Thermal Analysis of Solenoid in Vacuum Environment

Joshua Goldstein, John Patrick Shivanandan, & Louis Dressel

High-Power Electric Propulsion Laboratory

Spring 2012
Dr. Mitchell Walker
Natalie Schloeder

Objective

Develop a quick, computational thermal model of a solenoid and verify if the simulation accurately predicts the heating due to power dissipation of an ion thruster.

Introduction

Ion thrusters use multi-turn coils or solenoids to generate a magnetic field inside of a discharge chamber, by running a current through the coils. The two factors that greatly influence the strength of the magnetic field are the number of turns of the coil and the current applied. The wire used for the solenoids is typically rated for a specific current or voltage, for operation in an atmosphere. However, for ion thrusters, these wires must perform in a vacuum environment, where convection will not be able to remove the heat generated in the wire. Electric propulsion engineers require accurate thermal models of the solenoids operating in a vacuum to prevent overheating and shorting.

Experimental Methods

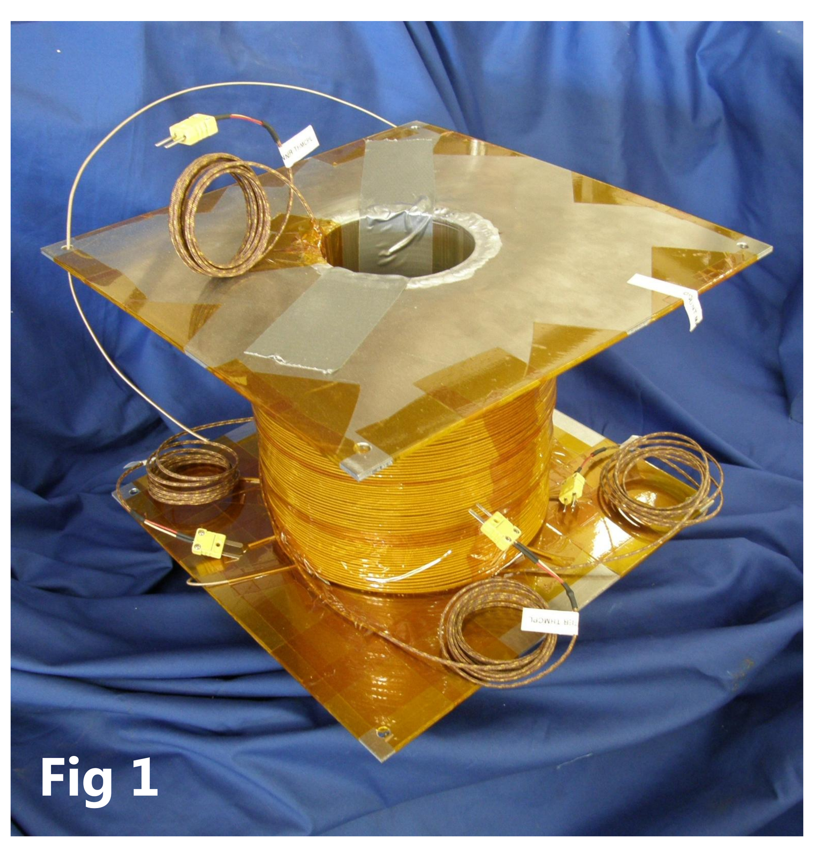

Fig 1

Figure 1: Five thermocouples were embedded at various positions within a two-thousand turn solenoid. The solenoid casing was made of aluminum. The solenoid winding consisted of fiberglass-insulated copper wire with Kapton tape separating the layers.

