Simulation of Thermal Images

Vijaya Priya Govindasamy

ECE 671

Spring 2003
Abstract

The threat of terrorist attacks has increased in the recent years. Therefore it has become important to inspect the underside of vehicles for special nuclear materials and explosives. Manual search for vehicle bomb detection is a time consuming process. A method has to be developed to reduce the time and increase the security. Since all objects emit radiations in the infrared band, there arises a necessary to simulate synthetic thermal images of real time data. It becomes easy for the inspector to compare the real time thermal data with the simulated data for explosives.

The objective of this project is to simulate infrared images using the infrared prediction software MuSES (Multi-service Electro-optic Signature). The project includes two steps: (a) Study and familiarize the software MuSES, and (b) Simulation of infrared images for objects of interest such as parts of under vehicle, human faces etc using MuSES.

MuSES is a cross-platform thermal modeling tool, which can be used to model the steady state and transient distribution of heat over complex surface descriptions of component systems. MuSES models 3-D conduction, convection and multi-bounce radiation. The output from MuSES is the temperature map of the component system which can be viewed using the integrated post-processor.

A sample model will be selected and its properties and characteristics will be altered using Muses. A complete infrared prediction will be carried out and the resulting output temperatures will be used in simulating infrared images of the sample. This study will be further extended for under vehicle inspection and other targets of inspection such as human faces or parts of ground vehicles etc.
Table of Contents

Abstract ........................................................................................................................................ i

1. Introduction ........................................................................................................................... 1
   1.1 Background ...................................................................................................................... 1
   1.2 Proposed Approach ......................................................................................................... 2

2. Theory and Methods .......................................................................................................... 2
   2.1 Thermal Imaging .............................................................................................................. 3
   2.2 MuSES (Multi-service Electro-Optic Signature) ............................................................ 4
      2.2.1 MuSES Graphical User Interface ............................................................................... 4
      2.2.2 Meshing in MuSES .................................................................................................. 10
   2.3 Rhino3d and Polytrans .................................................................................................... 15
   2.4 Simulation of Biomaterials ............................................................................................. 16

3. Experimental Results ........................................................................................................ 17
   3.1 Simulation of Exhaust System ......................................................................................... 18
   3.2 Simulation of Car Underbody System ........................................................................... 20
   3.3 Simulation of Laser scanned 3d Models ......................................................................... 24

4. Conclusions .......................................................................................................................... 26

References .................................................................................................................................. 27

Appendix A : Glossary .............................................................................................................. 27

Appendix B: File Types Supported By MuSES ...................................................................... 31
1. Introduction

Due to increasing terrorist attacks, it is necessary to detect nuclear and other harmful weapons. The weapons can be placed anywhere in the environment. Thermal image of the object under consideration is unique. Hence this project aims in simulating the thermal images of under vehicle parts and the simulation is extended for LASER scanned images. The software MuSES is used to simulate infrared thermal images. MuSES is a cross-platform tool used for modeling steady state and transient state thermal analysis of objects.

1.1 Background

Thermal images are unique for each and every object based on the infrared radiation emitted by the object. The object, which is not able to radiate energy in the visible spectrum, radiates in the infrared band. Thermal imagers are used to detect the infrared radiation and convert them to visual band to visually see the images. Thermal imagers are detector and lens combination, which detect the infrared radiation and convert the radiation to the visual band.

Ref [MuSES98] MuSES is a cross-platform thermal modeling tool used in Windows/UNIX computers to model the steady state and transient state thermal analysis of the object under consideration. MuSES models 3D conduction, convection and multi bounce radiation. The output of the system is a temperature map of the component system, which can be viewed using the post processor screen.

Need for simulation:

Effectiveness of thermal cameras depends on their operators. The three possible cases which results in poor results are

1) The spectral range of the thermal cameras differs from the spectral range of human eyes, and so the thermal image differs from the visual image of the same scenario.

2) The thermal cameras are not stereoscopic systems like human eyes and it is difficult to estimate the range to different object seen in the thermal image

3) There is no shades in the thermal images which provides us information about the source of illumination of the scene
Hence it is difficult to interpret thermal images manually. MuSES can simulate infrared prediction of the object in observation conditions specified by the user (the thermal properties of the materials, bounding conditions, etc.)

### 1.2 Proposed Approach

In MuSES, the thermal properties are assigned and the simulation of the thermal model is carried out. MuSES will provide a virtual prototyping platform for both concept and retrofit design efforts. Because of the strong connections between signature management for military ground vehicles and heat management for commercial automobiles, there is considerable dual-use collaboration and commercial commitments from the automobile industry. The approach is to design simple thermal models of under vehicle images and simulate the thermal images of the models created.

The section follows summarizes about the working of MuSES and other software used. The section following that discusses about the experimental results. Final section deals with the conclusion of the project and discusses the scope of improvement.

### 2. Theory and Methods

The MuSES interface is optimized for engineers who need to incorporate signature management treatments or heat management solutions into vehicle designs. It is necessary to study and analysis the heat distribution and radiation in complex component systems. All objects radiate energy in the infrared band and the most important feature of infrared energy is that it is unique for every object. The simulated thermal images of under vehicle parts of various vehicles like car, truck etc. can be simulated using MuSES and then it can be compared with real time data for any weapon detection. This helps in saving time and energy. Weather data file can be used to input thermal properties to the model. Hence it is easier to compare real time data of any date with the simulated model.

