Project description

Numerical modelling of erosion damages during floods

Introduction

The climatic change in the coming years are believed to increase the flooding in our rivers. The increase relates to the duration, magnitude and frequency of the floods. The damage from a flood can be caused by the water inundation itself, but there are also other effects. The sediment transport capacity of the river increases strongly during a flood, and this can lead to particular damages to infrastructure. The main cause is scour in the river, both local scour around abutment and bridge piers and general lateral movement of a river, removing foundations for roads, houses etc. The present research proposal investigates numerical models for prediction of scour in rivers. The goal is to be able to predict damages due to the floods and to aid the hydraulic engineer in designing safer structures in rivers, including measures to reduce erosion.

Surveys of damages during floods shows the problem in USA (Lagasse, P. F., 2003): *"Hydraulic factors such as stream instability, degradation, contraction scour, and local scour account for more of our bridge failures (approximately 60 percent) than all other factors combined."... "On-going screening and evaluation of the vulnerability of the nations' highway bridges to scour by State Departments of Transportation have identified more than 18,000 bridges that are considered scour-critical and in need of repair or replacement."*. Lagasse, P. F. (2003) also looked at flood damages in Europe. As an example, the floods in 1997 and 1998 in the Oder River in Poland showed hundreds of bridges damaged or destroyed due to inadequate foundation design or inadequate waterway opening. Over 8,000 bridges were in need of repair.

In Norway we had a large flood in 1995. Substantial damages were done from the flooding of the river Moksa at the town Tretten. This town mainly lies on an alluvial fan that Moksa has deposited over the years at the main valley. The natural process on an alluvial fan is that the river deposits sediments and changes its location now and then, unless proper bank protection structures are present. On the 2nd of July, the river Moksa moved back to one of its former locations, doing severe damage at Tretten by removing several houses and damaging others. Another example from Norway is the construction of the new bridge over the river Verdalselva at Verdal. The river was narrowed to give room for the bridge foundations, resulting in a scour hole that was measured to be 27 meters deep (Sæterbø, 2005). Such a deep scour hole was dangerous with respect to failure of the bridge piers.

Local erosion during floods may also have severe secondary effects. The Norwegian Water Resources and Energy Directorate (NVE) considers landslides in areas of quick clay deposits one of the flood processes with the highest potential for damages in Norway (Sæterbø, 2005). Such landslides are most often triggered by local erosion in a river. An example is the land slide at Verdal in 1893, where 116 people were killed. This was triggered by erosion of the river Verdalselva during a flood. The most commonly used method to prevent landslides in areas of quick clay deposits is construction of structures to prevent erosion in the river where the landslide is assumed to start.

The present project proposes to improve an engineer's capability to investigate water flow and erosion around structures in rivers by use of computational fluid dynamics (CFD). Over the last years this science has gained increased use in research on modelling rivers and sediment transport. This is partly due to increased speed of computers, making it feasible to model rivers with a 3D CFD program for time-dependent cases. Numerical models of rivers are essential tools for river engineers in assessing the performance of hydraulic structures and environmental effects of changing the flow regime. Limitations in the practical use of the models are related to numerical stability, accuracy and the ability to model all the physical processes in rivers. The present project proposes several sub-projects aimed at improving this situation, including improvements of numerical algorithms and generating new ones. This will be useful in predicting river movements during floods, where bank erosion may damage infrastructure as roads, buildings, railways etc, or cause indirect damage through landslides. The current project will improve prediction methods for these problems, enabling engineers to make civil structures safer.

The local scour process

Local scour is one of the classic topics in hydraulic engineering, and has received substantial attention over the last 50 years, both in form of PhD studies and publication. The reason is the large practical impact and consequences for construction of infrastructure near or in the river. Local scour around for example bridge piers is characterized by a complex three-dimensional velocity field. The water in front of the pier forms a wave on the surface due to the stagnation pressure, and a vertical component arises. When encountering the bed, a horse-shoe vortex is formed at the base of the pier. Behind the pier, a vortex street is formed. The complex flow field affects the scour hole around the pier, both in its magnitude of depth and its general geometry. The flow pattern around the pier gives rise to increased shear stress at the bed. Experiments (Dargahi, 1990) indicates that the local bed shear stress can be up to 10 times higher than the average value in the river. The increased shear stress leads to erosion. The depth of the scour hole around the pier can be estimated by simple empirical formulas, usually a factor times the water depth or the pier width. The factor is often in the range of 0.5 - 1.5. This means that the magnitude of the scour hole can be equal to the water depth.

