

# Editing a Case - Part 1

## Introduction

A Case is edited by changing the values of the model parameters. These parameters are accessed in the 'Edit Settings' area of the LDAT Dashboard where the parameters are grouped under the Tabs **Model**, **Active elements**, **Waste type**, **Boundary elements** and **Calculation data**.

Clicking on one of the Tabs exposes details of the current values of the parameters. Change the values by clicking again on one of the Edit buttons. A popup dialog appears and displays the values again in editable text boxes. These dialogs may also have an 'Info' section which contains advice on the ranges of values that the parameters usually take.

In the case of the individual active and boundary element editors you need to select the element that is related to the parameter values. When the same set of parameter values is to be applied to a group of elements it is possible to change the parameter values in all of the elements at the same time. This is done by selecting the Range of the group of elements in the popup dialog.

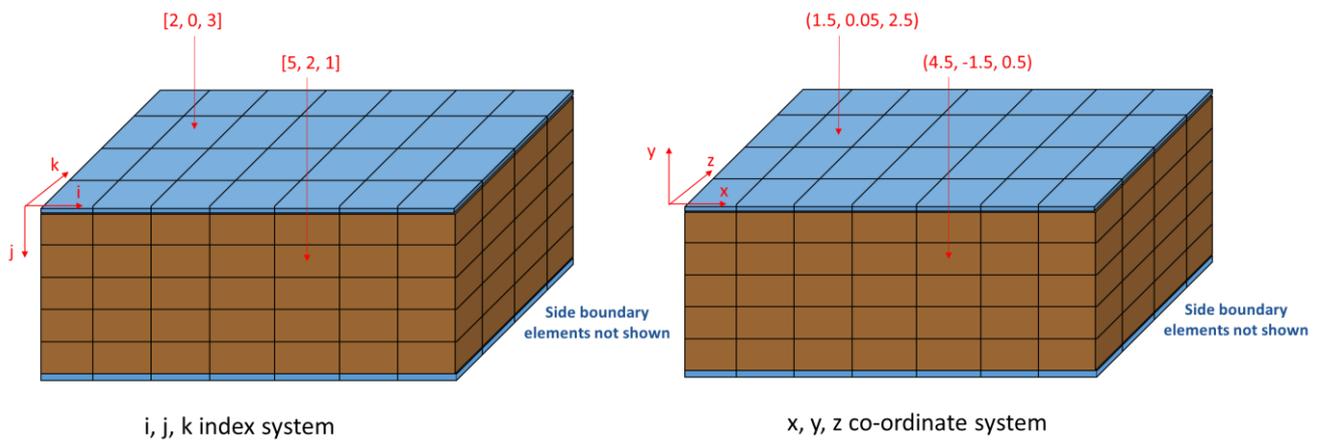
## Set the model scale

The overall size of the model is controlled by editing the dimensions of the individual rows and columns in the element framework. This is done by editing the parameter values in the **Slice dimensions** group found by clicking on the **Model** Tab.

The rows and columns in the model are identified using the indices  $i$ ,  $j$  and  $k$ . These indices are used to identify the individual element 'Slices' as well. Thus element  $i, j, k$  is the element in row  $j$  of column  $(i, k)$ . In this editor you set the widths and depth of the elements vertical Slice by vertical Slice, and horizontal Slice (or layer), by horizontal Slice. This of course also sets the widths and depths of all of the elements contained by the Slice. This means that you only need to set the widths and depths of the Slices that intersect along a diagonal of the element array, in order to set the widths and depths of all of the elements in the array.

The index origin of the element array is at the nearest upper left hand corner of the element array. From here the zero based element integer indices  $i$ ,  $j$ , and  $k$  increment across the element array in orthogonal directions, the  $j$  index being vertically downwards. LDAT uses an  $x, y, z$  orthogonal co-ordinate system to calculate the geometrical position of various points within the element array. The position of the centroid of each element is provided as part of the results output. The origin of the  $x, y, z$  system can be set by the user by editing the Origin co-ordinates parameter values group found by clicking the Model Tab. Note that LDAT assumes that the Boundary elements form an outer layer around the Active elements.

This information is summarised in the following Figures.



In this Figure the side boundary elements are not shown so the number of elements in the element array is 9 elements in the  $i$  direction, 7 in the  $j$  direction and 6 in the  $k$  direction. The origin element is the nearest left upper element and has  $i, j, k$  indices  $[0, 0, 0]$ . This element is not actually shown in the Figure as it is a side element. The boundary element  $[2, 0, 3]$  and the active element  $[5, 2, 1]$  are picked out.

In the example above, the element array origin in the  $(x, y, z)$  co-ordinate system has been set at  $(-0.1, 0.1, -0.1)$  and the active elements all have the unit size of  $1\text{m} \times 1\text{m} \times 1\text{m}$ . Remember that the thickness of the boundary elements is 0.1 metres. This results in the centroid of element  $[2, 0, 3]$  being at  $(1.5, 0.05, 2.5)$ , and the centroid of element  $[5, 2, 1]$  being at  $(4.5, -1.5, 0.5)$ .

### Set the initial leachate concentration and bacteria fractions in the waste

The groups of parameter values that determine the initial waste characteristics of the waste contained in an element are to be found by clicking the edit settings area **Waste type** Tab. Here we draw attention to the phase fractions and density group, where three important parameters are set. These are:

1. Initial leachate concentration
2. Initial bacteria fraction
3. Waste material density

Leachate concentration is the fraction of the pore space volume in the element that contains leachate and can in theory range in value between 0 and 1. Whatever is set for the leachate fraction determines the gas fraction since the two fractions add up to one. So you only have to set the leachate fraction.

