# 3D NUMERICAL SIMULATION OF SUPERCRITICAL CHANNEL JUNCTION FLOW

**Robert Feurich[1] & Nils Reidar B. Olsen[1]**

[1]Department of Hydraulic and Environmental Engineering, Norwegian University of Science and Technology, Norway, 7491 Trondheim

E-mail: robert.feurich@ntnu.no

## Abstract

Supercritical flows often appear in spillway channels, chutes and other man-made structures. One way for assessing the effects of supercritical flows on water levels is by conducting physical model tests, but those results are often influenced by scale effects, especially in cases where air entrainment is involved. Another possibility, which became more and more important and reliable over the last years, is using numerical models to obtain data from flow fields. Two commercial, three dimensional codes, both using the Volume of Fluid method for tracking the free surface, have been tested, simulating the supercritical flow in a channel junction. The main purpose was to see how they can be applied to predict extreme wave values. The calculated free surface elevations have been compared with measurement data from literature. In the chosen test case two rectangular channels are joining together. The combining angle is 22.5° and the total length is app 0.6 m with a channel width of 0.1 m. In each of the two inflow branches the Froude number is app. 4.5 which results in a flow rate of 6.2 l/s. The results showed clearly that numerical simulations can provide reliable data for supercritical flow situations in channel junctions. Using numerical tools can help finding optimized technical solutions fast for channel construction, either by demanding additional freeboard or other measures to reduce cross-waves.

## Introduction

Knowing the increased free surface elevation due to supercritical flows is of immense importance especially in case of changing cross section geometries, friction, slope or small obstacles within the flow area. The resulting standing water waves are also called shock-waves and can cause damage to hydraulic structures and the surrounding environment unless they are considered during construction phase.

Physical model tests are the common way of investigating different flow and geometry setups. Empirical formulas for e.g. deflection angles or height of the shock front can also be found (Ippen and Knapp 1936, Ippen and Dawson 1951, Rouse et al. 1951, Bowers 1950, Hager 1989, Mazumder and Hager 1993).

Computational fluid dynamics (CFD) has almost completely replaced experimental investigations in areas like mechanical or aerospace engineering. The techniques have also become popular and reliable in hydraulic engineering over the last years. However, the use of numerical models in this field has much been restricted to one (1D) or two dimensional (2D) computations. CFD has also been used to calculate supercritical flows, mainly using 2D depth-averaged approaches (e. g. Jimenez and Chaudhry 1988, Soulis 1991, Krüger and Rutschmann 2006, Mignot et al. 2008, Ying et al. 2009). Despite the fact that the results of the 2D simulations were quite reasonable, the flow situation of supercritical flows with oblique shockwaves is in fact a complex three dimensional (3D) problem.

Stamou (2008) applied the commercial 3D numerical model Flow-3D from Flow Science to successfully simulate the expansion channel by Mazumder and Hager (1993) using the Volume of Fluid (VOF) approach to track the free surface. The algorithms to find the position of the free water surface are a particular challenge. First tests using a 3D approach were performed using a simplified potential flow approach as described in Chan et al. (1973 a, b). First results using the Navier-Stokes equations were obtained using the marker and cell (MAC) method (Harlow and Welch 1965). Later this method was improved using on a fixed mesh either the VOF method (Hirt and Nichols 1981) or the level set method described in Sethian (1996) which are somewhat similar. Mnasri et al. (2010) successfully used the VOF method in 2D for simulating the free surface behaviour when horizontal cylinders are exiting and entering. An interesting approach is presented in Krüger et al. (1998) and Krüger (2000) who derived the extended shallow water equations proving 3D flow features on a 2D computational mesh. One of the advantages of numerical modelling is the time and money saving aspect compared to physical model studies. Chandler et al. (2003) and Gessler and Rasmussen (2005) reported that the costs are reduced to 20 - 25% and the time to 25 - 33%.

## Test Case Setup

A small channel junction experimentally investigated for different flow situations by Hager (1989) has been used for the numerical modelling (Fig. 1). That case has also been investigated by Krüger and Rutschmann (2006) using an extended shallow water approach. In the chosen test case the upstream Froude number ($F_0$) is 4.5 in both branches connected by an angle ($\delta$) of 22.5°. The total length is 0.6 m with a channel width of 0.1 m. The inflow flow depth is 0.0268 m and the flow rate is app. 6.2 l/s in each branch.

