JV create parameters.



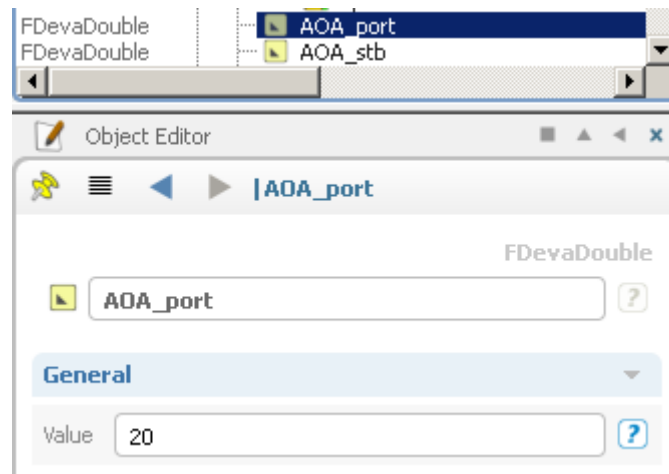
8. Now add additional parameter PD that will represent delivered power using the following formula



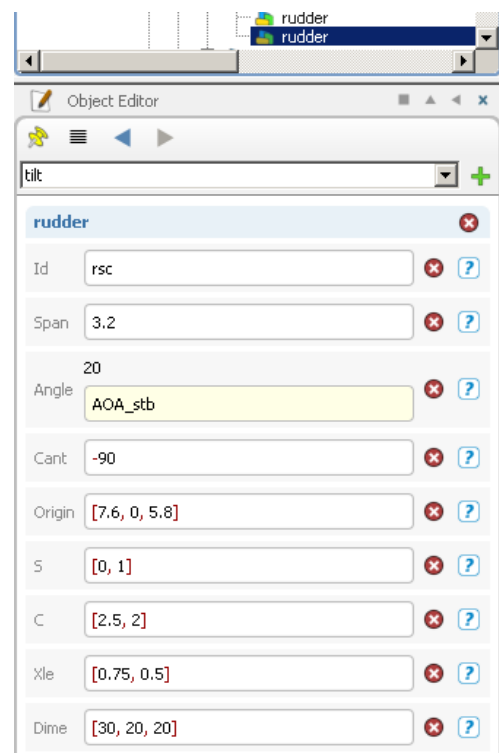
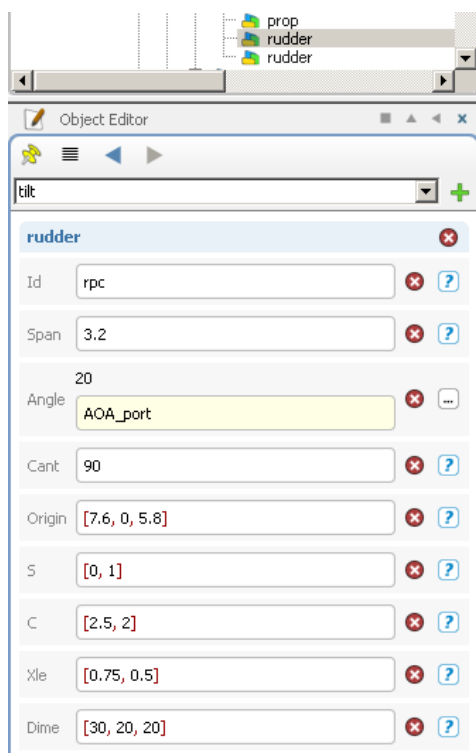
9. Create two Design Variables which will be used as input for angle of attack, name them



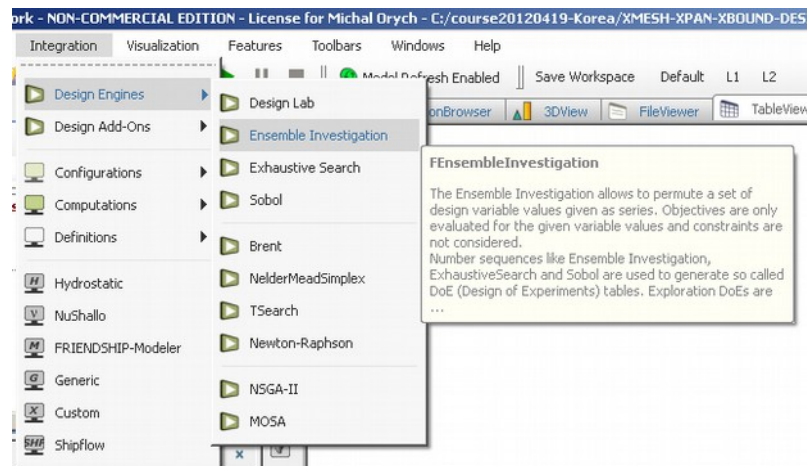
AOA\_port and AOA\_stb and set both to 20.



