Final Manuscript for presentation at the 1995 SAE International Congress & Exposition,
Underhood Thermal Management Session,Paper Number 951012, Thursday 1:00 p.m. March 2, Cobo Center Room W2-70, Detroit, Michigan

A Methodology for Rapid Calculation of Computational Thermal Models

Keith R. Johnson, Bobbi S. Wood, Peter L. Rynes, and Ban K. Yee
Michigan Technological University

Frank C. Burroughs and Tim Byrd
Lydall, Inc.

ABSTRACT

Too often many heat management problems are not solved with thermal analysis because of excessive complexity, time, and cost. A method for quickly solving a sophisticated thermal/fluid system with minimal user interaction and with common desktop computer resources is presented. A desktop (Microsoft Windows^{TM}) thermal analysis package, WinTherm, consists of the Generic Processor (pre-processing software), the 3-D Thermal Model (a finite difference nodal network solver), and an Image Viewer (wireframe and animated thermal display). The theoretical basis for this thermal analysis toolkit will be discussed as well as examples of its implementation.

INTRODUCTION

Detailed thermal models using nodal finite difference or finite element techniques are well known and applied in various areas of engineering design. The computer resources and analyst experience that are required for this type of modeling are extensive. An alternative analysis tool has been developed that provides speed and simplicity within a sophisticated modeling environment. The software is developed for the Windows operating system so that your desktop computer can be used for thermal modeling.

By representing a generic geometry within a preassigned set of rules, the user can manipulate the structure and material choices to closely conform to the desired system or component. The analyst simply selects values for the dimensional and material parameters thus creating a specific instance of the generic model with the Generic Processor software. The Generic Processor generates a set of input files for this model which are ready for immediate analysis with the thermal modeling software. The output of the thermal model can then be viewed directly in its numerical tabular form, plotted with your spreadsheet or plotting package, or thermographically visualized with the Image Viewer.

This technique has been used for solving various heat management issues such as carpet/shoddy/insulation design requirements, heat shield performance, and exhaust component configuration. The analytical model has been extensively validated and refined through comparison of measurements taken from an actual physical testbox that has been used by Lydall, Inc. for material performance evaluation. Future plans will include other applications for the WinTherm software such as underhood thermal modeling.

USING WINTHERM

Computational thermal analysis typically requires a primary thermal analyzer code plus several pre or post-processors to support model building (input files) and visualization (geometric and temperature output). This often leads to confusion regarding what sequence to follow in the analysis as well as the location of the codes or data files. By integrating all of the tools and file control into a single program, WinTherm, a logical and intuitive analysis session will follow.

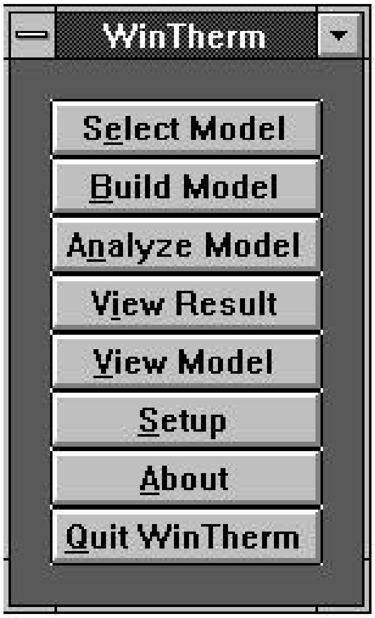


Figure 1: WinTherm Toolkit

WinTherm allows easy accesses to the three primary modeling modules: Generic Processor, Thermal Model, and Image Viewer; and it maintains consistent data file control. Any auxiliary modules that are needed for the analysis are called automatically. Figure 1 illustrates the simplicity of selecting a particular model and accessing the required tools.

The analysis proceeds in the following fashion and is illustrated further in Figure 2: 1) Select the Model to Process/Analyze; 2) Build the Model with the Generic Processor; 3) Analyze the Model with the 3-D Thermal Analyzer; 4) View the Tabular Results; and 5) View the geometry or thermal history with the Image Viewer.

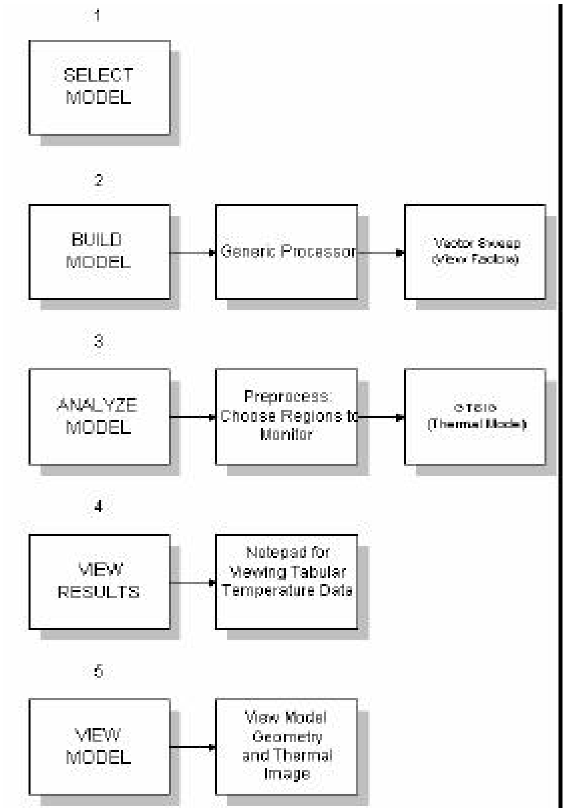
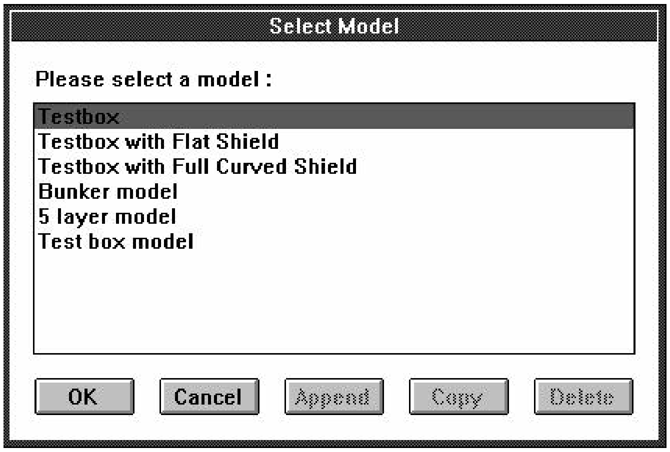


Figure 2: The Flow of WinTherm Processes

SELECT MODEL - The initial activity is selecting a model from the database as shown in Figure 3. This sets up all of the file control for further processing (modifying) and analyzing the model. This list represents different models in their baseline configuration. The append and delete buttons allow more or less models in this selection list window.



