**CGU HS Committee on River Ice Processes and the Environment**
13th Workshop on the Hydraulics of Ice Covered Rivers
*Hanover, NH, September 15-16, 2005*

# Investigation of Ice Block Stability – Numerical Modeling Issues

**Karen E. Dow, Peter Steffler and Faye Hicks**
*Department of Civil and Environmental Engineering*
*University of Alberta, Edmonton AB, T6G 2W2*
*kdow@ualberta.ca ; peter.steffler@ualberta.ca ; faye.hicks@ualberta.ca*

The hydrodynamic forces acting on individual ice floes are examined using a three-dimensional computational fluid dynamics package. Understanding the complex fluid dynamics is of relevance to the physics of ice cover development and ice jam formation and release. The objective of this study is to examine the steady state stability of floating ice blocks that have come to rest against an intact ice cover using 3-D $k$-$\varepsilon$ modelling. Preliminary results on the modeling efforts are presented with focus on a sensitivity analysis of the inlet development length, inflow boundary condition and shape of the leading edge of the ice cover on the pressure distribution beneath the ice cover.

It was found that the longer the development length the greater the pressure reduction at the leading edge of the ice cover. The results suggested that the magnitude of the surface velocity may be important to the resulting force acting on the ice cover. The sensitivity analysis of inflow velocity profile further corroborated this finding as three different velocity profiles with the same surface velocity, different average velocity, produced similar pressure distributions beneath the ice cover.

The effects of shape of the leading edge were examined by comparing model results for a sharp rectangular edge to those for a beveled edge. The bevel affected the separation zone and the local pressure reduction at the leading edge of the ice cover creating a smaller force on the ice cover. Future work includes validating the numerical results, using the numerical model as an investigational tool, and using the numerical results to plan an experimental study.

## 1. Introduction

The transport and accumulation of ice is one of the more complicated problems in river ice hydraulics, because of the complex fluid dynamics surrounding individual ice floes. It is of relevance to the physics of ice cover development and ice jam formation, but is of particular in the context of ice jam release. Jasek (2003) notes that when large ice floes are transported downstream under an ice jam past its toe, then it is likely that these floes will be propelled upwards, impacting the underside of the intact solid ice cover. He suggests that such occurrences have the potential to crack and weaken the restraining ice cover, initiating the open leads which are believed to play an important role in the occurrence of ice jam release (Jasek, 2003). If true, specific knowledge of the hydrodynamic forces acting on individual ice floes will be an important component of any model which attempts to predict the occurrence of ice jam release events.

In the practical context of this problem, there are a number of component phenomena to consider. For example, the initial question is whether or not discrete ice floes approaching an ice jam accumulation from upstream will contribute to lengthening, or will be entrained in the flow and transported beneath the ice jam. For the latter case, the further question is whether or not the entrained floe will be transported all the way past the ice jam toe, to be in a position to rise under the intact restraining ice cover.

At present, much of our knowledge of these processes is necessarily qualitative, due to the inherent logistical difficulties and safety issues which arise when trying to measure dynamic ice processes in the field. This is particularly difficult for ice floe transport under ice jams. As a consequence, we must rely in large part on experimental and numerical work to further understand the mechanics of ice floe entrainment and transport, and that is the purpose of this investigation. Here we discuss some preliminary results of the first phase of this investigation, in which we focus on the issue of ice floe entrainment at the leading edge. Current theory and observations (e.g. Beltaos, 1995) suggest that the leading edge of an ice jam accumulation behaves as a narrow jam, with floe entrainment or juxtapositioning being the dominant local processes. Healy and Hicks (2001) observed this same tendency near the leading edge of ice jams forming in a laboratory flume.

Numerous studies have already been conducted to examine this problem of ice block stability. Early investigations of this phenomenon focused on defining the critical approach velocity or critical densimetric Froude number (based on approach flow velocity and block thickness) at which floating ice blocks at leading edges of intact ice covers are submerged (Pariset and Hausser, 1961; Ashton, 1974; Uzuner and Kennedy, 1972; Larsen, 1975). Daly and Axelson (1990) examined the problem analytically and determined that instability was reached when the underturning moment exceeded the righting moment. Coutermarsh and McGilvary (1991, 1993, 1994) attempted to measure the two dimensional pressure distribution along the bottom surface of a floating block and found both positive (stabilizing) and negative (destabilizing) pressures acted on the block, with a characteristic saddle shape in the pressure distribution. They observed that changes in flow velocity had little effect on the pressure distribution but primarily changed the pressure magnitude. More recently Hara *et al.* (1996) and Kawai *et al.* (1997) conducted a series of experiments investigating the movement of ice floes at the edge of an ice cover characterizing the movement and the critical densimetric Froude number at movement. They focused on the shape of the edge of the ice cover and the thickness of the ice block.

With recent advances in numerical and experimental technology that allow for better flow visualization and determination, more information about the mechanics of the problem can now be now realized. The objective of this first phase of our study is to examine the steady state stability of floating ice blocks that have come to rest against an intact ice cover using a three-dimensional (3-D) computational fluid dynamics package. We seek to increase our knowledge of the stability behaviour of floating ice floes and the hydrodynamic forces that act upon them. The results of this study will be used to assist in the design of an experimental study, and will hopefully ultimately prove valuable to the advancement of discrete particle models of river ice. At this stage, preliminary results of the numerical modeling effort are presented with a focus on the effects of the assumed inlet velocity profile and the shape of the leading edge of the ice cover on the pressure distribution under the ice cover.

## 2. Dimensional Analysis

When an ice floe comes to rest against a floating obstacle, it can remain in place or it can become submerged. Buoyancy is the resisting force against motion, while the submerging forces are due to the pressure reductions under the ice floe caused by flow separation and acceleration. The significant variables in this analysis are (Figure 1):

- the mean flow velocity under the block, $V_u$;
- the depth of the approach flow, $H$;
- the block length L, block thickness, $t$;
- the block width b, block density, $\rho'$;
- the fluid density, $\rho$; and
- the acceleration due to gravity, $g$,

such that,

$$V_u = f(b, t, L, \rho', \rho, g, H) \qquad [1]$$

Dimensional analysis leads to the following relationship:

$$\frac{V_u}{\sqrt{gt}} = f\left(\frac{b}{L}, \frac{t}{H}, \frac{\rho'}{\rho}, \frac{t}{L}\right) \qquad [2]$$

which is the same result as in Beltaos (1995).