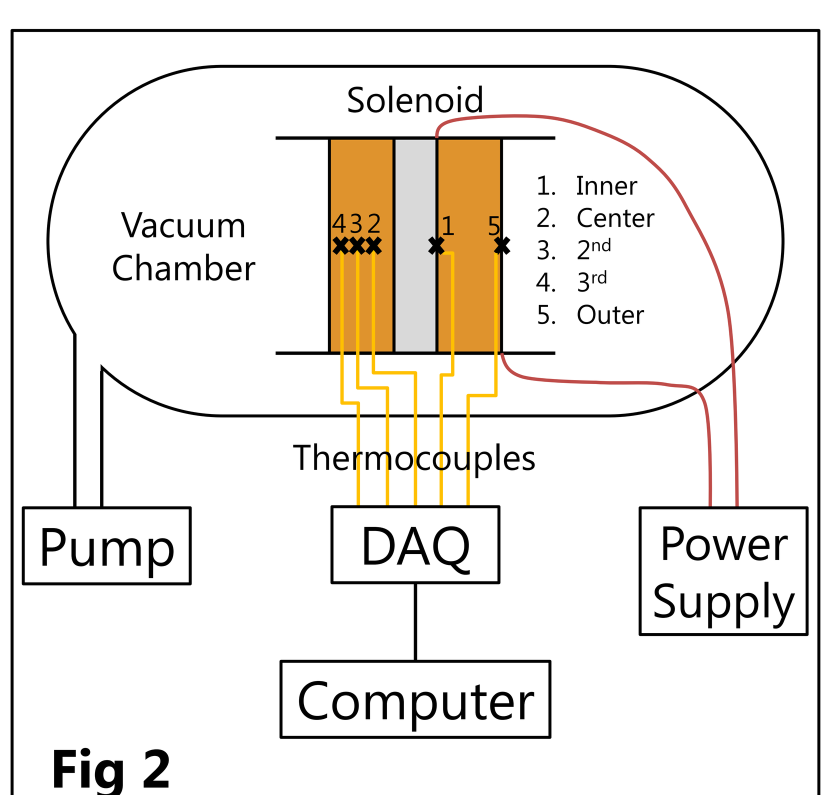

Fig 2

Figure 2: The completed solenoid was setup in a vacuum chamber and the thermocouples were connected to a data-acquisition unit (DAQ). Tests were performed at 2, 4, 6, and 8 Amps, both in atmospheric and vacuum conditions.

Computational Methods

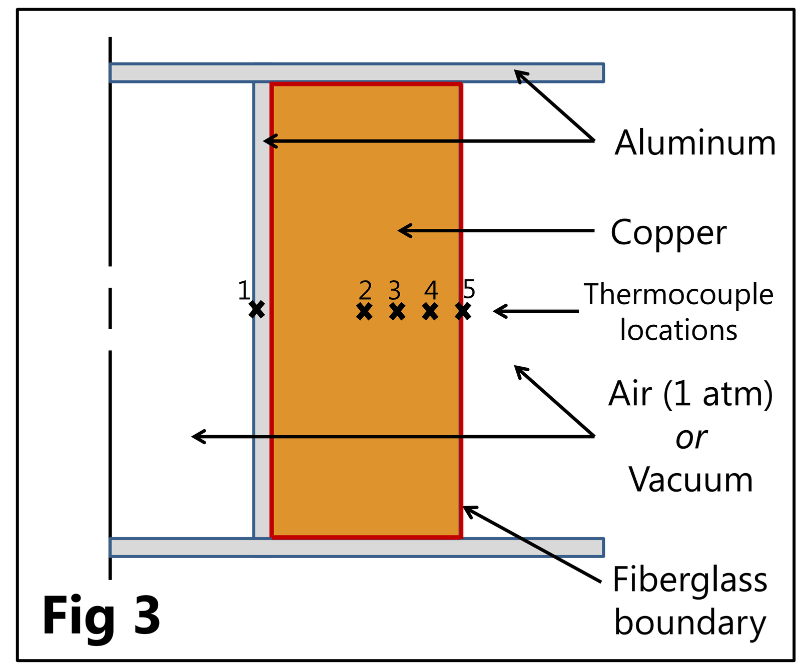

Fig 3

Figure 3: Geometry and material domains for *Comsol* analysis.

A 2D-axisymmetrical model of the solenoid was created within *Comsol Multiphysics*. The heat transfer and AC/DC physics modules were used to analyze the heating of the solenoid due to power dissipation. The material properties were taken from the built-in database within *Comsol* and a simple triangular mesh was created for the analysis.

