ČESKÉ VYSOKÉ UČENÍ TECHNICKÉ V PRAZE

Fakulta stavební

Ing. Jaroslav Pollert, Ph.D.

Matematické modelování překážek ve stokovém systému

Mathematical modelling of obstacles in sewer system

Summary

Sewer and wastewater systems suffer from insufficient capacity, construction flaws and pipe deterioration. This paper presents the results of a research developed inside the CARE-S project (Computer Aided Rehabilitation of Sewers) focused on sewer and storm water networks management.. Hydraulic simulations are usually done running commercial models that apply, as input, default values of parameters that strongly influence results. Using CCTV inspections data to catalogue failures affecting pipes, a 3D model was used to evaluate their hydraulic consequences. The translation of failures effects in parameters values producing the same hydraulic conditions caused by failures was done through the comparison of laboratory experiences and 3D simulations results. A Visual Basic routine was developed in order to automatically calculate the effects of temporal decline in terms of pressure drops in the system components: coefficients of local or distributed head loss. Those parameters could be the input of 1D commercial models instead of the default values commonly inserted.

This will move the results from 1D models closer to reality and decrees a number of calibrations needs. It can be also used during the operation time for changing performance capacity due to deduction of failures, or for prediction of capacity due to predicted failures on a sewer system.

Souhrn

Stoky a stokové systémy často trpí nedostatečnou kapacitou způsobenou vlivem stárnutí nebo poruchou struktury potrubí. Výsledky tohoto výzkumu, který byl uskutečněn v rámci CARE-S projektu (Computer Aided Rehabilitation of Sewers), se zaměřují na stoky jednotného a dešťového systému. Hydraulické simulace v jednorozměrných komerčních modelech, které používají defaultní nebo modifikované hodnoty ztrátových součinitelů, velmi silně ovlivňují výsledky kapacitního plnění. Využitím kamerové inspekce se dají zaznamenat překážky, které se následně katalogizují dle normy EN 13508. Použitím matematického modelu se dají nasimulovat ztrátové součinitele jednotlivých typů a velikostí překážek. Spojením těchto dvou metod můžeme získat ztrátový součinitel v reálném stokovém systému. Data byly samozřejmě ověřovány i experimentálně.

Nyní je možné data z kamerové inspekce použít ke zpřesnění 1D modelů díky automatickému programu pro přepočet ztrát. Každé potrubí má poté jiný ztrátový součinitel bližší realitě. Je tak možné lépe určit problematická místa stokového systému.

Keywords

Failures in sewer system; CCTV inspection; FLUENT; CARE-S

Klíčová slova

Překážky ve stokovém systému, kamerová inspekce, FLUENT, CARE-S

České vysoké učení v Praze

Název: Mathematical modelling of obstacles in sewer system

Autor: Ing. Jaroslav Pollert, Ph.D.

Počet stran: 33

Náklad:

ISBN

Contents

Introduction

The research project CARE-S (Computer Aided RE-habilitation of Sewer Networks) financed by the European Council under the Fifth Framework Programme deals with the study of urban drainage systems in infrastructural and in economic terms. A decision support system applicable to various local conditions will be developed. It will establish cost-effective rehabilitation strategies for public sewer and storm water networks of any dimension.

The hydraulic performance is the most important parameter for the analysis of the service level of the wastewater collection system. It depends on the dimension of the pipes, but also on temporal effects of structural deterioration, blockages, roots, sags, etc., which affect the capacity to transport wastewater and runoff, and to avoid local floods and excessive pollution discharges. Software for network flow capacity are commonly used, but those packages do not consider the gradual capacity reduction, after a long period of time.

WP3.2 ('FLUENT' model) assesses the effects of an expected future structural deterioration of hydraulic performance.

Failures, on the CCTV inspection data coding system(European Standard – prEN 13508-2 – Condition of Drain and Sewer systems outside buildings – Visual inspection coding system) are divided into groups. The hydraulic capacity of sewer affected by failures is simulated using a mathematical 3D model. Each failure is simulated separately. Trough the comparison of a new/clean pipe and a pipe with some failures hydraulic parameters are evaluated. Those parameters describe the real pipe condition better than the default values usually applied in 1D models such as MOUSE, SWMM or Hydroworks. Thus, the model of the sewer system is closer to the reality and the mathematical 1D model can produce more accurate results that, hopefully, require less effort and data for calibration.

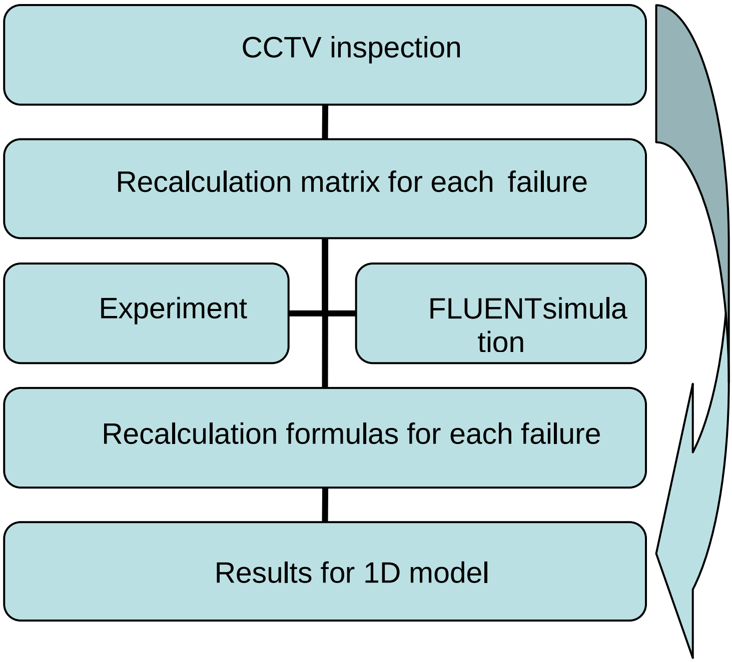
The methodology is based on pressure losses on fully filled pipe compared to clean new pipe without any obstacles. The differences between pressure losses of those two pipes gives head loss due by failure.

The capacity of the pipe is define when the pipe becomes pressurised, so free surface flow conditions are not considered. During free surface condition the pipe does not have full capacity and obstacles in sewer will only increase water level in the system.

A numerical 3D model was used for simulating the hydraulic effects of each failure on the sewer flow. The main outputs of those simulations are the values of hydraulic parameters that could produce the same effects of the failures in the system The 3D model used is FLUENT. The 1D models considered are MOUSE, Info Works and SWMM.

