Improvement of a Steady State Method of Thermal Interface Material Characterization by use of Three Dimensional FEA Simulation in COMSOL Multiphysics

Benjamin N. Sponagle*, Dominic Groulx
Mechanical Engineering Department, Dalhousie University
*Corresponding author: P.O.Box 15 000, Halifax, Nova Scotia, Canada, B3H 4R2, BenSponagle@dal.ca

Abstract: An FEA model of a steady state thermal interface material characterization apparatus was created in COMSOL Multiphysics 4.2a. This model was then fitted using three convection heat loss coefficients and the conductance of the TIM layer to a set of experimental measurements made using a steady state apparatus. It was shown that the model successfully matched the measured temperature values and the TIM conductance determined using the one dimensional assumption in the case which was examined.

Keywords: Thermal Interface Materials, Electronics Cooling, Material Characterization, Heat Transfer.

1. Introduction

Interface conductance refers to the ease with which heat can be transferred across an interface between two surfaces which are in contact. When two surfaces are pressed together, only a small percentage of the nominal area will be in direct contact due to surface imperfections such as flatness and roughness [1]. Air gaps formed as a result, serve to decrease the thermal conductance associated with transferring heat across the interface. A common engineering problem where interface conductance is of interest is electronics cooling. More powerful chips tend to dissipate more heat. In order to, increase the power of a microchip design, while maintaining the operating temperature, the rate at which heat is removed must be optimized. The conduction path from the microchip to the heat sink will include at least one interface (more if a heat spreader is used). The conductance of an interface is improved by the use of thermal interface materials (TIMs). TIMs are deformable materials which can be placed into the interface. The TIM will conform to the surface irregularities filling the air gaps [1].

The performance of a particular TIM is determined by both its effective thermal conductivity (TIMs are rarely homogeneous), its ability to conform to the surface irregularities, and the thickness of the layer it forms within the interface. Therefore, TIM performance cannot be defined using a simple bulk property such as thermal conductivity. It must be tested in an interface in order to get a true measure of its performance [1-2].

One method for characterizing the conductance of an interface with a TIM applied is the steady state method characterized by ASTM D5470 – 06. The premise of this experiment is to setup a controlled heat flow through an interface with an applied TIM and then measure the temperature drop across that interface [3], as illustrated in Fig. 1.

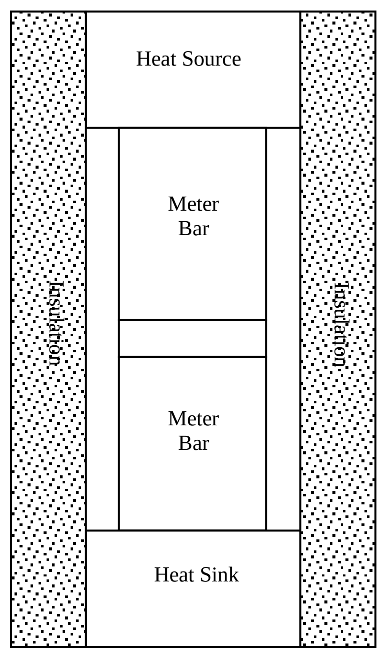

Figure 1. Illustration of a steady state TIM characterization device.

In the experimental method, the temperature gradient along two meter bars is measured and then the temperature at the hot side (T_H) and cold side (T_C) of the interface are extrapolated from measured temperature at known location in the meter bars. The conductance of the interface θ (W/cm^{2}K) can then be calculated using the following Eq (1):

$$\theta = QA^{-1}(T_H - T_C)^{-1} \quad (1)$$

Where Q is the heat transferred through the interface and A is the area of the interface. This calculation assumes that the test system is one dimensional. Heat losses from the sides of the assembly or non-uniformity in the heat source and sink will introduce experimental bias.

The authors used COMSOL 4.2a to build a three dimensional FEA simulation of the above test apparatus with the goal of being able to calculate the interface conductance of a TIM based on the experimental measurements made in a steady state test apparatus without using the one dimensional assumption.

2. Geometry and Materials

The simulation was based on a steady state test apparatus built at the Lab of Applied Multiphase Thermal Engineering (LAMTE) at Dalhousie University. The apparatus consists of three basic components: a set of meter bars, a heater and a heat sink. A simplified illustration is shown in Fig. 1 and a photograph of the setup without insulation is shown in Fig. 2.

