Ref. Ares(2020)546558 - 29/01/2020

AQUA EXCEL 2020

AQUAculture infrastructures for EXCELlence in European fish research towards 2020 — AQUAEXCEL2020

Final flow field model after testing period

Morten Omholt Alver

This project has received funding from the European Union's Horizon 2020 research and innovation programme under grant agreement No 652831. This publication reflects only the view of the author, and the European Commission cannot be held responsible for any use which may be made of the information contained therein

Executive Summary

Objectives:
The purpose of this document is to describe the functionality and technical implementation of the flow field model. This numerical model is one of the main components in the AQUAEXCEL2020 virtual laboratory developed in WP5: "Virtual laboratories and modelling tools for designing experiments in aquaculture research facilities".

The main components of the virtual laboratory are:

- Growth, nutrition and waste production models for different fish species
- Water quality and water treatment model
- Model of hydrodynamic flow fields in tanks and cages

The objective of the flow field model is to represent the water currents within the production unit - fish cage or tank, presenting key information relating to the current to the other model components.

Rationale:
One of the main research activities in AQUAEXCEL2020 is to develop a virtual laboratory system that enables virtual experiments in aquaculture research facilities. This system will feature a framework that allows the integration of mathematical models of different subsystems in common simulations, replicating the system operation of research laboratories.

Main Results:
The flow field model is developed and tested, and it is shown that this model component can be integrated with the other main components. The flow field model uses one approach for current in tanks – precomputed flow fields from a Computational Fluid Dynamic (CFD) model – and another for open sea net cages – current profiles depending on ambient current conditions. The model interacts with the other model components either through providing the current speed and direction vector for given locations, or through providing descriptive numbers for the overall flow field in the production unit.

Authors/Teams involved:
Morten Omholt Alver, Martin Føre (NTNU)
Eleni Kelasidi, Finn Olav Bjørnson, Gunnar Senneset (SINTEF)
Štěpán Papáček (USB)

Table of contents

1. Background

This document is part of the AQUAEXCEL2020, WP5/Joint Research Activity 1 – Virtual laboratories and modelling tools for designing experiments in aquaculture research facilities.

Experiments with fish usually involve extensive use of laboratory facilities and run for long periods of time. Both from an ethical perspective (3R's) and from a cost perspective, tools for design and planning of experiments are increasingly important. In aquaculture research as well as other domains, numerical models are increasingly used preparatory to the actual experiments.

One of the main research activities in AQUAEXCEL2020 is to develop a virtual laboratory system that enables virtual experiments in aquaculture research facilities. This system features a framework (see Bjørnson et al., 2016; 2019) that allows the integration of mathematical models of different subsystems in common simulations, replicating the system operation of research laboratories. The overall system architecture is shown in Figure 1.

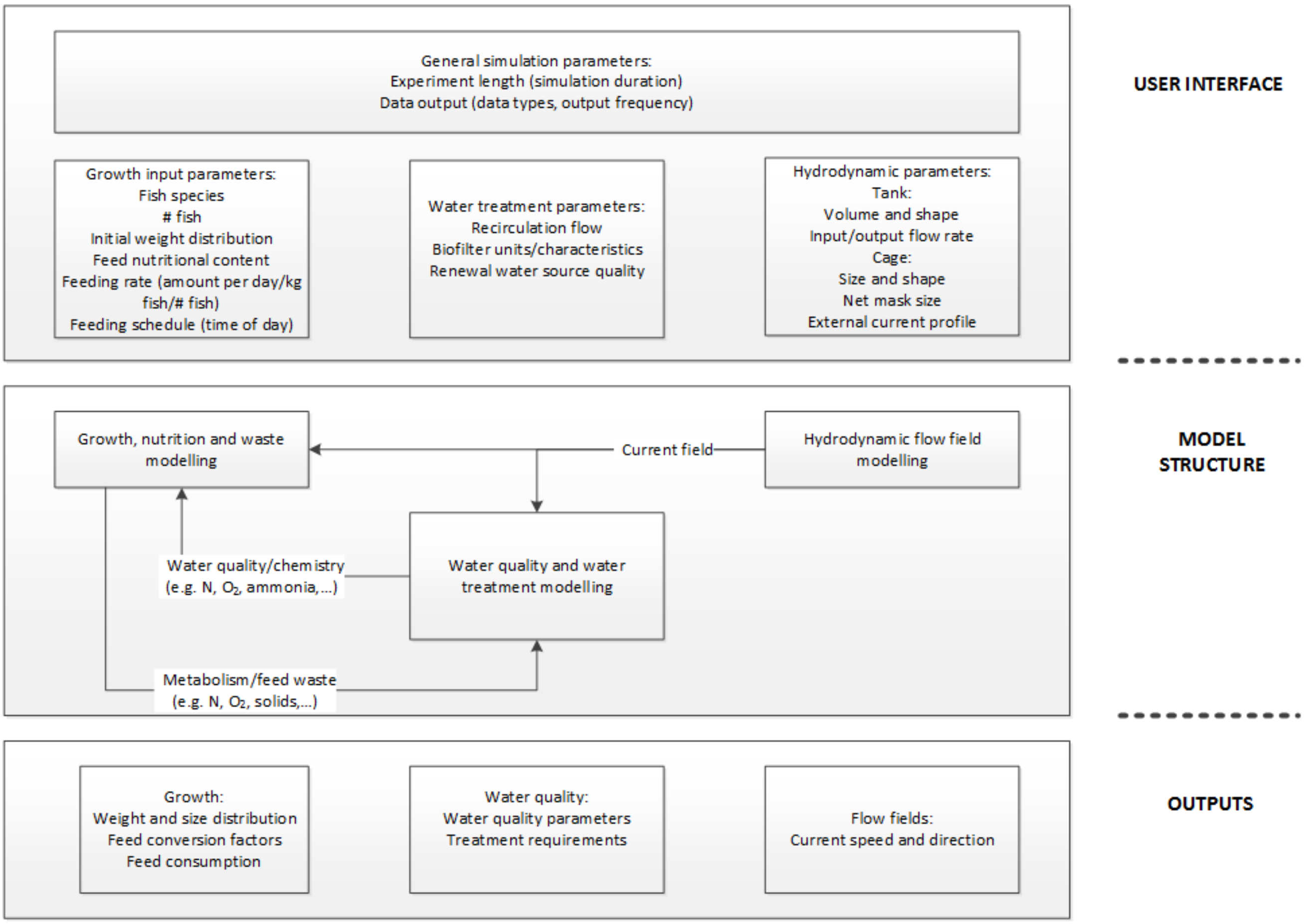

Figure 1 Virtual laboratory system architecture

This document describes the final version of the flow field model in terms of technical implementation and functionality. The earlier report *D5.4 First prototype flow field model established* (Alver et al. 2018) described the initial model version. For completeness, this report repeats the parts of D5.4 that are still relevant, and adds descriptions of new and modified functionality.