Please select a model :

- Testbox
- Testbox with Flat Shield
- Testbox with Full Curved Shield
- Bunker model
- 5 layer model
- Test box model

Figure 3: Selecting a Model to Process/Analyze in WinTherm

BUILD MODEL - Once a model has been selected, it can then be modified from its baseline configuration with the Generic Processor by clicking the Build Model button. The flow of activities for the Generic Processor is illustrated in Figure 4.

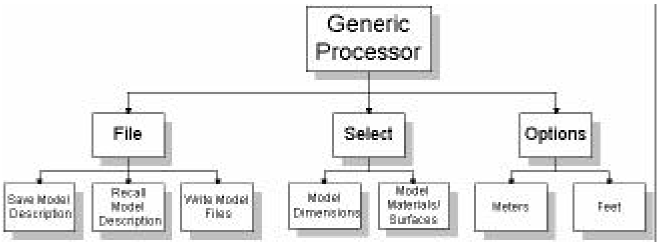
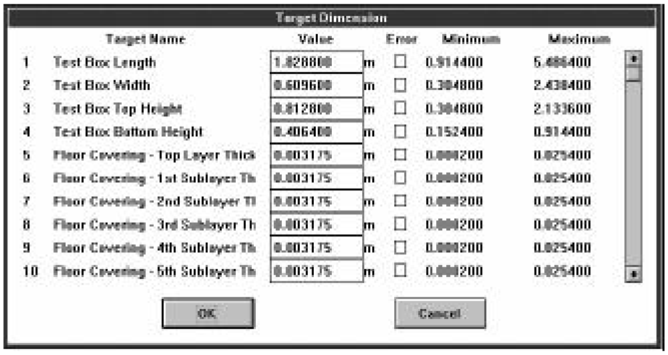


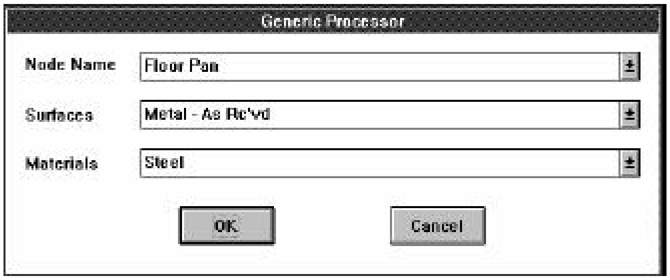
Figure 4: Building a Model with the Generic Processor

The option of specifying the dimensions with units of meters or feet is available and should be considered first. Then the *Select* button can be clicked to obtain the choice of modifying the dimensions of any of the components (Figure 5) or the surface/material selections for any of the components (Figure 6). Note that established minimums and maximums are in force at all times for the component dimensions and will change based on dependencies between the various components (e.g., if one exhaust section of this particular *testbox* model is changed, then another section will also indicate that it must change to conform to the length of the box). The surfaces and materials for each component that are available for selection can be modified in the GMD file (see Theory of the Generic Processor).



	Target Name	Value		Error	Minimum	Maximum
1	Test Box Length	1.828800	m	☐	0.914400	5.486400
2	Test Box Width	0.609600	m	☐	0.304800	2.438400
3	Test Box Top Height	0.812800	m	☐	0.304800	2.133600
4	Test Box Bottom Height	0.406400	m	☐	0.152400	0.914400
5	Floor Covering - Top Layer Thick	0.003175	m	☐	0.000200	0.025400
6	Floor Covering - 1st Sublayer Th	0.003175	m	☐	0.000200	0.025400
7	Floor Covering - 2nd Sublayer Th	0.003175	m	☐	0.000200	0.025400
8	Floor Covering - 3rd Sublayer Th	0.003175	m	☐	0.000200	0.025400
9	Floor Covering - 4th Sublayer Th	0.003175	m	☐	0.000200	0.025400
10	Floor Covering - 5th Sublayer Th	0.003175	m	☐	0.000200	0.025400

Figure 5: Adjusting the Dimensions of the Model Components



Node Name: Floor Pan

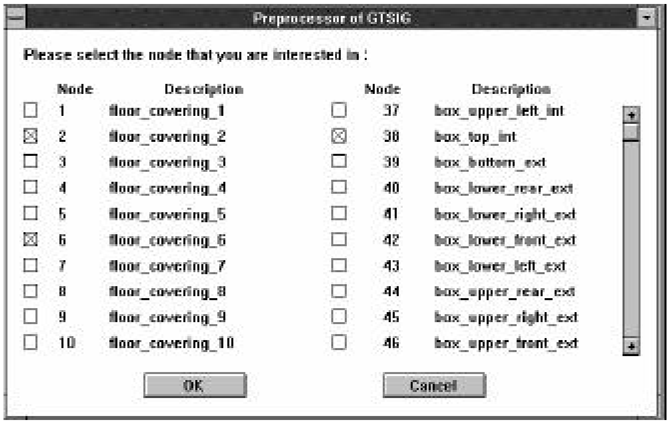
Surfaces: Metal - As Rc'vd

Materials: Steel

Figure 6: Changing the Surface and Material Properties of the Model Components

Once all of the dimension and surface/material modifications have been processed, then this new version of the model can be saved and written out for subsequent analysis. During this file generation step, the new geometry and nodal network are calculated. This process produces a new facet file and other files that contain the new nodal capacitances and conductance linkages (conduction, convection, and radiation) for the model. The Save/Recall Model Description provides a convenient shortcut and bookkeeping function for doing case studies.

ANALYZE MODEL - Once a model has been processed and the files generated, it can then be analyzed with the Thermal Model to determine temperatures and heat rates. A pre-processor is provided to easily select various input parameters that the Thermal Model (GTSIG) has available. Selecting nodes to write out to a data file for subsequent inspection or graphing as shown in Figure 7 is one function of this pre-processor.



