Influence of Pipe-Junctions on Downstream Measuring Sections, predicted by a numerical model

Influence des jonctions de réseau sur les sections de mesure avales (traitement numérique)

R. Mohn[1*], N. Vosswinkel[1], A. Schnieders[1], M. Henrichs[1], M. Uhl[1]

[1]Muenster University of Applied Sciences, Dept. of Civil Engineering, Laboratory of Water Resources Management, Corrensstr. 25, 48149 Muenster, Germany
*Corresponding author, e-mail mohn@fh-muenster.de

RÉSUMÉ

La mesure de débit en réseaux d'assainissement permet d'optimiser les opérations de planification dans la gestion urbaine des eaux usées, des stations d'épuration et des rivières. La pertinence d'une campagne de mesure de débit est étroitement liée à la localisation du point de mesure en relation avec les singularités du réseau.

Ce papier s'étend particulièrement sur la confluence de deux canalisations circulaires idéales pour deux configurations données (l'une avec une arrivée de 45° par rapport à la canalisation principale, l'autre 90°), des incertitudes de calcul de débit liées à la dissymétrie du champ de vitesse en aval de la jonction. L'intérêt s'est tout particulièrement focalisé sur la problématique de l'influence de la distance entre la confluence et le point de mesure sur les modèles de calcul de débit utilisés (le débit est déduit des mesures de vitesse et de hauteur).

Cette investigation est traitée essentiellement numériquement (simulations 3D). Ces simulations ont été réalisées par le code de calcul 3D FLUENT, logiciel commercial bien souvent utilisé en mécanique des fluides.

Les résultats de cette étude montrent que les recommandations usuelles entre la distance d'une confluence latérale et d'une section de mesure (avec les conditions limites appropriées) conduisent à d'importantes incertitudes quant au débit évalué. Sur l'ensemble des méthodes de mesures investiguées, la mesure par profil de vitesse était la plus robuste. Les autres méthodes étudiées utilisant une mesure unique de la vitesse (Doppler continu) peuvent présenter de bons résultats mais uniquement après étalonnage du modèle de mesure.

ABSTRACT

Measuring discharge data in sewer networks serves to optimize planning and operation processes of water management in the urban sewer system, treatment plants and rivers. The efficiency of a measuring campaign is strongly dependent on positioning of the measuring section in relation to non-uniformities in the system.

This paper concentrates on combining junctions of circular pipes in an idealized form of two configurations (Junction with lateral inflow under 45°- and 90°-angle), and the uncertainties in discharge calculation, resulting from asymmetry in the velocity distribution downstream of the inflow. The interest is focused on the question: how strong does the distance of the measuring sensor from the disturbance influence the calculation of the discharge out of the velocity distribution by means of a measuring model (procedure for deduction of discharge from velocities)?

The investigation is done completely numerically with 3D flow simulations. The simulations were performed with FLUENT, a commercial software-package for computational fluid dynamics (CFD).

The results of the study show that the usual recommendations in practice for the distance between measuring sections and lateral inflows, for the examined hydraulic boundary conditions, lead to significantly increased uncertainties of deducted discharges. From the applied methods the one with a measuring path (velocity profile) was the most robust. The other examined methods with single -point characteristic-measuring may also give good results, but only after expert calibration of the measuring model.

KEYWORDS

Open channel flow, measurement model uncertainty, pipe-junction, 3D-CFD, VOF

1 INTRODUCTION

Computational fluid dynamics (CFD) is an established design method for complex machine constructions. It also helps to visualize the three-dimensional flow patterns in pipes and sewers of civil engineering supply networks. In this study CFD is used as a modeling device for the three-dimensional velocity distribution and its spatial development. From the resulting velocity arrays data were taken like measurements, at locations corresponding to the specific characteristic of selected sensor types.

Measurement campaigns aim for the improvement of operational performance or economical benefit of water management systems. For the success of such campaigns, activities have to be concentrated on the highest possible reduction of (systematical) uncertainties. Pressure flow at pipe measuring sections is not always possible, due to local constraints or cost efficiency, and therefore the applicability of classical magnetic-inductive devices is limited. Alternative sensors are distributed, that are able to cope with free surface flow and which are more cost efficient. These sensors take velocities either punctually in a control volume or linearly along a defined measurement section (Kölling/Valentin [1995]). An important pre-condition for the application of such methods is the choice of an appropriate location in the sewer network and the optimal sensor position in the pipe gauging section (Erb [1998]). The evaluation of flow conditions and patterns in the pipes are jeopardized by disturbances like lateral inflows, bends, etc., which cause asymmetries in the velocity distributions.

Bonakdari et al. [2007] have previously used CFD in their study, to investigate the effect of a bend on the downstream velocity profile. Beneath other issues velocity profiles are shown in their study, that are located 10·B (B: width of the free water surface in the pipe) downstream of the bend. The distance of 10·B is also recommended by the sensor producer NIVUS (NIVUS [2005]) for the positioning of sensors behind bends with the angle 45°<α<90°. The velocity profiles show a distinct influence of the upstream bend. According to Bonakdari et al. [2007] an influence could be detected up to a distance of 20·B downstream. They defined the water level as a rigid roof and not as a free surface level as in the following study. The procedure that Bonakdari et al. used reduces the number of the cells in the grid, however a computationally intensive model for the multiphase simulation is not required. A disadvantage is that if e.g. a symmetry plane instead of a free surface level is used, the turbulent exchange of momentum is not impaired but the free turbulence at the free surface level is blocked. (Kölling [1994])

