

MelMACCS Version 2.0.2 User's Guide

K. McFadden, Sigma Software LLC
N. E. Bixler, Sandia National Laboratories

1 Introduction

This document provides directions on how to use MelMACCS in both interactive and batch modes to create a MACCS source-term file from a MELCOR plot file. The standard method of processing a MelMACCS output in MACCS is to use the WinMACCS interface (Bixler et al., 2015) either using the import file option or the cyclical file set option, which can be used to process multiple MelMACCS output files. Section 6 describes the requirements for creating the MACCS input files directly without using the WinMACCS interface.

Not all MELCOR plot files have the information that MelMACCS needs. The release paths must be defined in the MELCOR input cards to create MACCS-specific data in the plot file. A description of the cards needed for MELCOR is included in this document.

MelMACCS comes with a set of default core inventories that can be used without modification. However, it is preferable that the user define his/her inventories specific to the reactor that is being evaluated. Information on how create an inventory file is provided in the MelMACCS Models Document that is provided with MelMACCS.

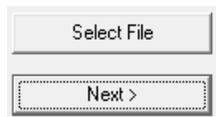
Section 2 of this document describes how to use MelMACCS in interactive mode. Section 3 describes the MelMACCS data files. Section 4 defines the format of a project file. Section 5 describes how to run MelMACCS in batch mode. Section 6 describes how to use a MelMACCS source term file without using the WinMACCS interface. Finally, Section 7 describes the inputs required to run MELCOR so that the plot file is compatible with MelMACCS.

2 MeIMACCS in interactive mode

MeIMACCS is started by selecting the MeIMACCS symbol from the Windows Start Menu. This application leads the user one window at a time through the process of creating a MELCOR/MACCS interface file, also referred to as a MeIMACCS output file or a MACCS source-term file.

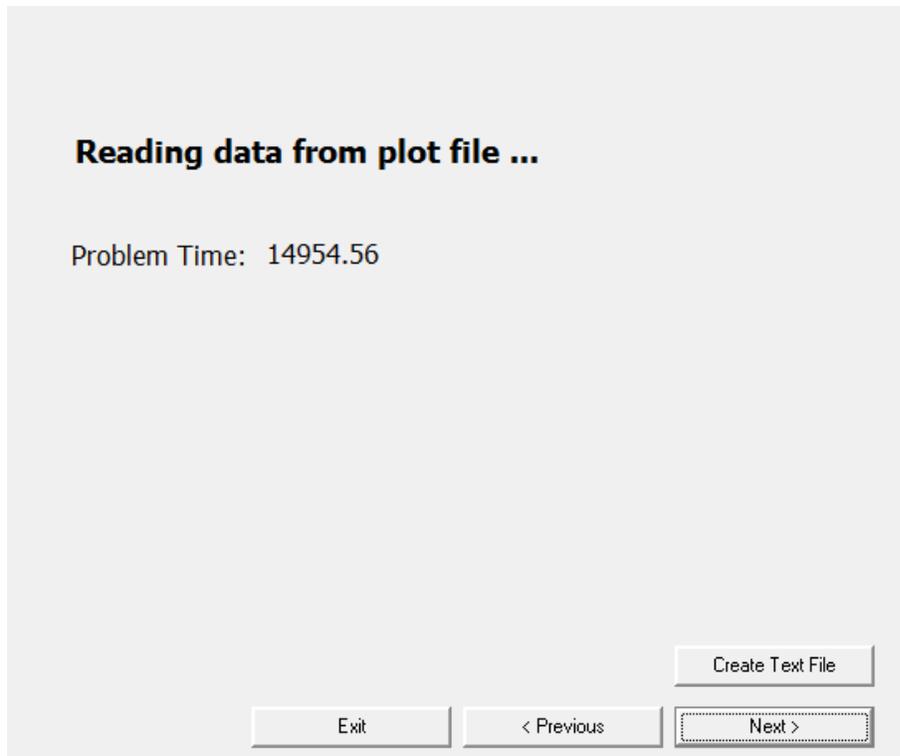
2.1 Selecting the MELCOR plot file

The first form that opens when the user starts MeIMACCS contains the text “Welcome to the MELCOR to MACCS wizard.” The *Select File* button is clicked to choose a MELCOR plot file. The user chooses the MELCOR plot file and clicks *Open*. The MELCOR plot file commonly has a .ptf extension. The user clicks *Next >* to continue.



2.2 Reading the MELCOR plot file

The next form contains the text “Reading data from plot file...” After this form is opened, MeIMACCS starts to read the plot file. When this step has completed, the last time read from the MELCOR plot file is displayed next to the text labeled *Problem Time*. The user clicks the *Next >* button to proceed to the next form. Optionally, the *Create Text File* button can be clicked to create an ASCII file containing a summary of the MELCOR data that are relevant to MACCS.



2.3 Entering the reference time and height

The default value of the MelMACCS variable labeled *Reference time...* corresponds to a MELCOR input variable indicating the reactor scram time. Typically, MELCOR input is set up so that reactor scram occurs at time zero, but this need not be the case. If either this value is missing from the plot file or if the value on the plot file is less than the time of the first data block written to the plot file, the value is set to zero. The primary significance of this parameter is to control the start of radionuclide decay in the MACCS calculation.

This time is also used to adjust the time of plume releases saved in the MelMACCS output file. For example, if the value of the time of accident initiation is input to be 100 s and the user selects a plume segment that starts at MELCOR time 300 s in a *Release Path* form, the starting time for the plume written to the MACCS Input file is $300 - 100 = 200$ s.

The text box labeled *Height of ground level in MELCOR model (m)* is the ground height in the MELCOR reference frame. This value is used to adjust the height of plume segments saved in the MelMACCS output file. For example, if MELCOR

records a release path height of -10 m and this corresponds to a height of 30 m above ground, then this variable should be set to -40. This value is used to calculate the MACCS variable PLHITE by subtracting it from the height for the release path in the MELCOR reference frame.

Reference time for inventory in MELCOR time frame (sec)	<input type="text" value="0"/>
Height of ground level in MELCOR model (m)	<input type="text" value="0"/>

The user clicks *Next >* to continue.

2.4 Entering the release path geometry

The grid shown on this form contains the release path identification number, the release path height, and the adjusted release height based on subtracting the user entered value labeled “*Height of ground level in MELCOR model*” from the release path height read from the MELCOR plot file. All of this information comes directly from the MELCOR plot file except the value used to adjust the release height.