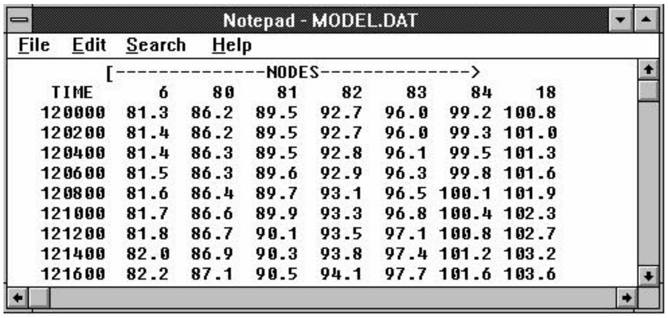
Please select the node that you are interested in :

	Node	Description		Node	Description
☐	1	floor_covering_1	☐	37	box_upper_left_int
☒	2	floor_covering_2	☒	38	box_top_int
☐	3	floor_covering_3	☐	39	box_bottom_ext
☐	4	floor_covering_4	☐	40	box_lower_rear_ext
☐	5	floor_covering_5	☐	41	box_lower_right_ext
☒	6	floor_covering_6	☐	42	box_lower_front_ext
☐	7	floor_covering_7	☐	43	box_lower_left_ext
☐	8	floor_covering_8	☐	44	box_upper_rear_ext
☐	9	floor_covering_9	☐	45	box_upper_right_ext
☐	10	floor_covering_10	☐	46	box_upper_front_ext

Figure 7: Selecting Nodes for the Thermal Model to Write to a Data File

After the preprocessing of the Thermal Model is completed, the Thermal Analyzer begins numerically processing the heat transfer solution to completion. Temperature output files are generated at this stage.

VIEW RESULTS - This function provides a convenient method for inspecting or editing the temperature data file generated by the previous Analyze Model activity. Figure 8 illustrates the Time versus Nodal Temperature table that was produced in the *model.dat* file. Another file is produced called *model.out* that contains a more robust listing of temperature and heat flow data for the model analysis.



```
        [--------------NODES-------------->
  TIME       6     80     81     82     83     84     18
 120000   81.3   86.2   89.5   92.7   96.0   99.2  100.8
 120200   81.4   86.2   89.5   92.7   96.0   99.3  101.0
 120400   81.4   86.3   89.5   92.8   96.1   99.5  101.3
 120600   81.5   86.3   89.6   92.9   96.3   99.8  101.6
 120800   81.6   86.4   89.7   93.1   96.5  100.1  101.9
 121000   81.7   86.6   89.9   93.3   96.8  100.4  102.3
 121200   81.8   86.7   90.1   93.5   97.1  100.8  102.7
 121400   82.0   86.9   90.3   93.8   97.4  101.2  103.2
 121600   82.2   87.1   90.5   94.1   97.7  101.6  103.6
```

Figure 8: Viewing Temperature versus Time for the Monitored Nodes

The *model.dat* file is an ascii text file that is configured for importing into a spreadsheet (Figure 9); otherwise, it can be tailored with the notepad for other graphing application software.

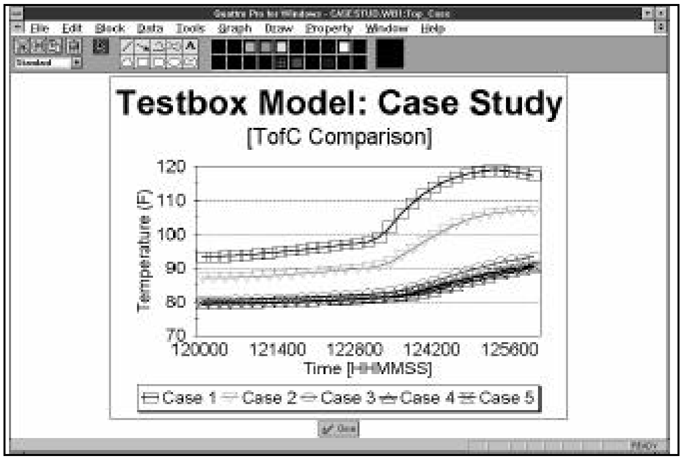


Figure 9: Directly Graphing Tabular Results with a Spreadsheet

VIEW MODEL - An Image Viewer (Figure 10) is provided that will allow 3-D display of your geometry (wireframe) or the shaded thermal image in gray scale (color intensity correlated to temperature). This option quickly allows the analyst to inspect the dimensional changes that were made during the Generic Processor session and the subsequent thermal performance impact of the additional material/surface changes. The spatial representation of temperature in the form of a calibrated thermal image is an effective way to assess the results of a transient heat transfer analysis. The operation of the Image Viewer is straightforward as outlined below.

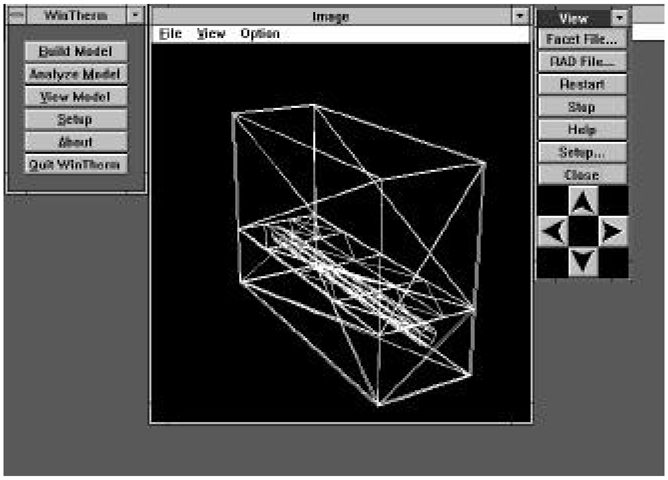


Figure 10: Viewing the Model Geometry or Animating the Temperature versus Time in Gray Scale (Thermography)

Load a Facet File - To load a facet file, you just click on the *Facet File* button. After making a selection, the facet file is read and an image window is produced. The wire-frame geometry of the model is displayed in the image window and the arrow buttons will rotate the model in all directions.

Load a Radiance File - Once you obtain your preferred view angle, you can load the radiance file to run the thermal simulation. A dialog box will be displayed to obtain the range of temperatures that you prefer to visualize. Default temperature values indicate the min and max temperatures in the file. If a narrower temperature range is chosen to increase visual sensitivity, any facet temperatures outside the new range will be simply saturated (black or white).