Local scour can also take place close to other constructions in a river. An example is bank protection structures located at the outside of a bend. A local example from Trondheim is the river Nidelva which turns two times on its pass through the city. The river will naturally erode at the outer bank of a bend, but this can not be allowed to happen in a city with major infrastructure. Erosion protection structures have therefore been built along the outer bank. The local branch of the Norwegian Water Resources and Energy Directorate (NVE) are currently investigating several scour holes at these locations. The water in the river have a higher velocity at the water surface, and the stagnation pressure at the bank protection structures will be higher at the water surface. This will lead to a downwards movement of the water, and a secondary current is formed. The secondary current sweeps the sediments away from the bank protection structure, leading to a scour hole. The structure must therefore be placed fairly deep in the bank, otherwise there is a risk of erosion undercutting the structure with corresponding collapse. This will most often happen at very high discharges during a flood, as the bed shear stresses increase with increasing water discharge.

Some engineering measures have been suggested to reduce the local scour depth. One is to add coarser stones around the pier foundations. The problem with this method is that the bed shear stress is also a function of the sediment size on the bed, so the larger stones give increased shear stress. There has also been observed erosion at locations where the grain size distribution change at the river bed. This mechanism is not understood completely at the moment, but it involves changes in the boundary layer and turbulence in this region. The current project aims at investigating this problem in more detail.

Another method to prevent local scour is to build a concrete foundation at the bed of the pier. The problem with this method is that the foundation itself can be undermined by local scour. If the scour depth is a function of the pier width, the wider foundation implies an even greater scour depth. The selected engineering solution to the local scour problem is to design the structure so that it will still be functional even if local scour occur. Then it is very important in the design of the structure to know the magnitude of the local scour hole. Up to now, this could only be done by use of fairly inaccurate simple formulas, or to do a costly physical model study in the laboratory. The present project proposes an alternative method, where the depth of the scour hole can be computed by a CFD program.

A factor affecting the magnitude of the scour hole is the alignment of the construction. A bridge pier is often designed with greater length in comparison with the width. Aligning the pier with the flow will then produce a smaller effective diameter, reducing the scour hole. The strategy, however, depends on the ability of the engineer to align the pier with the water flow. If the bridge is located in a part of the river which is not straight, the water velocity direction may be different on low flows than on high flows. The most severe erosion occur during the highest flows, where measurements of the velocities are difficult and often not done. Also, in planning of new structures, the structure itself will affect the direction of the water velocities.

The complexity of the natural river bed and the water flow field may also produce different water velocities upstream of the construction than the average velocity over a cross-section. The risk of erosion of a particle is a function of the bed shear stress, and this can again be a function of the local velocities. The complexity of the flow field in a natural river is one of the problems when applying simplified formulas for the local scour depth. To take these complexities and other factors into account, one needs to use a physical model or a CFD program as proposed investigated in the present study. As a physical model is fairly expensive, the CFD approach has great promise for future use in hydraulic engineering.

Current state-of-the-art in CFD modelling of erosion

Computational fluid dynamics is currently used as a standard tool in mechanical engineering. In civil engineering hydraulics this is not the case. General purpose CFD programs are often limited when it comes to modelling the boundary conditions at the bed and the free water surface. Using commercial CFD programs, local erosion has only been partially done by the program FLOW3D (Brethour, 2002), but only one sediment size was used. A number of commercial programs also exist that are tailor-made for civil engineering hydraulics. The most well-known are Telemac, MIKE3 and Delft3D. These models are mostly used for the marine environment, and applications on rivers are rare. The models use the hydrostatic pressure assumption in the vertical direction, which works well in lakes and in the sea, but are not be suitable for modelling local scour due to problems of predicting secondary currents in sharp bends and vertical recirculation zones.

