

# USER'S GUIDE

FOR THE

## UMTRI AIRBAG SKIN BURN MODEL

RELEASE 9809

Steven J. Reed  
Matthew P. Reed

Biosciences Division  
University of Michigan  
Transportation Research Institute  
2901 Baxter Rd.  
Ann Arbor, MI 48109-2150

September 1998

## 1.0 INTRODUCTION

This manual describes the implementation and use of the UMTRI Airbag Skin Burn Model as an ANSI C computer program. This model is an analytical formulation and solution of equations that simulate the exposure of bare skin to high-temperature airbag exhaust gas and evaluate the potential of that exposure to produce burn injury. The development of the original UMTRI Airbag Skin Burn Model was reported by Reed and Schneider (1994) and Reed, Schneider, and Burney (1994). A more portable and flexible version of the model, written in FORTRAN, was reported by Bowman and M. Reed (1996). In the 1994 model, the solution to the heat conduction problem used both closed-form equations and a Fourier series approximation. In the 1996 model, a completely closed-form solution was used. Both the 1994 and 1996 studies were applicable only to systems with convective boundary conditions.

In the current model release, the method of solution has been revised and the model has been completely rewritten in C. The new version is based on a generic, one-dimensional finite-difference solution to the heat conduction problem. This generic model core allows the program to simulate a variety of different time-varying surface boundary conditions, and provides the flexibility to handle a number of different simulation tasks.

One important change from the previous versions is that the model no longer includes an airbag inflation simulation module. The potential users of the model generally have much more sophisticated airbag inflation models available, so the current release of the UMTRI Airbag Skin Burn Model is focused exclusively on the skin. The model is configured to be used with output data from other airbag simulations.

The following sections of this User's Guide give an overview of the model and provide instructions for its use. For technical support or more information concerning the model, see the Contacts section at the end of the Guide.

## 2.0 MODEL OVERVIEW

### 2.1 Method

The model is based on a one-dimensional, finite-difference conduction model. Figure 1 shows the model schematically. The skin is modeled as a series of nodes, each having thickness  $\Delta X$ , except for the surface node, which has thickness  $\Delta X/2$ . The nodes represent the locations in the skin at which the temperature is calculated during the simulation. Heat conduction is represented by the Fourier heat equation:

$$\frac{\partial^2 T}{\partial x^2} = \frac{1}{\alpha_s} \frac{\partial T}{\partial t},$$

which indicates that the rate of heat conduction through a material is proportional to the second derivative of the temperature gradient. For a finite-difference solution, the differential terms in the heat equation are replaced by linear approximations based on the temperature differences between adjacent nodes. General presentations of finite difference methods applied to heat conduction can be found in Incropera and Dewitt (1986) and many other heat transfer texts.

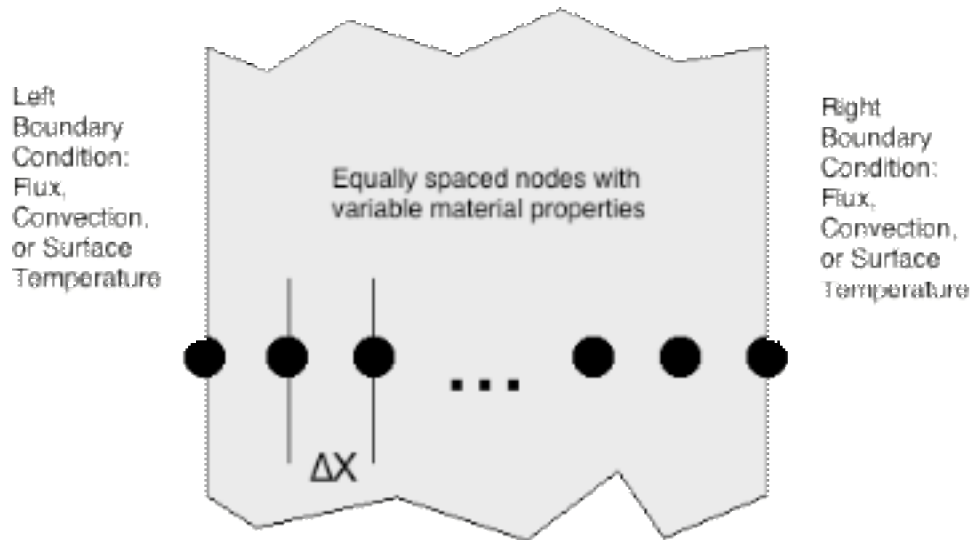


Figure 1. Schematic of the finite-difference skin conduction model.

An implicit solution method is used in the current model. The implicit method represents the node temperatures as a set of interrelated equations, one for each node. These equations are solved simultaneously to determine the node temperatures at the next time step. Because each node is affected only by the adjacent nodes, the solution matrix is tridiagonal (only the main and adjacent diagonals have nonzero entries), and an efficient solution method can be used which is linear in the number of nodes.

The simulation proceeds by a marching solution. The solution interval is divided into time steps ( $\Delta t$ ), and the temperature at each node is calculated at each time step in sequence. This method allows the boundary conditions to be changed at the granularity of the time step, typically one millisecond.