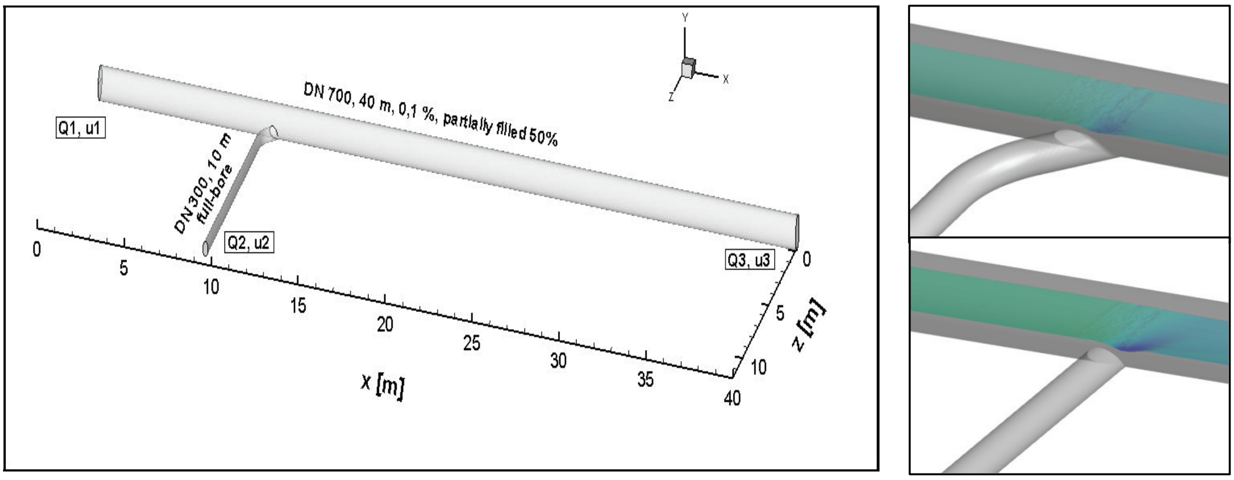

Figure 1: Geometry of lateral inflow

Main Pipe DN 700						Inflow Pipe DN 300			
Q_1	Q_3	u_1	u_3	$h_{surf,out}$	Fr_{out}	Q_2	u_2	h_{surf}	Re
[l/s]	[l/s]	[m/s]	[m/s]	[m]	[-]	[l/s]	[m/s]	[m]	[-]
114	145	0.59	0.75	0.35	0.46	31	0.43	full	99,300

Table 1: Variation of hydraulic boundary conditions

Comparable to the recommendations concerning bends there are also recommendations (practical experience) for the distance of measuring sections from pipe junctions with lateral inflows (DN: pipe diameter):

point measurement: 15·DN
profile measurement: 10·DN

Fig. 1 shows the geometry of the lateral inflow under 45°-angle and the 90°-angle. The main pipe reach is modeled in a length of 40 m and has a diameter of 700 mm. After 10 m the lateral inflow follows with a diameter of 300 mm, modeled in a reach of 10 m length. The inflow pipe bottom level is placed 1 cm above the main pipe bottom level. From the inflow boundary condition there are in both pipes 10 m flow length for the boundary layer to develop until the junction disturbs the flow patterns. The slope in both pipes was chosen to be 1.0 ‰. The wall roughness was assumed to be 1.5 mm. The origin of the coordinate system was placed in the middle of the inflow cross section of the main pipe. The x-axis describes the flow direction, the y-axis the height and the z-axis the width of the pipe.

In the scope of this study a numerical model was set up to investigate the influence of these inflows on the velocity measurement and discharge calculation. Aim of this study is to demonstrate the effect of varying distances to the disturbance on the determination of the discharge. This is done for the above mentioned two lateral inflows under selected hydraulic boundary conditions (see Table 1).

2 METHODS

In this paper the emphasis is laying on numerical simulations to evaluate different measurement models. Of course it is possible to derive the single velocity to feed the measurement models directly from measurements. However each method of velocity measurement has its own characteristic uncertainties or technical limitations, additional to the restrictions of the used test flume. Considering this, the authors decided to use a CFD-model in this study. The detail-models and numerical methods implemented in the used commercial software package (FLUENT) were verified by their developers.

The scope of hydraulic conditions in this study was restricted to subcritical equilibrium pipe flow where the water level develops parallel to the pipe bottom (without confluence at the junction). Therefore it was not necessary to measure the water level development to calibrate the wall roughness or parameterise the boundary layer modelling.

The velocity profiles resulting from the simulations, which of course may include uncertainties, were then used to test different measurement models (algorithms to calculate discharges from velocity profiles) and compare them with each other. The known discharge of the simulation served as reference. Experimental verification of the flow deflection by confluence is left to future studies, but the authors have no doubt, that the simulations are the better guess compared to existing standards.

2.1 CFD Model-Theory

In this study the FLUENT-software-package in the version 12.0 is used to solve the Reynolds-averaged Navier-Stokes-Equations numerically, which are based on the assumption of conservation of mass and momentum within a moving fluid (Ferziger/Peric [1997]).