2. MODEL DESCRIPTION

Background

The flow field of production units, either tanks or net cages, may affect the fish in a number of ways. Most obviously, the current at the location of each fish represents a motion superimposed on the fish's own active movement. The current also serves to transport and mix particulate and dissolved matter in the water, such as feed pellets and concentrations of oxygen and ammonia, all of which may have effects on fish welfare and performance.

The flow fields are very different in tanks and in net cages. In tanks, the walls are closed so water only enters and leaves the tank through designated water inlets and outlets. In net cages, water flows through the cage sides, causing a net transport across the cage, typically supplemented by complex turbulent variability caused by the cage netting and the fish movement. In case of strong current, the fish may be forced to swim up against the current to keep their position in the cage. Tanks and net cages need to be handled using different tools, and in both cases exact currents are hard to measure, and hard to model mathematically.

Flow fields in tanks

Flow fields in tanks, disregarding the effect of the fish, can be computed using Computational Fluid Dynamics models (e.g. Sayma *et al.* 2009). The fundamental basis of these models is the Navier-Stokes equations which are used to describe the motion of viscous fluids. CFD models solve a form of Navier-Stokes numerically after discretizing the simulation volume. CFD models typically use non-uniform discretization schemes where the spatial resolution varies over the simulated volume, with higher resolution around features such as inlets. The process of designing the model grid is a separate operation that may require a large number of computations. Numerical solution of the equations on the model grid is also computationally demanding, and models are typically parallelized and run on high performance computing centres. The output, resulting 3D flow field and related hydrodynamic quantities, may be either steady (independent on time) or unsteady (time-dependent). For the purpose of flow fields in tanks, we are looking for the steady-state field for a given tank setup and water in-flow rate, so the CFD model will be run to produce a constant 3D field representing steady state flow (see examples of flow field and streamlines in Figure 2 and Figure 4, respectively).

Figure 2 shows velocity contours in three planes above the tank bottom for the Research Institute of fish Culture and Hydrobiology Vodnany (VURH) tank. The upper contour plot shows the higher velocities originating from a water inlet. The other dominating features are a lower current velocity closer to the centre of the tank, and a slight reduction of current near the tank wall except those areas affected by the water inlet. Figure 3 shows velocity contours in three planes, and Figure 4 shows streamlines, for the HCMR tank. The streamlines indicate how water moves through the volume, with the color of each line indicating speed. The figure by necessity just shows a small subset of all possible streamlines in the tank. Figure 5 shows velocity contours in three planes for the NOFIMA tank. The flow field displayed in these figures may be accompanied by a computed turbulence kinetic energy (TKE) field describing the level of turbulence estimated over the volume.

There is a wide selection of open source software (e.g. OpenFoam, Typhon and Reef3D) and commercial software (e.g. Ansys Fluent, CFX and COMSOL) that basically solve the same mathematical equations but have different features and different learning curves.

CFD code ANSYS Fluent was used in order to provide numerical simulations of the stationary flow of incompressible viscous fluid (water) in 3-D domain corresponding to three different tanks located at (i) VURH Vodnany, (ii) HCMR Heraklion (see e.g. Lika et al. 2015), and (iii) NOFIMA Sunndalsøra. The properties of the tanks are summarized in Table 1. All calculations were performed using a Reynolds-averaged Navier-Stokes equation system and the Intermittency k-omega SST turbulence model. Two typical velocity flow field visualizations, i.e., velocities in three horizontal sections (bottom, medium, surface), see Figure 2 and Figure 5, and streamlines in whole volume, are presented, see Figure 4.

Table 1: Properties of the simulated tanks

Tank	Type	Width (m)	Height (m)	Volume (l)
VURH	Circular flat bottom tank	1.465	0.84	Ca. 1400
HCMR	Cylindroconical tank	1.01	0.93	Ca. 545
NOFIMA	Octagonal exp. tank	1.8 / 1.75	1.0	Ca. 3000

Validation of the VURH tank model output has been made by comparing velocity measurements using Acoustic-Doppler Velocimeter (ADV). These measurements are pointwise with a range of ±1 cm s−1 (accuracy of is ±0·5% of the measured value). The mean deviation is on the order of ten percent (Hanak 2016). Based on the comparison of experimentally measured and simulated values of velocity profiles, we argue that CFD code ANSYS Fluent represents a modern and reliable tool for the simulation of hydrodynamic flow field within a production unit although it does not have the option of simulating the effect of fish swimming activity. However, by making assumptions of simplified fish swimming patterns (e.g. with the fish swimming in circular motion against the current direction at their preferred speed) and based on recent experimental measurements, cf. Gorle *et al.* (2018), one may include the effects related to fish biomass, making the CFD results more accurate.