The output is seen using the post processor screen. The results can be exported as a PRISM file and the temperature can be plotted for a particular element.

The MuSES solution procedure is as follow as
1. Group geometry
2. Assign material properties
3. Set boundary conditions
4. Set solution parameters
5. Run simulation
6. Run Signature simulation (optional)
7. View results in Post process screen

Elements with similar properties are grouped into parts. In the resulting simulation, the thermal properties for each individual element can be seen.

Elements consist of one or more thermal nodes based on the part type assigned. The results can be analyzed for each node. For example, a three-layer part type has three thermal nodes named as front, middle and back layer.

The thermal simulation is also tried for other models like ship and human thermal models. The results of these models are discussed in Section 3.

2.1 Thermal Imaging

Thermal imaging is the quantification and recording of the surface temperature of objects by non-contact sampling and analysis of the infrared emissions from that surface. This data is compiled by a computer into a thermal image.

Thermal imaging is non-contact, emits no ionizing radiation, and as such, is non-invasive. Because it is non-contact, there is no pain in thermal imaging.

A Thermal image Scanner (or camera) measures infrared energy (heat) radiated from a body, accurately converting it to thousands of equivalent temperature readings.

On a thermogram, Whites, reds and yellows typically represent the warmer end of the temperature range being displayed. Blacks, blues and greens generally represent the cooler end of the same range.

A computer program assigns particular temperature ranges a particular color, and then displays the new colorized temperature readings as a color image, which is a thermogram or thermograph.

Infrared Spectrum
Infrared radiation lies between the visible and microwave portions of the electromagnetic spectrum. Infrared waves have wavelengths longer than visible and shorter than microwaves, and have frequencies, which are lower than visible and higher than microwaves. Infrared is broken into three categories: near, mid and far infrared. Near infrared refers to the part of the infrared spectrum that is closest to visible light and far infrared refers to the part that is closer to the microwave region. Mid-infrared is the region between the near and far infrared regions.

Heat is the energy that an object has because of the motion of its molecules - which are continuously jiggling and moving around. When energy is added to an object, its molecules move faster, creating more heat. Compared to a warm object, the molecules in a cold object have less molecular motion. Heat is the total energy of molecular motion in a substance.

Heat can be transferred from one place to another by three methods: conduction in solids, convection of fluids (liquids or gases), and radiation through anything that will allow radiation to pass. The method used to transfer heat is usually the one that is the most efficient.

In infrared astronomy, we measure heat, which has traveled by radiation. Infrared radiation (also called heat or thermal radiation) is a type electromagnetic radiation (or light). Radiation is a form of energy transport consisting of electromagnetic waves traveling at the speed of light. Temperature is a measure of the average heat or thermal energy of the molecules in a substance.

2.2 MuSES (Multi-service Electro-Optic Signature)

The MuSES interface is optimized for engineers who need to incorporate signature management treatments or heat management solutions into vehicle design. The software is a cross platform tool for both UNIX and WINDOWS. PRISM is an infrared signature prediction program with an inherent thermal analysis capability that is used predominately for military defense applications throughout the US and NATO countries.

2.2.1 MuSES Graphical User Interface

MuSES interface screen has three major areas.
1. The selection window
   This window allows to perform operations like creating and editing, building the thermal model, analyzing the results, running signature simulation and to view the results of simulation.

2. Graphics window
   This is where the model geometry and the results of simulation are viewed.

3. Status window
   The status windows keeps the commands executed while the modeling process is done.

**Selection Window**

The selection window has five major tabs and sub tabs for each major tab.

- **Geometry Tab**
  The geometry tab acts as the starting point for all thermal simulations. This tab has some sub tabs to create and edit the geometry. Processes that can be done from the geometry tab are

  1. Open an existing geometry file: The native file type of MuSES is .tdf (Thermal data file). MuSES requires either tri’s or quad’s. For elements with more than four vertices, MuSES reads only the first four and ignores any additional vertices. Appendix B lists the file formats that are supported by MuSES.

  2. Create new geometry
     MuSES provides two geometry primitives to create geometry and the created geometry will be positioned at the origin. MuSES does not allow any modification in the mesh once the geometry is created.
     The two primitives are
     Cylinder: The parameters required are length and diameter and also the number of elements along the length and diameter
3. Edit the geometry

The geometry editing functions that are available in MuSES are

i. Scale: Allows increasing or decreasing the size of the currently selected parts (s) or elements (s). The geometry can be scaled using a single scale factor or multiple scale factors.

ii. Translate: Allows moving the currently selected part(s) or elements(s) to a new location relative to the rest of the geometry. The geometry can be translated in any of the three directions X, Y, and Z. Translation connected option maintains connections to other parts by stretching the geometry.

iii. Rotate: Allows rotating the currently selected part(s) or elements(s). The geometry can be rotated in any degree (positive or negative) about any of the axes.

iv. Copy: Allows copying of the currently selected part(s) or elements(s). The copied geometry can be copied into the same location or can be offset from the original geometry by specifying the X, Y, and Z translation vectors.

v. Condense: Allows reducing two or more vertices into a single vertex. Since MuSES will calculate conduction between parts that share common vertices condensing is required. A maximum vertex separation value is used as the threshold for determining how far vertices can be apart that will be condensed into a single vertex. When the maximum vertex separation is greater than the shortest distance (0.01mm), a message box opens up saying that “the vertices cannot be condensed because elements will be condensed down to zero area”
vi. **Delete**: Allows deleting the currently selected part(s) or elements(s)

vii. **Diagnostics tab**: the check distance option allows checking the distance between elements. To use this option Element mode should be selected instead of part mode.