The value of the adjusted release height can be edited by the user. For each plume segment, MACCS requires a value of building height, variable BUILDH (Bixler et al., 2015), to evaluate whether a buoyant plume is entrained in the building wake. Initial plume dimensions, SigmaY and SigmaZ, are also needed by MACCS. These values represent the standard deviation of the Gaussian plume in the cross-wind and vertical dimensions, respectively. The MACCS User’s Guide recommends values as follows:

$$\text{SigmaY} = W/4.3 = 0.23 \times W$$
$$\text{SigmaZ} = H/2.15 = 0.47 \times H$$

Here, W is the building width and H is the building height. These values are not always trivial to select since most buildings are rectangular or have even more complex footprints and have multiple roof heights. In reality, the best choice for the building dimensions often depends on wind direction, but single, representative values are required by MACCS. Fortunately, these values only influence the plume concentrations at relatively short distances and generally have modest influence at distances of several kilometers downwind.

Enter building height and initial plume parameters.
SigmaY and SigmaZ define the initial plume width and height, respectively.

Reset

	Release Path	Release Path Height (m)	Adjusted Release Height (m)	Building Height (m)	SigmaY (m)	SigmaZ (m)
	51	0	5	12	9	6
▶	99	0	5	14	9	6.5

Clicking the button labeled *Reset* removes all the values entered by the user and replaces the Adjusted Release Height with the default values.

If a row in the grid is defined, clicking the button labeled *Auto Fill* copies the row to vacant entries in subsequent rows until a completed row is encountered. The row that is copied is the last defined row.

The user clicks *Next >* to continue.

2.5 Specifying the deposition velocity algorithm

This form allows the user to select the method of calculating MACCS variable VDEPOS. There are two model choices. The first is based on gravitational settling. The default model is based on a correlation of expert elicitation data (Bixler et al., 2013).

Deposition Velocities

Gravitational settling

Expert elicitation/gravitational settling hybrid

Cutoff Aerodynamic Diameter (micrometers)

Deposition Velocity Parameters

Surface Roughness (m)	<input type="text" value="0.1"/>
Wind Speed (m/s)	<input type="text" value="5"/>
Quantile	<input type="text" value="0.5"/>

A higher number for quantile increases the deposition velocity; a lower number decreases it. All values are within expert bounds.

MELCOR aerosol density (kg/m³)

Disable deposition velocity results in MACCS file

Exit

< Previous

Next >

When the expert elicitation option is chosen, additional inputs are required as follows:

- *Cutoff Aerodynamic Diameter*: This is the point at which the deposition velocity calculation is switched from the expert elicitation correlation to gravitational settling. The default and recommended value is 20 μm . An exception to this rule is made when the gravitational settling result for a particle diameter is less than the value calculated using the expert elicitation correlation for the cutoff diameter, in which case the velocity calculated for the cutoff diameter using the expert correlation is used.
- *Surface Roughness*: This is a measure of the terrain roughness. By default this is set to 0.1 m. This value is allowed to range from 0.01 m to 1.0 m. A value of 0.03 m represents grasslands; a value of 0.1 m to 0.8 m represents suburban terrain; a value of 0.2 m to 1.0 m represents forested land.
- *Wind Speed*: The default wind speed is 5 m/s. The range of values are 0.5 to 10 m/s.

- *Quantile*: This is a measure of degree of belief in deposition velocity, as expressed by a group of experts. A value of 0 represents the smallest value that the experts thought to be possible; a value of 0.5 represents the best guess; a value of 1 represents the highest value that the experts thought to be possible. This value is 0.5 by default.

Two of these parameters, wind speed and surface roughness, must be specified as representative values over the grid and for all weather trials. Surface roughness often varies over the grid and wind speed varies from hour to hour and weather trial to weather trial. As a result, the user needs to choose a reasonable, representative value in light of the expected variability at a site.

The *MELCOR Aerosol Density* is read from the MELCOR plot file. If this value is not included on the plot file, the number 1000 kg/m³ is used as a default.

If the check box titled "*Disable deposition velocity results in MACCS file*" is checked, the deposition velocity is calculated, but when the values are written to the MelMACCS output file they are written as comments. Thus, both WinMACCS and MACCS ignore these cards and deposition velocities must be specified separately. This option is useful for uncertainty analyses for which deposition velocity is treated as an uncertain input.

For example, the input image containing the deposition velocity is disabled in the following line because it starts with the * character:

```
*DDVDEPOS001 8.7563E-04
```

If the check box is not selected, then the input image containing the deposition velocity is enabled, as shown in the line below:

```
DDVDEPOS001 8.7563E-04
```

The user clicks *Next >* to continue.

2.6 Entering the mass threshold fractions

This form allows the user to filter out release paths and plume segments that are insignificant. There are two threshold values that can be entered to allow this filtering as follows:

Mass threshold fraction for path to be used**0,01**

Value is the threshold fraction of the mass release of a chemical group for a given release path to the total mass release for that same chemical group summed over all release paths. If any chemical group release in the release path is equal to or exceeds this fraction, then the window for that release path is displayed. Release paths with mass releases that fall below this threshold for every chemical group are not available for further processing.

Mass threshold fraction for plume segment to be used**0.05**

Value is the threshold fraction of the mass release of a chemical group for a given plume segment to the total mass release for that same chemical group summed over all release paths. If any chemical group releases in the plume segment exceeds this fraction, then the segment is saved when Apply is clicked. Segments with mass releases that fall below this threshold for every chemical group are not saved.

When zero is entered for *Mass threshold fraction for path to be used*, every *Release Path* form is active. Otherwise, only forms with a release fraction equal to or exceeding the threshold are active. Inactive forms do not appear on the screen.

When zero is entered for *Mass threshold fraction for plume segment to be used*, then when the *Apply* button is clicked in the *Release Path* form, every plume segment has an entry in the form showing the plume segment parameters. When a positive value is entered, each release fraction for each chemical group of a plume segment is tested against the threshold value. If a plume segment has at least one release fraction for a chemical group that is equal to or greater than the threshold, it is applied to the form showing the plume segment parameters. The user clicks *Next >* to continue.

2.7 Entering the options that affect the core inventory

This form allows the user to specify the set of chemical groups to be included, the core inventory, and the ring (if the MELCOR plot file contains release data for more than one ring).

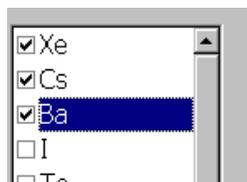
All available chemical groups found on the MELCOR plot file that are not specifically excluded are shown on this form. The user can eliminate chemical groups from this form by editing the MelMACCS.usr file in an ASCII text editor

such as Notepad in the section labeled /EXCLUDE-GROUPS. Groups listed in this section do not appear in the MelMACCS user interface.