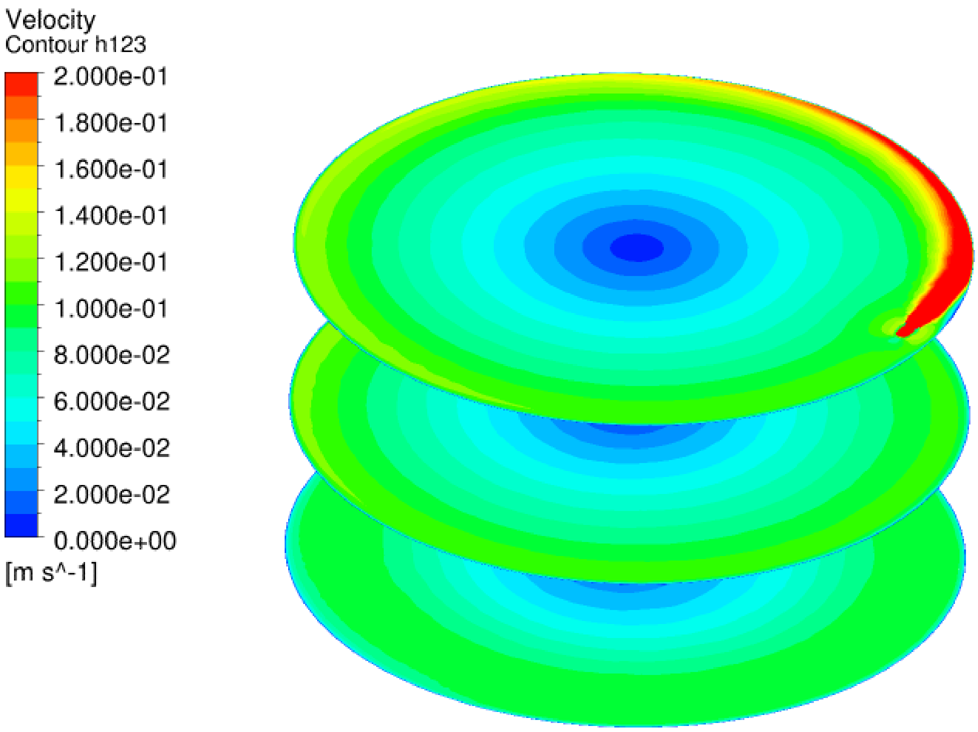

Figure 2: Example of velocity flow field in VURH tank, computed using Ansys Fluent.

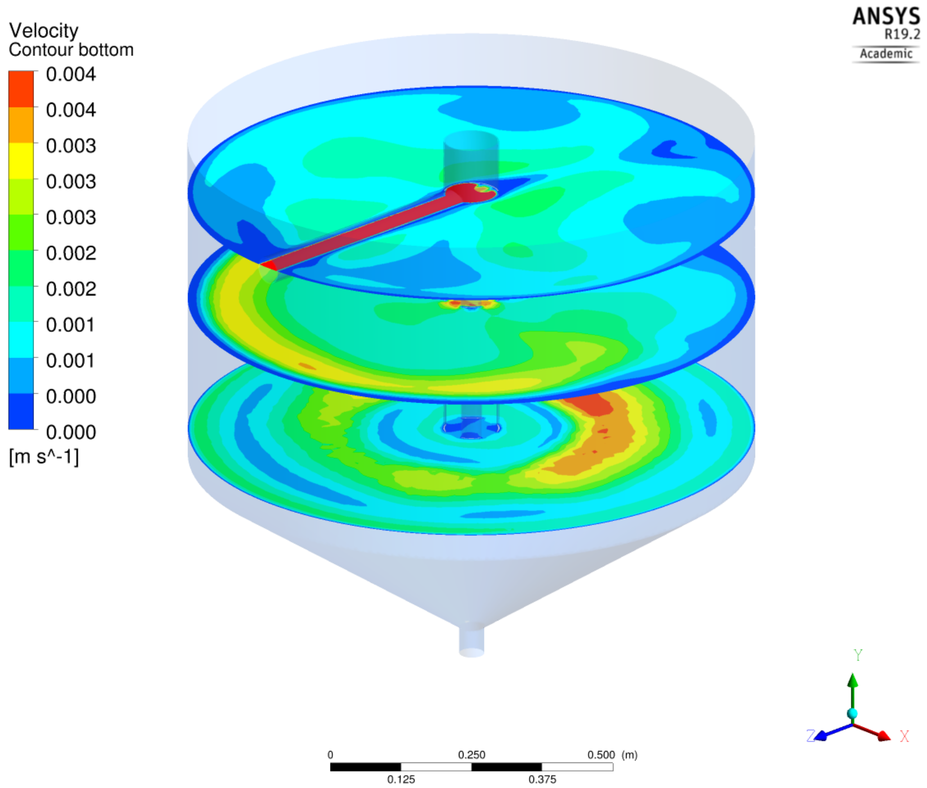

Figure 3: Example of velocity flow field in HCMR tank, computed using Ansys Fluent.

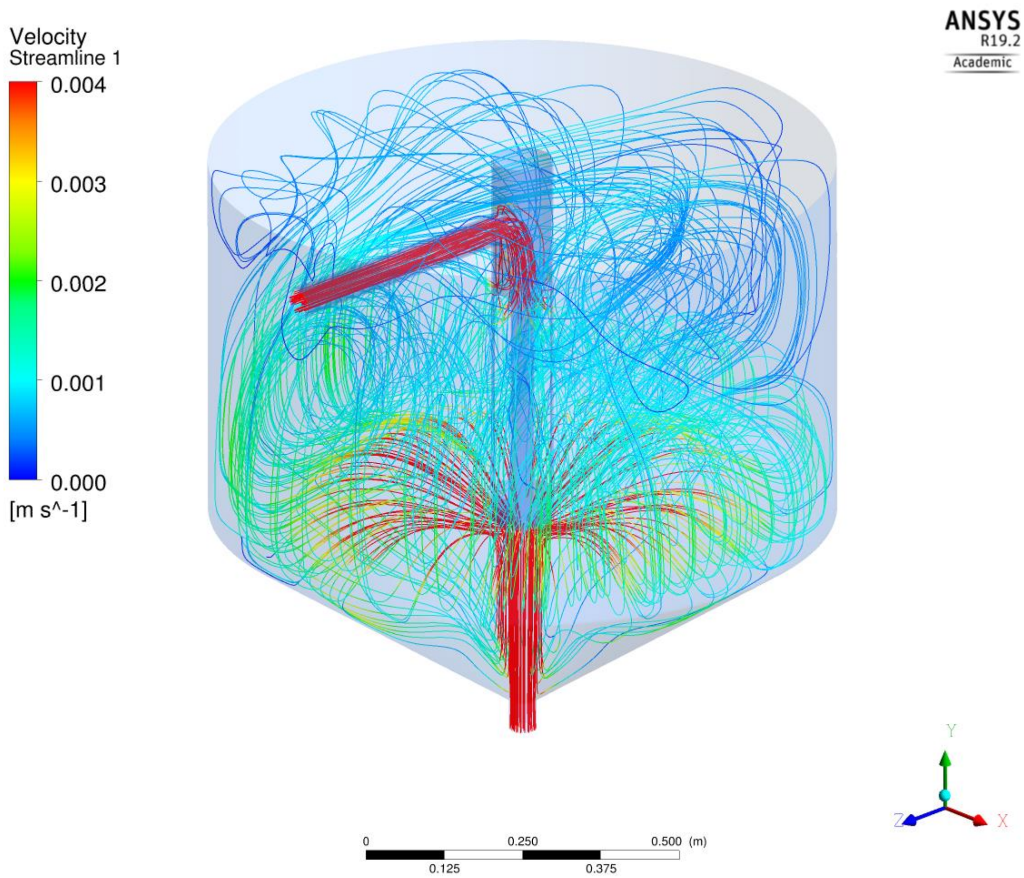

Figure 4: Streamlines illustrating the flow pattern in the HCMR tank.