---

**Figure 1**: MuSES GUI in the Geometry tab mode

---

**Editor tab**

This tab is used to setup or modify the thermal properties of the model so that the thermal analysis can be performed.

The editor tab allows performing the following:

i. Grouping of elements with common thermal properties into parts

ii. Assigning of appropriate part type, properties, and boundary conditions to each part

iii. Specify other thermal properties including environmental conditions, curve types, material properties and surface properties
Analyze tab

The parameters for the thermal solution (e.g. steady state vs. transient state) are set and the simulation is run.

The analyze tab allows performing the following

i. Set the solution parameters (Start time and end time, tolerance, maximum number of iterations etc.)

ii. Run the simulation

iii. View convergence meter

iv. Analyze the results
Signature Tab

The parameters for the signature solution are set and the signature solution is run based on the results of the thermal solution. The signature solution is solved for the selection solution times. The available solution times are determined by the parameters that were set for the thermal solution. The signature solution is calculated for the selected radiance band.

Post processor tab

This tab is used for viewing the simulation and analyzing the results after the initial thermal solution is complete.

The Post Process tab allows performing the following

i. Animate the simulation

ii. View detailed thermal element/node and part information

iii. Plot the results

iv. Export the results to an ASCII file.
2.2.2 Meshing in MuSES

Meshing can be defined as the process of breaking up a physical domain into smaller sub-domains (elements) in order to facilitate the numerical solution of a partial differential equation.
Meshing can be used for a variety of applications. Finite element meshing is the basis for most of the applications. Surface domains are normally subdivided into triangle or quadrilateral shapes, whereas volumes are subdivided primarily into tetrahedra or hexahedra shapes. Automatic meshing algorithms ideally define the shape and distribution of the elements. The automatic mesh generation problem attempts to define a set of nodes and elements in order to best describe a geometric domain, subject to various element size and shape criteria. Geometry is most often composed of vertices, curves, surfaces and solids as described by a CAD or solids modeling package.

The process of creating a thermal mesh is a complicated task, which requires modeling expertise and advanced tools. The process of generating meshes is not a linear process.

Mesh generation starts with an existing geometry and produces a quality thermal model depending on the:

1. Geometry type (mesh or CAD source)
2. The modeler’s preferences
3. The condition of the source geometry
4. Special thermal modeling techniques
5. The desired final model resolution

**Mesher:**

It is the tool, which is used to generate meshes for the given geometry. Filling the surface creates the mesh.

The common factors in the meshing process are:

1) Import the mesh or CAD data into the mesher
2) Clean up the geometry so that edges and vertices line up (heal, repair, or create geometry)
3) Identify the hard (large angle feature) edges
4) Mesh using an algorithm that preserves the geometry as best possible while minimizing the element count and maintaining element quality.

**Examples of a good mesh**
MuSES requires either triangular or quadrilateral meshes, other polygonal meshes are not supported. The majority of meshed geometries found on the Internet that are output from CAD programs do not meet the requirements for being a good mesh for thermal analysis.

1. **Unconnected Elements:**

   In the figure shown below, the IC chip contains unconnected elements. In order for heat to conduct between elements, the edge that the polygons share in common must reference the same vertices.

![Figure 5: Unconnected parts in an IC chip](image)

In the example shown there will be no heat conduction between the pins and the case and the upper part, even though they seem to be attached. MuSES will consider them as disjoint parts and no conduction occurs between those parts. Consider the figure shown, the connections between the pins and case have been defined by subdividing the polygons constituting the case so that the pins and case share vertices. Now there is heat conduction between these elements. Still there won’t be any conduction between the
upper and lower parts of the pins; the lower polygon in the upper part of the pins would have to be redefined before such conduction can occur. Radiation is unaffected by this constraint - radiation will occur between unconnected facets as long as the facets have a view of each other.

MuSES conducts through edges. Conduction through faces, such as between the bottom of the box and the floor, is not included in the current release of MuSES. When this feature is implemented, the code will automatically connect parts through their common faces. If all edges of a polygon in each part share common vertices, MuSES will assume that the parts are in contact across that face. Thus the mesh of a box that sits on a floor should share common vertices with the mesh of the floor, the box and floor mesh should each include the common face/polygon, and the box and floor should be defined as separate parts.

2. Overlapping Elements

Another situation of no heat conduction arises when modelers add windows to buildings or vehicles. A common technique is to overlay the window facets onto the wall facets. Since the window facets are not embedded into the wall facets (i.e. they do not share common vertices) heat cannot conduct between the windows and the wall.
3. Unconnected Parts

One more type of mesh deficiency arises when the parts are meshed individually. In this example of a mesh of a tank the turret mesh is not connected to the hull mesh, the meshes for the hatches are not connected to the hull, and the gun parts are not connected to each other. No heat will be conducted between these elements.

