Pressure-Flow Scour Prediction Using Computational Fluid Dynamics with a Moving-Grid Approach

August 31, 2010

Dr. Steven Lottes

Transportation Research and Analysis Computing Center (TRACC)

Argonne National Laboratory

slottes@anl.gov

www.tracc.anl.gov

In cooperation with the U.S. Department of Transportation, Northern Illinois University and the University of Illinois

Pressure-Flow Scour Prediction using Computational Fluid Dynamics with a Moving-Grid Approach

Dr. Kornel Kerenyi
FHWA's Turner-Fairbank Highway Research Center

Prof. Junke Guo
University of Nebraska-Lincoln

Dr. Tanju Sofu
Argonne National Laboratory (TRACC)

Dr. Bushra Afzal
University of Nebraska-Lincoln

Bhaskar Tulimilli
Northern Illinois University

In cooperation with the U.S. Department of Transportation,
Northern Illinois University and the University of Illinois

Purpose

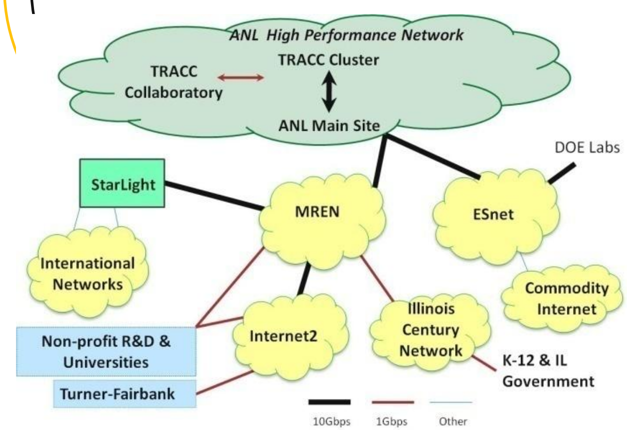
Researchers at Argonne's USDOT Transportation Research and Analysis Computing Center (TRACC), FHWA's Turner-Fairbank Highway Research Center, and universities are collaborating in evaluating CFD methods available in commercial CFD software for application to hydraulic problems of practical importance in transportation.

TRACC Is a National USDOT Supercomputing Facility

TRACC is a user facility
for transportation researchers and engineers

TRACC High Performance Compute Cluster

- 1024 cores /128 compute nodes
- 180TB Global Parallel File System Disk Storage
- 160TB Archive/Backup Tape Storage



High-bandwidth connectivity is provided via the Argonne high-performance network to world-wide research and education networks (Internet2 and ESnet)

TRACC Collaboratory - Visualization, Access Grid, and Digital Conferencing

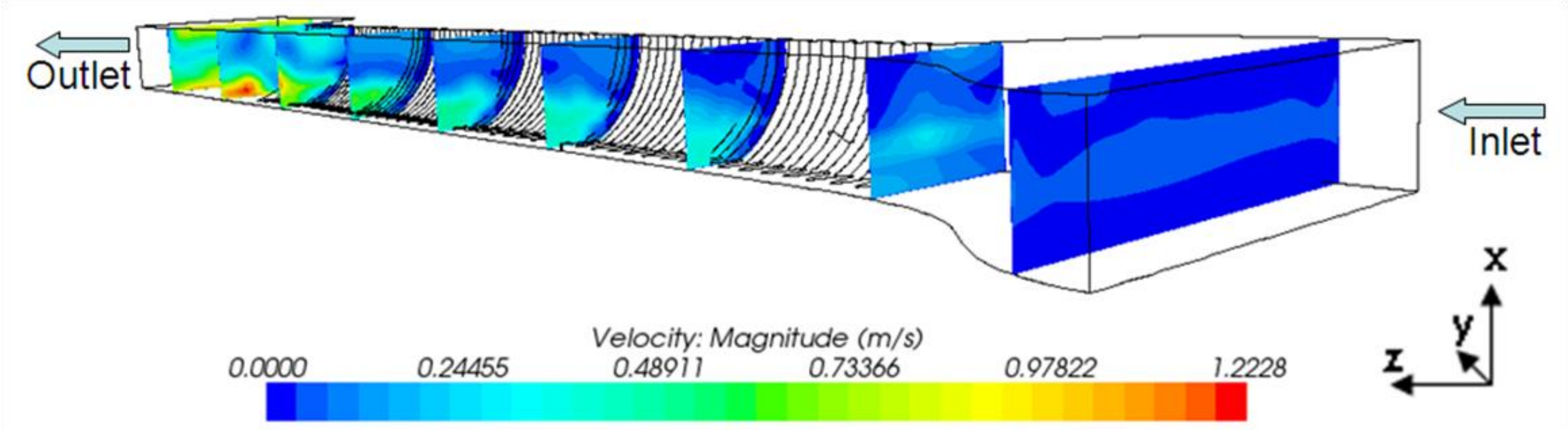
What Is CFD: Computational Fluid Dynamics

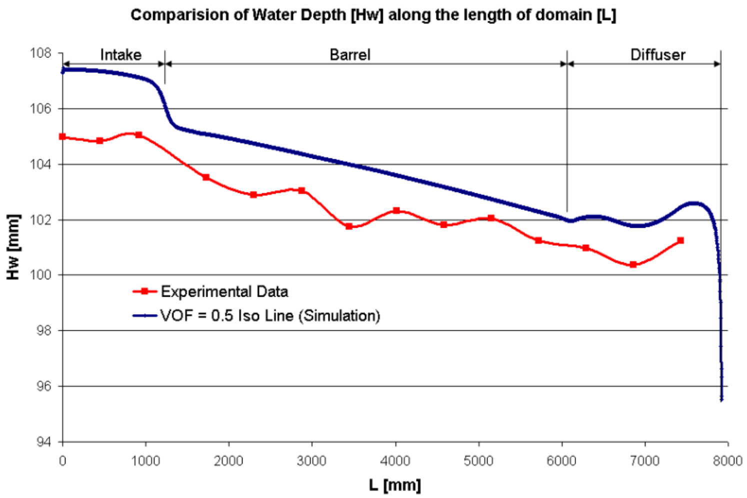
- CFD is a collection of numerical procedures for solving the governing equations of fluid flows and the processes that occur in these flows using computers.
  - Processes in the flow may include multi-phase particle transport, transport of chemical species, reaction, heat transfer, etc.
  - The primary governing equations are the conservation of mass and Newton's second law (the rate of change of momentum is equal to applied force)
  - Except for laminar flows with simple geometries and boundary conditions, the equations do not have closed form solutions and must be solved numerically
  - CFD comes as close as possible to solving the fundamental physics of a flow problem
- CFD software normally includes extensive software tools to set up problems and analyze results
  - Import or build and manipulate the problem geometry in 3-D
  - Specify boundary conditions, material properties, additional physics, and solver settings
  - Make sense of the often huge solution data files that are produced
    - Reduce data via post processing
    - Create visualizations of the solution variables

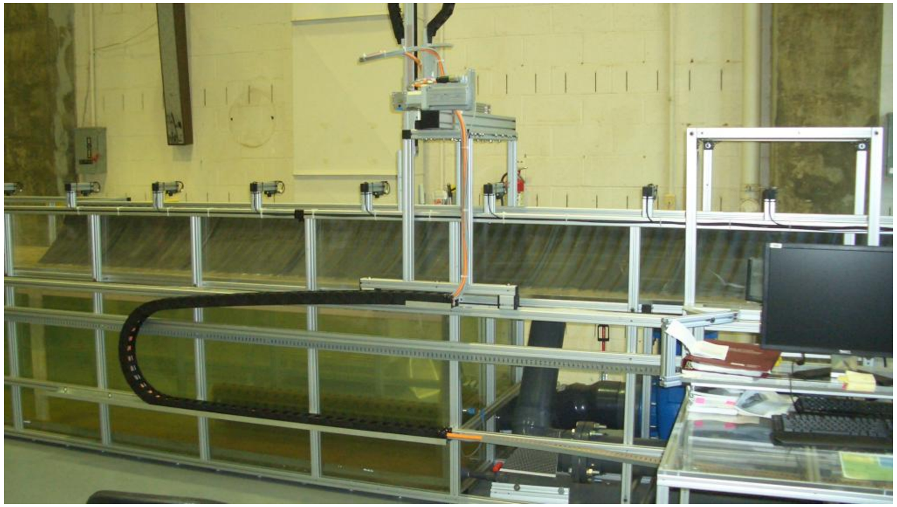
Challenges in Hydraulic Analysis Using 3-D CFD Software

- Everything that affects the flow field should be represented in the geometry
  - Becomes a big data collection and surface and solid generation problem
  - Newer tools may help with providing a very detailed geometry (every tree, etc.)
  - Surface wrapping helps the CFD analyst include sufficient detail for CFD analysis
- Turbulence will remain a challenge for the foreseeable future
  - Direct numerical simulation (DNS) must resolve eddies over 4 or more orders of magnitude in size, which may require computational grids with 10^{14} points or more
  - Large eddy simulation (LES) is needed for many problems but is just becoming feasible for reasonably sized and formulated problems
  - Reynolds-averaged Navier-Stokes (RANS) equations with a two equation turbulence model formulation is adequate for many engineering applications
- Scour simulations must span flow time scale (milliseconds) and scour time scale (hours)
- As the bed is displaced in scour simulations, high computational mesh quality must be maintained.

CFD Modeling of Culvert Flow Experiments at the Turner-Fairbank Highway Research Center Has Been Successful

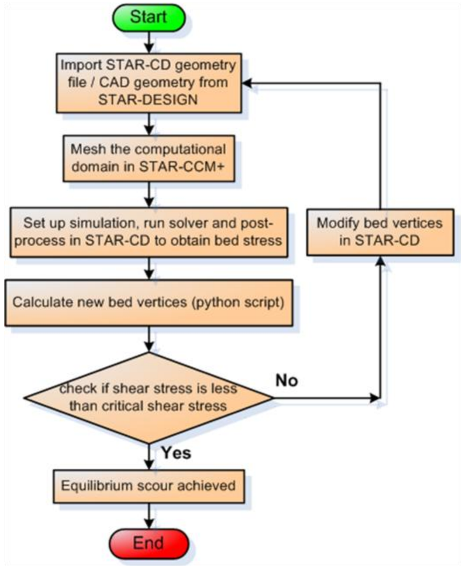






Goals of CFD Scour Analysis Effort

- Develop a basic approach to 3-D scour analysis using commercial CFD software that can be used as a foundation for testing increasingly advanced physics models
  - Use minimal physics: single phase, no sediment transport, critical shear force criteria
- Concentrate on automating methods to
  - Displace the bed in a 3-D domain
  - Re-mesh the domain to maintain mesh quality as the bed is displaced
  - Perform a large sequence of quasi-steady CFD runs to obtain bed shear stress
  - Complete the analysis within 1 or 2 days
- Compare results with TFHRC pressure flow scour experiments and look for
  - Conservative prediction of scour hole depth
  - 3-D effects in the scoured bed surface topology
- Document procedures for further use and development



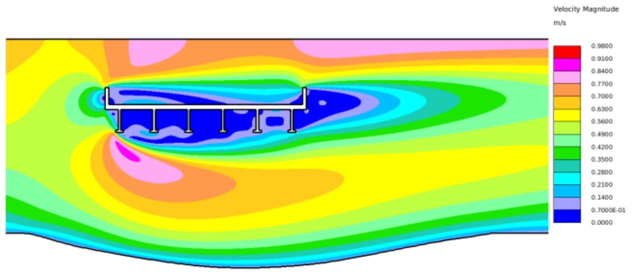
Scour Critical Bridge in Illinois Region 1

Abutment contraction is clearly visible.

Counter measures under consideration but not designed yet.

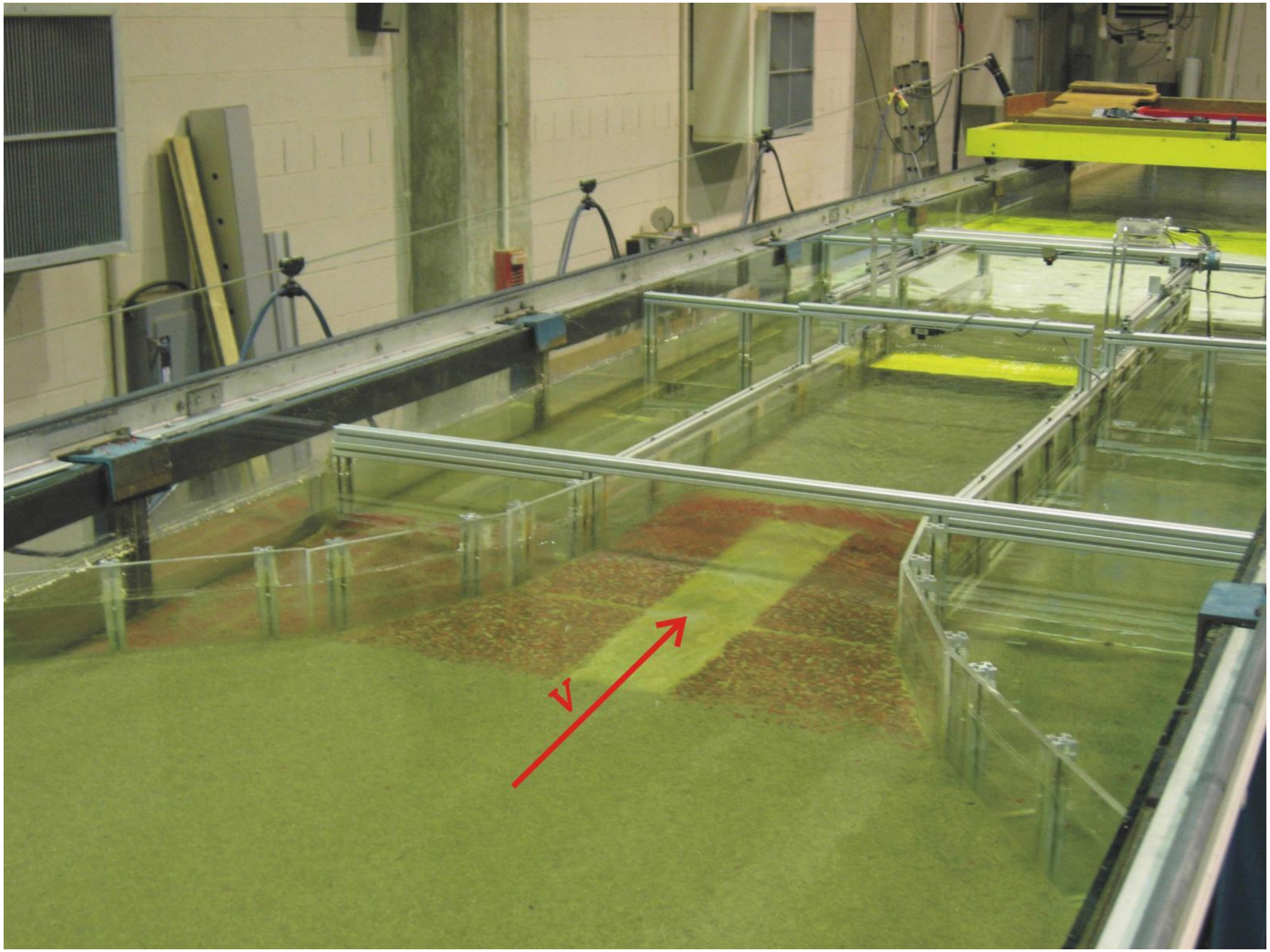
CFD Scour Analysis Modeling Alternatives

- Single-phase flow with a “moving boundary” formulation based on comparison of the local shear stress at the bottom surface with the “critical shear stress” at which the scour is assumed to initiate.