The investigated flow conditions do obviously have high turbulence and the interesting discharge conditions do have usually high Reynolds numbers. The Reynolds-averaging makes it therefore necessary to include a turbulence model for the effect of small-scale turbulence on the averaged velocity fields (Lecheler [2009]). There is no turbulence model valid for every flow configuration. Each model has its characteristic advantages and disadvantages. In a first stage in this study the k-ε-model together with the standard wall functions (Launder/Spalding [1974]) was used. This two-equation-model has two separately solved transport equations for the kinetic energy k and the turbulent energy dissipation ε (Salaheldin et al. [2004]). The k-ε-model is most widely used due to its high convergence stability and comparably low computational resources demand, combined with good performance in the free flow (Lecheler [2009], Patel/Gill [2006]). Disadvantageous is the prediction of secondary flow patterns due to local anisotropy of turbulence (Wintterle [2008]), since the k-ε-model is based on the eddy viscosity concept that includes the assumption of isotropy of turbulence.

The interaction of water and air at the free surface in a partially filled pipe is considered by two-phase-modeling. The Volume-of-Fluid-(VOF-) method (Bardiaux et al. [2006]) is able to localize timely varying phase boundaries by means of additional phase interaction functions. The VOF-method is an interface-capturing method and is based on the assumption that two or more phases are not mixable and each grid cell (control volume) includes a specific fraction of all phases. This model setup is necessary because of the variation of water surface levels along the flow reaches. A simulation with uniformly defined and fixed water levels is not expected to give reliable results.

2.2 CFD Model-Set-Up

Two variants are considered to investigate the effect of a combining junction, one lateral inflow with 90°-angle and one with 45°-angle. All pipe profiles are circular. An inspection chamber, that might be located at the junction, was not considered or modeled. Table 1 contains a list of the hydraulic boundary conditions for both variants. The numerical grid, consisting of roughly 815.000 cells, was produced with the preprocessor GAMBIT. Step by step the grid was refined until a grid-independency was achieved. Around the lateral inflow the grid was formed with triangular pyramids (tetrahedrons). In the remaining area cubes (hexahedrons) were used to build the grid. Figure 2 shows, how the near-wall boundary zones and especially the expected water level are gridded with cells of high quality. The boundary conditions are based on the results of Schnieders [2009] and Vosswinkel [2010].

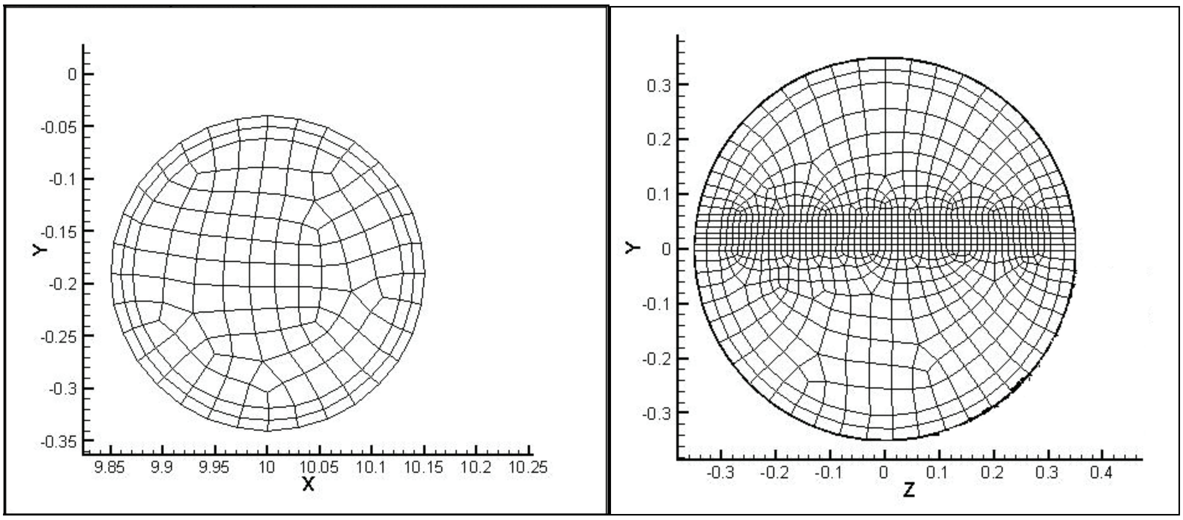

Figure 2: Numerical grid (left Inflow Pipe; right Main Pipe)

The boundary conditions were chosen as follows:

I. **Main Pipe Inflow**: Two separate inflow areas for air- and water-phase were defined. The inflow area type for the water-phase was defined as mass flow inlet with open channel boundary condition. The open channel boundary condition allows the definition of a free water surface. The inflow type of the air was defined as pressure outlet to allow a free development of the air flow.

II. **Lateral Pipe Inflow**: The lateral inflow was defined as massflow inlet with specified discharge as full-bore.

III. **Outlet**: The type chosen here is a pressure outlet, also with open channel boundary condition. Atmospheric pressure (gauge pressure = 0) was assumed here. The outlet boundary condition is defined by the free water surface level. Assumed here was uniform equilibrium flow calculated with the Prandtl-Colebrook-equation (Colebrook [1939]).

IV. **Pipe walls**: the walls were defined with a no-slip condition. Additionally the roughness was set to 1.5 mm.

2.3 Measurement models for discharge calculation

There is no direct method to measure discharge. It is only possible to measure features that are inherent to the specific measurement system. These have to be performed in mathematical relations, called the measuring model that allows calculating the velocities, the area of the flow section and finally the discharge (DWA-M 181 [2008]).