Methodology

Simulations using FLUENT were performed on fictitious pipes, trying to represent all the possible failure conditions. Comparing a new/clean pipe with a pipe affected by failure we derived hydraulic parameters that describe the real pipe condition better than the default values. The methodology is based on pressure losses on filled pipe compared to clean new pipe without any obstacles. Differences between pressure losses of these two pipes can be interpreted as a local head loss due to failure.



Methodology of overall calculation

The capacity of the pipe is evaluated when it becomes pressurised, so we didn't consider the free surface flow conditions. During free surface condition the pipe does not have full flow capacity and the presence of eventual obstacles can only increase the internal water. For the calibration and the validation of FLUENT results an experimental pipe, divided into three sections, was used. In the middle section obstacles were put. Measuring pressure differences between the beginning and the end of the pipe we compared that value with pressure differences received from FLUENT modelling.

For the simulations and study of failure effects the fictitious pipe used was of 20 m in length and with a circular section with a diameter of 1m.

Simulation done with FLUENT on the fictitious pipe without failures gave pressure and flow conditions. Starting from the initial condition of flow 0,785 [m^3/s] and roughness 0,001 m which correspond to an old pipe in concrete material.

Obstacles and failures were simulated considering they occurred in the middle of the pipe. From difference of pressure losses between the beginning and the end of the fictitious pipe and the pipe with failures, we receive local losses due by the failure.

The following step was the definition of a new pipe that will be insert in the network, simulated by the 1D model, instead of the pipe with failures. This new pipe can be considered like an "equivalent pipe" of the real one: it will be described by the hydraulic parameters, defined with the 3D simulations, which will produce in the pipe the same hydraulic performances of the specific failure studied.

For calibration and verification of FLUENT results we used an experimental pipe, which was divided into three sections. into the middle section some obstacles were put as in sewer should be. the pressure differences measured at the end stream of the pipe was compared with the pressure differences resulting from FLUENT modelling.

Then, from the difference of pressure losses between the beginning and the end of the fictitious pipe and the pipe with failures, we receive local losses due by failure.

The equation known as the Darcy-Weisbach formula expresses the losses of head in pipes as given by

$$\Delta h = j \cdot l = \lambda \cdot \frac{l}{D} \cdot \frac{v^2}{2g} \qquad (1)$$

Where λ is a friction coefficient. This equation applies to turbulent flow.

Comparing this equation with equation (1), (3) and (4):

$$\frac{\Delta p}{\rho \cdot g} = \lambda \cdot \frac{l}{D} \cdot \frac{v^2}{2g} = \beta_r \frac{Q^2}{D^5} \cdot l \qquad (2)$$

$$\Delta p = \rho \cdot \lambda \cdot \frac{l}{D} \cdot \frac{v^2}{2} \qquad (3)$$

We will consider as roughness coefficient the Manning parameter „n“ [$m^{1/3}$/s] and as friction coefficient the λ. The 1-D model we will use permit to insert the following roughness coefficient:

- MOUSE: the Manning coefficient [$m^{1/3}$/s];
- InfoWorks: the Manning coefficient [$m^{1/3}$/s], the Gauckler Strickler coefficient [s/$m^{1/3}$]; friction factor λ; InfoWorks can use either the Colebrook-White equation or the Manning formula to calculate hydraulic roughness. You may use two values, one for the bottom third of the link and one for the rest of the cross section, which is usually smoother. The default value for an individual conduit is the global value specified for the drainage system.
-

To use the Colebrook-White equation, select **CW** as the hydraulic roughness type. Typical values are as follows:

Description (Pipe sewers in good condition)	**ks mm**
Surface water	0.6

Foul or combined	1.5
Smooth concrete	1.5
Smooth brick	3.0
Rough concrete or brick	15.0
Old sewers -- part blocked	15.0 to 300.0
Smooth earth channels	60.0
Rough earth channels	300.0
Overgrown earth channels	600.0

To use the Manning equation, select **Manning** or **N** as the hydraulic roughness type.

Historically, HydroWorks treated the Manning's value input by the user as 1/n. We retain this treatment of Manning's for former HydroWorks users. You can now select **N** as roughness type and enter the normal Manning's n value.

Typical values are:

Description	1/n (metric) (select Manning)	n (select N)
Smooth concrete	83	0.012
Rough concrete or brick	50	0.02
Smooth earth channels	33	0.03
Rough earth channels	5 to 25	0.2 to 0.04

If you define a large depth of sediment it is recommended that the bottom part of the link be made rougher to represent the high roughness of irregular sediment deposits. A ks value of 30 to 50 mm would be appropriate.

- SWMM: the „n“coefficient [$s/m^{1/3}$].

Relationship between Manning roughness factor “n” and the friction coefficient “λ“.

Starting from the between the beginning section and the end section of the fictitious pipe and pressure difference in the same sections of the pipe with failure, it is possible to evaluate the new “n” or “λ” values. that will be used to simulate the failure presence in the pipe. Instead the pressure difference as a local head loss can be introduced. Local headloss can be calculated using the following equation:

$$\Delta h_{loc} = \frac{kv^2}{2g} \qquad (4)$$

Where “k” is the headloss factor . “k” can be estimated with:

$$k = \frac{\Delta h_{loc} \cdot 2g}{v^2} = \frac{2\Delta p_{loc}}{\rho v^2} \qquad (5)$$

If you need to recalculate for some reason Manning coefficient instead of k, the following procedure can be done:

$$n_{new} = \sqrt{n_{orig}^2 + \frac{k \cdot D^{\frac{4}{3}}}{124{,}5 \cdot l}} \qquad (6)$$

Results

Results we can divided into 3 parts – modelling of flow using 3D model, experimental data and sensitivity analysis. Results from each part was compared with results from the other parts.

Mathematical modeling

Important note: all of the results are included in a file called "FLUENT results.xls" due to the saving space.

Mathematical modeling of flow in pipe was done with program FLUENT, which contains several turbulent models. Before any simulation was started the turbulence models was tested on fictitious pipe and compared with Darcy-Weisbach formula. As a best model the k-ω turbulent model was chosen. It is very similar to well known k-ε model. One of the advantages of the chosen model is also wall roughness model, which has model for shear flow correction. It helps to save memory due to the wider calculation mesh.

This section presents the standard and shear-stress transport (SST) k-ω models. Both models have similar forms, with transport equations for k and ω. The major ways in which the SST model differs from the standard model are as follows:

- gradual change from the standard k-ω model in the inner region of the boundary layer to a high-Reynolds-number version of the k-ε model in the outer part of the boundary layer
- modified turbulent viscosity formulation to account for the transport effects of the principal turbulent shear stress

The transport equations, methods of calculating turbulent viscosity, and methods of calculating model constants and other terms are presented separately for each model.