The model is based on a generic solver core which solves the heat conduction problem for a series of  $N$  nodes with some initial temperature distribution and specified boundary conditions. For convenience, the boundary conditions are referred to as “Left” and “Right” and correspond to the outside surface of the skin and the inside of the body, respectively. Note, though, that the model can be used to simulate a variety of different heat transfer problems that can be adequately modeled by a finite slab.

The model supports three types of boundary conditions: conduction (surface temperature), convection, and flux. For the conduction condition, the temperature history of the surface node is specified. This is equivalent to the surface node being in contact with a surface at the specified temperature. The conduction boundary condition is used at the right (internal) boundary for skin simulations. The convection boundary condition simulates heat transfer from a gas characterized by a free-stream temperature ( $T_{inf}$ ) and a convection coefficient ( $h$ ). The thermal flux from a convection boundary condition is equal to the temperature difference between the surface and  $T_{inf}$  multiplied by the convection coefficient. This boundary condition is used to simulate airbag exhaust gas impinging on the surface of the skin. The thermal flux boundary condition imposes a specified flux on the surface node. Note that flux is positive to the right, so that a positive flux at the left boundary represents heat entering the model, while a positive flux at the right boundary removes heat from the model. The flux boundary condition is used with experimental data from heat flux sensors to calculate burn potential.

The heat transfer model calculates a temperature history at each node for the duration of the simulation, based on the initial temperature distribution and the boundary conditions. The burn injury integral function ( $\omega$ ) is applied to the temperature history at the critical skin depth (basal layer of the epidermis) to determine if a threshold second-degree burn would have occurred. For more information on the burn integral function, see Reed et al. (1994).

## 2.2 Software

The computer model currently consists of one executable program file, labeled BURN9809.EXE, that was compiled as a project within the Visual C++ programming environment running under WindowsNT. However, the file BURN9809.CPP can be compiled to run in any operating system using an appropriate C/C++ compiler. The BURN9809 program takes the directory location of the input data files as an argument and should be called from a batch file (BURN.BAT).

The BURNMODL program reads from one to 11 input files, depending on the type of simulation and available input data. The output can be set to either print to a single, summary output file or to the monitor screen. The BURNMODL program can be run in either batch or standard mode. Standard mode requires the user to press the <ENTER> key at the beginning and end of each program module; whereas, batch run mode requires no key strokes from the user. Both of these provide the user with visual feedback verifying the model is running and not in a 'hung' state; however, the standard mode allows the user to exit the model at an intermediate stage if desired.

The user-provided input data for all studies must be provided in one ASCII (text) file named DATA.DAT. This file must adhere to the specified format (see below). Additional input data may be provided in any of nine optional data files named INITTEMP, CONDUCT, HEATCAP, DENSITY, LEFTBC, LEFTAMB, RIGHTBC, RIGHTAMB, LEFTIMP, and GASPROP.

The main data file DATA.DAT contains system constants and user controls, and may contain all of the input data for a simulation. LEFTIMP and GASPROP must be used for Impinging Jet Convection simulation, which simulates airbag exhaust gas from a discrete vent impinging on the skin. The remainder of the input files may be used to input large streams of depth/time dependent data, such as flux versus time data from a thermal flux sensor used in a deployment test.

The simulation output file is OUTPUT.TXT. It contains a tabular summary of the desired output, either Temperature versus Time or Temperature versus Depth. The burn injury integral,  $\omega$ , at the specified depth and the depth where  $\omega$  is equal to one conclude OUTPUT.TXT for any Temperature versus Time output.

The input and output files discussed above are summarized in Table 1. An example of DATA.DAT is shown in Figure 1. All of the input and output files are described more fully in Section 5.0, and listings of example files are included in Appendices B to F. Source code and executable program files, as well as data files, are on an MS-DOS distribution diskette.

Table 1  
Input and Output Data Files

Data File Type	File Name	Type of Simulation	Status	Contents
Input	DATA.DAT	All	Required	Controls/constants
Input	INITTEMP	Any	Optional	Depth vs. initial node temperatures
Input	CONDUCT	Any	Optional	Depth vs. initial node temperatures
Input	HEATCAP	Any	Optional	Depth vs. material heat capacity
Input	DENSITY	Any	Optional	Depth vs. material density
Input	LEFTBC	Any	Optional	Time vs. left boundary condition
Input	LEFTAMB	Any	Optional	Time vs. left side ambient temperature
Input	RIGHTBC	Any	Optional	Time vs. right boundary condition
Input	RIGHTAMB	Any	Optional	Time vs. right side ambient temperature
Input	LEFTIMP	Impinging Jet	Required	Controls/constants
Input	GASPROP	Impinging Jet	Required	Temperature vs. gas properties
Output	OUTPUT.TXT	All	Optional	Summary of results
Output	Screen	All	Optional	Summary of results

### 3.0 USING THE MODEL

The general procedure for using the model is:

1. Copy the distribution files into a directory on a computer hard drive. The model is compiled to run under Windows 95 or Windows NT.
2. Configure the batch file BURN.BAT with the appropriate paths to the executable and data files (see the ReadMe file on the distribution diskette).
3. Configure the input data files (text files) to specify the simulation conditions.
2. Execute the model by running the batch file BURN.BAT, which calls the executable. The program runs in a text console window. If not running in batch mode, press the <enter> key to proceed through the program modules.
5. Examine the OUTPUT file to obtain temperature histories, temperature distributions, or burn integral (omega) values.

