# VTP: An open-source software to determine the correction factor

## 1 Introduction

VTP is the open-source software which aims to determine the correction factor of the Vimoke—Taylor concept for 3D models of prefabricated vertical drains (PVDs) systems. The theory behinds the software is mentioned in the manuscript named *"Application of the Vimoke–Taylor concept for fully coupled models of consolidation by prefabricated vertical drains "*. VTP uses mesh information, boundary conditions and nodal loads from other software (ANSYS, ABAQUS…) as input files.

VTP was developed using C++ programming language with Qt framework. To recompile the code or to add more features, the following components need to be installed:

- Qt-5 (static version for FEFLOW Plug-in): https://www.qt.io/download
- Eigen library: http://eigen.tuxfamily.org/index.php?title=Main_Page
- Intel MKL: https://software.intel.com/en-us/mkl
- Visual Studio C++ or g++ as the compiler

For the Windows end-users, *VTP.exe* can be run directly without any configuration.

## 2 VTP: Main features

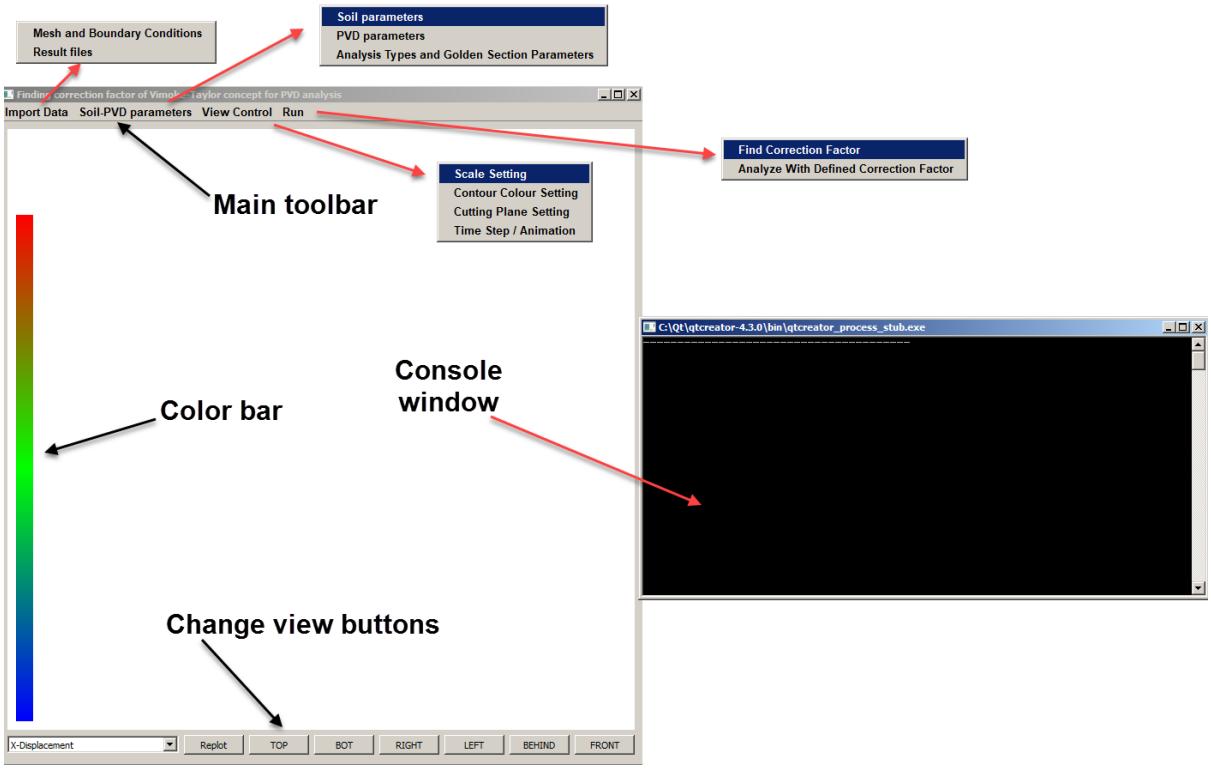



***Figure 1**: Main window and the toolbar of VTP*

VTP (**Figure 1**) includes:

- The main window shows the mesh, and the results (displacements in x, y, z-direction and the pore pressure).
- The console window shows more information. It mainly can be used for debugging.
- The main toolbar contains all the program functions that are: (1) Import Data, (2) Set up the parameters of PVDs and soils, (3) Set up 3D view, (4) Determine the correction factor.

## 2.1 Import Data

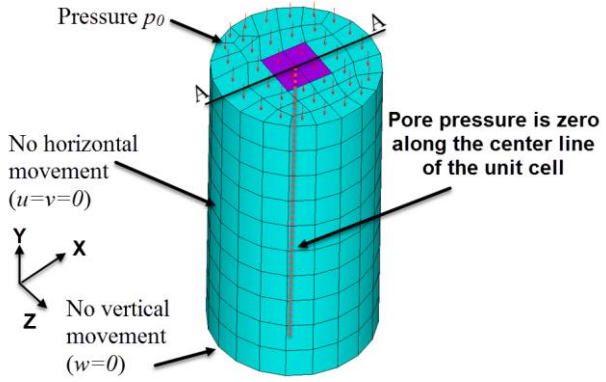



*Figure 2: Unit cell model to determine the correction factor*

Figure 2 presents a unit cell model that can be created by any software. The unit cell model has two different material types that represent the drained zone (purple colour) and the soil zone (cyan colour). The top is assigned a reference vertical pressure of 100 kPa. The bottom boundary is fixed in Y-direction, and the outer boundary is fixed in both X and Z directions. Along the centre line of the unit cell, the pore pressure is zero (drained boundary condition). The mesh and the boundary condition information of the above model then is exported to ASCII file used for VTP. VTP requires the following file to perform a simulation (**Figure 4**):

- Coordinate file (*coordinates.dat)* that has four columns: (1)-Node number, (2)-X coordinate, (3)-Y coordinate and (4)-Z coordinate (Figure 5).
- Element file (*elements.dat*) that has 14 columns. Depending on the element types, details about the element file is presented in section 3.
- Dirichlet boundary conditions in x,y,z-direction are *fix.dat*, *fixy.dat* and *fixz.dat,* respectively. Each file has two columns: (1)-Node number, (2)-Boundary condition value.
- Similarly, *fixh.dat* contains boundary conditions of the pore pressure.
- *forcey.dat* file has two columns. The first column is the node numbers belonging to the top surface. The second column is the nodal force that is converted from 100 kPa vertical pressure on the top surface.