Calculation mesh

For simulations in 3D is necessary to create a calculating mesh. This mesh must comply with these criteria:

- Triangle mesh was used – better fits into circular pipe than squares
- Number of cells around walls must be thick enough, because for pressure losses are important boundary layer.
- Ratio between sizes of smallest and biggest triangle cannot be higher than 10
- Other criteria for stable calculations – quality of the mesh

For each obstacle separated mesh was created. Next pictures show some examples of mesh quality for intruding pipe D=0.7 m, 0.5 m of intrusion:

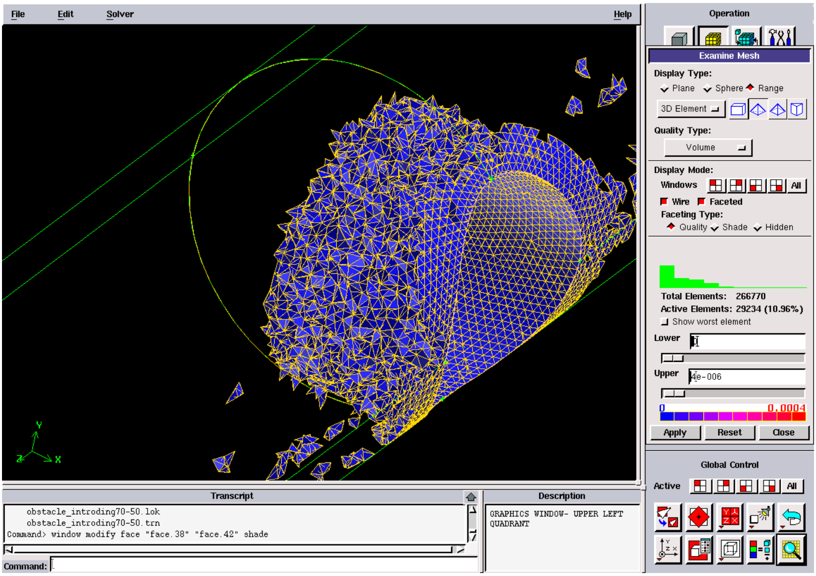


Figure 1 minimal volume of triangle mesh

Simulation of obstacles

Obstacles were divided into groups and these groups were simulated separately. Due to the different dimensions, the relative sizes to diameter were created. Each type was simulated in several different sizes. Failures were simulated as if they occurred in the middle of the pipe. From difference of pressure losses between the beginning and the end of the fictitious pipe and the pipe with failures, it is possible to evaluate the local head loss due to the failure. Next chapters present some examples of obstacles simulated.

Displaced pipe

This simulation of obstruction in sewer system was simulated as two pipes of 10 m in length with different levels of displacement connection. The example figure (Figure 1) shows pressure drop on 20 m length pipe due to two types of displacements: 20% and 10% of displacement. Depending on the percentage of displacement the final formula obtained to calculate the local head loss due to this kind of failure is:

$$k = 0{,}0122 \cdot e^{12{,}295\%}$$

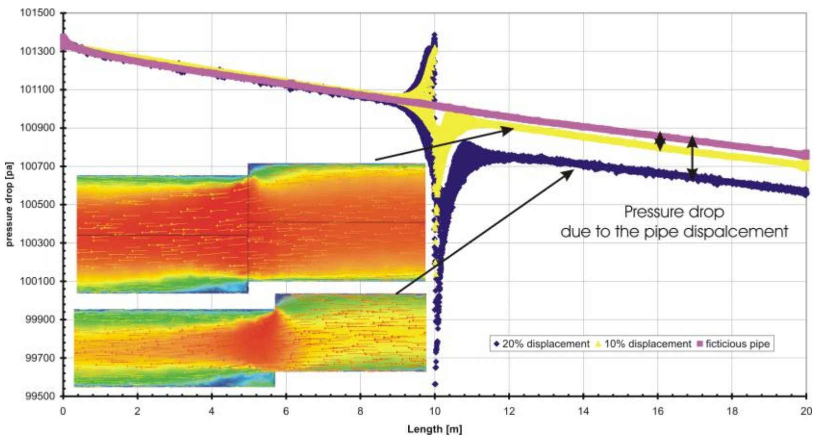


Figure 2 pressure losses due to pipe displacement

Obstacle "brick"

This obstacle is like a brick lying on the pipe surface. The Last simulation is a brick inside the pipe, where water can flow under the brick also. It was only for testing if this will change the flow conditions.

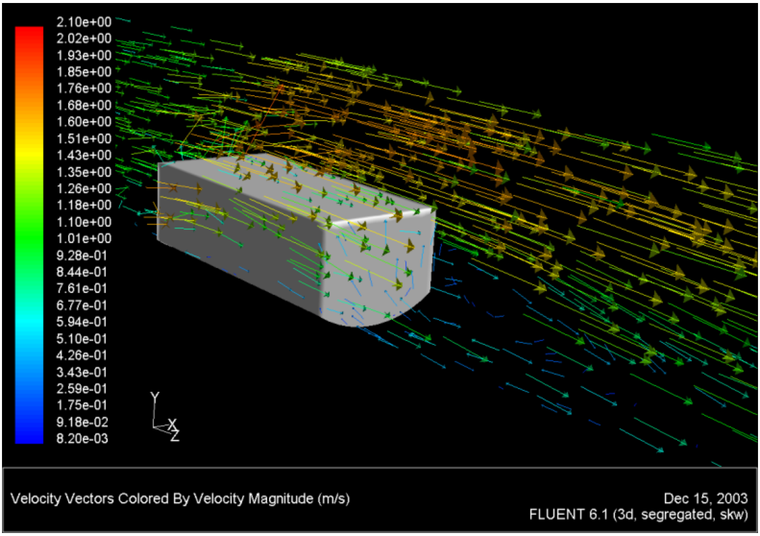


Figure 3 velocity vectors for brick

Obstacle "Pipe through pipe"

This is the simulation of pipe or cables intruding through sewer.