The velocity-area-methods do belong to the usual class of measuring models. They derive the discharge from the measured water surface level, resp. the flow section area and the mean velocity. This is done for one or more vertical measuring sections, each giving a velocity-area that is representative for the part of the control section (cross-sectional flow area) that is adjacent to the vertical measuring section under consideration. There may be chosen as many vertical measuring sections as necessary to represent the velocity profile in the whole control section.

$$Q = u_{mean} * A; \qquad u_{mean} \left[\frac{m}{s}\right] \text{ mean velocity}; \qquad A\,[m^2] \text{ cross sectional flow area}$$

This calculation is based on the one-dimensional continuity equation and is only valid for rectangular flow through the cross-sectional area A and in a steady flow field (Siedschlag [2004]). These limitations are usually postulated and then assumed to be insignificant. The mean velocity results from integration of the velocity field over the control section (cross-sectional flow area). If the pipe is filled completely (pressure flow) the cross-sectional area is given directly through the pipe geometry. In free surface flow the water level (h) has to be measured additionally and similar to the main process of velocity detection (DWA-M 181 [2008]).

The following methods are used in this paper to compute the mean velocity u_m, here applied on simulation results instead of measurements. These methods have one feature in common, the factor x_i that can be chosen (calibrated) in a specific range (see EN ISO 748 [2005]) or can be given by the producer of the sensor. In practice this factor can be calibrated with other measurement methods after the installation of the sensor.

The study deals with mathematical measuring models of the sensors. The systematic effects of the distance between disturbance and measurement section are analyzed. Further uncertainties during the measurement process are not considered.

Single-Point-Surface Method

The velocity is measured at one location close to the water surface. The mean flow velocity is calculated by multiplication with the factor $x_{surface}$, which is depending on cross-sectional area. The standard range according to EN ISO 748 [2005] is between 0.84 and 0.90, but practical experience in many cases gives values below that range, so in this study it was chosen to be $x_{surface}$ = 0.80.

$$u_{mean} = x_{surface} \cdot u_{surface}$$

Maximum-Velocity Method

In this method the maximum velocity (u_{max}) of a vertical measuring section is determined. The mean velocity u_{mean} is then calculated by multiplication with a factor x_{max} in a range from 0.80 to 0.90. In this study the value was chosen to be 0.89.

$$u_{mean} = x_{max} \cdot u_{max}$$

Nivus Method, NIVUS [2005]

The sensor NIVUS PCM Pro consists of an ultrasonic signal with digital pattern recognition. Velocities are determined from specific levels on the vertical measuring section that are varying with the water depth ($\frac{1}{n}\sum_{i=1}^{n} u_i$). From the averaged punctual velocity values the mean velocity is calculated with the following equation (Teufel/Solliec [2010]):

$$u_{mean} = x_{Nivus} \cdot \frac{1}{n}\sum_{i=1}^{n} u_i;$$ *u_i punctual velocity at a specific level [m/s]; n number of levels in the vertical measuring section under consideration*

Investigation method of this study

The data from the numerical simulation are used as "artificial measurement" for the above mentioned measurement models for discharge calculation. Here the flow development over increasing distance to the lateral inflow is shown over a dimensionless x-variable, defined as multiples of the pipe diameter (0·DN to 40·DN). In each of these (eight) positions a discharge depending on the respective section is calculated using these measurement models. The calculated discharges are compared with the reference discharge of the simulation in the main pipe (145 l/s). This part of the procedure is here labeled as “uncalibrated”, because the discharge is calculated with the chosen (recommended) factor x_i . So the robustness of each method for calculating the discharge without calibration of the factors x_i is pointed out.

The calibrated procedure goes beyond this. Here the power of numerical hydromechanics (CDF) is used, that there are no random errors or systematic aberrations of the electronic signal processing system in the resulting "measurement" data (see CEN ISO/TS 25377 [2007], Uhl [2000]). Apart from the uncertainties resulting from turbulence modeling in CFD the data can be named ideal, since the discharge is not only calculated from the simulation results but is also a necessary boundary condition of the flow configuration. For this reason it is possible to calibrate the factors x_i with the discharge boundary condition of the CFD-simulation. The calibration is of course done with a completely developed, symmetrical and undisturbed velocity profile – here at the end of the model reach at 40·DN.

3 RESULTS

Fig. 3 shows the location of the horizontal (lateral to the main flow direction) maximum velocity of the simulated profile over increasing distance from the lateral inflow. For the 90°-inflow there is a stronger deviation of the maximum from the center observed compared to the 45°-inflow. The lateral deviation is very strong especially between 10·DN and 20·DN. The maximum could be found up to 12 cm apart from the pipe axis. Fig. 4 shows the distribution of x-velocities with isolines in fictive measurement profiles for selected locations (multiples of DN). There is a distinct difference between the two observed cases of lateral inflow. While the flow of the 45°-inflow is almost symmetrical after 20·DN, for the 90°-inflow this is the case not before 35·DN resp. 40·DN. This is confirmed by the complete horizontal velocity profile that is including the maximum x-velocity-value (Fig. 5). Especially in the 90°-variant the velocity profiles are deformed strongly. The influence of the lateral deviation of the maximum velocity as well as the influence of deformation of the velocity profile on the calculation of the discharges is shown in Fig. 6 and in Fig. 7. The discharges were calculated with the measurement models of chapter 2. Shown are the relative discharge errors (in percent) compared to the reference discharge and over the increasing distance to the inflow location. First the relative errors are shown for not calibrated measurement model factors (x_i) (Fig. 6) and then for calibrated model factors (Fig. 7). The calibrated factors x_i are shown in Table 2.