Run Simulation - After you have entered the temperature range, you can run the simulation by clicking the *Start* button in the view window. The model is shaded in a gray scale with all the hidden surfaces removed. You will notice the gray scale of the model changing and the PC speaker beeping as the Image Viewer steps through time with the corresponding image frame. There are options to pause, stop, or restart the simulation at any time. If you rotate the model within the simulation, the processing will stop and the wire-frame of the model is displayed. You can then change to another angle of view and continue the simulation or start over. When the simulation is restarted, you can change the temperature range by selecting the *Setup* button in the view window. This feature will aid in obtaining the best resolution display of the thermal image.

Multiple Image Windows - The Image Viewer supports multiple image windows (maximum of four). With multiple image windows, you can have more than one model displayed at anytime or you can have a model displayed in more than one angle of view. Click the *Facet File* button in the view window to open another facet file and the new image will be displayed in another image window.

THEORY OF THE GENERIC PROCESSOR

The goal is to design an analytical model that can be easily adjusted to suit a particular analysis. The approach taken was to create a tool that will read a special type of file, offer a menu of options that can be executed, and then produce the new geometry and nodal network for subsequent thermal analysis. This tool is called the Generic Processor since it builds a model from a generic configuration of a particular class of models [1,2]. The file that contains the instructions is the GMD (Generic Model Definition) file and is partially illustrated in Figure 11. It should be noted that the GMD file originates from an experienced model developer who creates the baseline geometry with an independent CAD tool (not part of WinTherm) and constructs the initial nodal network with a thermal model editor (also not part of WinTherm). A key to the philosophy behind the Generic Models is that the connectivity graph for the vertices that defines the facets remains constant and the connectivity graph between the nodes in the model that defines the thermal conductances also remains constant regardless of the geometric parameters or materials specified. This feature allows new instances of models to be built by the casual user with simple menu selection rather than the traditional thermal model construction from scratch by an experienced heat transfer specialist.

CREATING THE THERMAL MODEL FILES - The GMD file contains the specifications for which components can be adjusted and the formulations for altering the various geometric and thermal entities based on the component changes. The geometry is stored as vertices defining surface facets. Transformation rules are set up to properly translate, rotate, and scale the facet geometry based on dimensional changes of the components. Nodal capacitances and conductances are formulated to allow recalculation based on dimensional, surface, or material values that were indirectly changed by the user.

The conductances associated with conduction heat transfer connect layers of the same element or elements which are in direct contact. The conductance is calculated from node material, and the parameterized area and thickness of the node. Convection conductances (forced or natural) connect nodes with exposed surfaces to surrounding fluids. The geometric portion of convection conductance can be precalculated from the parameterized node surface area.

The radiation conductances are most difficult to precompute since they depend on more complex quantities than simply the thickness or surface area of a single element. View factors which depend on the geometry of the surfaces relative to one another must be computed. These view factors are then used with the radiative surface properties to compute the required radiation exchange factor that is contained in the radiation conductance. Two view factor calculation options are contained in the Generic Processor that will be further discussed in the next section.

VIEW FACTOR CALCULATIONS - Radiation heat transfer calculations require view factors, F_{ij}, for all surface nodes (or facets). Heat transfer books contain formulations and charts for obtaining view factors for simple cases. These values are then incorporated in the radiation conductance terms, C_{ij}. The Generic Processor will allow imbedding these analytical formulas into the GMD file for calculation of the view factors. It is done by predetermining the connectivity graph and then formulating the required view factor equations with adjustable parameters that are based on geometry changes. This option is sufficient for small models with simple geometry. A more general purpose methodology option is described next.

Since the geometry of most models is too complex for computing view factors analytically, the geometry model VECTOR SWEEP [3] is utilized to numerically calculate all radiation view factors (accounting for mutual shadowing) based on the new geometry facet file. The view factor F_{ij} is determined by numerically integrating the following equation

$$A_i F_{ij} = \int_{A_i} \int_{A_j} \frac{\cos\theta_i \cos\theta_j \, dA_i \, dA_j}{\pi r_{ij}^2} \qquad \textbf{(1)}$$

where A, θ, and r are shown in Figure 12.

DIFFUSE RADIANT EXCHANGE - The rate at which radiant energy leaves a surface per unit area and time is its radiosity, J. The object's radiosity is composed of two components: self emission and reflection. The self emission quantity is obtained from its emissivity, ε, and its blackbody emissive power, E_b. The reflection quantity is obtained from its reflectivity, ρ, and incident irradiation, G. Total radiosity for each surface node is expressed by:

$$J_i = e_i E_{bi} + r_i G_i \qquad \textbf{(2)}$$

The blackbody emissive power can be calculated with the Stefan-Boltzmann equation,

