Application of Computational Fluid Dynamics to Optimization of Hydraulic Design and Structure Operation in Everglades Restoration Projects

Jie Zeng, Matahel Ansar, Zubayed Rakib, Seyed Hajimirzaie
Applied Hydraulics, Hydrology and Hydraulics Bureau
South Florida Water Management District

Background

- Great deal of hydraulic designs are carried out in support of the Everglades Restoration Projects. Reduced-scale physical models typically implemented: reliable but costly.
- Computational Fluid Dynamics (CFD): Evaluate and optimize hydraulic performance and design of hydraulic structures in Everglades Restoration projects

Governing equations, NS :

$$\frac{\partial u_j}{\partial x_j} = q$$

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_i u_j) = \frac{\partial}{\partial x_j}[\mu_e(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j})] - \rho g \frac{\partial \zeta}{\partial x_i} + F_i$$

Turbulence model: k-ε closure

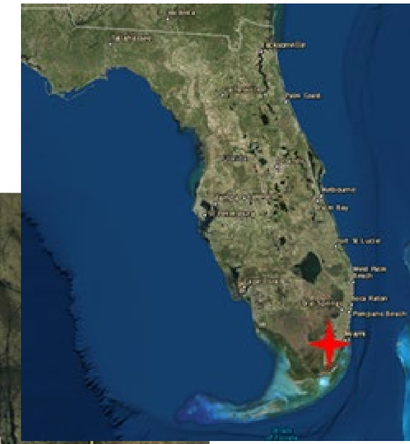
$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_j k) = \frac{\partial}{\partial x_j}(\frac{\mu_e}{\sigma_k}\frac{\partial k}{\partial x_j}) + G_k - \rho\varepsilon$$

$$\frac{\partial(\rho\varepsilon)}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_j \varepsilon) = \frac{\partial}{\partial x_j}(\frac{\mu_e}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial x_j}) + \frac{\varepsilon}{k}(C_1 G_k - C_2\rho\varepsilon)$$

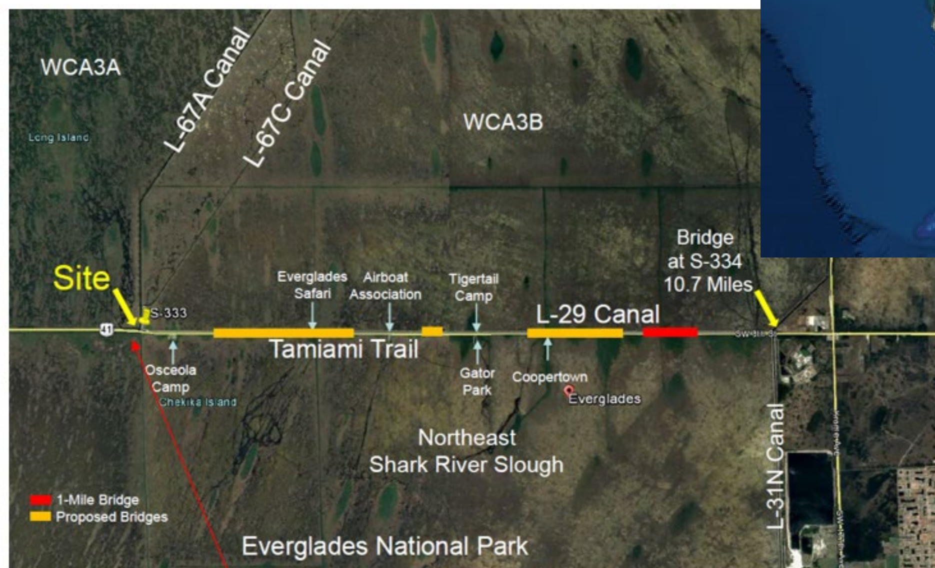
Commercial CFD-software package ANSYS FLUENT

Case Study I:

S333N Spillway Design, Layout and Impact Assessment



- S333 is a trapezoidal-sill reinforced concrete spillway, located at the intersection of L-29 and L67 canals
- Proposed new S333N spillway to accommodate additional discharge



<u>Objective:</u>
Determine the layout, the design, operation criteria, and impact of a newly proposed spillway

Case Study I:
S333N Spillway Design, Layout and Impact Assessment

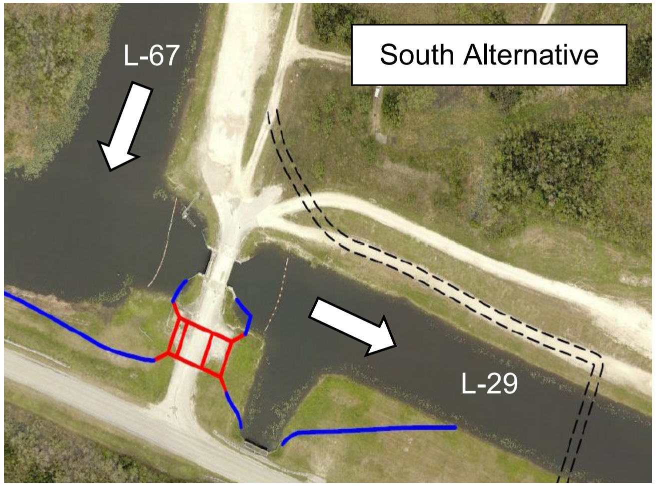
S333N Sizing

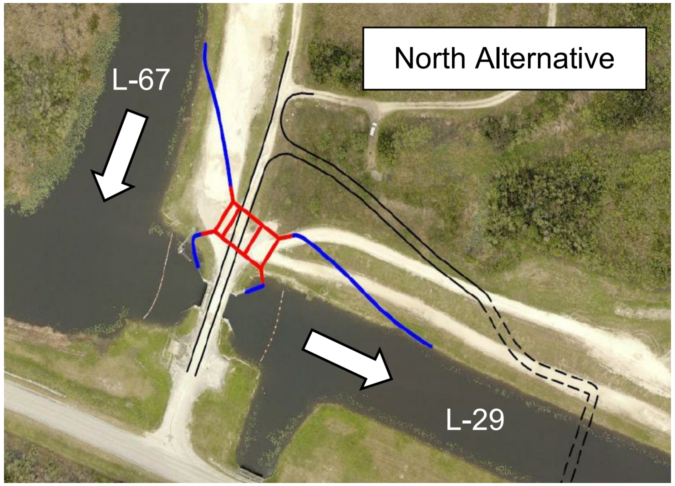
S333 Capacity: 1,350 cfs, One gate 29 ft wide

$$\frac{y_c}{G_0} = a\left(\frac{H-h}{G_0}\right)^b \qquad y_c = \frac{Q^{2/3}}{L^{2/3} g^{1/3}} \qquad \frac{h}{G_0} \geq 1.0$$

S333N Proposed Capacity: 1,150 cfs, Two gates each 14 ft wide
S333N Required gate opening: 2 x 6.40 ft at design HW of 9.5 ft-NGVD, and TW of 9.0 ft-NGVD)