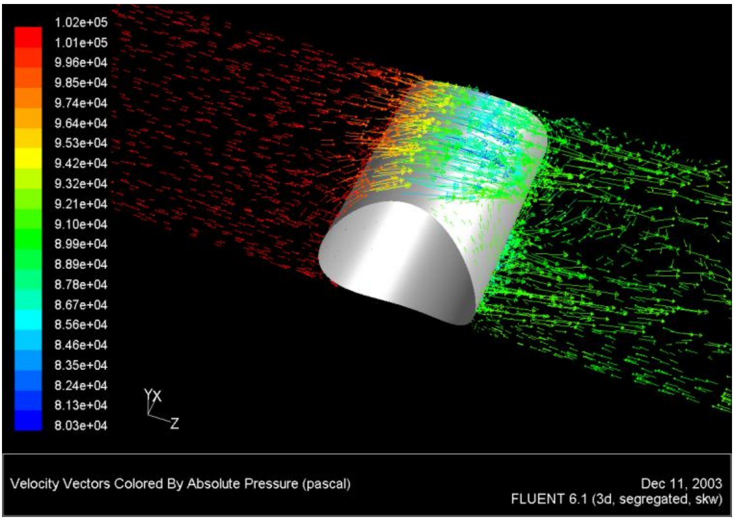


Figure 4 velocity vectors colored by pressure around the 0,8 m pipe intruding into the transversal direction

Obstacle "Partly intruding pipe"

This is the simulation of pipe or cables intruding sewer. This model is used for intruding connection and for obstacle model. Several different sizes and distance of intrudsiont was simulated.

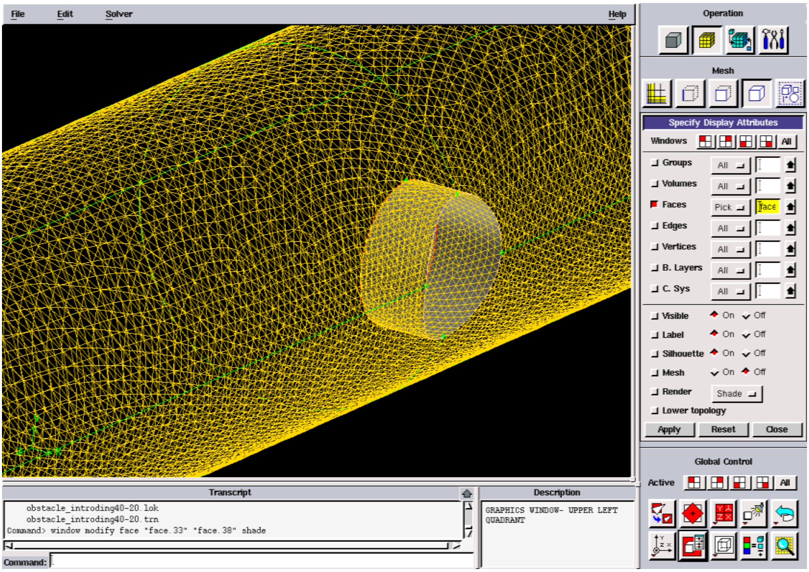


Figure 5 mesh of 0,4 m pipe intruding into main pipe (0,2 m)

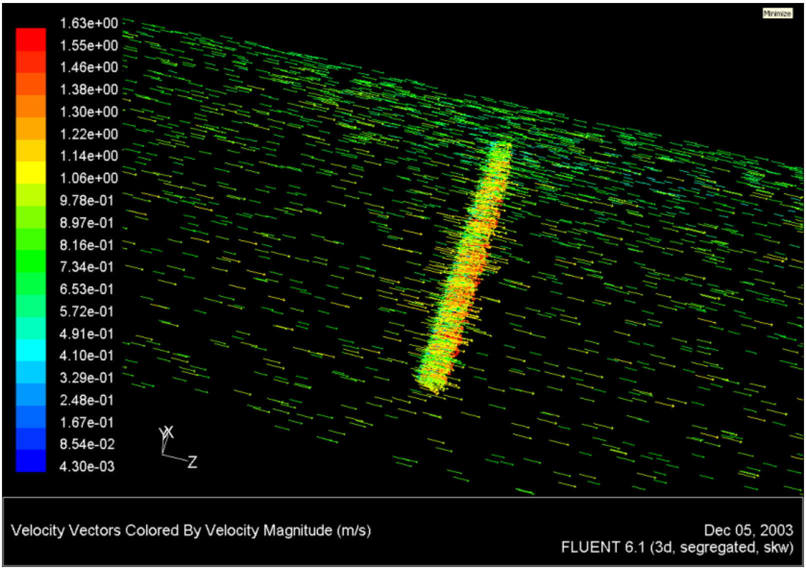


Figure 6 velocity vectors for 0,2 m pipe intruding 0,7 m inside

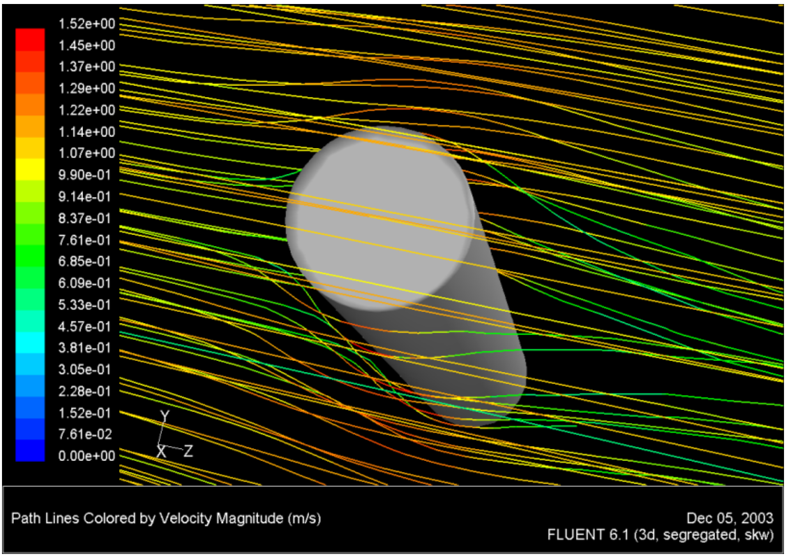


Figure 7 path lines colored by velocity magnitude around 0,4 m pipe partly intruding into sewer

Obstacle "Local solid sediment"

This type of obstacle is like local solid sediment on bottom.