The following sections describe the configuration of the input data files. All of the model features are accessed through the input data files.

### 3.1 Quick Start

This section provides a condensed guide to configuring the input files, listing only the data input elements that normally should be configured. Consult the sections below for more detailed information.

1. Identify the simulation type and find the appropriate preconfigured data directory:

Simulation Type	Description	Directory
Discrete Airbag Vent	Simulates the heat transfer due to a round jet of airbag exhaust gas impinging on the skin.	DISCRETE
Flux	Simulates the heat transfer due to a measured flux imposed on the skin. Used for interpreting data collected from heat flux sensors.	FLUX
Contact Burn	Simulates a contact burn by setting the skin surface temperature. This will generally be valid only for high-conductivity contact, such as by hot metal or liquids.	CONTACT

2. Back up the selected directory, then make changes to the files as follows:

#### DISCRETE

- a. No changes are needed to DATA.DAT.
- b. Configure GASPROP and LEFTIMP files (see appropriate sections below). If the impinging gas is reasonably approximated by nitrogen, configure only LEFTIMP.
- c. Edit the data path in BURN.DAT to match the data directory. Execute BURN.DAT.

#### FLUX

- a. No changes are needed to DATA.DAT.
- b. Configure LEFTBC.DAT with {time, flux} pairs. See sample files in Appendix A.
- c. Edit the data path in BURN.DAT to match the data directory. Execute BURN.DAT.

## CONTACT

- a. No changes are needed to DATA.DAT.
- b. Configure LEFTBC.DAT with {time, surface temperature} pairs. See sample files in Appendix A.
- c. Edit the data path in BURN.DAT to match the data directory. Execute BURN.DAT.

For more information on varying simulation parameters and controlling the simulation, see the sections below.

### 3.2 DATA.DAT

DATA.DAT is the primary input file, and contains all system constants and user controls for the simulation. All other input files are read only if DATA.DAT is configured to require those files.

The comment lines at the start of the file describe some of the input quantities that comprise the data set. Some of these quantities need little explanation beyond the short one given in the comment lines. Additional clarification of most quantities is given below. For dimensional quantities, units are given in parentheses at the end of the input line. It is important to remember that input data is required on each line even if the specific input is not utilized in calculation of the output. The best procedure is to save the distribution version of the input file for use as a template for simulations. Several different versions of DATA.DAT are included with the distribution.

The lines in DATA.DAT are of the general form ! KEYWORD: VALUE, where '!' identifies a data line with multiple data lines to follow and KEYWORD is a word or series of words that indicate to the program the type of data that follows. For some keywords the value is a flag (e.g., -1, 0, or 1) that indicates a specific action. For other keywords, the value is an input parameter, such as an initial temperature.

The following section describes each input line. The input lines are divided into blocks. The first block specifies the desired output and sets several key parameters.

```
OUTPUT:
OUTPUT TEMP vs. DEPTH @ time:      0      (sec)
OUTPUT TEMP vs. TIME @ node:      9
OUTPUT TO FILE(1) or SCREEN(0):    1
OUTPUT A-MATRIX:                   0
OUTPUT INIT-NODE-TEMPS:            0
OUTPUT NODE WHERE OMEGA = 1:       0
BATCH RUN:                          0
OUTPUT TRIAL:                       1
TRIAL EXECUTION DATE:              11/14/98
TRIAL EXECUTED BY:                 SJR
```

```
Line 1: OUTPUT TEMP vs. DEPTH @ time:      0      (sec)
```

The output file can contain either the temperature distribution in the model at a specified time, or the temperature history at a specified node. To output a temperature distribution (temperature vs. depth) set this line equal to any multiple of the time increment delta-t, and set the next line equal to zero. Alternatively, if a temperature history is to be output, set this line equal to zero.

Line 2: OUTPUT TEMP vs. TIME @ node: 9

To obtain the time history at a specified node (typical setting for determining burn potential at the critical skin depth), set this line to the desired node number. Use the equation  $\text{nodenumber} = (\text{Critical Depth} / \text{Delta-x}) + 1$  to obtain the node number. Set this line to zero if a temperature distribution (first line) is desired instead.

NOTE: To obtain temperature versus depth for all times (i.e., all node temperatures at all time steps), set both Line 1 and Line 2 inputs to zero. This may result in a very large output file, depending on the number of time steps and nodes specified.

Line 3: OUTPUT TO FILE(1) or SCREEN(0): 1

Specify whether the desired output should be displayed on the screen (0) or written to a file (1).

Line 4: OUTPUT A-MATRIX <1/0>

A one should be entered here for outputting the A-matrix. A zero input will not output the matrix. This option is used primarily for debugging and is not generally useful for simulations.

Line 5: OUTPUT INIT-NODE-TEMPS: 0

Input 1 to output the initial temperature distribution, or 0 for no initial node temperature output.

Line 6: OUTPUT NODE WHERE OMEGA = 1: 0

Set Line 6 input to 1 if the program should determine at which depth  $\Omega = 1$ . Otherwise, set this value to zero and  $\Omega$  will be calculated only at the node in Line 2. This is only valid if Line 2 is non-zero.

Line 7: BATCH RUN: 1

Set line 7 equal to 1 for running in batch mode. Batch mode requires no user key strokes while the model is running. Enter a 0 to user key stroke prompts. The model is generally run in non-batch, or standard, mode. This allows the user to exit the program gracefully should a problem arise.