The temperature in each of the meter bars is measured using resistance temperature detectors (RTDs) in three locations. The RTDs are evenly spread out in each of the meter bars with a 0.5 in gap between each. The six will be labeled from 1-6 going from the top to the bottom of the test assembly (see Fig. 3). Figure 3 also shows the locations and naming protocols for: T_H, T_C, and the three main temperature differences used in extrapolating T_H and T_C using the one dimensional assumption. For the model creation, the geometry was cut down the center along the line of symmetry which can be seen in Fig. 2. Figure 4 shows the COMSOL geometry; note that it has been subdivided several times to accommodate meshing.

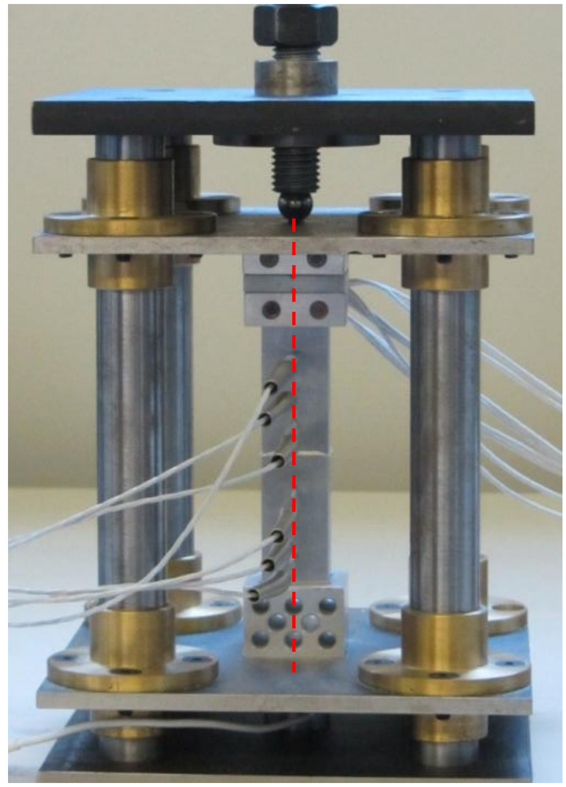

Figure 2. Photograph of the steady state TIM characterization device built at LAMTE. The red dotted line shows the symmetry cut line of the system.

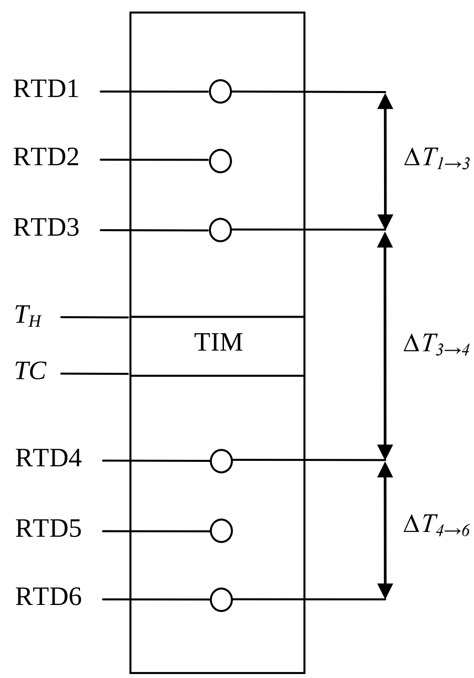

Figure 3. Schematic of the temperature probe locations and naming protocols

Figure 4. Geometry used for the COMSOL Multiphysics model.

Four materials were used in the simulation. The meter bars, heater and heat sink were Al 6061 T6. Ceramic fiber insulation (Superwool) was used to insulate the outside of the test assembly and a piece of ceramic Macor was used at the top between the heater and guard heater. Finally, there is an air gap between the fiber board insulation and the meter bars. Table 1 summarizes these materials and their properties.

Table 1: Material properties used in the simulation.