Example from the MelMACCS.usr file. In this example, chemical group B is excluded from the MelMACCS interface.

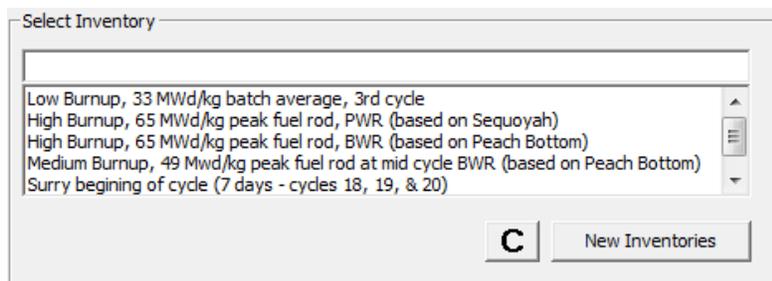
```
/EXCLUDE-GROUPS  
B  
/END
```

Select the chemical groups that are to be included in the MACCS analysis using the check boxes. By default, all chemical groups are included and the user must deselect groups to be excluded.



Next, select the inventory that is closest to the inventory modeled by MELCOR. A description field is shown in the user interface to describe each inventory. Ideally, the same ORIGEN calculation is used to define the MELCOR inputs and to create an inventory for MelMACCS. This is done by creating one of more inventory files, as discussed in Section 6 and also in the MelMACCS Models Manual.

The inventory list shows the core inventories read from the MelMACCS.inv file found in the installation folder and additional inventory files created by the user. User-defined inventory files must be located in the MelMACCS_Docs\Inventories folder in the Windows user area. Under Windows 7, the inventory folder is C:\Users\username\MelMACCS_Docs\Inventories. All inventory types defined in these files are displayed in the user interface and can be selected, as shown below:



New inventory files can be added by clicking the button labeled *New Inventories*. When a new inventory file is added, a copy of that file is placed in the MeIMACCS_Docs\Inventories folder. The next time MeIMACCS starts, the new inventory file is automatically loaded.

Inventories can be prevented from loading into MeIMACCS by removing the inventory file from the MeIMACCS_Docs\Inventories folder.

All inventories can be removed from the current MeIMACCS run by selecting the button labeled *C*. This does not change the files that will be loaded the next time MeIMACCS is run.

MELCOR is commonly run with multiple rings, but in most cases a single release fraction is recorded in the plot file for all of the rings. However, MELCOR can be run in a mode that records release fractions on the plot file for each ring. This is especially useful for spent fuel pools, where different rings can represent fuel of different ages. MeIMACCS processes multi-ring release fractions when they are present in the MELCOR plot file. When this is done, each ring has a set of initial masses and released masses on the plot file. The same set of chemical groups is common for all rings. Because the rings may represent fuel of different ages, each ring may be associated with a different inventory. When the MELCOR plot file only contains a single ring, the selection has already been made and the user does not need to do anything further.

The user may select the ring to consider in any order.

Ring Processed	Ring Name
<input checked="" type="checkbox"/>	RING_1
<input type="checkbox"/>	RING_2
<input type="checkbox"/>	RING_3

After the plots associated with a ring have been displayed in the *Release Path* windows, the *Ring Processed* check box is checked the next time this window is displayed. The ring can be reevaluated. This check box is only a visual guide to help the user keep track of which rings have already been evaluated.

The default value of the inventory scaling factor is 1.0, which is usually the desired value, but this value can be changed and defines the value of the MACCS parameter, CORSCA. This parameter is useful when the MELCOR calculation has been done for a reactor that is similar, but different in size and power rating, than the reactor that is to be evaluated with MACCS.

Inventory Scaling Factor

1

The user clicks *Next >* to continue.

2.8 The Master Form

A form with the text *Select plume segments and click "Apply..."*, referred to as the *master form*, is opened along with supporting *Release Path* forms, where each window corresponds to a MELCOR release path written to the plot file. Each release path is displayed provided that the mass release threshold requirements are met. After defining plume segments from the release path windows, as described in this section, the MelMACCS output file is created.

Plume segments can be selected from the *Release Path* windows, and applied to the *master form* in the manner described in section 2.9. When these segments are applied in the *master form* from the *Release Path* windows, the data are summarized in the table titled *Plume Segment Parameters*.

Segment	Release	Start(s)	Duration (s)	Release Height(m)	Adjusted Height (m)	Heat (J/s)	Flow Rate (kg/s)	Gas Density
1	51	3677.367	1499.109	0	5	3.180091E+07	37.36899	0.3394516
2	51	5176.476	1500	0	5	7610.368	4.156428E-02	0.6580326
3	51	6676.476	1507.287	0	5	2910296	7.658202	0.4931791
4	51	8183.763	1500	0	5	289641.9	1.301917	0.5875116
5	51	9683.763	1490.797	0	5	8273083	9.898834	0.3991854
6	51	11174.56	1490	0	5	70577.66	0.3195424	0.5935096
7	51	12664.56	1530	0	5	419500	2.105535	0.6048939
8	51	14194.56	760	0	5	531822.5	2.779543	0.6037541

Select MACCS model to specify the plume segment parameters.

Heat
 Density
 Do not specify model

Indicate which plume segment is risk dominant. This is often based on its potential for causing early fatalities.

Segment 1

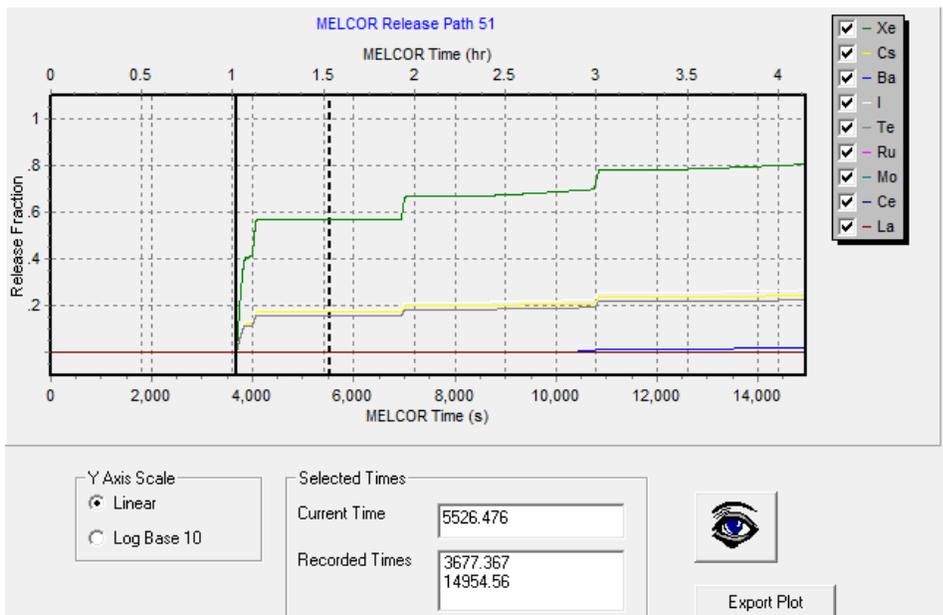
Next Ring Create MACCS File
< Previous Exit