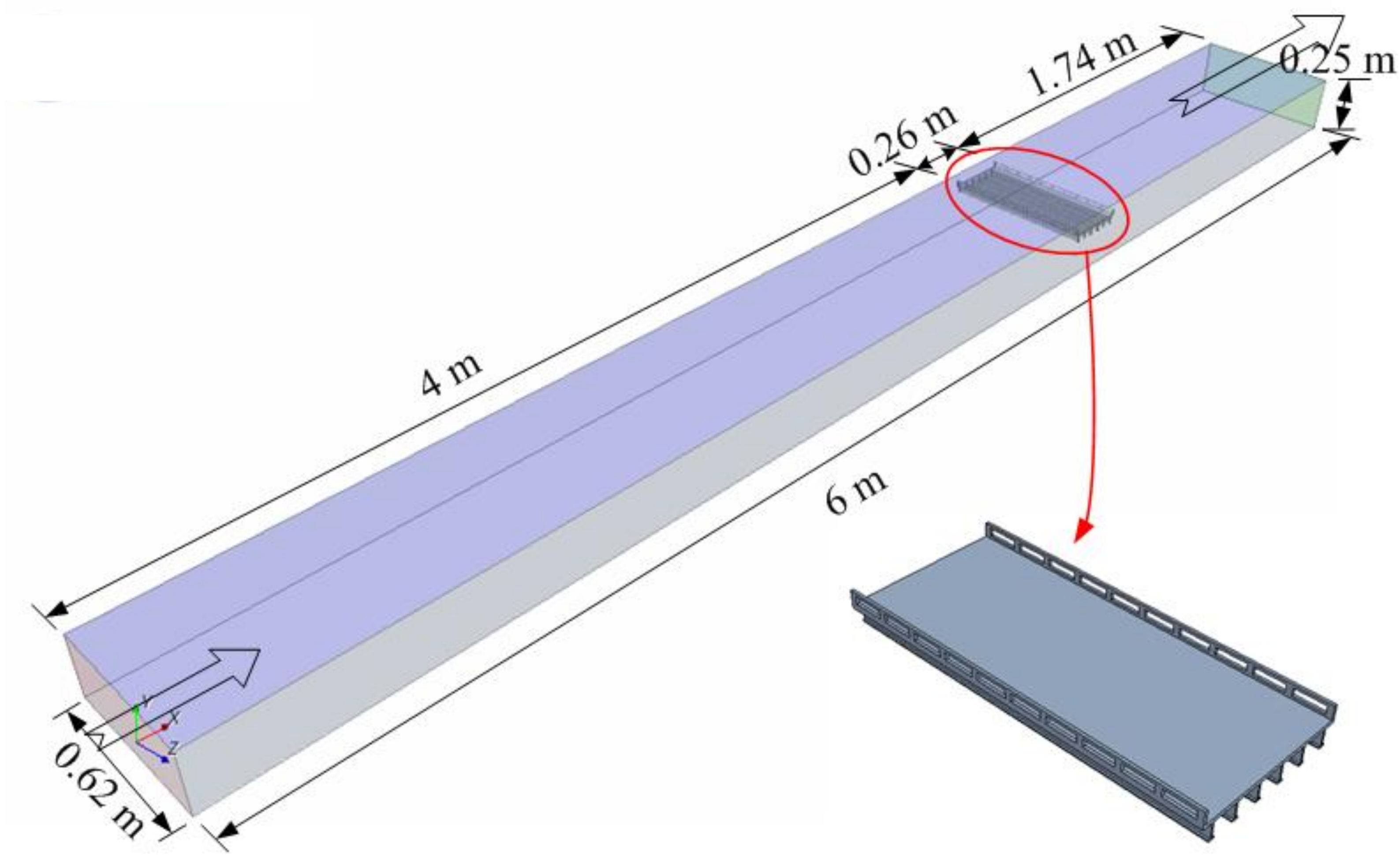


- Multi-component flow technique
  - Flow field is a single continuum
  - Sediment transported as a scalar concentration
    - Scour rate source at packed bed interface
    - Settling rate
  - Fluid properties are a function of sediment concentration
- Full multiphase flow – water and particles treated as interpenetrating continua
  - Model is the most promising (most fundamental physics)
  - Most computationally intensive
  - FLUENT and STAR-CD do not have models (terms in PDE’s) to account for stationary beds
- Large Eddy Simulation (LES) to accurately predict vortex structures in the flow that cause scour – requires bigger computers

Turner-Fairbank Highway Research Center Pressure Scour Experiments – Scour Flume



3-D Computational Domain



Critical Shear Stress is Used as Scour Criteria

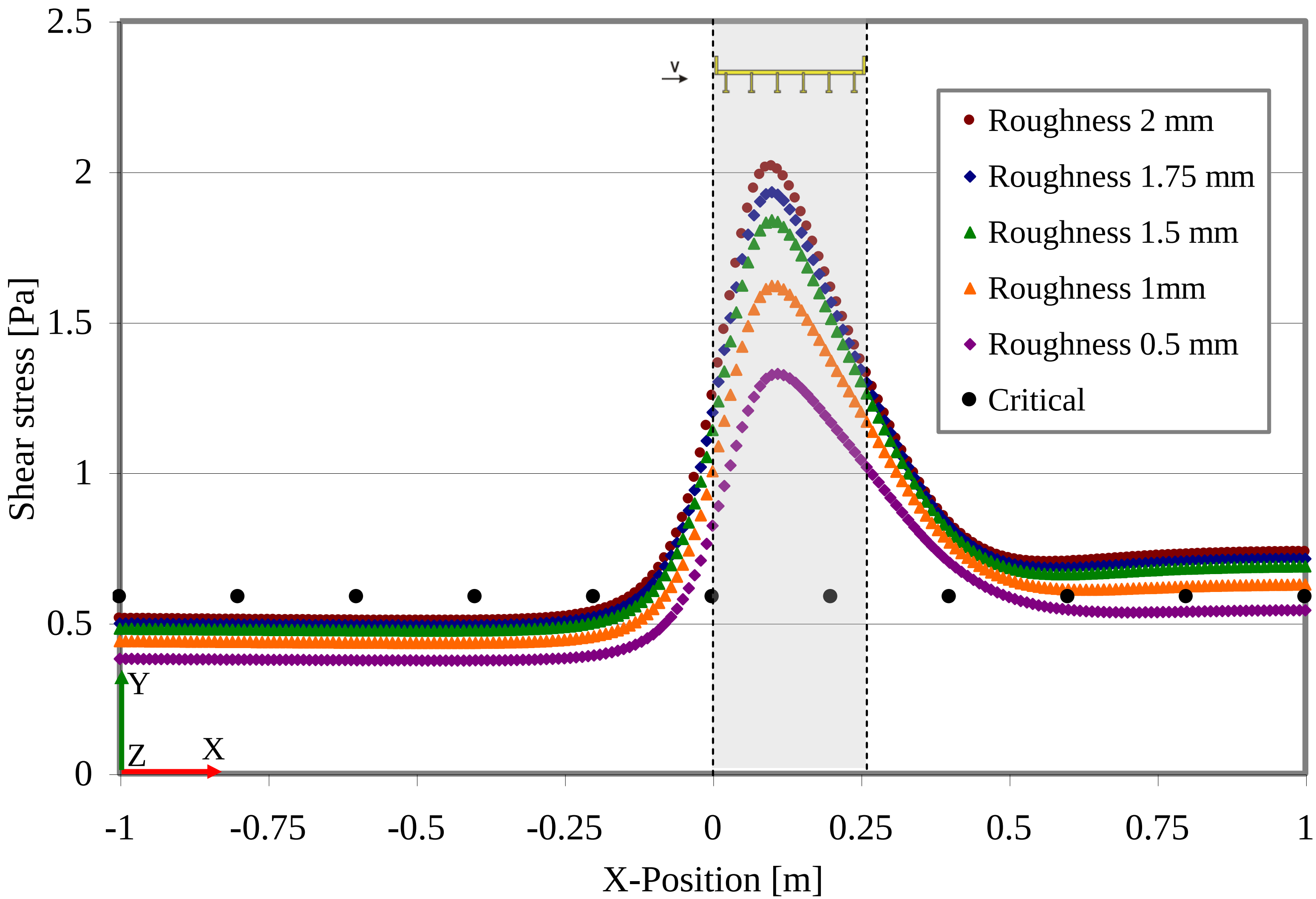
The bed is displaced when the bed shear exceeds critical shear stress

$$\frac{\boxed{\tau_c}}{(\rho_s - \rho)gd_{50}} = \frac{0.23}{d_*} + 0.054\left[1 - exp\left(-\frac{d_*^{0.85}}{23}\right)\right]$$

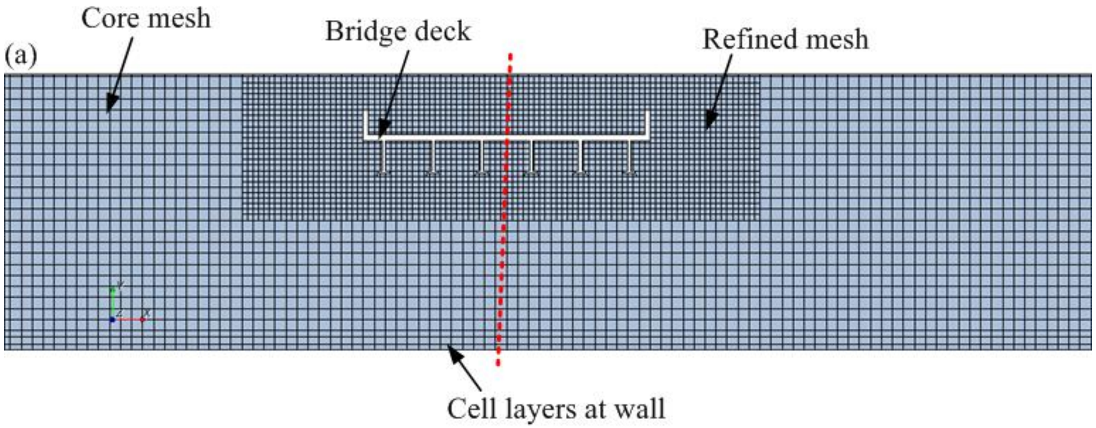
ρ_s	=	Density of sediment
g	=	Acceleration due to gravity
d_{50}	=	Median size of the bed sediment
$d_* = \left[\frac{(\rho_s/\rho - 1)g}{v^2}\right]^{1/3} d_{50}$	=	Dimensionless diameter
$u_c^* = \sqrt{\frac{\tau_c}{\rho}}$	=	Critical velocity

For 1 mm sand, critical shear stress is 0.586 Pa.
For 2 mm sand, critical shear stress is 1.410 Pa.
Bed roughness is taken to be twice the mean diameter of sand.

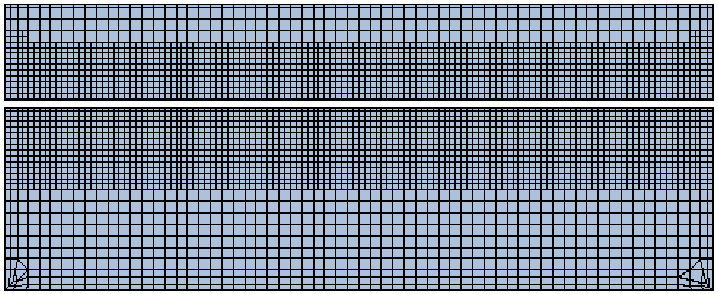
CFD Scour Analysis is Sensitive to Bed Roughness: Variation of Shear Stress with Roughness



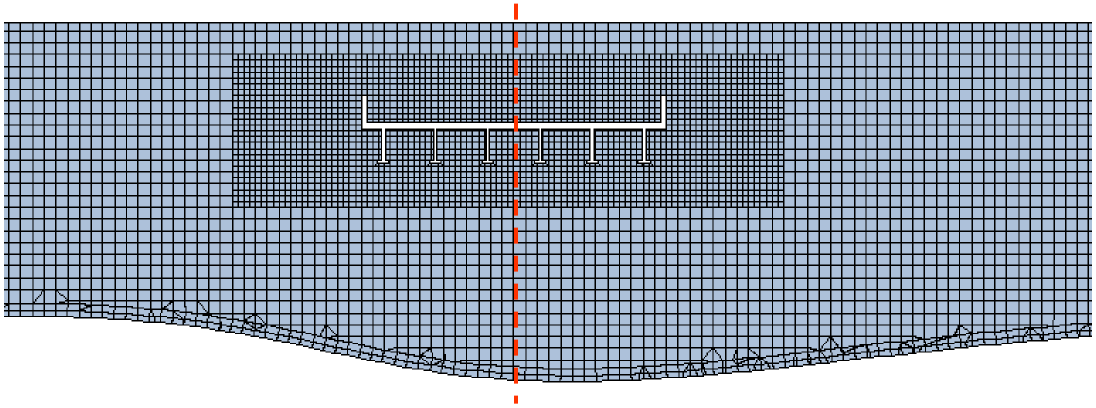
STAR-CD Solver Yields Bed Shear, pro-STAR Used to Displace Bed, STAR-CCM+ Mesher Used to Maintain Mesh Quality