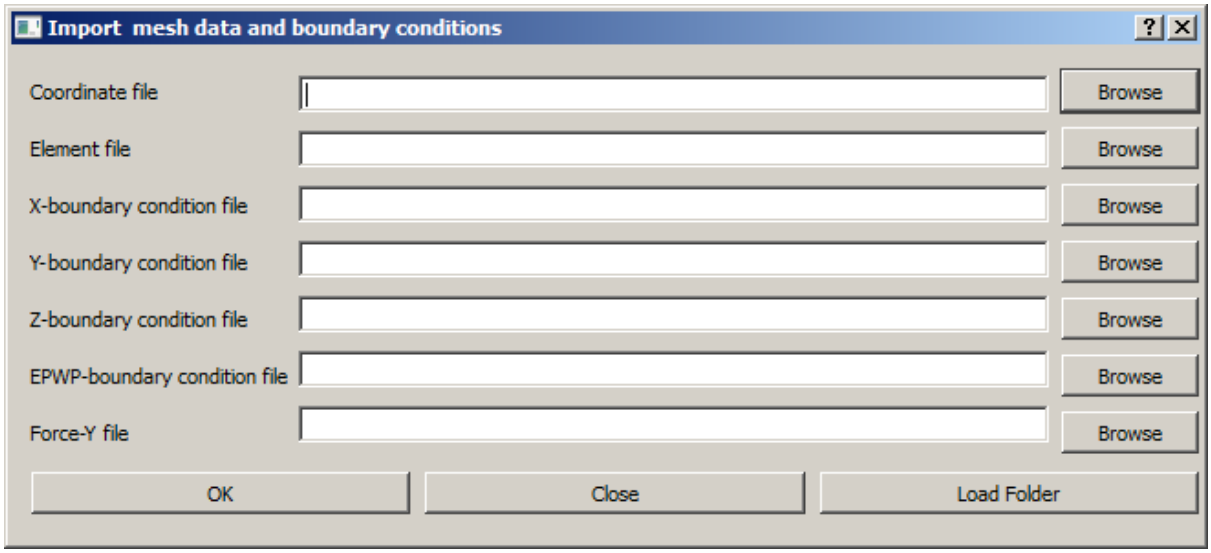



***Figure 3**: Import mesh and boundary condition data*


***Figure 4:*** *Required files for VTP*

| Node number | X-Coordinate | Y-Coordinate | Z-Coordinate |
|---|---|---|---|
| 1 | 0.00000E+00 | 0.50000E+01 | 0.00000E+00 |
| 2 | -0.15000E+00 | 0.50000E+01 | -0.15000E+00 |
| 3 | -0.75000E-01 | 0.50000E+01 | -0.75000E-01 |
| 4 | 0.15000E+00 | 0.50000E+01 | -0.15000E+00 |

*Figure 5: Example of coordinates.dat*

| Node number | Boundary condition value |
|---|---|
| 1 | 0.00000E+00 |
| 155 | 0.00000E+00 |
| 156 | 0.00000E+00 |
| 157 | 0.00000E+00 |

*Figure 6: Example of fixx.dat, fixy.dat, fixz.dat and fixh.dat*

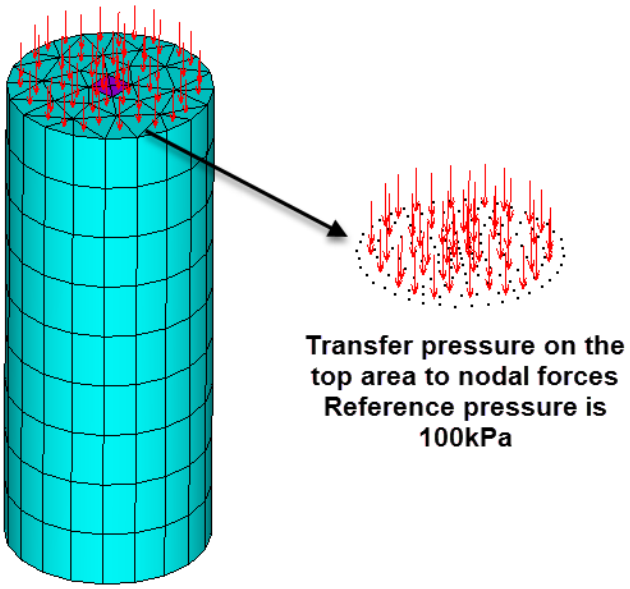



*Figure 7: Transfer pressure on the top pressure to nodal forces with the reference pressure is 100 kPa.*

In Figure 8, for result visualisation, VTP can import the nodal result files (displacement of X, Y, Z-direction and the pore pressure).