Data Analysis

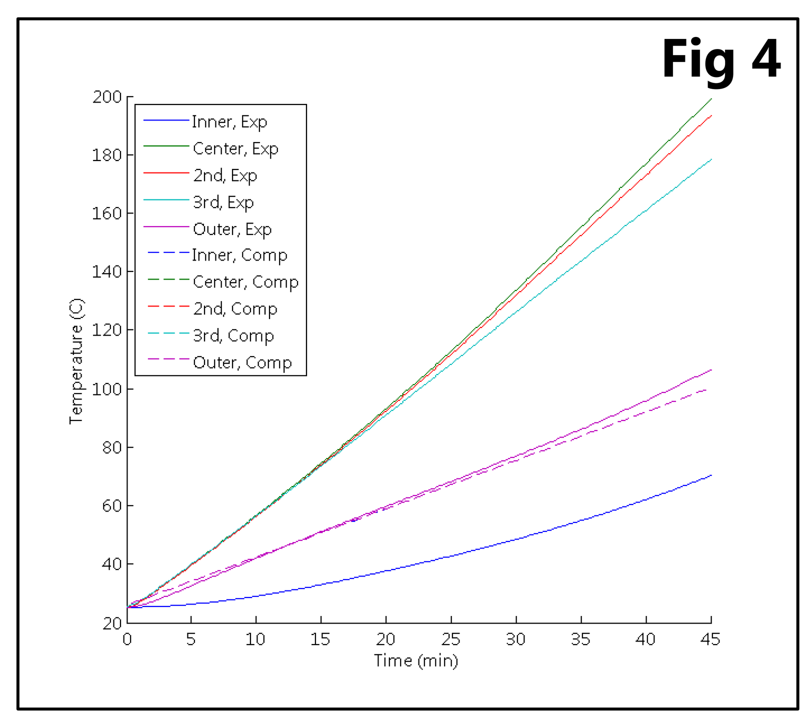

Fig 4

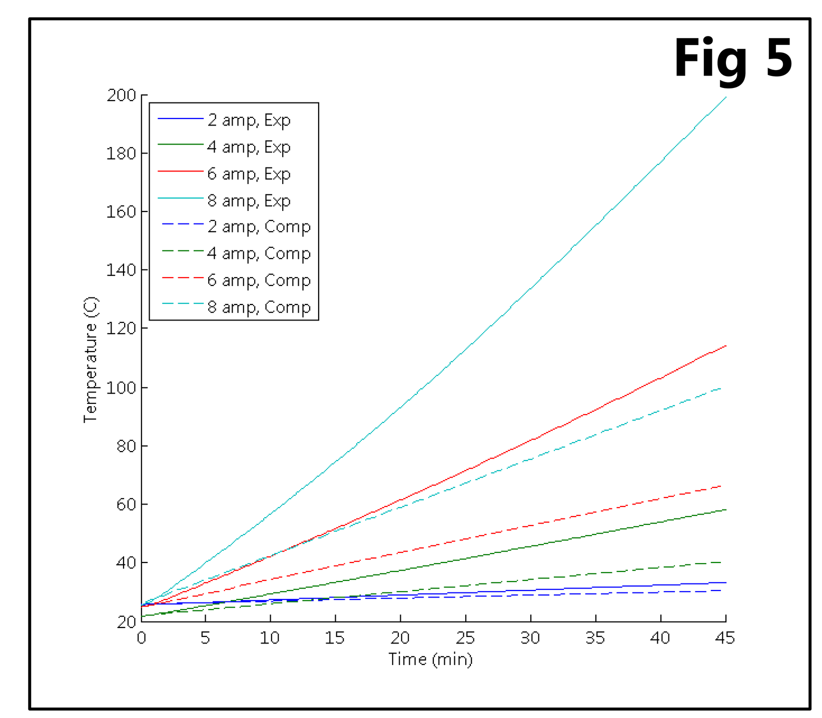

Fig 5

Figure 4: 8 Amp test at vacuum (Computational vs. Experimental)

Figure 5: Center Thermocouple temperature at vacuum (Computational vs. Experimental)