Some research groups have instead developed in-house 3D CFD programs. Here in Europe the most well-known group in this field of research is headed by Prof. Wolfgang Rodi at the University of Karlsruhe. This group modelled bed deformation in a 180 degree bend (Wu et. al., 2000). Recently, they also modelled sediment deposition in the Three Gorges Dam in China (Fang and Rodi, 2003). In USA, the most well-known research groups in numerical modelling of sediment transport is lead by Prof. Sam S. Y. Wang at the University of Mississippi. The has made 3D CFD models for rivers where bed changes are computed although most of their journal publications are on a 2D depth-averaged model. Lateral movements of meanders was computed by Duan et. al. (2001) using a 2D model and a structured grid. This meant that only small lateral changes could be computed. A more advanced approach was used by Olsen (2003), computing the formation of a meandering channel using a fully 3D model, including secondary currents in the model. The lateral movement of the channel was computed using an unstructured grid, where the total number of grid cells changed during the computation. This enabled a much more complex geometry to be modelled. Also, since the model was 3D, local scour effects could potentially be included in the computations. Our research group published the first journal paper in the world (Olsen and Kjellesvig, 1998) where the local scour hole depth was predicted using our own CFD program. We are the leading group in the world in many aspects of the CFD modelling of sediment erosion.

Present project

The goal of the present project is to improve existing numerical algorithms and develop new numerical methods in order to improve the stability and accuracy of the CFD predictions for erosion around hydraulic structures. This includes both mathematical/numerical topics and projects to develop a better understanding of the physics of the processes in the river: mostly of boundary layers in flow with rough walls and sediment transport. Development of algorithms for how the processes can be included in the numerical model is also a part of the project.

The present project builds on our current knowledge of CFD modelling using the finite volume approach, the SIMPLE method for pressure coupling and the k-epsilon turbulence model. Algorithms for the free water surface have been developed, and also sediment transport algorithms that can predict bed changes over time. This is also done by using the finite volume method. Both suspended load and bed load are computed, and a roughness predictor based on computed bed form height can be used. Our models uses both structured and non-structured grids, with dominantly hexahedral cells. Algorithms for wetting and drying are used to model lateral movements of the grid.

The present project is divided in four parts:

1. Selection of test cases to compare the numerical model with field/lab data
2. Testing and improvement of numerical algorithms for sediment transport
3. LES study of boundary layer flow with large roughness
4. Publication and knowledge transfer

The points are described in more detail in the following.

1. Selection of test cases

It is very important to test how well the numerical model is able to predict the shape and magnitude of the local scour hole. This testing has to be done against measurements in the field or in the laboratory. A large number of laboratory studies of local scour have been done over the years, and the results are published and available. The present project must begin with selecting a number of these for testing. The selection must be based on the quality of the study and the availability of the necessary data for the numerical model. Also, it would be an advantage to select test cases with a high degree of variation in the different parameters: sediment composition at the bed, including cohesive materials, and different shapes and alignments of the constructions. Both bridge piers and groyne cases should be selected, and one should try to find a large number of constructions with different shapes. The selection should be based on a literature search, and on personal communications with leading researchers in this field.

Here in Norway we have field data from several cases of local scour. Currently, a new project is starting up initiated by the Department of Geography in Trondheim and the local branch of the Norwegian Water and Energy Directorate. In the project, several local scour holes in rivers near Trondheim will be mapped using advanced instruments. This will give a detailed geometry of the scour holes and possibly also the changes over time, if more surveys can be done. The project will also use the instruments and other data collection methods to estimate the grain size distribution in the scour hole. The data from this project will be very useful for testing the numerical model.

The project will also obtain data from international collaboration. We are currently starting up a cooperation with the Federal Waterways Engineering and Research Institute (Bundesanstalt für Wasserbau, BAW) in Karlsruhe, Germany. This laboratory is one of the largest in the world, and they are currently conducting several studies related to local scour. In Germany, river bank protection is often carried out using groynes. The method has also been used in Norway, called “buner”. An example of a case modelled by BAW is the river Oder, where the physical model is 80 meters long and 10 meters wide. Several groynes are located at the outside of a bend, and detailed data for the bed topography around the groynes are measured.