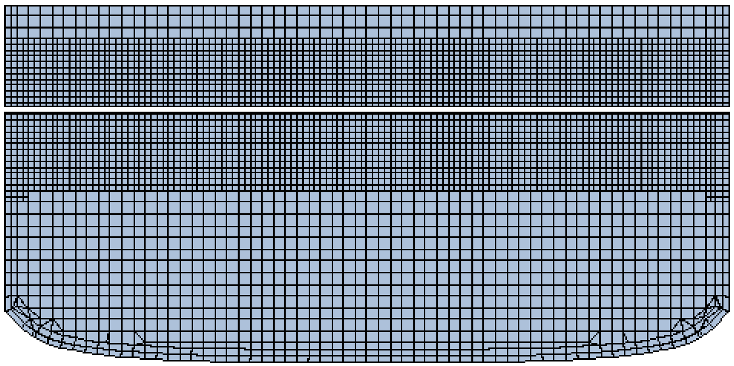
Zoomed front view flat bottom



Flat bottom cross section

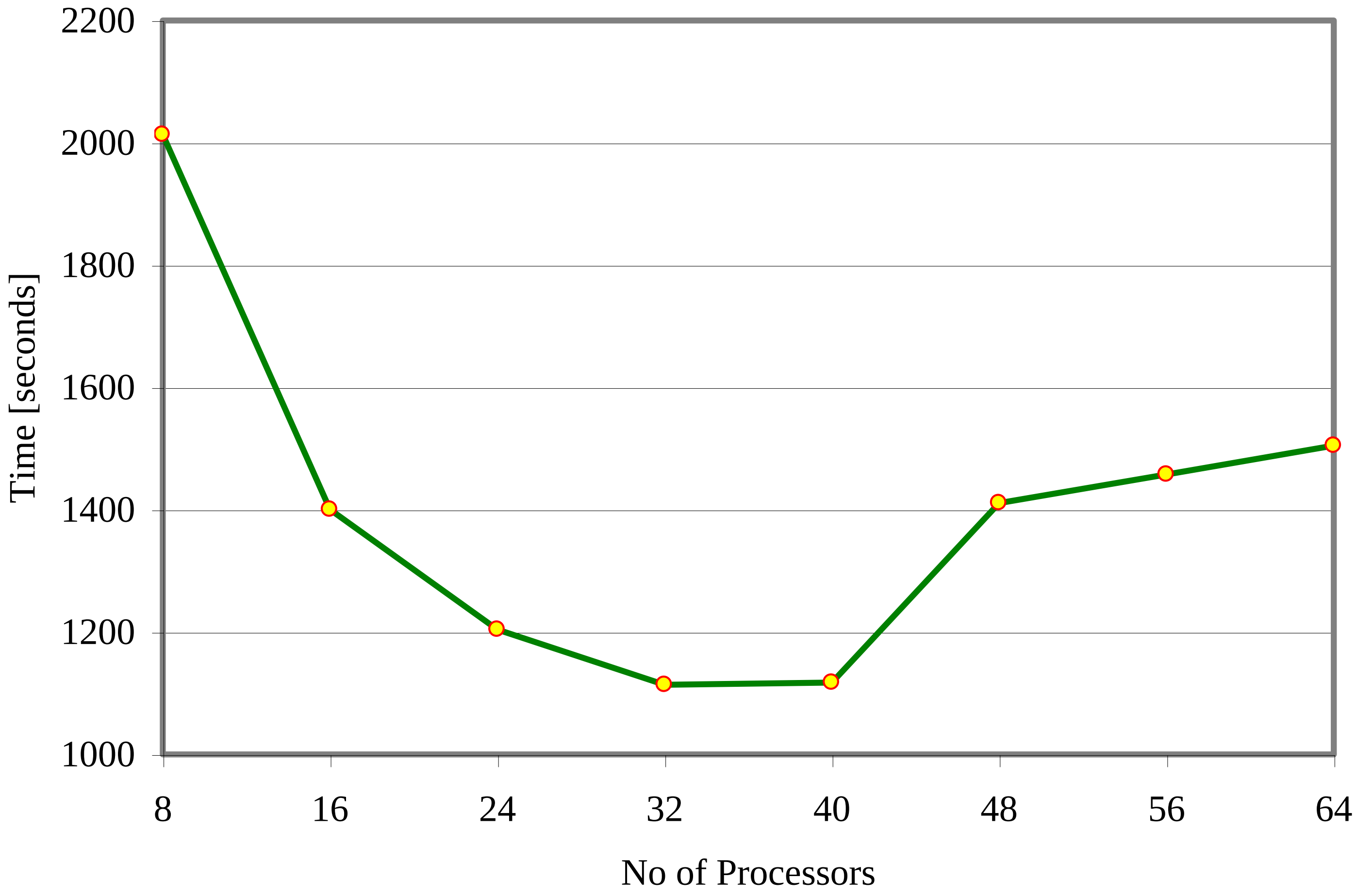


Zoomed front view scoured bottom



Scoured bottom cross section

Solver Time for Quasi-Steady Run versus Number of Processors



Simulations performed on 1.3 million cell computational domain

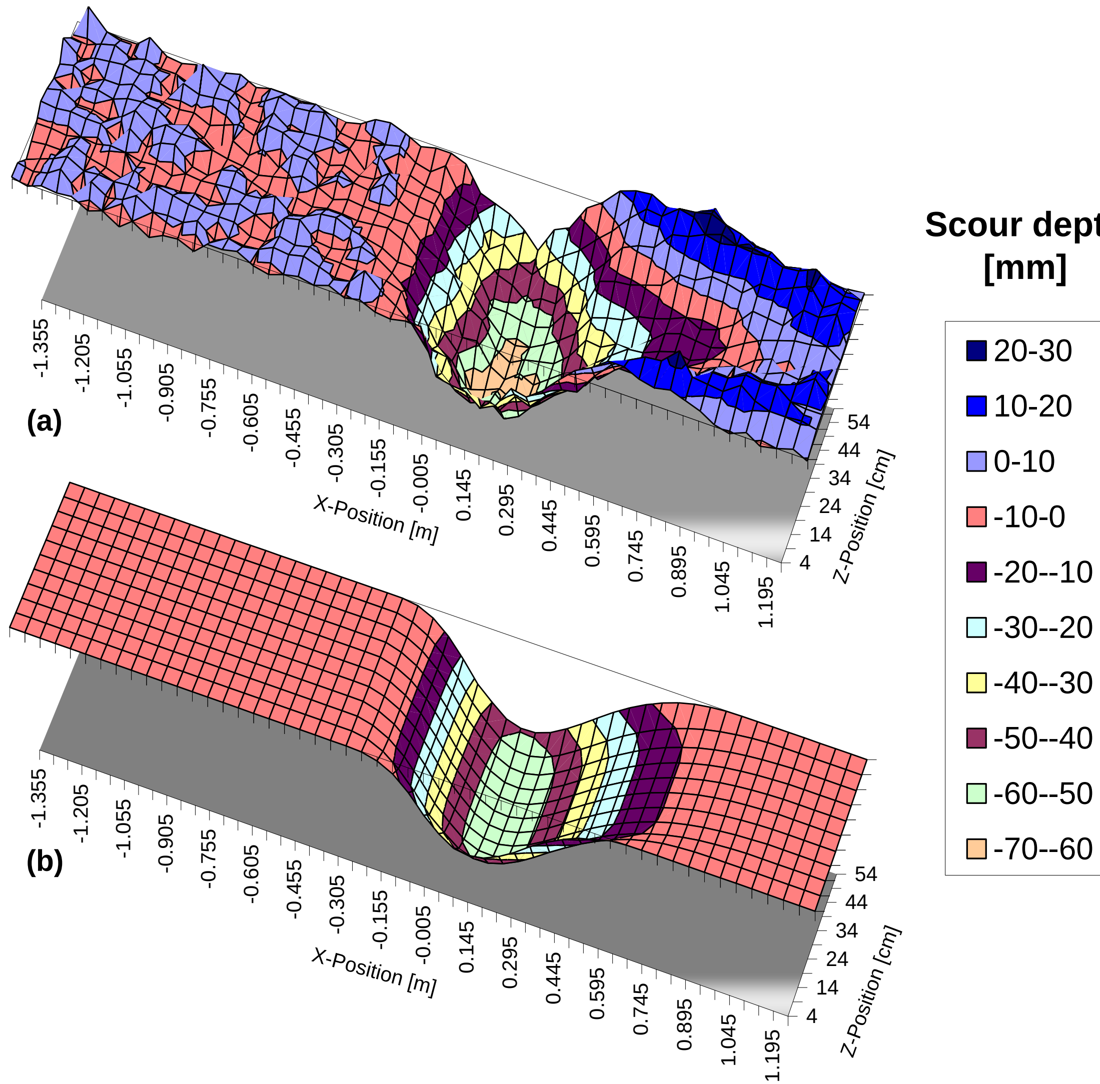
3-D SCOUR CONTOURS

h_b=11.5 cm

(a) Experimental

(b) Simulation

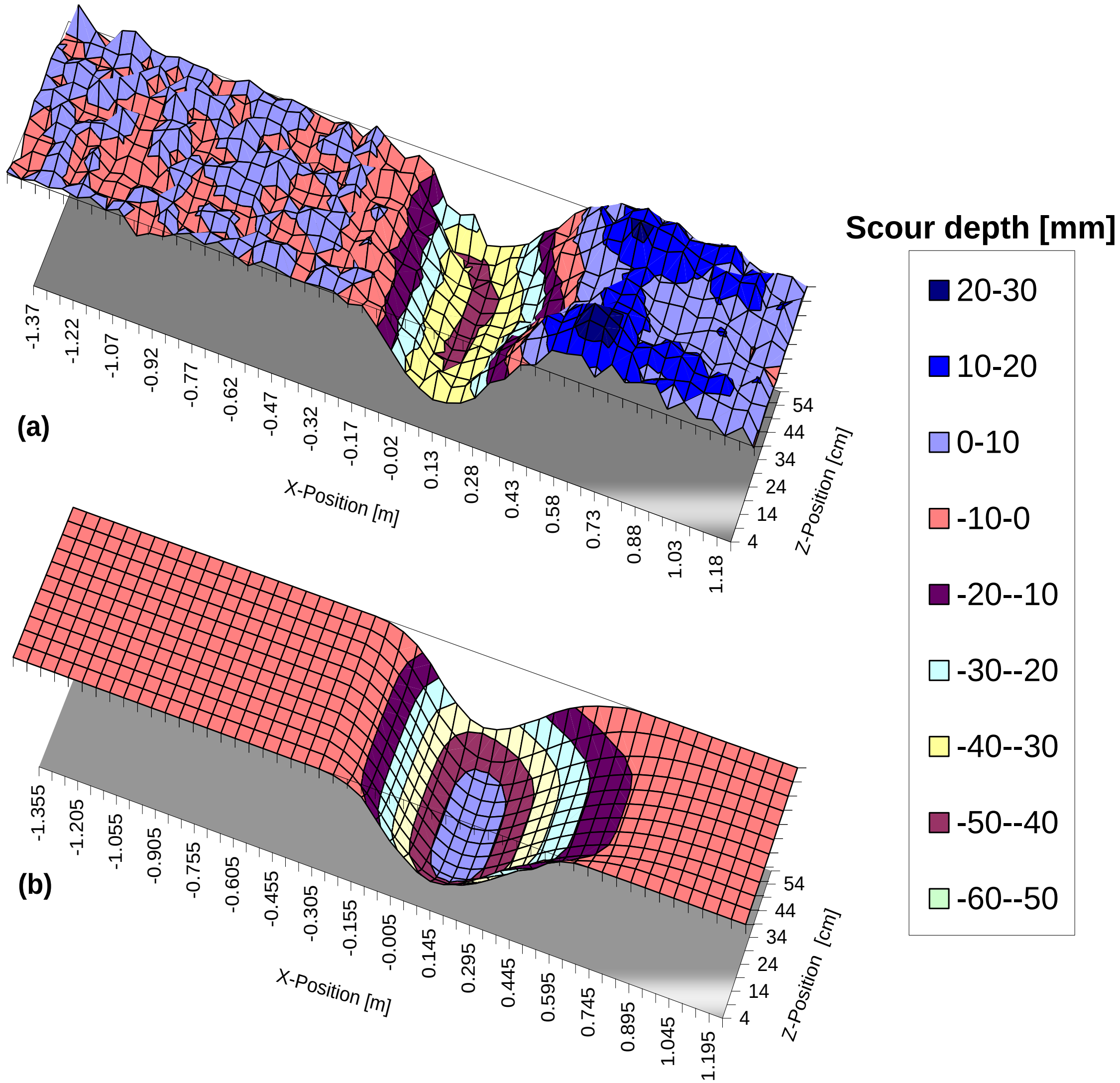
2 mm sand



3-D SCOUR CONTOURS h_b=13.0 cm

(a) Experimental

(b) Simulation