However, most authors have expressed their results in terms of a densimetric Froude number which would be in the form of:

$$F = \frac{V_u}{\sqrt{\left(\frac{\rho - \rho'}{\rho}\right) gt}} = f\left(\frac{b}{L}, \frac{t}{H}, \frac{t}{L}\right) \qquad [3]$$

## 3. Model Implementation

### 3.1 Modeling Platform

A three dimensional (3-D) computational fluid dynamics package, ANSYS CFX-5, was used to investigate the steady state stability of ice blocks resting against an ice cover. CFX-5 is based on the finite volume technique which solves the Navier Stokes equations in their conservation form. For steady state, inviscid flow the equations of motion presented in the CFX-5 Solver Theory Manual in differential form reduce to:

$$\nabla \cdot \left(\rho \vec{U} \otimes \vec{U}\right) = \nabla \cdot \left(-\rho\delta\right) + S_M \qquad [4]$$

where:

$\nabla = \left(\frac{\partial}{\partial x}, \frac{\partial}{\partial y}, \frac{\partial}{\partial z}\right)$

$\vec{U}$ = velocity vector
$\rho$ = density
$\otimes$ = tensor product
$\delta$ = Kronecker delta or identity matrix
$S_M$ = momentum source

Turbulence models are necessary in CFD simulations to enable the effects of turbulence to be predicted without requiring a prohibitively fine mesh and computing power that does not yet exist. There are many turbulence models available in CFX-5 and can be divided into eddy viscosity models, Reynolds stress models, Large Eddy Simulation (LES) and Detached Eddy Simulation (DES). The turbulence models equations presented here are described in the CFX-5 Solver Theory Manual while general description can be found in the CFX-5 Solver Modelling Manual. Within the eddy viscosity models, the *k*-*ε* model is considered the industry standard turbulence model as it is good for many engineering flows. The term *k* is the turbulence kinetic energy and ε is the turbulence eddy dissipation. This model introduces two variables to the system of equations, specifically the effective viscosity $\mu_{eff}$ and the modified pressure $p'$ as:

$$\mu_{eff} = \mu + \mu_t \qquad [5]$$

$$p' = p + \frac{2}{3}\rho k \qquad [6]$$

where $\mu_t$ is the turbulence viscosity which in the *k-ε* model is assumed to be linked to the turbulence kinetic energy and dissipation via the relationship:

$$\mu_t = C_\mu \rho \frac{k^2}{\varepsilon} \qquad [7]$$

where $C_\mu$ is a constant equal to 0.09 (CFX-5 Solver Theory Manual).

The Large Eddy Simulation model is for transient large scale fluctuating flows. It filters the velocity field so that it contains only the large scale components of the total field as it is the large scale motions that are more energetic and effective transporters. LES filters the equations of movement and decomposes the flow variables into a large scale (resolved) and a small scale (unresolved) parts. The LES model is used primarily for research purposes and is not practical because of the fine grid and time step requirements. It must be run in a transient mode and gives detail on the structure of turbulent flow such as pressure fluctuations that would not be obtained from a Reynolds Averaged Navier Stokes formulation.

Boundary conditions in CFX-5 can be modeled as an inlet, outlet, opening (fluid can simultaneously flow both in and out of the domain), wall, or symmetry plane. The most robust boundary configuration is to specify a velocity / mass flow at an inlet with a static pressure at an outlet (CFX-5 Solver Modelling Manual).

### 3.2 Test Case for Ice Block Stability

A key component of any numerical modeling study is validation data, and at present the most comprehensive validation data found in the literature are the detailed pressure measurements by Coutermarsh and McGilvary (1991,1993,1994). Although at this time, these data are not yet available to us, our preliminary test cases were modeled after that study in the hope that ultimately a verification comparison might be conducted. Their experiments were conducted in a warm flume with cross section of 0.91 by 0.91 m and length of 7.32 m with a variable pumping capacity of 0.3 $m^3$/s. A hollow Plexiglas square ice floe 61.6 by 61.6 cm and thickness of 7.62 cm with 91 pressure taps on the bottom surface was fastened to the flume to hold the block rigid through the test. The upstream end of the block was fixed with a threaded rod that allowed the block angle of attack to be varied. The *t/H* ratio was varied by changing the water depth as the thickness of the ice floe remained constant.

For this preliminary study component, simulations were limited to the following test cases:

- angle of attack of 0°,
- *t/H* of 0.1, and
- approach flow mean velocity, *V*, of 0.45 m/s.

in order to conduct detailed sensitivity analyses of the following parameters:

- inflow boundary conditions (i.e. assumed velocity profile)
- shape of the leading edge of the ice floe;
- location of outlet boundary
- use of a symmetry plane
- domain setup

### 3.3 Model Implementation

#### Geometry

Figure 2 illustrates the geometry of the modeled domain, which was created to match that of the apparatus used by Coutermarsh and McGilvary (1994), as discussed earlier. The pressure measurements made by Coutermarsh and McGilvary (1994) verified that the pressure distributions were always symmetrical about the centerline of the ice floe and initial runs of our numerical model also confirmed this. Therefore, a symmetrical boundary condition was used at the centerline of the flume (x = 0.455 m), to minimize computational memory and processing requirements.

The regions of fluid flow and/or heat transfer in CFX-5 are called domains. Fluid domains define regions of fluid flow while solid domains are regions occupied by conducting solids. As there was no need to model thermal heat transfer between the ice and the water and the ice remains stationary, the ice floe and ice cover were modeled as cavities rather than as solid domains. By modeling as a cavity, the physical properties of the ice are not modeled (density, heat transfer, etc.). This saves on computational time as there is no computational mesh over the ice cover. The specific gravity of ice was assumed to be 0.92 and was used to determine the vertical displacement of the ice.

As this simulation was steady state and the ice floe remained stationary there is little advantage to modeling the ice floe and ice cover separately. Rather the ice floe and ice cover were modeled as one continuous piece of 1 meter length. This simplified the meshing procedure and geometry setup and had the advantage of effectively modeling an ice block of infinite length.