2.9 Navigating a release path form

The following options are available on a release path form:

- A dashed, vertical line tracks the mouse movement as the mouse moves across a plot. The time corresponding to the location of the mouse is displayed adjacent to the label *Current Time*. Clicking on the plot records a time in the *Recorded Times* window. A series of n recorded times, t_1, t_2, \dots, t_n corresponds to plume segments $[t_1, t_2], [t_2, t_3], \dots, [t_{n-1}, t_n]$.

- A time can also be recorded by typing a value in the text box labeled *Current Time* and pressing the *Enter* key on the keyboard. Clipboard functionality is also supported in this text box.
- To select an entry in the *Recorded Times* list, the user can click on the entry in the list. This value is copied to the clipboard using Ctrl-C. This function allows easy duplication of recorded times between *Release Path* forms.



In the above figure, Times 3677.367 and 14954.56 are recorded. The mouse pointer is positioned over 5526.476, but the user has not recorded this time by clicking the left mouse button.

- A single selected time can be deleted by clicking the time in the list labeled *Recorded Times* and pressing the *Delete* key on the keyboard.

All user recorded times are deleted if the user clicks the button *Clear*. The time of the first and last mass released is retained.

Selected Times

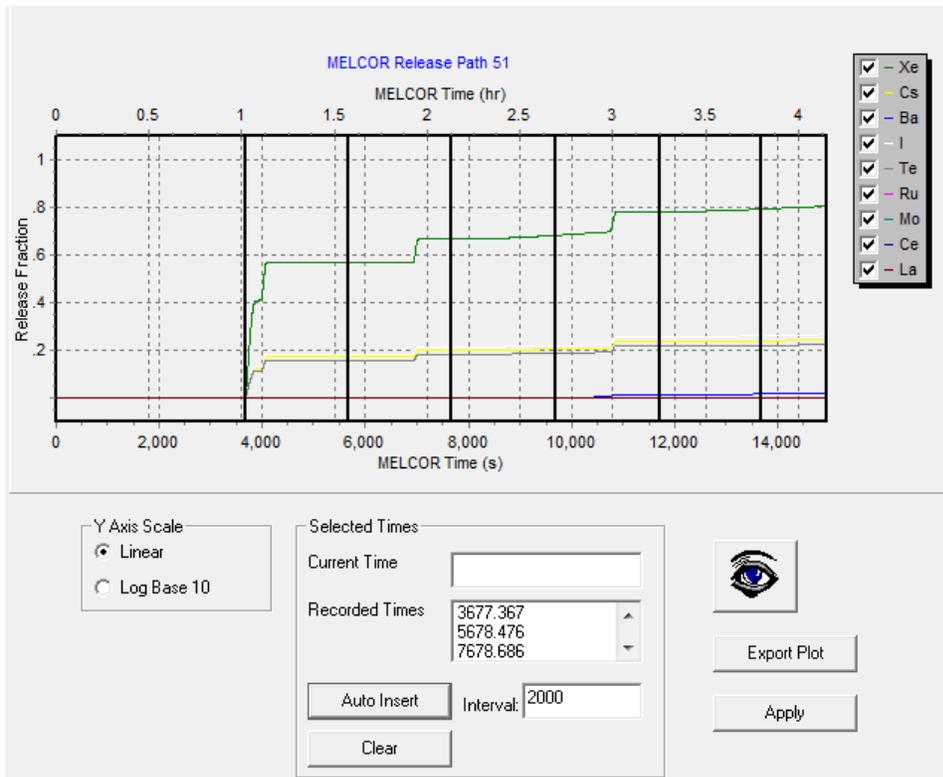
Current Time

Recorded Times

Interval:

Time 7678.686 is selected. Remove by pressing the Delete key.

- A sequence of times is inserted by typing in a time interval in the *Interval* box and clicking the *Auto Insert* button. To use this feature, at least two times must be recorded. Insertion is between the first and last recorded times at the interval indicated. The last interval is usually shorter than the selected time interval.



In the illustration, auto Insert is used to insert lines at intervals of 2000 s.

- Toggle the zoom mode by clicking on the zoom button. This prevents recorded times from being selected.

If zoom mode is on, the graph can be magnified by drawing a box from top left to bottom right by a drag and release motion using the left mouse button. To return to the original graph magnification, repeat this process, but move the mouse from bottom right to top left.

Times can be recorded while the graph is magnified provided the zoom mode is off.

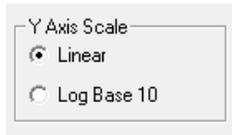


Zoom Off

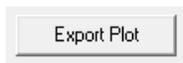


Zoom On

- Y-axis scale can be changed from linear to log by choosing the appropriate radio button in the group labeled *Y Axis Scale*.



- The plot can be exported in various formats by clicking *Export Plot* button. The formats include graphics formats, such as Windows bmp (bitmap) and jpeg, and text formats such as ASCII, HTML, and XML. All MELCOR data used in MELMACCS calculations can also be exported as text by clicking the *Create Text File* button on the form that contains the text “Reading data from plot file...” .. The *Export Plot* option only exports the data shown in the plot window.



- The *Apply* button can be clicked to apply the recorded times to the *master form*. When this button is clicked, previously applied plume segments associated with the current release path are replaced.



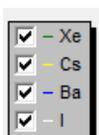
The time preceding the first nonzero flow greater than or equal to the reference time and the time corresponding to the last nonzero flow are automatically recorded for the user, although these can be manually deleted. These times

mark the logical beginning and end of a set of plume segments for this release path.

The time corresponding to the first nonzero flow is recorded on the bottom left of the graphics window adjacent to the label *Initial release time (sec)* and similarly for the *Final release time (sec)*.