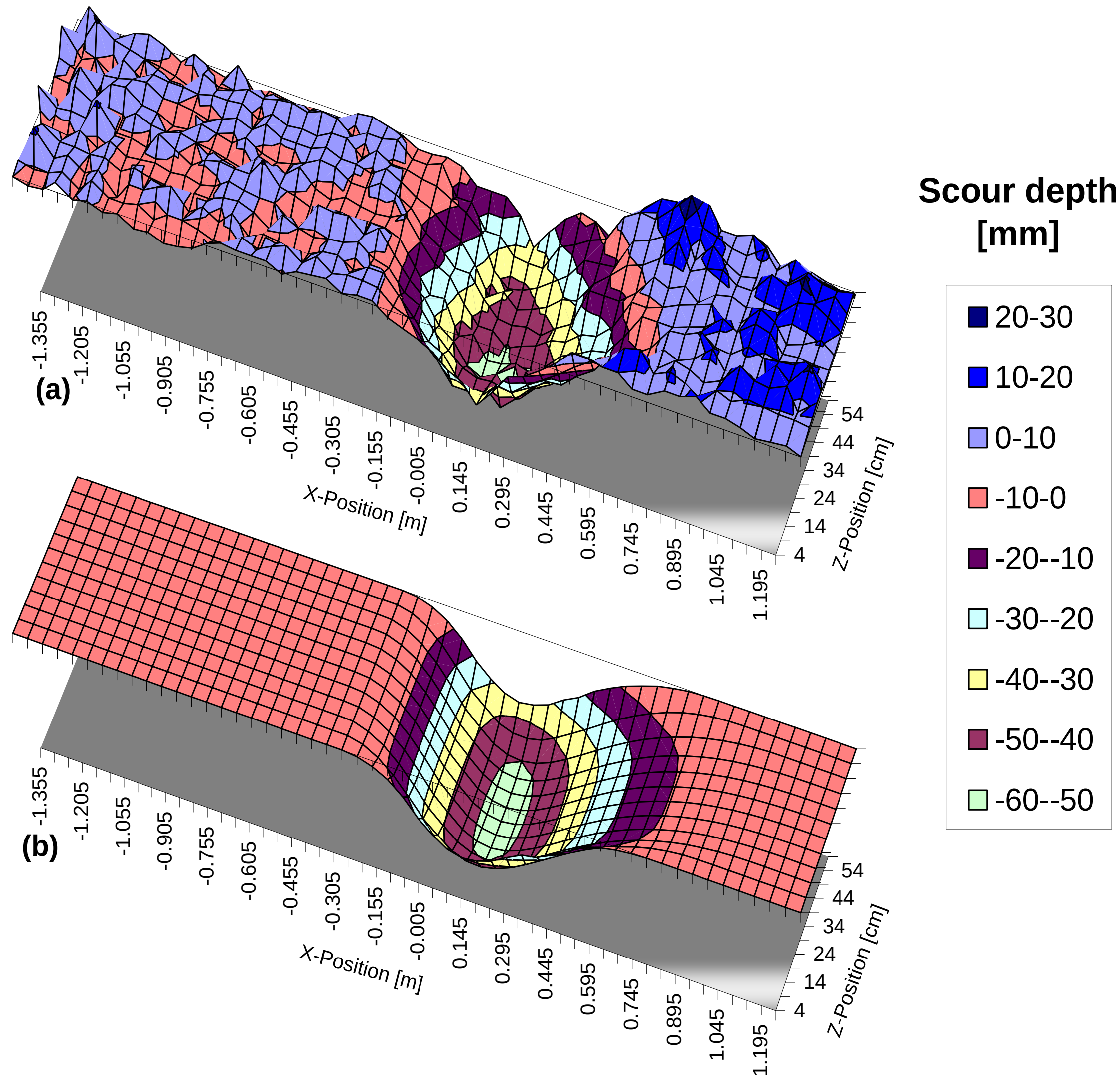
2 mm sand

3-D SCOUR CONTOURS h_b=14.5 cm

(a) Experimental

(b) Simulation

2 mm sand

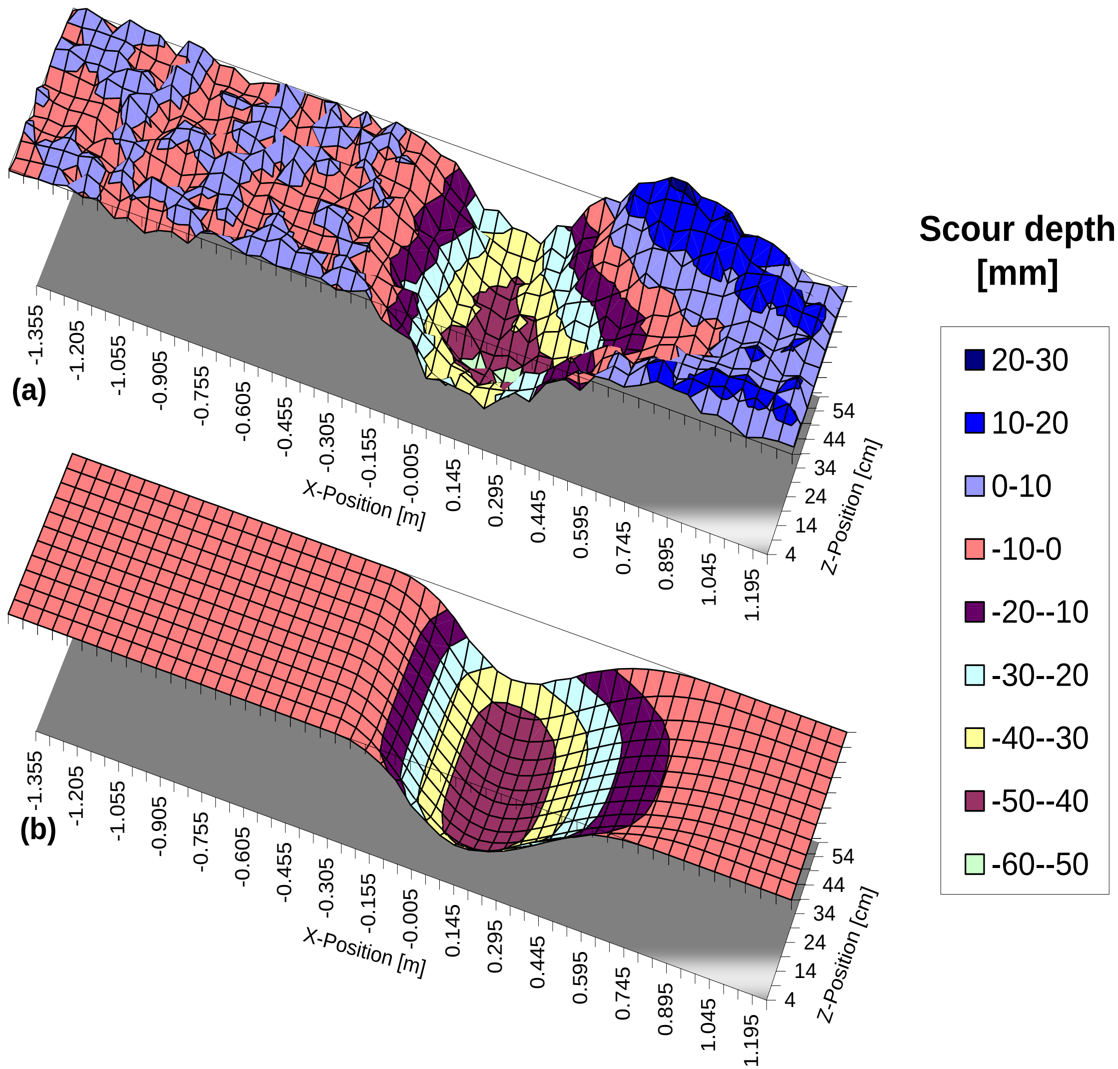


3-D SCOUR CONTOURS

h_b=16.0 cm

(a) Experimental

(b) Simulation

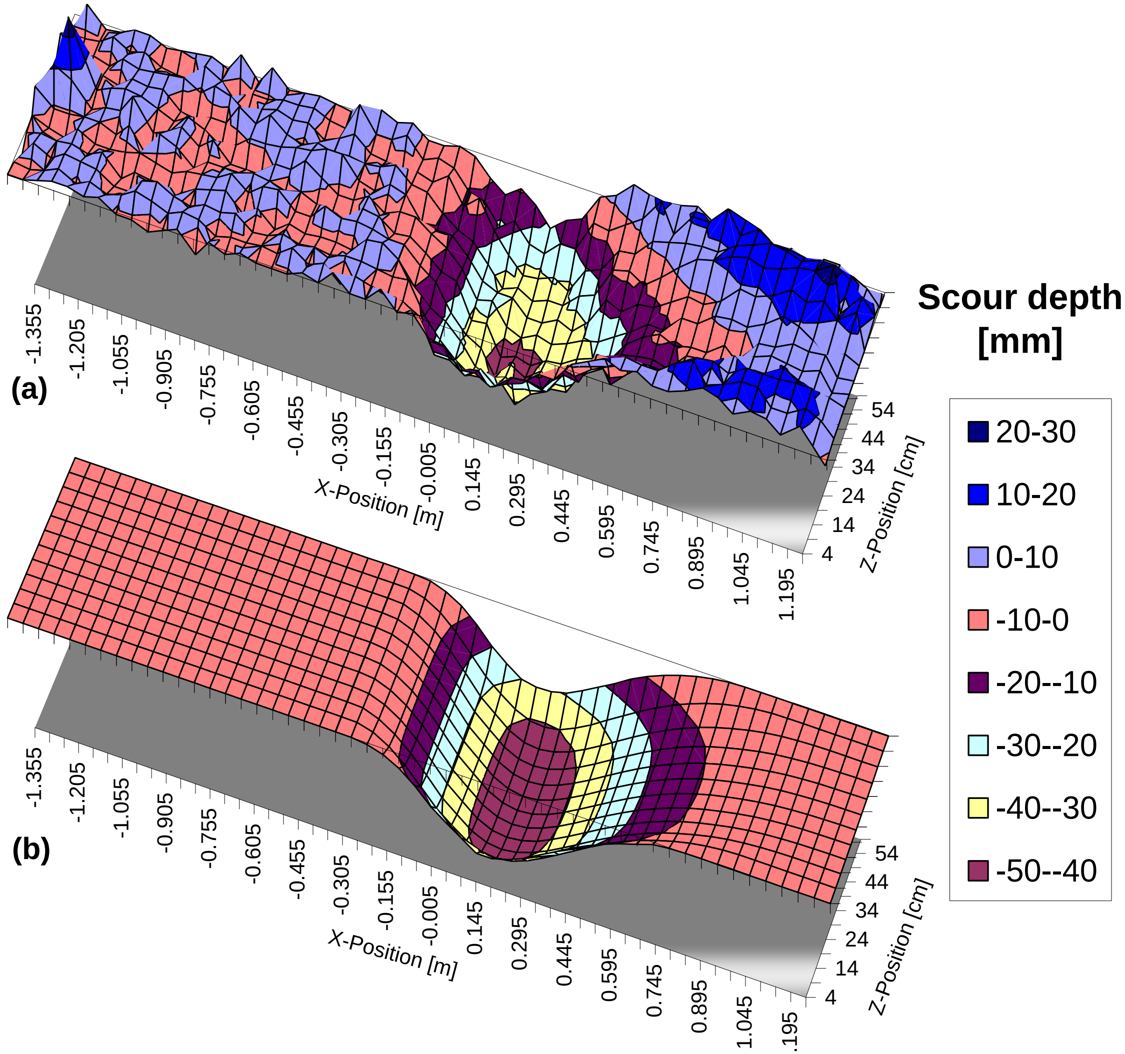


2 mm sand

3-D SCOUR CONTOURS h_b=17.5 cm

(a) Experimental

(b) Simulation



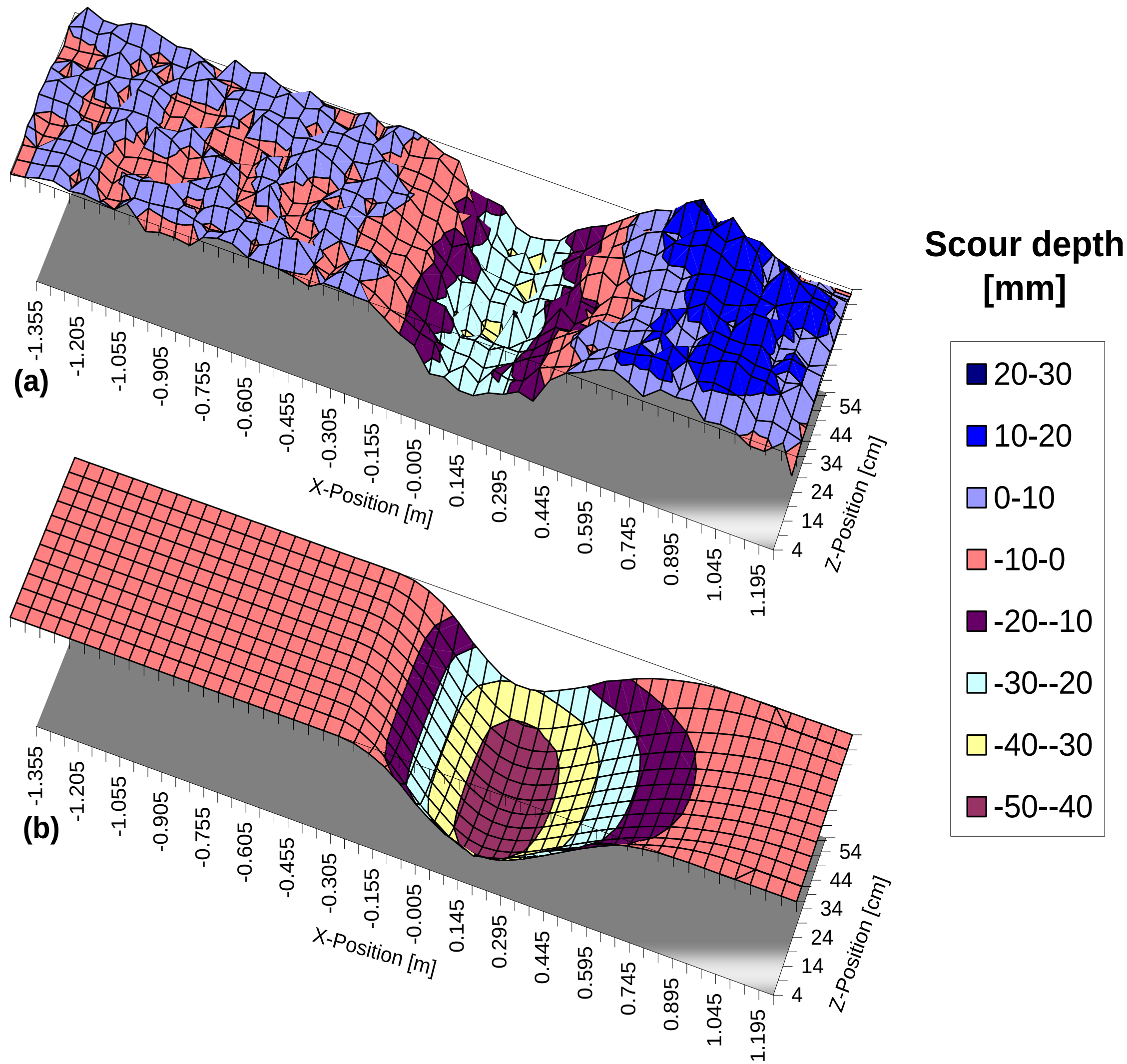
2 mm sand

3-D SCOUR CONTOURS h_b=19.0 cm

(a) Experimental

(b) Simulation

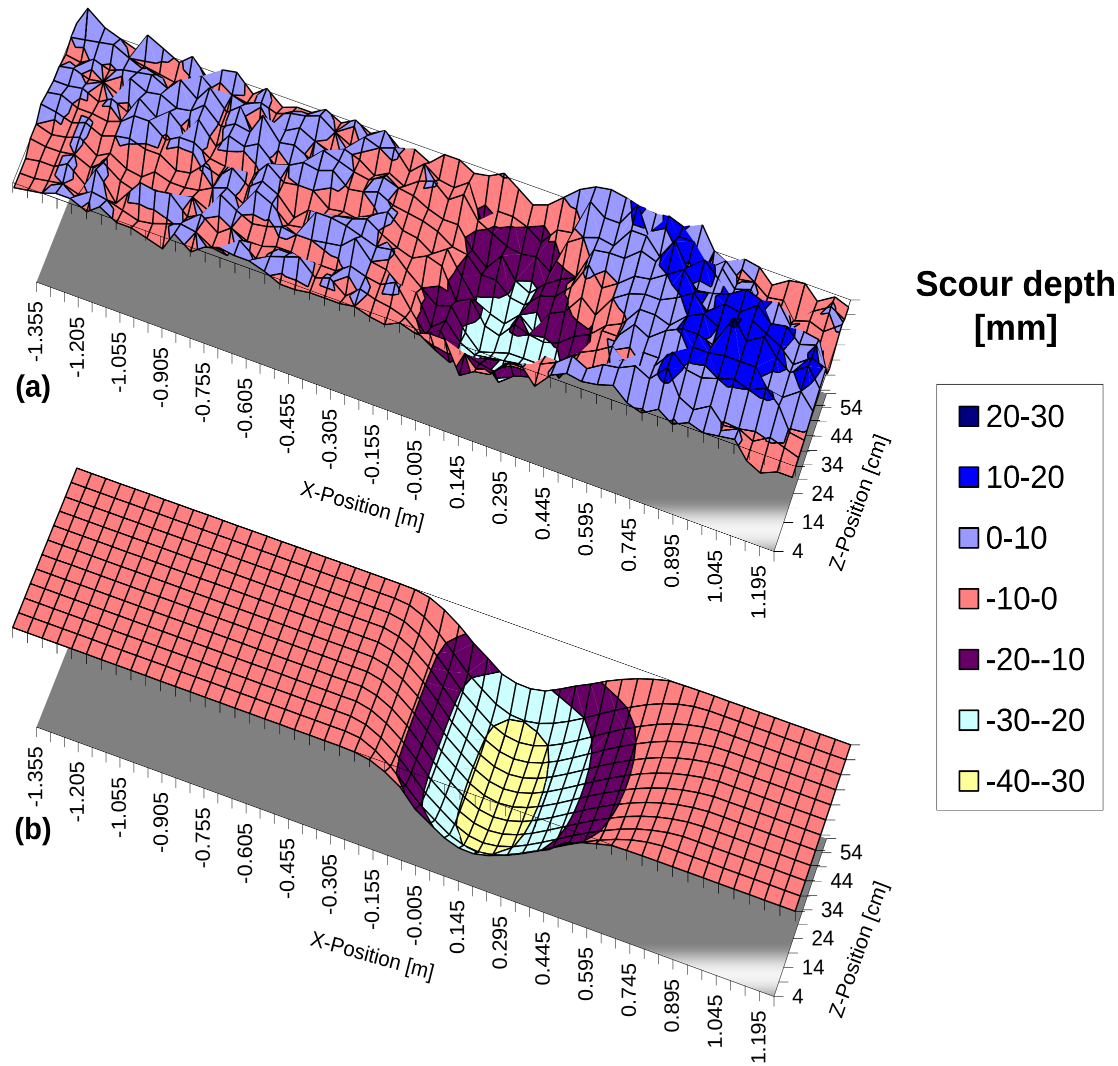
2 mm sand



3-D SCOUR CONTOURS h_b=20.5 cm