Layout Alternatives

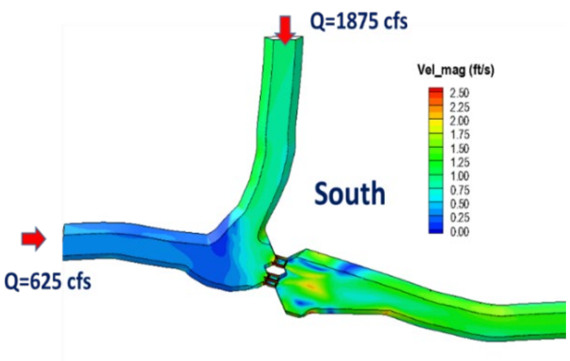


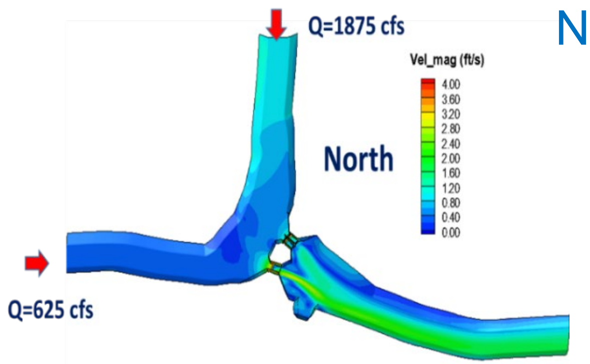


Case Study I:

S333N Spillway Design, Layout and Impact Assessment

Flow Scenario A: 75% flow from L-67





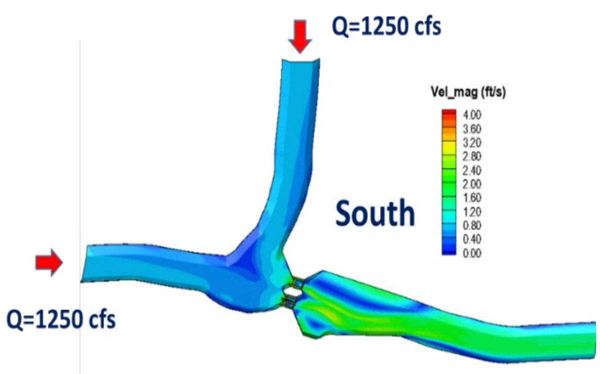
H=9.5 ft, T=9.0 ft

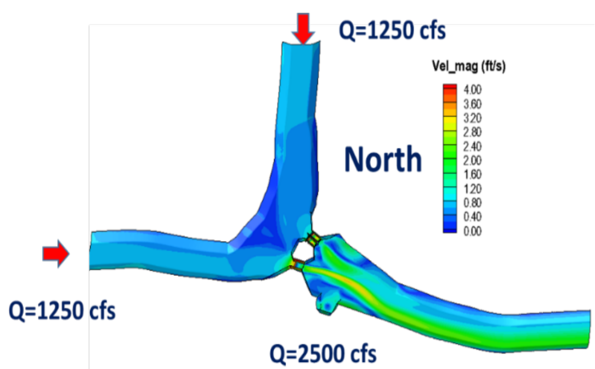
Near Bed Velocity Contours

South alt: 1.0-2.0 ft/s
North alt: 1.0-3.0 ft/s

Limestone layer:
scouring not likely

Flow Scenario B: 50% flow from L-67





South alt: 1.5-3.2 ft/s
North alt: 1.8-3.2 ft/s

Eddy formation downstream,
Flow bias towards east bank in L-29 Canal

Place proposed S333N structure north of the existing structure S333 at angle 25-30 degrees with S333

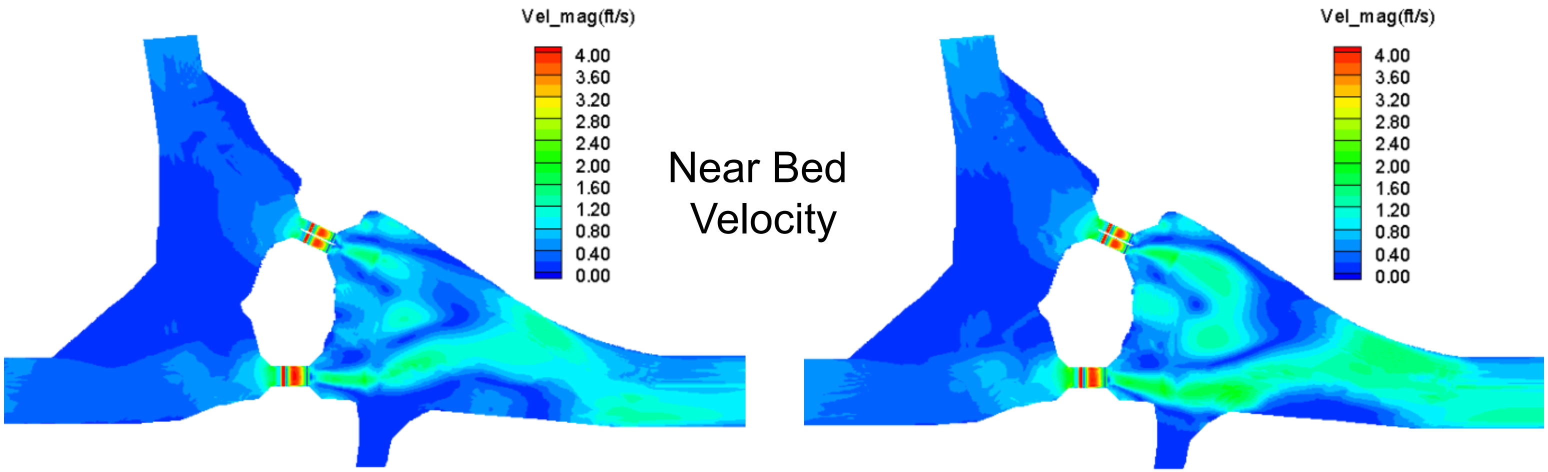
Case Study I:
S333N Spillway Design, Layout and Impact Assessment

Further Analysis

- Extreme scenarios analysis: high flows + low tailwater

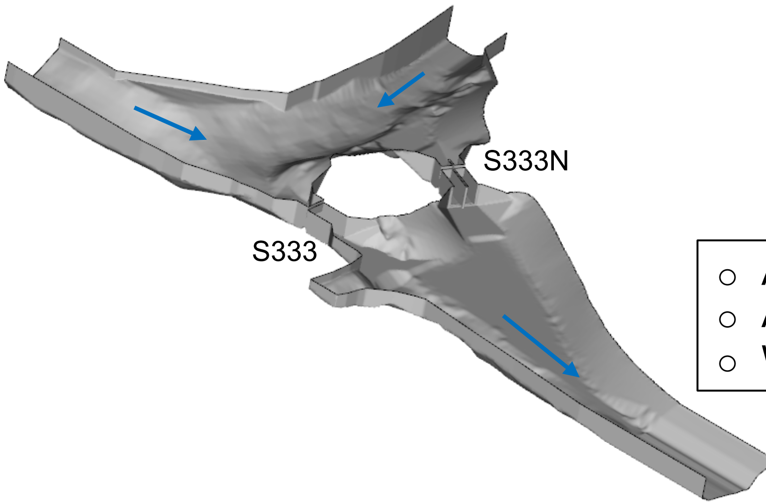
H=10.5 ft, T=8.5 ft,
Q_{333}=1,350 cfs, Q_{333N}=1,150 cfs