#### Computational Mesh

A key aspect of developing a meaningful 3-D numerical model for this study was the determination of an appropriate computational mesh. Ideally, a fine mesh throughout the computational domain would be best, but this is impractical both in terms of the memory requirements and the length of time required to conduct a 3-D simulation. Consequently, a more practical overall meshing strategy is to have a relatively coarse mesh in areas where the solution does not changing rapidly and a finer mesh in areas with large gradients in velocity or pressure. In this context then the computational mesh development is an iterative process which can be automated in the CFX-5 software using a built-in feature known as *"mesh adaptation"*. Figure 3 illustrates an example of a cross section (YZ plane) of the mesh at the edge of the ice block both before and after mesh adaptation.

Normally, for optimal results, the mesh should be refined to a point at which a grid independent solution is reached. In other words, to a point at which the solution no longer changes with further mesh refinement. However, this case provided an interesting situation because of the sharp leading edge on the ice cover. It was found that if the mesh became too refined, the steady state solution would no longer converge, and oscillations would result both in the residuals (solution error) and in the solution itself. Figure 4 illustrates an example of this. It is suspected that this behaviour is evidence of either vortex shedding beneath the ice cover, or pressure oscillations rebounding from the outlet. These possibilities will be investigated further at a later time using transient large eddy simulation (LES). However, at this preliminary stage, the mesh was refined as far as possible with the additional criterion of achieving a converged steady state solution. In addition, the effect of smoothing the leading edge of the ice was examined.

**Boundary Conditions**

For this preliminary investigation, the flume walls and bed as well as the ice underside were modeled as smooth, no slip, walls while the open water surface was modeled as a free slip surface. The outlet boundary condition was specified as an average static pressure of 0 Pa. This is an average over the whole outlet which is the most commonly used option. The average constraint is applied by comparing the area weighted pressure average over the outlet to the user specified value of 0 Pa. A sensitivity analysis was carried out to ensure that the location of the outlet was not affecting the solution in the region of interest. The length of the ice cover was varied from 1 to 5 m. It was found that an ice cover of 1 m yielded similar results to longer ice covers, which gave confidence that the location of the outlet was not adversely affecting the pressure distribution beneath the ice cover.

The inlet boundary condition was a specified as Cartesian velocity components with a medium turbulence intensity of 5%, which is the recommended option if there is no information available about the inlet turbulence. This preliminary study was primarily focused on conducting a detailed sensitivity analysis examining the effects of the inlet velocity profile shape and inlet boundary location (flow development length) on the pressure distribution and force calculation under the ice floe.

## 4. Model Results

### 4.1 Sensitivity Analysis on Inlet Development Length and Inflow Boundary Condition

A sensitivity analysis was performed on the inlet boundary conditions by systematically changing the location of the inlet and the inlet velocity profile to see the effect on the resulting pressure distribution beneath the ice cover. In order to examine solution sensitivity to the location of the inlet, the distance from the inlet boundary to the upstream edge of the ice cover was varied from 0 to 50 meters holding all other variables constant. The inlet velocity profile was examined by comparing results for uniform, power law, and log-law velocity profile distributions while keeping all other variables constant. A one-seventh power law profile was prescribed as:

$$u = u_{\max}\left(\frac{z}{H}\right)^{1/7} \qquad [8]$$

where $u_{\max}$ is the surface velocity.

The log-law velocity profile was prescribed as:

$$u = u_* \left( \frac{1}{\kappa} \ln \frac{z u_*}{\nu} + 5.0 \right) \qquad [9]$$

where $\kappa$ = von Karman's constant (0.41)
$u_*$ = shear velocity
$\nu$ = viscosity

**Effect of Varying Development Length**

In this first series of tests, the inlet velocity profile was set to be a uniform velocity of 0.45 m/s with a medium turbulence intensity of 5%. The distance from the inlet boundary to the upstream edge of the ice cover was varied as 0, 5, 10, 20, 30, 40 or 50 m. The outlet boundary condition was set as 0 Pa, the *k-ε* turbulence modelling option was selected, and all walls were modeled as smooth walls. Figure 5 presents the results of this sensitivity analysis, illustrating the centerline pressure profiles along the ice cover. As the figure illustrates, these tend to have a similar shape regardless of development length. However, as the development length increased the magnitude of the pressure drop increased. This suggests the importance of the surface velocity on the magnitude of the pressure reduction at the leading edge of the ice cover. The velocity profile becomes increasingly developed from the uniform profile as the development length increases. This would lead to a higher surface velocity as the average velocity would be kept constant at 0.45 m/s. So one would expect that the surface velocity with a development length of 50 m to be higher than the surface velocity with a development length of 5 m.

**Effect of Varying Inlet Boundary Condition**

Three different inlet velocity profiles were tested, specifically: uniform, power law and log-law profiles. In this case, the results were compared based on a 1m development length (chosen to minimize computational requirements). Only the inlet velocity profile was changed, all other variables were kept constant. The outlet boundary condition was set as 0 Pa, the *k- ε* turbulence modelling option was used, all walls were modeled as smooth walls, and a medium intensity turbulence inlet boundary condition was chosen. As the results of the development length sensitivity analysis indicated that the surface velocity was important in the pressure reduction at the leading edge of the ice cover, the surface velocity of all three profiles was set to 0.45 m/s which gave an average velocity of 0.45 m/s, 0.41 m/s and 0.39 m/s for the uniform, power and log law profiles, respectively.

Figure 6 presents the results of this sensitivity analysis, illustrating the centerline pressure distributions. The figure shows that the greatest pressure reduction is for the uniform velocity profile and the least is for the power law profile. However, the difference between the three profiles is rather small compared to the development length investigation. To further investigate the surface velocity effect a one seventh power law profile with an average velocity of 0.45 m/s (surface velocity of 0.5156 m/s) was tested and is also shown in Figure 6. The pressure reduction is significantly larger for this profile than the other three.

### 4.2 Sensitivity Analysis on Shape of the Leading Edge of the Ice Cover

As discussed earlier, assuming a sharp leading edge on the ice cover creates a strong separation and recirculation zone that is difficult for the numerical model to resolve, particularly in a steady state simulation. To explore this effect further, additional simulations were conducted in which the leading edge of the ice cover was beveled (as illustrated in Figure 7). In this case, results were compared to that for the sharp leading edge, based on a 5 m development length (chosen to minimize computational requirements).