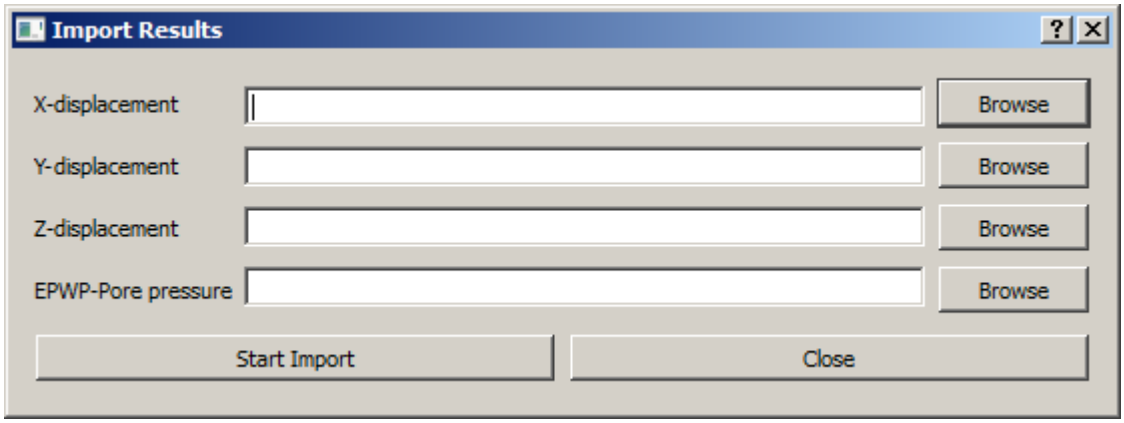



*Figure 8: Import result data*

## 2.2 Input soil parameters

The requirement soil parameters are (Figure 9):

- The vertical hydraulic conductivity $k_v$ (m/s).
- The ratio between the horizontal hydraulic conductivity and the vertical hydraulic conductivity.
- The ratio between the horizontal hydraulic conductivity of the smear zone and the undisturbed zone.
- The void ratio.
- The horizontal consolidation coefficient $c_h$ ($m^2/s$). Alternatively, the bulk modulus $K$ ($kN/m^2$) can be inputted instead.
- Finally, the Poisson's ratio.

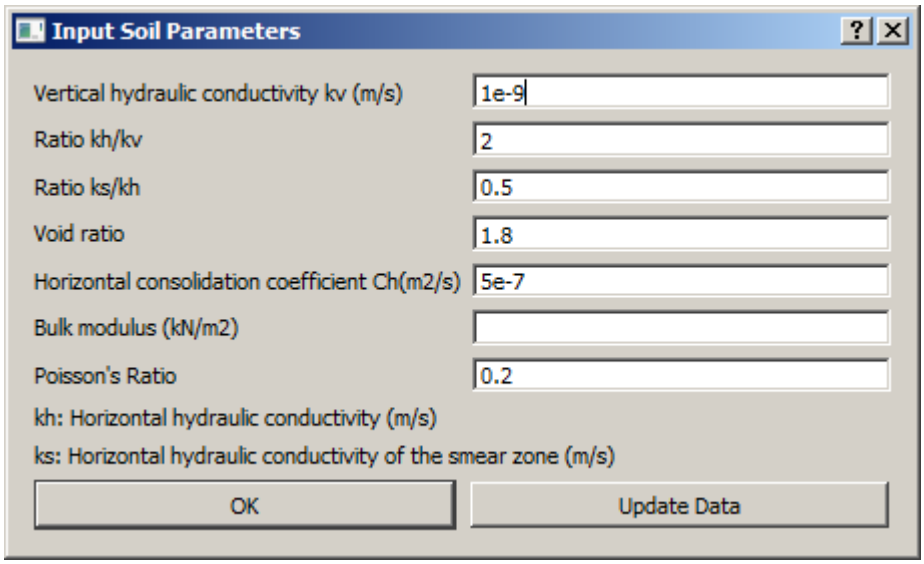



*Figure 9: Input soil parameters*

## 2.3 Input PVD parameters

Figure 10 shows the necessary information for the PVD:

- The unit cell radius $r_e$ (m) calculated from the installation pattern of PVDs.
- The equivalent PVD radius $r_w$ (m) calculated from sizes of the PVD.
- The equivalent radius of the smear zone $r_s$ (m).
- If the well resistance is considered, the PVD discharge capacity $q_w$ ($m^3/s$) and the unit cell height $H$ (m) is also needed.