H=10.5 ft, T=8.5 ft,
Q_{333}=1,620 cfs, Q_{333N}=1,380 cfs

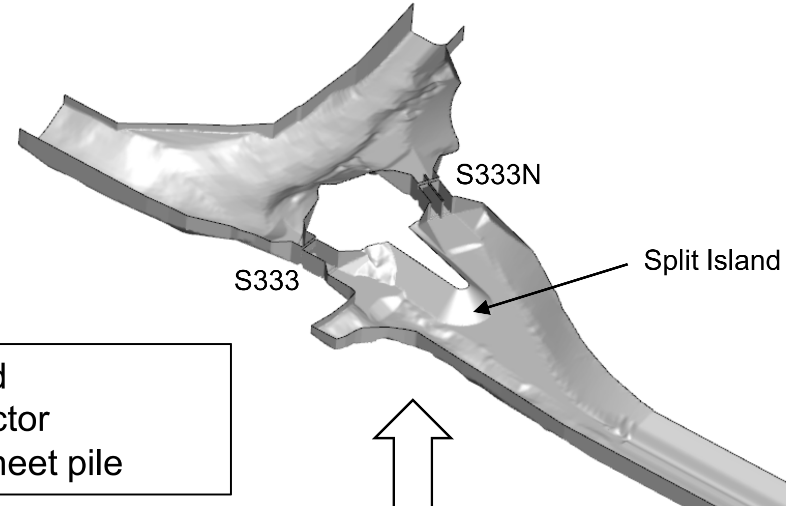


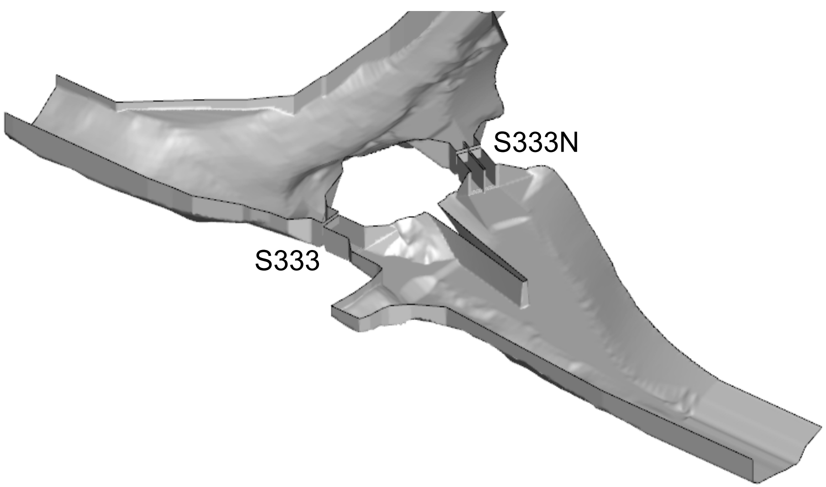
- With the adjusted angle of S333N spillway, flow jets are evenly distributed at downstream, without any severe potentials of eddy formation or scouring
- As conditions became extreme, flow jet downstream of the structures began to oscillates between north and south bank of L-29 Canal