Initial release time (sec): 3677.367

The plot legend shows the chemical groups chosen in the form “*Choose options that affect the core inventory*”. The legend entries can be unchecked in the legend, which causes the corresponding release fraction curve to become invisible in the plot. This option is used to help identify the data series in the plot. This option does not modify the chemical group information saved in the MelMACCS output file. If one or more chemical groups are to be removed from consideration in the MelMACCS output file, the user must navigate back to form titled “*Choose options that affect the core inventory*” by clicking the <Previous button on the *master form*.



2.10 Creating the MelMACCS Output File

For the user to create a MelMACCS output file:

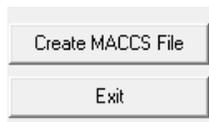
- The user records times in the *Recorded Times* window of the *MELCOR Release Path* forms as described in section 2.9. These times define the beginning and end of the plume segments associated with this release path.
- The user clicks the *Apply* button in a *Release Path* form to apply the times record in the plot to the *master form*.
- The user selects the model for the MACCS plume buoyancy calculation. All of the parameters, namely heat, flow and density, for each plume segment, are calculated by MelMACCS and are written to the output file. When this file is read into WinMACCS, the parameters imported are dependent on the model setting defined here. If Heat is selected, the heat values are imported into WinMACCS. If Density is selected, the flow rate and gas density values are imported into WinMACCS. If Do not specify model is selected, the parameters used by WinMACCS are dependent on the buoyancy model defined on the Plume/Source tab on the WinMACCS Project Properties form. Selecting Do not specify model offers the most flexibility.

Formatted: Font: Italic

- The user then indicates in the pull down menu with the title “*Indicate which plume segment is risk dominant...*” the risk dominant plume segment. This can also be chosen by selecting the row containing that segment.
- After all release paths have been considered, the user clicks the *Create MACCS File* button to create the MELMACCS output file. Navigate to a folder and type a name for the output file.
- To start over again, the user can click the *Reset* button in the *master form* to clear all plume segments from the grid.
- Clicking the *Next Ring* button returns to the form containing options that define the core inventory.

If the header area on any given column heading of the table is clicked on the *master form*, the data in the table are reordered. The order is toggled between ascending and descending order each time the heading is clicked. However, the values in the MelMACCS output file is always created in ascending order of plume release time (the column labeled *Start (s)* in the table).

Clicking the *Exit* button closes MelMACCS.



3 MeIMACCS data file format

A project file is defined in an easily editable format. A general format is used for all data files read by MeIMaccs. This includes the user defined inventory files (.inv), the user project files (.mel), the user settings file, MeIMACCS.usr. With the exception of the MELCOR plot file, the format of data files read by MeIMACCS all follow the same language rules. This format supports editing in an ASCII text editor, such as Microsoft Notepad. Microsoft Word and Microsoft Word Pad can add extra invisible formatting characters that are not compatible with MeIMACCS, so these editors should not be used to modify these files. Additionally, new lines in the MeIMACCS data files are indicated by the Windows standard carriage return (ASCII 13) and line feed (ASCII 10) characters. Because UNIX files have only the line feed character, files created on a UNIX box are not compatible with MeIMACCS.

The following components are found in a MeIMACCS data file:

- A comment is indicated by an asterisk, "*", in column one. Comments can occur anywhere in the file.
- Comments following data in a data line are not supported. Place all comments on their own line beginning with an asterisk in column one.
- A keyword starts with a slash, "/", in column one, followed by a keyword followed by value(s) associated with that keyword. The number of values required depends on the keyword.
- Values are separated on data or keyword lines using spaces or tabs.
- A data block begins with a keyword line. This is followed by a series of data lines. The end of the data block is indicated by the following character sequence starting in column one, "/END".
- Some keyword lines are not associated with a data block.
- Blank lines are permitted in the file. Blank lines can occur anywhere.
- If a value is a string containing spaces, it must be enclosed by the double quote characters (" ").
- White space can be any number of spaces, tabs or otherwise unreadable characters. More precisely, characters with an ASCII code less than or equal to 32 are considered white space.

The following is an example of a keyword taken from a MeIMACCS.usr file. The keyword is "DEFAULT-DIR". The value is enclosed in double quotes because it contains a space.

```
/DEFAULT-DIR "C:\MeIMACCS Results"
```

The following is an example of a data block from the file MeIMACCS.ini. The keyword is "COMPOUND-GROUPS". The data block contains four lines of data.

```
/COMPOUND-GROUPS  
CSI I 0.488444  
CSI Cs 0.511556  
CSM Cs 0.73478922  
CSM Mo 0.26521078  
/END
```

4 Format of a project file

A project file is used to define settings of a MelMACCS project. If a keyword or data block is not defined in a project file, a default value is used if the keyword or data block is optional. However, some keywords and data blocks are required.

The keywords in a MelMACCS project file are not case sensitive.

4.1 Chemical groups

List of chemical groups to be considered, matching the entries in the MELCOR plot file.

This is an optional data block.

Keyword	Default	Example
Chemical_Groups	All available chemical groups are selected	/Chemical_Groups Xe Cs Ba /End

4.2 Ring number

Ring number associated with the MELCOR plot file. If there is only one ring or the ring referenced is out of range, the value is set to 1.

The value following the keyword is as follows:

- Ring number, integer

This is an optional variable.

Keyword	Default	Example
Ring	1	/Ring 2

4.3 Inventory scale factor

The inventory scale factor defines the value to write to output file for MACCS variable CORSCA.

The value following the keyword is as follows:

- Scale factor, positive real

This is an optional variable.

Keyword	Default	Example
Inventory_Scale	1.0	/Inventory_Scale 1.2

4.4 Reactor type

Value indicates which inventory to use when calculating the core inventory. The value of the Reactor_Type must be an inventory defined in the file MeIMACCS.inv or in an inventory file found in the MeIMACCS_Docs\Inventories folder in the user area that are loaded into the user interface when MeIMACCS starts. Under Windows 7, the inventory folder is C:\Users\username\MeIMACCS_Docs\Inventories.

The value following the keyword is as follows:

- Inventory identifier, inventories provided in the MeIMACCS.inv file or another user loaded inventory file. LOW, HIGH_PWR, HIGH_BWR and MEDIUM_BWR are provided in MeIMACCS.inv.

This is a required variable.

Keyword	Default	Example
Reactor_Type	None.	/Reactor_Type HIGH_PWR

4.5 Reference time

This is used to adjust the plume release time, PDELAY. The reference time is subtracted from the start time of each plume segment to calculate the time written to the MeIMACCS output file.

The value following the keyword is as follows:

- Time (sec), real, not to exceed the last time on the plot file

This is an optional variable.