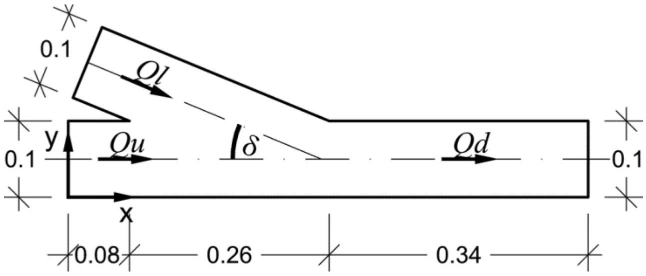


Figure 1: Channel junction geometry [m]

## Numerical Setup

The supercritical flow configurations in the current study are characterized by a strong three-dimensional structure and a free water surface including standing waves. Therefore the 3D simulation tools STAR-CCM+ and Flow-3D were chosen because they can handle complex three dimensional flow situations and by using the Volume of Fluid (VOF) method every kind of free water surface can be reproduced.

The principle of the VOF method is to use additional scalar information in each cell to track the ratio of water and air within the whole domain. The exact position where the free surface cuts through a cell ($0 < f < 1$) is determined using the scalar ($f$) information in the cell itself and its neighbouring cells. Similarly, the normal vector on the plan can be determined. An additional transport equation has to be solved for $f$.

$$\frac{\partial f}{\partial t} + U_i \frac{\partial f}{\partial x_i} = 0 \tag{1}$$

**STAR-CCM+**

STAR-CCM+ is a multipurpose computational fluid dynamics (CFD) software produced by CD-adapco. STAR-CCM+'s numerical solver is based on the finite volume method (FVM) and handles structured and unstructured grids. Different mesh types are available in the software. When investigating the free surface flow, the hexahedral (trimmed) mesh was found optimum because the grid lines are more aligned with the flow direction, causing less false diffusion. That produces a smoother water surface compared to tests with polyhedral and tetrahedral meshes.

A trimmed mesh (1.3 million cells) (Figure 2) with Dirichlet boundary condition (inflow velocity) was used and a time step of 0.001 s was necessary to obtain reliable results.

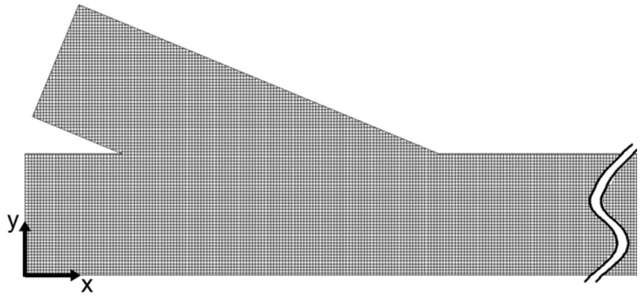


Figure 2: Plane view of the STAR-CCM+ grid

The domain was filled with water up to the inflow boundary water depth (h = 0.0268 m) as initial condition. The flow volume with the channel bed as bottom wall boundary was imported to STAR-CCM+ using a stereolithography (STL) file, where the solid object surface is represented by triangles. The Reynolds-Averaged Navier Stokes (RANS) equations were solved using the standard k-ε model in combination with the SIMPLE method and wall laws at bed and side boundaries. An isosurface has to be defined to visualize the free surface in STAR-CCM+ using the VOF method. The VOF value for that isosurface was set to 0.5 in the current study, which means that the free surface is placed at cells that are 50% filled with water and air.

**Flow-3D**

Flow-3D is a 3-D commercial numerical code produced by Flow Science Inc. With this tool it is possible to achieve the flow field three dimensionally and the water surface elevation very accurate. It solves the RANS equations using a finite difference (control volume) method. Flow-3D uses a very simple structural mesh consisting of a hexahedral block subdivided into variable-sized hexahedrons. In order to account for complex geometries, a multi-block mesh consisting of several adjacent blocks can be built. Into this simple mesh the geometry is inserted as obstacles, which can either be:

- Simple elements termed "Flow-3D primitives"
- XYZ-data
- 3D CAD file formats (e.g. IDEAS, STL)

The Fractional Area Volume Obstacle Representation (FAVOR) approach (Hirt 1992) is used to represent complex geometries in Flow-3D which principle is similar to the VOF method but representing the terrain and obstacle/water ratio instead of the fluid/water ratio. When using the FAVOR method the mesh resolution has to be chosen properly in order to rebuild the geometry in the numeric model sufficiently. Figure 3 shows exemplary how the same geometry is represented differently when using a coarse (a) or a fine (b) grid.