Material	Density (kg/m^3)	Thermal Conductivity (W/mK)	Specific Heat (J/kgK)
ASIS 4340 Steel	2700	167	900
Superwool 607	335	0.06	0.243
Macor	2520	1.46	790
Air	COMSOL Materials Database		

3. Use of COMSOL Multiphysics

The setup was modeled in three dimensions using COMSOL Multiphysics. Half of the actual geometry was modeled as was discussed in the previous section. All of the boundary conditions in the model were set as convection to an ambient temperature of 25 ºC except for the top of the Macor insulation. All of the heat transfer within the test assembly is modeled as conduction. Convection within the air gap between the meter bars and the insulation was not required because the heater is located above the air gap and a stable convection current will not be able to form. Three heat losses from the setup were considered: $h_{lateralheatloss}$, $h_{heatsink}$ and $q_{topheatloss}$. $h_{lateralheatloss}$ is the heat transfer coefficient associated with the outside walls of the insulation jacket and $h_{heatsink}$ is the heat transfer coefficient at the surface of the heat sink. The heat flux $q_{topheatloss}$ is a simple heat flux boundary condition at the top of the Macor insulation used to simulate the loss of heat through the top of the assembly.

The RTDs used in the experimental setup are not point probes. The specific sensors used had an element 0.5 inches (1.27 cm) long. In order to derive comparable values from the simulation the average temperature was taken along 0.5 inch lines located at the same locations as the sensors: every 0.25 inches along the meter bars.

Actual TIM layers are very thin and the bond line thickness was not measured during the experimental testing. Only the conductance values of the TIM layers were measured. This was simulated by placing a thin domain between the meter bars and then making its conductivity a function of the domain thickness as follows:

$$k_{TIMdomain} = d_{TIMdomain} \times \theta \qquad (2)$$

Using this method the conductance of a TIM can be simulated without having to simulate the actual thickness of the layer. This assumes that the edge effects associated with the thin domain are negligible. Edge effects, refers to any additional heat transfer at the edges of the TIM layer where it borders the air gap, as a result of the domain being thicker than the actual TIM layer. The validity of this assumption can be tested by running several different simulations and checking to see how the solution result changes as a function of the domain thickness.

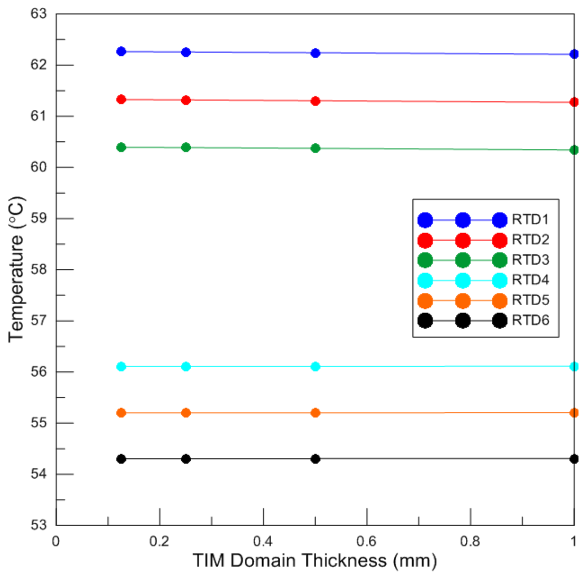

Figure 5. Simulated temperature output with a TIM domain thickness ranging from 0.125 mm to 1 mm.

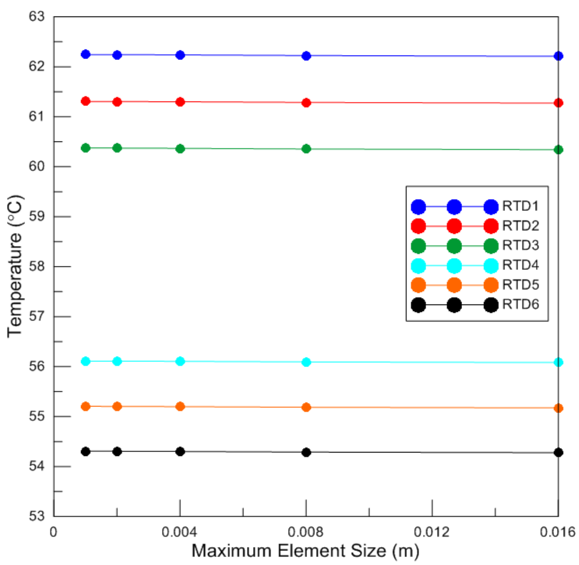

Figure 6. Convergence study showing simulated temperatures, at the sensor locations, while varying the maximum element size of the square mesh.

