Structures in IBER: Bridges, Culverts and Dikes in 2D Hydraulic Simulations

Authors: Georgina Corestein, Enrique Escolano, Ernest Bladé, Luis Cea

References: Bladé, E., Cea, L., Corestein, G., Escolano, E., Puertas, J., Vázquez-Cendón, M. E., Dolz, J., and Coll, A. (2012). "Iber - Herramienta de simulación numérica del flujo en ríos." Revista Internacional de Métodos Numéricos para Cálculo y Diseño en Ingeniería, en Prensa.

Iber (www.iberaula.es)(Bladé et al. 2012) is a numerical model for simulating turbulent free surface unsteady flow and environmental processes in river hydraulics, developed by Flumen Institute (Universitat Politècnica de Catlunya and International Center for Numerical Methods in Engineering) and GEAMA group (Universidade da Coruña). The ranges of application of Iber cover river hydrodynamics, dam-break simulations, flood hazard assessment, sediment transport processes and wave flows in estuaries. Iber has 3 main computational modules: a hydrodynamic module, a turbulence module and a sediment transport module. All of them use the finite volume method on non-structured meshes made up of triangles or quadrilaterals. In the hydrodynamic module, which is the base of Iber, the depth averaged two-dimensional shallow water equations are solved.

A key issue for a correct hydrodynamic simulation, for example for flood hazard or flood risk assessment, is the incorporation of special structures like bridges, culverts and dykes. These structures usually are not included in de Digital Terrain Models (DTM) that are used as a base for the mesh generation. The reason is that most times these structures have a sub-grid size, but also because the common DTM file formats, with a raster structure, do not allow to include the details of such structures.

In version 2 of Iber new methods have been included to incorporate these kind of structures and the conditions defining their properties in a friendly and detailed way. As for bridges, in previous versions they were modelled as a combination of a weir and a gate, which could work in submerged or unsubmerged flow, through an internal condition. An internal condition is a condition assigned to pre-existing mesh element sides. For that reason, when generating the mesh some tedious geometry or mesh editions had to be performed. The new method allows defining the bridge position and bridge conditions after the generation of the mesh, and it automatically adapts the mesh to include the bridge line as a series of element sides with the corresponding conditions.

Culverts represent a flow bypass between elements that are not contiguous. For that reason, culverts are not included as a condition, but specific tcl code has been developed to consider them. A culvert is defined with a specific window in which it is possible to introduce the culvert ends coordinates and its main properties (roughness coefficient, cross section shape and dimensions, end elevations) and it is automatically drawn on the model (geometry or mesh) and when the simulation is run a special file with all this data is written and subsequently read by the execution file.