The centerline pressure results are compared in Figure 8, where it is seen that the pressure distribution obtained for the beveled leading edge is significantly different than obtained for a rectangular leading edge. The negative peak pressure near the leading edge is much larger in magnitude, occurs further downstream (just after the edge of the bevel) and recovers more rapidly, as compared to the behaviour for the sharp leading edge case. This increased magnitude of the negative peak pressure actually seems to go against intuition, as one might expect the beveled edge to produce a smaller pressure drop (as the flow separation would not be as strong). The most likely explanation for this is that the results for the sharp leading edge case are not realistic. Clearly then, this behaviour cannot be resolved properly using this steady state simulation. Further analyses, employing the transient large eddy simulation available in CFX-5 are planned to investigate this further.

This limitation, however, has little effect on the overall force calculation. The peak negative pressure occurs over a small area so it has little effect on the overall force acting on the ice cover. Thus, even though the peak negative pressure for the beveled edge is greater, the magnitude of the calculated vertical force acting on the ice cover for the sharp leading edge is actually larger (-6.13 N) than for the beveled edge( -3.73 N). It is the overall distribution of the pressure which is important.

## 5. Summary and Conclusions

Preliminary results of a numerical study on the steady state stability of floating ice blocks that have come to rest against an intact ice cover have been presented. This knowledge of the hydrodynamic forces that act on individual ice floes is crucial to the prediction of ice jam release events as much of the current knowledge of these processes is necessarily qualitative. A 3-D computational fluid dynamics package, ANSYS CFX-5, was used for this study. The logistics of setting up the model were discussed including meshing, boundary conditions, and physical model selection. Sensitivity analyses on the inlet development length and inflow boundary conditions were conducted.

The sensitivity analysis showed that as the development length increased, the velocity profile became more developed and the pressure reduction at the leading edge of the ice cover was increased. This suggested the importance of the surface velocity on the pressure reduction. The inflow boundary condition (velocity profile) sensitivity analysis showed that three different profiles with the same surface velocity (different average velocities) produced a similar pressure reduction on the ice cover. This is important not just for a numerical study but also for an experimental study in a shorter flume. If the velocity profile is not fully developed before reaching the region of interest the results may not be accurate. Also when modeling experimental results it may be important to use the actual velocity profile from the experiment as an inlet boundary condition rather than assuming a fully developed profile. A good practice would be for experimenters to measure the velocity profile in the flume experiment.

The effect of beveling the leading edge of the ice cover was a reduction in the overall force acting on the ice cover. This suggests, as other researchers have found, that the local pressure reduction caused by the separation zone is important. Most current ice models assume a sharp leading edge on the underside of the ice cover, which may not be realistic in all circumstances. The effect of different leading edge shapes will be investigated further in the next phase of this study.

Plans for further numerical investigations also include switching the simulation to a transient large eddy simulation which will allow for more detailed modeling of the vortex shedding at the leading edge of the ice cover. This can then be used to determine whether it is the instantaneous pressure distribution due to vortex shedding or the overall average pressure acting on the block that is important in determining whether the ice block will underturn. The final step of the numerical modeling will attempt to model the actual block movement under the ice cover. This will involve a transient simulation with moving mesh techniques to track the movement of the block with the appropriate hydrodynamic forces.

In the future, the detailed pressure measurements of Coutermarsh and McGilvary (1994) could provide excellent validation data for these numerical results. From this the model can be used as a testing device to investigate a wider range of variables (such as block geometry and flow characteristics) in much less time than would be feasible with an experimental model. The results of this testing will be further used to design an experimental program employing particle image velocimetry to obtain detailed velocity field measurements, and digital imaging techniques to track the movement of ice blocks at the leading edge.

## Acknowledgments

This research is supported by scholarships to the first author from the Natural Sciences and Engineering Research Council of Canada (NSERC) and Alberta Ingenuity, and through NSERC research grants to the other two authors. This support is gratefully acknowledged.

## References

Ashton, G.D., 1974. Froude criterion for ice-block stability. Journal of Glaciology, 13(68), pp. 307-313.

Beltaos, S. (1995). River Ice Jams. Water Resources Publications, LLC, Highlands Ranch, Colorado, 372 pp.

CFX-5 Solver Modelling Manual

CFX-5 Solver Theory Manual

Coutermarsh, B. & McGilvary, W.R., 1991. Pressure distribution of an underturning ice floe. Proc. 6th Workshop on River Ice, pp. 143-163.

Coutermarsh, B.A. and McGilvary, W.R., 1993. Static analysis of floating ice block stability. Journal of Hydraulic Research, 31(2), pp. 147-160.

Coutermarsh, B and McGilvary, R., 1994. Analyzing the stability of floating ice floes. CRREL Report 94-13, 19 pp.

Daly, S.F. and Axelson, K.D., 1990. Stability of floating and submerged blocks. Journal of Hydraulic Research, 28(6), pp. 737-752.

Hara, F., Kawai, T., Hanada, M., Nishihata, A., and Saeki, H., 1996. Movement of ice blocks under an ice cover. Proceedings of IAHR Ice Symposium, Beijing, pp. 769-778.

Healy, D. and Hicks, F. (2001) "Experimental Study of Ice Jam Shoving" Proc. 11th Workshop on River Ice, Ottawa, May, pp. 126-144.

Jasek, M., 2003. Ice jam release and break-up front propagation. Proceedings of the 12th workshop on the hydraulics of ice covered rivers, The CGU HS Committee on River Ice Processes and the Environment. June 19-20, 2003, Edmonton, AB, pp. 348-368.

Kawai, T., Hara, F., Masaki, S., Nishihata, A., and Saeki, H., 1997. Experimental study on the process of ice jam development. Proceedings of 9th CRIPE, pp. 245-256.

Larsen, P., 1975. Notes on the stability of floating ice blocks. Proceedings of the IAHR third international symposium on ice problems. Hanover, NH, IAHR, pp. 305-313.

Pariset, E. and Hausser, R., 1961. Formation and evolution of ice covers on rivers. Transactions of the Engineering Institute of Canada, 5(1), pp. 41-49.

Uzuner, M.S. and Kennedy, J.F., 1972. Stability of floating ice blocks. Journal of the Hydraulics Division, 98 (HY12), pp. 2117-2133.

