SUBMISSION ID #
200

Analysis of Alignment of Comite Diversion Channel and Associated Drop Structure

Christopher Denney, Research Engineer, USACE-ERDC-CHL, MS,
Christopher.r.denney@erdc.dren.mil

Extended Abstract

To prevent flooding downstream of the Comite and Amite River Basins, a 12 mile long diversion channel has been designed to divert flow from the Comite River into the Mississippi River. The United States Army Corps of Engineers, New Orleans District has tasked the Engineer Research and Development Center to quantify the effects of changes made to the initial diversion structure design on important flow parameters within the initial channel. A previous three dimensional model using the proprietary software FLOW3D was performed by an independent third party. However, this model features differences in structure shape, a different alignment angle between the planned diversion channel and the Comite River, and introduction of a lateral drainage path introducing additional flows upstream of the first control structure. Therefore, performance of the proposed diversion structure is evaluated by the development of a three dimensional, multiphase computational fluid dynamic model. The model will evaluate the new alignment of the channel by monitoring the amount of flow diverted, and velocity profiles within the diversion channel and subsequent drop structure located just downstream of the initial diversion channel. Special analysis will be placed on the drop structure flow dynamics and the impact of flows from the lateral drainage ditch just upstream of the drop structure. Monitoring of the shear forces on the structure walls and any Eddy formations can help determine the potential danger to sediment buildup within the channel. The support structures for a bridge located just downstream of the drop structure are also included within the model domain to ensure they pose no significant effect to the flow profiles within the drop structure. The model is constructed using the open source OpenFOAM library and will make use of the finite volume solver interFoam which is designed to solve the incompressible Navier-Stokes equations for two isothermal immiscible fluids. The fluid interface is captured using the volume of fluid method. The model will evaluate the steady state results for three different flow discharges which represent potential significant flood events. Significant flow parameters regarding the turbulence coefficients, boundary roughness, and boundary wall functions are perturbed within reasonable limits to help quantify uncertainty associated in the quantities of interest. Grid Independence studies are performed for the three flow parameters to provide further confidence in the results.