Figure 5 shows four consecutive tests with a TIM domain thickness varying from 0.125mm to 1 mm, doubling the thickness each time. Each of the plot lines represents the output of the simulation which would correspond to one of the experimental sensors (temperature measurement with the RTDs). It is seen from Fig. 5 that changing the domain thickness within the range studied does not affect the temperature output obtained numerically. The edge effects of the TIM domain do not affect the temperatures simulated at the sensor locations.

A hybrid mesh was used in this simulation. A swept quad mesh was used on the meter bars, heater and insulation and a tetrahedral mesh was used on the heat sink. This was a compromise between the accuracy and reliability of the quadrilateral elements and the difficulty of meshing the complex geometry of the heat sink. Figure 6 shows simulated temperatures, from the six sensor locations, while changing the maximum elements size from 0.001 m to 0.016 m, doubling the elements size each step.

The simulation converges easily. A maximum elements size of 0.002 m was used in the simulation. This was a good compromise between a fine mesh and computation time which was of the order of 2 minutes using a Intel i7 quad core computer with 8 GB of RAM.

The vast majority of heat transfer within the simulation occurs within the main assembly (heater, meter bars, heat sink). For this reason, fewer elements are required in the outer insulation layer. Figure 7 shows the effect of changing the number of elements in the insulation layer going out from the main assembly. Again, convergence is easily obtained and 4 elements going laterally out through the insulation layer were used in the simulation. Figure 8 shows the final mesh used in the model.

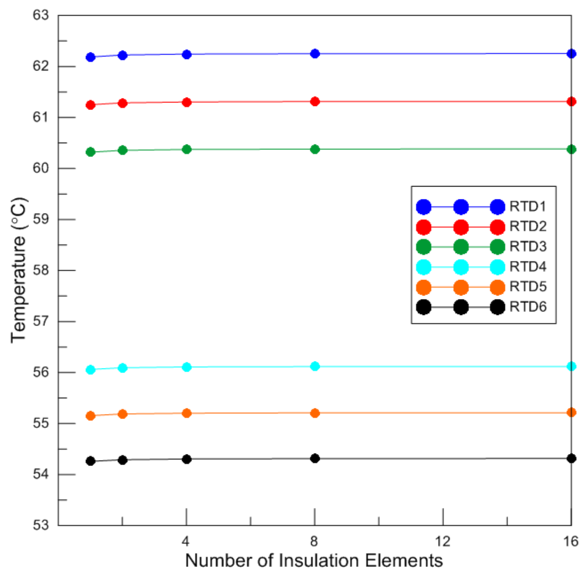

Figure 7. Convergence study showing simulated temperatures, at the sensor locations, while varying the number of elements in the insulation layer from 1 to 16.

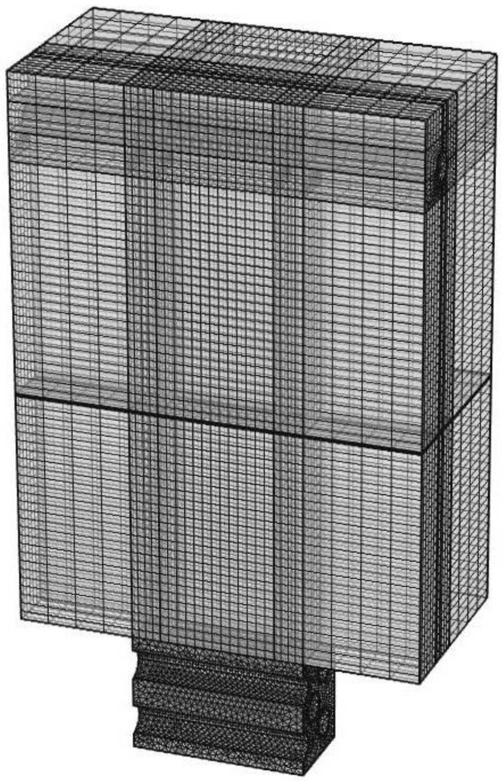

Figure 8. Hybrid mesh used in the model.

5. Results

The three heat loss coefficients introduced in section 3 and the conductance of the TIM layer were used to fit the FEA model to a set of experimental results. For the initial testing of the simulation a simple experimental result was chosen. Specifically, the case of no TIM in the test interface at 0.50 MPa (73psi) of applied pressure to the system. This was chosen mainly because there was little heat loss in the experiment and it was expected that if the simulation was working correctly it should be able to match the results found using the one dimensional assumption. The heat transfer coefficients used to match this experimental result are shown in Table 2.