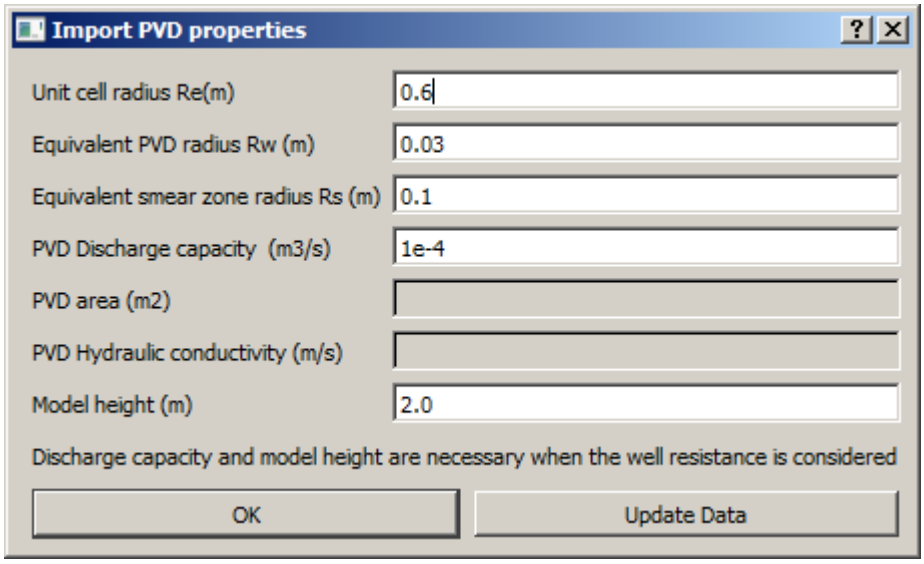



*Figure 10: Input PVD parameters*

### 2.4 Analytical solutions and the golden section method

VTP calculates the pore pressure of any node at any given time using the analytical solutions from either Barron or Hansbo. The golden section method is used to minimise the error between numerical results and the analytical results by adjusting the correction factor $C_d$. The tolerance and the search range of the golden section can be modified (Figure 11).

The initial guess Cd0 is used to calculate the *error* with a defined correction factor.

