Flow Analysis of Drain Structure with Flow-3D

2017. 10

GS E&C Research Institute
Contents

1. Project Overview
2. Computational Domain
3. CFD Result
4. Future Works
1. Project Overview

- Project: OOMW Class CCPP
- Contract Price: OOO Billion
- Location: Chungnam OO Si
- Contract type: EPC Turnkey
- Technical Support: Discharge Box & Seal Pit CFD Analysis
2. Computational Domain

Structure Dimension

1580m

375m

25m
2. Computational Domain

**Structure Dimension**

On Shore

Off Shore

Discharge Drain #1

Discharge Drain #2

Weir
2. Computational Domain

**Computational Mesh**

- The real structure has been modeled with 8 blocks that consist of cubic meshes of 0.2 m.
- Some mesh blocks have been stretched larger than 0.2 m to reduce the number of meshes.

**Weir & Onshore**

Total Number of cell: 50,687,658  
Total Number of fluid cell: 3,719,915  
Total Number of solid cell: 1,219,568  
Max aspect ratio: 2.5
2. Computational Domain

- The real structure has been modeled with 8 blocks that consist of cubic meshes of 0.2 m.
- Some mesh blocks have been stretched larger than 0.2 m to reduce the number of meshes.

**Computational Mesh**

- Mesh size set properly
- Mesh size set improperly

**Weir & Onshore**

- Mesh size = 0.2
- Mesh size set properly
- Mesh size set improperly

Total Number of cell: 50,687,658
Total Number of fluid cell: 3,719,915
Total Number of solid cell: 1,219,568
Max aspect ratio: 2.5
2. Computational Domain

**Boundary Condition**

- Basically, boundary condition of inter-mesh blocks is set to “Symmetry”.
- The boundary conditions of discharge drain are set to “Volume Flow Rate” separately, because of different flow rate.
2. Computational Domain

**Boundary Condition**

- The boundary condition of outflow position is set to “Specified Pressure” and the mesh block becomes indirect to the structure for the stability of simulation.
2. Computational Domain

**Initial Condition**

- In order to prevent surge of water at the early stage of calculation, hydrostatic pressure is applied to entire structure.
- The hydrostatic pressure in the structure is applied simply \( P = \rho gh \), because the ends of discharge channel are exposed to the atmosphere.
2. Computational Domain

**Case Study**

- To analyze the effect of the level of seawater, 3 types of seawater level are applied separately.

---

**High Water Level**

**Medium Water Level**

**Low Water Level**
3. CFD Result

**Drain Flow Rate**

- The largest difference between in-flow and out-flow of each mesh block is less than 1%, that means this simulation is physically reasonable.

<table>
<thead>
<tr>
<th>Mesh Block</th>
<th>HWL</th>
<th></th>
<th></th>
<th>MWL</th>
<th></th>
<th></th>
<th>LWL</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1&amp;2</td>
<td>0.0%</td>
<td>0.0%</td>
<td>0.0%</td>
<td>0.0%</td>
<td>0.0%</td>
<td>0.0%</td>
<td>0.0%</td>
<td>0.0%</td>
<td>0.0%</td>
</tr>
<tr>
<td>3</td>
<td>0.0%</td>
<td>0.0%</td>
<td>0.0%</td>
<td>0.0%</td>
<td>0.1%</td>
<td>-0.1%</td>
<td>0.0%</td>
<td>-0.9%</td>
<td>0.9%</td>
</tr>
<tr>
<td>4</td>
<td>0.0%</td>
<td>0.0%</td>
<td>0.0%</td>
<td>0.1%</td>
<td>0.1%</td>
<td>0.0%</td>
<td>-0.9%</td>
<td>-0.6%</td>
<td>-0.4%</td>
</tr>
<tr>
<td>5</td>
<td>0.0%</td>
<td>0.0%</td>
<td>0.0%</td>
<td>0.1%</td>
<td>0.0%</td>
<td>0.0%</td>
<td>-0.6%</td>
<td>-0.7%</td>
<td>0.1%</td>
</tr>
<tr>
<td>6</td>
<td>0.0%</td>
<td>-0.1%</td>
<td>0.0%</td>
<td>0.0%</td>
<td>0.0%</td>
<td>0.0%</td>
<td>-0.7%</td>
<td>-0.3%</td>
<td>-0.4%</td>
</tr>
<tr>
<td>7</td>
<td>-0.1%</td>
<td>0.9%</td>
<td>-0.9%</td>
<td>0.0%</td>
<td>0.9%</td>
<td>-0.8%</td>
<td>-0.3%</td>
<td>0.3%</td>
<td>-0.6%</td>
</tr>
<tr>
<td>8</td>
<td>0.9%</td>
<td>0.9%</td>
<td>-0.1%</td>
<td>0.9%</td>
<td>1.0%</td>
<td>-0.1%</td>
<td>0.3%</td>
<td>-0.3%</td>
<td>0.6%</td>
</tr>
</tbody>
</table>
3. CFD Result

**Flow velocity**

- The flow velocity from the difference of hydraulic head between final & initial condition is almost identical with the velocity calculated from FLOW-3D, which means the boundary condition and initial condition are correctly set.

<table>
<thead>
<tr>
<th></th>
<th>HWL</th>
<th>MWL</th>
<th>LWL</th>
</tr>
</thead>
<tbody>
<tr>
<td>Difference of Hydraulic Head</td>
<td>0.12</td>
<td>0.13</td>
<td>0.17</td>
</tr>
<tr>
<td>between final &amp; Initial Condition</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Depth averaged velocity from FLOW-3D</td>
<td>1.52</td>
<td>1.58</td>
<td>1.83</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Difference between velocity from</td>
<td>0.00</td>
<td>0.00</td>
<td>0.01</td>
</tr>
<tr>
<td>hydraulic head difference and velocity calculated from FLOW-3D</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
3. CFD Result

Pressure in the structure

• The pressure in the structure is proportional to the level of seawater from 0 sec to 300 sec.
3. CFD Result

Pressure in the structure

- The pressure in the structure is proportional to the level of seawater from 0 sec to 300 sec.
3. CFD Result

**Depth Averaged Velocity**

- Depth averaged velocity of each case is different according to the level of water inside the discharge box.

- **High Water Level**
  - Time: 300s

- **Medium Water Level**
  - Time: 300s

- **Low Water Level**
  - Time: 300s
3. CFD Result

**Movies**

- These movies show that the pressure and the depth averaged velocity in the drain structure.

**High Water Level**
3. CFD Result

**Movies**

- These movies show that the pressure and the depth averaged velocity in the drain structure.

**Medium Water Level**
3. CFD Result

Movies

- These movies show that the pressure and the depth averaged velocity in the drain structure.

Low Water Level
4. Future Works

- There need to find more stable way to solve in order to prevent surge of water.
- The calculation time of these simulation is too long to get the steady state, so it is necessary to find the way to solve faster.
- The change of the drain water flow in the structure need to be shown, with respect to the change of seawater level between High Water Level and Low Water Level.
- The comparison with the result of other CFD code will help to show advantages of using FLOW-3D, especially for modeling the complex geometry and generating meshes.