The bacteria fraction is multiplied by the initial total volume of solids in an element to determine the amount of bacteria present. This has a strong influence on the initial rates of solids degradation. The range of values used is typically between 0.01 and 0.05, and should be proportionate to the amount of degradable solids that are present. These are setup using the solid waste characteristic parameters which are discussed in Editing a Case – Part 2.

The waste material density is the density of the solid fraction and is usually set at around  $1050 - 1200 \text{ kg/m}^3$  in the absence of information to the contrary.

You can look at (but not edit) the current values for these parameters in each element after clicking the **Waste type** button, by selecting the element using the element index selector located above the

tables of parameter values. If you want to change a value click on the 'Edit' button. A popup editor dialog containing a table of editable text boxes will appear. The dialog also contains two element selectors, both set to the element selected in the previous step. Edit these values and click the button to save the changes and close the popup. This procedure applies to all of the parameter value editors. It is possible to apply your changes to a range of elements at the same time. The range is specified by selecting the two elements in the popup to be at the extremities of the range.

Further details about how these parameters are used in the LDAT algorithm maybe found by downloading the reference: **LDAT code Note - initial mass calculation**.

In most editing pages, guidance is given on the typical range of values that a parameter might take. This guidance may be accessed by clicking on 'Info' in the popup dialog.

### **Set the key calculation parameters**

The calculation parameters are accessed by clicking the **Calculation data** Tab. Start with the **Time iteration** settings which control the calculation start and finishing times, the time step intervals, and the frequency of recording the results.

At the moment the value of the **Start Time** should be set to zero. In future the value will be selectable from a drop-down selector. This will allow a user to select one of the times in the records contained in the existing Merged Results Table for the Case. The calculation will then start from the Start time selected, and append the new records below the existing records up to and including the finishing time record. If the selected **Start Time** is zero then the whole merged results Table will be over-written from time zero with a fresh set of results. This facility of starting a calculation from a point within an existing set of results will enable a Case to be built up in stages, with the possibility of each stage having different boundary conditions.

The calculation will run until the model time reaches the value set as the **Finish time**. The results are recorded in the Merged Results Table at the regular intervals specified by the **Record interval** parameter value.

The time increment of the calculation starts at the **Minimum time increment** and is expanded after each iteration by the multiplier specified by the **Expansion Factor** up to a maximum specified by the **Maximum time increment**.

Another group of parameters that are accessed by way of the **Calculation data** button, are the **Calculation options** which enable you to choose which landfill processes are included in the model. The Liquid and Gas flow sub-models should always be on together with the Settlement sub-model. The default of course is to have all of the sub-models running at the same time. However it is sometimes useful to build up the functionality of a new Application by activating one sub-model at a time. You can do that by using the tick boxes.

### **Change key boundary conditions**

When you run and obtain the results for different Cases of a numerical simulation model like LDAT, you are running the same underlying data process system, but with different boundary conditions (defined by parameter values).

There are in fact two main types of 'boundary' condition. One is the set of 'initial conditions' (or the time boundary condition), which need to be set for the start time of a calculation run. These are what you are setting up when you edit the parameter values accessible through the **Active element** and **Waste type** Tabs. The other type of 'boundary' conditions are the conditions applied at the

actual physical boundaries that are defined by the boundary elements that wrap around the space that contains your model. You access these through the **Boundary elements** Tab.

Start with applying the boundary pressures and temperatures in the **BC Stresses** group of parameters. These are the Total stress, Temperature, Liquid Pressure, and Gas pressure applied at the location of a boundary element.

Of course these stresses only influence the space occupied by the waste in the **Active elements** if their affect can be transmitted to that space. Total stresses can only be applied through the upper layer of boundary elements. Liquid and Gas pressures are only transmitted if the boundary permeability is greater than zero. Similarly the boundary Temperature only has an effect if the Thermal conductivity of the boundary is greater than zero.

These 'transmission' parameter values are contained in the **BC Permeability/Thermal conductivity** group, where you will see the current boundary parameter values for Liquid permeability, Gas permeability, and Thermal Conduction.

If a boundary element is connected to the internal Active element space then Liquid and Gas will flow between the two. You can setup an initial reservoir of both fluids by setting the Liquid initial mass and Gas initial mass parameter values found in the **Boundary elements, Liq/Gas mass v/v** group.

As a detail, there is a requirement to specify the amount of the interface area between the boundary element and the waste that is shared between the Liquid and Gas phases when they flow into the waste. Use the parameters Liquid open area fraction and Gas open area fraction to do this. These are set to be the fraction of the interface area and should add up to 1.0. Usually they would be set at 0.5 (Liquid), 0.5 (Gas). However, if for example you set the share to be 1.0 (Liquid) and 0.0 (Gas), no gas would be able to flow into the waste from the boundary element although gas would be able flow into the boundary from the waste.

### **Estimate waste settlement**

The parameters accessed through the Boundary elements button include the group **BC Inert stock**, which has a single parameter, Mass of inert stock. This parameter only requires setting for boundary elements in the upper layer of the model.

When the waste degrades settlement takes place. The LDAT framework is rigid and cannot follow the reduction in solids volume that accompanies settlement. This is addressed by ensuring that the elements are topped up from a source of inert solid material that is stored in the upper boundary elements. To ensure that this does not run out, set the Mass of inert stock parameter value in an upper boundary element, to at least the fraction of degradable solid material in the waste contained in the Active elements below that upper boundary element.