Summary

AMBER is a compartment modelling software tool developed by Quintessa Ltd. This is the user guide for the demonstration version of the AMBER 6.7 software.

AMBER 6.7 is a flexible software tool that allows the user to build their own dynamic compartment models to represent the migration and fate of contaminants in a system.

© Quintessa Limited. All rights reserved. Quintessa Limited owns the rights to AMBER. See Help | About AMBER for licence information.
## Contents

1 **Introduction**

2 **Compartment Modelling with AMBER**
   2.1 Compartment Modelling ........................................... 3
   2.2 Modelling with AMBER ........................................... 4
   2.3 Spatial Modelling with AMBER ................................. 6

3 **AMBER Overview**
   3.1 Opening Case Files ............................................. 8
   3.2 General Features ................................................ 9
   3.3 Left Panel ....................................................... 9
   3.4 Main Panel ..................................................... 10
   3.5 Spatial View .................................................... 11
      3.5.1 Specifying X-Y Points and Defining Shapes ............. 12
      3.5.2 Specifying Z Points and Defining Cells ................. 13
      3.5.3 3D View .................................................. 14
   3.6 Contaminants and Decays ....................................... 15
   3.7 NameSets and Mappings ......................................... 15
   3.8 Parameters .................................................... 16
   3.9 Features to Support the Review of Parameter Definitions in AMBER .................................................... 17
   3.10 Calculating a Case ............................................ 18
   3.11 Results Viewer .................................................. 19
      3.11.1 Reporting Results in a Text File ....................... 20
      3.11.2 2D Graphs ................................................ 20
      3.11.3 Rendering of Results in 3D View ....................... 20
   3.12 Use of Tabs in the Main Panel ................................ 21
   3.13 References for Section 3 ..................................... 26

4 **Example Case Files**
   4.1 IAEA Trench Training Case .................................. 27
   4.2 IAEA ISAM Vault Test Case .................................. 28
   4.3 IAEA Waste Acceptance Criteria Example .................. 30
   4.4 IAEA BIOMASS Example Reference Biosphere 2A ............ 32
   4.5 IAEA BIOMASS Example Reference Biosphere 2B ............ 32
4.6 Point Source NORM Example .................................................. 34
4.7 Area Source NORM Example ................................................. 35
4.8 Landfill Gas Model .............................................................. 36
4.9 Biokinetic Model for Lead ...................................................... 37
4.10 Gaussian Plume Atmospheric Dispersion Model ...................... 37
4.11 PSACOIN 1B Probabilistic Biosphere Model ......................... 38
4.12 Model for C-14 in a Canadian Shield Lake .............................. 39
4.13 References for Section 4 ....................................................... 41

5 Implementing a New Case ....................................................... 42
  5.1 The Demonstration Model .................................................... 42
  5.2 Create a New Case ............................................................. 43
  5.3 Defining the Base Units ...................................................... 43
  5.4 Adding Contaminants and Decays ......................................... 43
    5.4.1 Adding Contaminants .................................................. 43
    5.4.2 Adding Decays .......................................................... 44
  5.5 Adding Compartments, Transfers and Submodels ...................... 45
    5.5.1 Adding Compartments ................................................ 45
    5.5.2 Adding Transfers ....................................................... 47
    5.5.3 Creating a Submodel .................................................. 49
  5.6 Adding a NameSet and Mapping ......................................... 51
    5.6.1 Adding a NameSet ..................................................... 51
    5.6.2 Adding a Mapping ...................................................... 52
  5.7 Setting Start Amounts ....................................................... 54
  5.8 Adding Parameters .......................................................... 55
    5.8.1 Adding Standard Parameters with No Multiplicity .............. 55
    5.8.2 Adding Standard Parameters with Single Multiplicity ........ 56
    5.8.3 Adding Standard Parameters with Dual Multiplicity .......... 59
    5.8.4 Adding Lookup Time-Dependent Parameters .................... 61
    5.8.5 Adding Observers ...................................................... 64
  5.9 Specifying Transfer Rates ................................................ 68
  5.10 Adding Result Times ....................................................... 69
  5.11 Calculating and Investigating Results ................................ 70
    5.11.1 Checking Parameters and Units .................................. 70
    5.11.2 Calculating the Case ................................................. 70
5.11.3 Investigating the Results ........................................ 71
5.11.4 Additional Changes ........................................... 73
5.12 References for Section 5 ........................................ 73

6 Feedback .................................................................. 74
6.1 Sales Enquiries ..................................................... 74

A Knowledge Base ..................................................... 75
1 Introduction

AMBER is a flexible software tool that allows users to build their own dynamic compartmental models to represent the migration and fate of contaminants in a system, for example the surface and sub-surface environments. Radioactive and non-radioactive contaminants in solid, liquid and gaseous phases can be considered. AMBER can be used to assess routine, accidental and long-term contaminant scenarios.

This is the user guide for the free demonstration version of AMBER. The capabilities of the software are briefly summarised below, before the structure of the document is described. The demonstration version of AMBER is also accompanied by the AMBER 6.7 Reference Manual, which includes comprehensive details on the way in which the unrestricted version of the software works. Both of these documents are also available as electronic help files from within the AMBER software via the Help menu.

AMBER gives the user the flexibility to define:

- any number of compartments and submodels;
- any number of contaminants and associated rates of degradation/decay and ingrowth;
- any number of transfers between compartments;
- algebraic expressions to represent transfer processes operating between compartments;
- algebraic expressions defining endpoints of interest, such as the uptake of contaminants by humans; and
- deterministic, probabilistic and time varying parameter values.

This flexibility means that users can construct case-specific models and tailor generic models to their specific needs. Key features incorporated into AMBER are summarised in Table 1 below.

The guide to the demonstration version of AMBER has the following structure.

- The principals of using compartment models to represent the source, fate and consequences of contaminants are described in Section 2.
- An overview of AMBER is provided in Section 3, including an introduction to the main features and the user interface.
- The example models (case files) provided with AMBER are described in Section 4.
- A tutorial for developing a simple model for disposal of radioactive waste is described in Section 5.
- Quintessa very much welcomes feedback and further enquiries about AMBER; contact information is provided in Section 6.

AMBER has been applied to a wide range of studies over the past twenty years. Reports and papers relating to those studies provide a ‘knowledge base’ for AMBER users and can be drawn on to provide inspiration, mathematical models and data for future studies. References for a selection of such studies are included in Appendix A for information.
Table 1. Key AMBER Features

<table>
<thead>
<tr>
<th>Features</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>A powerful, flexible, user-friendly graphical interface for Windows and Linux</td>
<td></td>
</tr>
<tr>
<td>Completely transparent text-based storage of model information (this simple feature facilitates robust quality assurance of models)</td>
<td></td>
</tr>
<tr>
<td>Spatial awareness and 3D visualisation of models, which are unrivalled in this calibre of contaminant transport software and major new features that were introduced in AMBER 6.0</td>
<td></td>
</tr>
<tr>
<td>Full probabilistic capabilities, allowing full or partial Monte Carlo or Latin Hypercube sampling and including a wide range of probability density functions</td>
<td></td>
</tr>
<tr>
<td>Fast and accurate Laplace-transform and numerical time-stepping solvers, which automatically determine solve steps</td>
<td></td>
</tr>
<tr>
<td>Support for complex time-varying source terms, environmental properties and transfer processes</td>
<td></td>
</tr>
<tr>
<td>Import and export of data via external files, allowing data to be exchanged with other software tools</td>
<td></td>
</tr>
<tr>
<td>Batch mode operation, facilitating automated running and outputs, along with integration with other software tools</td>
<td></td>
</tr>
<tr>
<td>In-built parameter checking and ‘units awareness’, which increase confidence in models</td>
<td></td>
</tr>
<tr>
<td>In-built graphing and reporting as well as Excel export capabilities</td>
<td></td>
</tr>
<tr>
<td>Provision of comprehensive tutorials and technical supporting documentation</td>
<td></td>
</tr>
<tr>
<td>A large active international user group</td>
<td></td>
</tr>
<tr>
<td>Continued active development</td>
<td></td>
</tr>
</tbody>
</table>
2 Compartment Modelling with AMBER

The concept of compartment modelling is discussed in subsection 2.1 and its application within AMBER introduced in subsection 2.2. The concept of spatially aware modelling in AMBER is introduced in subsection 2.3.

2.1 Compartment Modelling

Compartment modelling is a flexible way of evaluating the migration and fate of contaminants in environmental systems. Contaminants are unwanted substances within a ‘system’ that may have the potential to do harm. In the context of this discussion, and for modelling with AMBER, the contaminants are present in insufficient quantity to modify the surrounding environment. The ‘system’ of interest will reflect the context being studied, but may be small scale (such as an individual plant or a single waste container), or large scale (such as global oceans). Systems that can be represented in compartment models may have uniform characteristics (such as targeted modelling of a relatively homogeneous aquifer) or may include a diverse array of different man-made and natural media (such as a total-systems model for geological disposal of waste). The migration of contaminants around the system being modelled is characterised by transfer/turnover rates between the compartments.

A key assumption for compartment models is that the distribution of contaminants within a single compartment can be considered to be uniform (either because relatively rapid mixing occurs in the compartment, or because the average concentration is a sufficient approximation). If this assumption does not hold, then the medium needs to be further split-up (discretised) into a number of compartments within which averaging is acceptable.

At any given time, the movement of contaminants from one compartment to another is represented with a transfer rate. This is the fraction of the inventory in a particular compartment that is transferred from that compartment to another per unit of time (per second or per year). Transfer rates can themselves change with time. Such changes may occur gradually (such as with changing climatic conditions) or suddenly (such as through sudden loss/failure of a barrier).

Contaminants may also change with time, for example through chemical/physical degradation or through radioactive decay. Contaminants may degrade/decay to substances that are no longer of interest, for example, if the resulting species is no longer hazardous (such as radioactive decay to a stable isotope).

The structure of a very simple compartment model is illustrated in Figure 1. The evolving amount of a contaminant N in compartment i (Ni, moles) can be calculated mathematically using a first order linear differential equation:

\[
\frac{dN_i}{dt} = \sum_{j \neq i} \lambda_{ji}N_j + \lambda_M M_i + S_i(t) - \sum_{j \neq i} \lambda_{ij}N_i - \lambda_N N_i
\]

(2.1)

where:

- \(i\) and \(j\) are the two compartments;
• $N$ and $M$ are the amounts (moles) of contaminants $N$ and $M$ in a compartment ($M$ is the precursor of $N$ in a decay/degradation chain);

• $S_i(t)$ is a time dependent external source of contaminant $N$ (moles $y^{-1}$);

• $\lambda_N$ is the decay/decay/degradation rate for contaminant $N$ ($y^{-1}$);

• $\lambda_{ij}$ is the transfer rate representing the loss of radionuclide $N$ from compartment $i$ to $j$ ($y^{-1}$); and

• $\lambda_{ji}$ is the transfer rate representing the loss of radionuclide $N$ from compartment $j$ to $i$ ($y^{-1}$).