Figure 8: Unconnected meshes in a tank
2.3 Rhino3d and Polytrans

MuSES has only two geometry primitives and so it is not easy to create complex surfaces using only those two primitives. Hence it is necessary to use other software to create the geometries and then import them into MuSES.

Rhino3d

This software is a 3d tool used for creating complex NURBS surfaces. 3D models, both simple and complicated, can be developed quickly and intuitively through its well-honed interface. The user operates within four windows: three windows show the model from the 3 cardinal directions while the fourth screen is a perspective view. Any of these views can be shaded. Once the NURBS geometry is created, an automated meshing algorithm creates a mesh that comes reasonably close to being suitable for MuSES. Rhino permits the import of NURBS surfaces created by CAD programs using the IGES file format.

Limitations in using Rhino3d

i. When an existing mesh is imported into Rhino, the triangles are converted into individual surfaces. It is hard to interpret the transformed NURBS surface.

ii. Multiplicity of the commands results in unexpected results.

Polytrans

Polytrans translates 3D geometry models, both in polygon mesh and NURBS formats. Polytrans reads in and converts the entire file contents so that textures, normals, animation, camera and light info, and comments are all fully translated. Objects (NURB surfaces or polygons) can be deleted or assigned to different parts. Simple 3D objects can be created and added. Objects from multiple files can be added together. Polytrans can polygonize or triangulate NURBS objects. When exporting polygon meshes to Wavefront OBJ format, the user can direct Polytrans to triangulate concave polygons and/or to triangulate any polygon with more than 4 sides.
2.4 Simulation of Biomaterials

Ref [Valvano] summarizes the ways of determining the thermal conductivity of the biomaterials. The paper discusses about the methods of calculating the bioheat properties of biomaterials. Ref [Kenneth] lists out in a table the thermal conductivity for various tissues.

The Pennes Equation

The fundamental equation of bioheat transfer, developed in 1948. Tissue is treated as a continuous material, which differs from inert engineering materials in that:

i. There is distributed generation of heat by metabolism

ii. There is distributed flow – perfusion – of blood throughout the tissue

\[
\rho \ c \ \frac{\partial T}{\partial t} = \nabla \cdot (k \nabla T) + \rho_b \omega_b c_b (T_{ar,i} - T) + q
\]

\(\rho\) → tissue density (kg/m\(^3\))

\(c\) → tissue specific heat (J.kg\(^{-1}\).oC\(^{-1}\))

\(T\) → tissue temperature (oC)

\(t\) → time (s)

\(k\) → tissue heat conductivity (W.m\(^{-1}\).oC\(^{-1}\))

\(\rho_b\) → blood density (kg/m\(^3\))

\(\omega_b\) → blood perfusion rate (s\(^{-1}\))

\(C_b\) → blood specific heat (J.kg\(^{-1}\).oC\(^{-1}\))

\(T_{ar,i}\) → arterial blood temperature inside the \(i\)th cylinder (oC)

\(q\) → metabolic heat production (W/m\(^3\))

The Human thermal model

Passive System: Described by equations resultant from the application of heat and mass balances to a tissue control volume.
Temperature control system: Responsible for the maintenance of the human body’s temperature.

The table shown below gives a list of thermal conductivity values for the tissues of biomaterials.

### THERMAL CONDUCTIVITY DATA FOR SPECIFIC TISSUES AND ORGANS

FOR HUMANS AND OTHER MAMMALIAN SPECIES

Professor Kenneth R. Holmes

<table>
<thead>
<tr>
<th>TISSUE</th>
<th>k (W/mK)</th>
<th>%H₂O</th>
<th>Reference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Kidney whole (rabbit)</td>
<td>.502</td>
<td>81</td>
<td>6</td>
</tr>
<tr>
<td>whole (rabbit)</td>
<td>.495</td>
<td></td>
<td>16</td>
</tr>
<tr>
<td>whole (human)</td>
<td>.543</td>
<td>84</td>
<td>4</td>
</tr>
<tr>
<td>cortex (rabbit)</td>
<td>.465 - .490 (n=7)</td>
<td>76.6 - 79.8</td>
<td>1</td>
</tr>
<tr>
<td>cortex (dog)</td>
<td>.491</td>
<td></td>
<td>16</td>
</tr>
<tr>
<td>cortex (human)</td>
<td>.499</td>
<td></td>
<td>16</td>
</tr>
<tr>
<td>medulla (rabbit)</td>
<td>.502 - .544 (n=7)</td>
<td>82.0 - 86.0</td>
<td>1</td>
</tr>
<tr>
<td>medulla (dog)</td>
<td>.507</td>
<td></td>
<td>16</td>
</tr>
<tr>
<td>medulla (human)</td>
<td>.499</td>
<td></td>
<td>16</td>
</tr>
<tr>
<td>Aorta human</td>
<td>.476 ± .041 (SD) (n=12)</td>
<td></td>
<td>2</td>
</tr>
<tr>
<td>Arterial plaque</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>fatty</td>
<td>.484 ± .044 (SD) (n=13)</td>
<td></td>
<td>2</td>
</tr>
<tr>
<td>fibrous</td>
<td>.485 ± .022 (SD) (n=12)</td>
<td></td>
<td>2</td>
</tr>
</tbody>
</table>