Line 8: OUTPUT TRIAL: 1

This integer will appear in the header of the output file and can be used for reference.

Line 9: TRIAL EXECUTION DATE: 08/01/98

This date will appear in the header of the output file.

Line 10: TRIAL EXECUTED BY: SJR

Additional trial identification. This value will appear in the header of the output file. No spaces are permitted in this field.

The next series of input lines supply constants to the burn integral calculations. These should remain unchanged, since they determine the sensitivity of the model to burn injury. These constants have been calculated by adjusting the burn model performance to match human burn threshold data. If G is set to 0.00 the values shown below will be used as system defaults.

```
BURN INTEGRAL:
G: 3.442707 x 10^94
Delta-E: 623580
R: 8.314390
```

The next set of input lines sets up the model. The boundary and initial conditions are specified, as are the node spacing (delta-x), the time step (delta-t) and the total simulation duration.

```
INPUT:
LBC: Convection
RBC: Temperature
TIME STEPS: 100
DELTA-t: 0.0100000 (sec)
DELTA-x: 0.0000100 (m)
NODES: 100
```

Line 14: LBC: Convection

Specify the left-side boundary condition by inputting one of: Flux, Convection, or Temperature. Convection is used for a typical airbag burn simulation in which an exhaust gas jet is impinging on the skin surface.

Line 15: RBC: Temperature

Specify the right-side boundary condition by inputting one of: Flux, Convection, or Temperature. The Temperature mode is typically used for the right side in skin simulations. This mode is used to hold the innermost node of the model at body temperature.



INITTEMP.DAT. Enter the corresponding number of rows of paired data in the lines immediately following this input. Refer to Appendix A for examples of tabular input. Data pairs are in units of meters and degrees Kelvin.

Line 21:

```
! CONDUCTIVITY:          4                (x, W/mK)
                        0.000000      0.209490
                        0.000080      0.209490
                        0.000081      0.379100
                        0.001000      0.379100
```

This line specifies the thermal conductivity of the material (nominally skin). These values represent conductivity for epidermis and dermis and should not be changed for routine skin burn simulations. Enter the number of pairs of data that will follow this line. Enter zero if the available data is in CONDUCT.DAT. Enter the corresponding number of rows of paired data in the lines immediately following this input. Refer to Appendix A for examples of tabular input. Data pairs are in units of meters and W/m/K.

Line 22:

```
! HEAT_CAPACITY:        4                (x, J/kgK)
                        0.000000      2882.760000
                        0.000080      2882.760000
                        0.000081      2651.050000
                        0.001000      2651.050000
```

This line specifies the heat capacity of the material. These values represent epidermis and dermis and should not be changed for routine skin burn simulations. Enter the number of pairs of data that will follow this line. Enter zero if the available data is in HEATCAP.DAT. Enter the corresponding number of rows of paired data in the lines immediately following this input. Refer to Appendix A for examples of tabular input. Data pairs are in units of meters and J/kg/K.

Line 23:

```
! DENSITY:              2                (x, kg/m^3)
                        0.000000      800.000000
                        0.001000      800.000000
```

This line specifies the density of the skin. These values have been set during model validation and should not be changed for routine skin burn simulations. Enter the number of pairs of data that will follow this line. Enter zero if the available data is in DENSITY.DAT. Enter the corresponding number of rows of paired data in the lines immediately following this input. Refer to Appendix A for examples of tabular input. Data pairs are in units of meters and kg/m<sup>3</sup>.

Lines 24 and 25 control the surface (left-side) boundary conditions for the model, and should be modified by the user as necessary.

Line 24:

```
! LEFT_SURFACE_BC-1:           -1           (t, W/m^2 | K | W/Km^2)
```

This line specifies inputs to configure the left-side boundary condition. Changes to this line are required whenever the left-side boundary condition is changed. If the input is a positive integer, it represents the number of data pairs following this line. Enter zero if the available data is in LEFTBC.DAT. The type of data is dependent on the type of left-side boundary condition specified earlier in the input file (line 13: LBC). If the boundary condition type is Convection, the inputs are taken to be {time, heat transfer coefficient} pairs. If the boundary condition type is Temperature, the data are taken to be {time, surface temperature} pairs. If the boundary condition type is Flux, the data are taken to be {time, thermal flux} pairs. Heat transfer coefficient, temperature, and thermal flux are in units of  $W/m^2/K$ ,  $K$ , and  $W/m^2$ , respectively.

For most airbag simulations, the discrete impinging jet flow model will be used. This model calculates the heat transfer coefficient as a function of time based on supplied values of gas velocity and temperature. To use the impinging jet model, enter -1 and configure the files LEFTIMP.DAT and GASPROP.DAT appropriately.

Line 25:

```
! LEFT_AMBIENT_BC-2:           4           (t, K)
                                0.000000      773.150000
                                0.155000      773.150000
                                0.155100      293.000000
                                1.000000      293.000000
```

The left side ambient temperature history is used when the left boundary condition type is Convection. Enter the number of pairs of data that will follow this line. Enter zero if the available data is in LEFTAMB.DAT. Enter the corresponding number of rows of paired data in the lines immediately following this input. If Line 23 is set to equal -1 to use the impinging jet model, this line supplies the impinging gas jet temperature versus time.