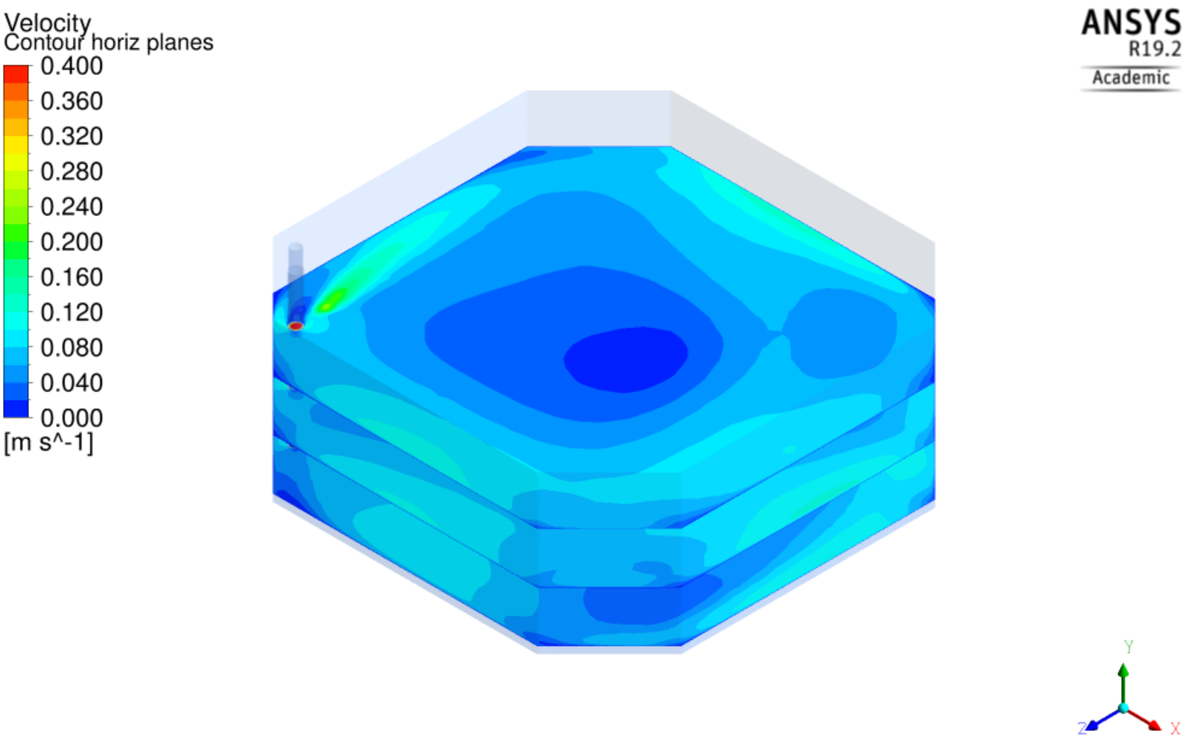

Figure 5: Example of velocity flow field in NOFIMA tank, computed using Ansys Fluent. The figure shows velocity contours in three planes above the tank bottom.

Flow fields in cages

In the case of cages, no established models exist to simulate currents as function of outside current and the activity of the fish. The current outside the cage at a given location may be monitored using point or profiling current meters, and modelled using regional ocean models such as SINMOD (Slagstad & McClimans 2005) or ROMS (Shchepetkin & McWilliams 2005). The vertical current profiles observed at an aquaculture location depend on which processes dominate the dynamics at the location. Some of the relevant processes are tidal dynamics, wind induced currents and stratification caused by freshwater run-off. Aquaculture farms may generally be categorized according to the type of current and hydrography conditions they experience. In the Norwegian salmon industry, for instance, there are some sheltered fjord locations, with limited wind and waves, high stratification and mainly tidal currents and estuarine circulation, and more exposed locations with stronger wind and wave effects, less pronounced tidal currents and with current influenced by the Norwegian Coastal Current.

The current speed inside of the cage is dependent on outside current and the properties of the cage (Løland 1993). The outside current for each individual cage in a fish farm depends also on the cage's position, as the wake from upstream cages will affect the current speed. Cages deform in strong currents, making the dynamics more complicated (see Figure 6). Løland (1993) developed a method for calculating the current forces on a net structure and the resulting wake, including estimates of current inside the cage, and compares results to laboratory scale model test. An approximate inside current speed of 85% of the ambient current speed is found for a reasonable set of parameters. Endresen et al. (2013) used a net cage model implemented in SINTEF's FhSim modelling framework (Reite et al. 2014) to estimate wake effects acting within an aquaculture net cage, comparing numerical estimates with experimental data. An approximate inside current speed of 80-85% of the ambient current speed was found.