Table 1: Thermal properties of Biomaterials

### 3. Experimental Results

The thermal models simulated, were created using Rhino3D and then imported into MuSES. The thermal properties were assigned and the simulation was run. The simulation duration is set for each type of model based on the nature of the model. The results of simulation and the properties and the duration of simulation are reported below. The original models are shown along with the results.
3.1 Simulation of Exhaust System

The exhaust system model was created using Rhino3d and the temperature curve for the model was set. The material properties and the bounding box parameters were set in the model builder and then the simulation was run for a period of 30 minutes. The properties assigned are listed in a separate table. The model developed as the following parts:

1. Floor pan
2. Inlet pipe
3. Muffler
4. Manifold
5. Outlet pipe

Exhaust System Model

Figure 9: Exhaust System Model
The Properties Table:

<table>
<thead>
<tr>
<th>Properties</th>
<th>Floor Pan</th>
<th>Inlet Pipe</th>
<th>Muffler</th>
<th>Manifold</th>
<th>Outlet Pipe</th>
</tr>
</thead>
<tbody>
<tr>
<td>Part type</td>
<td>Calculated Standard</td>
<td>Assigned</td>
<td>Assigned</td>
<td>Calculated Standard</td>
<td>Assigned</td>
</tr>
<tr>
<td>Material</td>
<td>Steel (Mild)</td>
<td>-</td>
<td>-</td>
<td>Steel (Mild)</td>
<td>-</td>
</tr>
<tr>
<td>Thickness</td>
<td>1.5</td>
<td>1.5</td>
<td>1.5</td>
<td>1.5</td>
<td>1.5</td>
</tr>
<tr>
<td>Surface condition</td>
<td>Steel as Rcvd,0.74</td>
<td>Steel as Rcvd,0.74</td>
<td>Steel as Rcvd,0.74</td>
<td>Steel as Rcvd,0.74</td>
<td>Steel as Rcvd,0.74</td>
</tr>
<tr>
<td>Temperature</td>
<td>20 degrees</td>
<td>Inlet curve 250 degrees</td>
<td>Muffler curve 225 degrees</td>
<td>20 degrees</td>
<td>Outlet curve 200 degrees</td>
</tr>
<tr>
<td>Convection coefficient</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>Fluid flow temp</td>
<td>20</td>
<td>Ambient air</td>
<td>Ambient air</td>
<td>20</td>
<td>Ambient air</td>
</tr>
</tbody>
</table>

Table 2: Thermal properties assigned for the Exhaust System Model

The simulation result is shown in the figure below. At the end of 30 minutes, the thermal analysis is done and the simulated thermal model output is given. Clicking on the element in the post process screen, the element properties can be analyzed. The results can be exported or plotted and also can be animated. Here the simulation result is found to be exact according to the temperature curve.
3.2 Simulation of Car Underbody System

The model is created using Rhino3D and the various elements are grouped into parts as shown below.

Parts included in the model:
1. Engine Block
2. Radiator
3. Air filter  
4. Intake and Exhaust Manifolds  
5. Electric Motor  
6. Catalytic converter  
7. Pipes connecting manifolds, converter and muffler  
8. Muffler  
9. Batteries  

The thermal properties are assigned to the parts and the simulation is run for a period of 20 minutes. The temperature values are given in the form of a curve. The original model is shown in the figure below. The properties set are listed in the table below.

Figure 11: Model of Car Underbody System
The Properties table:

<table>
<thead>
<tr>
<th>Properties</th>
<th>Part type</th>
<th>Material</th>
<th>Thick -ness</th>
<th>Surface condition</th>
<th>Temperature</th>
<th>Convection coefficient</th>
<th>Fluid flow temp</th>
</tr>
</thead>
<tbody>
<tr>
<td>Air filter</td>
<td>Standard</td>
<td>Frame</td>
<td>1.5 mm</td>
<td>Steel(Rcv d) 0.74</td>
<td>20 degree</td>
<td>0</td>
<td>20 degree</td>
</tr>
<tr>
<td>Air filter hose</td>
<td>Standard</td>
<td>Frame</td>
<td>1.5 mm</td>
<td>Steel(Rcv d) 0.74</td>
<td>20 degree</td>
<td>0</td>
<td>20 degree</td>
</tr>
<tr>
<td>Engine block</td>
<td>Assigned</td>
<td>-</td>
<td>1.5 mm</td>
<td>Steel(Rcv d) 0.74</td>
<td>93.33 degree</td>
<td>0</td>
<td>Ambient air</td>
</tr>
<tr>
<td>Intake manifold</td>
<td>Standard</td>
<td>Frame</td>
<td>1.5 mm</td>
<td>Steel(Rcv d) 0.74</td>
<td>20 degree</td>
<td>0</td>
<td>20 degree</td>
</tr>
<tr>
<td>Exhaust manifold</td>
<td>Standard</td>
<td>Frame</td>
<td>1.5 mm</td>
<td>Steel(Rcv d) 0.74</td>
<td>20 degree</td>
<td>0</td>
<td>20 degree</td>
</tr>
<tr>
<td>Inverter</td>
<td>Assigned</td>
<td>-</td>
<td>1.5 mm</td>
<td>Steel(Rcv d) 0.74</td>
<td>65.55 degree</td>
<td>0</td>
<td>Ambient air</td>
</tr>
<tr>
<td>Radiator</td>
<td>Assigned</td>
<td>-</td>
<td>1.5 mm</td>
<td>Steel(Rcv d) 0.74</td>
<td>37.77 degree</td>
<td>0</td>
<td>Ambient air</td>
</tr>
<tr>
<td>Catalytic converter</td>
<td>Assigned</td>
<td>-</td>
<td>1.5 mm</td>
<td>Steel(Rcv d) 0.74</td>
<td>37.77 degree</td>
<td>0</td>
<td>Ambient air</td>
</tr>
<tr>
<td>Pipe connecting CC and manifold</td>
<td>Standard</td>
<td>Frame</td>
<td>1.5 mm</td>
<td>Steel(Rcv d) 0.74</td>
<td>20 degree</td>
<td>0</td>
<td>20 degree</td>
</tr>
<tr>
<td>Pipe connecting CC and muffler</td>
<td>Standard</td>
<td>Frame</td>
<td>1.5 mm</td>
<td>Steel(Rcv d) 0.74</td>
<td>20 degree</td>
<td>0</td>
<td>20 degree</td>
</tr>
<tr>
<td>Batteries</td>
<td>Assigned</td>
<td>-</td>
<td>1.5 mm</td>
<td>Steel(Rcv d) 0.74</td>
<td>21.111 degree</td>
<td>0</td>
<td>Ambient air</td>
</tr>
<tr>
<td>Muffler</td>
<td>Assigned</td>
<td>-</td>
<td>1.5 mm</td>
<td>Steel(Rcv d) 0.74</td>
<td>65.55 degree</td>
<td>0</td>
<td>Ambient air</td>
</tr>
<tr>
<td>Floor pan</td>
<td>Standard</td>
<td>Frame</td>
<td>1.5 mm</td>
<td>Steel(Rcv d) 0.74</td>
<td>20 degree</td>
<td>0</td>
<td>20 degree</td>
</tr>
<tr>
<td>Transmission</td>
<td>Standard</td>
<td>Steel mild</td>
<td>1.5 mm</td>
<td>Steel(Rcv d) 0.74</td>
<td>20 degree</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>Motor</td>
<td>Assigned</td>
<td>-</td>
<td>1.5 mm</td>
<td>Steel(Rcv d) 0.74</td>
<td>65.55 degree</td>
<td>0</td>
<td>Ambient air</td>
</tr>
</tbody>
</table>

Table 3 Thermal properties assigned for the Car Underbody System Model
The simulated model shows the radiation and conduction effects. The element properties are analyzed in the post processor. The engine block is at the high temperature and the other parts are heated accordingly based on the radiation and conduction effects.

Simulated Thermal Model

Figure 12: Simulated Result of Car Underbody System
3.3 Simulation of Laser scanned 3d Models

The thermal simulation is tried for scanned and reconstructed real data. The model shown here consists of the Waterneck model. The Waterneck is a part of the cooling system. The model consists of the following parts

1. Radiator
2. Engine block
3. Waterneck
4. Pipe connecting the Waterneck and radiator

The model consists of the Waterneck connected in between the radiator and the engine block. The thermal analysis is done and the result is analyzed for the conduction effects in the Waterneck. The figure below shows the original geometry used for simulation of thermal model.

![Waterneck Model System Model](image)

Figure 13: Model of a Waterneck System
The simulation is run for a period of 10 minutes and the engine block and the radiator are assigned the maximum temperature and the Waterneck is at the ambient temperature.

**The Properties Table:**

<table>
<thead>
<tr>
<th>Parts</th>
<th>Part type</th>
<th>Temperature</th>
<th>Thickness</th>
<th>Material</th>
<th>Fluid flow temp</th>
</tr>
</thead>
<tbody>
<tr>
<td>Engine block</td>
<td>Assigned</td>
<td>200 F</td>
<td>1.5 mm</td>
<td>-</td>
<td>Ambient air</td>
</tr>
<tr>
<td>Radiator</td>
<td>Assigned</td>
<td>100 F</td>
<td>1.5 mm</td>
<td>-</td>
<td>Ambient air</td>
</tr>
<tr>
<td>Waterneck</td>
<td>Standard</td>
<td>68 F</td>
<td>1.5 mm</td>
<td>Frame</td>
<td>68 F</td>
</tr>
<tr>
<td>Pipe</td>
<td>Standard</td>
<td>68 F</td>
<td>1.5 mm</td>
<td>Frame</td>
<td>68 F</td>
</tr>
</tbody>
</table>

Table 4: Thermal properties assigned for the Waterneck System Model

The simulated result is shown in the figure below. The conduction of heat from the engine block to the Waterneck can be clearly seen. The result confirms that efficient geometry models of real data can be used for simulation of thermal models.
4. Conclusions

The models simulated in this project are relatively simpler. The simulation results were found to be exact for some models and in case of other models they still need to be fine tuned to get the exact simulation result.