$$E_b = sT^4 \qquad \textbf{(3)}$$

```
C ****************************************************************************
C * Definition of surfaces, materials, substructures and node areas.         *
C ****************************************************************************
C_NO-SURF 5     ! Number of surface treatments allowed in model.
Ca Steel - Clean
Cb 0.100 0.540  #2
Ca Steel - Oxidized
Cb 0.800 0.740  #3
Ca Carpet
Cb 0.940 0.800  #4
{break}
C_NO-MTRL 7     ! Number of bulk thermal material properties allowed in model.
Ca Steel
Cb 3642345 54.0         #1
Ca Insulation
Cb 22720 0.045          #2
{break}
C_NO-SS 9       ! Number of substructures.
Ca Exhaust Pipe
Cb 4                    ! Number of nodes in substructure.
Cc 25 26 27 28          ! Substructure node numbers.
Cd 1 3                  ! Default surface treatment & number of permitted treatments.
Ce 1 2 3                ! Permitted surface treatments.
Cf 1 1                  ! Default bulk material & number of permitted bulk materials.
Cg 1                    ! Permitted bulk materials.
{break}
C ****************************************************************************
C * Set up the transformation rules. Translate, rotate, and scale.            *
C * APPLY the above transformation rules to the proper facets.                *
C ****************************************************************************
C_Scale 1, &a1 1.8288 /, 1.0, 1.0, 0.0, 0.0, 0.0
C_Xlate 9,  0.0, 0.0, &a12 &a13 > &a14 > &a15 > -0.1143 + -2 /
C_Apply 1,  1,   48,  -9999, 9999, -9999, 9999, -1, 9999  ! Scale Test Box length
{break}
C ****************************************************************************
C * This is the section for describing the Nodal Capacitance,                 *
C * Conductors, and other required entites of the Thermal Model.              *
C ****************************************************************************
C ***************** Number of nodes, conductors, etc. *****************
C_p  123    8   50    0    0  NN  NCT  IRCNT NZS   NQCRV
C_p   25   0.00   4 exhaust_pipe_inlet_#1
C_p   26   0.00   5 exhaust_pipe_#2
C_p   51   0.00   3 air
{break}
C *********** Capacitance Section *************
C_p    1    1;       97.9446 &a5 0.003175 / *;  0 !floor_covering_1
C_p    2    2;       99.3801 &a5 0.003175 / *;  0 !floor_covering_2
C_p   52   -29 0;    513.2562 &a4 0.4064 / *;   0 !Bottom_Air
C_p   53   -38 0;   1018.4927 &a3 0.8128 / *;   0 !cabin_air
C_p   55   -1 4;     97.9446 &a6 0.003175 / *;  0 !floor_covering_1(1-1of5)
C_p   56   -1 5;     97.9446 &a7 0.003175 / *;  0 !floor_covering_1(1-2of5)
{break}
C ******************* Conductance Section ******************
C_p    1    1; 1.7628 &a5 0.003175 / /; 55 -1 4; 3.5256 &a6 0.003175 / /;  0 (1.1757) floor_covering_1
C_p    1 -1 0; 0.0887;                 53  0 0; 0.0000;                   103 (0.0887)
C_p    1    1; 0.0002 &a1 1.8288 / / &a1 1.8288 / / &a5 0.003175 / *;    2    2; 0.0002 &a1 1.8288 / / &a1
1.8288 / / &a5 0.003175 / *;  0 (       0.0001)
C_p    1    1; 0.0009 &a2 0.6096 / / &a2 0.6096 / / &a5 0.003175 / *;    5    5; 0.0009 &a2 0.6096 / / &a2
0.6096 / / &a5 0.003175 / *;  0 (       0.0005)
C_p    2    2; 1.7886 &a5 0.003175 / /; 60 -2 4; 3.5773 &a6 0.003175 / /;  0 (1.1921) floor_covering_2
{break}
C ******************* Forced convection cards ***************
C_p  7; 0.0; 0.0; &a1 &a2 * &a4 * 0.3333333 ^; 0; Wind Convection
C_p -4; 0.0; 0.0; &a1 1.8288 / 1.8162 *; 0; Fan Convection
C_p -8; 0.0; 1177.; 0.0; 0; Fan Mass Flow Convection
```

Figure 11: The GMD File Contains All of the Baseline Geometry, Nodal Network Attributes, and Adjustment Rules for a Particular Generic Model

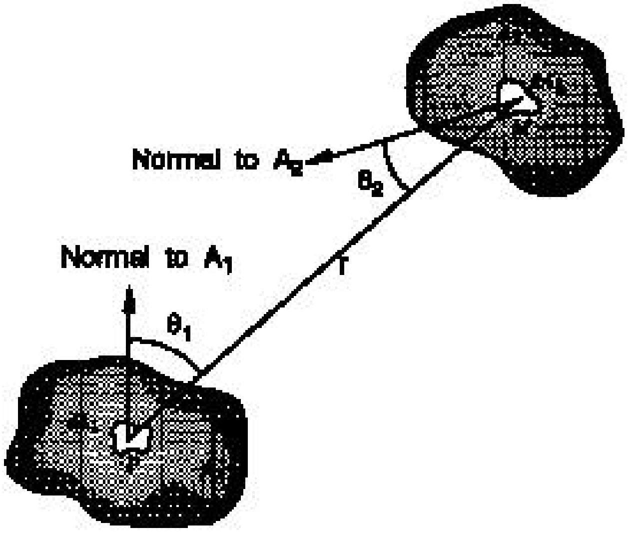


Figure 12: Geometrical Notation for View Factor Formulations (where i=1 and j=2).

Two methods of solution for the radiosity matrix could be considered. One method uses the following set of simultaneous equations for solution:

$$J_i = e_i E_{bi} + r_i \sum_{j=1}^{N} F_{ij} \; J_j \qquad \textbf{(4)}$$

This method requires a matrix solution and is not practical for large systems. A second method is used within the Generic Processor that efficiently computes greybody radiation exchange factors, from emissivity and view factors. The theoretical discussion of the methodology of radiation exchange factor calculations can be obtained in reference [4]. With these exchange factors only the simple arithmetic equation is required:

$$J_i = e_i \; E_{bi} + \frac{1-e_i}{e} \sum_{j=1}^{N} —_{ij} \; E_{bj} \qquad \textbf{(5)}$$

THEORY OF THE THERMAL MODEL

The Thermal Model code can be viewed as a thermal network solver in which the network is a collection of interconnected nodes [5]. The nodes are characterized by thermal properties such as heat capacity, temperature, etc. and are interconnected by heat flow paths characterized by heat conductance (via solid conduction, fluid convection, or thermal radiation) between two temperature potentials. The Thermal Model is based on the solution to the basic energy balance for a volume element that is simply stated as

heat in - heat out = heat stored **(6)**

The differential form of this generalized statement is

$$\frac{d}{dx}\left(k_x \frac{dT}{dx}\right) + \frac{d}{dy}\left(k_y \frac{dT}{dy}\right) + \frac{d}{dz}\left(k_z \frac{dT}{dz}\right) + Q_v = r\, C_p \left(\frac{dT}{dt}\right) \qquad \textbf{(7)}$$

where the net heat in and out in each direction is represented by the differential conductive heat rates, $\delta/\delta x(k_x \delta T/\delta x)$ etc.; the internal heat source is Q_v; and the heat storage term is $\rho C_p(\delta T/\delta t)$. When the appropriate boundary conditions are added, the temperature distribution and thus the heat rates can be calculated. These boundary conditions include radiation, convection, heat fluxes, contact interfaces, phase change, etc.