This part of the project will be done by the doctoral student in cooperation with the project manager and our national and international collaborative partners.

2. Testing and improvement of numerical algorithms for sediment transport

Our research group has done initial work on erosion modelling by computing local scour depth around a circular cylinder for clear-water scour (Olsen and Kjellesvig, 1998). Only uniform sediments were used and the geometry was fairly simple. The scour depth was reasonably well predicted, but the shape of the scour hole was not predicted completely accurately according to experiments. This is probably possible to improve by including more accurate algorithms for critical erosion on sloping beds. Most numerical models for sediment transport use the Shield's (1937) curve to compute the critical shear stress for erosion of a particle. This is the most accepted approach in our scientific community. However, physical model tests show that it is difficult to determine this value. The turbulence causes the bed shear stress to fluctuate over time, and although the average bed shear stress is below the critical value for movement of a particle, some times a turbulent burst can increase the shear stress to a level where the particles move. An alternative approach based on statistics may therefore give a better results in cases where the bed shear stress is close to the critical value. The results from sub-project 3 will be used in this analysis, as the Large Eddy Simulations compute the turbulent shear stress in more detail.

For the use of CFD in practical engineering, a relatively coarse grid has to be used. It is then necessary to use wall laws to incorporate the effects of the boundary layer close to the wall. There exist well accepted wall laws for smooth boundaries and boundaries with relatively small roughness. However, less is known about the velocity and turbulence close to the wall for large roughness. The sediments on a natural river bed consists of stones of different shapes and sizes, and form a roughness pattern that is highly complex. The current project will test different new wall laws for rough walls based on the findings of the LES results of sub-project 3.

Early investigations of CFD modelling of local scour (Olsen and Kjellesvig, 1998) found that the magnitude of the scour hole can be underpredicted if the computational grid does not have sufficient resolution. On the other hand, Olsen and Kjellesvig (1998) reported a computational time of 2 months for one case with a fine grid. If grid size is chosen too fine, the computational time could be too long for practical engineering purposes. One of the goals of the present study is to investigate the resolution of the grid and its relation to the accuracy of the results, and get experience for the most appropriate number of grid cells to use and the accuracy this involves. Experience from many other CFD studies shows that the inaccuracies in coarse grids are due to false diffusion. Instead of using more cells, it may be possible to use higher-order discretization schemes. This may be more effective in reducing the total computational time. The use of different high-order schemes is also an interesting topic in that will be investigated in the present study.

One of the main uncertainties in practical use of the simple empirical scour prediction formulas are the complexity of the geometry and flow field around the structure. The formulas were made for uniform flow over a flat bed, and in natural river the bed is not flat. Also, curves in the river and other complex shapes can lead to a highly complex non-uniform flow around the bridge pier. An advantage with a CFD model is that it is possible to compute a longer part of the river with in the grid. Then the complex shape of the bed and the corresponding flow field can be computed. However, a problem is that the grid resolution for the flow around the bridge pier can be different from the grid of the main river. To predict the local scour, a fairly fine grid is required. It may not be possible to use a grid with such fine resolution for the whole river, which may be modelled with a coarser grid. Different solutions to this problem has to be investigated in the present study. One option is to use a nested grid. Two grids can be placed inside one another, as shown in Fig. 1 a. Here, a coarse grid models the main river and a fine grid models the scour around the bridge pier. Algorithms are used to transfer variable values such as velocities and possible sediment concentrations from the coarse grid to the boundaries of the fine grid. Another strategy is to use a local grid refinement in the coarse grid. This is shown in Fig. 1 b.

Finally, it is interesting to look at the accuracy of the water flow module and the turbulence on the results. Investigations of turbulent diffusion in steady uniform flow (Miller, 1971) showed that the assumption of an isotropic eddy-viscosity is not valid. The most used turbulence model, the k-epsilon model, uses this assumption. Miller's (1971) experiments revealed that the diffusion in one direction could be nine times higher than in another direction. Preliminary studies with very simple non-isotropic turbulence models show that improved results are obtained for bed movements in meandering channels (Rüther, 2003, personal communication). More advanced turbulence models will therefore be tested in the present study (Naot and Rodi, 1982; Speziale, 1987). The results from sub-project 3 using Large Eddy Simulations will be used in the evaluation of the different models.