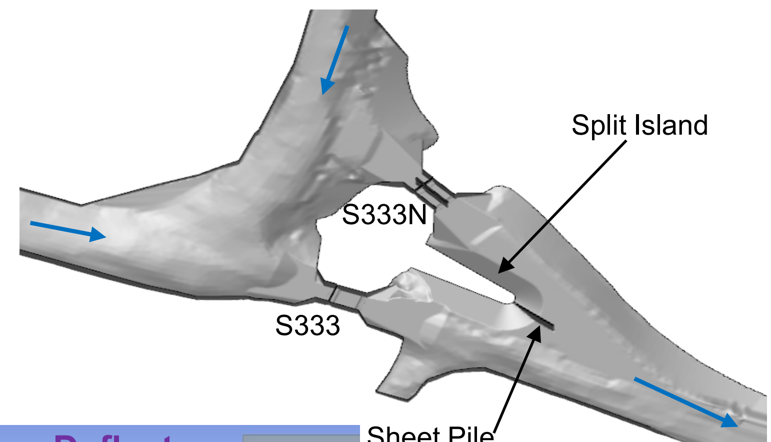
Design Optimization

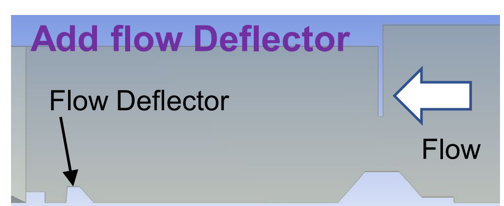


- Add split Island
- Add flow deflector
- With/without sheet pile

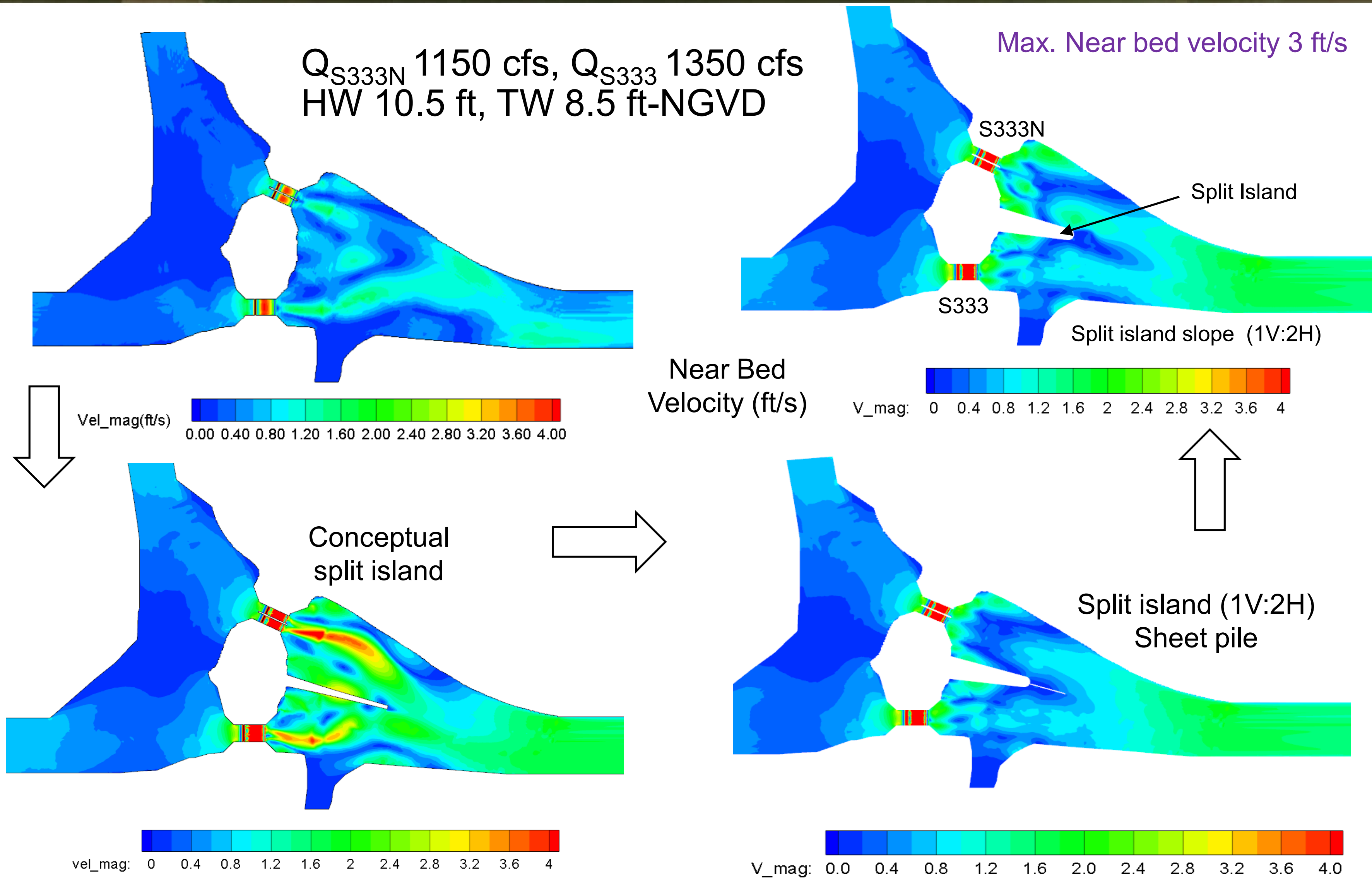




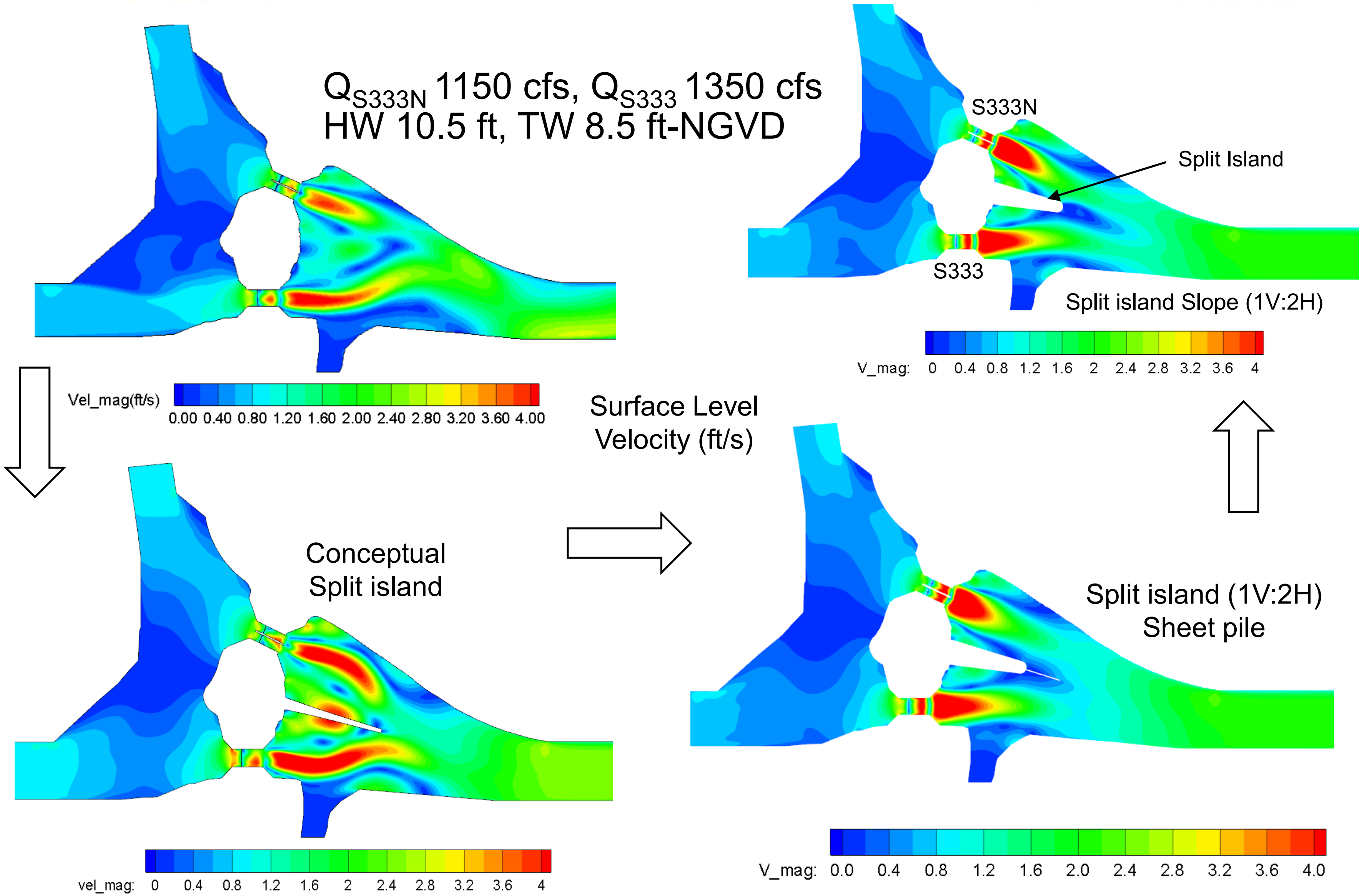




Near Bed Flow Field



Surface Flow Field



Case Study I:

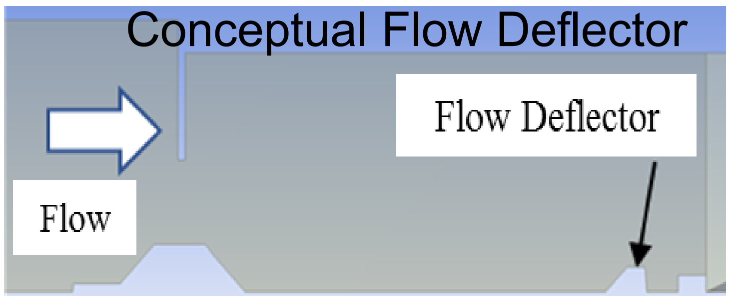
S333N Spillway Design, Layout and Impact Assessment

H=10.5 ft, T=8.5 ft,
Q_{333}=1,350 cfs, Q_{333N}=1,150 cfs

Design Improvements

- Installation of flow deflectors at both end-sills, raised by 1.5-2 ft

(a)