Line 26:

```
! RIGHT_SURFACE_BC-1:          2           (t, W/m^2 | K | W/Km^2)
                                0.000000      310.000000
                                1.000000      310.000000
```

The right surface boundary condition is set in the same manner as the left. For typical skin burn simulations, these values should not be changed. Values for this line are identical to the left boundary condition (see line 23). Enter the number of pairs of data that will follow this line. Enter zero if the available data is in RIGHTBC.DAT. Enter the corresponding number of rows of paired data in the lines immediately following this input.

Line 27:

```
! RIGHT_AMBIENT_BC-2:          2           (t, K)
                                0.000000      310.000000
                                1.000000      310.000000
```

This line supplies the right-side ambient temperature in the same manner as line 24 supplies the ambient temperature for the left side. These values are only used if the RBC is of type Convection. These values should not be changed for typical skin burn simulations. Enter the number of pairs of data that will follow this line. Enter zero if the available data is in LEFTAMB.DAT. Enter the corresponding number of rows of paired data in the lines immediately following this input. If Line 23 is set to equal -1 to use the impinging jet model, this line supplies the impinging gas jet temperature versus time

Line 28:

! END

This string sequence is used to identify the end of the file.

### 3.3 OPTIONAL INPUT FILES

Example data for INITTEMP, CONDUCT, HEATCAP, DENSITY, LEFTBC, LEFTAMB, RIGHTBC, RIGHTAMB are shown in Appendix A. These data sets contain the depth versus initial temperature distribution, depth versus material property distribution, and time versus boundary condition heat transfer value and ambient temperature distribution. The format for these files is similar to that of DATA.DAT, where the header section can have as many rows as possible as long as they begin with a '#' symbol and are limited to 80 characters each. The first line of each file contains the data line identifying character (!) and specifies the number of pairs of data to be read. The remainder of the file should be pairs of data in two columns, separated by at least one blank space followed by '! END' string sequence.

### 3.4 DATA FILES FOR IMPINGING GAS JET MODEL

The impinging jet model allows the user to simulate a round airbag exhaust gas jet impinging on the skin surface. This is the primary method for simulating airbag-induced skin burns. The files LEFTIMP.DAT and GASPROP.DAT provide input data specific to the impinging jet model. The simulated jet and target geometry are specified, as are the gas jet velocity, temperature, and thermal properties.

#### 3.4.1 LEFTIMP.DAT

Appendix A and the distribution disk include a properly configured DATA.DAT data file for using the impinging jet convection model, as well as a sample LEFTIMP.DAT file. Explanation of the file items follows:

Line 1:        On axis bump term:        <1/0>

This value is set to one to include the 'bump' term in the on-axis calculation. This is provided for historical reasons and should be set to zero for routine calculations. The “bump” refers to a trend in the heat flux data at some Reynolds numbers for a slight increase in Nusselt number with increasing distance at about

4 diameters. The model performance has been validated without the bump term, so it should not be used for most purposes.

Line 2:        Reynolds number:            <real number/0>

If the Reynolds number of the gas is known to be constant, enter that value here. This option is used at UMTRI for heat gun test conditions that produce a constant-velocity gas jet. For airbag simulations, enter a zero and be sure to enter the gas jet velocity history and the gas properties in GASPROP.DAT.

Line 3:        Vent diameter:            <real number>

Enter the diameter of the airbag vent, in meters. This is the gas jet diameter used in the calculations.

Line 4:        Target diameter:           <real number>

Enter the diameter of the skin target in meters. The heat transfer coefficient is averaged over the target diameter. For the most conservative analysis, enter 0.000 to use the on-axis heat transfer coefficient, which is generally higher than off-axis.

Line 5:        Target distance:           <real number>

Enter the distance from the jet outlet to the skin in meters. The impinging jet model is valid at distances from 0.5 to 10 times the jet diameter.

Line 6:        ! LBC\_convection\_input: <integer>

This line provides a means of specifying a portion of the simulation over which the impinging jet convection model is to operate. For a typical simulation, the impinging jet model will be used only during the gas flow interval, but the simulation will be carried out to at least one second to allow time for the heat to be conducted through the skin. Configure this line by entering {time, integer} pairs, where the integer 1 indicates that the impinging model should be used during this interval, or zero to revert to a constant convection coefficient.

Line 7:        ! Gas\_Velocity:            <integer>

Enter the number of pairs of gas jet velocity data that will be entered. Input the paired data in the subsequent lines in the form {time, velocity}.

NOTE: When using the Impinging Jet Convection Model make sure that first term on each pair of data in Line 25 of DATA.DAT and Lines 5 and 6 in LEFTIMP.DAT correspond. This means that there is a temperature value that corresponds to each velocity value.

### 3.4.2 GASPROP.DAT

The GASPROP.DAT file specifies the gas properties used in the impinging jet model, and is currently configured for nitrogen gas. The gas characteristics in GASPROP.DAT need little explanation beyond the short ones given in the comment lines. Note the units. Configure the file for any gas that is desired by simply replacing the current data with similarly formatted data that corresponds to the alternative gas. The file is currently limited to eighteen lines of inputs. Note that the model does not include condensation effects, so simulations using water vapor are likely to be inaccurate.