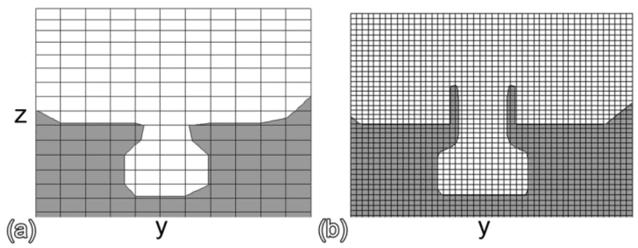


Figure 3: FAVOR geometry representation, (a) coarse, (b) fine grid

In the present study 1.3 million cells have been used for the simulations in order to have similar conditions as in the simulations done with STAR-CCM+. Figure 4 shows the grid and the two blocks, one (block 2) for the main channel and the other (block 1) for the incoming side channel. Due to the rectangular blocks of Flow-3D the inflow geometry has to be slightly different compared to the STAR-CCM+ setup. But that is not affecting the results because the inflow velocity can be rotated to the correct direction in Flow-3D too.

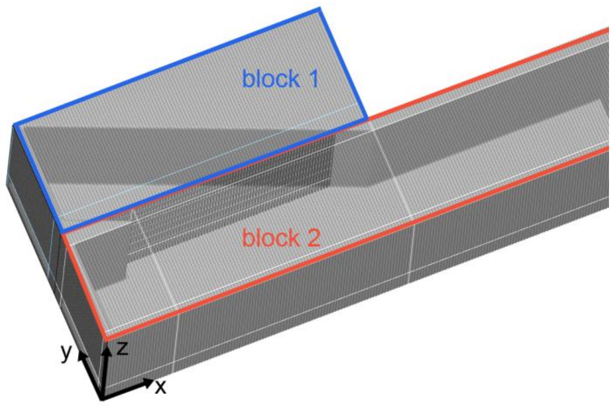


Figure 4: 3D view of the Flow-3D grid

It can be seen in Figure 5, where the FAVOR representation of the channel geometry is shown, that the amount of cells, used in this work, is large enough to characterize the channel geometry correctly.

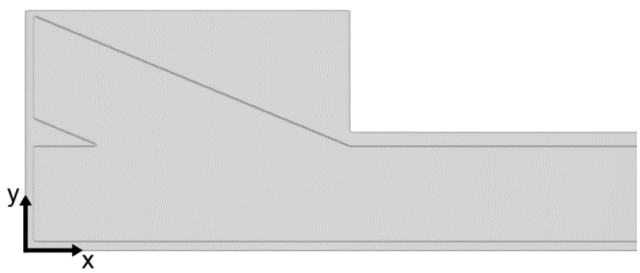


Figure 5: Channel junction FAVOR representation

Flow-3D has an air entrainment model included, which has been used for the present study to see if air entrainment is affecting the water levels in the channel junction. There are two possibilities how the software is handling the air entrainment feature. One way, for small amounts of entrained air, is to look just at the total amount of entrained air within the flow volume without affecting the volume or the surface elevation itself. The second way, which has been used in the present work, is to consider the amount of entrained air in the increase of the total fluid/mixture volume fraction. The principle idea of the air entrainment model in Flow-3D is that turbulent eddies are able to trap air at the surface of the flow and carry them back into the water body. That can happen when the turbulence intensity is high enough to overcome the surface stabilizing forces of gravity and surface tension. Hirt (2003) describes the way air entrainment is handled in Flow-3D in more detail.

## Results and Discussion

### STAR-CCM+

When looking at the results of the STAR-CCM+ simulation first, it can be seen that the shape, angle and location of the wave front within the channel junction show a very good agreement between measurement and simulation (Figure 6).