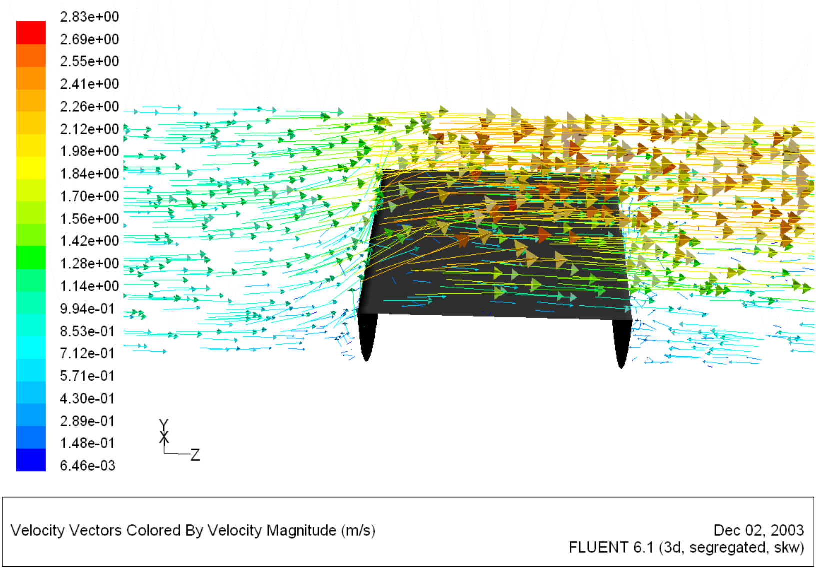


Figure 8 velocity vectors on 50% sediment

Roots

For simulation roots intruding into sewer was developed a different model. Roots were simulated as a porous media.

Inertial Losses in Porous Media

At high flow velocities, the constant C_2 in Equation (7) provides a correction for inertial losses in the porous medium. This constant can be viewed as a loss coefficient per unit length along the flow direction, thereby allowing the pressure drop to be specified as a function of dynamic head.

For modelling a perforated plate or tube bank, we can sometimes eliminate the permeability term and use the inertial loss term alone, yielding the following simplified form of the porous media equation:

$$\nabla p = -\sum_{j=1}^{3} C_{2_{ij}} \left(\frac{1}{2} \rho v_j v_{\max} \right) \tag{7}$$

or when written in terms of the pressure drop in the *x, y, z* directions:

$$\Delta p_x \qquad -\sum_{j=1}^{3} C_{2_{xj}} \Delta n_x \frac{1}{2} \rho v_j v_{\max}$$

$$\Delta p_y \qquad -\sum_{j=1}^{3} C_{2_{yj}} \Delta n_y \frac{1}{2} \rho v_j v_{\max}$$

$$\Delta p_Z \qquad -\sum_{j=1}^{3} C_{2_{zj}} \Delta n_z \frac{1}{2} \rho v_j v_{\max} \tag{8}$$

Again, the thickness of the medium (Δn_X , Δn_y , or Δn_Z) is the thickness defined in our model.

Coefficient used in our roots model for inertial resistance:

$$\Delta p = C_2 \cdot \frac{1}{2} \cdot \rho \cdot v \cdot v_{\max} \tag{9}$$

$$C_2 = \frac{\frac{1}{2} \cdot \rho \cdot v \cdot v_{\max}}{\Delta p}$$

C_2=5 in all directions *x, y, z*

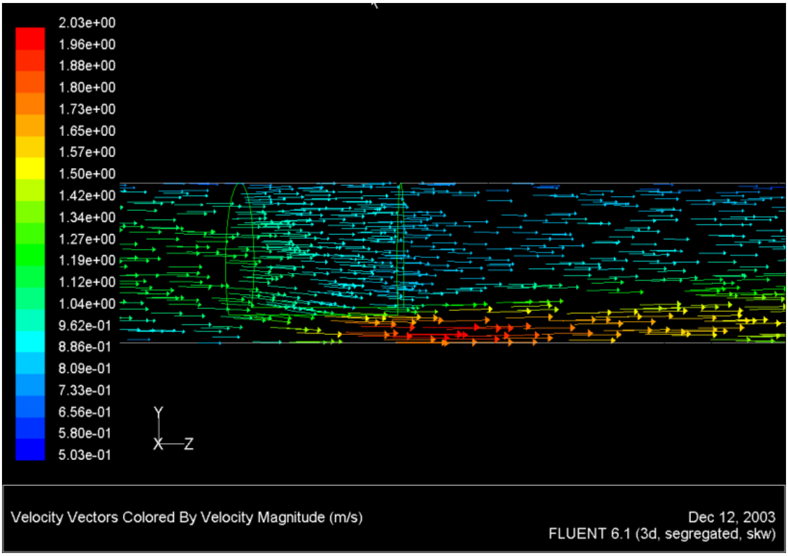
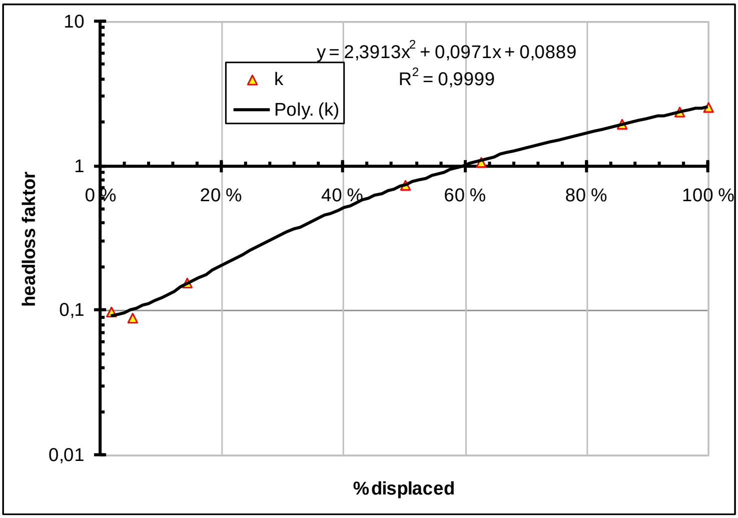


Figure 9 velocity vectors of flow through intruding roots



Graph 1 root loss factor

Obstacle "Partly intruding pipe"

This is the simulation of pipe or cables intruding sewer. This model is used for intruding connection and for obstacle model.

Several different sizes and distance of intrusion were simulated. Depending on the percentage of intersection area occupied by intruding pipe the final formula were obtained to calculate the local head loss:

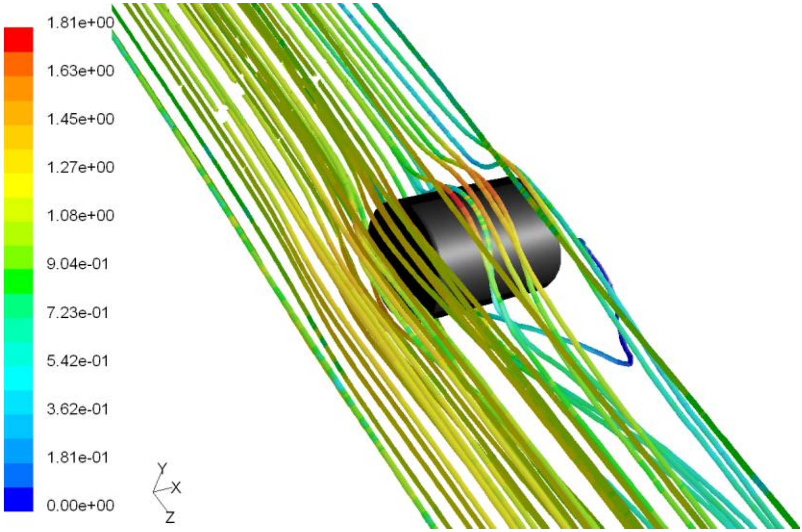


Figure 10 Path lines of velocity field around 50% intruded pipe

$$k = e^{14,673\%^3 - 17,544\%^2 + 12,324\% - 3,6518}$$

Sags

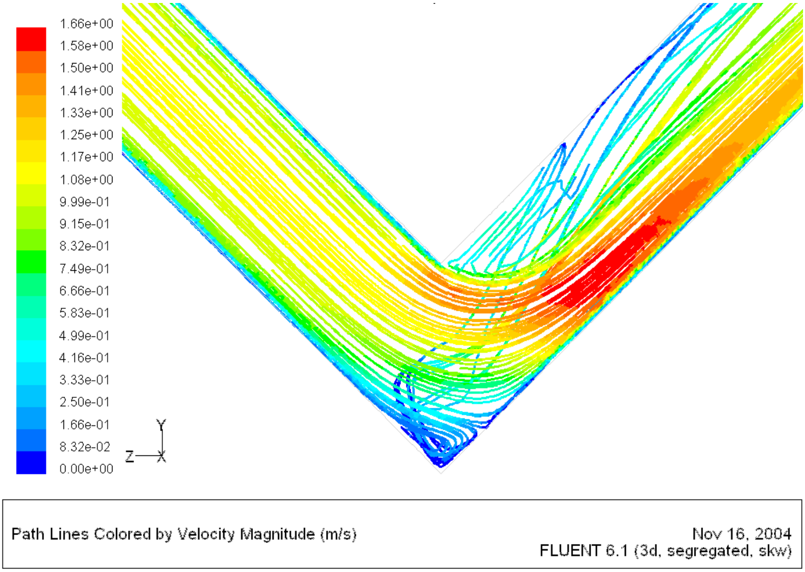


Figure 11 Path lines colored by velocity in 90° elbow

Sags are caused by failure of the pipe bedding on the bottom of the trench. This type of failure causes a section of the pipe to drop below proper grade. Water remains trapped in the sag and solids suspended in the water tend to settle in the sag area.

$$k = 3 \cdot 10^{-5} \cdot \alpha^2 + 0{,}0047 \cdot \alpha + 0{,}0921$$

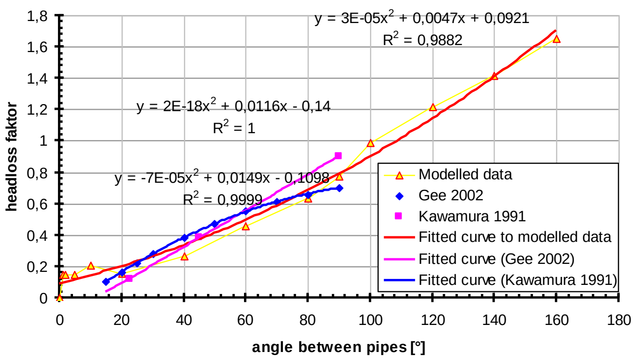


Figure 12 Sags pressure losses compared with literature

Study of sags was focused mainly to small angles (ά), because these are more obvious. However, larger angles were studied too to compare it with literature review. Difference between literature and modeled data are mainly due to different shapes of elbow. We are considering sag to be sharp, but in literature the elbow is not too sharp (elbow in water pipe).

Obstacle brick

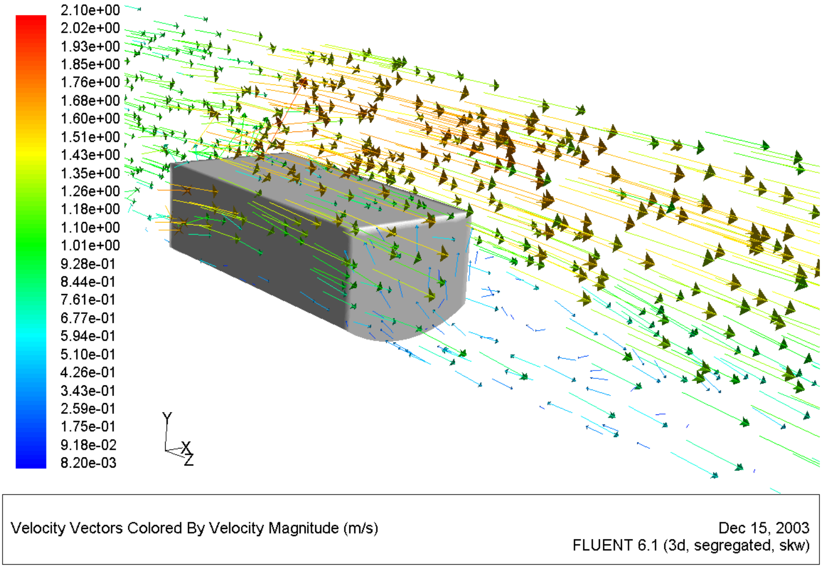
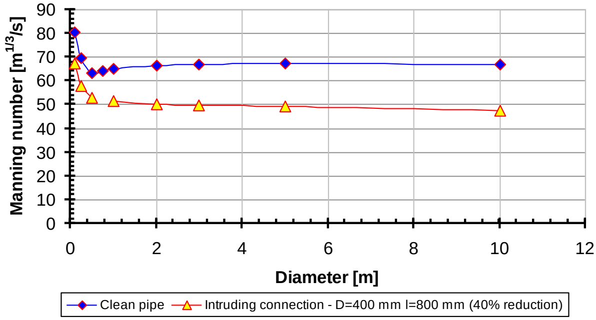


Figure 13 – brick in a sewer

Study of velocity and diameter influence

All previous examples are using 1 m pipe diameter and 1 m/s velocity inflow. So next logical step is to prove low influence of these values to be able omit them. Sometimes it is problematic, because we are running out of validity of mathematical k-ω model (low velocities, small diameter). We also study shape influence (oval, rectangular, egg shape) to pressure



losses.