Keyword	Default	Example
ATime	0	/ATime 100.0

4.6 Ground height relative to MELCOR

This is used to adjust the MELCOR height associated with each release path considered. The value is used to define the MELCOR height associated with ground level, i.e., the height of the ground (grade) in the MELCOR reference frame. See section 2.3 for more information about how this variable is used.

The value following the keyword is as follows:

- Height (meters), real, range [-1000.,1000.]

This is an optional variable.

Keyword	Default	Example
Ground_Height	0.	/Ground_Height 5.0

4.7 Building height, sigma y and sigma z

This data block has an associated data line for every path defined on the MELCOR plot file. See section 2.4 for more information about how these variables are used.

The lines in the data block contain the following:

- MELCOR release path, integer
- Building Height (meters), real, range [1.,1000.]
- Initial SigmaY (meters), real, range [0.1,1000]
- Initial SigmaZ (meters), real, range [0.1,1000]

This is a required data block. There must be an entry for every path on the MELCOR plot file.

Keyword	Default	Example
Path	None	/ Path 51 10.0 0.2 0.3 99 4.0 0.5 0.6 101 7.0 0.8 0.9 /End

4.8 Adjusted release height

This keyword allows the adjusted release height to be set for each release path. The automatic adjustment for all paths using the value entered for /Ground_Height is ignored for paths listed in this data block.

The lines in the data block contain the following:

- MELCOR release path, integer
- Release height (meters), real, suggested range [0.,1000.]

Not all paths are required to be listed.

This is an optional data block.

Keyword	Default	Example
Adjusted_Release_Height	Height of Release Path minus Ground_Height	/Adjusted_Release_Height 51 0.0 99 10.0 /End

4.9 Deposition velocity

This indicates the method used to estimate deposition velocity. See section 2.5 for more information about how this variable is used.

Possible values are EXPERT and GRAVITY.

This is an optional variable.

Keyword	Default	Example
Deposition_Velocity	EXPERT	/Deposition_Velocity EXPERT /Deposition_Velocity GRAVITY

4.10 Aerodynamic cutoff diameter

The aerodynamic particle size above which the gravitational settling method is used to calculate the deposition velocity is defined on this line. This variable is used only if the deposition velocity method is set to EXPERT. See section 2.5 for more information about how this variable is used.

The value following the keyword is as follows:

- Cutoff diameter (μ meters), real, positive number, range (0,25]

This is an optional variable.

Keyword	Default	Example
Cutoff_Diameter	20 μ meters	/Cutoff_Diameter 25.0

4.11 Surface roughness

A representative value of surface roughness is entered on this line. This variable is used only if the deposition velocity method is set to EXPERT. See section 2.5 for more information about how this variable is used.

The value following the keyword is as follows:

- Surface roughness (meters), real, range [0.01,1.]

This is an optional variable.

Keyword	Default	Example
Surface	0.1 (m)	/Surface 0.2

4.12 Wind speed

The wind speed is entered on this line. This variable is used only if the deposition velocity method is set to EXPERT. See section 2.5 for more information about how this variable is used.

The value following the keyword is as follows:

- Windspeed (meters/second), real, range [0.5,10.]

This is an optional variable.

Keyword	Default	Example
Wind_Speed	5.0 (m/s)	/Wind_Speed 4.0

4.13 Quantile

The quantile level determines the expert coefficients to use. This variable is used only if the deposition velocity method is set to EXPERT. See section 2.5 for more information about how this variable is used.

The value following the keyword is as follows:

- Quantile, real, range [0.,1.]

This is an optional variable.

Keyword	Default	Example
Quantile	.5	/Quantile 0.76

4.14 Output format of deposition velocity

This value determines whether the deposition velocity values are written to the MeIMACCS output file as comments. The value is set to True when the

WinMACCS user wants to override the deposition velocities with other values entered in WinMACCS.

This is an optional variable.

Keyword	Default	Example
Disable_Deposition_Velocity	False	/Disable_Deposition_Velocity True

4.15 Release path threshold

This variable is used to eliminate paths that have an insignificant release. See section 2.6 for more information about how this variable is used.

The value following the keyword is as follows:

- Release path threshold, real, range [0.,1.]

This is an optional variable.

Keyword	Default	Example
ReleasePathThreshold	0	/ ReleasePathThreshold 0.001

4.16 Plume release threshold

This variable is used to eliminate plume segments that have an insignificant release. See section 2.6 for more information about how this variable is used.

The value following the keyword is as follows:

- Plume release threshold, real, range [0.,1.]

This is an optional variable.

Keyword	Default	Example
ReleaseThreshold	0	/ ReleaseThreshold 0.0001

4.17 Default path boundary

The time boundaries for each path can be determined based on the first and last mass released, or can be based on specific user entered times. When running MelMACCS in interface mode, the boundaries are initially determined by the first and last mass released.

This keyword allows the user to override the boundary values with constants. These boundaries are used for every path. See section 4.18 to override the boundary determination for specific paths.

Possible values associated with the boundary assignment are as follows:

- AutoBoth: MelMACCS determines the upper and lower bound based on the first and last mass released. No further parameters are needed.
- AutoBeginOnly: MelMACCS determines the lower bound. The upper bound needs to be supplied.
- AutoEndOnly: MelMACCS determines the upper bound. The lower bound needs to be supplied.
- AutoNone: Both the lower and the upper bounds need to be supplied.

This is an optional variable.

Keyword	Default	Examples
Default_Plume_Boundaries	AutoBoth	Default_Plume_Boundaries AutoBoth /Default_Plume_Boundaries AutoBeginOnly 12000. /Default_Plume_Boundaries AutoEndOnly 4000. /Default_Plume_Boundaries AutoNone 4000. 12000.

4.18 Path boundaries

The determination of the plume boundaries may be specified per path. This takes precedence over the default method for determining the boundaries. See section 4.17 for a description of the possible default methods.

The first value on a line is the MELCOR release path. The values following the release path are identical to the values following the Default_Plume_Boundary keyword described in section 4.17.

This is an optional data block. If path boundaries are not defined for a path the default boundaries are used.

Keyword	Default	Examples
Plume_Boundaries	Value of Default_Plume_Boundaries	/Plume_Boundaries 51 AutoNone 4000. 13000. 99 AutoBeginOnly 13000. /End /Plume_Boundaries 99 AutoEndOnly 4000. /End

4.19 Default plume segment interval

A time interval can be specified to define plume segments for all release paths. The insertion of this time interval occurs between the path boundaries for every release path that has not been eliminated because of mass threshold limits as defined in section 4.15. This operation is similar to the Auto Insert function in the user interface; however, it is applied to all release paths.

See section 4.20 to specify the release interval by path.