(a) Experimental

(b) Simulation



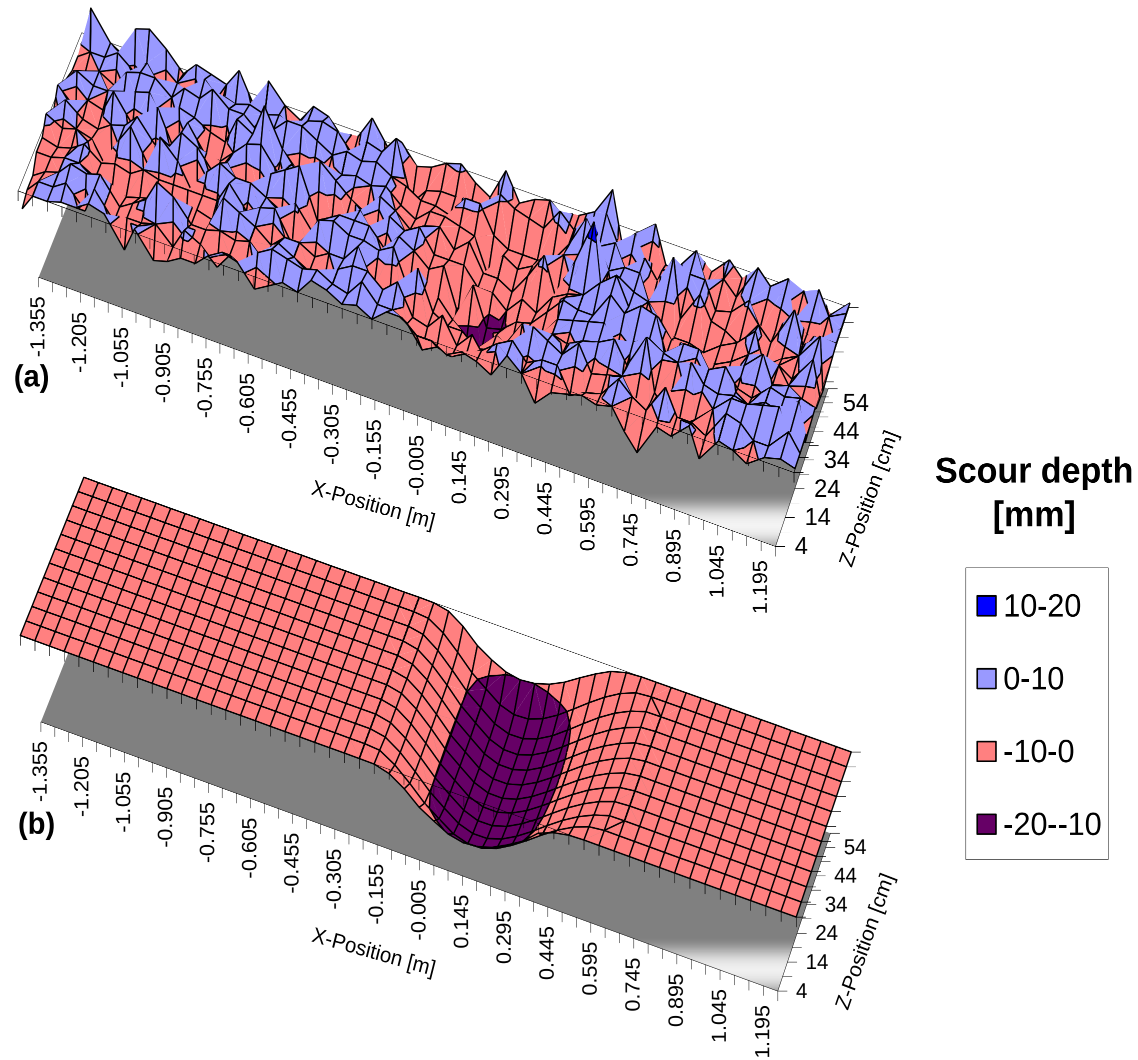
2 mm sand

3-D SCOUR CONTOURS h_b=22.0 cm

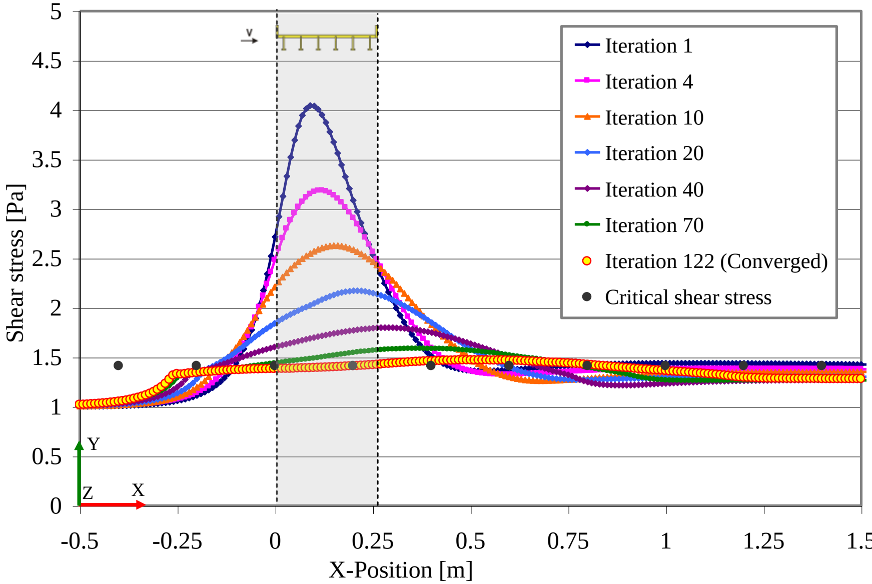
(a) Experimental

(b) Simulation

2 mm sand



Shear Stress and Scour Profiles During Iteration for h_b=13.5 cm, 2 mm sand

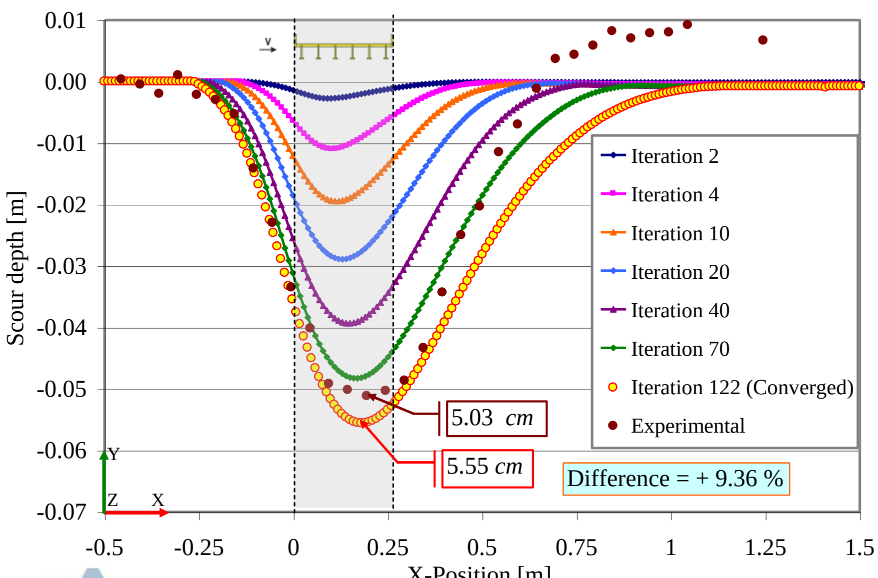


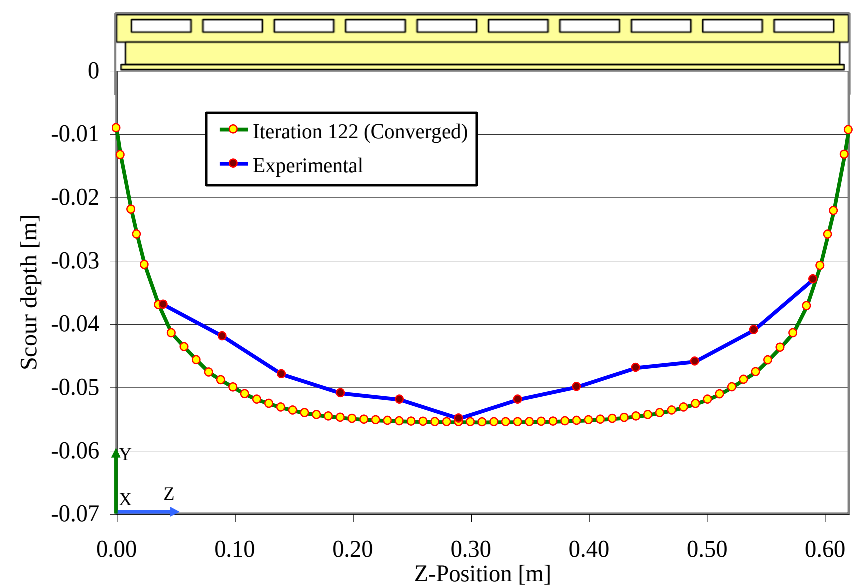
h* = 2.068

Experimental = 5.03 *cm*

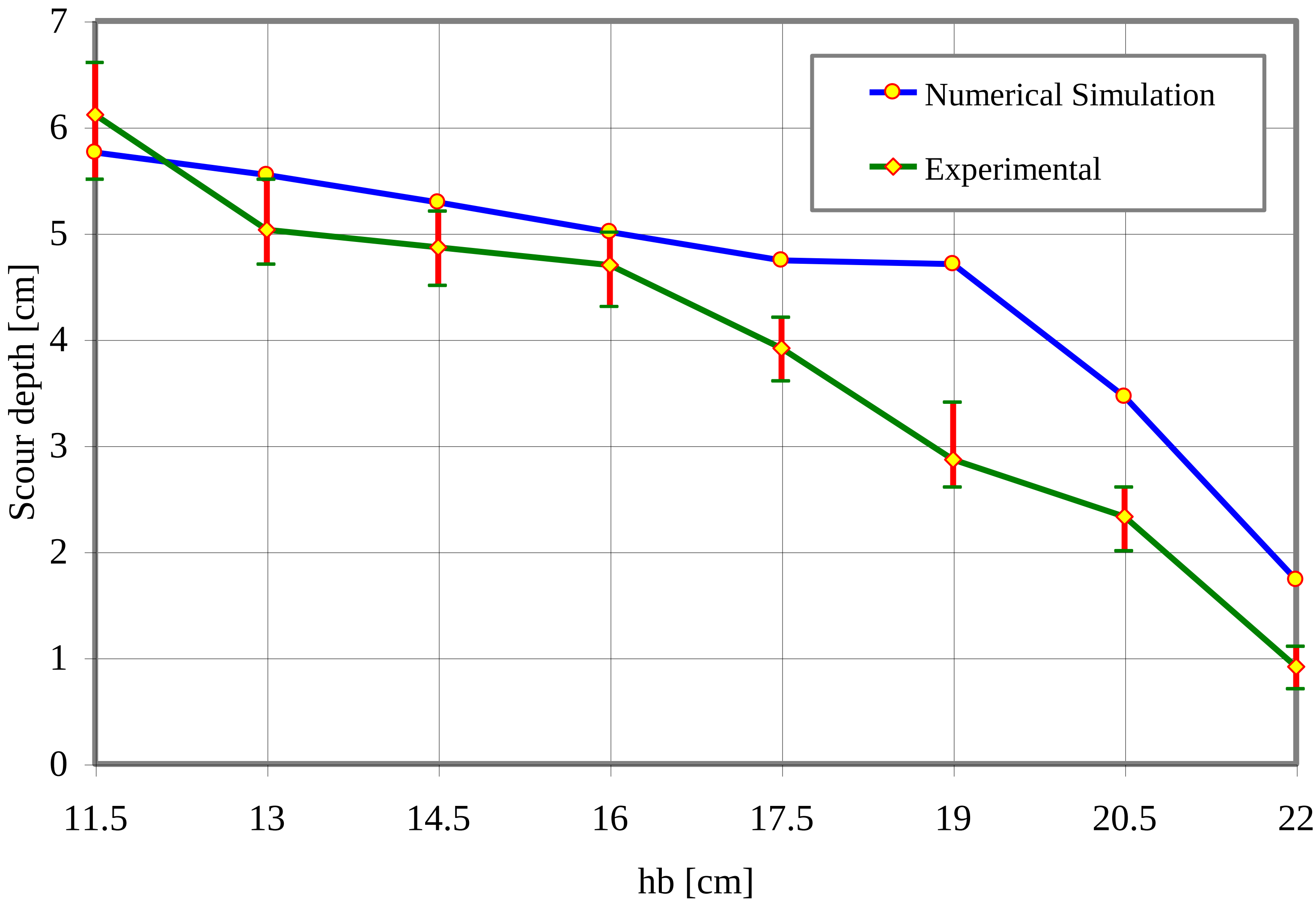
Simulated = 5.55 *cm*

Difference = 9.36%





Maximum Scour Depth vs. Bridge Height Above the Bed, h_b (for 2 mm Sand)