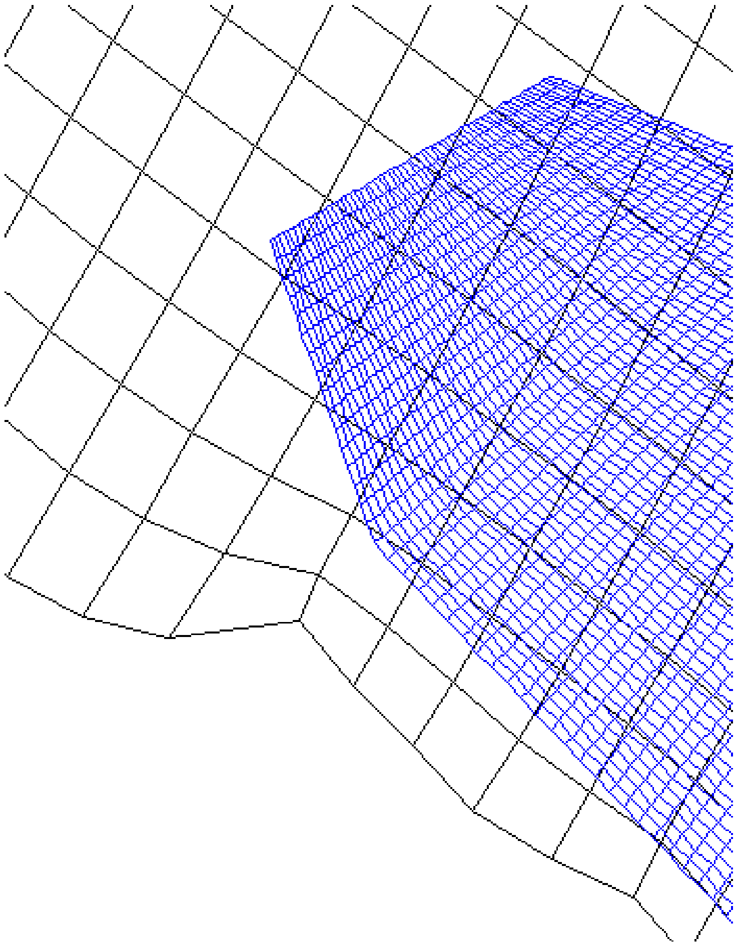
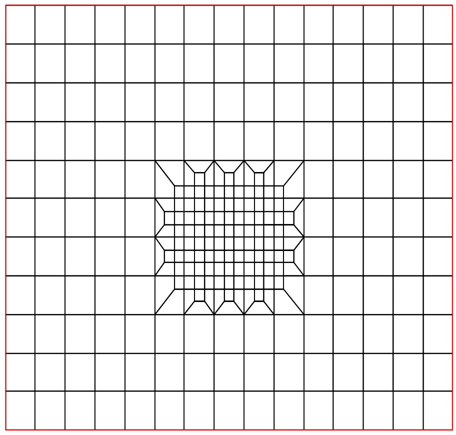


Figure 1a (left) *Detail of a nested grid from a computation of pollutant spill in Lake Tyrfjorden (Olsen and Tjomsland, 1998). The circulation in the lake was modelled with the coarse grid, and the dispersion of the pollutant was modelled in the fine grid.*

Figure 1b (under) *Local grid refinement using quadrilateral cells.*



This part of the project will be carried out by the doctoral student, in cooperation with the project manager.

3. Large eddy simulation (LES) of boundary layer flow with large roughness

The flow in an open channel, especially near a rough channel bed, will be investigated with the help of Large-Eddy Simulations (LES). The main objective is to get further insight into the physical mechanisms for incipient sediment transport. The project will enhance the general understanding of the turbulent flow processes near rough channel beds in order to enlighten the physical mechanisms of the bed destabilization processes and to improve existing numerical modelling tools for the computation of sediment transport. This will be done by gaining insight into the velocity and turbulence profiles in the roughness region at the wall. Experience in CFD modelling of sediment transport has shown that there are large uncertainties in modelling flow with large roughness.