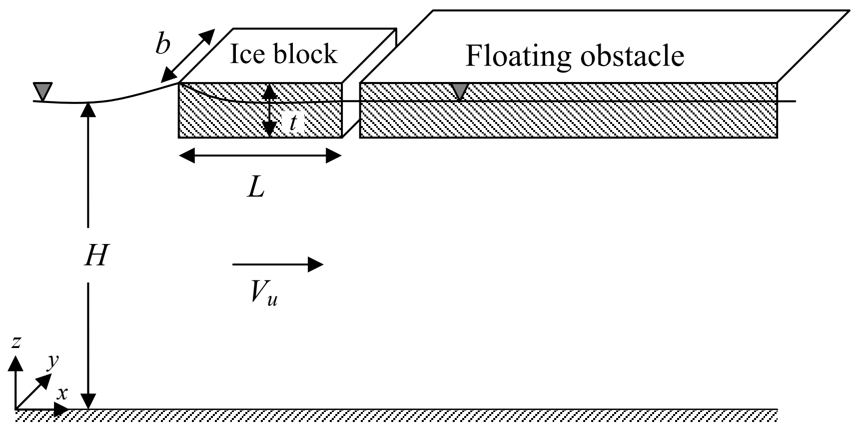


Figure 1. Defining sketch for ice floe stability problem.

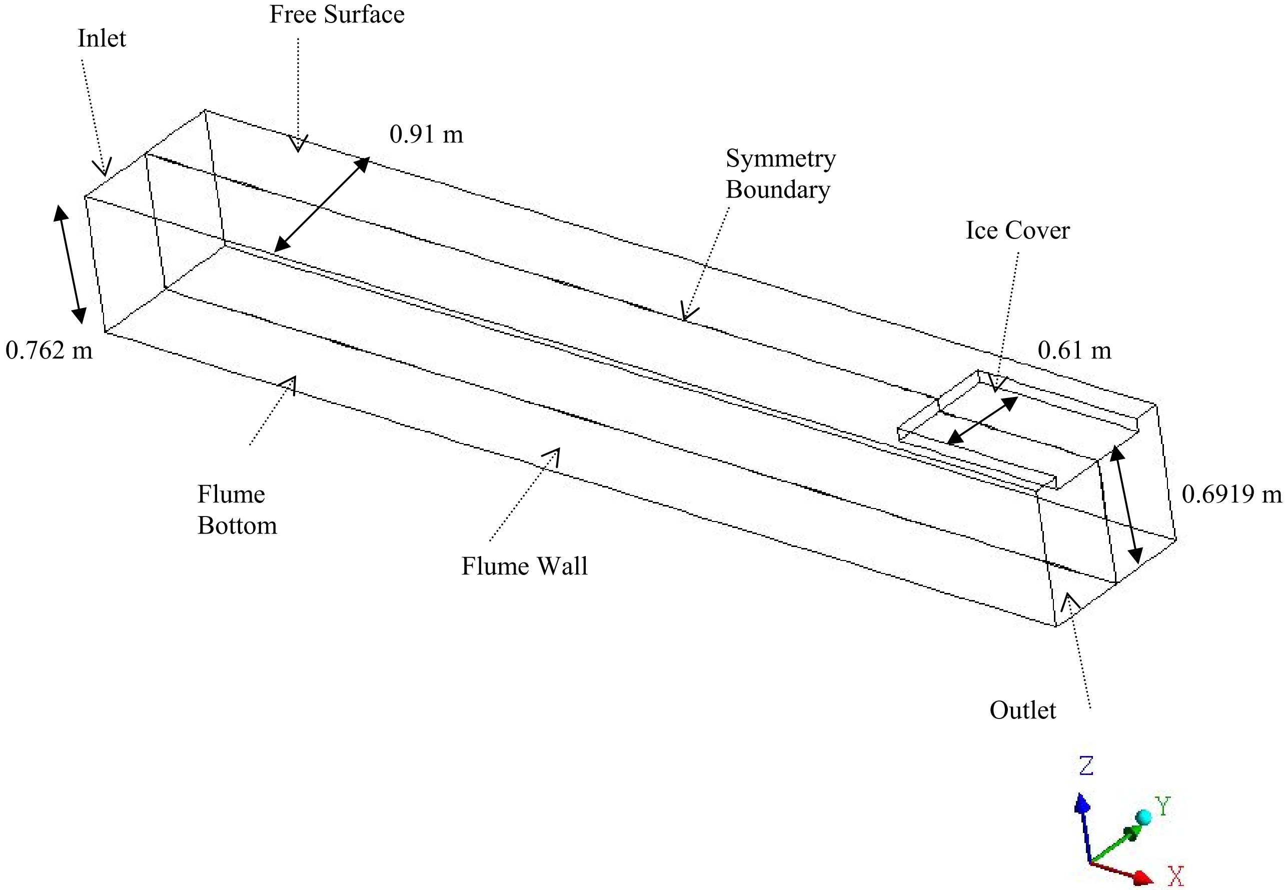


Figure 2: Model geometry for preliminary simulations.