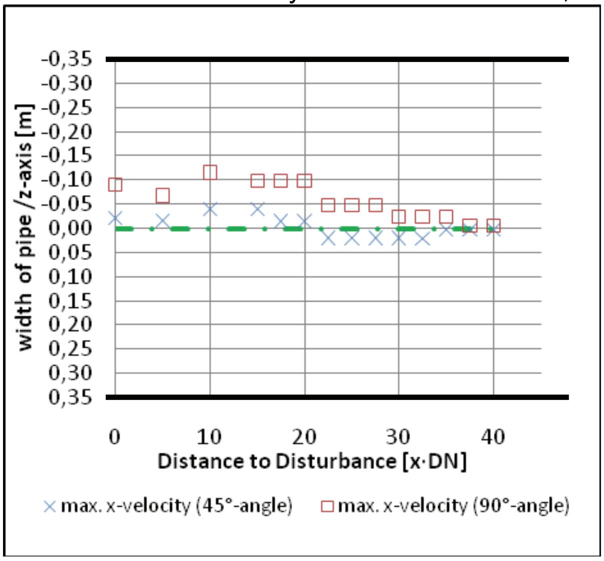

Figure 3: Location of maximum x-velocity in the width of pipe

The Nivus method shows for uncalibrated measurement models (Fig. 6) only slight differences in a magnitude of not more than +2% from the reference discharge, already after a distance of 20·DN from the inflow. However the uncertainty at the recommended range for a measurement section location (>10·DN) could be more than this (+2...>5%). The deviations of the two other methods do reach a constant value only after 30·DN. The discharge calculated by the Single-point-surface-method differs up to -3% and the maximum-velocity-method up to 8% from the reference value at 40·DN. Closer to the inflow location the differences at the recommended minimum distance from the disturbance (15·DN) are +2% (single-point-method) up to +13% (max-velocity-method) for 45°-inflow respectively -4% (single-point-method) up to +6% (max-velocity-method) for 90°-inflow.

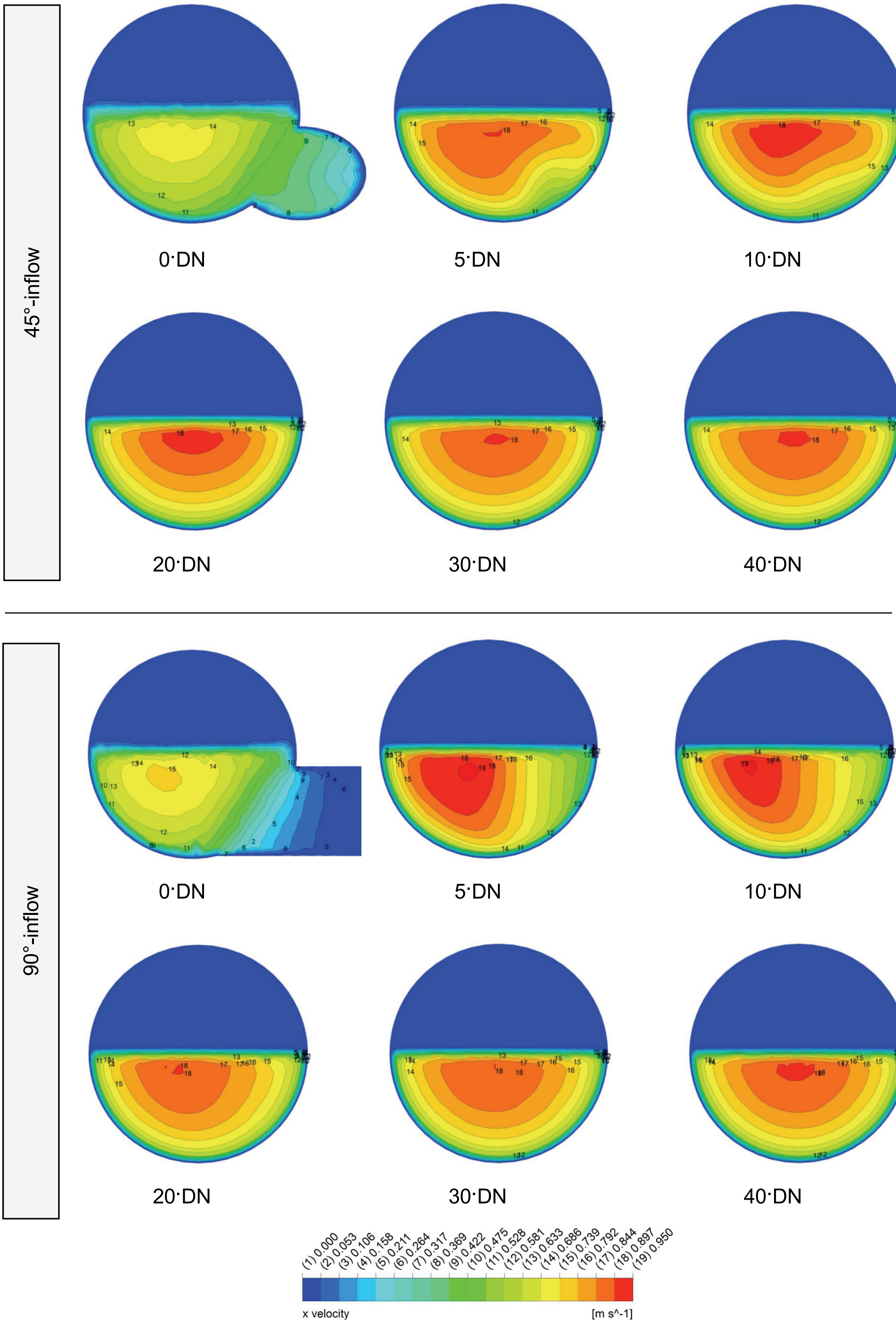

Figure 4: Isolines of x-velocity in increasing distance to the lateral inflow (upper 45°; lower 90°)

The qualitative development of the uncertainties along the flow direction is nearly similar for calibrated point measurement models (single-point and max-velocity). Fig. 7 shows nearly coinciding developments for the two models, so that only the initially assumed x_i -factors were reasons for the high absolute uncertainties in Fig.6. Both methods are comparably sensible on deformations of the velocity profile.