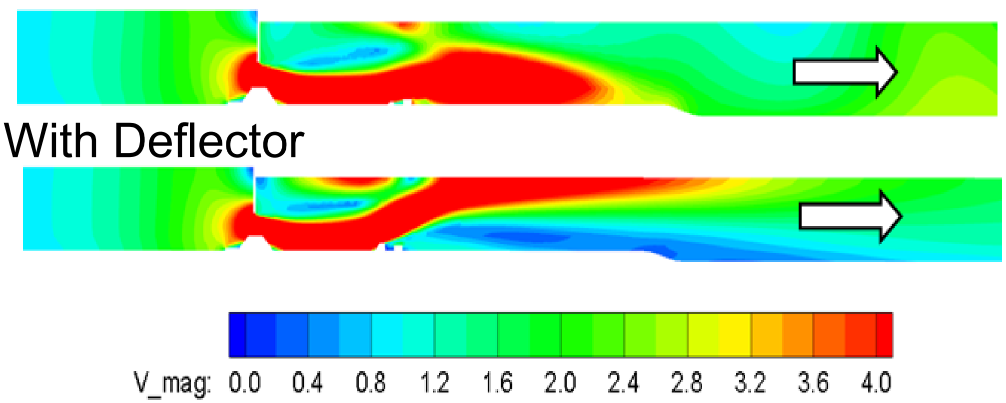
Lateral Velocity Profile (ft/s)

Without Deflector

(b)

With Deflector

(c)



- Flow jet travels longer and expands slower for energy dissipation without the flow deflector
- Deflector directs the discharge upwards, reduces near bed velocities
- Near bed velocities significantly reduce from 4 ft/s to 2.5 ft/s
- Reduces riprap protection requirements in L-29 canal

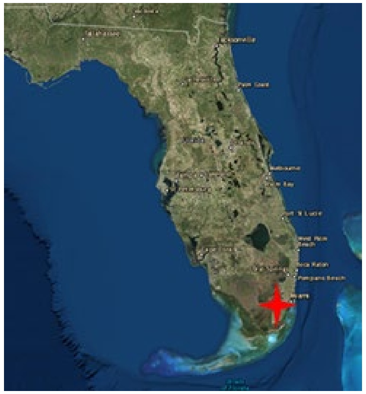
Case Study II:

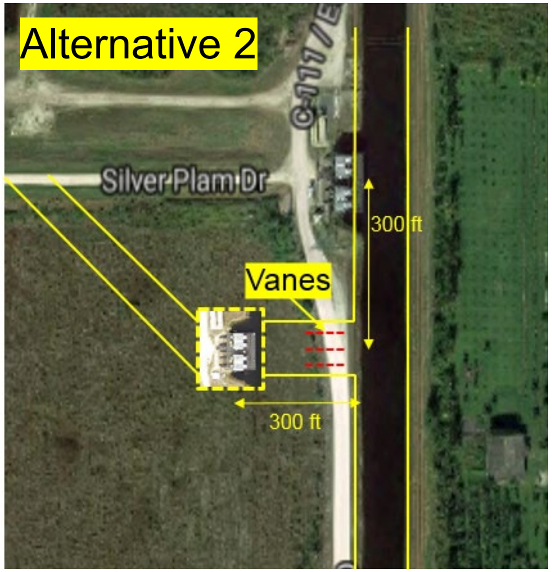
S332B and C Pump Stations Refurbishment Design

- S332B/C pump stations are located south of Pump Station 331, along the L-31N canal
- Construction did not adhere to District standards, meant to be temporary, Frequent repair works
- Inflow canal leading to the pump is oriented at 90º with the pump: flow field biased

S332B Layout Alternatives

1) Move pump Downstream
2) Add vanes
3) Move pump further west



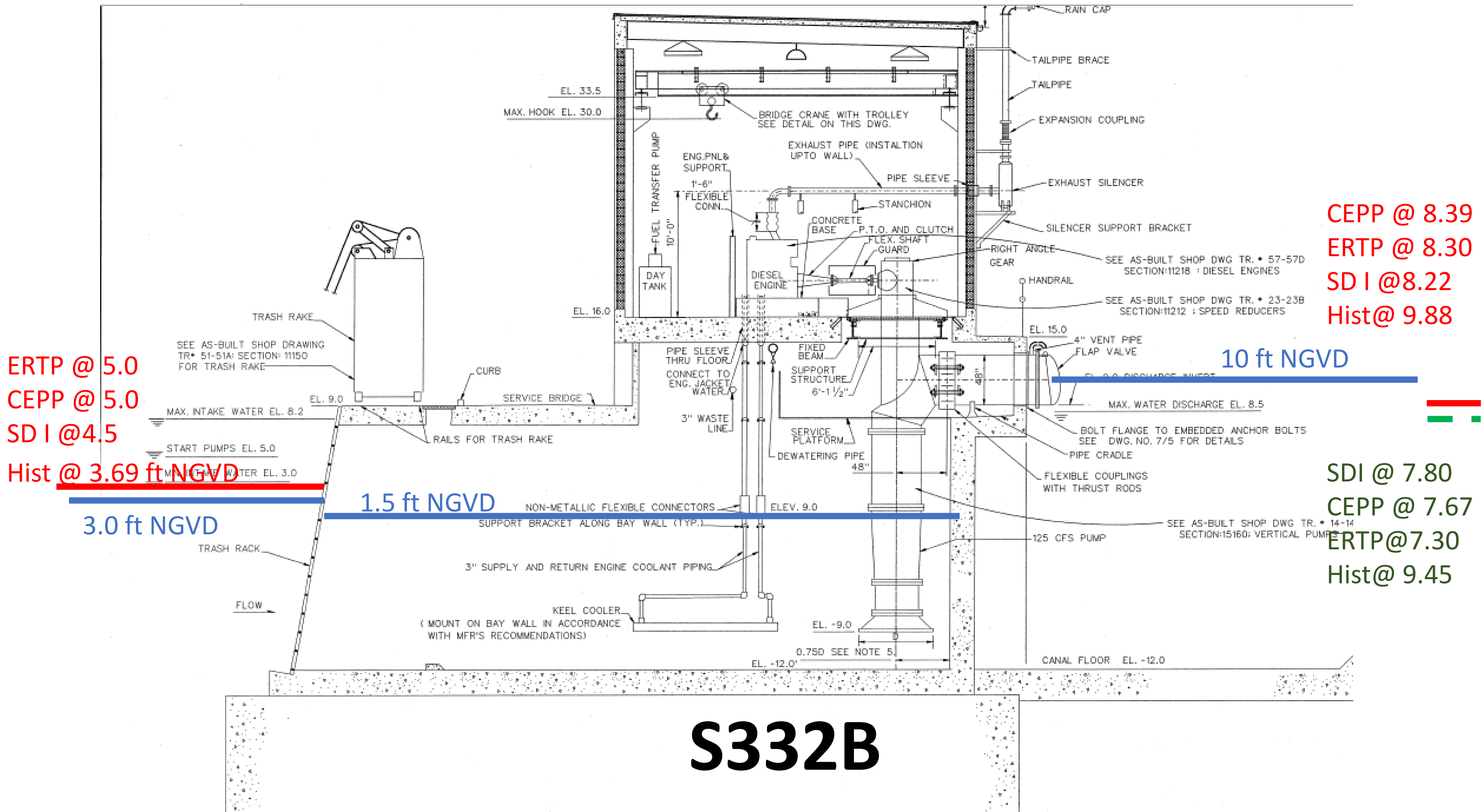


Objective:

Apply CFD model to optimize the refurbishment of S332B/C Pump Stations for improving hydrodynamic performance