## 4.0 INTERPRETING THE OUTPUT

The output file contains a temperature history at a selected node, or a temperature distribution at a selected time, depending on the settings in the input file. The burn prediction is accomplished by computing the burn integral,  $\omega$ , from the temperature history at the critical skin depth. The value of  $\omega$  is appended to the bottom of the output file. The critical skin depth is usually taken to be the depth of the basal epidermal layer, which averages about  $80\ \mu\text{m}$ . This is the most superficial layer of the skin at which there are living cells that can sustain burn injury. The value of  $\omega$  calculated for the specified node (skin depth) is printed at the bottom of the output file, in scientific notation and to a fixed number of decimal places.

When  $\omega$  is less than 1, no burn is predicted. When  $\omega$  is greater than 1, skin burn is predicted for the specified depth. Thus,  $\omega = 1$  is the threshold for burn injury. When  $\omega = 1$ , the thermal exposure is predicted to result in a minimal severity second-degree burn.

$\omega$  cannot be interpreted as an index of burn severity. An  $\omega$  value of 2 is not twice as severe a burn as an  $\omega$  value of 1. Because the  $\omega$  function is highly nonlinear, very large or very small  $\omega$  values are common in simulations.  $\omega$  has meaning only at the burn threshold.

The best way to gauge burn severity and to rank different exposures in terms of burn potential is to compute the depth of the skin at which  $\omega = 1$ . Generally speaking, a deeper burn is a more severe burn. The input file DATA.DAT includes a flag that will prompt the program to search for the node at which  $\omega$  is closest to one. If this flag is set, the program will find the two nodes one either side of  $\omega = 1$  and will linearly interpolate for the depth at which  $\omega = 1$ . This depth is then appended to the end of OUTOUT.TXT. It should be noted that the depth calculation is linear whereas the burn integral is exponential; therefore, any interpolated depth is only an approximation. If this depth is less than the critical skin depth, no burn is predicted. The  $\omega = 1$  at a depth greater than the critical skin depth, a burn is predicted, with greater depths indicating more severe burns. Note that for very low thermal exposures, no node will experience a temperature history such that  $\omega$  exceeds one.

## 5.0 CHECKING RESULTS AND TROUBLESHOOTING

The current release of the model (9809) is the second release of the new C-language version and does not yet contain substantial error checking and useful diagnostic messages. The user must exercise care to verify that the results are accurate. This section discusses some checks that can be made and addresses some common types of errors.

## 5.1 Simulation Duration

The simulation duration must be sufficient that the burn integral value at the critical depth is no longer increasing with increasing time. This duration needs to be about 1 second for a typical airbag simulation with 100 ms of gas flow, and longer for a longer exposure duration. The extended duration is required to allow the cool below the threshold for burn injury. Verify that the simulation duration is sufficient by adjusting the number of time steps in DATA.DAT until the calculated omega value does not increase appreciably with additional time. The program will continue to function, but inaccurate results may be obtained.

## 5.2 Node Depth

The total depth of the model must be sufficiently large that no appreciable temperature gradient develops at the interior nodes. In effect, this causes the model to simulate a semi-infinite solid. The adequacy of the model depth can be tested by selecting a node at about 90 percent of the depth and outputting the temperature history at that node. Ideally, the node should remain at the starting temperature and show no temperature increase during the test.

## 5.3 File Not Found: GetData() Errors

When the program is not able to find the required input files, a File Not Found or GetData() error may result. Check the path names in the BURN.BAT file. Ensure that the files are permitted for read access and are not open in any other application. Check also to be sure that the files are text only and were not saved in any other format. Be certain that all of the files required for the selected simulation type are available in the specified directory.

## 5.4 Other Errors

Most other errors can be traced to an improperly formatted input file. Try starting with one of the demo files from the original distribution and making incremental changes to identify the problem. If no solution is found, please contact the UMTRI researchers listed below. We will be happy to make changes to fix problems with the model.

## 6.0 SUPPORT

Questions concerning the model should be directed to the contacts listed below. Users of the model should feel free to contact us with problems, comments, or suggestions. Input from users helps us to improve the usefulness and efficiency of the model.

Steve Reed	sjreed@umich.edu	734-936-1114
Matt Reed	mreed@umich.edu	734-936-1111

## 6.0 REFERENCES

Reed, M.P., and Schneider, L.W. (1994). *Skin Burns from Airbag Exhaust Gas: Laboratory Experiments and Mathematical Modeling*. Technical Report UMTRI-94-24. University of Michigan Transportation Research Institute, Ann Arbor, MI.

Reed, M.P., Schneider, L.W., and Burney, R.E. (1994). Laboratory investigations and mathematical modeling of airbag-induced skin burns. Technical Paper No. 942217.



*Proc. 38th Stapp Car Crash Conference*, pp. 177-190. Society of Automotive Engineers, Warrendale, PA.

Bowman, B.M., and Reed, M.P. (1996). *User's Guide for the UMTRI Airbag Skin Burn Model*. Technical Report UMTRI-96-5. University of Michigan Transportation Research Institute, Ann Arbor, MI.

Incropera, F.P. and DeWitt, D.P. (1985). *Introduction to Heat Transfer*. New York: Wiley.