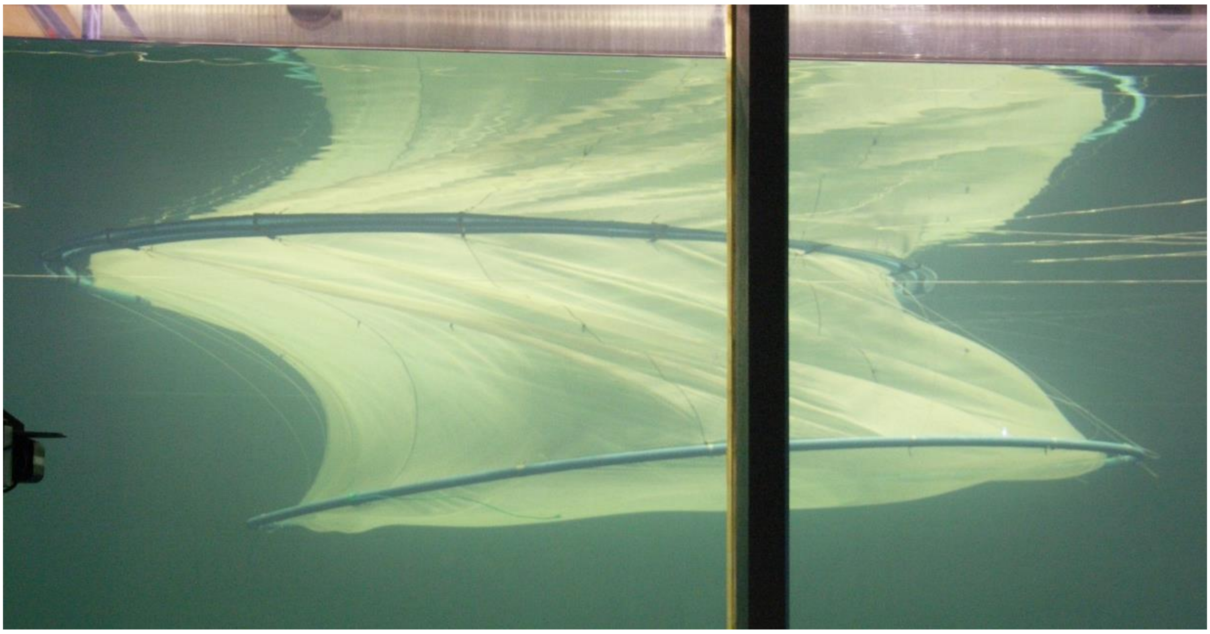

Figure 6: Photograph of downscaled net cage model in a laminar current field (Photo: SINTEF Ocean).

As with CFD modelling of tanks, the effect of the fish is difficult to account for. Models describing the current flow field inside net cages in more detail may become available through ongoing research, but at present the best approach is to do a simple parameterization based on the ambient current profile to estimate the flow field inside.

Average profiles

To obtain representative current speed values for Atlantic salmon cage culture in Norway, we have used numerical ocean model results calculated using the SINMOD model (Slagstad & McClimans 2005). The regions of Middle Norway were simulated at 160 m horizontal resolution for a full year, and current statistics were found for actual aquaculture locations in the area. The simulations were done by SINTEF for the aquaculture industry, and model results are presented at the web page http://midtnorge.sinmod.com (login required for data access).

We made a selection of open locations and fjord locations, where open refers to relatively exposed locations along the coast, and fjord refers to locations inside fjords. For each of the locations, we recorded the highest and lowest monthly average current speed at every 5th m through the water column from 0-35 m. This resulted in a "high" and a "low" current profile for each location, representing high and low average values, but not extreme conditions; one can expect all of these locations to have significantly higher maximum current speeds. For each category of locations (open and fjord), we computed the mean of the high and low profiles to obtain reasonable high and low current profiles for each.

Figure 7 summarizes the high and low speeds estimated at the selected locations, as well as the mean values over the locations in each case. Both types of locations tend to have decreasing current speeds from the surface downwards, but fjord locations show a sharper transition from a shallow surface layer. This is due to stronger stratification in fjord locations compared to open locations.

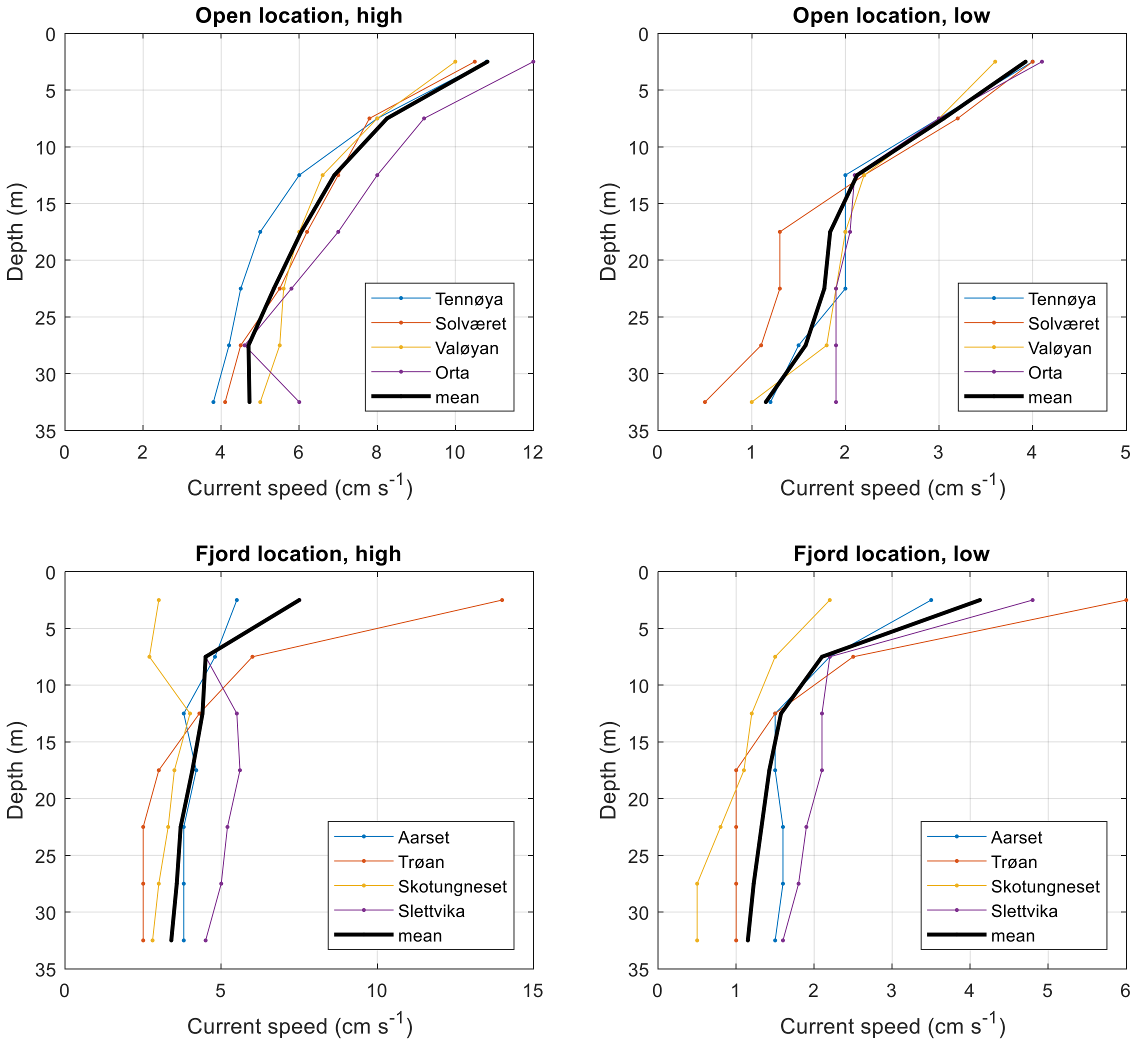

Figure 7: Representative high and low current profiles at four open locations and four fjord locations, and profiles averaged over the four locations in each case.