The limitations for the simulation are meshing, and the time of simulation. Normally it takes more time to simulate the result.

The simulation process can be extended for complex surfaces and also for human models.
References


MuSES manual Thermoanalytics INC.  [http://www.thermoanalytics.com](http://www.thermoanalytics.com)


Professor Kenneth R. Holmes (paper). “Thermal Properties of Biomaterials”.

Appendix A : Glossary

This glossary defines some of the terms commonly used by MuSES.

**Assigned Temperature Part** - A part that has a constant (known) temperature. The temperature can be a fixed value or can be described by a time dependent curve.

**Back Side** - The side of an element that does not have the surface normal directed out of it.

**Bounding Box** - MuSES uses a box to simulate the environment of the model. This box bounds the geometry with an offset that is specified by the user. The walls of the box are assumed to be constant temperature with an emissivity of 1.0 (blackbody). The user can specify individual temperatures for each box face.

**Boundary Conditions** - Boundary conditions are known conditions that can be applied to a part. Assigned Temperature parts allow the specification of a temperature boundary condition. Calculated Temperature parts allow the specification of an imposed heat source, a convection coefficient, and a fluid temperature.

**Calculated Default Insulated Part** - A part that has no heat transfer (adiabatic) on the back side. MuSES will compute a temperature for each element on the front side.

**Calculated Default Two Sided Part** - A two sided calculated part that is made up of one material. This part may have a temperature gradient through its thickness.
**Calculated Part** - A part that has an unknown temperature. Temperatures for elements of these parts will be calculated by MuSES. Parts of type Default Insulated, Default Two Sided, and 3-Layer are all called Calculated Parts.

**Coincident Vertices** - Vertices that occupy the same space (on top of one another) but are disconnected and will not have any conduction. Coincident vertices can be "condensed" to a single vertex by using the Condense function in MuSES.

**Convection Coefficient** - A type of boundary condition that can be applied to the front or backside of calculated parts. Convection coefficients are normally applied in conjunction with a fluid temperature.

**Computed Group** - Another name for a Calculated Part.

**Conduction, Face** - Union of two parts or elements that share a common polygon (defined by common edges sharing the same vertices) that allows heat to be conducted across the common face.

**Conduction, Lateral** - Union of two adjoining elements that allow heat to be conducted across the elements' common edge.

**Constant Temperature Part** - A part that has a constant (known) temperature. The temperature can be a fixed value or can be described by a time dependent curve.

**Curve** - A set of data points that defines a time dependent property or boundary condition. Curves can be created with the Curve Editor. Curves can also specify other (e.g. temperature) dependent properties.

**DXF File** - An ASCII file format specified by AutoCAD for storing geometry information. MuSES can import geometry from a DXF file. If you have a file with meshed 3DFACE geometry MuSES will import different layers as separate parts. MuSES will only open DXF files that contain meshed 3DFACE geometry.

**Element** - A planar surface with 3 or more vertices. Element data is imported from the supported geometry files, or it can be created within MuSES. Elements = facets = polygons = triangles or quadrilaterals = face. MuSES requires convex elements.

**Fluid Temperature** - A type of boundary condition that can be applied to either side of calculated parts. The fluid temperature is normally applied in conjunction with a convection coefficient.

**Front Side** - The surface normal of an element is directed out of the front side.

**Group** - Another name for a Part. A collection of elements having the same material properties and boundary conditions.
**Group Name** - Another name for the Part Name. An alphanumeric tag that is associated with a Part.

**Group Number** - Another name for the Part Number. The identification number for each Part.

**Group Type** - Another name for the Part Type. Specifies the thermal type of the Part.

**Highly Conductive Part Type** - A two sided computed part that is assumed to have no temperature gradient through its thickness. The thickness will be used for computing conduction heat exchange from element to element.

**Material Properties** - A general name for the type of material, the material thickness, and the surface condition.

**Material Type** - A material identification number that references the material being used. Specifying the Material Type is a specification of the material density, conductivity, and specific heat. The properties of a material can be examined by clicking the Material tab in the editor.

**NTL File** - The .ntl file is an ASCII file format specified by MSC/PATRAN for storing geometry information. MuSES can import geometry with part numbers and associated elements from a Patran Neutral file, but will not import any other part information.

**OBJ File** - The .obj file is an ASCII file format specified by Wavefront/Alias for storing geometry information. MuSES can import geometry with part numbers, names, and associated elements from a WaveFront file, but will not import any other part information.

**Patch or Radiation Patch** - A collection or grouping of elements that are assigned a common view factor to other radiation patches.

**Part** - A collection or grouping of elements having the same material properties and boundary conditions.

**Part Name** - An alphanumeric tag that can be associated with a part.

**Part Number** - The identification number for each part.

**Part Type** - Specifies the thermal type of the part. Assigned temperature, calculated temperature, radiation on one side or both sides, etc. The possible part types are Assigned Temperature and Calculated Temperature.

**Plus Junction** - An intersection of four elements in which there is a union at a common edge. Heat is conducted through an area based on length of the common edge and a user-specified thickness for the element.
**Patran Neutral File** - An ASCII file format specified by MSC/PATRAN for storing geometry information. MuSES can import geometry with part numbers and associated elements from a Patran Neutral file, but will not import any other part information.