This line causes plume segments to be defined for each path at the interval specified. If the user wants to define plume segments for a limited number of paths, then this line should not be used. However, the time interval for a specific release path overrides this time interval, as described in the following subsection.

The value following the keyword is as follows:

- Time interval for insertion in seconds, real

If this line is not present, then there is no default interval insertion. In this case, all insertions are specified by path as described in section 4.20 or by specifying each release point manually in section 4.21.

This is an optional variable.

Keyword	Default	Example
DefaultAutoInsert	none	/DefaultAutoInsert 1500.0

4.20 Plume segment interval

A time interval can be specified to determine plume segments. The insertion of this time interval occurs between the path boundaries for the path specified. This operation is similar to the Auto Insert function in the user interface.

This line causes the default insertion interval for the specified path defined using the /DefaultAutoInsert line to be replaced.

The values following the keyword are as follows:

- MELCOR release path, integer
- Time interval for insertion in seconds, real

Not all paths are required to be listed.

This is an optional variable. There can be multiple entries of this line.

Keyword	Default	Examples
AutoInsert	Time interval defined on the DefaultAutoInsert line is applied to all paths	/AutoInsert 51 1000.0 /AutoInsert 99 2000.0

4.21 Manual plume segment definition

Time points can also be specified for individual release paths. This can be used in place of the /AutoInsert or /DefaultAutoInsert lines, but if auto insertion is defined for a path and specific release times using the /ManualInsert data block are defined for that same path, then all times are applied.

This data block can be repeated for multiple release paths.

The value following the keyword is as follows:

- MELCOR release path, integer

The lines in the data block contain the following:

- Time (seconds), real

This is an optional variable.

Keyword	Default	Examples
ManuallInsert	none	/ManuallInsert 99 1515. 4000. 10000. 25000. /End

4.22 Plume segment of maximum risk

This line identifies which plume segment is considered to have the maximum risk in the MACCS calculation. This value defines the MACCS parameter MAXRIS. For the MACCS calculation the plume segments are ordered by start time of release, independent of the release path. MaxRisk identifies the plume segment ordered by start of the release.

The values following the keyword are as follows:

- Plume release number identifying interval, integer

This is an optional variable.

Keyword	Default	Examples
MaxRisk	1	/MaxRisk 14

Formatted: Heading 2

4.23 Select model to define plume buoyancy

This line identifies which of the plume buoyancy parameters to consider in the MACCS calculation. All of the parameters are calculated by MeIMACCS and are written to the output file. However, when this file is imported in WinMACCS, the parameters read into the WinMACCS interface are dependent on the model setting defined on this line. If *Heat* is selected, the heat values are imported into WinMACCS. If *Density* is selected, the flow rate and gas density values are imported into WinMACCS.

The values following the keyword are as follows:

- Heat
- Density
- None

Formatted: Normal, Bulleted + Level: 1 + Aligned at: 0.25" + Indent at: 0.5"

Formatted: Normal, Indent: Left: 0.5"

Formatted: Normal

This is an optional variable. If this line is absent or if the value is set to None, which parameters read into the WinMACCS interface is dependent on the buoyancy model defined on the Plume/Source tab on the WinMACCS Project Properties form.

Formatted: Font: Italic

Formatted: Font: Italic

Keyword	Default	Examples
<u>Buoyancy</u>	<u>None</u>	<u>/Buoyancy Heat</u> <u>/Buoyancy Density</u> <u>/Buoyancy None</u>

5 Running MeIMACCS in batch mode

MeIMACCS can be run from a command prompt window (cmd.exe) or from a batch file (.bat file). The parameter values are read from a MeIMACCS project file (.mel extension). A sample project file, sampleProject.mel, based on the sample plot file demo.ptf, is provided with MeIMACCS. The format of this file is described in section 4.

The format of a command is as follows:

```
MeIMACCS.exe projectfile [-i melcorplotfile] [-o maccsfile][[-r]]
```

The order of the optional parameters as indicated by the brackets is not important.

The following is an example of a line to run MeIMACCS in batch mode:

```
MeIMACCS.exe sampleProject.mel -i "D:\Demo.ptf" -o MACCSInterface.inp -r
```

There are three files specified. If full paths are not specified, then it is assumed that the files are in the users default directory. File paths and names should be delimited by quotes when there are embedded spaces in the file names.

If any of the optional arguments are specified, then the project file, in this example sampleProject.mel, must be specified.

The meanings of the arguments are as follows:

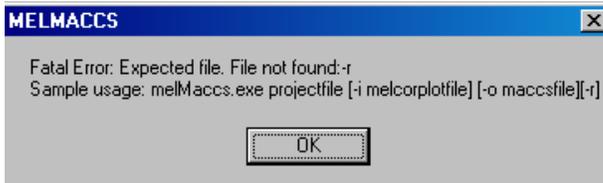
- MeIMACCS project file: This must be the first argument. This file has a special format containing all information that would normally be entered in the user interface via mouse clicks and keyboarding. All values specified in the project file are loaded into the interface.

The interface opens with the preloaded values if the -r option is not used. In this case, the values specified in the project file are viewed in the user interface.

If the -r option is used and an error occurs, the message is appended to the log file, AutoRun.log, located in the same folder as the MeIMACCS project file.

A MeIMACCS project file can be debugged by omitting the -r option, which allows the user to review the settings.

- MELCOR plot file: The plot file is specified with the optional entry, “-i melcorplotfile”. The plot file normally ends with a .ptf extension. This must be defined when the -r option is used. If missing, a visual dialog box displays an error message as follows:



- MelMACCS output file: The output file is specified in the optional entry, “-o maccsname”. This entry defines the MACCS interface file that MelMACCS creates. If not specified, then the file is called MACCS.inp and is placed in the users default folder.
- Run in background: Using the “-r” option indicates that MelMACCS is to be run without any user input.

When the -r option is not used, a dialog box pops up explaining any errors, such as data validation errors. This allows the user to adjust values in the interface. If the -r option is used, any error other than an informative message in MelMACCS causes the batch mode to abort. Messages are appended to the AutoRun.log file found in the same folder as the MelMACCS project file. In this case, the user should open this file to view the error message.

When using the -r option, any plume segment deemed illegal is not applied. This does not cause an error, but a message is written to the log file. Illegal plume segments are those with time durations greater than 86400 s, segments or paths not meeting the threshold criteria, and segments with zero flow or zero gas density. The user can force these segments to be written to the MelMACCS output file only when running in user interface mode. Additionally, plume segments are not created when there is no mass released.