Tidal cycles

It may be desirable for the modeler to include tidal variations in the current speed in a net cage. Tides are caused by tidal forcing from the Sun and Moon, and propagate depending on the bathymetry and coast lines in the ocean. Due to the difference in period of the forcing from the Sun (12 h) vs. the Moon (12.42 h), the weighted sum of these components leads to a near-monthly oscillation in the tidal amplitude, causing what is known as *spring* and *neap* tides. The daily oscillations in surface elevation have a period of approximately 12.2 hours. Effects with other periods also affect the signal, usually to a smaller degree. The specific tidal signal at specific locations varies greatly, and can be estimated using mathematical models. For the purposes of the Virtual Laboratory, a tidal cycle representing a typical situation is considered sufficient.

Disregarding the oscillation between *spring* and *neap* tides, a typical situation can be represented with a *sine* shaped variation in sea surface elevation. There is no need to explicitly represent the elevation, but rather the variation in current speed caused by it. Tidal current variations can be approximated by a multiplicative factor proportional to the derivative of the elevation (which is a *cosine* wave, or a *sine* since the phase of the signal is not important here); the speed is 0 at high and low tide, and reaches a maximum as the ebb or flow rate is highest at medium elevation. When considering the absolute value of the current speed, the variation is shaped as the absolute value of a *sine* wave.

Since tidal variation typically accounts for only a part of the current speed at a given location, it is reasonable to add a constant value to the multiplier to ensure the current speed is not set to 0 at medium tide. Taking all this into consideration, the function for the current speed multiplier (M_{speed}) can be formulated like this:

$$M_{speed} = K_{const} + K_{tidal}|\sin(2\pi \cdot t_{hours}/12.2)| \quad (1)$$

Where K_{const} is the minimum value of the multiplier, K_{tidal} is the amplitude of the variations and t_{hours} is the time in hours.

The average value of M_{speed} is K_{const} + K_{tidal} * 2/π. Since the average current speed is set according to location type, it makes sense to design for an average multiplier value of 1. To achieve this, the parameters should be chosen according to this relationship:

$$K_{const} = 1 - K_{tidal}\frac{2}{\pi} \quad (2)$$

Figure 8 shows a time plot over 24 hours of the multiplier with K_{tidal} = 1, and K_{const} = 0.3634, chosen to get an average value of 1.

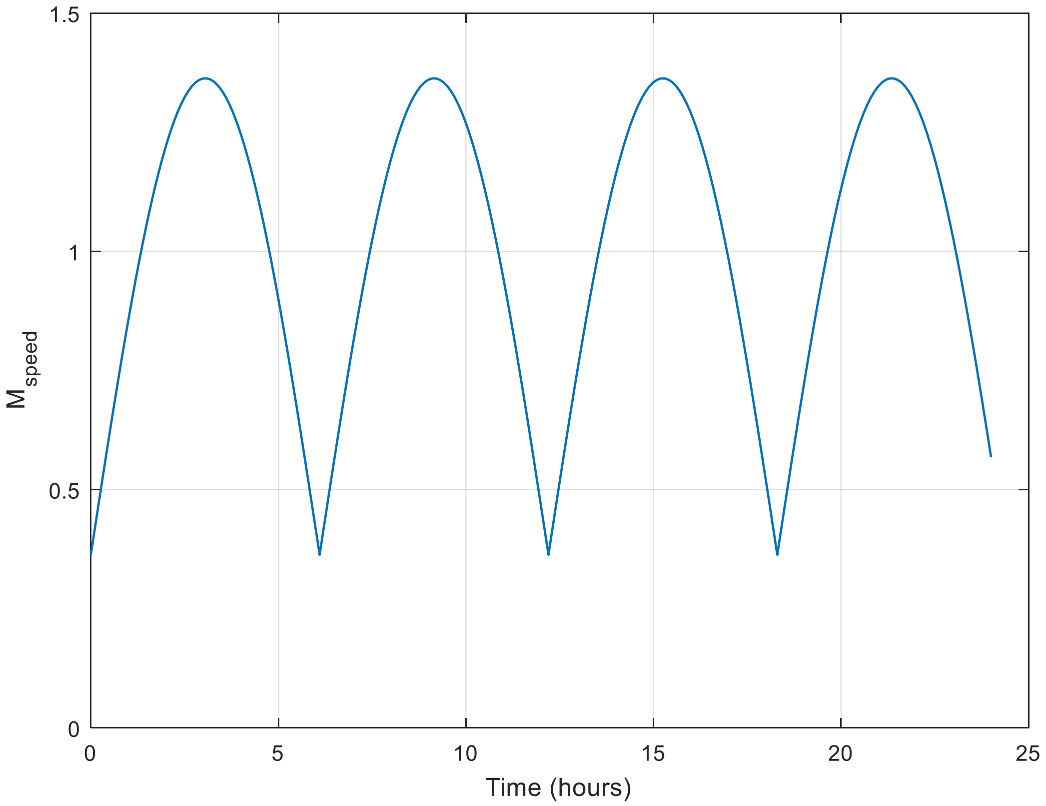

Figure 8: Tidal variations over 24 hours with K_{tidal} = 1 and K_{const} = 0.3634. The plot shows the multiplier applied to the original speeds of the current profile to get tidally influenced speeds.

Model functionality

In the case of tanks, due to the computational cost, computing a flow field based on user inputs is time consuming and requires high performance hardware. Therefore, it is not practical to run the current model on demand through the virtual laboratory web interface. The solution is to provide precomputed flow fields for a selection of tank designs and operating conditions. These precomputed fields are stored in NetCDF files[1] structured in a way suitable for the flow field model to read. As long as the output is stored or convertible to a suitable format, any CFD model may be used for this purpose.

The precomputed flow fields for tank designs at VURH, HCMR and NOFIMA will be available to the user for selection. Additional precomputed flow fields may be added in the future without change to the flow field model code. The web interface may include a mechanism for users to request runs of new tank designs.

In the case of cages, the flow field is estimated based on outside current conditions in the form of vertical profiles of current direction and speed. The user will be able to either choose and adjust typical profiles, or input their own. The vertical current profile to use will be written in NetCDF format for access by the model. Again, additional precomputed profiles may be added in the future without changing the model code.

