Proceedings of the ASME 2013 International Mechanical Engineering Congress and Exposition
IMECE2013
November 15-21, 2013, San Diego, California, USA

IMECE2013-62278

THREE-DIMENSIONAL MESH MORPHING METHODOLOGY FOR SCOURING AROUND BRIDGE PIERS BASED ON COMPUTATIONAL FLUID DYNAMIC SOLUTION

Chris Edwards
Northern Illinois University
DeKalb, IL, U.S.A.

Steven A. Lottes
Argonne National Laboratory
Argonne, IL, U.S.A.

Pradip Majumdar
Northern Illinois University
DeKalb, IL, U.S.A.

Flow scour is the engineering term used to describe the erosion of a sediment bed due to fluid flow. Local scour occurs around objects placed in the path of flow, such as bridge piers and abutments. Severe damage or even failure of structures may occur if the amount of scour is too great. Due to the complexity of the fluid/structure interactions and cost of experiments, Computation Fluid Dynamics (CFD) methods are under development to predict the shape and depth of a scour hole. This study extends a previous 3-D iterative methodology, with several improvements to the scouring physics models, implemented in the commercial CFD software STAR-CCM+ to predict the scour hole formation around circular bridge piers. These improvements are inclusion of a variable critical shear stress (VCSS) for the initiation of motion of bed sediment, scouring normal to the sediment bed, and a sand slide model. Reynolds Averaged Navier-Stokes (RANS) equations and a k-ε turbulence model are used to resolve the flow field. The methodology uses a single phase implicit unsteady approach to obtain sediment bed shear stress values. Two moving boundary relations are employed to model the erosion and sand slide physics. One for the erosion rate is based upon an empirical correlation for critical shear stress combined with a sediment entrainment function of Van Rijn, and the other uses the slope of the sediment bed, to iteratively displace the sediment bed in a way that decreases slope as long as it exceeds the angle of repose of the sediment. This is accomplished by a user defined function to move the sediment bed at each time step and the mesh morphing procedure built into STAR-CCM+ to solve fluid-structure interaction problems to stretch the existing mesh to maintain cell quality throughout the flow domain as the bed is displaced. Simulation results have been compared to experimental data found in literature. It was found that simulations over predict the maximum scour depth by up to 35%, but show a large improvement in capturing the overall shape of the scour hole in comparison to models that do not include a sand slide model.

INTRODUCTION

Scour of riverbed material away from bridge foundations during floods is a major problem that accounts for about sixty percent of bridge failures. Of the 600,000 bridges in the National Bridge Registry in the U.S. about 26,000 (1 in 20) are classified as scour critical. Over the past 30 years approximately 20 bridges per year on average fail due to scour (one every twenty days on average). Many more fail in major flood years and far fewer in years without major floods. Federal guidelines for evaluating scour risk are highly conservative by design to ensure public safety and use simple engineering correlations and related methods in evaluations.

Three-dimensional computational fluid dynamics (CFD) methods are being developed to include much more fundamental physics in scour analysis. Argonne National Laboratory's Transportation Research and Analysis computing Center (TRACC) [1], Northern Illinois University, and the Federal Highway Administration's Turner-Fairbank Highway Research Center [2] have been collaborative partners in this effort. The analysis methods developed have been for highly benchmarked commercial CFD software that may offer the reliability required for improved analysis of scour in engineering design of bridge foundations and placement in and over rivers. Earlier efforts included Biswas [3] who developed an automated approach to computing hydrodynamics and then displacing the bed and adjusting the volume mesh for the analysis of pressure flow scour under flooded bridge decks using STAR-CD CFD software. Bhaskar [4] extended the work of Biswas from 2D to 3D and used the STAR-CCM+ preprocessor auto-meshing capability to generate a better quality mesh after the bed was displaced due to scour. Elapolu

Proceedings of the ASME 2013 International Mechanical Engineering Congress and Exposition
IMECE2013
November 15-21, 2013, San Diego, California, USA

IMECE2013-62278

THREE-DIMENSIONAL MESH MORPHING METHODOLOGY FOR SCOURING AROUND BRIDGE PIERS BASED ON COMPUTATIONAL FLUID DYNAMIC SOLUTION

Chris Edwards
Northern Illinois University
DeKalb, IL, U.S.A.

Steven A. Lottes
Argonne National Laboratory
Argonne, IL, U.S.A.

Pradip Majumdar
Northern Illinois University
DeKalb, IL, U.S.A.

