30. <u>Design and Optimization of a Thermoelectric Generator Cooling</u> System using Computational Fluid Dynamics

Derek Lastiwka

Vivid Numerics Inc.

SUMMARY

The use of CFD and FEA can give valuable insight into systems where measurements are difficult or costly. The following project deals with the analysis of a Thermoelectric Generator, comparisons between CFD to experimental results and optimization of the design. The analysis process described here starts with an initial CFD model for verification and validation (V&V) which was then fed into a 2D optimization study. The final 2D optimized geometry was then optimized further taking into account 3D effects. A final V&V is planned to be completed on the final geometry.

A Thermoelectric Generator (TEG) generates power utilizing the Seebeck effect. This effect is the conversion of a temperature gradient (across many thermopiles arranged in a power unit) into electrical energy. The temperature gradient is created from a combustion process (hot side) and air cooling using fins (cold side). This presentation will focus on the design and optimization of the cooling system (cooling fins).

The analysis was conducted using computational fluid dynamics (CFD). The CFD analysis was completed using ANSYS CFX 14.0. The results of the initial CFD analysis of an existing prototype (from here on referred to as Prototype A) were compared to test data obtained on the same prototype. The test data was used for validation and verification (V&V) of the CFD model.

The results from Prototype A did not result in a satisfactory temperature gradient and additional design iterations were required. The fin geometry was identified as the primary component for improving the TEG performance. In order to optimize the fin design in a time efficient manner a series of Quasi 2D CFD analyses were completed. These analyses were done assuming that the fins were arranged in a radial pattern around the annular power unit all equally spaced. This reduced the mesh size significantly compared to a fully 3D analysis and facilitated the analysis of many fin design iterations. A total of 21 fin design iterations were modeled and analysed. From these simulations it was determined the critical design variables (besides the fin area) were:

- Length of fin leading edge below power unit
- Height of fin block (a block that supports the fin and bridges the gap to the power unit)
- Ducted versus non-ducted

After the above the design variables were optimized, an approximate 12°C reduction in temperature at the cold side junction was realized for each variable compared to

prototype A. Previous client research indicated extending the leading edge and ducting the cooling fins was beneficial and this was confirmed with the CFD simulations.

After completing the 2D analyses a fully 3D model was analysed. It was determined that a 3D optimisation would be required but with fewer iterations. Five geometry variations using the following design variables were simulated:

- Fin width
- Fin height
- Duct height above fins

Prototype B utilized the final optimized geometry. CFD results showed that the cold side junction temperatures (the critical temperature specified by the client and located at the outer surface of the power unit) for the Prototype A and Prototype B design were approximately 200 °C and 135 °C respectively. This significant reduction met the client's requirements. The total number of fins did not increase between the two prototypes and the cost to produce and assemble is expected to drop.

Testing will be conducted on the Prototype B design to validate and verify the CFD results. If needed, the CFD models will be improved and the design altered for a third prototype, Prototype C. The complete project from start to manufactured prototype, including the CFD analysis was completed in a 1 month time frame. This timeframe could not have been met without virtual prototyping simulations (CFD).