In either case, the key functionality of the flow field model component is to provide information about the water current in the tank or cage. The model provides two sets of outputs – one that contains 3D current components (e.g. current upwards, towards the east and towards the north) at a requested position, and another that provides overall descriptive values characterizing the flow field.

Model interface

The initialization of the model requires a path to a data file in NetCDF format containing the precomputed flow field or current profile. This file will be opened upon startup of the model, and will be accessed through the course of the simulation to extract the information needed to compute the requested model output values. The model automatically recognizes whether the file contains a tank flow field or a cage profile, and configures itself accordingly.

Table 2 lists the initialization parameters required by the model. This list may be expanded in future versions of the model.

Table 2: Model initialization parameters

Name	***Type***	***Data type***	***Description***
filename	Input	String	Path to NetCDF file containing flow field or current profile
tidal	Input	Numerical	0 to disable tides, 1 to enable

Input values

The input values to the model are the x, y and z coordinates of a requested position (listed in Table 3).

Table 3: Input values

Name	***Type***	***Unit***	***Description***
xPos	Input	m	Requested position along the *x* axis
yPos	Input	m	Requested position along the *y* axis
zPos	Input	m	Requested position along the *z* axis

[1] For information about the NetCDF file format, see https://www.unidata.ucar.edu/software/netcdf/

Output values

The module outputs the current vector for the requested position, as well as a selection of descriptive values for the flow field. Output values are listed in Table 4.

Table 4: Output values. Values marked with (*) are only available if they can be read or calculated from the input NetCDF file.

Name	***Type***	***Unit***	***Description***
xVel	Output	m/s	Current speed along the *x* axis
yVel	Output	m/s	Current speed along the *y* axis
zVel	Output	m/s	Current speed along the *z* axis
vAvg	Output	m/s	Average speed
vMin	Output	m/s	Minimum speed
vMax	Output	m/s	Maximum speed
minX	Output	m	X position of minimum speed
minY	Output	m	Y position of minimum speed
minZ	Output	m	Z position of minimum speed
maxX	Output	m	X position of maximum speed
maxY	Output	m	Y position of maximum speed
maxZ	Output	m	Z position of maximum speed
*avgTke**	Output	J/kg	Average turbulence kinetic energy
*minTke**	Output	J/kg	Minimum turbulence kinetic energy
*maxTke**	Output	J/kg	Maximum turbulence kinetic energy
*minTkeX**	Output	m	X position of minimum TKE
*minTkeY**	Output	m	Y position of minimum TKE
*minTkeZ**	Output	m	Z position of minimum TKE
*maxTkeX**	Output	m	X position of maximum TKE
*maxTkeY**	Output	m	Y position of maximum TKE
*maxTkeZ**	Output	m	Z position of maximum TKE

Technical implementation

The flow field model component is written in C++ and packaged as a Functional Mock-up Unit (FMU). The model utilizes the NetCDF 4 library[2] which provides functionality for reading and writing NetCDF files.

Model initialization

At startup, the FMU opens the NetCDF file and reads the metadata defining the grid or the vertical profile. The NetCDF format allows for metadata describing units and the geometry of the data contained. Provided the input NetCDF file is well formatted, no additional information is needed for the model to relate the flow field to the tank geometry. The layout of a NetCDF file can be shown using the CDL (network Common data form Description Language) format, in which file dimensions and variables with attributes are listed. The following listing shows the CDL of a file containing a three dimensional current flow field (given as components *u*, *v* and *w*), with metadata providing grid and variable units:

```
netcdf datafile {
dimensions:
        xc = 400 ;
        yc = 400 ;
        zc = 200 ;
variables:
```

[2] NetCDF libraries are open source and can be downloaded at https://www.unidata.ucar.edu/downloads/netcdf/index.jsp.

```
                                    float xc(xc) ;

                                    xc:units = "m" ;
                                    float yc(yc) ;

                                    yc:units = "m" ;
                                    float zc(zc) ;

                                    xc:units = "m" ;
    float u(zc, yc, xc) ;

                                    u:units = "m/s"
    float v(zc, yc, xc) ;

                                    v:units = "m/s"
    float w(zc, yc, xc) ;

                                    w:units = "m/s"
}
```

In the case of net cages, the vertical profile is read upon startup, and stored in memory. In the case of tanks, no further data is read upon initialization of the model. All output values, except the current vector for a requested position, are calculated upon initialization of the FMU and stored for output in all subsequent time steps.

Calculation of current vector
When the Functional Mock-up Interface (FMI) master calls for the model to advance to the next time step, new *xPos*, *yPos* and *zPos* values are provided. In the case of net cages, the *zPos* value is used to extract the appropriate current vector from the vertical profile, and the vector components *xVel*, *yVel* and *zVel* are returned. In the case of tanks, the appropriate data cell is found based on the given coordinates, and the current velocity of the cell is read from the NetCDF file. If the file provides vector components, these are returned directly. If the file provides axial, radial and vertical velocities, vector components are calculated from these and returned.

Calculation of current descriptor values
To calculate current descriptor variables, the entire flow field is read from the NetCDF file, and the appropriate calculations are performed, such as finding the minimum, mean and maximum speeds. All of this is done at model initialization, and no further calculations are done when the model advances in time steps.

Integration into Virtual Laboratory

When setting up an experiment in the final version of the VL, the user will make selections that determine the setup of the flow field model:

- In the case of tanks, the user will choose which supported tank design to use
- In the case of sea cages, the user will choose the type of location (fjord or exposed), high or low current scenario and to activate/deactivate tidal variations

Based on the experiment setup, the flow field model will operate according to the descriptions in this report, and provide the defined set of output values. If another model module provides the *xPos*, *yPos* and *zPos* input values, it may read back the outputs *xVel*, *yVel* and *zVel*.

The flow model may be linked to the growth (AquaFishDEB) model by providing velocity information that can affect the apparent feed availability for the fish (representing increasing difficulty of feeding), and the maintenance cost of the fish (representing increased swimming activity). It may also potentially be linked to the water quality model, providing information on mixing rates and retention times for particular and dissolved matter in the rearing units.

As part of the model output summary, the VL will display and visualize key outputs from the flow field model.

3. CONCLUSIONS

The final version of the flow field model has been developed and integrated into the Virtual Laboratory (Bjørnson et al., 2019). The key functionality of the flow field model component is to provide information about the water current in the tank or cage. The model provides two sets of outputs – one that contains 3D current components (e.g. current upwards, towards the east and towards the north) at a requested position, and another that provides overall descriptive values characterizing the flow field.