The pre-assumed x_i -factors do thus have a strong influence on the reliability of the measurement result and should be calibrated for non-uniform flow situations extremely carefully, preferably after application of an independent reference measurement with a well described method.

This statement is confirmed by the range between upper and lower limit in Fig. 7, calculated with the respective span of x_i -factors of the measurement models. These show – comparably to an uncertainty range – how neglecting of calibration or mistaken choice of the x_i -factor can influence the results. Conversely, the Nivus method performs much more robust. The x_i -value, implemented internally in the firmware of the sensor, gives good results for the discharge without calibration and therefore a time-consuming and expensive calibration is not necessary.

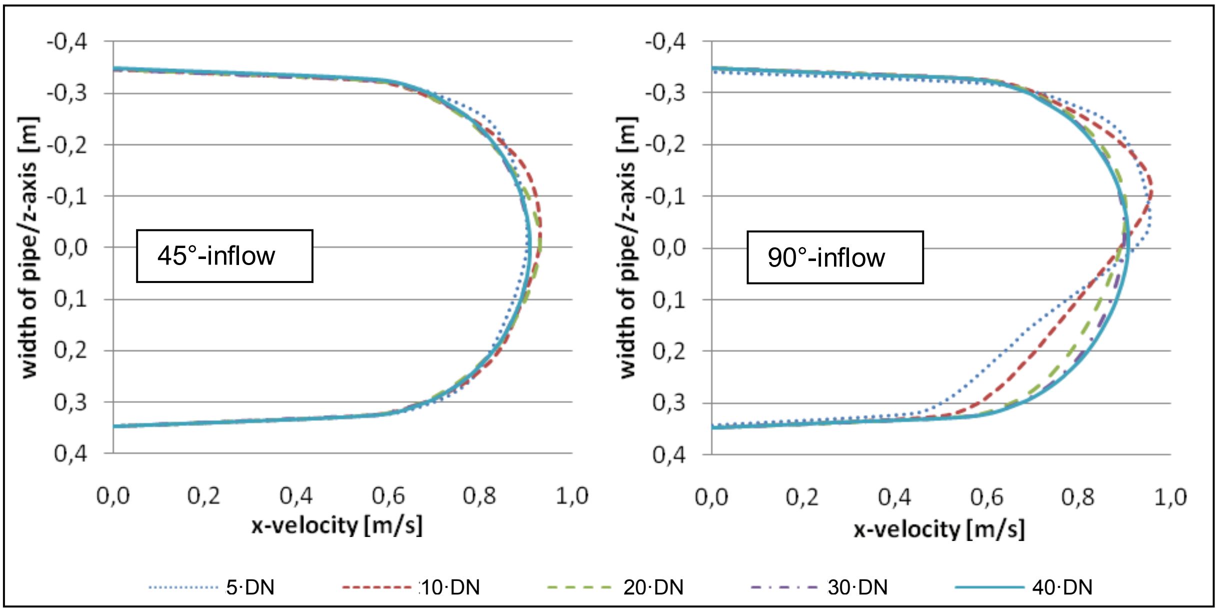

Figure 5: Horizontal velocity profiles including the maximum velocity of the entire cross section (left 45°; right 90°)