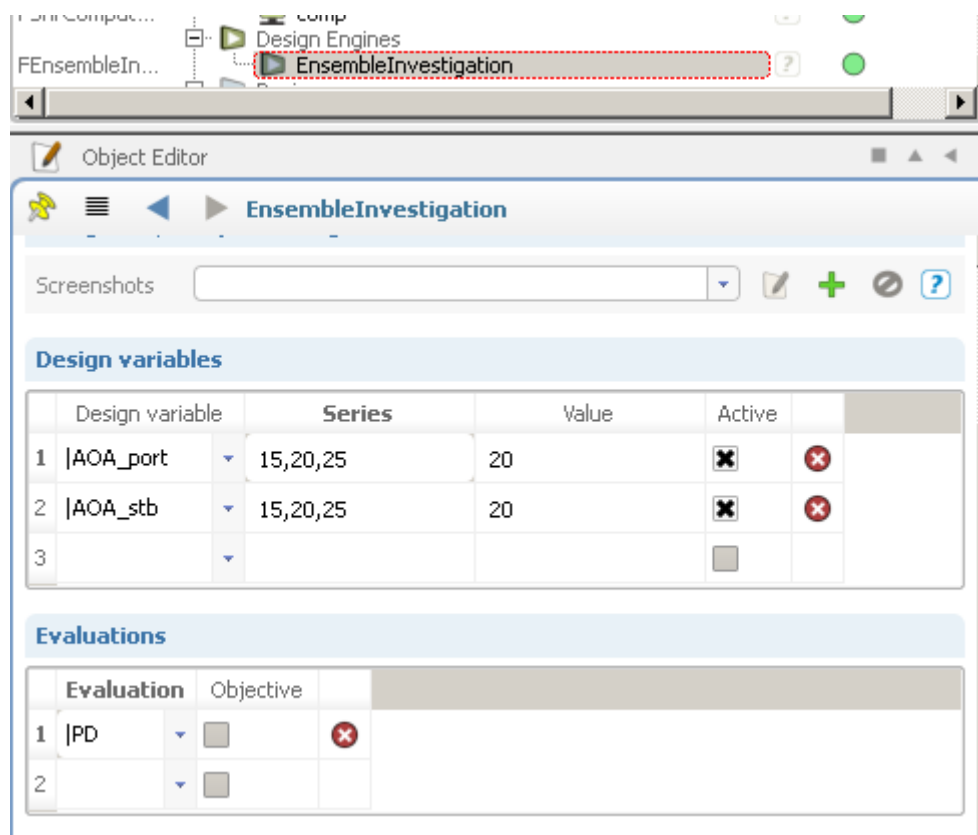
10. In the xflow | rudder configurations replace the Angle with the Design Variables



11. Add Ensemble Investigation

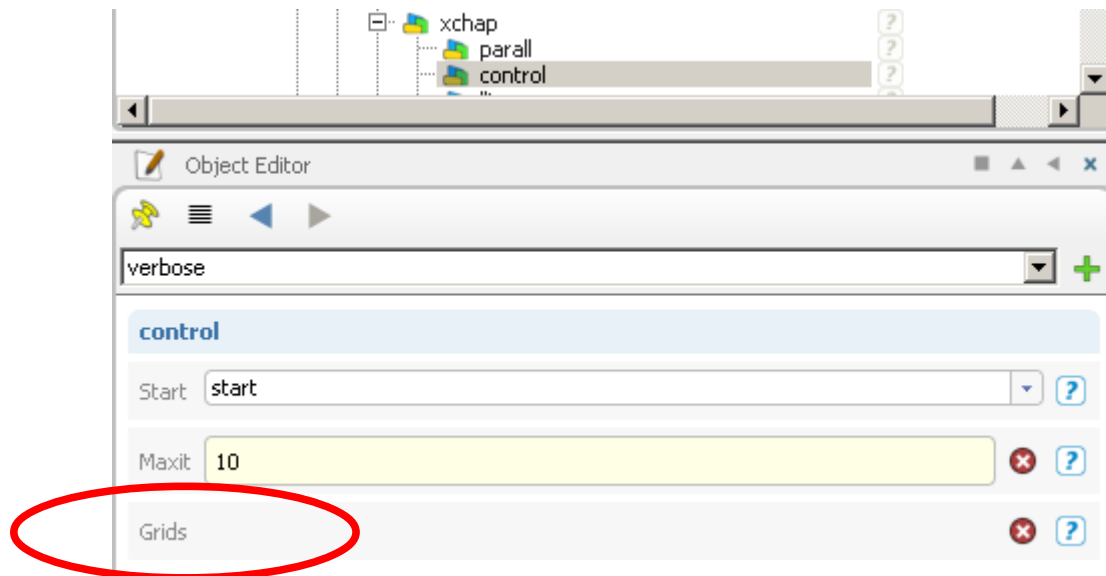


12. Use both AOA\_port and AOA as Design Variables and add variation +/- 5 degrees
13. Use PD for Evaluation of the results.



14. There will be 9 different variants created if you run this case and each should be run until convergence. Moreover, the grids should be much finer to give good results so we will only look at the grid modifications.
15. Go to xchap configuration and in control add grids command to prevent from running the

solver. Also use xchap only in the xflow|program configuration. Now you can start the Ensemble Investigation.



16. When the computations are finished check the different variants and verify that the angle of attack was varied appropriately. Since the actual xchap solver was not run there will be no valid flow calculations.

EnsembleInvestigation_1_results				
	AOA_port	AOA_stb	PD	
Attribute	Active	Active		
Name	AOA_port	AOA_stb	PD	
Scope				
Reference				
Lower Bound	15	15	nan	
Upper Bound	25	25	nan	
Feasible Designs: 0 %				
Mean Utilization Index				
Mean	20	20	0	
Sample Standard Deviation	4.330127	4.330127	0	
Error-free: 0 %	100 %	100 %	0 %	
EnsembleInvestigati...	15	15	nan	
EnsembleInvestigati...	15	20	nan	
EnsembleInvestigati...	15	25	nan	
EnsembleInvestigati...	20	15	nan	
EnsembleInvestigati...	20	20	nan	
EnsembleInvestigati...	20	25	nan	
EnsembleInvestigati...	25	15	nan	
EnsembleInvestigati...	25	20	nan	
EnsembleInvestigati...	25	25	nan	

- \* The calculation execution with the Grids keyword will result with PD = nan.
- \*\* Keyword Grids is used only to check correctness of the grids, RANS solver is not executed
- \*\*\* Note that the grids used in this tutorial is not fine enough for design applications.