The numerical method of Large-Eddy-Simulation (LES) to be used here is especially suitable for this project (see e.g. Stoesser et al., 2003). With LES the large, energetic vortex structures, strongly depending on the geometric boundary conditions, are directly computed. The smaller, dissipative vortices, which normally do not depend on the geometry, are accounted for in a subgrid scale model. In comparison to statistical models, i.e. the temporally averaged RANS-Model, LES offers the advantage, that the unsteady three dimensional flow field and consequently locally prevailing pressure fields, frictional forces and roughness induced stresses are directly computed in dependence on the local flow field.

Mean statistical data are difficult to interpret with regard to the dynamic flow field at channel beds since sediment transport processes strongly depend on the instantaneous flow field. In this respect, coherent turbulent structures play a significant role. So far, there are unsatisfactory quantitative or even qualitative statements on their influence on particle movement in the transition area between the free-surface flow and the subsurface pore flow of the channel.

In the framework of this proposal the turbulent flow over an immobile natural channel bed is calculated by means of LES with a high spatial and temporal resolution. This is accomplished for the configuration shown in Figure 2, that has been already investigated by Aberle (2005) in an experimental study. The channel bed consists of a well developed armoured layer of natural gravel. The roughness height of the gravel bed is approximately 90mm and various water depths are available (ranging from 150 mm to 500 mm) in order to validate the LES code for these flow configurations. The objective is to analyse the vortex structures as well as the unsteady forces on the discrete grains of the channel bed. The numerical simulations produce a full spatial and temporal picture of the flow. The following project aims are to be pursued:

1. Advancement of a numerical program, to be intended for further research demands in the area of numerical modelling of sediment transport in open channels over rough beds. The data of the LES will be evaluated with special regard on instantaneous bed shear stresses, flow velocities, turbulent kinetic energy and pressure forces. The physical mechanisms which lead to the destabilization of the bed material can be investigated.

2. Improvement of statistical numerical model approaches for the prediction near bed turbulent flow as well as bed shear stresses and incipient sediment transport for practical CFD models.

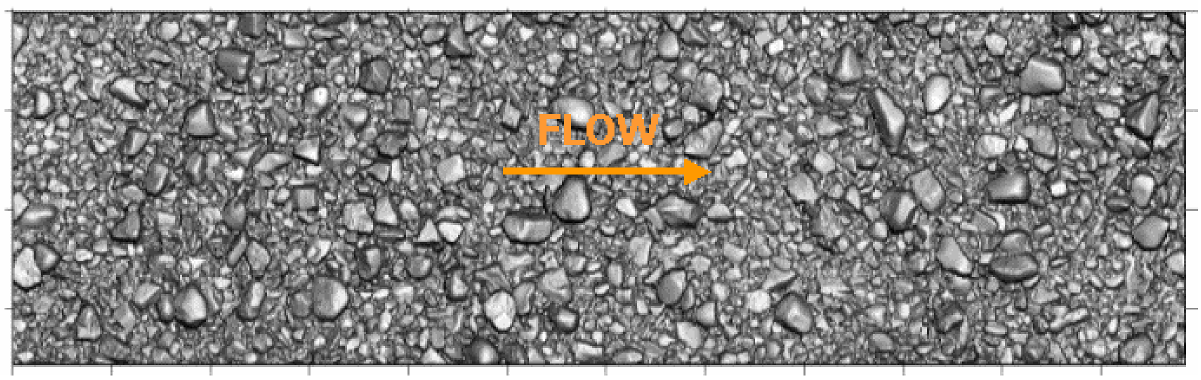


Figure 2: *Experimental setup of the flow over a well developed gravel bed by Aberle (2005)*

This project will be carried out by a post-doctoral researcher in cooperation with Dr. Thorsten Stösser, who is working with LES in the research group of Prof. Wolfgang Rodi in Karlsruhe, Germany. Dr. Stösser will work on the project for shorter time periods as a visiting scientist.