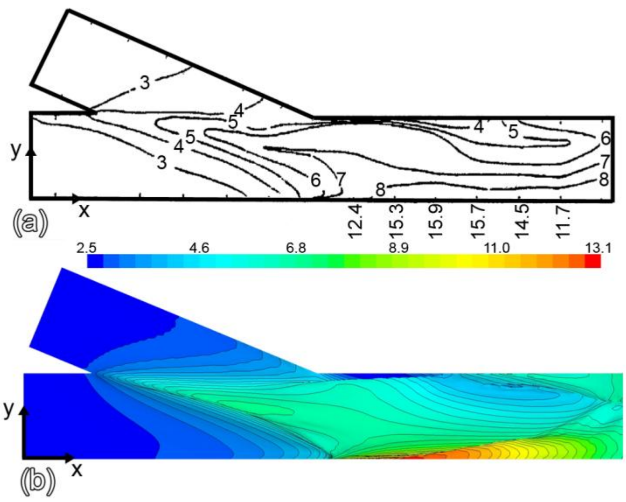


Figure 6: Free surface elevation [cm]; (a) physical model (Hager 1989), (b) simulation

The overall pattern of the two results fit quite well, although the maximum water height at the right side of the channel, downstream the junction, is too low compared to the measurements. The location of the maximum water level on the right side matches.

However, the height is in the same range as the results of Krüger and Rutschmann (2006) using an extended shallow water approach (Figure 7). The three dimensional code, used in the current study, improved the results compared to Krüger and Rutschmann (2006) on the left side of the channel right behind the side channel inflow. There it fits well in the present study. When looking at the water surface in the middle of the channel, just upstream of the channel outflow, it can be seen that the elevation and the shape are matching well.

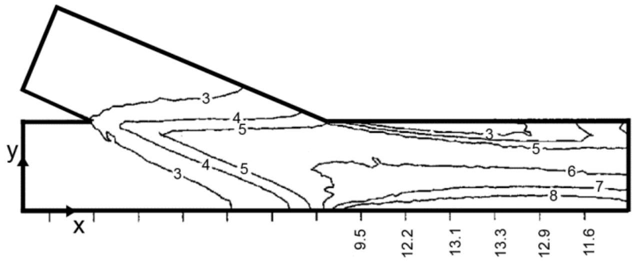


Figure 7: Free surface elevation [cm]; simulation results from Krüger and Rutschmann (2006)

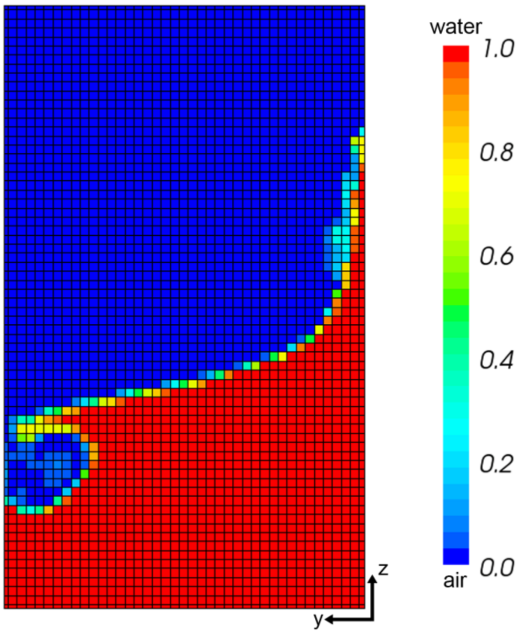


Figure 8: Cross section showing the volume fraction of fluid, looking downstream

Figure 8 shows the cross section where the maximum surface elevation on the right side occurs. It can be seen that the boundary between air and water is in the range of one cell size and no false diffusion can be observed.

When looking at the 3D-shape of the free surface in the junction area (Figure 9) it can also clearly be seen that the numerical simulation is representing the flow situation reasonably well.

In a next step it should be clarified if air entrainment, caused by the supercritical flow conditions within the channel junction, is the reason for the difference in the maximum water levels close to the wall on the right side. Additional simulations of the same case using Flow-3D, which has an inbuilt air entrainment model, have been conducted.