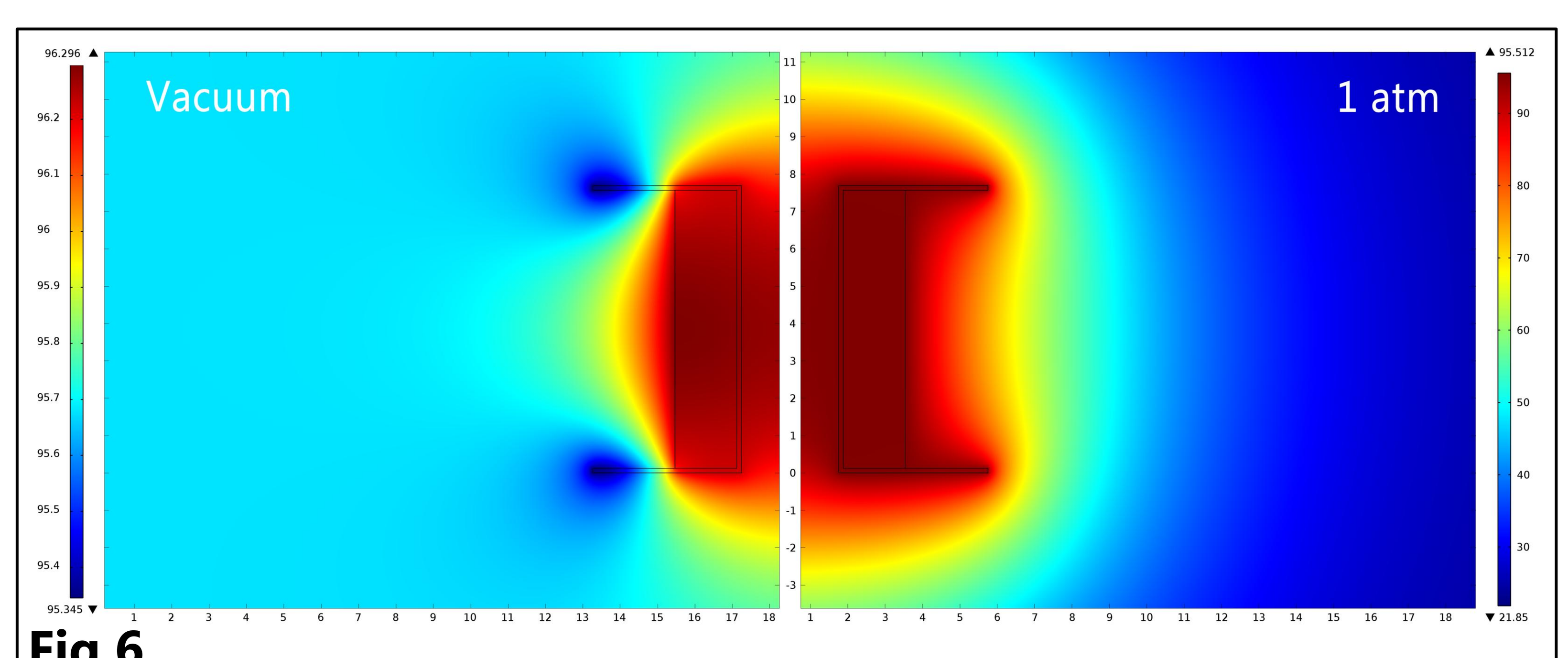

Fig 6

Figure 6: Computational thermal contours in °C for vacuum (left) vs. atmosphere (right), (8 Amp, 45 min)

Data Analysis (cont).