This equation is linear, so that, for example, if there is twice as much of a contaminant in the system initially, then the calculated concentrations in the various compartments will all be doubled at each time of interest. Note that the sums represent potential for transfers between many compartments.

The solution provides the time-dependent inventory of contaminant $N$ in each compartment. Assumptions for compartment sizes allow estimates of the associated concentrations to be made.

### 2.2 Modelling with AMBER

AMBER provides a simple and flexible environment in which compartment models can be developed (see, for example, Figure 2). There are no significant limits on the number of contaminants, compartments or transfer processes, beyond the practical constraints imposed by computer processing power. A typical computer is sufficiently powerful to permit models with several tens of compartments and contaminants to be solved within minutes.

The key aspects of AMBER that aid the development of compartment models are:

• a simple and intuitive user interface;

• a powerful system for defining parameters and expressions, which in turn can be used to specify the characteristics of contaminants, compartments or transfer processes;
• capability to represent the spatial geometry of the modelled region;
• probabilistic modelling capabilities;
• consistency checking, for example, to ensure that equations defined in AMBER are dimensionally correct;
• flexible methods for outputting data, enabling models and their behaviour to be explored.

Although the models that can be set up in AMBER are limited to those of the linear donor controlled compartment type, this class of models can be applied to a very wide range of problems as a result of the flexibility with which transfers between compartments can be specified. For example, some diffusive like processes depend upon the concentrations of contaminants in both the donor and receiving compartments, but these can readily be represented in AMBER by including a ‘forward’ transfer from the donor to the receiving compartment and a ‘backward’ transfer from the receiving compartment to the donor. The combination of these two transfers will correctly model the net transfer between compartments.

AMBER has been applied to a wide range of problems concerned with the way that radionuclides and other contaminants move through different parts of the environment. For more information on AMBER applications, please have a look at the ‘knowledge base’ of reports describing studies involving the software, which is provided in Appendix A.

![Biokinetic Model for Lead](image)

**Figure 2. An Example of an AMBER Representation of a Compartment Model**
2.3 Spatial Modelling with AMBER

AMBER includes the capability to represent model compartments spatially. This brings considerable power to the model-building process, enabling users to visualise models in three dimensions and allowing AMBER to automatically calculate various geometric properties (e.g. areas, volumes, distances) that are frequently used in model expressions.

The spatial structure for a model can be set-up through defining x, y and z coordinates, which can be grouped to define three-dimensional ‘cells’. The spatial cells can be linked to compartments in the contaminant transport model, allowing associated properties to be shared. This flexible approach is used to allow contaminant transport models to comprise both spatial and non-spatial components.
3 AMBER Overview

AMBER uses the standard conventions of Microsoft® Windows® and Linux operating systems and so anyone with experience of running applications in such environments will find it easy to use. The main elements of the AMBER interface are illustrated in Figure 3. Both the left panel and main panel can have multiple panes, each accessed through a named tab. This section summarises some of the main features of the AMBER interface. More details are included in the AMBER 6.7 Reference Manual.

![Figure 3. Main Features of the AMBER Interface](image)

AMBER has a menu bar (see Figure 4) that allows access to all of its main features. The most commonly used commands can also be accessed via buttons on the toolbar. ‘Tool tips’ are available when hovering over a button to help with navigation.

![Figure 4. The AMBER Menu and Toolbar](image)
3.1 Opening Case Files

Upon launching the demonstration version of AMBER 6.7, an ‘About’ window will appear in addition to the AMBER ‘splash’ screen emphasising that this is a restricted version of AMBER. The user is restricted in terms of the case files that can be opened and the changes that can be made. These are briefly described in under the ‘Permits’ tab of the About window.

The demonstration version is accompanied by twelve example case files, which provide an illustration of a range of models that have already been implemented in AMBER. These are described in more detail in Section 4.

The demonstration version also allows users to implement a relatively simple pre-defined compartment model for a radioactive waste disposal facility. The implementation of this model is described in more detail in Section 5.

AMBER models are saved in text-based ‘case files’ of the type “[filename].cse”. To open any of the example case files, click on File | Open or click on the ‘Open case’ icon on the toolbar. This will open a ‘Select a case file to open’ window. The required case file can be opened either by double-clicking, or by selecting the file and clicking on ‘Open’. After installation, the example cases and a completed tutorial can be found under an ‘AMBER Example Files’ folder in your user ‘Documents’ directory on Windows computers.

It is emphasised that users are limited in terms of the changes that they can make to the example case files and tutorial when using the demonstration version of AMBER. Changes beyond those permitted and attempts to open case files not associated with a permit will result in an ‘AMBER Error’ either when the changes are made or when the user attempts to calculate the case. On opening any case or creating a new case for the tutorial, a ‘Permit Selection’ dialog is shown that summarises the associated restrictions.

If you attempt to open a case file that is not automatically linked with a permit, then a ‘Permit Selection’ window will appear. Select the permit that is associated with the file that you are trying to open in order to proceed.
3.2 General Features

Various tools are available for building, arranging and annotating models in the main panel (see Figure 5). There are two main views – the model layout and the spatial view. The tools available depend on the view.

Model layout tools allow the user to develop, interact with and annotate the structure of the compartment model. The model can be organised hierarchically into submodels containing groups of related compartments (e.g. soil layers).

The spatial view tools enable the user to manipulate a spatial model in order to build and visualise it. The ‘select’ mode allows points to be placed and connected to form the polygons that make up a spatial model. Other tools allow the 3D model to be zoomed, panned and rotated.

<table>
<thead>
<tr>
<th>Model Layout View</th>
<th>Spatial View</th>
</tr>
</thead>
<tbody>
<tr>
<td>Toggle Side Panel</td>
<td>Select Mode</td>
</tr>
<tr>
<td>Select</td>
<td>Zoom In</td>
</tr>
<tr>
<td>Compartment</td>
<td>Zoom Out</td>
</tr>
<tr>
<td>Source</td>
<td>Pan</td>
</tr>
<tr>
<td>Transfer</td>
<td>Rotate</td>
</tr>
<tr>
<td>Submodel</td>
<td>Info</td>
</tr>
<tr>
<td>Insert Text</td>
<td></td>
</tr>
<tr>
<td>Insert Picture</td>
<td></td>
</tr>
<tr>
<td>Annotate</td>
<td></td>
</tr>
</tbody>
</table>

![Figure 5. AMBER Model Tool Palettes](image)

3.3 Left Panel

The left panel allows the information that supports a contaminant transport model (including parameters, NameSets etc.) to be viewed at the same time as the main panel. Different sets of information are viewed on separate tabs within the left panel (see Figure 6). The left panel can be shown or hidden, depending on user preference; the panel is automatically hidden when exploring a spatial view.
3.4 Main Panel

The main panel can show the Model view, the Spatial view or a range of specific tabs (e.g. for editing an individual parameter). The structure of the contaminant transport model is developed within the Model view. Pictures, text and colour can be added to the Model view to make models clear and more readily understandable.

The tool palette enables compartments, transfers and sources to be added in the model view. Selecting an option from the tool palette and then clicking in the Model view will add the model component at the location of the mouse.

The submodel tool palette option allows a part of a model to be grouped together into a submodel. When this is selected, a rectangle can be drawn in the Model view and all components within the highlighted area will be included in the submodel. Submodels are used for organising the layout of the compartment model, e.g. grouping a number of related compartments together. The compartments and their properties remain available to other parts of the model without restrictions. Submodels are displayed with rounded corners and without black orders or shadows.

Submodels are useful for organising the layout of a model (see Figure 7). There are various other options for arranging and organising the model components. The components can be moved around by selecting them. Multiple model items can be selected by using the pointer and holding <Shift> whilst clicking on the desired items. The layout of selected items on the Model view can be aligned, using View | Align. The Left, Centre, Right options align items in the horizontal plane, whilst the Top, Middle, Bottom options align items in the vertical plane.

Colour can be introduced to any of the components on the Model view (see Figure 8) by using the pointer tool, right clicking over the item (such as a source, compartment or transfer) and selecting Colour... from the context-sensitive menu. This calls up a dialogue box in which a colour can be selected or defined. The relative size of Model view components can also be adjusted via the context-sensitive menu by selecting Relative Size and choosing a size when one or more components have been selected. Finally, the font displayed in the Model view can be changed by right-clicking, selecting Font and then choosing the appropriate font system, style, and size.
Text, images and annotations can be added to the model using the relevant options in the tool palette.

![Figure 7. Use of Three Submodels to Organise a Model](image)

![Figure 8. An Example of the Use of Colour in a Model](image)

### 3.5 Spatial View

In the main panel, the Spatial view enables the user to build 3D representations of some or all of the model compartments. When a new spatial model is created, the user is first prompted to define the dimensions of the model, in terms of the maximum and minimum dimension in the X, Y and Z direction in metres. The user can also define any axes exaggeration, to aid the visualisation of a system with differing length scales in the x-y-z planes, of which the details can be found in Section 8.8 of the AMBER 6.7 Reference Manual. A background image can also be added, such as a map of aerial view, as well as a grid (helping points to be placed at precise locations).

Once the spatial model domain is defined the user can then interact with it in three modes, available on the right panel:

- X-Y point selection;
- Z point selection; and
- 3D view.
3.5.1 Specifying X-Y Points and Defining Shapes

All spatial compartments are made from user-specified points in the X-Y plane (see Figure 9). These are used as vertices for polygons, to which depth (in the Z direction) is subsequently added. All points have to be within the model domain.

Points can be defined by clicking the mouse at a location within the model domain, or specifying their coordinates manually. A cell can then be formed by selecting “New” cell then connecting the relevant points by clicking on or clicking and dragging around the points sequentially. A minimum of 3 points is needed to define a spatial cell. Once created, the area within a cell will be coloured green and the user prompted to assign a name for the cell. Cells are linked to the contaminant transport model by name, so a drop-down option allows users to select a name from the compartment model.

Note that users are unable to create new spatial models in the demonstration version of AMBER, but can explore existing spatial models associated with the example case files (see Section 4).

Figure 9. Specifying X-Y Points in the Spatial View
3.5.2 Specifying Z Points and Defining Cells

Volume-filling cells are defined by adding Z coordinates to the shapes drawn in the X-Y plane. This is done in the “Z Point Selection” mode (see Figure 10). Clicking a shape shows its X and Y points in the panel and allows the user to define a minimum (bottom) and maximum (top) for each point that makes up the shape. Once three minima or three maxima are added the top or bottom surface is defined. Note that each surface is defined by 3 points only, so as to ensure that the surface is planar.

In many cases users may want to define a “stack” of cells, one on top of another, in the Z-direction. This is easily done by creating a single cell covering the total depth and then clicking “Split Cell”. This will create the relevant number of stacked cells. The stack of cells will have the same top and bottom as the original cell, but each will have an equal fraction of the height, which can subsequently be modified.