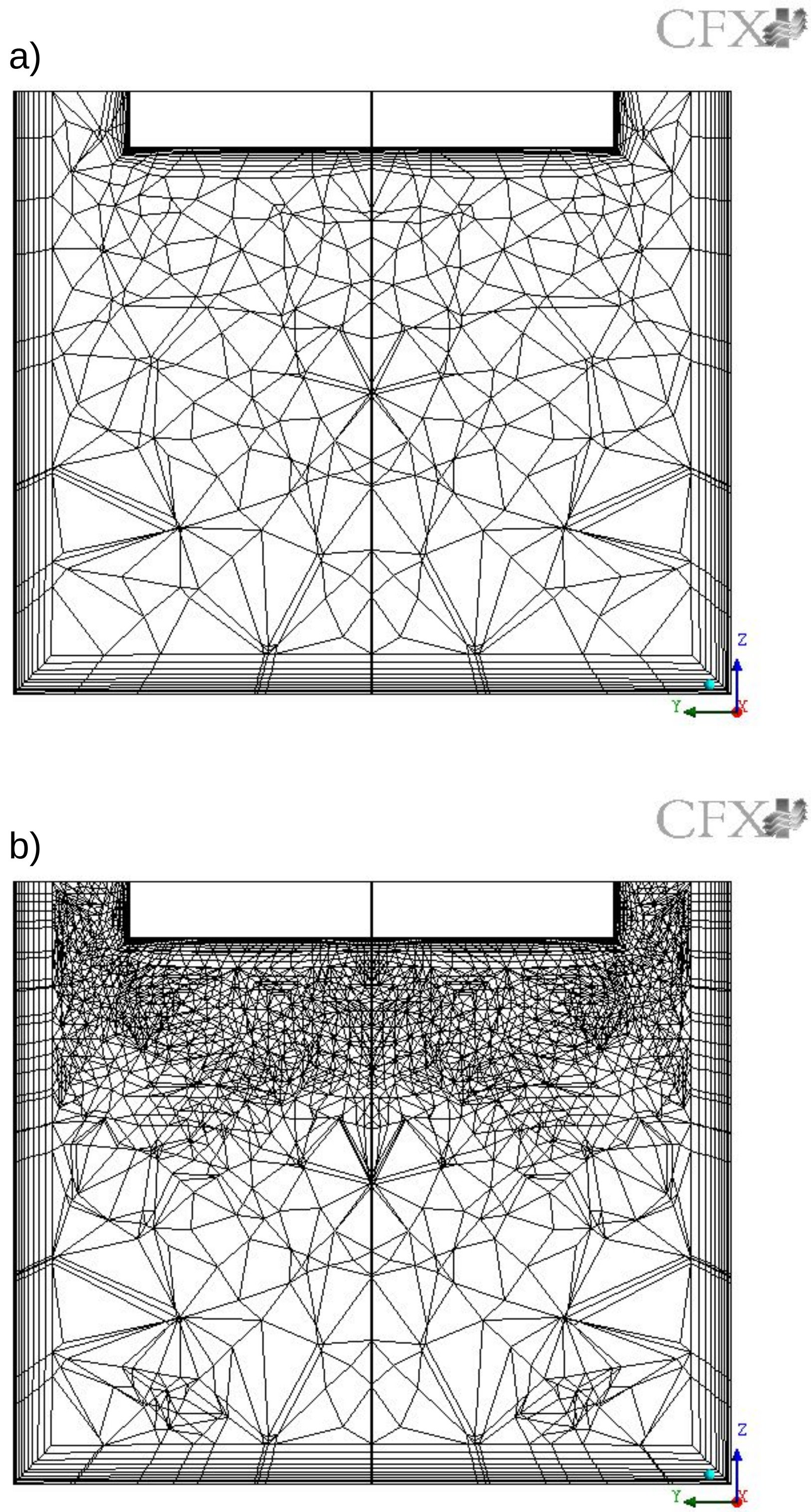


Figure 3. YZ plane at edge of ice cover a) before mesh adaption, and b) after mesh adaption.

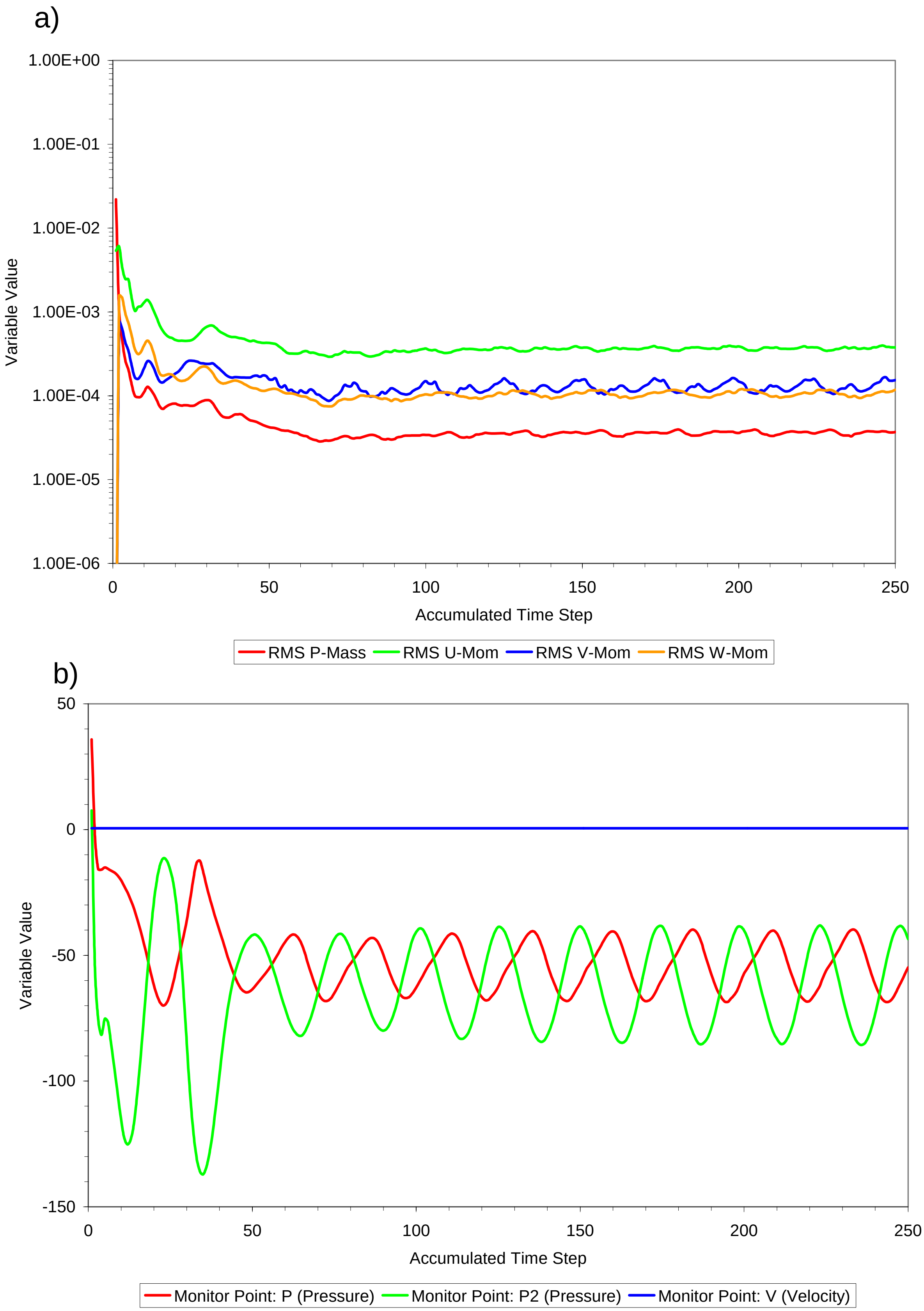


Figure 4. Refined mesh solution a) residual plot and b) monitor points on ice cover for pressure.