4. Publications and knowledge transfer

The leading journals in the world in this field of engineering is Journal of Hydraulic Engineering and Journal of Hydraulic Research. The results from the project will be published in these journals, in addition to presentations at international conferences. The project leader has previous experience in publishing for these journals, and he has several publications from the two previous times he has received funding from the Norwegian Research Council. The project leader is currently associate editor of the Journal of Hydraulic Engineering and has detailed knowledge about the publication process and the requirements to get a paper published in this journal. The topic of local scour has received a large amount of attention in this field of science, and new publications expanding the state-of-the art will be well received by these leading journals. Results will also be presented at international conferences on hydraulic engineering.

The results from the project will also be published on our web pages. Our research group on CFD has currently published a large number of research results from our previous projects on our web pages (www.ntnu.no/~nilsol/cfd).

The knowledge generated by the project will also be transferred to the students of hydraulic engineering at our department. This will be done by including results from the projects in the courses we teach on Hydromechanics and Hydroinformatics. Students will also be involved in the project as a part of their project and MSc thesis work. This will both benefit the project and be an effective method to transfer scientific knowledge to the students.

Research leadership

The proposed project leader, Nils Reidar Bøe Olsen, finished his PhD in 1991 on the topic of CFD modelling of sediment transport. He then worked for six years as a researcher in the River Section of the Norwegian Hydrotechnical Laboratory, a part of the SINTEF group. He was then project leader for several research projects on 3D CFD modelling of various hydraulic phenomena: Trap efficiency of sand traps, bed movements in sand traps, turbidity currents, local scour, spillways and prediction of coefficients of discharge, head losses in tunnel constrictions etc. The projects lead to several international publications.

In the years 1997-1999, Olsen worked on a post-doc project funded by the Research Council of Norway. The project was very successful, and a large number of publications were made. More information about this project is on the web page: www.bygg.ntnu.no/~nilsol/gren.

In the year 2000, Olsen got a permanent position at NTNU. He then started a research group on CFD modelling of hydraulic engineering, which he is currently leading. The group is one of the most active in research at the river hydraulics section of the Department of Water and Environmental engineering at NTNU. In 2004, our section was evaluated by the Norwegian Research Council in a Research Assessment Exercise. The section got the highest score, 5 (Excellent), in all three evaluation categories. Only three of all engineering research groups in Norway obtained a similarly high score. The evaluation committee stated: *"This is an excellent group with world-class quality that is well recognised internationally. Good publication culture, publication rate, including many journal papers while also being responsive to industry. They develop models to understand the basics such as: CFD, physical simulation in laboratory and in the field, eco-hydrology, river engineering/erosion and sediment transport, cold climate issues, climate change and hydrological modelling - a variety of high quality products"*. Looking at the five last years from 2000-2004, Olsen has produced 40 % of all the refereed international journal papers of the section. Most of the papers are in the two leading journals in the world in his field: Journal of Hydraulic Engineering and Journal of Hydraulic Research.

National and international cooperation

In the current project, Dr. Thorsten Stösser of the University of Karlsruhe, Germany, will take part in the project on LES modelling. There will be a close cooperation between him and with the post-doctoral researcher working on Large Eddy Simulation on boundary layer flow with rough surfaces. Dr. Stösser is part of Prof. Wolfgang Rodi's group working on CFD modelling, and this group is one of the leading research groups on the world in this topic. The project will also work together with Bundesanstalt für Wasserbau in Germany and get data from local scour cases from their large laboratory. Here in Norway we will work on the new project of the Department of Geography in Trondheim and the Norwegian Water and Energy Directorate, where we will get data from scour holes in local rivers around Trondheim.

Budget

The notation ‘ is used for 1000 NOK in the text below. The allocations for the guest researcher, post-doctoral researcher and doctoral researcher follow the funding guidelines of the research council.

In addition, there is some funding for travel expenses for the visiting researcher (10’ pr. year) and for travel to two conferences (2*20’) pr year. There is also some allocation for project administration.