When cells are split, those adjacent to each other in the Z-direction will have linked vertices to ensure that there are no gaps in the model. Changing the Z-value for one cell will result in the vertex of the adjacent cell being modified. Linked points are indicated with chain icon adjacent to their value.

As noted above, users are unable to create new spatial models in the demonstration version of AMBER, but can explore existing spatial models associated with the example case files (see Section 4).

Figure 10. Specifying Points in the Z Direction
3.5.3  3D View

Once one or more cells have been defined, the model can be viewed in the “3D View” mode (see Figure 11). This does not enable the spatial model to be changed, but provides a convenient way of inspecting the model. Cells can be highlighted by clicking on the list in the right panel. Cells can also be hidden or shown in the shape preview mode via the check boxes in the list.

![Figure 11. Visualising a 3D Model](image)

The user can save views of their spatial model in the 3D View mode, such that camera angle, zoom, selected cells and hidden cells can be saved. This is useful if you wish to render a series of results on the 3D view, with the option to save multiple views in a single case file.

Once the user has a spatial view they wish to save for a given composition of selected cells, degree of zoom and camera angle used, it is possible to save that view by right-clicking your mouse when hovered over the 3D spatial view. A context menu will pop-up in which you can select to Save View, Reset View or Open List of Views (see Figure 12).
3.6 Contaminants and Decays

The list of contaminants included in a case can be viewed by clicking on the Contaminants icon on the toolbar. This brings up the ‘Contaminants’ tab on the left panel, which displays a list of the contaminants. Double-clicking on a contaminant will bring up a Contaminant ‘Item Info’ section on the left panel.

Similarly, the list of degradations/decays in a case can be viewed by clicking on the Decays icon on the toolbar. This brings up the ‘Decays’ tab on the left panel, which displays a list of the decays. Double-clicking on a decay will bring up a Decay ‘Item Info’ section on the left panel.

Note that the contaminants and decays can also be explored via a ‘Contams and Decays Tree’ tab on the left panel. This can be viewed by selecting Panels | Contaminants and Decays | Show Tree...

3.7 NameSets and Mappings

AMBER uses the concept of NameSets to list items over which information will be stored/indexed in the case files. Examples of internal NameSets include Contaminants, Compartments and Transfers. Users can add NameSets of their own via the NameSets tab on the left panel, which can be viewed by clicking on the NameSets icon in the toolbar or via Panels | NameSets. Double-clicking on a NameSet opens an ‘Item Info’ section on the NameSets tab, which lists the individual items in the selected NameSet.
In some cases, it is useful to ‘map’ from one NameSet to another. For example, a NameSet can be added listing the media of a model to enable parameters with media-dependency to be defined. A mapping from the internal Compartments NameSet to the new ‘media’ NameSet will allow the media properties to be assigned to individual compartments. The ability to define NameSets and Mappings is a powerful feature in AMBER and greatly aids in the efficient management of data within a model.

Mappings can be viewed by clicking on the Mapping icon on the toolbar or via Panels | Mappings, which opens the Mappings tab on the left panel. Individual mappings can be viewed on the main panel by double-clicking on the mapping name in the Mappings tab.

3.8 Parameters

The list of parameters used in an AMBER case file can be viewed by clicking on the Parameters icon in the toolbar or via Panels | Parameters. This calls up the ‘Parameters’ tab on the left panel (see Figure 13). Parameters can be highlighted by clicking on them and/or scrolling through the list with the up and down arrow keys. Summary information is available if the mouse is held over a parameter name and is displayed on the bottom of the Parameters tab when a parameter is highlighted in the list. The full information about a parameter is available as a tab on the main panel if the user double-clicks on a parameter in the list.

There are ten types of parameter available in AMBER:

- Internal (Read Only) – internal AMBER parameters that can be viewed but not changed;
- Internal (Editable) – internal AMBER parameters that can be modified by the user;
- Standard – parameters used in calculating initial amounts, source terms and transfers;
- Observer – parameters that derive endpoints of interest from compartment amounts and transfer fluxes;
- Imported Parameter – parameters that are imported from external files;
- Lookup Time-dependent – one method of defining time-dependent parameters (another method is by directly using the internal (read only) parameter for time ‘t’);
- Sampled – sampled parameters that can be selected from a wide range of probability density functions (PDFs);
- Derived Parameters – parameters that can be derived from observers (such as ‘maximum dose’ or ‘time of peak concentration’);
- NameSet Option – user-defined drop-down lists that enable users to select particular model options (e.g. choosing between different scenarios in the same case file); and
- Spatial – parameters that provide spatial information obtained by linking compartments to 3D cells within a spatial model.
Note also that it is possible to ‘lock’ parameters in a case file. Locked parameters can be viewed by checking ‘Show Locked Parameters’ in the Parameters window, but they cannot be edited.

The concept of NameSets is introduced in Section 3.7. Parameters can be defined (indexed) over zero, one or two NameSets, allowing ‘arrays’ of data to be entered. This concept is called ‘Multiplicity’ in AMBER terminology.

### 3.9 Features to Support the Review of Parameter Definitions in AMBER

When implementing a mathematical model and supporting data in a software program, it is recommended that the definition of expressions (equations) and the transcription of numerical values from the supporting documentation is subject to a review and approval process. This form of Quality Assurance (hereafter referred to as QA) lies at the heart of AMBER. Indeed, AMBER was developed to have text-based case files to support review of Parameters, Expressions, and wider aspects of the model (e.g. Transfers, NameSets). The text-based case file can be annotated by a reviewer using the Parameter Description...
field (Figure 14a), adding comments to individual entries in a Parameter with multiplicity (Figure 14b), or by annotating a print-out of the case file.

![Figure 14. Examples of pre-existing means to record review of parameters. (a) Using the Parameter Description box. (b) Using the Comments box for a parameter with multiplicity.](image)

AMBER includes a number of features that aim to better support the management and recording of the QA status of parameters. The functionality is designed to be a user-friendly means of demonstrating that the model has been subject to due review. Note that the marking of a parameter as having been reviewed and approved for use in the model, or lack of marking, does not ensure or prohibit the use of the parameter in any calculations.

There are three possible QA states for a parameter that can be subject to the QA functionality: “Approved”, “Needs Review” or “No QA status”. In the context of AMBER,

- “Approved” means that the Parameter definition has been reviewed and approved.
- “Needs Review” means that the parameter definition is pending review and approval. It may be in this state after being marked as such directly on the parameter or implicitly through other changes within the case file.
- “No QA status” means that the Parameter QA status has not been set.

Though disabled by default, these features are accessed by selecting Options | Enable QA Features (Figure 15). When the QA functionality is enabled, the QA status of parameters can be viewed in a variety of ways:

- Viewing an individual Parameter tab in the Main Panel.
- Viewing the Parameter list in the Left Panel, and also reported in the Parameter Info box when a parameter is selected in the Parameter list.
- Export of a .csv file of the QA records and subsequent viewing in Excel, or similar software package.

Detailed information regarding the approval process for parameters and how the QA status of parameters can be viewed are given in Section 9.5 of the Reference Manual.

### 3.10 Calculating a Case

When a case is calculated and saved, AMBER will store the time-history of the compartment amounts and transfer fluxes at user-defined times. These times are called ‘Result
The QA functionality is accessed via the Options drop-down menu.

Times’ and can be viewed and edited by clicking on the Result Times icon on the toolbar or via Panels | Result Times.

Case files can be calculated by clicking on the Calculate icon in the toolbar or via Calculation | Calculate. This brings up a ‘Calculate’ dialog, where the user can select the solver to be used:

- Laplace solver – for cases that do not involve time-dependency (Robinson and Maul, 1991); and
- Time-step solver – for cases that involve time-dependency (Robinson, 2001).

Both solvers intelligently determine the required calculation time-steps, so the user need not input solver time-steps. Note that the resulting compartmental amounts and transfer fluxes are stored by AMBER at each of the defined Result Time.

### 3.11 Results Viewer

AMBER provides three main methods of presenting model results: via text-based reports, via 2D graphs and via rendering results that are indexed over compartments in the 3D View mode of the Spatial View. When a model is calculated all parameters are available to output (provided there are no errors in the expressions). Before calculating the model input parameters can be reported, including any equations that do not rely upon the model to be run (e.g. the expressions to calculate transfer rates). This can be useful to check the model has been correctly set up.
3.11.1 Reporting Results in a Text File

The report dialogue (Figure 16) enables the user to select the parameters to output and also some aspects of the reporting format (e.g. the number of significant figures and the delimiter between numbers).

![Figure 16. Reporting Dialogue](image)

3.11.2 2D Graphs

Figure 17 shows the dialogues for specifying the information to be graphed. AMBER includes a simple graphing tool, or the user can choose to output data to a graph in Microsoft® Excel®. If “Output to Excel” is selected for a given AMBER case file, this is then remembered for all other figures plotted from that case file.

3.11.3 Rendering of Results in 3D View

It is also possible to render parameters, both input and results, onto the 3D View mode (see Figure 18). Those parameters may be indexed over something else in addition to compartments. While rendering results, the model cells become opaque instead of transparent. It is possible to modify the colour scheme used, and attributes associated with the minimum and maximum values of the range (i.e. if they are based on a given time, all times, are arbitrarily fixed, or if non-spatial compartments values are accounted for). Further, the user can play through, and export, a video of results rendered in AMBER’s 3D View.

These are described more in Section 11.6.3 of the AMBER 6.7 Reference Manual.
3.12 Use of Tabs in the Main Panel

As with the left panel, different sets of information are viewed on separate tabs within the main panel (see Figure 19). This includes:

- submodels (see Section 7.2 of the AMBER 6.7 Reference Manual);
- parameters you are editing (see Section 3.8);
- mappings (see Section 3.7);
- NameSet Options (see Section 9.1.10 of the AMBER 6.7 Reference Manual);
- Check Parameters (see Section 5.11.1);
- graphs of results (see Section 3.11); and
- Spatial View (see Section 3.5).

It is also possible to view tabs in a tile arrangement in the main panel. This is achieved by clicking and holding the mouse down on a tab, and then dragging the cursor to the left, right or top sides of the main panel. When you have reached a position where a tile can appear that part of the main panel will go grey (Figure 20). Release the mouse to position the tab (Figure 21). Once you have created two tiles, it is possible to split any of those tiles into smaller tiles, each with a number of tabs (such as Figure 22). Note that if you try to place a tile at the top of the “Model” tab, it will always span the whole of the main panel. You can also drag tabs from one tile to another without creating new tiles by clicking and holding the mouse down on a tab and dragging that tab to the tab listing on a different tile.

To return to the standard tab view, drag the tabs you wish to keep open down to the bottom of the tile which has the Model tab on it. Alternatively, you can close the tabs in the tile that you wish to remove by clicking on the X buttons.
Figure 18. Rendering parameters in 3D View

Figure 19. Tab view in Main Panel
Figure 20. Positioning the tiles in the main panel
Figure 21. Tiled main panel
Figure 22. More complex tiled main panel
3.13 References for Section 3


4 Example Case Files

AMBER is supplied with twelve example models, which help to illustrate its application, though each is simple in scope. The example cases are installed with the software. Brief summaries are provided within the AMBER implementations and in the sub-sections below, together with references that describe the underlying models.