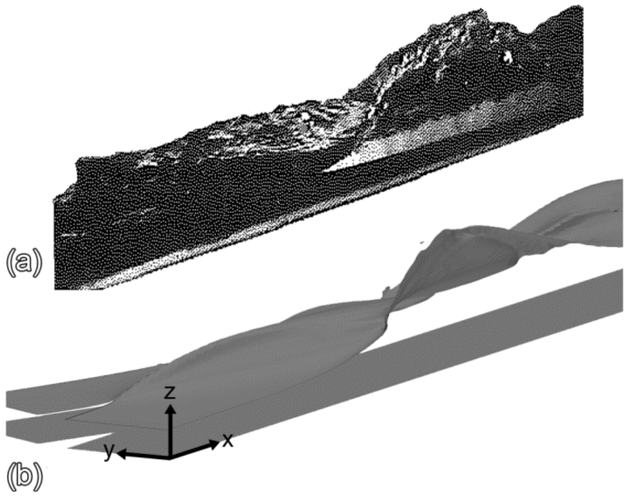


Figure 9: free surface elevation [cm]; (a) measurement (Hager 1989), (b) simulation

**Flow-3D**

Figure 10 shows the Flow-3D results of the channel junction free surface elevation. It is obvious that the free surface looks the same as in Figure 6, where the STAR-CCM+ results are presented. Only the area at the wall on the left channel side, behind the side-channel inflow, is slightly, but negligible, different. That can be explained by small differences in the codes like the generated grid.

It is also obvious that air entrainment, which has been activated in the Flow-3D model, is not the reason for the different maximum elevations in the physical and numerical model.

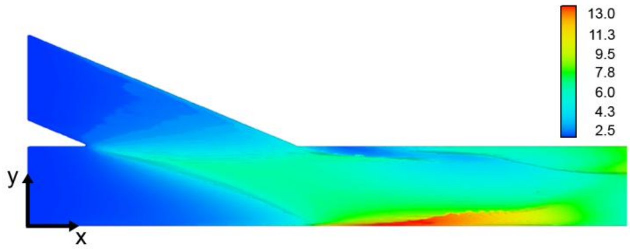


Figure 10: Free surface elevation [cm]

In Figure 11 the volume fraction of entrained air is presented. It can be seen that it is nearly zero within the whole channel.

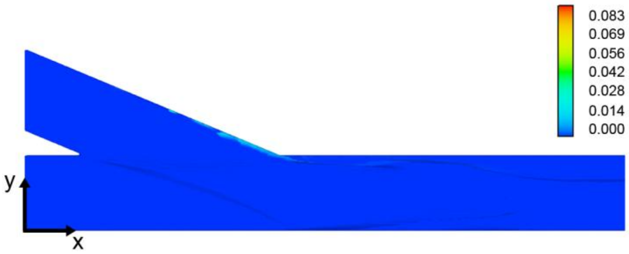


Figure 11: Volume fraction of entrained air

## Conclusions

Three dimensional numerical simulations of a supercritical channel junction flow have been conducted using two commercial codes, both applying the Volume of Fluid method for representing the free surface. The results have been compared with literature data of measurements from a physical channel junction.

All results show that the numerical models (STAR-CCM+, Flow-3D) are capable of reproducing the free surface of such flows reasonably well. Only the maximum elevation on the right side of the channel could not be reproduced well. Simulations with Flow-3D and activated air entrainment model showed that air entrainment is not the reason for that difference.

### References


Bowers, C. E. (1950). *Studies of open channel junctions*. Technical Paper Nr. 6, Series B, (Part V of Hydraulic model studies for Whiting Field Naval Air Station, Milton, Florida, University of Minnesota, St. Anthony Falls Hydraulic Laboratory) Technical Paper No. 6, Series B.

Chan, S. T. K., & Larock, B. E. (1973a). Fluid Flows from Axisymmetric Orifices and Valves. *Journal of the Hydraulics Division*, 99(1), pp. 81-97.

CD-Adapco (2011). *User Guide*. STAR-CCM+ Version 6.04.014.

Chan, S. T. K., Larock, B. E., & Herrmann, L. R. (1973b). Free-Surface Ideal Fluid Flows by Finite Elements. *Journal of the Hydraulics Division*, 99(6), pp. 959-974.