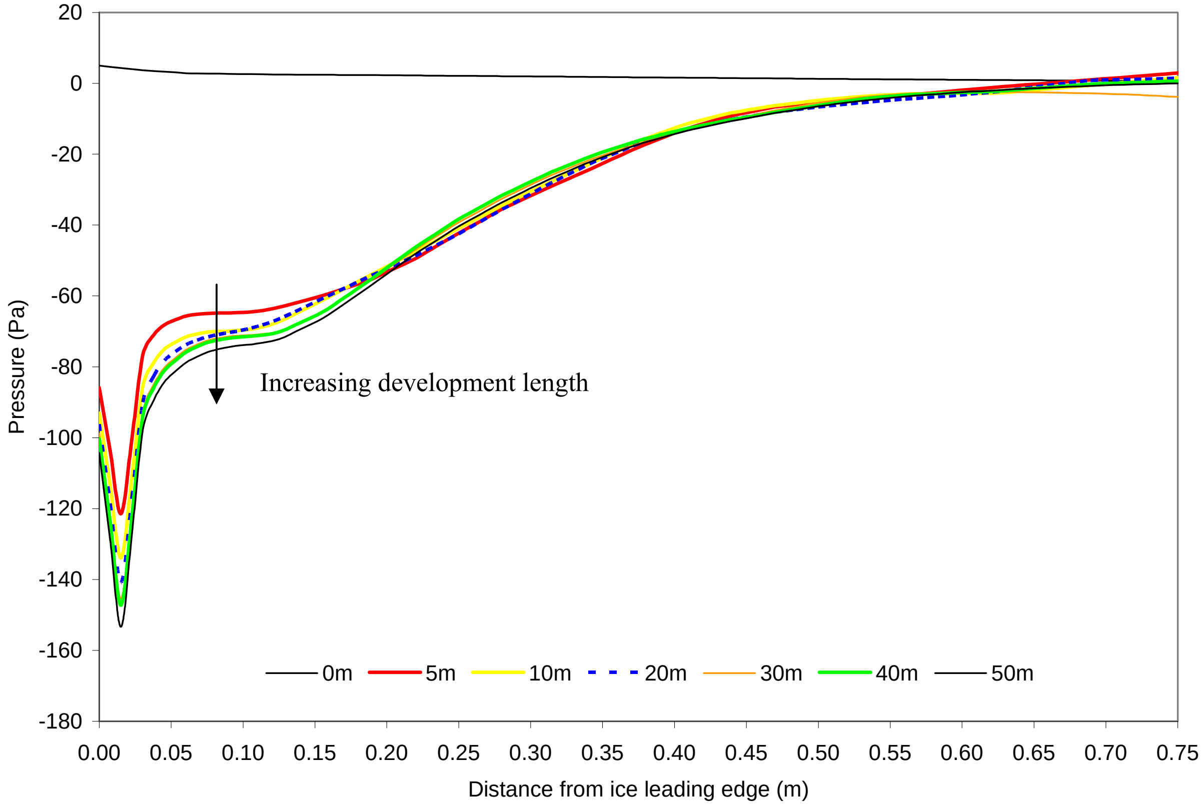


Figure 5. Sensitivity of model results to variation in flow development length.

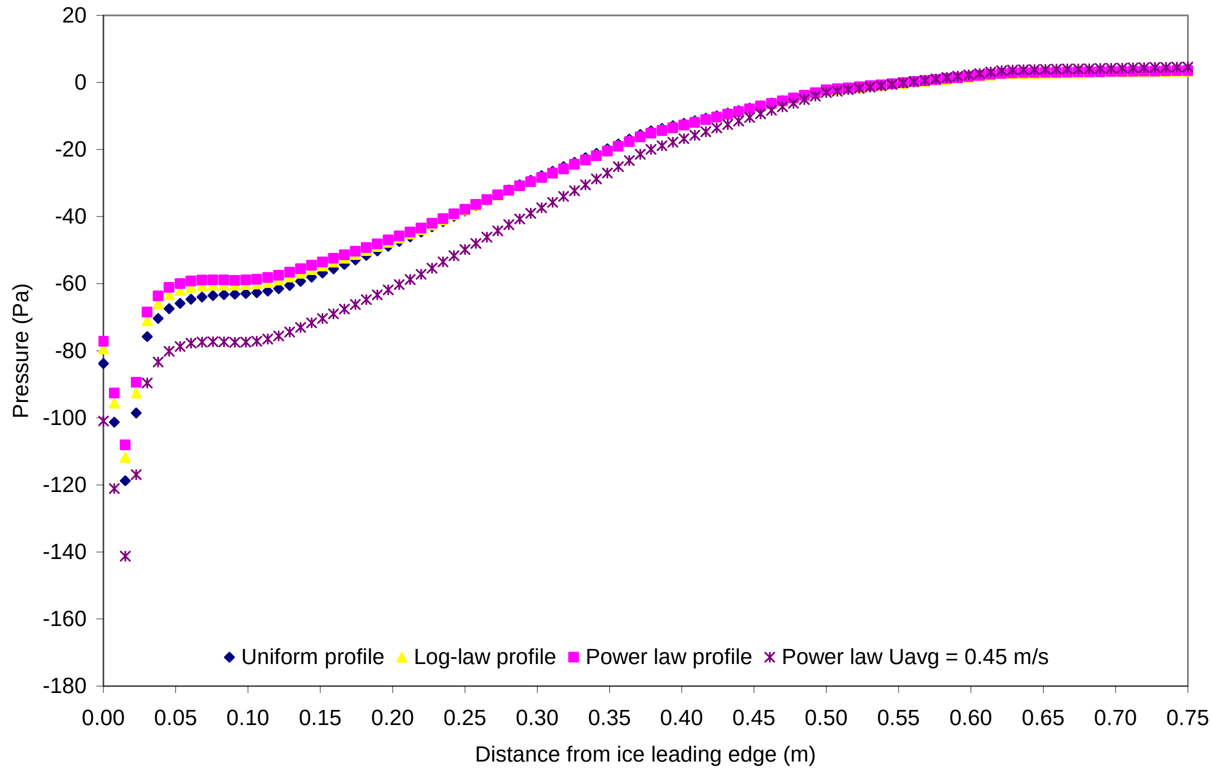


Figure 6: Sensitivity of model results to variation in inlet velocity profile.