The example cases are:

- an implementation of a model developed for use within an IAEA training course on Safety Assessment of Near Surface Low and Intermediate Level Radioactive Waste Disposal Facilities (file Trench.cse);
- an implementation of the vault test case (file Vault.cse) from the IAEA’s co-ordinated research programme on Improvement of Safety Assessment Methodologies for Near Surface Disposal Facilities (ISAM);
- an example of the use of AMBER to calculate waste acceptance criteria (WAC) based on an IAEA technical report (file WAC.cse);
- an implementation of Example Reference Biosphere 2A (file ERB2A.cse), produced within the IAEA’s BIOMASS programme;
- an implementation of Example Reference Biosphere 2B (file ERB2B.cse), produced within the IAEA’s BIOMASS programme;
- a model of dispersion and deposition of naturally occurring radioactive material (NORM) discharged from a chimney stack at a mineral smelting plant (file PointSource.cse), based on a scenario considered within the IAEA programme on Environmental Modelling for Radiation Safety (EMRAS);
- a model of historical NORM wastes containing enhanced concentrations of naturally occurring radionuclides (file AreaSource.cse), based on a scenario considered within the IAEA programme on Environmental Modelling for Radiation Safety (EMRAS);
- a scoping model of release of gases from a historical landfill site to buildings (file Landfill.cse);
- a biokinetic model for lead (file PbBiokinetic.cse);
- a Gaussian plume model for I-129 (file R91.cse);
- an implementation of a biosphere model developed within the NEA PSACOIN model intercomparison exercise (file Level1B.cse); and
- a simple model of C-14 migration in a Canadian Shield lake (file C14.cse).

Note that many of the parameters in the examples have been locked and can be viewed by checking ‘Show Locked Parameters’ under the ‘Filters’ option on the Parameters tab.

4.1 IAEA Trench Training Case

The trench case (Trench.cse) illustrates a simple ‘total systems’ model for the assessment of a generic near-surface radioactive waste disposal facility (see Figure 23 and Figure 24).
The model was developed for use within an IAEA training course on Safety Assessment of Near Surface Low and Intermediate Level Radioactive Waste Disposal Facilities, and is described in IAEA [2002].

The example provided relates to a generic ‘trench style’ disposal facility (‘Waste’) in which unit activity, $10^{12}$ Bq, of a variety of typical radionuclides has been disposed. Two potential exposure scenarios are considered:

- A groundwater pathway scenario, whereby radionuclides are leached from the wastes into the unsaturated zone and aquifer, through which the radionuclides may be transported. Contaminated water is abstracted from a well, situated 100m from the centre of the disposal facility. Exposure of humans by contaminated soil and irrigation water is then calculated, considering a variety of exposure pathways.

- A human intrusion scenario, whereby waste material is abstracted and dispersed on land that is subsequently used for agriculture.

The user can choose which scenario is assessed in a calculation via the NameSet Option tab on the main panel. The permit for the demonstration version of AMBER also allows an additional compartment, two additional transfers and two additional parameters to be created. The initial inventory of radionuclides in the waste can be edited via the Inventory parameter. Any other changes to the model will result in a permit violation error.

4.2 IAEA ISAM Vault Test Case

This model is based on the Vault Test Case from the IAEA’s coordinated research project concerning Improvement of Safety Assessment Methodologies for Near Surface Disposal Facilities (ISAM) and is fully documented in IAEA [2004].
The case (Vault.cse) represents a simple model for a surface concrete vault disposal facility for low activity radioactive waste. The waste is grouted into drums, the drums are emplaced into concrete cubes, which are then backfilled with grout before being stacked into concrete vaults. The waste is therefore represented as a monolith of concrete. Over time radionuclides are taken to leach from the wastes, through an unsaturated zone to an underlying aquifer. The aquifer is taken to flow a few hundred metres to a groundwater well that is used for domestic and agricultural purposes after a few hundred years.

The permit for the demonstration version of AMBER allows an additional compartment, two additional transfers and two additional parameters to be created. The amount of each radionuclide initially disposed can also be changed via the Disposal parameter. Any other changes to the model will result in a permit violation error.

**Figure 25. Screenshot showing the Submodels of the ISAM Vault Test Case**

**Figure 26. Screenshot of the Spatial Model for the ISAM Vault Test Case**
4.3 IAEA Waste Acceptance Criteria Example

This case is based on one of the scenarios described in IAEA [2003] as an illustration for how to calculate waste acceptance criteria (WAC) for radioactive waste disposal. The case represents analysis of an off-site, post-closure scenario and the derivation of associated acceptance criteria. The IAEA report describes how to assess a range of scenarios, with the WAC being based on the most limiting scenario for each radionuclide.

The model that is included as an example AMBER file (WAC.cse) includes leaching of radionuclides from the waste, transfer via an aquifer pathway and evaluation of potential exposures arising from use of well water for domestic and agricultural purposes. Two interesting features of the model are (i) that it tracks radioactive decay so that contributions from daughter radionuclides can be accounted for when calculating activity limits for parent radionuclides, and (ii) the compartments representing the aquifer pathway get wider with distance from the disposal facility to simulate transverse dispersion.

The permit for the demonstration version of AMBER allows an additional compartment, two additional transfers and two additional parameters to be created. The amount of each radionuclide disposed can also be edited by changing the Disposal_Act parameter. Any other changes to the model will result in a permit violation error.
Figure 28. Screenshot of the Spatial Model for the Waste Acceptance Criteria Case
4.4 IAEA BIOMASS Example Reference Biosphere 2A

Example Reference Biosphere 2A (ERB2A.cse) is a simple model of radionuclide migration around an agricultural system following groundwater release to a well (see Figure 29). It was developed as part of the IAEA’s Biosphere Modelling and Assessment (BIOMASS) research programme and is described fully in Section C3 of IAEA [2003b].

The permit for the demonstration version of AMBER allows an additional compartment, two additional transfers and two additional parameters to be created. The radionuclide concentration in the well water can be set in the C_w parameter. Any other changes to the model will result in a permit violation error.

![Figure 29. Screenshot of the BIOMASS Example Reference Biosphere 2A Case](image)

4.5 IAEA BIOMASS Example Reference Biosphere 2B

Example Reference Biosphere 2B (ERB2B.cse) is a model of radionuclide release to the biosphere via groundwater discharge to a small surface water catchment (see Figure 30). It was developed as part of the IAEA’s Biosphere Modelling and Assessment (BIOMASS) research programme and is described fully in Section C4 of IAEA [2003b].

The permit for the demonstration version of AMBER allows an additional compartment, two additional transfers and two additional parameters to be created. The radionuclide concentration in the groundwater discharging to the surface can be set in the C_gw pa-
parameter. Any other changes to the model will result in a permit violation error.

Figure 30. Screenshot of the BIOMASS Example Reference Biosphere 2B Case

Figure 31. Screenshot of the Spatial Model for the Example Reference Biosphere 2B Case
4.6 Point Source NORM Example

This case (PointSource.cse) provides an example of modelling Rn-222, Pb-210 and Po-210 following discharge from a single chimney stack at a mineral smelting plant. The model calculates doses to two potential exposure groups (PEGs) residing on fields close to the stack. The model uses a probabilistic approach coupled with Gaussian plume dispersion to evaluate the average concentration above each field, which is then used as input to a model of deposition, transport and potential exposures. The main inputs are:

- the release rate from the stack (Q, Bq s\(^{-1}\)); and
- the Pasquill Atmospheric Stability category (P).

The main outputs are the calculated doses to the two PEGs (D\(_{\text{tot}}\), mSv y\(^{-1}\)).

The case represents one of the hypothetical scenarios modelled within the Naturally Occurring Radioactive Material (NORM) working group of the IAEA’s programme on Environmental Modelling for Radiation Safety (EMRAS). The model is described in Walke (2015) and the scenarios are described in more detail in Appendix III of IAEA (2007).

The permit for the demonstration version of AMBER allows two additional parameters to be created. The Pasquill stability category can be changed via the ‘P’ parameter and the

![Hypothetical Point Source Model](image)

**Environmental Modelling for Radiation Safety (EMRAS)**

*Naturally Occurring Radioactive Materials (NORM)*

- The main inputs are the effective release rate (Q) and the Pasquill Stability Category (P).
- The main outputs are calculated doses (D\(_{\text{tot}}\)) to two potential exposure groups (PEGs) associated with fields close to the stack.
- Note that all of the model parameters can be viewed by checking the ‘Show Locked Parameters’ box under ‘Filters’ on the Parameters tab.


release rate for each radionuclide from the stack can be modified via the ‘Q’ parameter. Any other changes to the model will result in a permit violation error.

4.7 Area Source NORM Example

The model (AreaSource.cse) is based on an area source consisting of a historical layer of waste contaminated with naturally occurring U-238 series radionuclides and covered by a layer of soil 2 m thick. The model (see Figure 33) represents infiltration to an aquifer below and subsequent advective flow with groundwater. Erosion of the cover is also represented. Potential exposures arise from the use of contaminated groundwater for domestic and agricultural purposes, as well as use of the soil directly above the waste and inhalation of dust and radon.

The case represents one of the hypothetical scenarios modelled within the Naturally Occurring Radioactive Material (NORM) working group of the IAEA’s programme on Environmental Modelling for Radiation Safety (EMRAS). The model is described in Walke (2015) and the scenarios are described in more detail in Appendix III of IAEA (2007).

The permit for the demonstration version of AMBER allows an additional compartment, two additional transfers and two additional parameters to be created. The initial inventory in the waste can also be set via the InitialInventory parameter. Any other changes to the model will result in a permit violation error.

![Figure 33. Screenshot of the Submodels in the Area Source NORM Example](image-url)
4.8 Landfill Gas Model

This case file (Landfill.cse) represents a scoping model exploring the use of AMBER to probabilistically simulate risks associated with the construction of new buildings above an old landfill site. The model (see Figure 34) allows an initial amount of gas to be defined within the landfill, along with the proportions of methane (CH$_4$) and carbon dioxide (CO$_2$). No further gas generation is assumed. Gas can migrate from the landfill via diffusion and through pressure gradients resulting from (i) a drop in atmospheric pressure, and (ii) rising groundwater. The model calculates resulting concentrations in compartments including the building as well as a cupboard.

The model is described in an accompanying technical note [Quintessa, 2015].

The permit for the demonstration version of AMBER allows an additional compartment, two additional transfers and two additional parameters to be created. The initial total gas in the landfill can be set via the TotGas_Start parameter, and the mass fraction of the gases set via the MassFraction parameter. Any other changes to the model will result in a permit violation error.