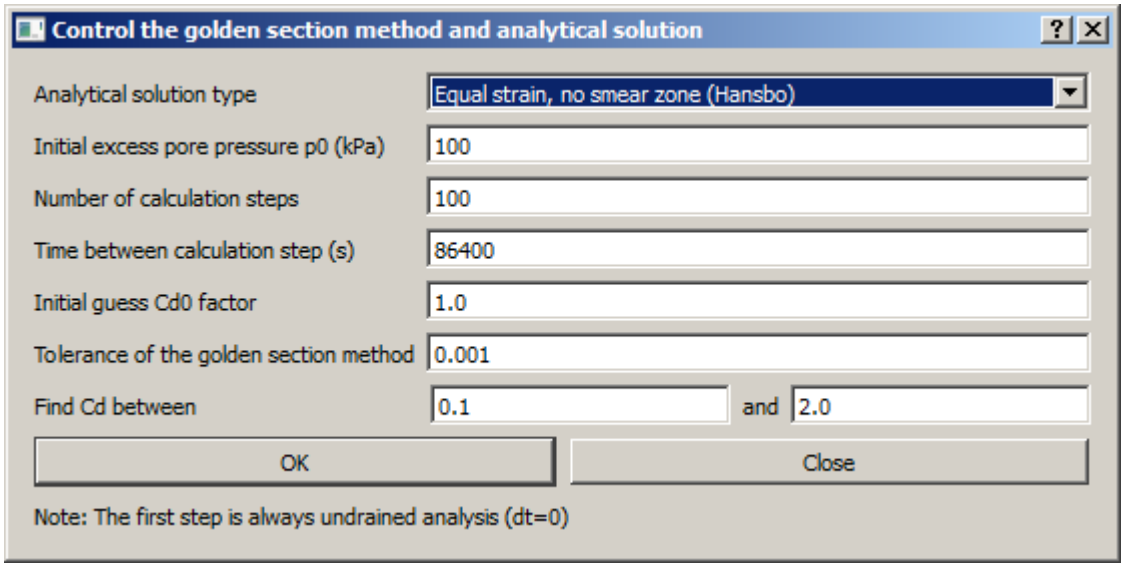



*Figure 11: Parameters for analytical solutions and the golden section method*

### 2.5 Visualisation

VTP can display 3D mesh and contour results (Figure 12). The mesh can be scaled. The results can be displayed for one calculation step, or the animation feature can be used to visualise all calculation steps.

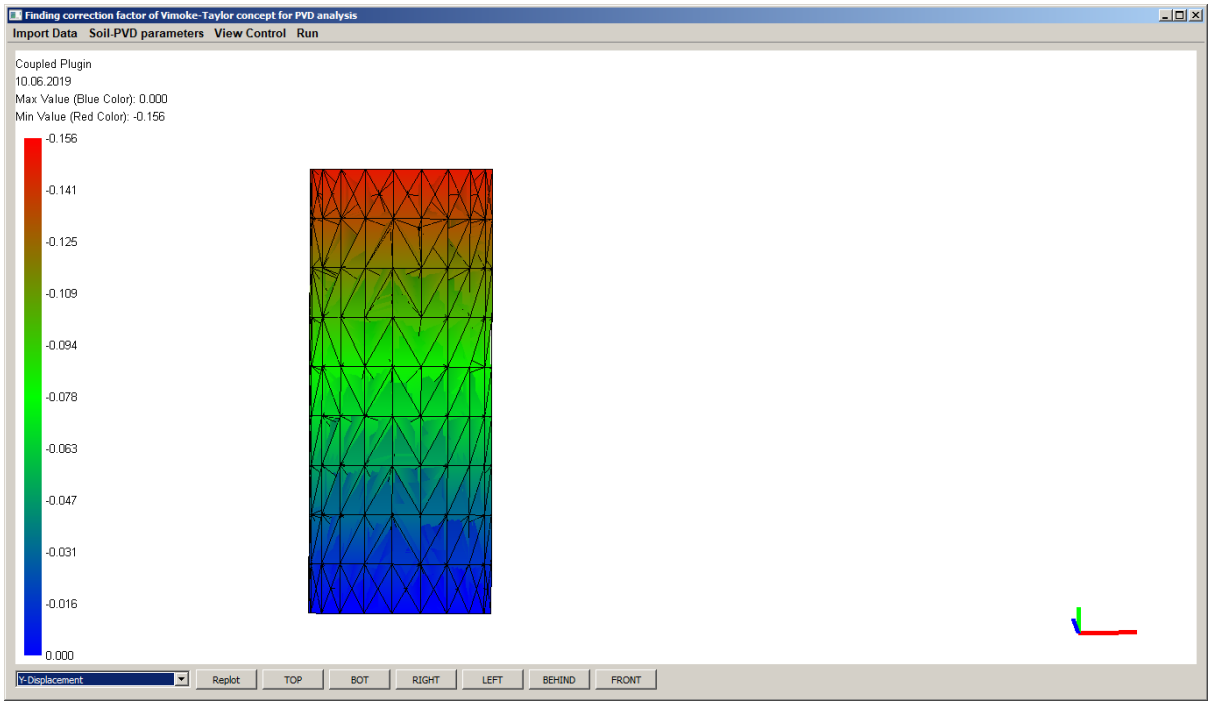



*Figure 12: Display mesh and results*

## 3 Element file structure

*elements.dat* has 14 columns. Column#1 is the element index, column#2—11 is the node indices of the element, column#12 is the number of nodes per element, column#13 is the material number (material#1 indicates the soil zone, and material#2 indicates the drained zone), and column#14 is the element type. Column#14 is a reserved feature and it can be ignored.