Flow scour is the engineering term used to describe the erosion of a sediment bed due to fluid flow. Local scour occurs around objects placed in the path of flow, such as bridge piers and abutments. Severe damage or even failure of structures may occur if the amount of scour is too great. Due to the complexity of the fluid/structure interactions and cost of experiments, Computation Fluid Dynamics (CFD) methods are under development to predict the shape and depth of a scour hole. This study extends a previous 3-D iterative methodology, with several improvements to the scouring physics models, implemented in the commercial CFD software STAR-CCM+ to predict the scour hole formation around circular bridge piers. These improvements are inclusion of a variable critical shear stress (VCSS) for the initiation of motion of bed sediment, scouring normal to the sediment bed, and a sand slide model. Reynolds Averaged Navier-Stokes (RANS) equations and a k-ε turbulence model are used to resolve the flow field. The methodology uses a single phase implicit unsteady approach to obtain sediment bed shear stress values. Two moving boundary relations are employed to model the erosion and sand slide physics. One for the erosion rate is based upon an empirical correlation for critical shear stress combined with a sediment entrainment function of Van Rijn, and the other uses the slope of the sediment bed, to iteratively displace the sediment bed in a way that decreases slope as long as it exceeds the angle of repose of the sediment. This is accomplished by a user defined function to move the sediment bed at each time step and the mesh morphing procedure built into STAR-CCM+ to solve fluid-structure interaction problems to stretch the existing mesh to maintain cell quality throughout the flow domain as the bed is displaced. Simulation results have been compared to experimental data found in literature. It was found that simulations over predict the maximum scour depth by up to 35%, but show a large improvement in capturing the overall shape of the scour hole in comparison to models that do not include a sand slide model.

INTRODUCTION

Scour of riverbed material away from bridge foundations during floods is a major problem that accounts for about sixty percent of bridge failures. Of the 600,000 bridges in the National Bridge Registry in the U.S. about 26,000 (1 in 20) are classified as scour critical. Over the past 30 years approximately 20 bridges per year on average fail due to scour (one every twenty days on average). Many more fail in major flood years and far fewer in years without major floods. Federal guidelines for evaluating scour risk are highly conservative by design to ensure public safety and use simple engineering correlations and related methods in evaluations.

Three-dimensional computational fluid dynamics (CFD) methods are being developed to include much more fundamental physics in scour analysis. Argonne National Laboratory's Transportation Research and Analysis computing Center (TRACC) [1], Northern Illinois University, and the Federal Highway Administration's Turner-Fairbank Highway Research Center [2] have been collaborative partners in this effort. The analysis methods developed have been for highly benchmarked commercial CFD software that may offer the reliability required for improved analysis of scour in engineering design of bridge foundations and placement in and over rivers. Earlier efforts included Biswas [3] who developed an automated approach to computing hydrodynamics and then displacing the bed and adjusting the volume mesh for the analysis of pressure flow scour under flooded bridge decks using STAR-CD CFD software. Bhaskar [4] extended the work of Biswas from 2D to 3D and used the STAR-CCM+ preprocessor auto-meshing capability to generate a better quality mesh after the bed was displaced due to scour. Elapolu et. al. [5] developed a method that uses the mesh morphing capabilities in STAR-CCM+, developed for fluid structure interaction problems, to morph the mesh to maintain volume mesh quality when the riverbed is displaced non-uniformly during the scour erosion process. This model used a sediment pickup function of Van Rijn [6] to determine erosion rate at a point on the bed. The goals for these previous efforts were to model scour using better methods to handle mesh motion resulting from the moving bed boundary as erosion occurred. The physics of the erosion models were limited in complexity in order to get the mechanisms in place for cycling through the flow solver, bed displacement, grid morphing, and periodic remeshing. Once the moving mesh methodology was working, it provided the framework for developing and testing more detailed and accurate scour erosion models. This work adds three enhancements to the physics models that greatly improve results of the simulations in achieving a bowl shaped scour hole around a cylindrical pier that is close to that observed in experiments. Roulund et.al. [11] present an excellent review of research on flow and scour around circular piers with reference to other review papers. Readers interested in the background of pier scour modeling efforts are referred to that paper.