APPENDIX A  
SAMPLE DATA FILES

```

#-----
#           Input File for
#           UMTRI Airbag Skin Burn Model
#           Release 9809
#
# File: DATA.DAT
#
# General:
#   All Title Block lines MUST begin with a pound sign (#)
#   Add as many lines as desired to this Title Block
#   Maintain this file's format for proper input
#
# Output:
#   Either TEMP VS. DEPTH OR TEMP VS. TIME must be set to 0
#   Given depth use NODE=(depth/DX)+1 for TEMP vs. TIME @ NODE
#   Set FIND NODE WHERE OMEGA = 1: to 1 for DO find; 0 for DON'T find
#
# Burn Integral:
#   Set Burn Integral values to 0.00 to use default values
#
# Input:
#   All nodes, times and depths start with zero
#   TOTAL TIME = DT * TIME STEPS
#   maximum TIME STEPS = 10001
#   NODES = (total depth/DX) + 1
#   maximum NODES = 10000
#   TIME STEPS must be greater than NODES
#   DX must be a factor of critical depth (usually .000072m)
#   or critical depth must be at a NODE
#   All data sets must have:
#       The number of data pairs you are providing
#       at least 2 pairs of data: first and last node/time step
#       For constant values, set second terms equal to each other
#
#   For Conductivity, Heat Capacity, and Density:
#       First data pair is for node 0 at depth 0.00
#       Last data pair is for node NODES at depth = DX*(NODES-1)
#
#   For Boundary Condition inputs:
#       Based on the Left or Right Boundary Condition,
#       Left/Right Boundary Condition 1 is time vs. (temp|flux|h) and
#       Left/Right Boundary Condition 2 is time vs. ambient temperature
#
#-----Note: The title block lines must be less than 80 characters in length.-----

OUTPUT:
OUTPUT TEMP vs. DEPTH @ time:      0      (sec)
OUTPUT TEMP vs. TIME @ node:      9 (dx*(CritDepth_node-1)=critical_depth)
OUTPUT TO FILE(1) or SCREEN(0):   1
OUTPUT A-MATRIX:                  0
OUTPUT INIT-NODE-TEMPS:           0
OUTPUT NODE WHERE OMEGA = 1:      1
BATCH RUN:                         0
OUTPUT TRIAL:                      1
TRIAL EXECUTION DATE:             09/22/98

```

TRIAL EXECUTED BY:

SJR

BURN INTEGRAL:

G: 3.442707 x 10<sup>94</sup>  
Delta-E: 623580  
R: 8.314390

INPUT:

LBC: Convection  
RBC: Temperature  
TIME STEPS: 100  
DELTA-t: 0.01000 (sec)  
DELTA-x: 0.0000100 (m)  
NODES: 100

! INITIAL\_NODE\_TEMPERATURES: 2 (x,T)  
0.000000 310.000000  
0.001000 310.000000

! CONDUCTIVITY: 4 (x,W/mK)  
0.000000 0.209490  
0.000080 0.209490  
0.000081 0.379100  
0.001000 0.379100

! HEAT\_CAPACITY: 4 (x,J/kgK)  
0.000000 2882.760000  
0.000080 2882.760000  
0.000081 2651.050000  
0.001000 2651.050000

! DENSITY: 2 (x,kg/m<sup>3</sup>)  
0.000000 800.000000  
0.001000 800.000000

! LEFT\_SURFACE\_BC-1: -1 (t,W/m<sup>2</sup>|K|W/Km<sup>2</sup>)

! LEFT\_AMBIENT\_BC-2: 4 (t,K)  
0.000000 773.150000  
0.155000 773.150000  
0.155100 293.000000  
1.000000 293.000000

! RIGHT\_SURFACE\_BC-1: 2 (t,W/m<sup>2</sup>|K|W/Km<sup>2</sup>)  
0.000000 310.000000  
1.000000 310.000000

! RIGHT\_AMBIENT\_BC-2: 2 (t,K)  
0.000000 310.000000  
1.000000 310.000000

! END

These four files are used to set the material properties as a function of depth.

```
#-----  
#  
#   File: INITTEMP.DAT  
#  
#   Initial Temperature Distribution  
#  
#   depth (m), Temp (K) datapairs  
#  
#-Note: The title block lines must be less than 80 characters in length.
```

```
! Number_of_datapairs: 5  
0.000000    310.00  
0.000050    310.00  
0.000100    310.00  
0.000500    310.00  
0.001000    310.00
```

```
! END
```

```
#-----  
#  
#   File: CONDUCT.DAT  
#  
#   Material Conductivity Distribution  
#  
#   depth (m), Conductivity (W/mK) datapairs  
#  
#-Note: The title block lines must be less than 80 characters in length.
```

```
! Number_of_datapairs: 4  
0.000000    0.209490  
0.000080    0.209490  
0.000081    0.379100  
0.001000    0.379100
```

```
! END
```

```
#-----  
#  
#   File: HEATCAP.DAT  
#  
#   Heat Capacity Distribution  
#  
#   depth (m), Heat Capacity (J/kgK) datapairs  
#  
#-Note: The title block lines must be less than 80 characters in length.
```

```
! Number_of_datapairs: 4  
0.000000    2882.760000  
0.000080    2882.760000
```

```
0.000081    2651.050000
0.001000    2651.050000
```

```
! END
```

```
#-----
#
#   File: DENSITY.DAT
#
#   Material Density Distribution
#
#   depth (m), Density (kg/m^3) datapairs
#
#-----Note: The title block lines must be less than 80 characters
in length.-----
```

```
! Number_of_datapairs: 2
0.000000    800.000000
0.001000    800.000000
```

```
! END
```