If the number of nodes of an element is less than 10 (no mid node), one row is used to store node indices. If the number of nodes of an element is larger than 10 (second order element with mid nodes), the first line is used to store the node indices of the major nodes, and the second line is used to store the node indices of the mid-nodes.

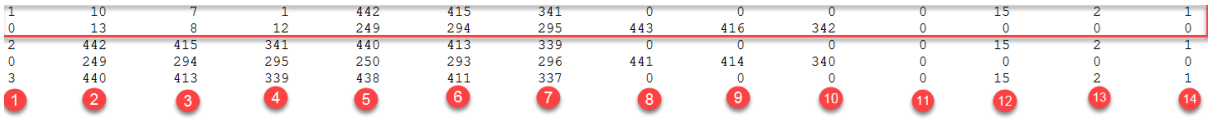



*Figure 13: Element file structure*

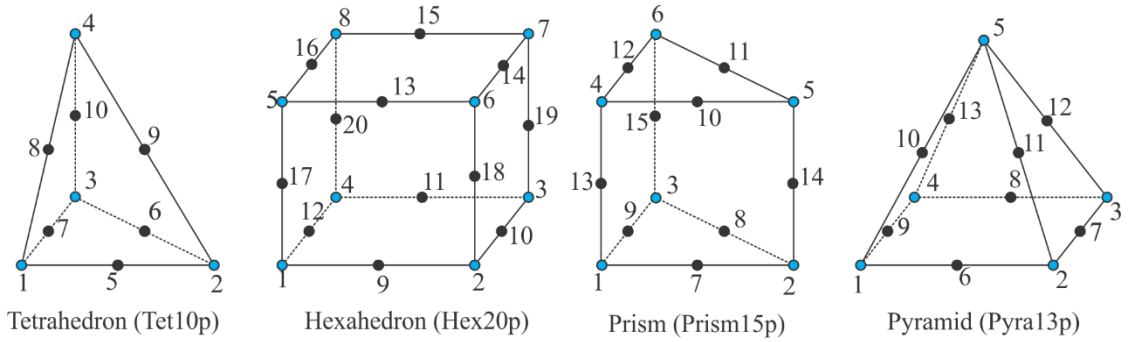
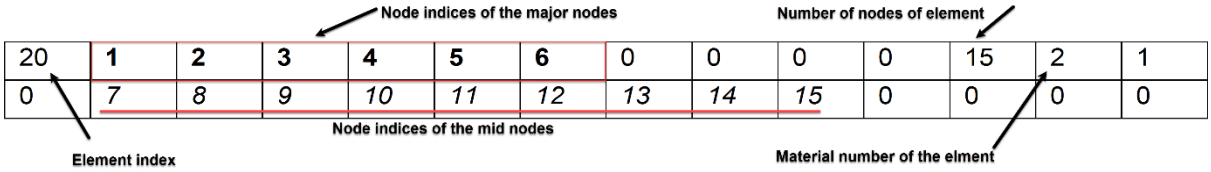



*Figure 14: Node indices of each element type*

For instance, in Figure 14, if a Prism15p with element index is 20 that belongs to the drained-zone, two rows that represent this element are shown in Figure 15. The “0” number means no data.




| | | | | | | | | | | | | |
|---|---|---|---|---|---|---|---|---|---|---|---|---|
| 20 | **1** | **2** | **3** | **4** | **5** | **6** | 0 | 0 | 0 | 0 | 15 | 2 | 1 |
| 0 | *7* | *8* | *9* | *10* | *11* | *12* | *13* | *14* | *15* | 0 | 0 | 0 | 0 |

*Figure 15: Two columns are used for a Prism15p*