Table 2: Summary of the heat transfer coefficients used in the simulation

$h_{lateralheatloss}$	2 W/m^{2}K
$h_{heatsink}$	36.1 W/m^{2}K
$q_{topheatloss}$	1.1 W
θ	0.5 W/cm^{2}K

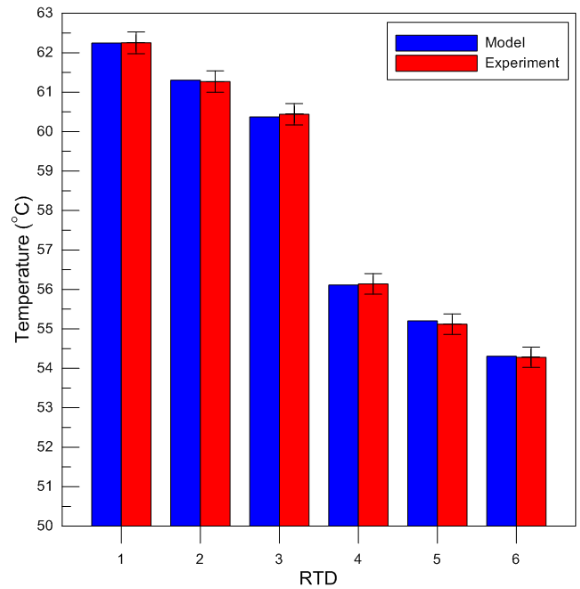

Figure 9. Comparison of the simulated temperatures to the experimental measurements.

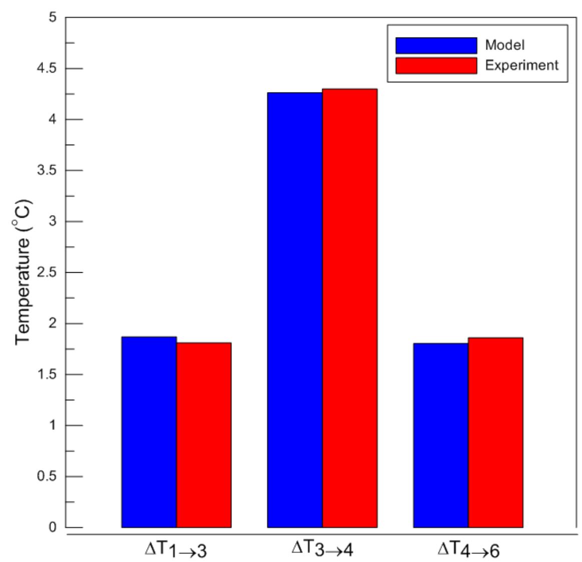

Figure 10. Comparison of the key temperature differences from the simulation and the experimental measurements.

Figures 9 and 10 compare the temperatures and key temperature differences produced by the simulation to the experimental results. The temperatures obtained numerically match up within the uncertainty of the experimental measurements.

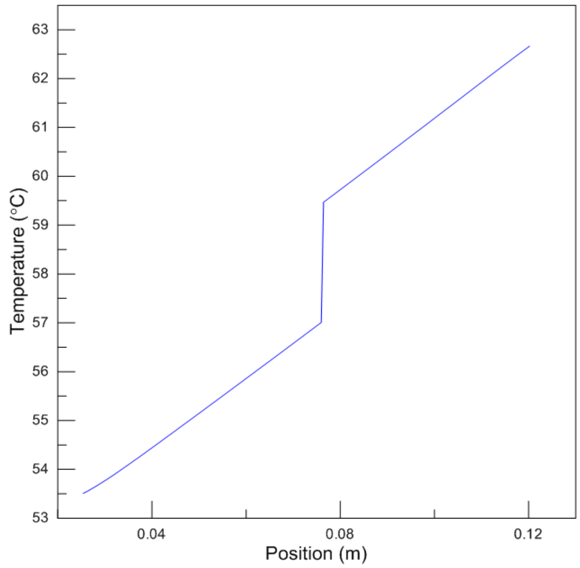

Figure 11. Simulated temperature distribution along a line through the center of the meter bars.