These four files are used to set the boundary conditions as a function of time.

```
#-----
#
#   File: LEFTBC.DAT
#
#   Left-side Boundary Condition Distribution
#
#   time (sec), Convection(W/m^2)/Constant
#               Temperature(K)/Conduction(W/m^2) datapairs
#
#-Note: The title block lines must be less than 80 characters in length.
```

```
! Number_of_datapairs: 2
0.000000    350.000000
1.000000    350.000000
```

```
! END
```

```
#-----
#
#   File: LEFTAMB.DAT
#
#   Left-side Ambient Temperature Distribution
#
#   time (sec), Temp (K) datapairs
#
#-Note: The title block lines must be less than 80 characters in length.
```

```
! Number_of_datapairs: 4
0.000000    773.150000
0.155000    773.150000
0.155100    293.000000
1.000000    293.000000
```

```
! END
```

```
#-----
#
#   File: RIGHTBC.DAT
#
#   Right-side Boundary Condition Distribution
#
#   time (sec), Convection(W/m^2)/Constant
#               Temperature(K)/Conduction(W/Km^2) datapairs
#
#-Note: The title block lines must be less than 80 characters in length.
```

```
! Number_of_datapairs: 2
0.000000    310.000000
1.000000    310.000000
```

```
! END
```

```
#-----
#
#   File: RIGHTAMB.DAT
#
#   Right-side Ambient Temperature Distribution
#
#   time (s), Temp (K) datapairs
#
#-Note: The title block lines must be less than 80 characters in length.
```

```
! Number_of_datapairs: 2
0.000000    773.150000
1.000000    310.000000
```

```
! END
```

These two files are used to set the additional impinging jet conditions.

```
#-----
#
#   File: LEFTIMP.DAT
#
#   Impinging Jet Convection Model
#
# Variables:
#   Use onaxis bump - (onaxisbump)
#       - 1 will force the code to use the bump term
#       - 0 will disregard bump code
#   Reynolds Number - (Re) - input for known Reynolds number. Any gas
#   Gas Jet Temperature - (Tout) - (degree K) temp at the jet outlet
#   Gas Jet Velocity - (constgasV) - constant (m/s) gas velocity
#   Vent Diameter - (ventD) - airbag vent diameter in (m)
#   Target Diameter - (targD) - surface target diameter in (m)
#   Target Distance - (targdist) - (m) dist. from gas jet to surface
#
#   LBC Convection input pairs:
#   arranged in (time (sec), true-false) pairs
#       - 1 denotes convection is affecting the system at time listed
#       - 0 denotes no convection is occurring during the interval
#
# Gas Jet Velocity:
#   first line is text; number of datapairs; units for data
#   arranged in (time (sec), Velocity) pairs
#
# Alternative Gas constants:
#   first line is text; true-false input
#       - 1 to verify use of alternative gas
#       - 0 for Nitrogen
#   if alternative gas is set to 1 the following data must be supplied
#   Conductivity - (W/mK)
#   Prandtl Number -
#   Kinematic Viscosity - (m^2/s)
#
# NOTE: The header files can ONLY use 80 COLUMNS for text
# Gas Jet temp          576.70 was line after Reynolds #
#-Note: The title block lines must be less than 80 characters in length.
```

```
Use onaxis bump-term      0
```

```
Reynolds Number          0.000000
Vent diameter             0.010000
Target diameter           0.000000
Target distance           0.040000
```

```
! LBC_Convection_input:      2          (t,W/m^2)
                             0.000000      1
                             1.000000      1
```

```
! Gas_Velocity:             5          (t,m/s)
                             0.000000      52.00000
                             0.155000      52.00000
                             0.155100      0.000000
                             0.700000      0.000000
```



1.000000 0.000000

! END

```
#-----  
#  
# File: GASPROP.DAT  
#  
# Impinging Jet Convection Model  
#  
# Nitrogen Gas constants  
#  
# Note: This file can be configured for any gas. But is limited  
#       to 18 input lines.  
#  
# This file input is limited to a 18 row x 4 column table. It  
# was originally taken from Introduction to Heat Transfer,  
# Incropera and DeWitt, 1985.  
#  
# Variables:  
# Temp (K)      Kin. Visc.(m^2/s)      Cond. (W/mK)      Pr  
#-Note: The title block lines must be less than 80 characters in length.
```

100	0.00000200	0.00958	0.768
150	0.00000445	0.01390	0.759
200	0.00000765	0.0183	0.736
250	0.00001148	0.0222	0.727
300	0.00001586	0.0259	0.716
350	0.00002078	0.0293	0.711
400	0.00002616	0.0327	0.704
450	0.00003201	0.0358	0.703
500	0.00003824	0.0389	0.700
550	0.00004486	0.0417	0.702
600	0.00005179	0.0446	0.701
700	0.00006671	0.0499	0.706
800	0.0000829	0.0548	0.715
900	0.0001003	0.0597	0.721
1000	0.0001187	0.0647	0.721
1100	0.0001382	0.0700	0.718
1200	0.0001586	0.0758	0.707
1300	0.0001799	0.0810	0.701