Figure 14 – diameter influence to manning number

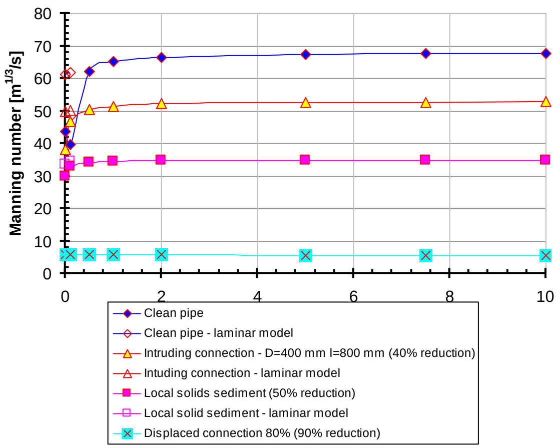


Figure 15 – velocity influence to manning number

Experiment

For the validation of result an experimental pipe was created. Experiments were performed in the Norges Teknisk-Naturvitenskapelige Universitet (NTNU), Trondheim, Norway.

The experimental pipe consists of (Figure 3):

1. inflow pipe D=0,05 m
2. flowmeter on inflow pipe
3. beginning of experimental pipe for development flow field

4. inflow piezometer
5. outflow piezometer
6. middle piece, which can be removed and obstacle can be inserted
7. removable obstacle
8. experimental pipe
9. outflow

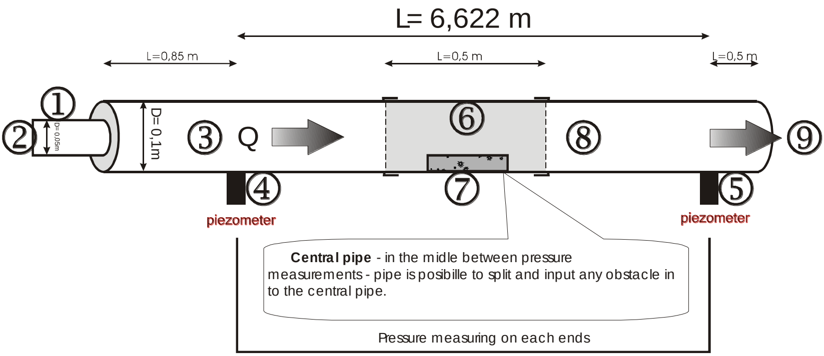


Figure 16 experimental pipe

The first experiment was run without any obstacle to compare pressure losses due to the wall friction. The second part of the experiment was performed with the insertion of obstacles in the middle of the experimental pipe. Two types of obstacles were inserted and compared with mathematical model. Pressure losses on experimental pipe were modeled. The first obstacle was 0,1 m in length and 0,05 in width and height (it filled pipe till the middle of the section). It was placed in the middle bottom of the experimental pipe.

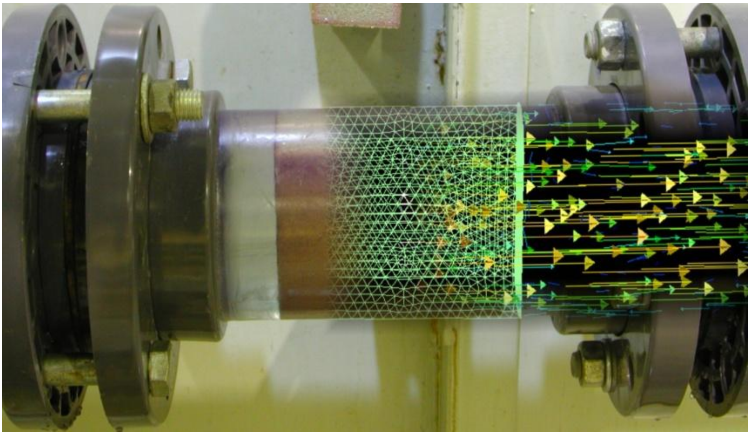


Figure 17 obstacle inside the experimental pipe

The second obstacle was 0,1 m in length and 0,05 in width and height (it filled pipe till the top). It was placed in the middle section of the experimental pipe.

The same procedure (same pipe, flow rate, geometry) was done with mathematical model FLUENT. The same turbulent model for flow as in the previous model was used to compare results from experiment. Pressure losses on experimental pipe were modelled. On the beginning of the curve you can see pressure drop due to the inflow. On zero is first pressure measurement and second pressure measurement is on the end. In the middle is pressure loss due to the obstacle.

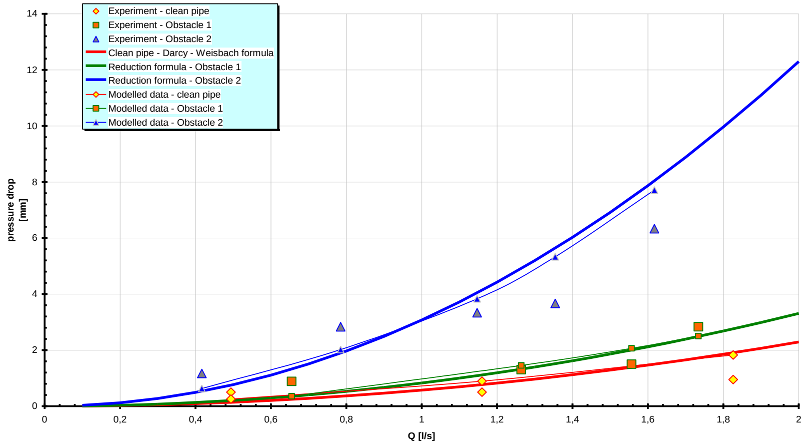


Figure 18 comparison of experiment results and mathematical modelling

Figure 18shows a confront between experiment and mathematical results on clean pipe, pipe with the first obstacle and pipe with the second obstacle. Results from experiment show a good accordance with the modelled data. Data from experiment and FLUENT simulations for the same flow rate were compared. Only the comparison between pipes without obstacles shows difference between experiment and modelled data. It was due to small difference and fluctuating inflow rate.

Results from experiment show a good accordance with the modelled data. Data from experiment and FLUENT simulations for the same flow rate were compared. Only the comparison between pipes without obstacles shows difference between experiment and modelled data. It was due to small difference and fluctuating inflow rate. (Pollert J, 2004)

Results

Figure shows the curves developed, equations to simulate the local head loss produced by different kind of failures: obstacles, pipe’s displacement, roots and sags. Result shows very similar local losses for similar kinds of obstacles for identical intersection area occupied by obstacle. Because of this we can group them and create more universal equations. This helps to convert CCTV inspection results to local losses.