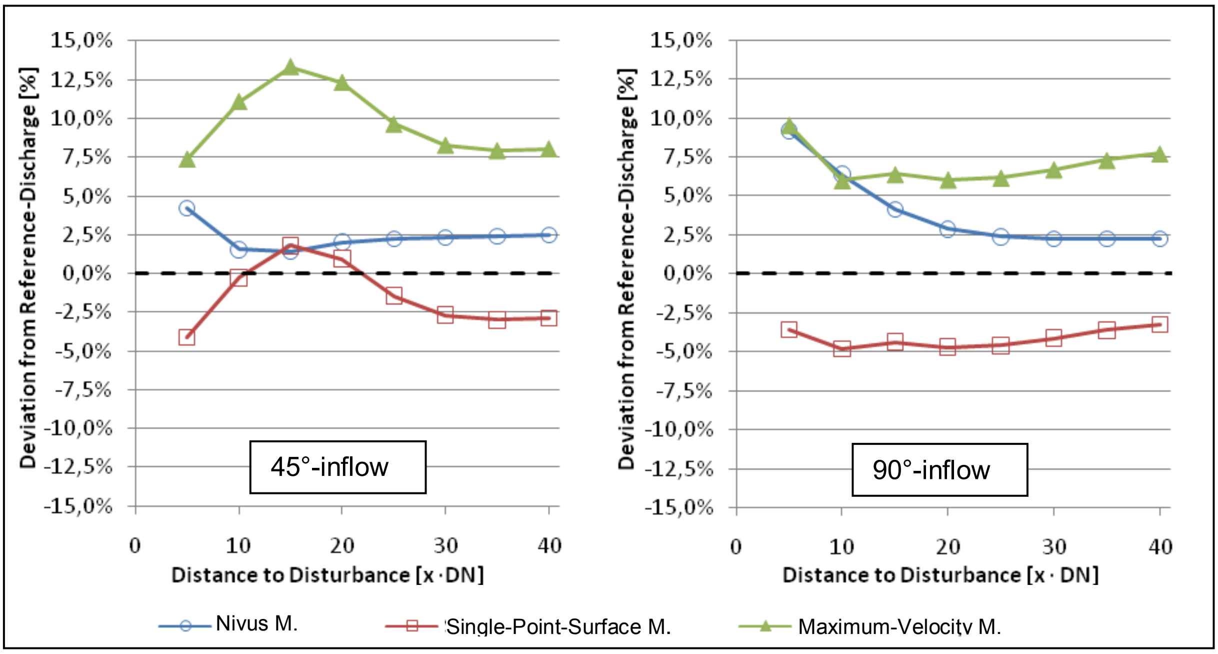

Figure 6: Uncalibrated deviation from reference-discharge over the distance from the inflow (left 45°; right 90°)

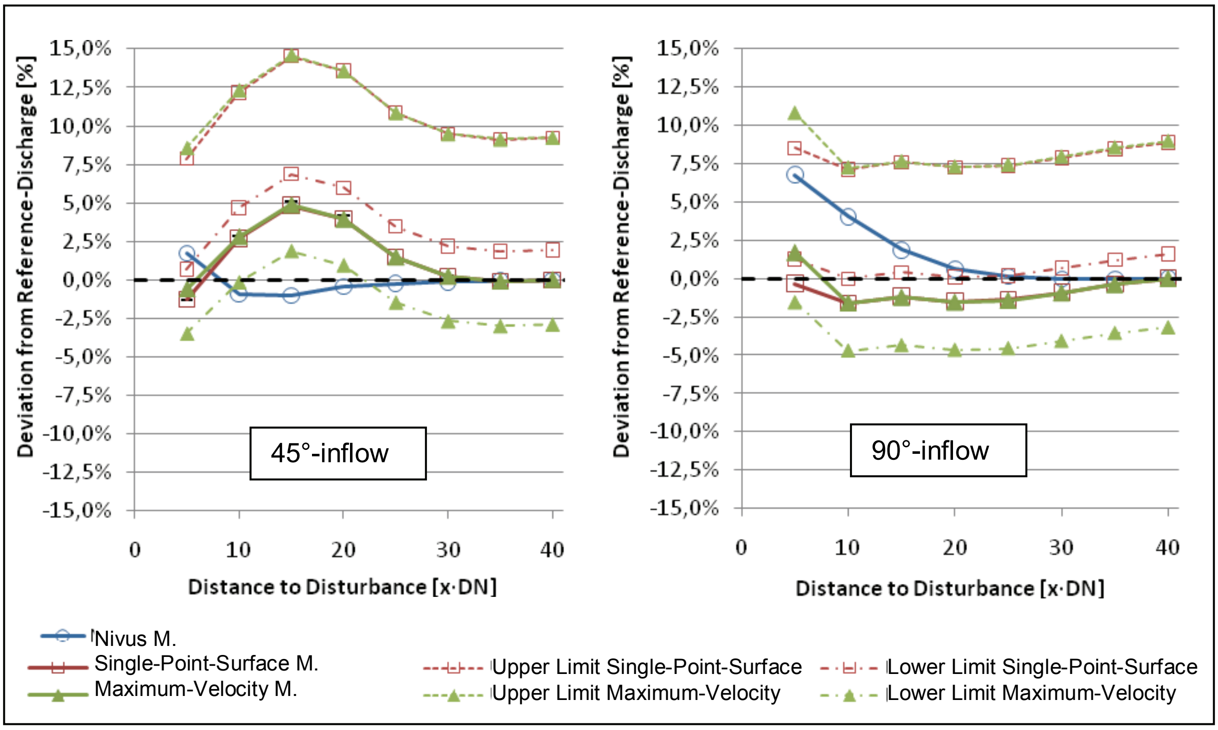

Figure 7: Calibrated deviation from reference-discharge over the distance from the inflow (left 45°; right 90°)

	Range for factor x_i	Factor x_i chosen	Factor calibrated at 40·DN	
			90°-inflow	45°-inflow
Nivus M.	-	-	0.94	0.94
Single-Point-Surface-M.	0.84 - 0.90	0.80	0.83	0.82
Maximum-Velocity-M.	0.80 - 0.90	0.89	0.83	0.82

Table 2: Calibrated x_i factors

4 CONCLUSIONS

In this study two variants of lateral inflow into a pipe (45° and 90°) were investigated by numerical flow simulation in regard to the calculation of mean velocities and discharges from disturbed velocity profiles. Special interest was given to the minimum necessary distance of the measuring section from the disturbances through the lateral inflow. The evaluation criterion was the dimensionless difference between reference discharge and calculated discharge, derived with the measurement models described above. The input data for the measurement models were produced by 3-dimensional CFD-simulations to avoid uncertainties that are inherent in real measurements.

The results show a distinct influence of the lateral inflow on the downstream pipe reach and potential measuring sections. Especially the 90°-inflow produces a strong deformation of the flow pattern.

For the boundary conditions used in this study (Chapter 1 and Table 1) it was shown, that the recommended minimum distance for placing a measuring section downstream of a lateral inflow is not sufficient.
The recommendation for single-point-measurements should be increased from 15·DN to 30·DN for comparable flow configurations.