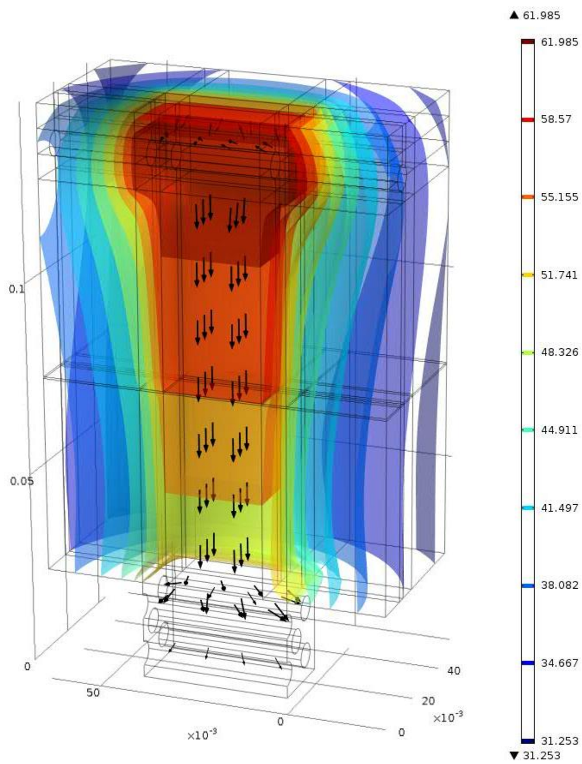

Figure 12. Isothermal surface plot of the simulation output.

For this simulation, the coefficients presented in Table 2 were manually fitted until the temperatures obtained matched the experimental data. It would be an interesting extension of this work to use a least squares approach to the fitting process. This could be done efficiently by using the interface between MatLab and COMSOL 4.2a or the optimization module.

For this fit of the data the conductance of the interface was found to be 0.5 W/cm^2K, very close to the value calculated experimentally using the one dimensional assumptions: $0.49 W/cm^2K$.

Figure 11 shows the simulated temperature distribution along a line passing through the center of the meter bars. When the simulation is fit to this set of experimental data it acts nearly perfectly one dimensional. Figure 12 shows the results when it is fitted to this data set, the colored regions show isothermal surfaces and the arrows represent the direction and magnitude of the heat transfer.

7. Concluding Remarks

This paper presented a three dimensional FEA model of a steady state TIM characterization device. The goal of this research was to develop a tool for calculating the TIM conductance from data measured during steady state characterization testing without using the one dimensional assumption. The model was compared to a set of experimental data for the case with no TIM in the interface and a clamping pressure of 73psi (0.5MPa). This set of data was known to have little lateral heat loss and would serve as a good initial verification of the model.

The model matches the experimental data well in the case examined. The temperatures were easily fitted to the experimental data using the three heat loss fitting parameters and the conductance of the TIM layer. When the model was fit to the experimental data it predicted a thermal interface conductance of 0.5 W/cm^2K which is within the experimental uncertainty of the value calculated using the one dimensional assumption. This set of experimental results was selected specifically because the heat losses were small and it was expected that if the model was valid that it should match well to those values calculated using the one dimensional assumption. The fact that the model and the experimental values matched so well does indicate that non uniformity does not play a large role in the test assembly which was simulated.

The next step for this work will be to fit the model to a larger number of experimental measurements and see how they compare. First, comparing the model to other sets of test data which have little heat loss, including several with a TIM in the interface, will increase confidence in the models accuracy. Then the model can be fitted to data sets where heat loss could be an issue and then analyze the difference between the model based calculations and the one dimensional assumption. This way, a better understanding of when the one dimensional assumption is valid can be gained.

8. References

1. J. P. Gwinn, R. L Webb, Performance and testing of thermal interface materials. Microelectronics Journal, 34, 215-222 (2003)
2. J. Gowda, D. Esler, et al., Micron and submicron-scale characterization of interfaces in thermal interface material systems, Journal of Electronic Packaging, 128 , 130-136 (2006)
3. C. J. M. Lasance, C. T. Murray, D. L. Saums, M. Rencz, Challenges in Thermal Interface Material Testing. 22nd Semiconductor Thermal Measurement and Management Symposium. 42-51 (2006)

9. Acknowledgements

The authors would like to thank Raytheon for their financial contributions to this work, as well as the Canadian Foundation for Innovation (CFI).