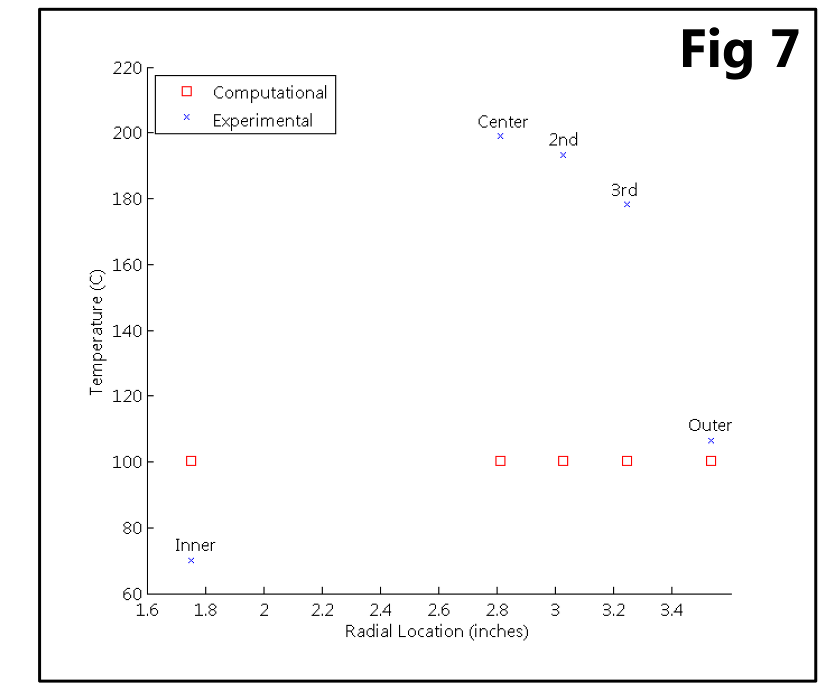

Fig 7

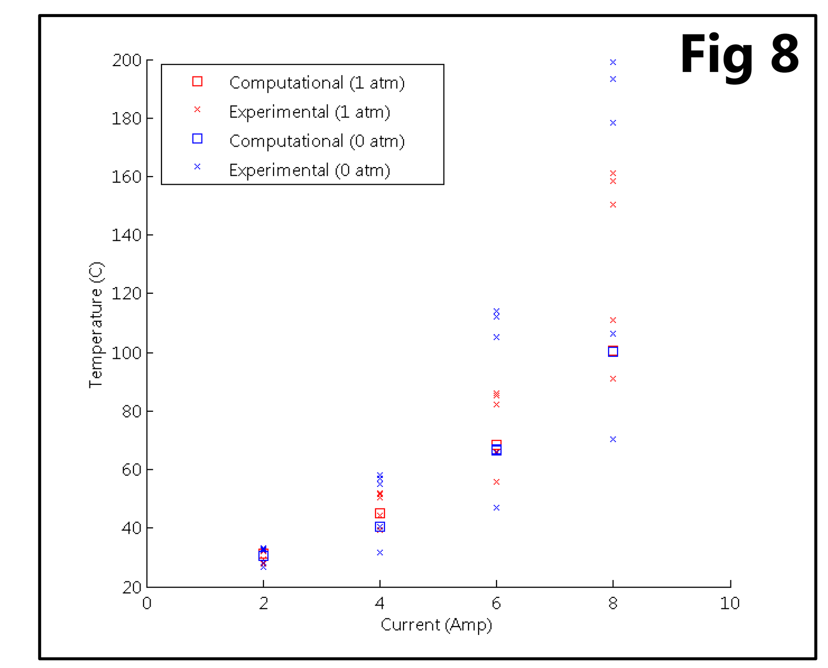

Fig 8

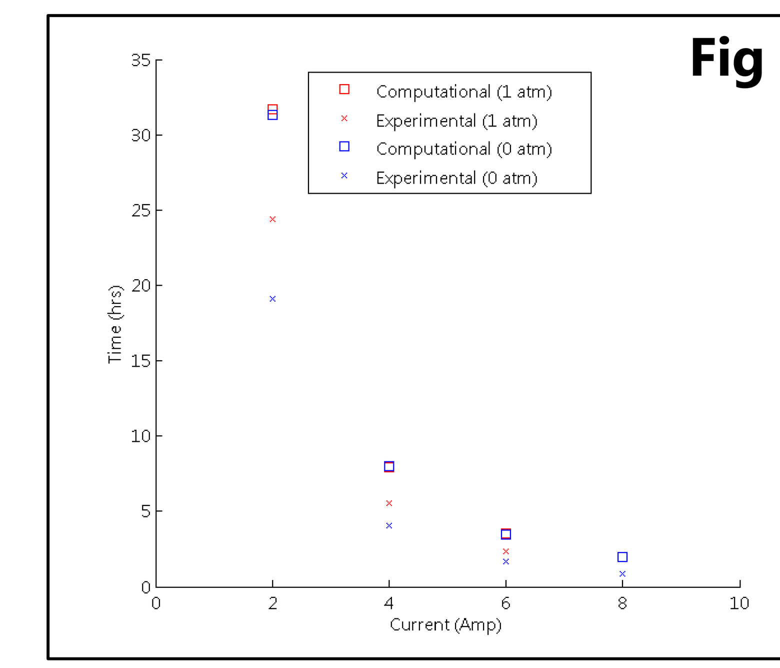

Fig 9

Figure 7: Temperature vs. radial thermocouple location (8 Amp, vacuum)

Figure 8: Computational vs. Experimental final temperature (45 min, 1 atm)

Figure 9: Estimated time to reach max temperature (220°C, Center Thermocouple)

Discussion

- The computational model:
  - Under predicts the heating created by the current in the electrical wires
  - Best predicts the temperature at the outer portion of the solenoid
  - Does not provide an accurate method of predicting the duration allowed before the wire will reach its maximum rated temperature
  - Is more accurate at lower currents
- Experimental:
  - Temperature peaks near the center of the solenoid
  - Temperature increases with increasing current
  - Hotter in vacuum than atmosphere
  - Thermocouple measurements are accurate to ±2 °C
- Problems with *Comsol* model:
  - Doesn't account for the fiberglass or Kapton tape
  - Doesn't accurately model vacuum conditions

Conclusion

This simple thermal model of a solenoid does not accurately predict the heating due to power dissipation in the coils, and thus a more detailed model needs to be developed.