6 Creating a complete MACCS input file

The MelMACCS output file created contains MACCS input cards that define the source term. This is not a complete MACCS ATMOS input file. To create a complete input file, the source-term file created by MelMACCS must be merged with another MACCS ATMOS input file.

One technique that can be used to incorporate the source-term information into an existing MACCS input file is to copy the contents of the MelMACCS output file and paste it at the end of an existing ATMOS input file, provided the file does not include multiple source terms. This works because MACCS uses the last data values encountered in the input file. The user needs to be cautious if there are fewer values for a given MACCS variable in the MELMACCS interface file than are in the existing MACCS input deck. For example, if the number of plume segments, chemical groups, radionuclides, pseudostable radionuclides, or particle size groups is less than the original set in the MACCS ATMOS file, errors may be generated or the calculation may not produce correct results.

To merge a MelMACCS output file with an existing MACCS ATMOS input file

- Open the MelMACCS output file in Notepad or another ASCII text editor. Select the contents of this file and copy to the windows clipboard (using the Copy function from the Edit menu).
- Open an existing MACCS ATMOS input file in a text editor. Paste the contents of the clipboard into an ATMOS file immediately preceding the first line consisting of a "." in column one. The amended file is a valid ATMOS file that can be run using MACCS.

A new input variable, GRPNAM, is included in the MACCS cards, one vector entry for each chemical group. This is required by WinMaccs, but at this time MACCS does not recognize this card and ignores it. If the output from MELMACCS is imported into WinMaccs, these cards should not be removed.

7 Creating a MELCOR plot file with MACCS information

There are two major versions of MELCOR available, 1.8.6 and 2.1. Input for 2.1 differs from 1.8.6. Directions are included in the MELCOR user's guide (Gauntt, R.O 2000). The MELCOR 2.1 users guide is currently under review and not available. The following excerpts have been copied and edited for this document.

7.1 Directions for MELCOR version 1.8.6:

When using 1.8.6 RG, created on November 20, 2002, or later, the following input cards should be included in the MELGEN input.

MACCSnn - MACCS Release Path Definition

1 <= nn <= 99 is the release path assigned by the MELCOR user.

This input card contains one parameter as follows:

(1) MCCSFP - MCCSFP is the number of the flow path; the sign is the sign of flow corresponding to "release".

(type = integer, default = none, units = none)

This record allows identification of flow paths that serve as "release" paths for the consequences code, MACCS. For these paths, and these paths only, fluid and radionuclide transport data are written to the plot file.

Example: This example is consistent with DEMO(rhonom=2500, the sample MELCOR input file released with MelMACCS.

```
MACCS51 399
MACCS99 398
```

7.2 Directions for MELCOR version 2.1:

The MELCOR version must be 2.1.1461 or later. Previous versions of 2.1 MELCOR create plot files that are not compatible with MelMACCS.

FL_MACCS – MACCS Release Paths Definition

This record allows identification of flow paths that serve as release paths for the consequences code MACCS. For these paths, and these paths only, data about fluid and radionuclide transport through the path are written to the plot file. The record can also be used simply to force such data to be written to the plot file for

other applications. However, all MACCS flow paths in a MELCOR plot file are evaluated by MelMACCS, so only the ones relevant to MACCS calculations should be defined if the plot file is to be processed with MelMACCS.

MACCSN - The number of MACCS release paths.
(type = integer, default = 0, units = dimensionless)

The following data are input as a table with length MACCSN. Variables are input in the order listed below.

Variable	Description
NFL	Table row index. (type = integer, default = none, units = none)
MACCSNAME	MACCS release path name (type = character*16, default = none, units = none)
MACCSNUMBER	MACCS release path (type = integer, default = none, units = none)
FPNAME	Flow path name (type = character*16, default = none, units = none)
DIRFL	Direction of flow corresponding to release. FROM TO (type = character*4, default = none, units = none)

MELCOR allows the flow path name to be any character string. However, MelMACCS requires this to be converted to an integer. This means that it is necessary to use an integer in the field FPNAME.

The field MACCSNAME is not currently used by MelMACCS.

When comparing the input format, a positive flow release path in 1.8.6 input corresponds to the keyword FROM, a negative number corresponds to the keyword TO.

Example corresponding to the same problem as referenced above in describing the sample input for 1.8.6:

FL_MACCS 2

```
1 'Release 51' 51 '399' FROM
2 'Release 99' 99 '398' FROM
```

7.3 Adding New Chemical Groups:

All chemical groups written to the MELCOR plot file are considered for processing. However, for a new chemical group to be accepted by MeIMACCS, the MeIMACCS.ini file needs to be edited. Each chemical group is associated with a set of elements. The association corresponding to the MeIMACCS.ini file is in a section labeled /CHEM-TO-ISO. This is necessary to calculate the core inventory by isotope.

Chemical groups are removed from consideration by MeIMACCS by modifying the section titled /EXCLUDE-GROUPS in the MeIMACCS.usr file found in the Windows user's folder. Under Windows 7, this folder is C:\Users\username\MeIMACCS_Docs, where username is the user's login name.

New compound groups such as Csl can be added by editing the MeIMACCS.ini file. See the MeIMACCS Models Document for more information.

References:

Bixler, N.E., E. Clauss, C.W. Morrow, J.A. Mitchell, C. Navarro, and J. Barr, 2013, "Synthesis of Distributions Representing Important Non-Site-Specific Parameters in Off-Site Consequence Analysis," NUREG/CR-7161, SAND2010-3380P, US Nuclear Regulatory Commission, Washington, DC.

Chanin, D., M. L. Young, J. Randall, and K. Jamali, 1998, "Code Manual for MACCS2: Volume 1, User's Guide," NUREG/CR-6613, SAND97-0594, US Nuclear Regulatory Commission, Washington, DC.

McFadden, K. and N.E. Bixler, 2015, "MelMACCS Models Document (MELCOR to MACCS Interface Description)," Letter Report to the NRC, Sandia National Laboratories, Albuquerque, NM.

Bixler, N.E., F. Walton, R. Haaker, K. McFadden and J. Barr, 2015, "MELCOR Accident Consequence Code System (MACCS) User's Guide and Reference Manual" to be published as a NUREG/CR report, US Nuclear Regulatory Commission, Washington, DC.

Gauntt, R.O., R.K. Cole, C.M. Erickson, R.G. Gido, R.D. Gasser, S.B. Rodriguez, M.F. Young, 2000, MELCOR Computer Code Manuals, Primer and User's Guide, Version 1.8.5, NUREG/CR-6119, Vol. 1, Rev. 2, SAND2000-2417/1, US Nuclear Regulatory Commission, Washington, DC.