Summing up the allocation for the different sub-projects:

1. Doctoral study on local scour: 3 years: 3 * 574’ = 1 722’
2. Post-doctoral project on LES: 2 years: 2 * 676’ = 1 352’
3. Visiting researcher, LES project: 4 years: 4 * 41’ = 164‘
4. Travel, administration etc. : 4 years: 4 * 80’ = 320’

Total: 3.558 mill NOK

The project will require substantial computer resources. The supercomputing facilities in Norway will be used, both in Trondheim and in other parts of the country where available. Applications will be made through the NOTUR organization, where the resources are funded by the Norwegian Research Council.

Conclusion

The present project will advance the state-of-the-art research on numerical modelling of erosion and sediment transport currently conducted at NTNU. This will have benefits to our society in the future in form of better engineering solutions to hydraulic problems related to prevention of damages from extreme floods.

References

Aberle, J. (2005). Personal Communication.

Brethour, J. M. (2002) “Transient 3-D Model for lifting, transporting and depositing solid material”, Hydroinformatics -2002, Cardiff, UK.

Dargahi, B. (1990) “Controlling mechanism for local scouring”, ASCE Journal of Hydraulic Engineering, Vol. 116, No. 10, Oct., pp. 1197-1214.

Duan, J.G., Wang, S.S.Y. and Jia, Y.F. (2001) “The applications of the enhanced CCHE2D model to study the alluvial channel migration processes”, IAHR Journal of Hydraulic Research, No. 5.

Fang, H.W. and Rodi, W. (2003) “Three-dimensional calculations of flow and suspended sediment transport in the neighbourhood of the dam for the Three Gorges Project (TGP) reservoir in the Yangtze River”, IAHR Journal of Hydraulic Research, No. 4.

Khan, A. A., Cadavid, R., and Wang, S. S. Y. (2000) “Simulation of channel confluence and bifurcation using the CCHE2D model” Proceedings of the Institution of Civil Engineers-Water, no. 2.

Lagasse, P. F. (2003) “Summary of 1998 Scanning Review of European Practice for Bridge Scour and Stream Instability Countermeasures”, US Department of Transportation, Federal Highway Administration, USA.

Miller, A. C. (1971) “Turbulent diffusion and longitudinal dispersion measurements in a hydrodynamically rough open channel flow”, PhD Thesis, Colorado State University, USA.

Morvan, H., Pender, G., Wright, N.G. and Ervine, D.A. (2002) “Three-dimensional hydrodynamics of meandering compound channels“, ASCE Journal of Hydraulic Engineering, No. 7.

Naot, D. and Rodi, W. (1982) “Calculation of secondary currents in channel flow”, ASCE Journal of Hydraulic Engineering, No. 8, pp. 948-968.

Olsen, N. R. B. (2003) "3D CFD Modelling of a Self-Forming Meandering Channel", ASCE Journal of Hydraulic Engineering, No. 5, May.

Olsen, N. R. B. and Tjomsland, T. (1998) "3D CFD modelling of wind-induced currents and radioactive tracer movements in a lake", 3rd. International Conference on Hydroscience and Engineering, Cottbus, Germany.

Olsen, N. R. B. and Kjellesvig, H. M. (1998) "Three-dimensional numerical flow modelling for estimation of maximum local scour depth", IAHR Journal of Hydraulic Research, No. 4.

Olsen, N. R. B. and Stokseth, S. (1995) "Three-dimensional numerical modeling of water flow in a river with large bed roughness", IAHR Journal of Hydraulic Research, Vol. 33, No. 4.

The Research Council of Norway (2004) “Evaluation of research in engineering sciences in Norway”.

Speziale, C. G. (1987) “On nonlinear K-l and k-e models of turbulence”, Journal of Fluid Mechanics, vol. 178, pp. 459-475.

Stoesser, T., Fröhlich J. and Rodi, W. (2003): "Identification of Coherent Flow Structures in Open-Channel Flow over Rough Bed Using Large-Eddy Simulation". XXX IAHR Congress, Thessaloniki, Greece.

Sæterbø, E, (2005) Personal communication.

Wu, W. M., Rodi, W. and Wenka, T. (2000) “3D numerical modeling of flow and sediment transport in open channels”, ASCE Journal of Hydraulic Engineering, Vol. 126, No. 1.