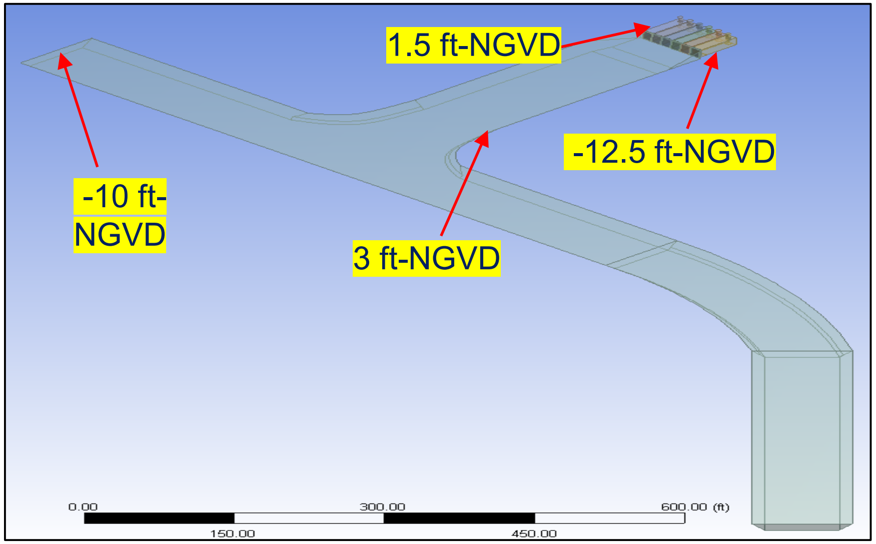
Case Study II:

S332B and C Pump Stations Refurbishment Design



Flow simulation uses H=3.00 ft, T=10.0 ft NGVD

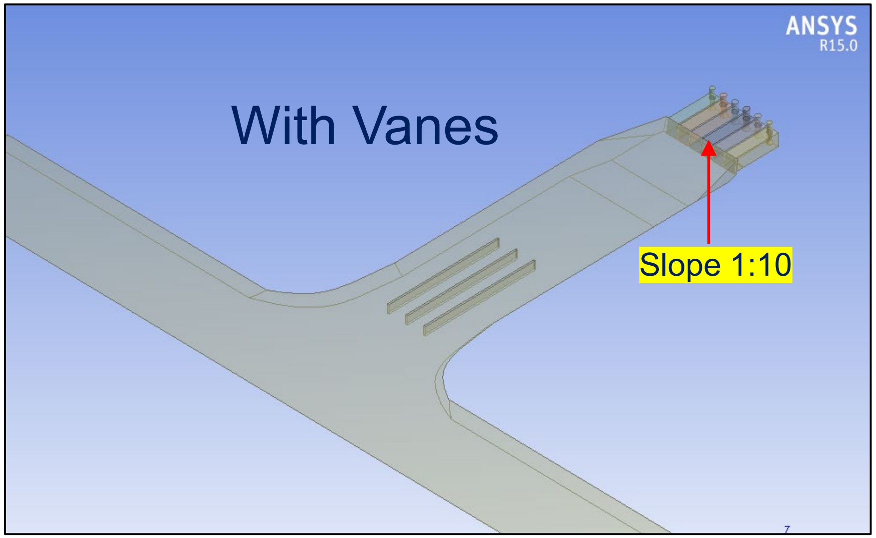
Case Study II:
S332B and C Pump Stations Refurbishment Design

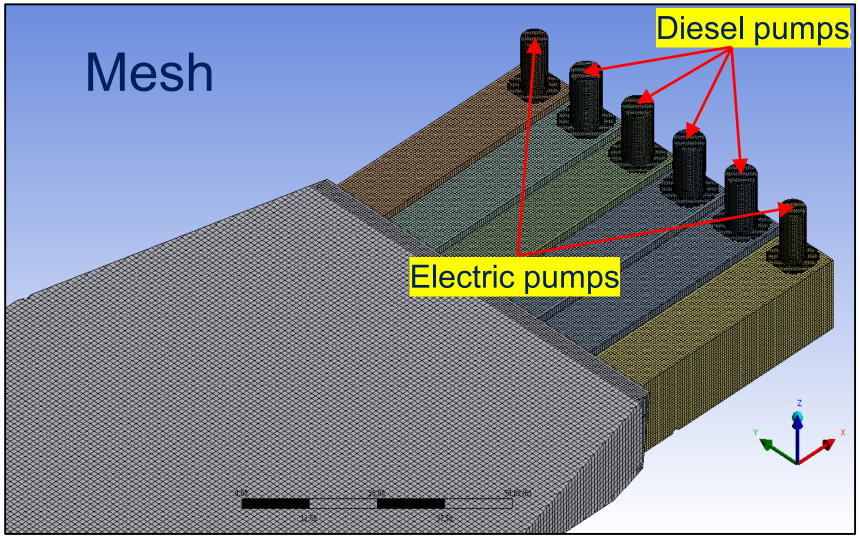


- Canal bottom @ -10 ft-NGVD based on as-builts
- Forebay extended 50 ft
- Slope 1:10 near forebay
- 4 Diesel pumps (125 cfs)
- 2 Electric pumps (75 cfs)