**Properties** - Attributes of parts: thickness, material, surface, temperature assignments, heat loads, convection coefficient, etc.

**Shared Vertices** - Vertices that have been "condensed" to create a single vertex which will calculate conduction between the elements that share the vertex.

**Shell Geometry** - A collection of elements and vertices that describe the surface of an object. This type of geometry has no thickness. MuSES uses shell geometry to build thermal models. Also known as surface geometry as opposed to Solid Geometry.

**Side A** - Another name for the front side. The surface normal of an element is directed out of Side A.

**Side B** - Another name for the back side. Side B is the "back side" of Side A.

**STL File** - The .stl file is a Stereolithography ASCII or Binary file format specified by many CAD programs. MuSES can import geometry from a Stereolithography file, but will not import any part information. Stereolithography files only use tri's in their mesh which can make for a poor thermal mesh.

**Surface Condition** - A surface type identification number that references the surface being used. Specifying the Surface Condition is a specification of the surface emissivity and solar absorptivity (future).

**Tee Junction** - An intersection of three elements in which there is a union along a common edge. Heat is conducted through an area based on length of the common edge and a user-specified thickness for the element.
Thermal Data File - A binary file format used by MuSES to save the thermal models. The file format is platform independent format, so that files written on one machine can be easily read on another machine that uses a different internal data representation.

Thermal Node - The smallest isothermal entity that temperature calculation (or energy balance) takes place. MuSES assigns one or more nodes to a single element. NOTE that the geometric concept of a node has a different meaning and is actually a vertex.

Thickness - The thickness of a part or layer.

Three Layer Part - A two sided computed part that is made up of three separate layers and four nodes. The middle layer material can be a solid, or it can be an air or vacuum gap. If air is specified, natural convection will be computed.

Two Sided Part - A part that has heat transfer on both sides. Parts of type Default Two Sided and Three Layer are called Two Sided Groups.

Vertex - A three dimensional geometric location, used for defining the edges of an element. Also known as coordinates, points, or nodes.

View Factor - A geometry exchange factor. These factors are computed when a model is run, and are used in the thermal solution’s calculation of radiation exchange.

Wavefront Object File - An ASCII file format specified by Wavefront/Alias for storing geometry information. MuSES can import geometry with part numbers, names, and associated elements from a WaveFront file, but will not import any other part information.

Appendix B: File Types Supported By MuSES

MuSES Files (.tdf): The .tdf file is the native MuSES file format, which stores both the geometry of the thermal model and the thermal analysis results. The MuSES file is a binary file format that is platform independent, so that files written on one machine can be easily read on another machine that uses a different operating system.

Patran Neutral Files (.ntl): The .ntl file is an ASCII file format specified by MSC/PATRAN for storing geometry information. MuSES can import geometry with part
numbers and associated elements from a Patran Neutral file, and will import H & Tfilm information, but will not import any other part information.

**Nastran Files (.nas):** The .nas file is an ASCII file format specified by NASTRAN for storing geometry information. MuSES can import geometry with part numbers and associated elements from a Nastran Neutral file and will import H & Tfilm information, but will not import any other part information.

**WaveFront Files (.obj):** The .obj file is an ASCII file format specified by Wavefront/Alias for storing geometry information. MuSES can import geometry with part numbers, names, and associated elements from a WaveFront file, but will not import any other part information.

**Stereolithography Files (.stl):** The .stl file is a Stereolithography ASCII or Binary file format specified by many CAD programs. MuSES can import geometry from a Stereolithography file, but will not import any part information.

Note: *Stereolithography files only use tri's in their mesh which can make for a poor thermal mesh. See the Meshing guide in Chapter 9 for recommendations and further information.*

**AutoCAD Files (.dxf):** An ASCII file format specified by AutoCAD for storing geometry information. MuSES can import geometry from a DXF file. MuSES will only open DXF files that contain meshed 3DFACE geometry. If you have a file with meshed 3DFACE geometry, MuSES will import different layers as separate parts.

Note: *If you have a DXF file that does not contain an entity with 3DFACE geometry, MuSES will not successfully import the geometry.*

**ACAD Facet Files (.facet):** The .facet file is an ASCII file format specified by ACAD for storing geometry information. MuSES can currently import geometry with part numbers, hierarchy, and associated elements from an ACAD file, but will not import any other part information.

**FRED Facet Files (.fac):** The .fac file is an ASCII or binary file format specified by FRED for storing geometry information. MuSES can currently import geometry with part numbers, hierarchy, and associated elements from a FRED file, but will not import any other part information.
**Rhino files (.3dm):** The .3dm is a binary file format specified by Rhino for storing geometry information. MuSES can currently import geometry with part numbers, hierarchy and associated elements from a Rhino File, but will not import any other information.

**Digital Elevation Map Files (.asc):** The DEM files are specific to the Montana Road Temperature project. These are files that are derived from ARC-info files. ARC-info is a very large software program that is used for many different things like tracking population densities and other functions of smaller government, storing aerial images, and more. The DEM files are derived from aerial terrain mapping.