The solution to the steady state or transient energy equation is achieved by representing the differential volume elements by finite nodal quantities with isotropic properties and isothermal temperatures. The steady state numerical solution is generalized to any node *i* connected to *N* other nodes:

$$T_i = \frac{\sum_{j\neq i}^{N} C_{ij} T_j + Q_i}{\sum_{j\neq i}^{N} C_{ij}} \qquad \textbf{(8)}$$

Since the unknown temperature of i is dependent on the unknown temperatures *j*, either a matrix solution or iterative solution is required. For small models without nonlinear conductances C_{ij}, such as radiation conductors or temperature dependent conductors, the matrix solution is ideal. General purpose requirements dictate that the iterative solution is used. This method is called the Gauss-Seidel procedure. Convergence is achieved when the greatest nodal temperature difference between the present iteration and the previous iteration is less than a specified amount.

The conductance term, C_{ij}, of Eq. (8) is used for all modes of heat transfer and will have units of W/ K for the SI system. C_{ij} is evaluated for the particular mode of heat transfer linkage.

$$C_{ij} = \frac{k\,A}{L} \qquad \textit{conduction} \;\; \textbf{(9)}$$

$$C_{ij} = h\,A_i \qquad \textit{convection} \;\; \textbf{(10)}$$

$$C_{ij} = s\;—_{ij}\,A_i\,(T_i^{\,2} + T_j^{\,2})(T_i + T_j) \qquad \textit{radiation} \;\; \textbf{(11)}$$

$$C_{ij} = \dot{m}\,C_p \qquad \textit{fluid flow} \;\; \textbf{(12)}$$

Note: The radiation conductance uses the previous iteration or time step value of temperature, T, for the new C_{ij} *value. The initial guess iterative temperature value is automatically set to air temperature unless modified by the user.*

3D TRANSIENT SOLUTION - Two methods of numerical transient solution are typically used in numerical heat transfer: an explicit technique or an implicit technique. The implicit transient solution is unconditionally stable since it uses the new temperatures *T* in its current time step temperature computation. It requires an iterative solution at every time step. The additional complexity in problem formulation and internal computations will still result in a faster solution with higher accuracy in most cases than the more simple explicit formulation.

The implicit difference expression is

$$\frac{CAP_i}{\Delta t}(T_i{}' - T_i) = \sum_{j \neq i}^{N} \left[C_{ij} \left(\frac{T_j{}' + T_j}{2} - \frac{T_i{}' - T_i}{2} \right) \right] + Q_i \qquad \textbf{(13)}$$

and solving for $T_I{}'$ results in

$$T_i{}' = \frac{\left[\sum_{j \neq i}^{N} C_{ij} T_j + \sum_{j \neq i}^{N} C_{ij} T_j{}' + 2Q_i - T_i \sum_{j \neq i}^{N} C_{ij} \left(1 - \frac{2}{STAB_i} \right) \right]}{\sum_{j \neq i}^{N} C_{ij} \left(1 + \frac{2}{STAB_i} \right)} \qquad \textbf{(14)}$$

where the quantity CAP_i is the thermal capacitance of node i:

$$CAP_i = r \; C_p \; V_i \qquad \textbf{(15)}$$

and $STAB_i$ is the stability constant of node i:

$$STAB_i = \frac{\Delta t}{CAP_i} \sum_{j \neq i}^{N} C_{ij} \qquad \textbf{(16)}$$

CONDUCTION BETWEEN FACETS - When a temperature gradient exists in a body, thermal energy is transferred from the high temperature region to the low temperature region. The rate at which this energy is transferred is a function of the material and the geometry. For the i^{th} node, conductivity to and from the adjoining nodes is derived from Fourier's equation or

$$Q_i = C_{i1}(T_1 - T_i) + C_{i2}(T_2 - T_i) + \ldots + C_{iN}(T_N - T_i) \qquad \textbf{(17)}$$

where N is the total number of nodes adjoining node i. The Conductance, C_{ij}, is the product of the thermal conductivity that is material dependent, and the cross-sectional area of the adjoining nodes, divided by the distance from the center of node i to the center of the node to node interface (i.e., Eq. 9).

When two dissimilar materials, 1 and 2, are in *good* thermal contact, C_{12} is given by

$$C_{12} = \frac{A_x}{\left(\frac{L_1}{k_1} + \frac{L_2}{k_2} \right)} \qquad \textbf{(18)}$$

where A_x is the cross-sectional area, k is the material thermal conductivity, and L is the distance from the node interface to the node center of mass [6].

CONVECTION MODEL - Newtonian cooling or convection is defined as

$$Q_c = h \, A_i (T_j - T_i) \qquad \textbf{(19)}$$

where T_j can be another internal node (typically fluid) or a boundary node (such as air). The calculation of the appropriate heat transfer coefficient, h, is the fundamental problem. Typically a set of dimensionless numbers are used to determine h such as the Nusselt Number:

$$Nu = \frac{h \, L_c}{k} \qquad \textbf{(20)}$$

<u>Forced Convection</u> - There are several types of forced convection that can be chosen in the thermal model based on geometry and flow conditions. An example of one type for *Flat Plate, Turbulent Flow, Average h (Re > 5x10*^{5}*)* is

$$h = \frac{0.036 \, Re^{0.8} \, Pr^{1/3} \, k}{L_c} \qquad \textbf{(21)}$$

where Re is Reynolds Number, Pr is Prandtl Number, and L_c is length of plate (m).

<u>Natural Convection</u> - In addition, different types of natural convection can be applied based on heat flow direction and geometry. One example is *Vertical Plate or Cylinder*

$$h = \frac{e \, k \, (Pr \;\; Gr)^f}{L_c} \qquad \textbf{(22)}$$

where Gr is Grashof Number, L_c is plate or cylinder height (m), and e,f are constants with values that depend on the product $PrGr$.

An overview of the Thermal Model capabilities is summarized in Table 1.