NOMENCLATURE

$\mathbf{A_b}$	m^2	Bed cell face area vector
d_{50}	m	Median sand diameter
d_*		Non-dimensional sand grain diameter
$\mathbf{E}_r$	m/s	Vector erosion rate normal to the bed
$\mathbf{g}$	m/s^2	Gravity vector
g	m/s^2	Acceleration due to gravity
k_s	m	Effective roughness
Δs_{max}	m	Maximum displacement
Δt_{k+1}	s	Size of the next time step,
V_b	m/s	Magnitude of the erosion rate at a point on the bed
$V_{b,k,max}$	m/s	Maximum erosion rate.
α	*radians*	Bed slope
γ	*radians*	Angle between the bed surface shear stress vector and the horizontal
ν	m^2/s	Kinematic viscosity of water
θ_s		Packing density
θ_w		Porosity of the bed
ρ	kg/m^3	Density of water
ρ_s	kg/m^3	Density of sediment
$\boldsymbol{\tau}_\mathbf{b}$	Pa	Shear stress vector at a point on the bed
τ_b	Pa	Bed shear at a point
τ_c	Pa	Critical shear stress
τ_{cf}	Pa	Flat bed critical shear stress
ϕ	*radians*	Angle of repose of the sand

DESCRIPTION OF THE PHYSICAL PROBLEM AND COMPUTATIONAL MODEL

The physical problem chosen for testing is the scour around a single cylindrical pier and uses the conditions from flume experiments done at Offshore Center Danmark [7]. To save computational time, a symmetric half flume cutting through the pier as shown in Figure 1 is used. The computational domain extends from -2 m to 2 m in the primary flow, or X-direction, -0.6 m to 0 m in the Z-direction and 0 m to 0.17 m in the vertical, or Y-direction. A fixed, solid cylinder is included in the computational domain and is cut along the symmetry plane in the same manner as the flume. The cylinder has a radius of 0.1m in the XZ-plane, has a vertical height of 0.47m, and runs from the top of the flume and ends 0.3m below the initial sediment bed height. The cylinder serves mostly as a marking post to visually inspect the amount of scour in relation to water depth and is meshed with a minimum number of cells. The domain is divided into two regions along the height of the cylinder in order to ensure that mesh morphing occurs only on the bottom region. This divided domain saves computational time in mesh morphing operations and prevents a morphing problem that generated negative volume cells at the upper boundary surface.

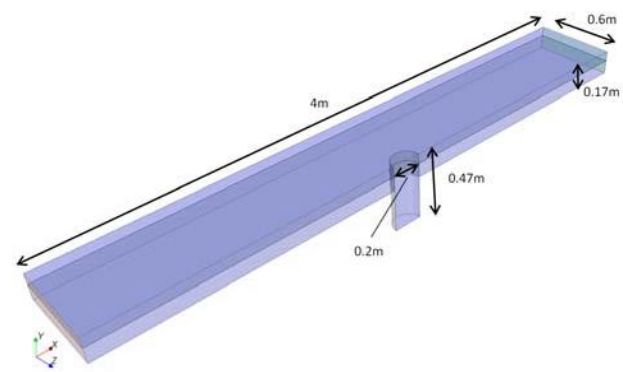

Figure 1: Schematic of modeled section of symmetric half of scour flume with cylindrical pier obstruction.

Boundary Conditions

The inlet is treated as a uniform velocity inlet. The flume side wall and half cylinder pier are taken to be smooth walls. The outlet is specified as a free flow zero gradient outlet. To limit the model to single phase, the free surface is modeled as a rigid lid with a flat surface with a symmetry boundary condition. This condition is close to the laboratory condition except for some variation of the free surface height in the immediate vicinity of the pier that does not greatly affect the bed shear computation and is therefore adequate for the present study. A symmetry condition is also applied at the flume centerline. The sediment bed is modeled as a rough wall with an effective roughness, k_s, of twice the sand diameter, d_{50} of 0.25 mm, or 0.5 mm.

et. al. [5] developed a method that uses the mesh morphing capabilities in STAR-CCM+, developed for fluid structure interaction problems, to morph the mesh to maintain volume mesh quality when the riverbed is displaced non-uniformly during the scour erosion process. This model used a sediment pickup function of Van Rijn [6] to determine erosion rate at a point on the bed. The goals for these previous efforts were to model scour using better methods to handle mesh motion resulting from the moving bed boundary as erosion occurred. The physics of the erosion models were limited in complexity in order to get the mechanisms in place for cycling through the flow solver, bed displacement, grid morphing, and periodic remeshing. Once the moving mesh methodology was working, it provided the framework for developing and testing more detailed and accurate scour erosion models. This work adds three enhancements to the physics models that greatly improve results of the simulations in achieving a bowl shaped scour hole around a cylindrical pier that is close to that observed in experiments. Roulund et.al. [11] present an excellent review of research on flow and scour around circular piers with reference to other review papers. Readers interested in the background of pier scour modeling efforts are referred to that paper.