![Figure 34. Screenshot of the Landfill Gas Model](image-url)
4.9 Biokinetic Model for Lead

The Leggett [1993] Biokinetic Model for Lead is that used as the basis for the ICRP lead model (see Figure 35). The model represents the age-dependent behaviour of lead, a heavy metal, in the human body after an initial amount has been deposited. The model (PbBiokinetic.cse) is currently set up with a nominal amount of lead (1 gram) in the stomach compartment. Initial amounts of lead can be specified for any of the compartments via the ‘InitialAmount’ parameter and the age of the individual can be set via the Age parameter.

This model demonstrates the application of AMBER in modelling the migration of a non-radioactive contaminant. Note that the model does not include the non-linearities relevant to high lead exposures as discussed in Leggett [1993].

The permit for the demonstration version of AMBER allows the InitialAmount and Age parameters to be modified, one additional transfer and two additional parameters. Any other changes to the model will result in a permit violation error.

![Biokinetic Model for Lead](image)

Figure 35. Screenshot of the Biokinetic Model for Lead

4.10 Gaussian Plume Atmospheric Dispersion Model

A simple compartment model (R91.cse, see Figure 36) has been developed to represent the deposition of I-129 onto a pond following its discharge into the atmosphere. The source term to the pond incorporates a Gaussian plume model to derive the atmospheric concentration at a defined position in relation to the release into the atmosphere. The model is based on the National Radiological Protection Board (NRPB) publication R91.
and is documented in Walke [2005]. The permit for the demonstration version of AMBER allows the atmospheric conditions to be modified along with the ground roughness and the position of the pond in relation to the discharge. This can be achieved through editing the parameters listed below. Any other changes to the model will result in a permit violation error.

- The Smith’s stability parameter ($P$).
- The roughness length in m ($z_0$).
- The effective release height in m ($h$).
- The horizontal distance from the source in m ($x$).
- The cross-wind distance from the centre of the plume in m ($y$).
- The observation height above ground in m ($z$).
- The deposition velocity in m s$^{-1}$ ($v_d$).

![AMBER Example Including Basic Atmospheric Dispersion Model](image)

**Figure 36. Screenshot of the Gaussian Plume Example Case**

### 4.11 PSACOIN 1B Probabilistic Biosphere Model

The PSACOIN 1B exercise [NEA, 1993] was an international code inter-comparison. The model for the comparison was devised to test models for a realistic case, which incorporated probabilistic treatment of some parameters. This model has been incorporated into AMBER (see Figure 37) and compares favourably with those reported. It is included here should the user wish to undertake their own comparison with PSACOIN results (Level1B.cse). This model is set up for a full Monte Carlo run, but it is possible to calculate the model deterministically by selecting ‘Best Estimates’ in the Calculate dialog.
The permit for the demonstration version of AMBER allows an additional compartment, two additional transfers and two additional parameters to be created. The initial amount of the contaminants in the 'source' compartment can be edited via the M_on parameter. Any other changes to the model will result in a permit violation error.

Figure 37. Screenshot of the PSACOIN Level 1B Example Case

### 4.12 Model for C-14 in a Canadian Shield Lake

A simple model (C14.cse) has been included to illustrate the basic method of compartment modelling – the concept of contaminants, compartments and transfers (see Figure 38). The model represents the migration and accumulation of C-14 in a Canadian Shield lake and is an implementation of a model of the second Biospheric Model Validation Study [BIOMOVS II, 1996]. Note that the half-life for C-14 has been updated from that used in the original study to 5730 years, consistent with [ICRP, 2008].

This model comprises a series of compartments representing the key parts of the system – water, sediment, fish and phytoplankton. Water outflow and exchange of $^{14}$CO$_2$ with the
atmosphere are also represented. The transfer rates between these compartments have already been computed, taking into account appropriate mathematical expressions and data. Consequently, only numerical values are given for transfers, and AMBER’s system of defining parameters is not used.

As well as serving to show the basic requirements for any compartment model, this example illustrates the benefits of being able to define parameters so the transfers etc. can be more easily understood and changed in a finished model.

The permit for the demonstration version of AMBER allows an additional compartment, two additional transfers and two additional parameters to be created. The initial amount of C-14 in the water column can be edited via the InitialAmount parameter. Any other changes to the model will result in a permit violation error.

---

Figure 38. Screenshot of the BIOMOVS II C-14 Example Case
4.13 References for Section 4


5 Implementing a New Case

A permit with the demonstration version of AMBER allows users to implement a simple compartment model themselves. The model to be implemented is restricted to a simple example of a post-closure ‘total-systems’ model for a radioactive waste disposal facility.

It is worth emphasising that AMBER is a compartment modelling tool that has been developed to enable users to implement their own compartment models from basics. This means that as an AMBER user, you start with a blank screen. Good modelling practice means that users should bring a pre-defined model to AMBER; systematic methodological approaches exist that help in the development and documentation of such models, for example, that described in IAEA Specific Safety Guide 23 [IAEA, 2012].

5.1 The Demonstration Model

The simple model represents a geological radioactive waste disposal facility that is positioned beneath the water table. The contaminants are radionuclides that leach from the waste and into the surrounding aquifer. The aquifer discharges contaminated groundwater into a river, from which drinking water is abstracted.

The model that will be implemented therefore includes:

- four contaminants – representing the Th-230 decay chain;
- eight compartments – one representing the radioactive waste, five representing the aquifer, one representing the river and one representing a sink (for radionuclides that leave the system); and
- seven transfers – one representing leaching from the waste into the aquifer, five representing advective groundwater flow through the aquifer and the discharge into the river, and one representing river flow to the sink.

The model includes a single NameSet (see Section 5.6) listing the compartment types to be modelled (nearfield, geosphere and surface water) and a Mapping from the compartments to the compartment types. The model requires the specification of:

- sixteen standard parameters;
- one lookup time-dependent parameter (representing the hydraulic conductivity of the waste); and
- three observer parameters (that use the calculated amounts to derive compartment concentrations and an effective dose rate to an individual consuming the abstracted water).

Note that the model must be implemented exactly as described; otherwise a permit violation will be encountered. Note also that AMBER is case sensitive so care is needed to ensure that the inputs are entered exactly as described. And note that space characters and ‘-‘ (minus) characters are not allowed in names.

The case cannot be calculated until it has been fully implemented. A fully implemented example (Demonstration.cse) is included for reference.
5.2 Create a New Case

1. Select File | New Case or click on the ‘New Case’ icon on the toolbar.

This will create a blank case that will allow development of the tutorial model to take place.

Note that good practice dictates that the tutorial case should be saved at regular intervals during its development. It is also recommended that the user includes some details in the ‘Case Information’ tab on the left panel; this is accessed via Panels | Case Information (note that you will need to click ‘Apply’ for text to be stored on this tab). The Demonstration permit will allow partially developed case files of this specific model to be saved and re-opened.

5.3 Defining the Base Units

Before setting the values for any parameters etc., the base units for the case need to be defined.

2. Select Options | Units....

3. Ensure that ‘Years’ are selected for the units of ‘Time’ and ‘Becquerels’ are selected for the ‘Amount’ units, then click on ‘OK’.

5.4 Adding Contaminants and Decays

Adding Contaminants is described in Section 5.4.1 and adding Decays is described in Section 5.4.2.

5.4.1 Adding Contaminants

The first step of this demonstration tutorial is to add the contaminants and decays to the blank case. Table 2 lists the contaminants to be included in the demonstration case.

<table>
<thead>
<tr>
<th>Name</th>
<th>Description</th>
<th>Atomic Mass</th>
</tr>
</thead>
<tbody>
<tr>
<td>Th_230</td>
<td>Thorium 230</td>
<td>230</td>
</tr>
<tr>
<td>Ra_226</td>
<td>Radium 226 (ingrown from Th-230)</td>
<td>226</td>
</tr>
<tr>
<td>Pb_210</td>
<td>Lead 210 (ingrown from Th-230)</td>
<td>210</td>
</tr>
<tr>
<td>Po_210</td>
<td>Polonium 210 (ingrown from Th-230)</td>
<td>210</td>
</tr>
</tbody>
</table>

4. Open the ‘Contaminants’ tab on the left panel by clicking on the Contaminants icon on the toolbar, or via Panels | Contaminants + Decays | Contaminants.

5. Open an ‘Add Contaminant’ dialog by clicking on the “Add…” button.

6. Enter the name, description and atomic mass for Th_230, as described in Table 2 and as illustrated in Figure 39, then click on ‘OK’.
7. Repeat steps 5 and 6 for the Ra_226, Pb_210 and Po_210 contaminants with the information described in Table 2.

The Contaminants tab on the left panel should now include all four contaminants, as illustrated in Figure 40.

![Figure 39. Add Contaminant Dialog for Th-230](image)

![Figure 40. Contaminants Tab showing all Four Radionuclides](image)

5.4.2 Adding Decays

Now the radioactive decays can be added to the case. These are listed in Table 3. Note that the NULL contaminant daughter denotes a species that is of no interest in the assessment such as a stable isotope, for example. The instructions below describe how to add decays.

Note that short-lived daughters are taken to be in secular equilibrium with Ra-226 so that the model represents decay directly to Pb-210. The contribution of the short-lived daughters is explicitly accounted for in the dose coefficients, described in Section 5.8.2.

8. Open the ‘Decays’ tab on the left panel by clicking on the Decays icon on the toolbar, or via Panels | Contaminants + Decays | Decays.
9. Click on the ‘Add…’ button to open a ‘Add Decay’ dialog.

10. Enter the information for the decay of Th$_{230}$ to Ra$_{226}$, as described in Table 3, by entering the 9e-6 in the ‘Rate (per Year)’ field; selecting Th$_{230}$ from the ‘Parent’ drop-down list; selecting Ra$_{226}$ from the ‘Daughter’ drop-down list; and then clicking on ‘OK’ (see Figure 41).

Note that the Th$_{230}$ to Ra$_{226}$ decay is now listed on the Decays tab and that it has automatically been assigned the name ‘Th$_{230}$$\rightarrow$Ra$_{226}$’.

11. Repeat steps 9 and 10 for the remaining three decays listed in Table 3.

![Figure 41. Add the Th$_{230}$ Decay](image)

## 5.5 Adding Compartments, Transfers and Submodels

The next stage of the demonstration tutorial is to add the compartments (Section 5.5.1) and transfers (Section 5.5.2), as well as to organise the layout using submodels (Section 5.5.3).

### 5.5.1 Adding Compartments

There are eight compartments to be added to the model, as described in Table 4.
Table 4. Compartments for the Demonstration Case

<table>
<thead>
<tr>
<th>Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Waste</td>
<td>Compartment representing the waste</td>
</tr>
<tr>
<td>Aquifer1</td>
<td>The first aquifer compartment</td>
</tr>
<tr>
<td>Aquifer2</td>
<td>The second aquifer compartment</td>
</tr>
<tr>
<td>Aquifer3</td>
<td>The third aquifer compartment</td>
</tr>
<tr>
<td>Aquifer4</td>
<td>The fourth aquifer compartment</td>
</tr>
<tr>
<td>Aquifer5</td>
<td>The fifth aquifer compartment</td>
</tr>
<tr>
<td>River</td>
<td>River compartment from which drinking water is abstracted</td>
</tr>
<tr>
<td>Sink</td>
<td>Compartment representing the environment downstream of the area of interest</td>
</tr>
</tbody>
</table>

12. Switch focus to the Model tab on the main panel by selecting Panels | Model or by clicking the ‘Model’ tab at the bottom of the main panel.