Table 1: Features of the Thermal Model

FEATURES	COMMENTS
THERMAL MODEL EXPRESSED WITH 3-D FINITE DIFFERENCE EQS.	*Provides Heat Transfer Linkages in any Direction*
NUMERICAL F.D.E. METHODOLOGY FOR TRANSIENT SOLUTION	*An Implicit Method Provides Optimum Numerical Solution*
INITIALIZATION SCHEME FOR TRANSIENT SOLUTION	*Initiate Simulation with a Calc. of Steady State*
FORCED AND NATURAL CONVECTION MODEL	*Convection Coefficients for Varied Applications*
MASS TRANSFER (EVAP., CONDEN., AND PRECIP.)	*Environmental Effects are Correctly Considered*
EXTENDED CONVECTION MODELS: (AERODYNAMIC AND FLUID FLOW)	*Fluid Flow is a Primary Interaction in Heat Management Analysis*
SHADOWING AND MULTIPLE BOUNCE RADIATIVE EXCHANGE	*Obscurations and Multiple Bounce Reflections are Correctly Included*

THEORY OF THE IMAGE VIEWER

When the Thermal Model calculates the temperatures of the nodes (or facets) for each designated time step, it records those temperatures (or radiances) into a data file. A visualization program is required to read the data file and show the temperatures of the object graphically for every time step. In the Image Viewer program [7], temperature is represented by a gray scale (0 - 255). The user can visualize the spatial temperature map over time (transient) with the gray scale (shade) correlated to a temperature (black is coldest and white is hottest).

The Image Viewer is divided into two sections: the 3-D graphics library and the WinImage module. The 3-D graphics library was written in-house and uses a special structure to represent 3-D objects. The 3-D graphics library is general purpose and environment independent. The WinImage module is dependent on Windows API (Application Programming Interface) functions.

3-D GRAPHICS LIBRARY - This library provides the required 3-D capability (i.e., Microsoft Windows^{TM} provides only 2-D graphics primitives). The 3-D graphics library was designed for portability and speed. Abstract data structures were used along with object-oriented programming written in C++.

The 3-D graphics library is divided into several modules (Table 2) for easy maintenance and general use. The operations and data are clearly defined for each module. The data structure of each module was hidden as much as possible, although direct access to other data structures is also used for maximum performance. Since an abstract data type is used in this library, changes in the data structure within a module will not affect other modules.

Table 2: The Modules in the 3-D Graphics Library

3Dmatrix	Defines 4x4 matrix and provides the basic matrix operations such as addition, subtraction, multiplication, and inverse.
3DVector	Defines 3-D vector and provides the vector operations such as addition, subtraction, dot and cross product, and length.
3DTrform	Provides the basic homogeneous transformation operations such as translations and rotations. A homogeneous transformation computes translations in multiplication form.
3DObject	Defines 3-D object, provides the functions to store the 3-D object in a hierarchical structure, and retrieves the object data.
3DView	Defines viewpoint in 3-D space and provides the functions to manipulate the viewpoint.
Show3D	Provides functions such as perspective projection and hidden line/surface removal.
WinLib	Environment intensive functions such as memory allocation, screen output, and file input/output.

WINIMAGE MODULES - This is the image generation program that displays the wireframe geometry and the gray scale thermographic history (3-D thermal image) of the model. It uses the 3-D graphics library and is also designed with abstract data types for easy upgrading. The individual modules are listed in Table 3.

Table 3: The Modules in WinImage

WPFile	Provides file services for WinImage.
WPWindow	Provides window creations and destructions for WinImage.
WPView	Reads facet file and displays the object with gray scale shading.
WinImage	Main program module.

HIDDEN SURFACE REMOVAL ALGORITHM - Creating realistic images of 3-D objects requires removal of hidden lines and surfaces. Hidden-surface removal algorithms typically require significant processing power and time. Existing algorithms can be broadly divided into two categories: the image-precision (or image-oriented) algorithm and the object-precision (or object-oriented) algorithm.

The image-precision algorithm works on a 2-D image to determine object visibility at each pixel in the frame. These algorithms require large memory, long processing time, and produce noticeable imperfections when enlargening the image. Commonly used image-precision algorithms are the Z-buffer algorithm, ray tracing algorithm, and scan-line algorithm.

The object-precision algorithm operates on the 3-D data directly to determine visible polygons. These algorithms are very fast, free from distortion, and require less memory. Object-precision algorithms however, do not always produce correct images and therefore need special consideration for overlapping polygons. Commonly used object-precision algorithms are the painter algorithm, back-face culling algorithm, and depth-sort algorithm.

To achieve optimum performance, the approach used in the Image Viewer is a hybrid of the image-precision and object-precision algorithms. The Z-buffer algorithm and the back-face culling algorithm have been used with some modifications. First the back-face culling, which is an object-precision algorithm, is used to remove those polygons invisible to the viewer. The remaining visible polygons are then stored in a list. Typically, about half the total polygons are visible. The image-precision algorithm is then applied to polygons on the list. Traditionally, the image-precision algorithm starts at the first pixel and proceeds to the last pixel in the frame. This procedure is modified to operate on polygons directly rather than pixels. The modified algorithm starts from the first polygon and works to the last polygon on the list. For each polygon, the smallest corresponding rectangular region occupied by the polygon is found (i.e., the extent of each polygon is calculated). Then the image-precision algorithm is applied in that small rectangular region. This modified algorithm is quite similar to the Z-buffer algorithm with the primary difference being that a traditional Z-buffer works on the entire image or frame. A large model with 1800 polygons takes about 23 seconds to display on a 25MHz 486 computer.

The pseudo code in Figure 13 describes the methodology for this hybrid hidden surface removal algorithm.

```
Initialize the distance array and image array;
Calculate the perspective projection for all vertices, and store the 2-D coordinates after projection;
Calculate the surface equation of all polygons for distance calculation;
for all polygons in the object do
            begin
            if the polygon is not facing away
                  insert it into list;
            end
for each polygon in list do
            begin
            get the projected rectangle region for this polygon;
            for each pixel in the region do
                  begin
                  calculate the distance from polygon to viewer;
                  if calculated distance less than the distance in distance array
                        begin
                        replace the distance in distance array with this calculated distance;
                        replace the index in image array with the index of this polygon;
                        end
                  end
            end
```

Figure 13: The Hybrid Hidden Surface Removal Algorithm Used in the Image Viewer

ANALYTICAL RESULTS FOR THE GENERIC TESTBOX MODEL

A validation of WinTherm was made by constructing a generic thermal model of a testbox apparatus used by Lydall Westex of Hamptonville, NC. The model was designed to simulate geometric and material variations of the Lydall Testbox.