NOMENCLATURE

$\mathbf{A_b}$	m^2	Bed cell face area vector
d_{50}	*m*	Median sand diameter
d_*		Non-dimensional sand grain diameter
$\mathbf{E}_r$	*m/s*	Vector erosion rate normal to the bed
$\mathbf{g}$	m/s^2	Gravity vector
g	m/s^2	Acceleration due to gravity
k_s	*m*	Effective roughness
Δs_{max}	*m*	Maximum displacement
Δt_{k+1}	*s*	Size of the next time step,
V_b	*m/s*	Magnitude of the erosion rate at a point on the bed
$V_{b,k,max}$	*m/s*	Maximum erosion rate.
α	*radians*	Bed slope
γ	*radians*	Angle between the bed surface shear stress vector and the horizontal
ν	m^2/s	Kinematic viscosity of water
θ_s		Packing density
θ_w		Porosity of the bed
ρ	kg/m^3	Density of water
ρ_s	kg/m^3	Density of sediment
$\boldsymbol{\tau}_\mathbf{b}$	*Pa*	Shear stress vector at a point on the bed
τ_b	*Pa*	Bed shear at a point
τ_c	*Pa*	Critical shear stress
τ_{cf}	*Pa*	Flat bed critical shear stress
ϕ	*radians*	Angle of repose of the sand

DESCRIPTION OF THE PHYSICAL PROBLEM AND COMPUTATIONAL MODEL

The physical problem chosen for testing is the scour around a single cylindrical pier and uses the conditions from flume experiments done at Offshore Center Danmark [7]. To save computational time, a symmetric half flume cutting through the pier as shown in Figure 1 is used. The computational domain extends from -2 m to 2 m in the primary flow, or X-direction, -0.6 m to 0 m in the Z-direction and 0 m to 0.17 m in the vertical, or Y-direction. A fixed, solid cylinder is included in the computational domain and is cut along the symmetry plane in the same manner as the flume. The cylinder has a radius of 0.1m in the XZ-plane, has a vertical height of 0.47m, and runs from the top of the flume and ends 0.3m below the initial sediment bed height. The cylinder serves mostly as a marking post to visually inspect the amount of scour in relation to water depth and is meshed with a minimum number of cells. The domain is divided into two regions along the height of the cylinder in order to ensure that mesh morphing occurs only on the bottom region. This divided domain saves computational time in mesh morphing operations and prevents a morphing problem that generated negative volume cells at the upper boundary surface.

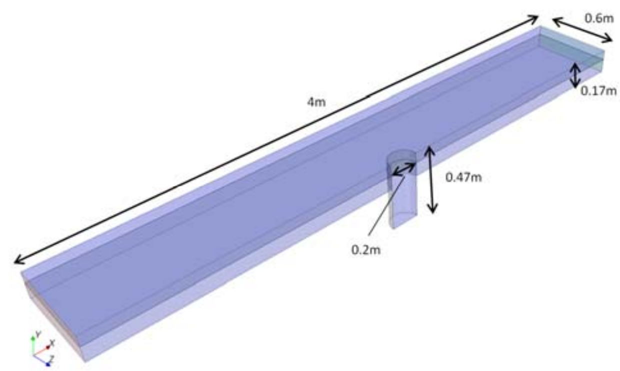

Figure 1: Schematic of modeled section of symmetric half of scour flume with cylindrical pier obstruction.

Boundary Conditions

The inlet is treated as a uniform velocity inlet. The flume side wall and half cylinder pier are taken to be smooth walls. The outlet is specified as a free flow zero gradient outlet. To limit the model to single phase, the free surface is modeled as a rigid lid with a flat surface with a symmetry boundary condition. This condition is close to the laboratory condition except for some variation of the free surface height in the immediate vicinity of the pier that does not greatly affect the bed shear computation and is therefore adequate for the present study. A symmetry condition is also applied at the flume centerline. The sediment bed is modeled as a rough wall with an effective roughness, k_s, of twice the sand diameter, d_{50} of 0.25 mm, or 0.5 mm.