Chandler, K., Gill, D., Maher, B., Macnish, S., & Roads, G. (2003). Coping with probable maximum flood - an alliance project delivery for Wivenhoe Dam. *Proceedings of the 43rd ANCOLD Conference*, Hobart, Tasmania, pp. 332–349.

Gessler, D., & Rasmussen, B. (2005). Before the flood. *Desktop Engineering Magazine*, http://www.deskeng.com/articles/aaabhe.htm, accessed 17/03/2011.

Hager, W. H. (1989). Supercritical flow in channel junctions. *J. Hydr. Eng.*, 115(5), pp. 595-616.

Harlow, F. H., & Welch, J. E. (1965). Numerical Calculation of Time-Dependent Viscous Incompressible Flow of Fluid with Free Surface. *The Physics of Fluids*, 8(12), pp. 2182-2189.

Hirt, C. W., & Nichols, B. D. (1981). Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries. *J. Comput. Phys.*, 39, pp. 201-225.

Hirt, C. W. (1992). *Volume-fraction techniques: Powerful tools for flow modelling*. Flow Science Report No. FSI-92-00-02, Flow Science, Inc., Santa Fe, N. M.

Hirt, C. W. (2003). *Modeling Turbulent Entrainment of Air at a Free Surface*. Flow Science Report No. FSI-03-TN61, Flow Science, Inc., Santa Fe, N. M.

Ippen, A. T., & Knapp, R. T. (1936). A study of high velocity flow in curved channels. *Trans. American Geophysical Union*, Part II, 17, pp. 516-521.

Ippen, A. T., & Dawson, J. H. (1951). Design of channel contractions. *Trans.*, ASCE, 116, pp. :326-346.

Jimenez, O. F., & Chaudhry, M. H. (1988). Computation of supercritical free-surface flows. *Journal of Hydraulic Engineering*, 114, pp. 377-395.

Krüger, S., Bürgisser, M. F., & Rutschmann, P. (1998). Advances in Calculating supercritical Flows in Spillway Contractions, *Hydroinformatics´98*, ed. Babovic and Larsen, Balkema, Rotterdam, The Netherlands.

Krüger, S, (2000). *Computational Contributions to Highly Supercritical Flows*. doctoral thesis ETH No.13838, Swiss Federal Institute of Technology, Zurich, Switzerland.

Krüger, S., & Rutschmann, P. (2006). Modeling 3D supercritical flow with extended shallow-water approach. *J. Hydr. Eng.*, 132(9), pp. 916-926.

Mazumder, S. K., & Hager, W. H. (1993). Supercritical expansion flow in Rouse modified and reversed transitions. *Journal of Hydraulic Engineering*, 119(2), pp. 201–219.

Mignot, E., Paquier, A., & Rivière, N. (2008). Experimental and numerical modelling of symmetrical four-branch supercritical cross junction flow. *Journal of Hydraulic Research*, 46(6), pp. 723-738.

Mnasri, C., Hafsia, Z., Omri, M., & Maalel, K. (2010). A Moving Grid Model for Simulation of Free Surface Behavior Induced by Horizontal Cylinders Exit and Entry. *Engineering Applications of Computational Fluid Mechanics*, 4(2), pp. 260-275.

Rouse, H., Bhoota, B. V., & En-Yun Hsu (1951). Design of channel expansions. *J. Trans.*, ASCE, 116, pp. 347-363.

Soulis, J. V. (1991). A numerical method for subcritical and supercritical open channel flow calculation. *International Journal for Numerical Methods in Fluids*, 13(4), pp. 437-464.

Sethian, J. A. (1996). Theory, algorithms, and applications of level set methods for propagating interfaces. *Acta Numerica*, 5, pp. 309-395.

Stamou, I., Chapsas, D. G., & Christodoulou, G. C. (2008). 3-D numerical modeling of supercritical flow in gradual expansions. *Journal of Hydraulic Research*, 46(3), pp.402-409.

Ying, X., Jorgeson, J., & Wang, S.Y. (2009). Modeling Dam-Break Flows using Finite Volume Method on Unstructured Grid. *Engineering Applications of Computational Fluid Mechanics*, 3(2), pp.184-194.