This test configuration thermally models the undercarriage and cab compartment of an operating vehicle. The testbox is vertically divided by a floor pan, one third of the distance above its base. Insulating materials, including various forms of Lytherm^{TM} (Lydall's insulating material), typical underlayment (shoddy), and carpeting, may be arranged above the floor pan. The floor pan, made of galvanized steel, is fixed in place above a stainless steel pipe, which spans the length of the testbox. This pipe is heated at one end to simulate the exhaust system of a vehicle. The compartment which houses the pipe is equipped with fans which force air through the lower section of the testbox at a fixed velocity.

It should be noted that although the geometry and types of materials used in the Lydall testbox are mostly fixed, the Generic Processor allows the physical and material parameters previously mentioned to be varied by the user with a great deal of flexibility.

To test the accuracy of the thermal model under a variety of different inputs, Lydall devised several tests in which the types and amounts of insulation above the floor pan were varied. The side of the floor pan facing the exhaust pipe was arranged with 55 evenly spaced thermocouples. This type of arrangement was repeated on the layers of insulation above the floor pan, and temperature measurements were taken every five minutes.

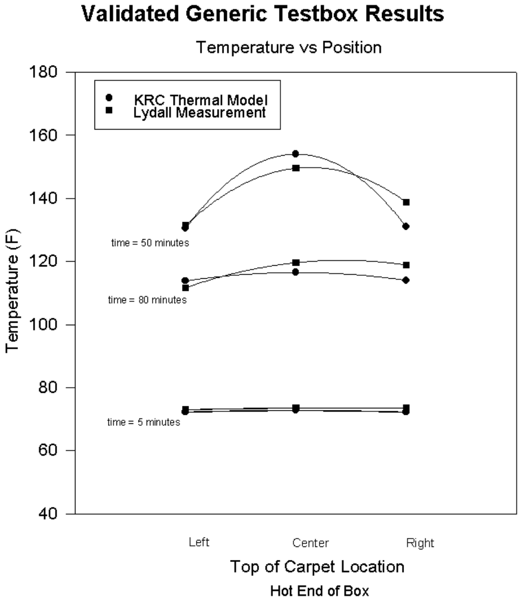


Figure 14: Validation of the Testbox Model

The validation procedure for the thermal model involved comparing measured and calculated data. The results in Figure 14 illustrate the accuracy of the testbox thermal model, as compared with the Lydall test case. This plot depicts a comparison of measured versus calculated temperature gradients across the width of the testbox, at different time intervals. Thermal input parameters for the Generic Testbox Model were selected that best approximate the configuration of the physical testbox. Once parameters were chosen for the other configurations, favorable comparisons such as the one in Figure 14 were obtained for each test.

SUMMARY

The thermal modeling package *WinTherm* is a comprehensive design and analysis toolkit that emphasizes ease of use and is based on rigorous computational heat transfer principles. This integral desktop modeling tool is composed of three primary modules: the *Generic Processo*r, the *Thermal Model*, and the *Image Viewer*.

The Generic Processor facilitates thermal/geometry model construction in a fast and orderly procedure. The heat transfer information, geometric relationships, and material properties are contained within the Generic Model Definition (GMD) file. A parameterized tradeoff study can be done significantly faster with this tool than with traditional methodologies. Custom designed *Generic Models* can be developed for virtually any of the heat management areas of the automobile such as underhood (engine compartment), underbody (exhaust components), interior cab (climate control), electronic/battery systems, engine block, etc.

The 3-D thermal model is a first principles nodal network solver that can be used in a variety of transient heat transfer problems. It includes sophisticated radiation, conduction, and conduction modules; flexible boundary conditions including environmental models; and detailed thermal output for a variety of user applications.

The Image Viewer provides a wireframe rotational display of the 3-D geometry and thermal display in an animated (time) mode for quick inspection of the thermal performance results.

To ensure that new design configurations are being accurately represented both by the physical equations in the Thermal Model and the modeling assumptions in the Generic Processor, a validation was performed with an actual testbox that directly compared measured data to the analytical results.

Future development will include an Expert System front-end for optimized design based on inputs of performance specification. A related program will include a Generic Engine Model that provides heat transfer and thermodynamic conditions of the engine based on limited user inputs.

NOMENCLATURE

Symbols

A	area, m^2
C_{ij}	conductance, W/K
C_p	specific heat, J/kg-K
L	length, m
E_b	blackbody emissive power, W/m^2
F_{ij}	view factor from surface i to surface j
$\mathscr{F}_{ij}$	radiation exchange factor from surface i to surface j
G	irradiation, W/m^2
h	convective coefficient, W/m^{2-}K
i,j	surface or nodal indices
J	radiosity, W/m^2
k	thermal conductivity, W/m-K
m	mass, kg
m	mass flow rate, kg/s
Q	heat flow, W
T	temperature, K
r	distance between emitter and receiver, m
t	time, s
Δt	time step, s
V	volume, m^3
ε	emissivity
θ	angle of incidence, degrees
ρ	mass density of node, kg/m^3, or
ρ	reflectivity, as determined by its context
σ	Stefan-Boltzmann constant, 5.668x10^{-8} W/m^{2}K^4

other variables defined in their application

REFERENCES

1. Rushmeier, Holly E. and Leonard J. Rodriguez, “Generic Models for Rapid Calculation of Target Signatures,” *Proceedings of the SPIE--the International Society for Optical Engineering*, Orlando, FL, April 1991.
2. Rodriguez, Leonard J. and Keith R. Johnson, “Menu Driven Construction of Generic Targets for the Infrared Tactical Decision Aid,” *Proceedings of the Eleventh Annual Symposium on Ground Vehicle Signatures*, Houghton, MI, August 1990.
3. O'Neill, Richard F. and Jon L. Zich, “Vector Sweep Engineering and Programmers Guide,” GDSS-SP-85-010, General Dynamics Space Systems Division, San Diego, CA, December 1985.
4. O'Neill, Richard F. and Jon L. Zich, “Computer Program Script-F,” Alcoa Defense Systems, San Diego, CA, November 1987.
5. Johnson, Keith R. and Leonard J. Rodriguez, “Technical Reference Guide for GTSIG/TCM2,” Georgia Tech Research Institute, Atlanta, GA, 1990.
6. Curran, Allen R., “User's Manual for PRISM 3.1,” Keweenaw Research Center, Michigan Technological University, Houghton, MI, 1993.
7. Yee, Ban K., “3-D Visualization of the Physically Reasonable Infrared Signature Model,” *Proceedings of the Fourth Annual Ground Target Modeling & Validation Conference*, Houghton, MI, August 1993.