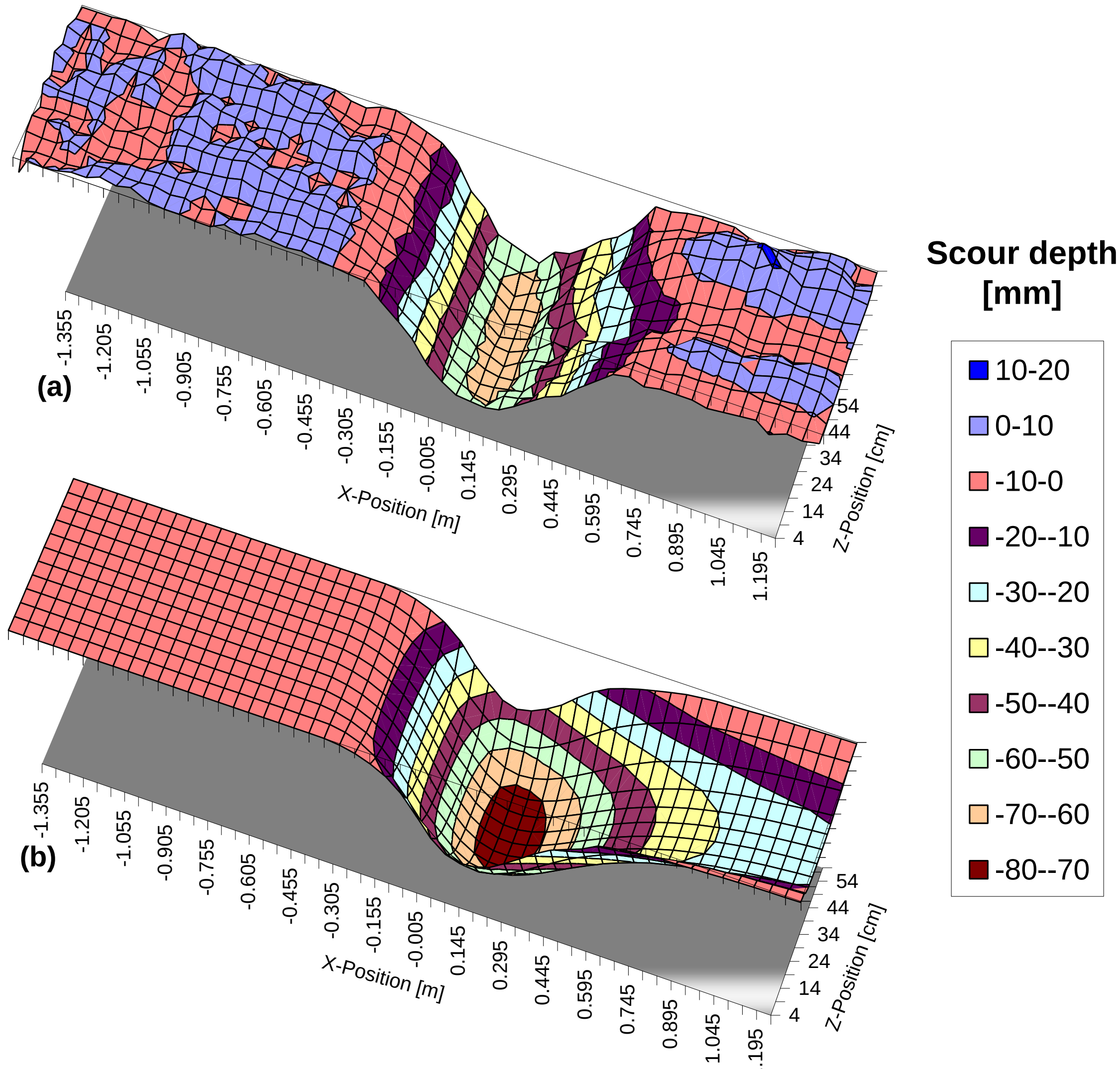


3-D SCOUR CONTOURS h_b=11.5 cm

(a) Experimental

(b) Simulation

1 mm sand

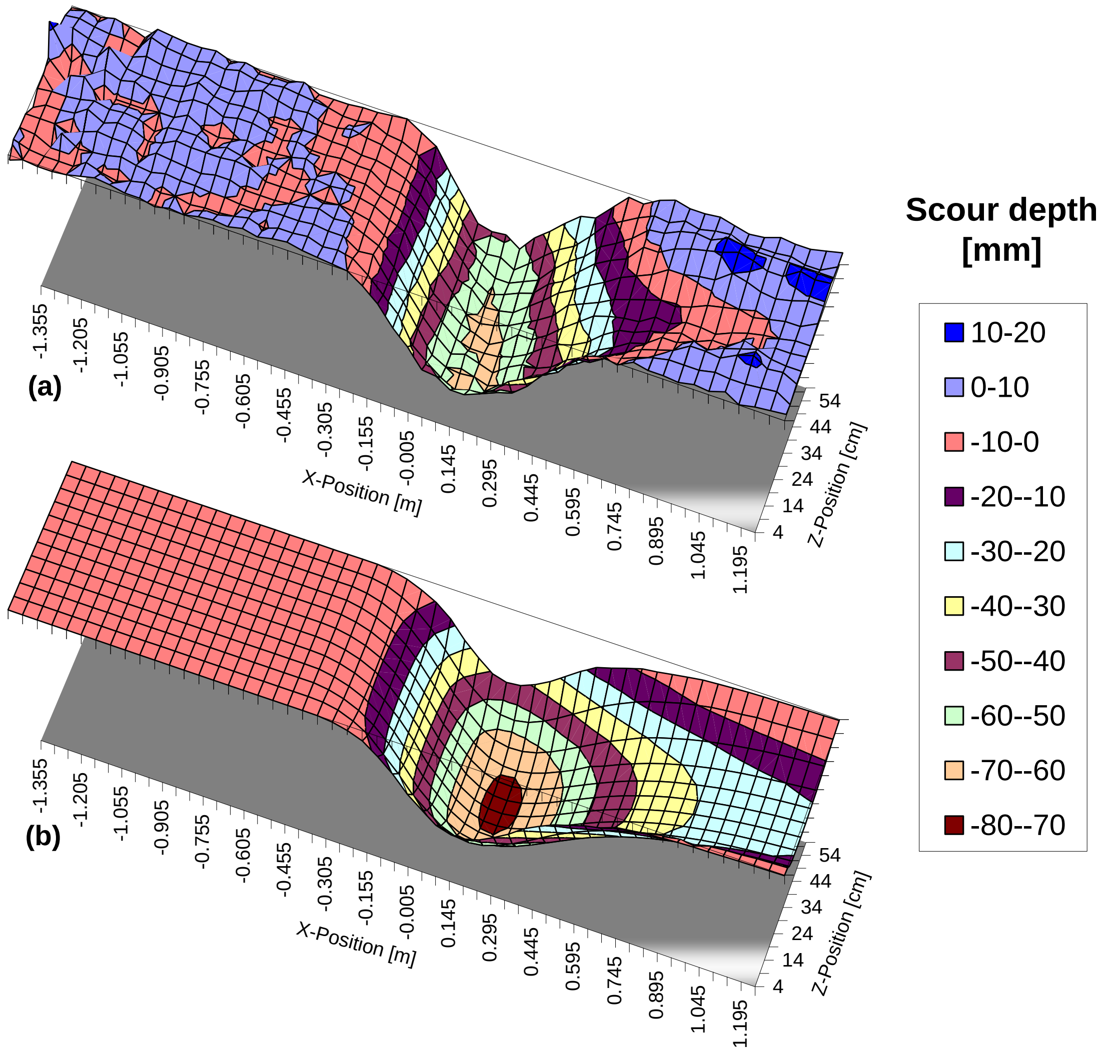


3-D SCOUR CONTOURS h_b=13.0 cm

(a) Experimental

(b) Simulation

1 mm sand

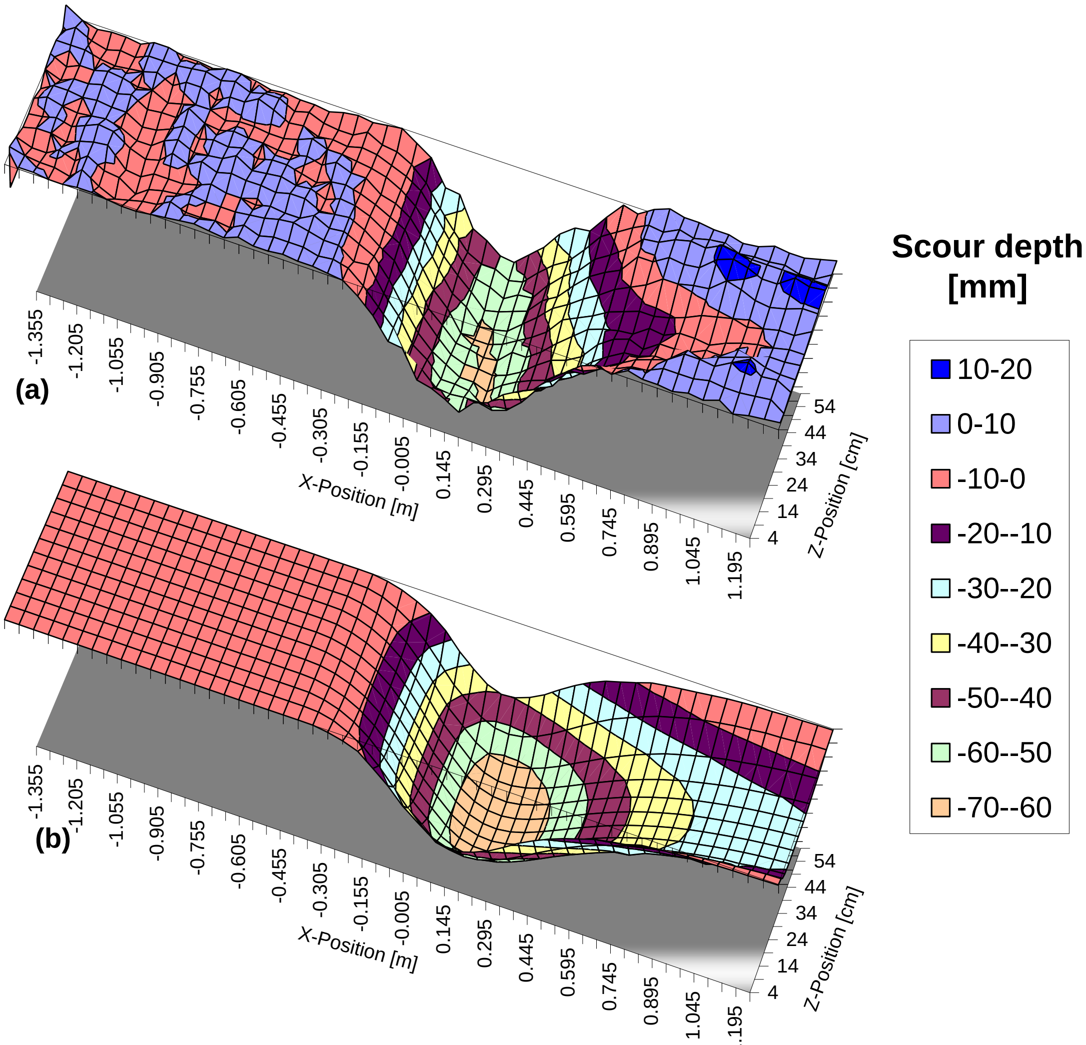


3-D SCOUR CONTOURS h_b=14.5 cm

(a) Experimental

(b) Simulation

1 mm sand

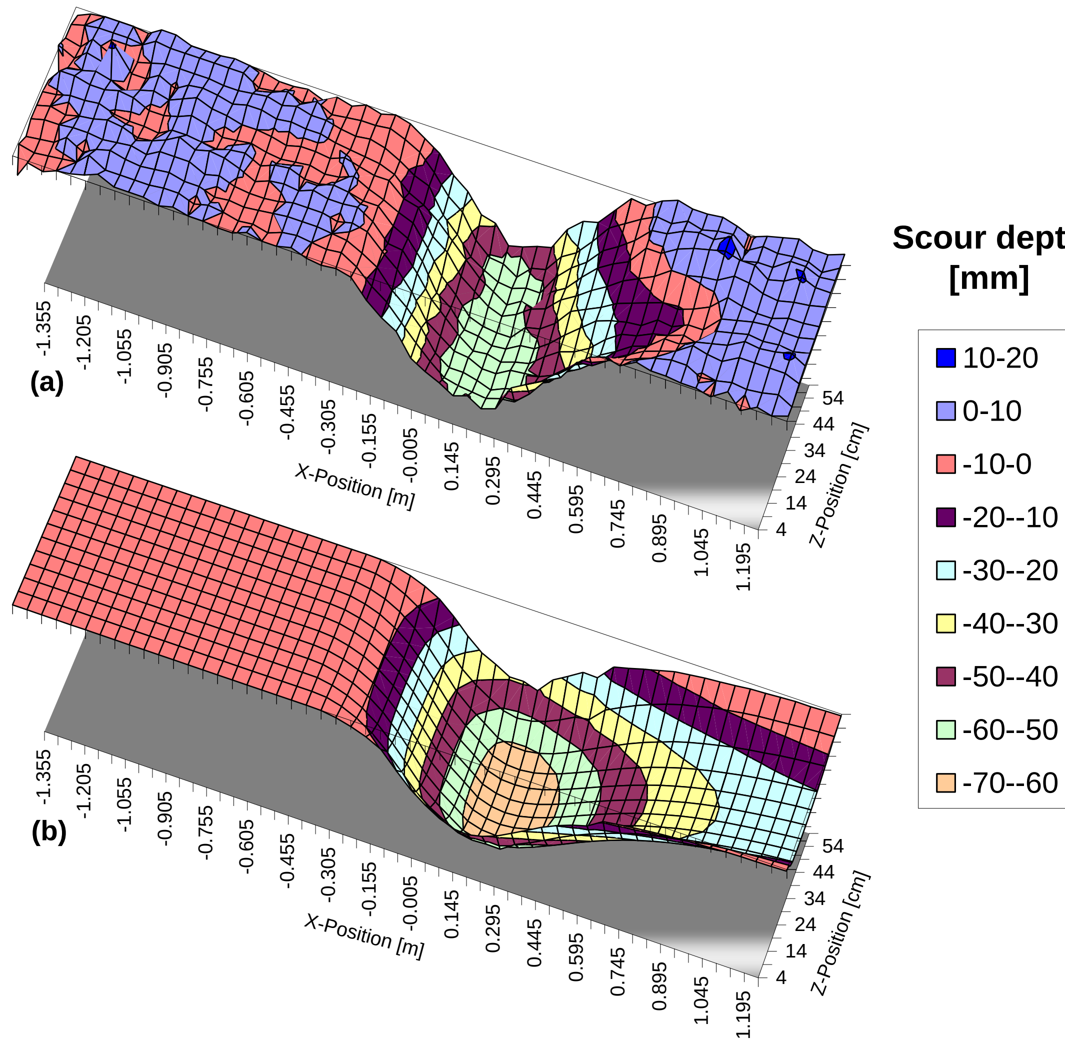


3-D SCOUR CONTOURS h_b=16.0 cm

(a) Experimental

(b) Simulation

1 mm sand

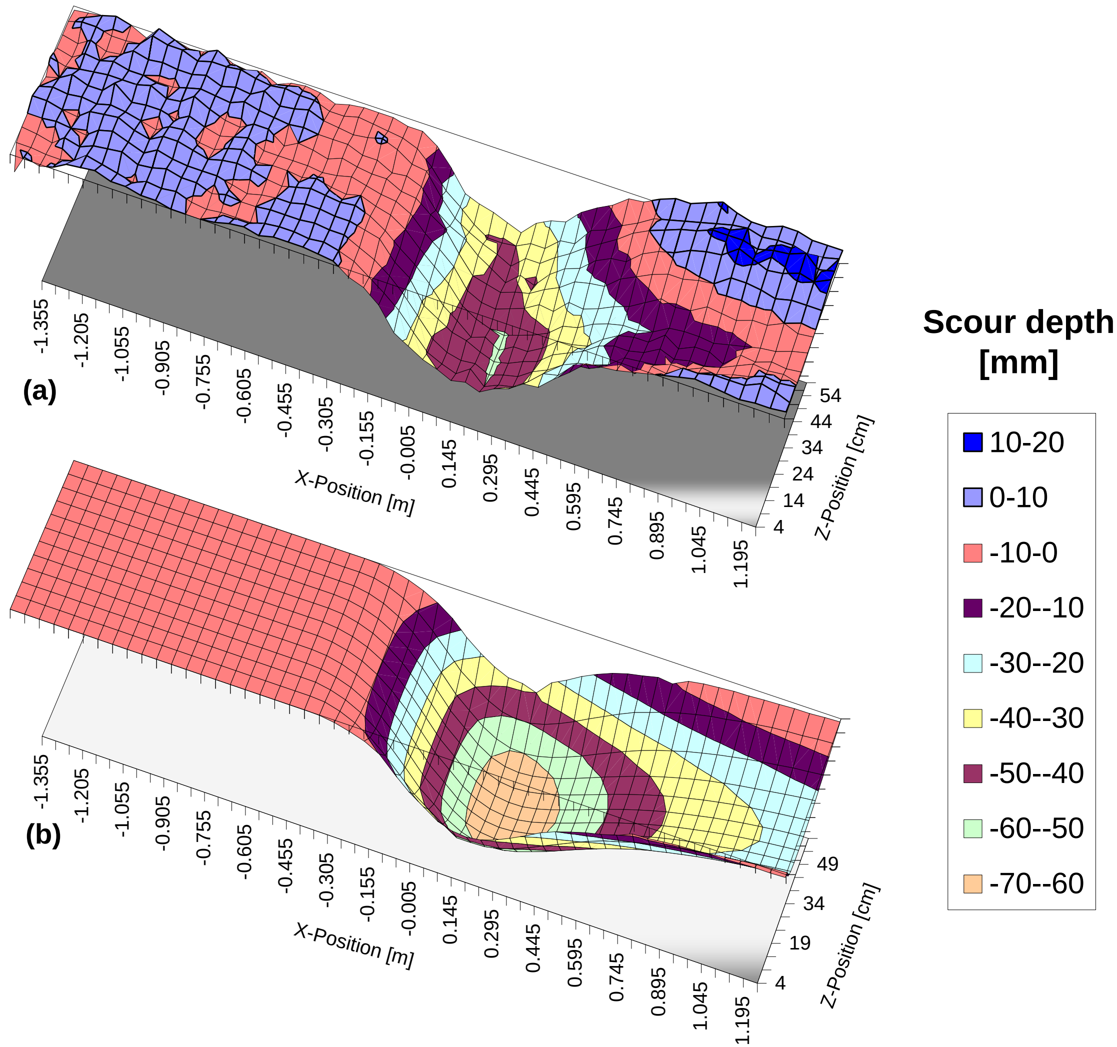


3-D SCOUR CONTOURS

h_b=17.5 cm

(a) Experimental

(b) Simulation



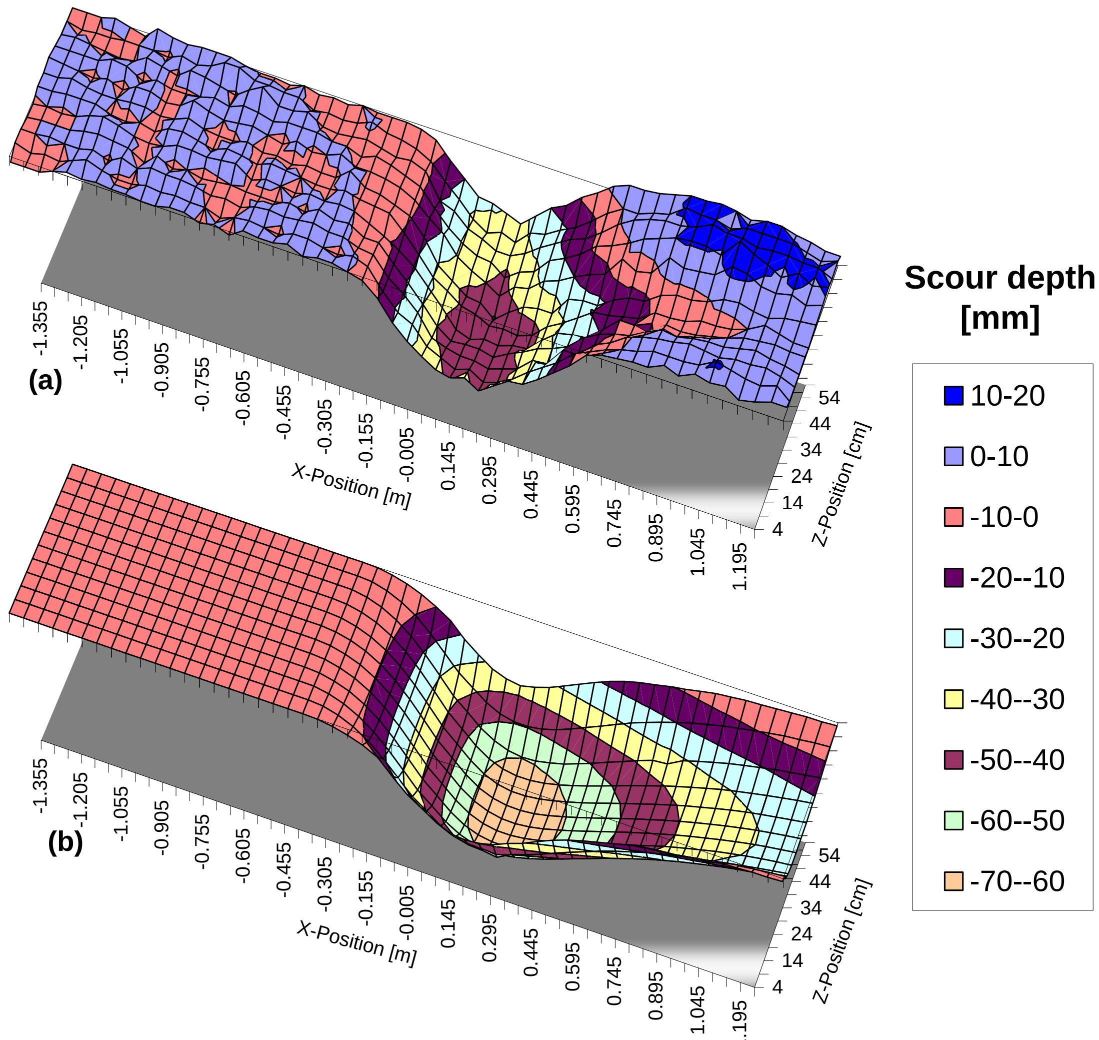
1 mm sand

3-D SCOUR CONTOURS

h_b=19.0 cm

(a) Experimental

(b) Simulation



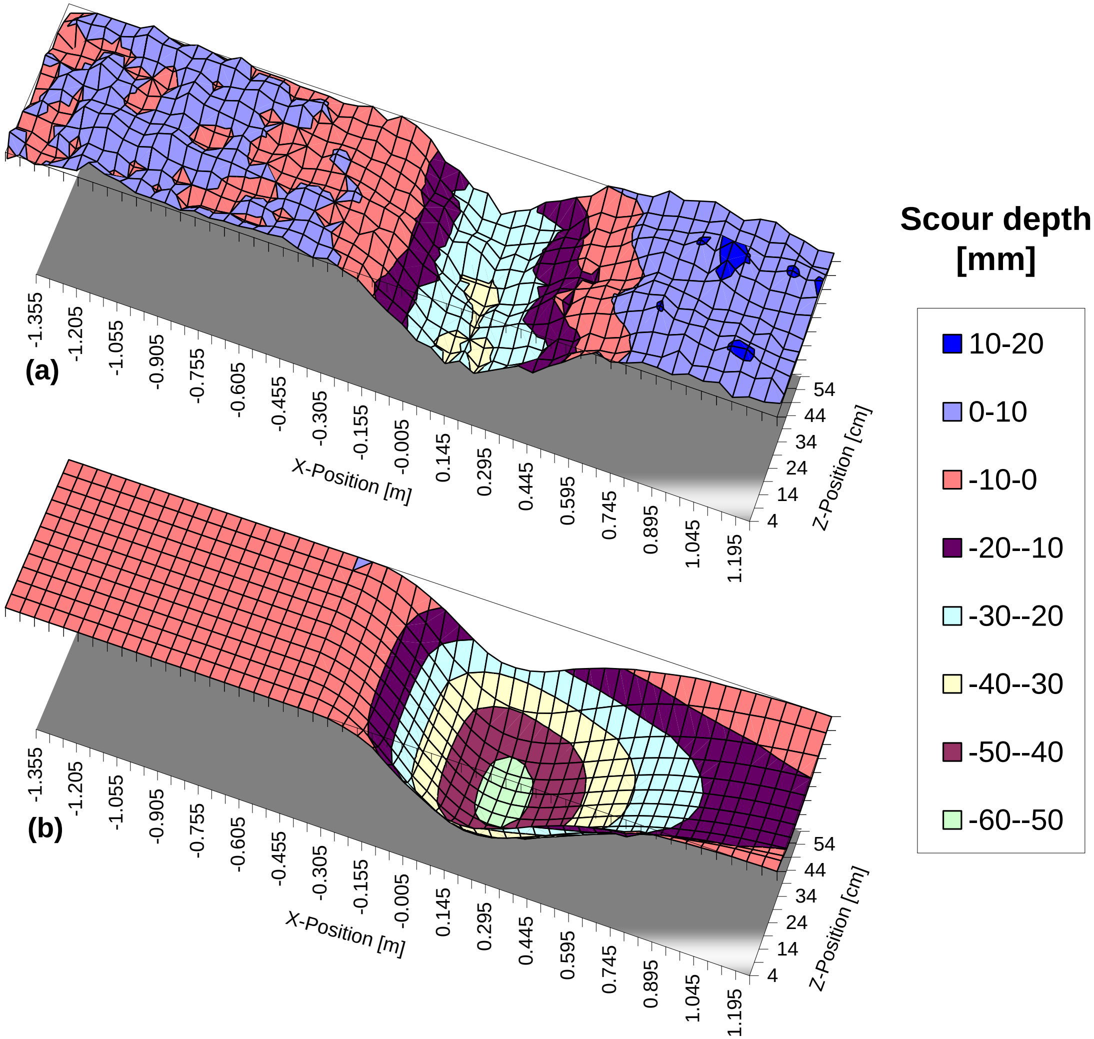
1 mm sand

3-D SCOUR CONTOURS h_b=20.5 cm

(a) Experimental

(b) Simulation

1 mm sand