For methods with velocity profile measurement like the Nivus method the recommendation should be increased from 10·DN to 20·DN.

It seems to be not recommendable to operate a single-point sensor without calibrating it for the specific measuring section. With calibration and under consideration of the above recommendations the discharge error resulting from the measurement model could be reduced below 1%.

In further studies the transferability of the above recommendations will be proved for extended hydraulic boundary conditions and modified geometrical configurations. Additionally the k-ε-turbulence model used in this study will be replaced by a turbulence model that considers anisotropy (i.e. Reynolds-Stress-Model).

LIST OF REFERENCES

Bardiaux, J.B., Bonakdari, H., Larrarte, F., Mose, R. und Vazquez, J. (2006): *Velocity Profiles through a sewer channel: Using CFD to obtain velocity fields*. In: Dresdner Wasserbauliche Mitteilungen, (32), S. 327 - 335.

Bonakdari, H., Larrarte, F. und Joannis, C. (2007): *Effect of a bend on the velocity field in a circular sewer with free surface flow.* In: Chocat, B. und Desbordes, M.: Novatech. Sustainable Techniques and Strategies in Urban Water Management. Lyon. S. 1401 - 1408. ISBN 2950933777.

CEN ISO /TS 25377 (2007): *Leitfaden zu Messunsicherheiten in der Hydrometrie (HUG)*. Berlin: Beuth Verlag, 58 S.

Colebrook, C.F. (1939): *Turbulent flow in pipes with particular reference to the transition region between the smooth and rough pipe laws.* In: J. Institution Civil Engineers, 1939 (4), S. 133-156.

DWA-M 181 (2008): *Messung von Wasserstand und Durchfluss in Abwasseranlagen*. DWA-Merkblatt 181, Hennef: DWA.

EN ISO 748 (2005): *Hydrometrie - Durchflussmessung in offenen Gerinnen mittels Fließgeschwindigkeitsmessgeräten oder Schwimmern*. Berlin: Beuth Verlag, 52 S.

Erb, H.G. (1998): *Durchflussmesstechnik für die Wasser- und Abwasserwirtschaft*. Essen: Vulkan-Verlag, ISBN 3-8027-2177-2

Ferziger, J. und Peric, M. (1997): *Computational Methods for Fluid Dynamics*. Berlin: Springer Verlag, ISBN 978-3-540-42074-3.

Kölling, C. (1994): *Finite-Element-Simulation der Geschwindigkeitsverteilung in Kanälen und teilgefüllten Rohrleitungen*. München, Technische Universität München, Lehrstuhl für Hydraulik und Gewässerkunde, Dissertation.

Kölling, C. und Valentin, F. (1995): *SIMK - Durchflussmessungen*. In: Wasserwirtschaft, 85/1995, S. 494 - 499.

Launder, B.E. und Spalding, D.B. (1974): *The numerical computation of turbulent flow*. In: Computer Methods in Applied Mechanics and Engineering, 1974 (3(2)), S. 269 - 289.

Lecheler, S. (2009): *Numerische Strömungsberechnung*. Wiesbaden: Vieweg+Teubner, ISBN 978-3-8348-0439-6.

NIVUS (2005): *Betriebsanleitung OCM Pro Aktiv*. Eppingen, Nivus GmbH, 37 S.

Patel, T. und Gill, L. (2006): *Volume of fluid model applied to curved open channel flows*. In: Rahman, M. und Brebbia, C.A. [Hrsg.]: *Advances in Fluid Mechanics VI.* Bd. 52, Southampton: WIT Press, ISBN 1743-3533, S. 362 - 369.

Salaheldin, T.M., Imran, J. und Hanif Chaudhry, M. (2004): *Numerical Modeling of three Dimensional Flow Field Around Circulaer Piers*. In: Journal of Hydraulic Engineering, 2004 (2), S. 91 - 100.

Schnieders, A. (2009): *Analyse von hydrodynamischen Vorgängen in Rohrleitungen mit Hilfe der numerischen Strömungssimulation*. Münster, Fachhochschule Münster, Labor für Wasserbau und Wasserwirtschaft, Diplomarbeit.

Siedschlag, S. (2004): *Horizontale Doppler Strömungsmessung in Flüssen und offenen Kanälen.* In: Lehrstuhl für Hydraulik und Gewässerkunde TU München: Q-Seminar: Neue Entwicklungen in der Durchflussmessung in natürlichen und künstlichen Gerinnen. Konferenz am 03.2004, München. S. 9.

Teufel, M. und Solliec, L. (2010): *Using velocity profiles to determine an accurate volume flow rate at small and large dimensions.* Novatech 2010. Konferenz am June 27 - July 1st 2010, Lyon.

Uhl, M. (2000): *Messbereiche und Messfehler*. In: ATV-DVWK [Hrsg.]: *Durchflussmessung in Regenüberlaufbecken.* ATV-DVWK-Schriftenreihe 19.Hennef: GFA - Gesellschaft zur Förderung der Abwassertechnik, ISBN 9783933707567, S. 111 - 132.

Vosswinkel, N. (2010): *Numerische Strömungssimulation zur Unterstützung von Messkampagnen mittels Kreuzkorrelation unter besonderer Berücksichtigung des Sensornahbereiches*. Münster Fachhochschule Münster, Labor für Wasserbau und Wasserwirtschaft, Masterthesis

Wintterle (2008): *Modellentwicklung und numerische Analyse zweiphasig geschichteter horizontaler Strömungen*. Stuttgart, Universität Stuttgart, Fakultät Maschinenbau Dissertation.