13. Select the ‘Compartment’ tool from the tool palette adjacent to the Model panel (see Figure 5). This tool allows you to add compartments by clicking in the Model panel.

14. Add a compartment in the Model panel by clicking anywhere in the Model panel. Double-click on the compartment to open the ‘Item Info’ section for the new compartment at the bottom of the left panel.

15. Add the name and description information for the Waste compartment described in Table 4, as illustrated in Figure 42, and click on ‘Apply’.

16. Repeat steps 13 to 15 for the seven remaining compartments listed in Table 4.

![Figure 42. Item Info for the Waste Compartment](image)

The position of items on the Model panel, such as compartments, can be changed by clicking on the Pointer tool in the tool palette and dragging items to their desired location. Note also that items in the model panel can be coloured by right-clicking with the pointer over the item and selecting Colour... from the resulting menu. Figure 43 shows the model window with the eight new compartments added.
### 5.5.2 Adding Transfers

There are seven transfers to be included in the case, these are described in Table 5. The instructions below describe how to add transfers.

#### Table 5. Transfers for the Demonstration Case

<table>
<thead>
<tr>
<th>Name</th>
<th>Donor</th>
<th>Recipient</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Leaching</td>
<td>Waste</td>
<td>Aquifer1</td>
<td>Leaching transfer from the waste to the aquifer</td>
</tr>
<tr>
<td>Advection1</td>
<td>Aquifer1</td>
<td>Aquifer2</td>
<td>Transfer via advective flow from Aquifer compartment 1 to compartment 2</td>
</tr>
<tr>
<td>Advection2</td>
<td>Aquifer2</td>
<td>Aquifer3</td>
<td>Transfer via advective flow from Aquifer compartment 2 to compartment 3</td>
</tr>
<tr>
<td>Advection3</td>
<td>Aquifer3</td>
<td>Aquifer4</td>
<td>Transfer via advective flow from Aquifer compartment 3 to compartment 4</td>
</tr>
<tr>
<td>Advection4</td>
<td>Aquifer4</td>
<td>Aquifer5</td>
<td>Transfer via advective flow from Aquifer compartment 4 to compartment 5</td>
</tr>
<tr>
<td>Discharge</td>
<td>Aquifer5</td>
<td>River</td>
<td>Discharge rate from the aquifer into the river</td>
</tr>
<tr>
<td>Flow</td>
<td>River</td>
<td>Sink</td>
<td>River flow downstream</td>
</tr>
</tbody>
</table>

17. Select the ‘Transfer’ tool from the tool palette (see Figure 5).

18. Click inside the Waste compartment on the Model tab of the main panel and drag the pointer to the Aquifer1 compartment.

This will add a new ‘Waste_TO_Aquifer1’ transfer to the model window, as illustrated in Figure 44.

19. An ‘Item Info’ section for the ‘Waste_TO_Aquifer1’ transfer should automatically appear at the bottom of the Transfers tab on the left panel when the new transfer is created.
20. Enter Leaching as the ‘Transfer Name’, include the description from Table 5, as illustrated in Figure 45, and then click on ‘Apply’.

21. Repeat steps 17 to 20 for the remaining six transfers described in Table 5, ensuring that the correct donor and recipient compartments are used in each case.

Figure 46 provides an illustration of the model window with the seven transfers added.
5.5.3 Creating a Submodel

AMBER includes the capability to create submodels to help with the management and presentation of complex models. Submodels allow sets of compartments to be enveloped within resizable submodel boxes. In the demonstration tutorial, a submodel will be used for the aquifer compartments as an example.

22. Select the ‘Submodel’ tool from the tool palette (see Figure 5).

23. Click and drag a submodel box over the five aquifer compartments, as illustrated in Figure 47.
This will replace the five Aquifer compartments and joining transfers with a blue sub-model box. Double-click on the submodel to view its contents in a new tab on the main panel (see Figure 48); note that the main panel has been split in the screenshot to view both the top level model as well as the submodel contents by clicking on the Submodel tab itself and dragging to the right of the main panel.

Note that submodels can be deleted by clicking on the submodel to highlight it and then pressing <Delete>. If the items within the submodel are to be kept, then click on ‘Keep contents’ when prompted.

![Figure 48. Submodel Tab Viewed Adjacent to the Model Tab](image)

24. Right-click on the ‘Submodel1’ box in the Model window and select *Edit Submodel*… to open the Item Info section for the submodel at the bottom of the left panel. Type *Aquifer* as the ‘Submodel Name’ and then click on ‘Apply’.

25. Resize the new ‘Aquifer’ submodel box by clicking on it and dragging the corners of the box (see Figure 49).

![Figure 49. Resizing the Submodel Box](image)
5.6 Adding a NameSet and Mapping

The next stage of the demonstration tutorial is to add a NameSet that describes the types of compartment in the model and to add a Mapping from Compartments to this new NameSet.

5.6.1 Adding a NameSet

26. Click on the ‘NameSets’ icon in the toolbar or click on Panels | NameSets to open the NameSets tab on the left panel.

27. Click on ‘Add’ to open an ‘Item Info’ section on the NameSet tab and type CompType as the NameSet name, along with Compartment Type under the NameSet description.

28. Click on ‘Add Name’ to open a ‘New NameSet Item Information’ dialog. Type Nearfield as the item name and click on ‘OK’.

29. Repeat step 28 to add the items Geosphere and SurfaceWater to the ‘CompType’ NameSet (see Figure 50) then click on ‘Apply’.

![Image of NameSet Item Info]

Figure 50. NameSet Item Info

Note that care is needed to specify these names correctly because AMBER is case sensitive. For example, mistyping SurfaceWater as Surfacewater will result in permit file errors later in the tutorial.
5.6.2 Adding a Mapping

30. Select the ‘Mappings’ icon on the toolbar or Panels | Mappings to open the Mappings tab on the left panel and click on ‘Add’ to open a ‘Set Multiplicity’ dialog.

31. Select ‘Compartments’ from the ‘Map from’ drop-down list and the new ‘Comp-Type’ NameSet from the ‘to’ drop-down list and click ‘OK’ (see Figure 51).

![Set Multiplicity Dialog for a New Mapping](image)

Figure 51. Set Multiplicity Dialog for a New Mapping

This opens a ‘New Mapping’ tab on the main panel, in which the model compartments can be assigned to compartment types. Note that comments can be assigned to each individual mapping in the table, although none are required for the tutorial case.

32. Enter Comp_to_CompType as the mapping name.

33. Click in the empty cell under ‘CompType’ next to the ‘default’ item.

34. Select ‘Geosphere’ from the drop-down list that appears in that cell.

This means that by default, the compartments are ‘mapped’ to the Geosphere compartment type. The next step is to assign the SurfaceWater type to the River compartment and the Nearfield type to the Waste compartment.

35. Scroll down the list of compartments and click on the blank cell under ‘CompType’ next to the River compartment.

36. Select ‘SurfaceWater’ from the drop-down list.

37. Repeat steps 35 and 36 to map from the Waste compartment to the ‘Nearfield’ type (see Figure 52).

38. Click on ‘OK’ in the New Mapping tab. This closes the tab on the main panel and the new Comp_to_CompType mapping will now be displayed on the Mappings tab on the left panel.
Figure 52. Mapping from Compartments to Compartment Types
5.7 Setting Start Amounts

The demonstration tutorial requires 1 TBq (1 \( \times 10^{12} \) Bq) of Th\textsubscript{230} to be specified as the start amount in the Waste compartment.

39. Navigate to the Model tab on the main panel and double-click on the Waste compartment to open the associated Item Info section at the bottom of the Compartments tab on the left panel.

40. Click on the ‘Set Start Amount...’ button to open the ‘Set Start Amount...’ dialog.

41. Click in the blank cell next to Th\textsubscript{230} in the Waste column, type 1e12 and click on the tick button (see Figure 53).

42. Click on ‘OK’ in the Set Start Amount dialog and ‘Apply’ on the Compartments tab.

![Figure 53. Set Start Amount Dialog](image-url)
5.8 Adding Parameters

AMBER allows the user to create any number of parameters, which can be indexed over any of the NameSets. The NameSets over which a parameter is indexed are referred to as its ‘multiplicity’ in AMBER terminology. The following subsections describe how to add parameters with no, single, and dual multiplicity.

5.8.1 Adding Standard Parameters with No Multiplicity

The model includes six parameters with no multiplicity; these are described in Table 6. The steps below describe how to add parameters with no multiplicity.

43. Click on the Parameters icon in the toolbar or Panels | Parameters to open the ‘Parameters’ tab on the left panel.

44. Click on the ‘Add’ button on the Parameters tab to open the ‘Type’ dialog. This is set to ‘Standard’ by default so click on ‘OK’ to open the ‘Set Multiplicity’ dialog.

45. Since the first parameters to be entered are not indexed over any NameSets, then both drop-down boxes should remain set to ‘None’. Self-referencing is not relevant to the tutorial, so the third drop-down box should remain set to nothing. Click on ‘OK’ to open a ‘New Parameter’ tab on the main panel.

46. Type $H_{\text{grad}}$ for the ‘Parameter Name’, type ‘-’ for the ‘Units’ (i.e. unit less), hydraulic gradient of aquifer as the ‘Parameter Description’ and 0.01 as the ‘Parameter Expression/Value’ (see Figure 54). Then click on ‘OK’. This closes the main panel tab and the new parameter is now listed on the Parameter tab on the left panel.

47. Repeat steps 44 to 46 for each of the five remaining parameters with no multiplicity described in Table 6. Note that the $K\text{_{Waste}}$ parameter (referenced in the algorithm for $q\text{_{Waste}}$) is a time-dependent parameter that is added later (see Section 5.8.4).

<table>
<thead>
<tr>
<th>Name</th>
<th>Units</th>
<th>Description</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>$H_{\text{grad}}$</td>
<td>-</td>
<td>Hydraulic gradient of aquifer</td>
<td>0.01</td>
</tr>
<tr>
<td>$K$</td>
<td>m/y</td>
<td>Hydraulic conductivity of the aquifer</td>
<td>315</td>
</tr>
<tr>
<td>$q$</td>
<td>m/y</td>
<td>Darcy velocity in aquifer</td>
<td>$K\cdot H_{\text{grad}}$</td>
</tr>
<tr>
<td>$q\text{_{Waste}}$</td>
<td>m/y</td>
<td>Darcy velocity in the waste</td>
<td>$K\text{<em>{Waste}}\cdot H</em>{\text{grad}}$</td>
</tr>
<tr>
<td>$Q\text{_{flow}}$</td>
<td>m$^3$/y</td>
<td>Flow rate of the river</td>
<td>1e6</td>
</tr>
<tr>
<td>$\text{Ing_Wat}$</td>
<td>m$^3$/y</td>
<td>Ingestion rate of drinking water by human</td>
<td>0.73</td>
</tr>
</tbody>
</table>
5.8.2 Adding Standard Parameters with Single Multiplicity

The model includes eight parameters with single multiplicity; these are described in Table 7. The steps below describe how to add parameters with single multiplicity.