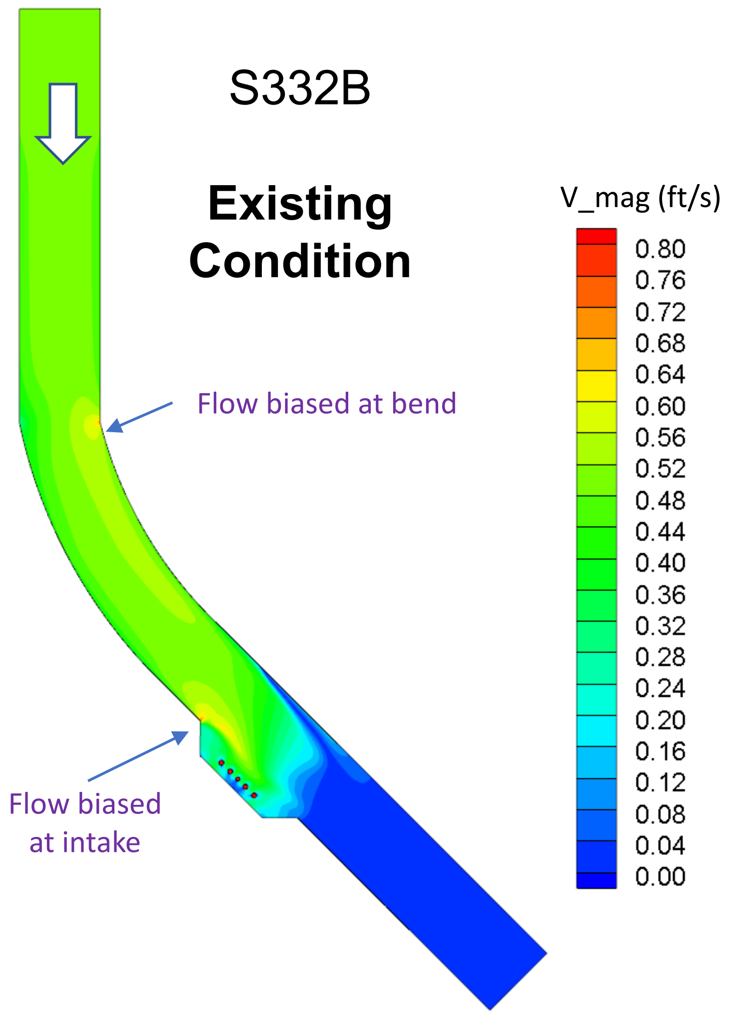
Without Vanes

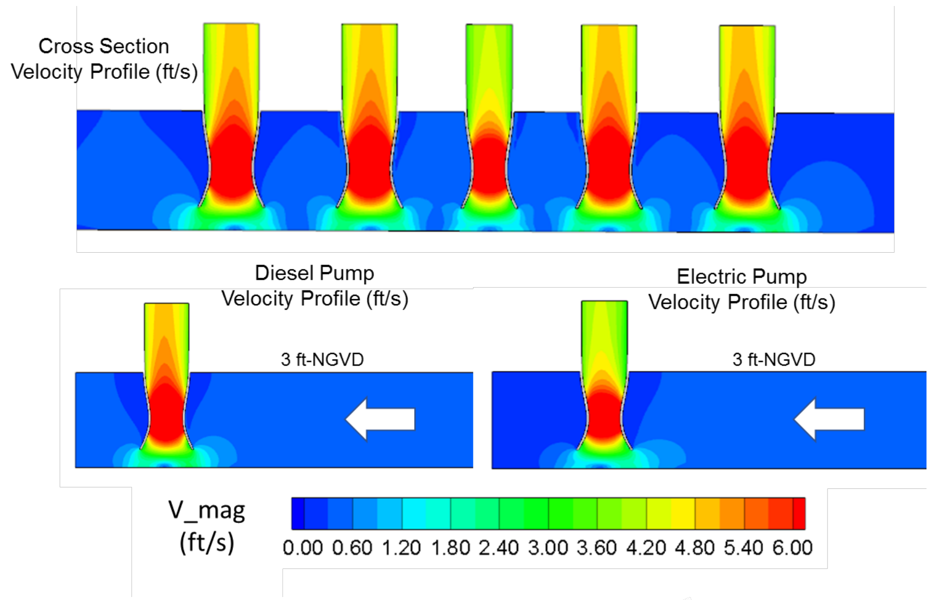
- Design Capacity 650 cfs, the bottom elevation is -12.5 ft NGVD