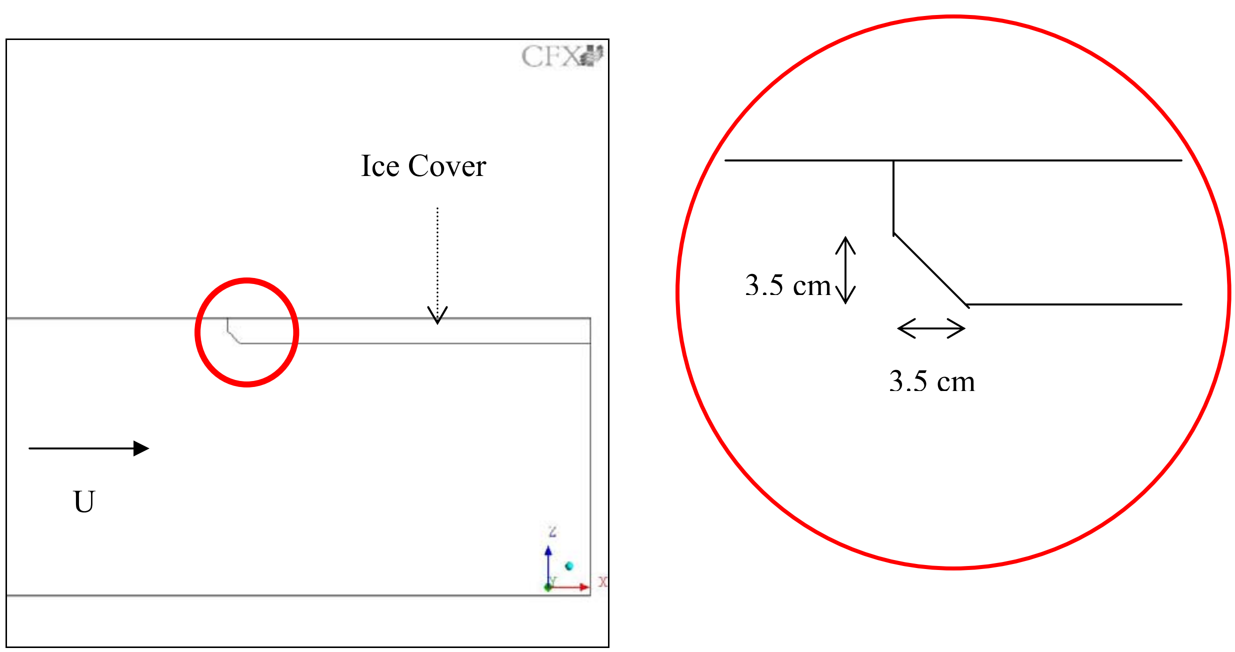


Figure 7: Model configuration for testing a beveled leading edge on the ice cover.

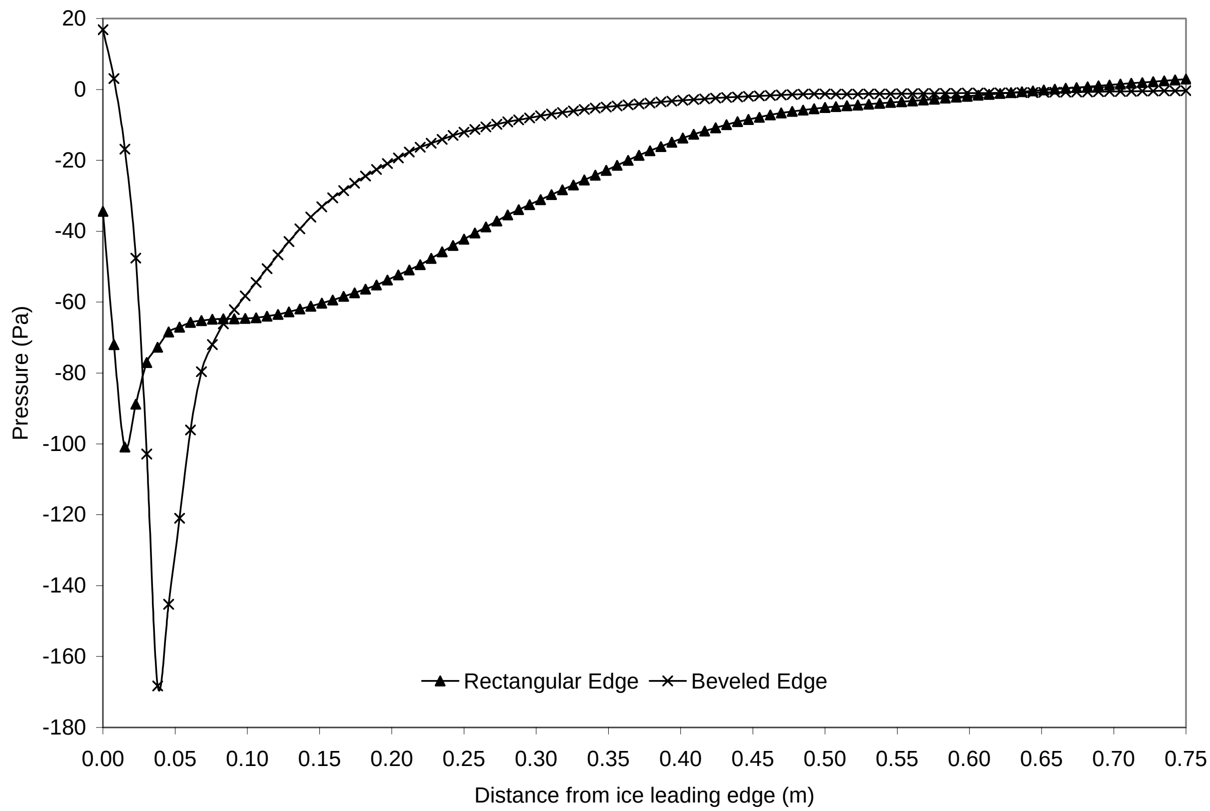


Figure 8: Effect of ice leading edge shape on computed under ice pressure distribution.