Note that the round brackets in the expression for $V$, $(\text{Comp}_\text{to}_\text{CompType})$, pick up the Comp_to_CompType Mapping that was implemented in Section 5.6.2. This is needed to convert parameters defined over compartment types (L and D) to compartment multiplicity.

48. On the Parameters tab, repeat step 45 to open the ‘Set Multiplicity’ dialog.

49. Select the ‘CompType’ NameSet from the drop-down menu under ‘Parameter defined for each’ (see Figure 55) and click on ‘OK’ to open the ‘New Parameter’ tab on the main panel (see Figure 56).

50. Type $D$ as the ‘Parameter Name’, $\text{m}$ as the ‘Units’ and type Depth/thickness of compartment for the ‘Parameter Description’.

51. Given that the parameter has multiplicity over the ‘CompType’ NameSet, it is possible to define values for each compartment type item. Enter the value 10 for the Nearfield, 10 for the Geosphere and a value of 2 for the SurfaceWater (see Figure 56). Note that comments can be assigned against each individual parameter entry, although none are required for the tutorial case. Click on ‘OK’ to apply the changes, this will close the Parameter tab for $D$ on the main tab.
52. Repeat steps 48 to 51 for each of the remaining seven parameters with single multiplicity described in Table 7, taking note of the multiplicity NameSet in each case.

Note also that it is possible to use the default cell in parameter tables to avoid repetition in input data. However, for the purpose of the demonstration tutorial, please take care to match the specification in Table 7 exactly, otherwise you will encounter permit errors (i.e. use of the default cell is only specified for the parameter $w$).

Note that users can navigate around entries within a parameter table by using the <Up> and <Down> arrow keys, and by using <Ctrl + Left> or <Ctrl + Right> arrows, or <Tab> and <Shift + Tab>.

### Table 7. Standard Parameters with Single Multiplicity

<table>
<thead>
<tr>
<th>Name</th>
<th>Multiplicity</th>
<th>Units</th>
<th>Description</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>D</td>
<td>CompType</td>
<td>m</td>
<td>Depth/thickness of compartment</td>
<td>Nearfield 10</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Geosphere 10</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>SurfaceWater 2</td>
</tr>
<tr>
<td>L</td>
<td>CompType</td>
<td>m</td>
<td>Length of compartment in direction of groundwater flow</td>
<td>Nearfield 100</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Geosphere 20</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>SurfaceWater 100</td>
</tr>
<tr>
<td>W</td>
<td>Compartments</td>
<td>m</td>
<td>Width of compartment</td>
<td>default 100</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>River 5</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Sink 1</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Waste 10</td>
</tr>
<tr>
<td>V</td>
<td>Compartments</td>
<td>m$^3$</td>
<td>Volume of compartment (default)</td>
<td>$L(\text{Comp_to_CompType}) \times W \times D(\text{Comp_to_CompType})$</td>
</tr>
<tr>
<td>Kd</td>
<td>Contaminants</td>
<td>m$^3$/kg</td>
<td>Distribution coefficients</td>
<td>Th$_{230}$ 1</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Ra$_{226}$ 0.1</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Pb$_{210}$ 0.1</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Po$_{210}$ 0.1</td>
</tr>
<tr>
<td>rho_b</td>
<td>CompType</td>
<td>kg/m$^3$</td>
<td>Bulk density of materials</td>
<td>Nearfield 2000</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Geosphere 2000</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>SurfaceWater 1</td>
</tr>
<tr>
<td>theta_w</td>
<td>CompType</td>
<td>-</td>
<td>Water filled porosity of each medium</td>
<td>Nearfield 0.25</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Geosphere 0.25</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>SurfaceWater 1</td>
</tr>
<tr>
<td>DC_Ing</td>
<td>Contaminants</td>
<td>Sv/Bq</td>
<td>Dose coefficients for ingestion</td>
<td>Th$_{230}$ 2.1e-7</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Ra$_{226}$ 2.8e-7</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Pb$_{210}$ 6.9e-7</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Po$_{210}$ 1.2e-6</td>
</tr>
</tbody>
</table>
Figure 55. Multiplicity Dialog in Setting-up Parameter D

Figure 56. New Parameter Tab with Single Multiplicity
5.8.3 Adding Standard Parameters with Dual Multiplicity

The model includes two parameters with dual multiplicity; these are described below.

Retardation

53. On the Parameters tab, repeat step 45 to open the ‘Set Multiplicity’ dialog.

54. Select ‘Contaminants’ from the drop-down menu under ‘Parameter defined for each’.

55. Select ‘Compartments’ from the drop-down menu under ‘and for each’ (see Figure 57) and click on ‘OK’ to open the ‘New Parameter’ tab on the main panel. Self-referencing is not used in this tutorial, so the third drop-down box should remain blank.

56. Type R as the ‘Parameter Name’, type ‘-’ as the ‘Units’ and Retardation as the ‘Parameter Description’.

57. In the ‘default, default’ cell of the table, enter the following expression (see Figure 58):

\[ 1 + \rho_b(\text{Comp}_\text{to}_\text{CompType}) \times K_d / \theta_w(\text{Comp}_\text{to}_\text{CompType}) \]

Note that round brackets are again used to pick up the ‘Comp_to_CompType’ mapping to convert parameters defined over compartment types (\(\rho_b\), \(K_d\) and \(\theta_w\)) to compartment multiplicity.

58. Click on ‘OK’ to apply the changes, which closes the New Parameter tab on the main panel.

![Set Multiplicity dialog](Figure 57. Multiplicity Dialog in Setting-up Parameter R)
59. On the Parameters tab, repeat step 45 to open the ‘Set Multiplicity’ dialog.

60. Select ‘Contaminants’ from the drop-down menu under ‘Parameter defined for each’.

61. Select ‘Compartments’ from the drop-down menu under ‘and for each’ and click on ‘OK’ to open the ‘New Parameter’ tab on the main panel.

62. Type $\lambda_A$ as the ‘Parameter Name’, $/y$ as the ‘Units’ and Transfer due to advective groundwater flow between compartments as the ‘Parameter Description’.

63. In the ‘default, default’ cell of the table, enter the following expression as a single line:

$$q / (\theta_w(\text{Comp_to_CompType}) \ast L(\text{Comp_to_CompType}) \ast R)$$

64. Scroll to the right, click on the ‘default’ cell for the ‘Waste’ compartment and enter the following:

$$q_{\text{Waste}} / (\theta_w(\text{Comp_to_CompType}) \ast L(\text{Comp_to_CompType}) \ast R)$$

65. Enter 0 in the ‘default’ cells for the ‘River’ and ‘Sink’ compartments (see Figure 59).

66. Click on ‘OK’ to apply the changes, which closes the New Parameter tab on the main panel.

**Advection Transport**

**Figure 58. New Parameter Tab for R**
5.8.4 Adding Lookup Time-Dependent Parameters

In this example case, the hydraulic conductivity of the waste is assumed to be time-dependent, with a value of 3.15 m/y for the first 100 years, followed by a linear increase to 315 m/y after 1000 years. This is achieved by adding a ‘Lookup Time-dependent’ parameter with the following steps.

67. From the Parameters tab on the left panel, click on ‘Add’ to open the ‘Type’ dialog.
68. Select the ‘Lookup Time-dependent’ parameter type and click on ‘OK’ to open a ‘New Parameter’ tab on the main panel.
69. Type in $K_{\text{Waste}}$ as the ‘Name’, m/y as the ‘Units’ and Hydraulic conductivity of the waste as the ‘Parameter Description’.
70. Check the ‘Linear Interpolation’ box.
71. Click on ‘Add time...’ to open the ‘Add switch time’ window, type 0 and click on ‘OK’.
72. Repeat step 71 to add 100, 1000 and 100000 as additional time points.
73. Click in the cell next to the ‘=0’ time, type 3.15 [m/y] in the ‘Parameter Expression/Value’ field and click the tick button.
74. Click on the cell next to the ‘=100’ time, type 3.15 [m/y] and click the tick button.
75. Click on the cell next to the ‘=1000’ time, type 315 [m/y] and click the tick button.
76. Click on the cell next to the ‘100000’ time and type 315 \([\text{m/y}]\) and click the tick button (see Figure 60).

77. Note that comments can be assigned to each individual value, although none are required for the tutorial case. Click on ‘OK’ to return to apply the changes, which closes the New Parameter tab on the main panel.

Note that when linear interpolation is used for a Lookup time-dependent parameter, it only uses values directly relating to the times you specify. It does not use the value before the first time point, nor the value after the last time point. For this reason those values are set to zero, and cannot be edited.
Figure 60. New Lookup Time-Dependent Parameter
5.8.5 Adding Observers

Parameters that depend, directly or indirectly, on either calculated transfer fluxes or compartment amounts need to be added as ‘observer’ parameters, since the case must be calculated for them to be evaluated. The example case includes three observer parameters, which can be added as described below.

**Compartment Concentrations (Dual Multiplicity)**

78. On the Parameters tab on the left panel, click on ‘Add ...’ to open the ‘Type’ dialog.
79. Select ‘Observer’ as the parameter type and click on ‘OK’ to open the ‘Set Multiplicity’ dialog.
80. Select ‘Contaminants’ from the drop-down menu under ‘Observer defined for each’.
81. Select ‘Compartments’ from the drop-down menu under ‘and for each’ and click on ‘OK’ to open a ‘New Parameter’ tab on the main panel.
82. Type C as the ‘Parameter Name’, Bq/m3 as the ‘Units’ and Concentration of radionuclides in compartment as the ‘Parameter Description’.
83. Enter Amount/V as the definition for the ‘default/default’ cell (see Figure 61), then click on ‘OK’ to apply the changes and close the New Parameter tab.

![Figure 61. New Parameter Tab for the Compartment Concentration Parameter](image)
Note that ‘Amount’ is the internal ‘read-only’ parameter that AMBER creates once a case has been calculated to store the time-history of the compartment inventories.

**Drinking Water Dose Rate (Single Multiplicity)**

84. On the Parameters tab on the left panel, click on ‘Add’ to open the ‘Type’ dialog.
85. Select ‘Observer’ as the parameter type and click on ‘OK’ to open the ‘Set Multiplicity’ dialog.
86. Select ‘Contaminants’ from the drop-down menu under ‘Observer defined for each’ and click on ‘OK’.
87. Type E_Wat as the ‘Parameter Name’, Sv/y as the ‘Units’ and Annual individual dose to humans from consumption of river water as the ‘Parameter Description’.
88. Type C[River]*Ing_Wat*DC_Ing as the parameter definition in the ‘default’ cell (see Figure 62) then click on ‘OK’.