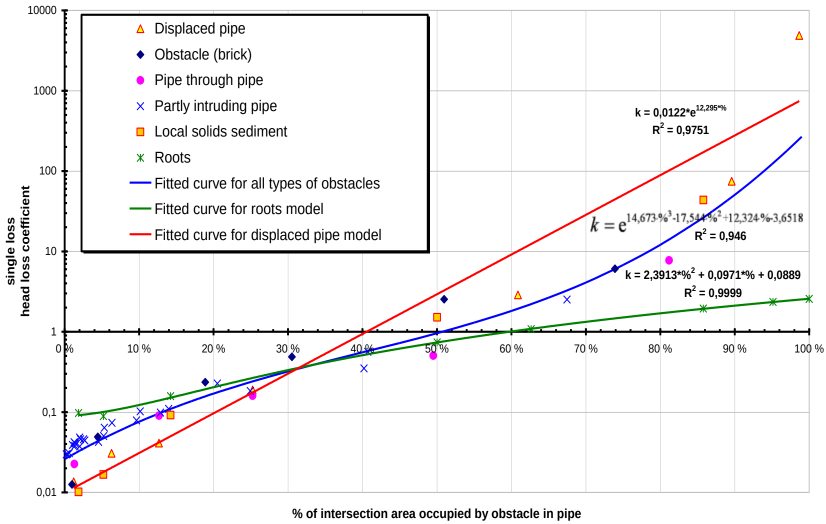


Figure 19 – overall results – several different types of simulated failures

Program for automatic conversion

For automatic conversion of CCTV inspection data to changed pipe parameters program, using conversion matrix, has been developed,. This matrix allows converting input form 1D UDM model, calculating their new values including temporal decline and fed them into the 1D model again.

The developed tool, named “Degradation”, is based on a Visual Basic 6.1 routine and is one of the tools included in the final CARE-S decision support system (DSS).

Conversion matrix

The final output of this study is a recalculation matrix to allow the translation of failures affecting pipes, recorded as required by the EN 13508, in parameters values producing the same hydraulic conditions caused by failures. Those parameters could be the input of 1D UDM instead of the default values commonly inserted. The matrixis divided in three fields, each one in a specific excel spreadsheet:

1. **Inputs from CCTV inspection** : CCTV data collected during inspections, as required by European Standard prEN 13508-2, are transferred from a CCTV database to the excel spreadsheet were pipes affected by failures are listed and their failures described. Failures are defined using the code system developed by the European standards: a 3 letters name, one characterization and quantification;
2. **Recalculation matrix** – the second spreadsheet is the field of quantification of temporal decline in each degraded pipe: failure hydraulic effects are calculated using the formula provided by the 3D numerical analysis. For each pipe the pressure drop coefficients are calculated like local headloss and distributed head loss, as well.
3. **Recalculation formula** –the last field includes the formulae developed using the 3D model in order to evaluate temporal decline: the spreadsheet gives the possibility to add new formulae.The Degradation tool is comprehensive of a detailed help file to support the user with tips for a better use of the code, methodology and results description, besides basic computer requirements.

Conclusion

Recalculation matrix for deteriorating sewer was developed. Using recalculation matrix is easy to change input values to 1D model using CCTV analysis and describe a real condition of the sewer system affected by temporal decline. Those values are used instead of the default hydraulic parameters commonly used for hydraulic simulations.

These equations are implemented in a program which automatically takes data from CCTV inspection, and, through the recalculation matrix, converts them into values closer to the reality.

Data were verified by experiments and results were tested by sensitive analysis.

References

Brebbia, Ferrante – Computational Hydraulics 1983 ISBN 0-408-01153-X

FLUENT users manual 2003

C. Frank Gee, Prepared by LOCATION AND DESIGN DIVISION, HYDRAULICS SECTION, Adopted April 2002, Virginia Department of Transportation

Clemens, F.H.L.R. (2001) Hydrodynamic Models in Urban Drainage: Application and calibration. PhD thesis. Technische Universiteit Delft, Delft.

FLUENT users manual 2003

H. Korving,P. van Gelder, J. van Noortwijk and F. Clemens (2002) Influence of model parameter uncertainties on decision-making for sewer system management

Harremoës, P. and Madsen, H. (1999) Fiction and reality in the modelling world – Balance between simplicity and complexity, calibration and identifiability, verification and falsification. Water Science and Technology. 39(9), 47-54.

Lei J.H. and W. Schilling (1994) Parameter uncertainty propagation analysis for urban rainfall runoff modelling. Water Science and Technology. 29(1-2), 145-154.

Pollert J, R. Ugarelli, S. Saegrov, W. Schilling, V. Di Federico (2004). "The Hydraulic Capacity of Deteriorating Sewer Systems" (2004). IWA 2004, Marrakech, September 2004.

Reda, A.L.L. and Beck, M.B. (1997) Ranking strategies for stormwater management under uncertainty: sensitivity analysis. Water Science and Technology. 36(5), 357-371.

Susumu Kawamura, Integrated Design of Water Treatment Facilities, John Wiley & Sons, Inc, Canada 1991, ISBN 0-471-61591-9

Acknowledgements

This works was supported by the project of Ministry of Education, Youth and Sport of Czech Republic No. MSM 211100002, by the Czech Grant Agency project No. 103/08/0134.

Životopis

Osobní údaje:

Jméno:	Ing. Jaroslav Pollert, Ph.D.
Narozen:	1.5.1971, Praha
Rodinný stav:	ženatý, dvě děti – Jáchym (2004) a Jonáš (2007)
Adresa bydliště:	182 00, Praha 8, Trojská 782/53
Občanství:	Česká republika
Jazyky:	anglicky, rusky

Vzdělání:

1989 – 1995 : Inženýrské studium - Fakulta stavební ČVUT v Praze, obor „Vodní hospodářství a vodní stavby“

1996 - 2001 doktorandské studium - Fakulta stavební ČVUT v Praze, obor „Vodní hospodářství a vodní stavby“

Zaměstnání:

1995 – dosud Fakulta stavební ČVUT v Praze, Katedra zdravotního a ekologického inženýrství

Studijní pobyty:

1994: 4 měsíční studijní pobyt The Ohio State University, Ohio, U.S.A.,

2004: 4 měsíční pobyt na The Norwegian University of Science and Technology, Trondheim, Norsko

2005: 4 měsíční pobyt na The Norwegian University of Science and Technology, SINTEF, Trondheim, Norsko

Odborné zaměření:

Mechanika tekutin, proudění kapalin a suspenzí v otevřených a uzavřených profilech, matematické modelování proudění kapalin a suspenzí, odvodnění urbanizovaných území, čištění odpadních vod.

Ostatní aktivity

Sport – 4 x mistr světa ve vodním slalomu v kategorii 3xC2 muži (1995 – Nottingham, 1999 - Seu d Urquel, 2003 - Augsburg, 2006 – Praha), 2 místo ve světovém poháru ve vodním slalomu v kategorii C2 muži (2003)