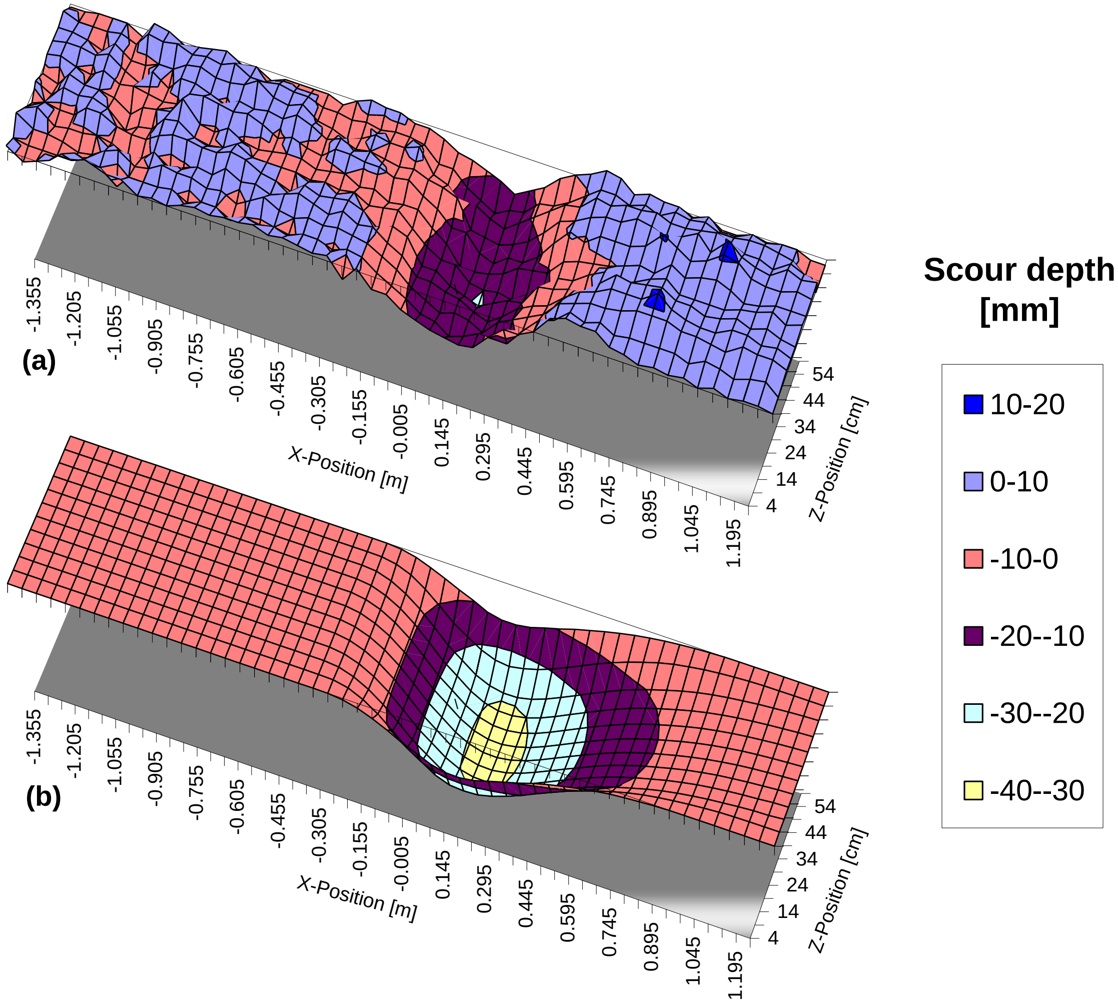
3-D SCOUR CONTOURS

h_b=22.0 cm

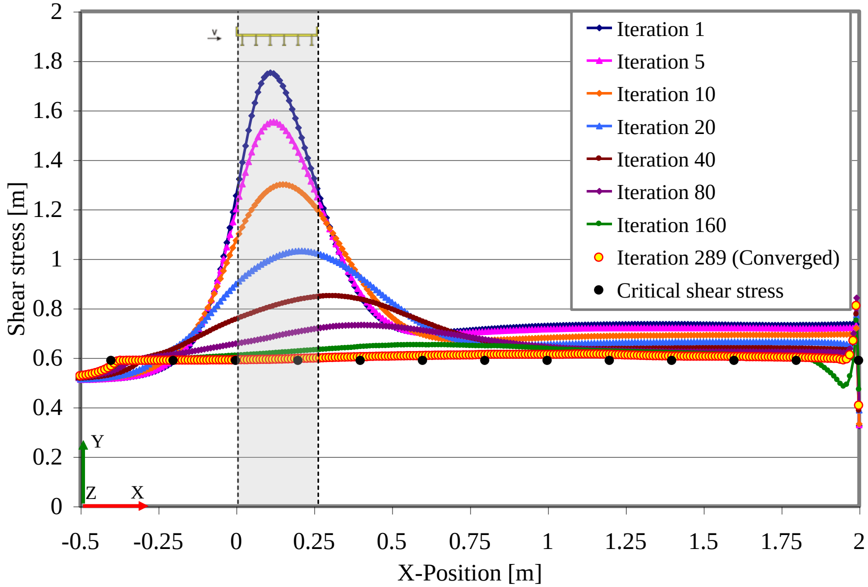
(a) Experimental

(b) Simulation

1 mm sand



Shear Stress and Scour Profiles During Iteration for h_b= 16 cm, 1 mm sand

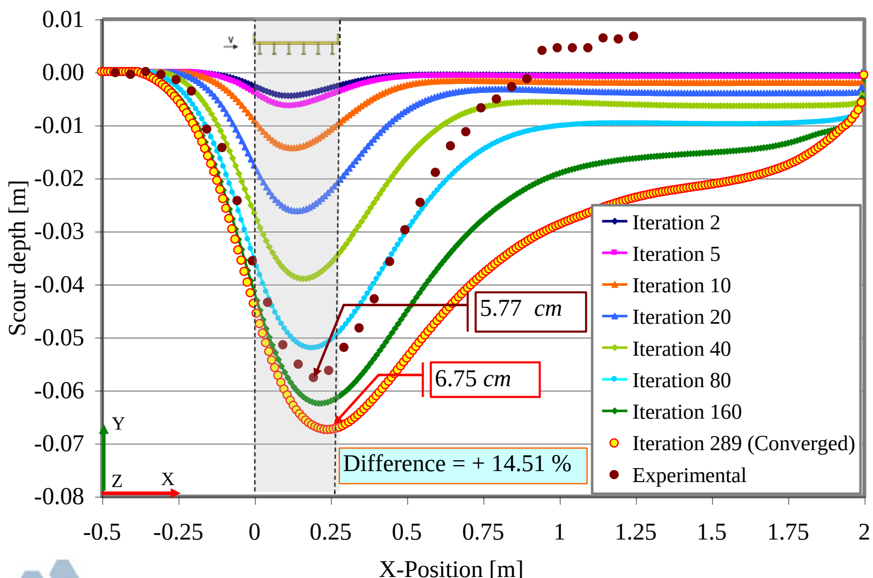


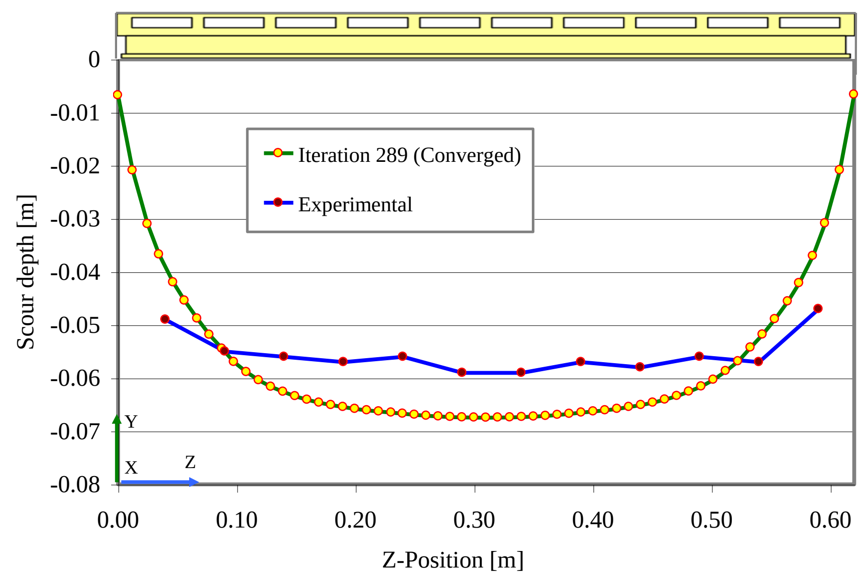
h* = 1.551

Experimental = 5.77 *cm*

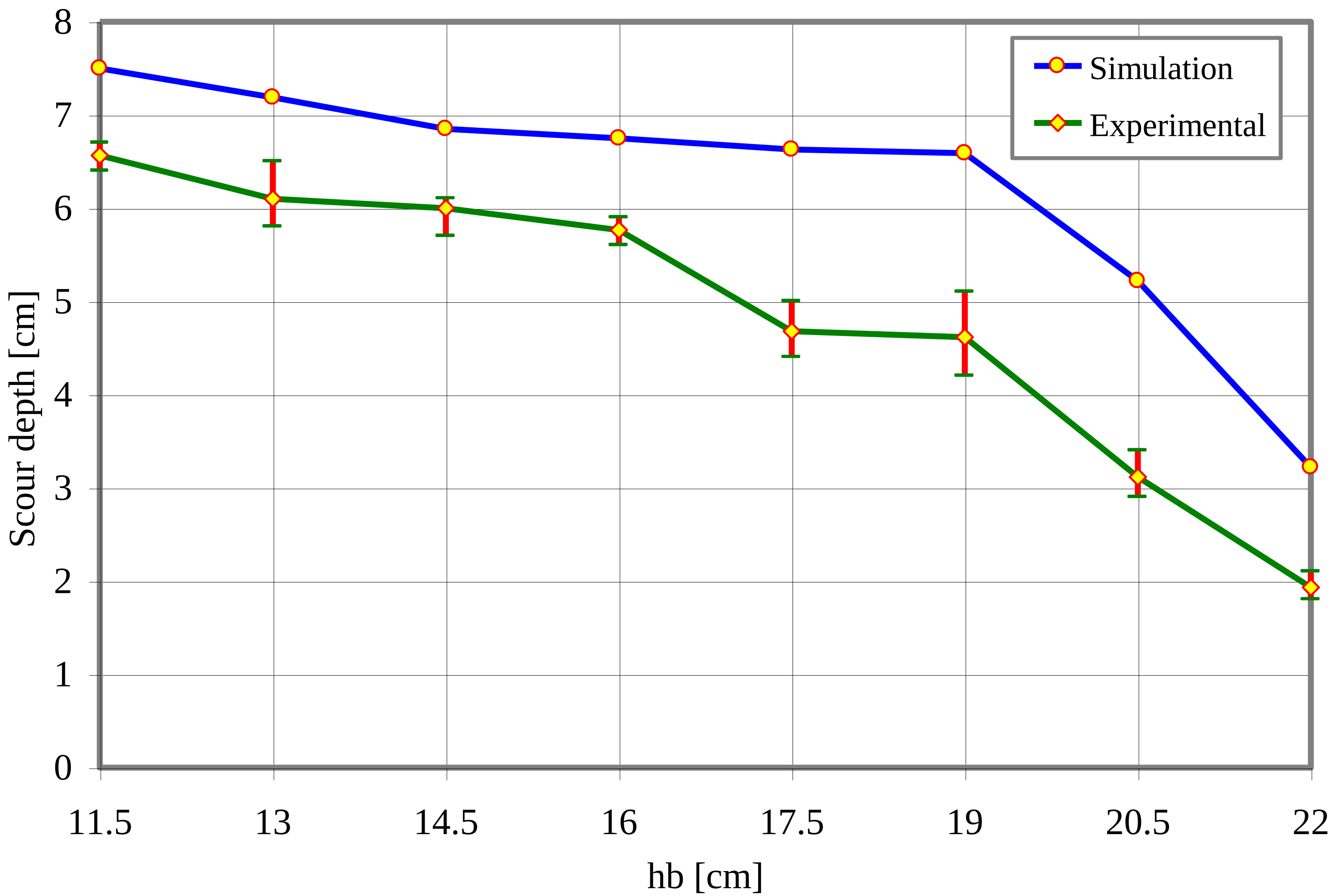
Simulated = 6.75 *cm*

Difference = 14.51%

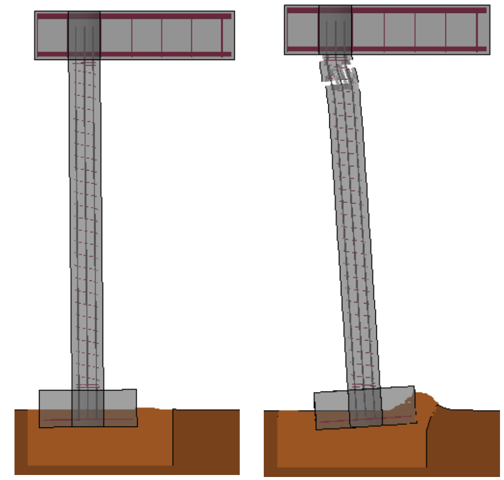




Maximum Scour Depth vs. Bridge Height Above the Bed, h_b (for 1 mm Sand)



Coupled Problems: Soil-Structure Interaction



With the advancement of computational methods and the increase in compute power, engineers can use numerical methods to reconstruct failure scenarios involving coupling between structural mechanics and soil mechanics.

The Oat Ditch Bridge on I15 in California failed from hydraulic loading on support piers during a flood in 2003. Large deformation soil-structural interaction failure analysis was able to capture the failure mode, and can be used to analyze other scour critical bridges.

Summary

- TRACC provides high performance computing resources funded by the Department of Transportation to perform CFD analysis of large problems of interest to the transportation community
- A basic methodology for 3-D scour analysis using commercial CFD software has been developed that can be expanded with more complex physics models
- Comparison with 3-D pressure flow scour experimental data shows expected 3-3-D effects and trends for the simple model physics used in the CFD analysis
- Coupling to structural mechanics and soil mechanics analysis can provide additional valuable information on failure risk