Note that the square brackets are used to define the value for a particular NameSet item. For example, C[River] is used to define the value of parameter C (concentration) for the ‘River’ compartment.

**Total Drinking Water Dose Rate (No Multiplicity)**

89. On the Parameters tab on the left panel, click on ‘Add’ to open the ‘Type’ dialog.
90. Select ‘Observer’ as the parameter type and click on ‘OK’ to open the ‘Set Multiplicity’ dialog.
91. Leave both multiplicity fields on ‘None’ and click on ‘OK’.
92. Type E_WatTot as the ‘Parameter Name’, Sv/y as the ‘Units’ and Total annual individual dose to humans from consumption of river water as the ‘Parameter Description’.
93. Type sum(E_Wat,Contaminants) as the parameter definition (see Figure 63) then click on ‘OK’.

Note that sum(Expression,NameSet) is a function that is available in AMBER to sum an expression over the defined NameSet. In this way, the drinking water dose is summed over all of the radionuclides to give the total dose.
Figure 62. New Parameter Tab for the Drinking Water Dose Parameter
Figure 63. New Parameter Tab for the Total Dose Parameter
5.9 Specifying Transfer Rates

The next stage of the tutorial is to specify the transfer rates between the model compart-
ments.

94. Navigate to the Model tab and double-click on the ‘Leaching’ transfer to bring up
the Item Info section for that transfer at the bottom of the left panel.

95. Click on the ‘Set Transfer Rate...’ button to bring up the Set Transfer Rate dialog for
the Leaching transfer.

96. Enter $\lambda_A$ (donor) into the default cell for the Leaching Transfer (see Figure 64).

97. Click on ‘OK’ to exit the Set Transfer Rate window and on ‘Apply’ at the bottom of
the left panel to confirm the changes.

Figure 64. Setting the Rate for the Leaching Transfer
This ensures that the lambda_A parameter (created in Section 5.8.3) is used in calculating the transfer rate of contaminants between the ‘Waste’ and ‘Aquifer1’ compartments. ‘(donor)’ is an internal mapping from Transfers to donor Compartments (click on the ‘Internal (Read Only)’ check-box on the Mapping tab to view it). The mapping ensures that the expression for the Waste compartment in lambda_A is used to define the Leaching transfer.

98. Repeat steps 94 to 97 to specify the transfer rates for the remaining six transfers described in Table 8. Note that the transfers within the Aquifer sub-model can be reached by double-clicking on Aquifer sub-model on the Model tab.

Table 8. Transfer Rates

<table>
<thead>
<tr>
<th>Transfer</th>
<th>Specification</th>
</tr>
</thead>
<tbody>
<tr>
<td>Leaching</td>
<td>lambda_A(donor)</td>
</tr>
<tr>
<td>Advection1</td>
<td>lambda_A(donor)</td>
</tr>
<tr>
<td>Advection2</td>
<td>lambda_A(donor)</td>
</tr>
<tr>
<td>Advection3</td>
<td>lambda_A(donor)</td>
</tr>
<tr>
<td>Advection4</td>
<td>lambda_A(donor)</td>
</tr>
<tr>
<td>Discharge</td>
<td>lambda_A(donor)</td>
</tr>
<tr>
<td>Flow</td>
<td>Q_flow/V[River]</td>
</tr>
</tbody>
</table>

5.10 Adding Result Times

Before the case can be calculated, AMBER needs to be told of the times at which to store calculated contaminant fluxes and compartment amounts. These times are called ‘Result Times’. Note that they are independent of the solve-steps that AMBER uses.

99. Click on the ‘Result Times’ icon on the toolbar or Panels | Result Times to bring up the ‘Result Times’ panel on the left panel.

100. Click on ‘Add Times >>’ to expand the ‘Add Result Times’ tab.

101. Select ‘Arithmetic Sequence’, type 1 as the ‘First Time (Years)’, 9 as the ‘Last Time (Years)’, 9 as the ‘Number of Times’ and then click ‘Add’.

102. Repeat steps 100 and 101 to include the remaining sequences described in Table 9.

Table 9. Result Times

<table>
<thead>
<tr>
<th>First Time</th>
<th>Last Time</th>
<th>Number of Times</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>9</td>
<td>9</td>
</tr>
<tr>
<td>10</td>
<td>90</td>
<td>9</td>
</tr>
<tr>
<td>100</td>
<td>900</td>
<td>9</td>
</tr>
<tr>
<td>1000</td>
<td>9000</td>
<td>9</td>
</tr>
<tr>
<td>10000</td>
<td>100000</td>
<td>10</td>
</tr>
</tbody>
</table>
5.11 Calculating and Investigating Results

The tutorial case is now ready to be checked, calculated and the results explored.

5.11.1 Checking Parameters and Units

Before calculating the case, users are recommended to use the ‘Check Parameters’ function to ensure that the model can be solved and to check the case for units consistency.

103. Click on the Check Parameters icon on the toolbar or Calculation | Check Parameters.

This results in a ‘Check Parameters’ tab on the main panel, which displays any parameter errors and units inconsistencies that have been encountered, or if there are any mapping problems. If errors are reported, the case has not been implemented correctly, please check the error message(s) and resolve the problem (see Section 14 of the AMBER 6.7 Reference Manual for tips on solving parameter errors).

5.11.2 Calculating the Case

The demonstration tutorial case is now complete and can be calculated.

104. Click on the Calculate icon in the toolbar or Calculation | Calculate to open the ‘Calculate’ dialog, then click on ‘OK’.

Note that the linearly interpolated hydraulic conductivity of the Waste compartment means that the intelligent ‘Time-Step’ solver is selected by default. A ‘Progress’ window reports the progress of the calculation (see Figure 65).

If the Demonstration tutorial has not been implemented exactly as specified, then a ‘Permit’ error is shown listing the items that need to be corrected before the case can be calculated.

![Figure 65. Progress Window](image)
5.11.3 Investigating the Results

Results can either be reported to a text-based file or plotted to graphs.

**Reporting Results**

105. Click on the ‘Write Report’ icon on the toolbar or Results | Report... to bring up the ‘Report Information’ dialog (see Figure 66).

106. Use the Report Information dialog to specify the results you are interested in. For example, parameters of interest (e.g. E_WatTot for the total dose) can be selected via the ‘Parameters...’ button.

107. Click on ‘OK’ to bring up a ‘Report file’ window through which it is possible to specify the name and location for the text based report file. Then click on ‘Save’.

108. Open the resulting Report File in a text editor to view the results.

Note that the solver controls permit very small negative results in relation to the total amount of the contaminants in the system; this feature is typical of numerical solvers. In the full AMBER release, the user can specify the minimum values for parameters as being zero, thereby avoiding reporting of negative observer amounts.
Graphing Results

109. Click on the ‘Create Chart’ icon on the toolbar or Results | Graph... to bring up the ‘Graph Type’ dialog (see Figure 67).

110. Select the options required and click on ‘OK’ to bring up the ‘Graph Plots’ dialog.

111. Select the required combination of parameters and NameSet items (where appropriate), then click on ‘OK’ to plot the graph. For example, you could try plotting E_Wat on the y-axis versus time on the x-axis (see Figure 68).

Note that AMBER has its own graphing package, which is used by default and results in the chart being presented on a new tab on the main panel. However, it is also possible to plot graphs directly to Microsoft Excel by selecting the ‘Output to Excel’ option in the ‘Graph type’ dialog. If the use wishes to output the data directly to file without graphing, the ‘Output to CSV’ option can be selected in the ‘Graph Type’ dialog.

Figure 67. Graph Type Dialog
5.11.4 Additional Changes

The permit file has been set up for the demonstration tutorial to enable users to make some changes to the completed case:

- the user is able to change the values for the $H_{\text{grad}}$, $K$, $K_{\text{Waste}}$ and $Q_{\text{flow}}$ parameters, though these must remain as Standard parameters;

- the user is able to add four new sampled parameters, which can be called upon by $H_{\text{grad}}$, $K$, $K_{\text{Waste}}$ or $Q_{\text{flow}}$.

These permitted changes allow the user to run AMBER in probabilistic mode and investigate the relative sensitivity of the model to these different parameters and parameter ranges. See Section 9.1.7 of the AMBER 6.7 Reference Manual for how to add a sampled parameter and Section 10 of the AMBER 6.7 Reference Manual for how to undertake probabilistic calculations.

5.12 References for Section 5

6 Feedback

AMBER is developed in response to user feedback; therefore, the AMBER support team encourages users to provide feedback about the software. In particular:

- if you identify a new feature that would enable AMBER to better address your modelling needs;
- if you have a suggestion to enhance existing AMBER features;
- indeed, we welcome any comments about the software.

The spatial awareness and 3D visualisation capabilities included in AMBER 6.7 represent key functions that are unique in this calibre of contaminant transport and risk assessment codes. The capabilities provide a basis for further enhancement, we therefore particularly welcome feedback and ideas for how to extend and further enhance these aspects of the software.

6.1 Sales Enquiries

We offer both commercial and academic licences for AMBER. Both licences give access to the full functionality of AMBER, with the academic licences restricted to use for teaching and non-commercial academic research. The licences are managed through USB hardware keys, with the option for remote licensing alongside the current local licensing system. In addition, support agreements entitle existing users to help from AMBER experts if they encounter any problems with developing cases.

We also offer training courses in AMBER, which cover aspects of safety assessment and can be tailored to your organisation’s needs.

For all enquiries please contact us via email.

AMBER website: https://www.quintessa.org/software/AMBER

AMBER Support Team:
e-mail: amber@quintessa.org
Telephone: +44 (0)1925 885956
Address: Quintessa Limited
First Floor, West Wing
Videcom House
Newtown Road
Henley-on-Thames
Oxfordshire RG9 1HG
United Kingdom
Website: https://www.quintessa.org/amber
A Knowledge Base

AMBER has been applied to a wide range of studies over the past twenty years. Reports and papers relating to those studies provide a ‘knowledge base’ for AMBER users and can be drawn on to provide inspiration, mathematical models and data for future studies. References for a selection of such studies are included below.

• 2023

• 2022

• 2021

• 2020


• 2019


• 2018


• 2017


- 2015

- 2014


- 2013


- 2012


• 2011


- 2009


- 2008


• 2007


• 2006


• 2005


2004


2003


• 2002


- 2001


- Venter A, Smith G M, Walden Bevan R (2001). Modelling the radiological impact of C 14 and tritium from Cardiff Waste Water Treatment Works; Assessment of exposure via the foodchain. Report to Amersham plc under contract number NYCO6239B.

• 2000


- Venter A, Smith G M, Loose M (2000). Radioactivity in food due to disposals from non licensed sites. Report to the Ministry of Agriculture, Fisheries and Food (MAFF) under project number RP 0441.

• 1999


• 1998


• 1997


• 1996