For flow fields in tanks, the model uses precomputed flow fields from a Computational Fluid Dynamic (CFD) model. Tank designs at VURH, HCMR and NOFIMA will be available to the user for selection, and additional precomputed flow fields may be added in the future without change to the flow field model code. For open sea net cages, the model estimates the flow field based on outside current conditions in the form of vertical profiles of current direction and speed. The user can choose high or low current speed conditions for fjord or exposed locations, and tidal variability can be applied to the current profiles.

The flow model has been integrated into the Virtual Laboratory. Information from the flow model is not currently used by other model modules, but there are potential links that could be introduced with both the growth and water quality models.

REFERENCES

Alver, M. O. et al. (2018). D5.4 First prototype flow field model established, AQUAEXCEL2020 Report.

Bjørnson, F. O. et al. (2016). D5.1 Model development guidelines, AQUAEXCEL2020 Report.

Bjørnson, F. O. et al. (2019). D5.5 Virtual laboratory version 1. AQUAEXCEL2020 Report.

Endresen, P. C., Føre, M., Fredheim, A., Kristiansen, D., & Enerhaug, B. (2013). Numerical modeling of wake effect on aquaculture nets. In ASME 2013 32nd International Conference on Ocean, Offshore and Arctic Engineering (pp. V003T05A027-V003T05A027). American Society of Mechanical Engineers.

Fluent: http://www.ansys.com/Products/Fluids/ANSYS-Fluent

Gorle, J.M.R., Terjesen, B.F., Mota, V.C., Summerfelt, S. (2018). Water velocity in commercial RAS culture tanks for Atlantic salmon smolt production. Aquacultural Engineering 81, 89-100. doi: 10.1016/j.aquaeng.2018.03.001.

J. Hanak: CFD simulation of flow in fish tanks, MSc. Thesis, CTU Prague, 2016.

Lika, K., Pavlidis, M., Mitrizakis, N., Samaras, A. and Papandroulakis, N., 2015. Do experimental units of different scale affect the biological performance of European sea bass Dicentrarchus labrax larvae? Journal of fish biology, 86(4), pp.1271-1285.

Løland, G. (1993). Current forces on, and water flow through and around, floating fish farms. Aquaculture International, 1(1), 72-89.

Reite, K.J., Føre, M., Aarsæther, K.G., Jensen, J., Rundtop, P., Kyllingstad, L.T., Endresen, P.C., Kristiansen, D., Johansen, V. and Fredheim, A, (2014). Fhsim—time domain simulation of marine systems. In ASME 2014 33rd International Conference on Ocean, Offshore and Arctic Engineering. American Society of Mechanical Engineers Digital Collection.

Sayma, A. (2009). Computational fluid dynamics. Bookboon

Shchepetkin, A. F., & McWilliams, J. C. (2005). The regional oceanic modeling system (ROMS): a split-explicit, free-surface, topography-following-coordinate oceanic model. Ocean modelling, 9(4), 347-404.

Slagstad, D., & McClimans, T. A. (2005). Modeling the ecosystem dynamics of the Barents Sea including the marginal ice zone: I. Physical and chemical oceanography. Journal of Marine Systems, 58(1-2), 1-18.

Glossary

AQUAEXCEL^{2020}: AQUAculture Infrastructures for EXCELlence in European Fish Research towards 2020

CFD: Computational Fluid Dynamics

FMI: Functional Mock-up Interface. A standard to support model exchange and co-simulation of dynamic models using a combination of xml-files and compiled C-code (http://fmi-standard.org/).

FMU: Functional Mock-up Unit. A model packaged to support the FMI standard. The flow field model described in this report is packaged as an FMU.

NetCDF: A set of software libraries and self-describing, platform independent data formats for array-oriented data.

Definitions

Document information

EU Project N°	652831	Acronym	AQUAEXCEL2020
Full Title	AQUAculture Infrastructures for EXCELlence in European Fish Research towards 2020		
Project website	www.aquaexcel.eu		

Deliverable	N°	D5.8	Title	Final flow field model after testing period
Work Package	N°	5	Title	JRA1 – Virtual laboratories and modelling tools for designing experiments in aquaculture research facilities

Date of delivery	Contractual	15/01/2020 (Month 51)	Actual	15/01/2020 (Month 52)
Dissemination level	PU	PU Public, fully open, e.g. web		
		CO Confidential, restricted under conditions set out in Model Grant Agreement		
		CI Classified, information as referred to in Commission Decision 2001/844/EC.		

Authors (Partner)	NTNU			
Responsible Author	Name	Morten Omholt Alver	Email	morten.alver@ntnu.no

Version log			
Issue Date	Revision N°	Author	Change
20/12/2019	1	Morten Omholt Alver	First version for internal review
27/12.2019	2		Feedback from WP leader
08/01/2020	3		Draft for 2nd reviewer
15/01/2020	4		Final version

Annex 1: Check list

Deliverable Check list (to be checked by the "Deliverable leader")

	Check list		Comments
BEFORE	I have checked the due date and have planned completion in due time	X	*Please inform Management Team of any foreseen delays*
	The title corresponds to the title in the DOW	X	*If not please inform the Management Team with justification*
	The dissemination level corresponds to that indicated in the DOW	X	
	The contributors (authors) correspond to those indicated in the DOW	X	
	The Table of Contents has been validated with the Activity Leader	X	*Please validate the Table of Content with your Activity Leader before drafting the deliverable*
	I am using the AQUAEXCEL^{2020} deliverable template (title page, styles etc)	X	*Available in "Useful Documents" on the collaborative workspace*
The draft is ready			
AFTER	I have written a good summary at the beginning of the Deliverable	X	*A 1-2 pages maximum summary is mandatory (not formal but really informative on the content of the Deliverable)*
	The deliverable has been reviewed by all contributors (authors)	X	*Make sure all contributors have reviewed and approved the final version of the deliverable. You should leave sufficient time for this validation.*
	I have done a spell check and had the English verified	X	
	I have sent the final version to the WP Leader, to the 2nd Reviewer and to the Project coordinator (cc to the project manager) for approval	X	*Send the final draft to your WPLeader, the 2nd Reviewer and the coordinator with cc to the project manager on the 1st day of the due month and leave 2 weeks for feedback. Inform the reviewers of the changes (if any) you have made to address their comments. Once validated by the 2 reviewers and the coordinator, send the final version to the Project Manager who will then submit it to the EC.*