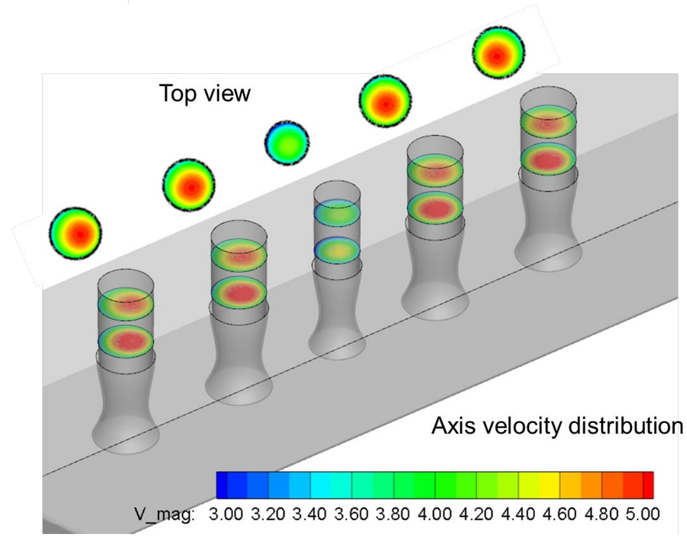




Case Study II:
S332B and C Pump Stations Refurbishment Design



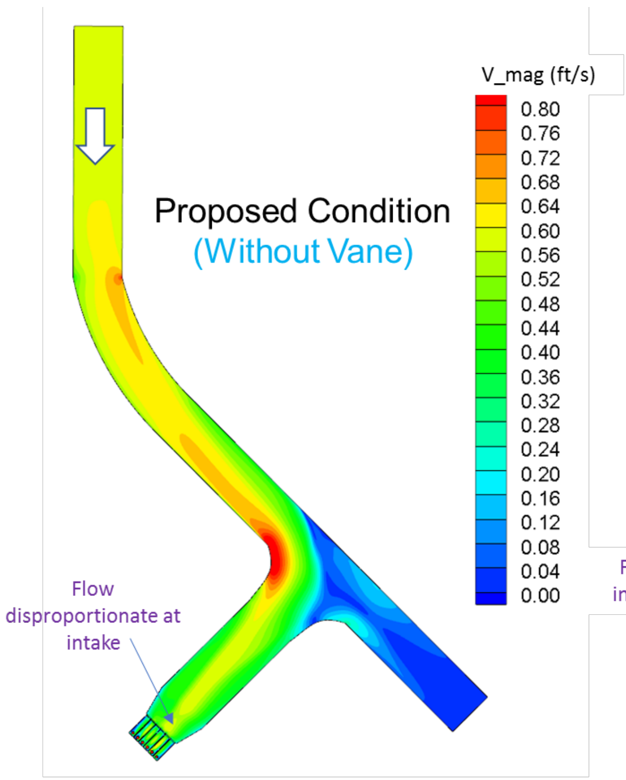


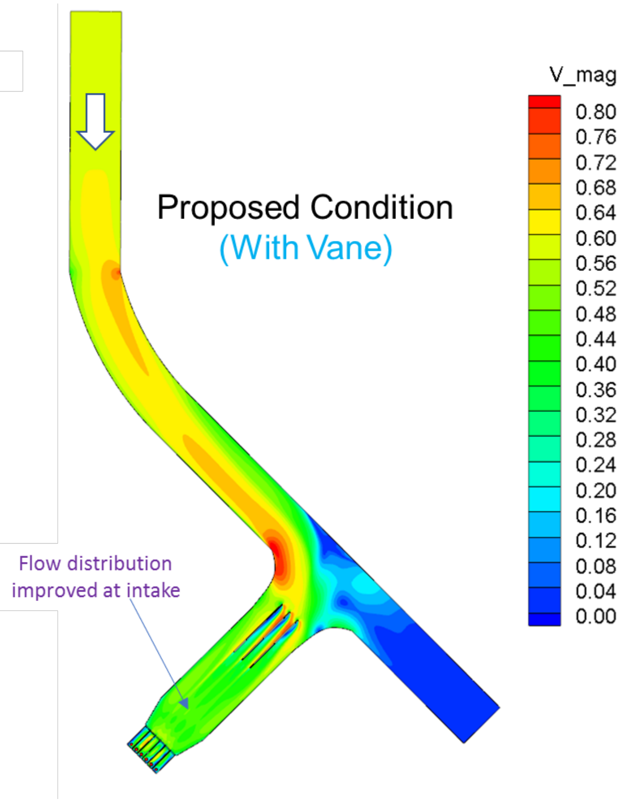


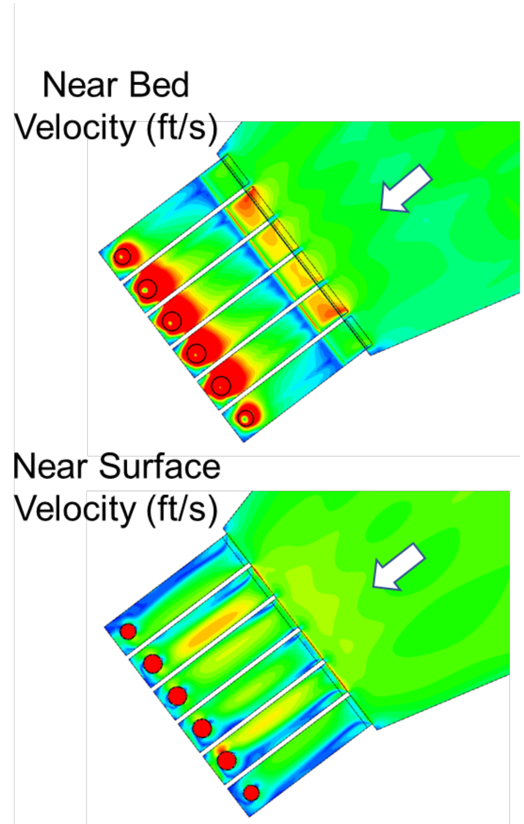
Case Study II:
S332B and C Pump Stations Refurbishment Design

Proposed Condition

Simulation with and Without Vanes





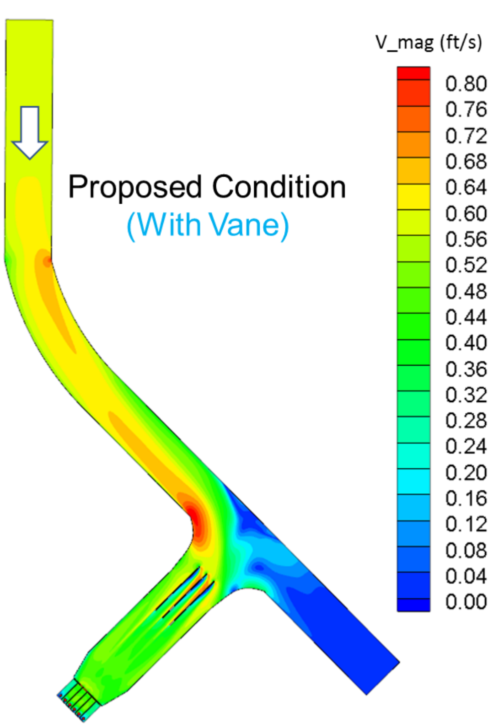


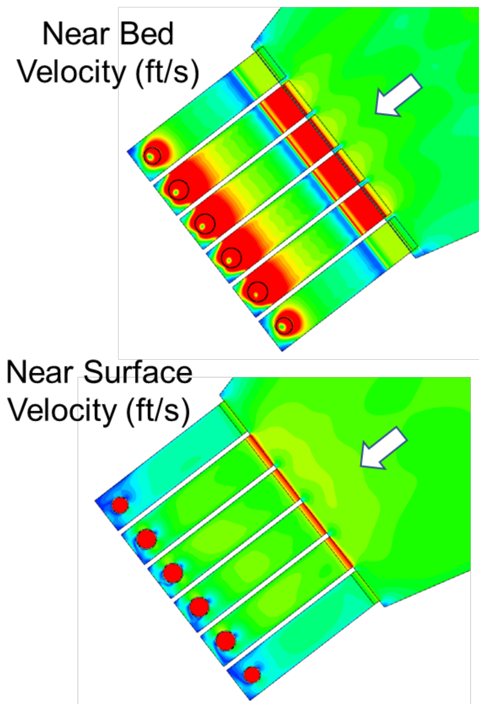
Flow vane improves pump approach flow distributions

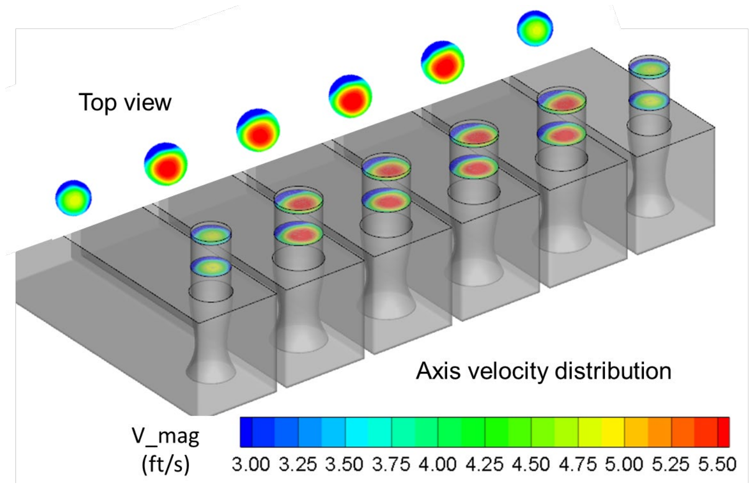
Case Study II:

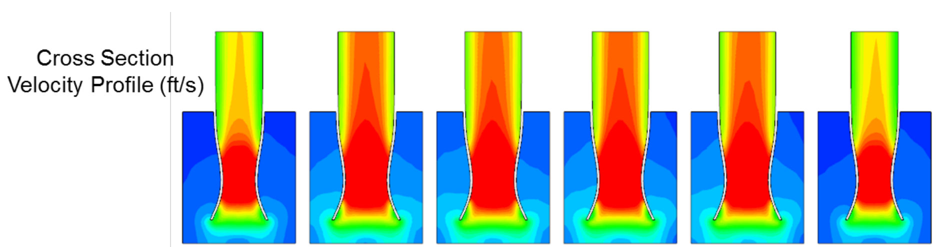
S332B and C Pump Stations Refurbishment Design

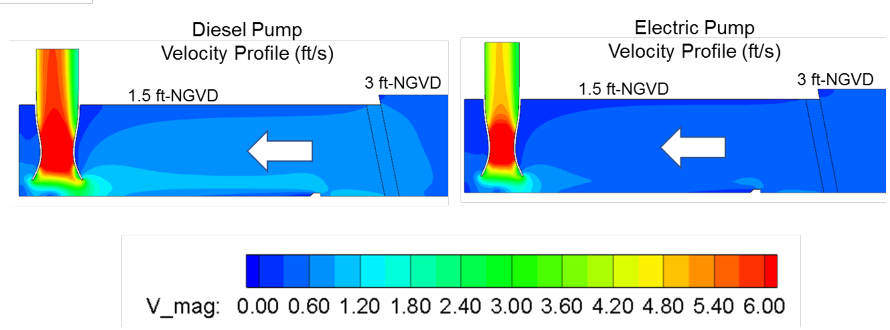
Proposed Condition: Simulation with Vanes and Trash Rack











Summary

- CFD successfully applied to hydraulic analysis of two water control structures in Everglades Restoration Projects
- CFD is used as a complement or alternative to physical model and prototype results
- CFD was systematically used to:
  - Evaluate structure performance and design
  - Predict flow behavior and operation risk
  - Optimize structure design