THERMAL AND STRUCTURAL ANALYSIS OF AN AUXILIARY VEHICLE HEATER SYSTEM

Rădoi Florin, Sorohan Ștefan, Vasiliu Nicolae, Călinoiu Constantin
University POLITEHNICA of Bucharest
313, Splaiul Independentei, Sector 6
RO 060042 Bucharest, Romania
E-mail: {radoi, sorohan, vasiliu, calinoiu}@fluid-power.pub.ro

KEYWORDS
Model design, numerical simulation, multiphysics FEM analysis, concurrent engineering by compatible software packages.

ABSTRACT
The paper deals with a steady-state thermal analysis and thermo-elastic stress analysis of an auxiliary vehicle heater system using finite element modeling. A detailed FE model has been created for this purpose. The FE model consists of the main parts of the assembly, including a description of thermal and mechanical loads, and contact interaction between its parts and external boundary conditions. The model considers heat transfer coefficients from current literature, which are iteratively corrected to estimate the temperature distribution in the main parts according to the target power of the heater estimated to be around 4kW. The design was developed using Catia v5 r18, and analysis was carried out using the FE program ANSYS Workbench v15. The analysis is a very useful step in a concurrent design process of such a device.

INTRODUCTION
This paper presents a three dimensional simulations of the temperature distribution within an automotive additional heater, and then the thermo-elastic stresses that appears due to the non uniform temperature distribution. The computational approach attempts to avoid the challenging aspects of complicated physical and chemical interactions of the burning and flow aspects.

The design and analysis of gas auxiliary vehicle heating systems is based on combined theoretical and empirical approach and the design of combustion chamber is less than exact science. Modern auxiliary vehicle heaters (fig. 1) are very compact and have an extremely high energy conversion rate. Operating temperature range is usually between -40 to +40 °C.
When the system is used as parking heater, it is activated by a remote control device, and it goes into start mode. The heater fan (blower) “washes” the burner with fresh air. Than the fuel is aspirated by a solenoid volumetric pump from the petrol tank and sent to the combustion chamber of the heater. The fuel-air mixture is ignited by a glow plug and the combustion chamber starts to warm up. A centrifugal circulating pump moves the cooling agent along a closed loop: from the parking heater to the car heater’s heat exchanger, then into the engine and back to the parking heater. The car heater’s heat exchanger send the heat into the cabin via the car’s fan. The cooling agent is also introduced in the car engine and warms it up as well.

Figure 1: Auxiliary water heater system (WEBASTO).

A detailed complex FE analysis provides valuable information about temperature distribution and mechanical stresses in the overall assembly of the auxiliary vehicle heater when are used together multi-physical (thermal and structural), combustion analysis and CFD. The biggest part of the thermal energy produced by the combustion process is transported with the exhaust gases and transferred to the heat exchanger boundaries of the combustion chamber by different heat transfer mechanisms: radiation, forced convection and conduction. For a heating output of about 4 kW, fuel consumption in a heating phase of 20 minutes at full load is around 0.2 l. The lower consumption of the preheated engine compensates up to 40 % of the heater fuel consumption.

MODELLING BY FEM
The target of the finite element analysis was to get the temperature distribution and maximum stresses generated in the components of the water heater system. Because the measuring of the temperatures is a very complex task and the CFD analysis is not yet available, for the first phase of
the design a simplified analysis was done. The complete temperature distribution in structural components can be obtained using an adequate finite element model in thermal analysis and neglecting the structural effects. Then, using the obtained temperature distribution as loads (Fig 2) a sequentially multi-physics coupled analysis may be developed.

![Diagram](Image)

Figure 2: Sequential coupling of thermal and structural problem

A sequentially coupled physics analysis is the combination of analyses from different engineering disciplines which interact to solve a global engineering problem (Ansys 2004). The term sequentially coupled physics refers to solving one physics simulation after another. Results from one analysis become partially inputs for the next analysis. If the analyses are fully coupled, results of the second analysis will change some input to the first analysis. Due to some unknowns concerning the real boundary conditions in the thermal analysis, the steady state thermal analysis has a supplementary loop for convective load improvement to fit the imposed thermal boundary condition for partial validate the thermic balance. The steady-state thermal analysis was simplified, only conduction and equivalent surface convection from the air, water and combustion gases being included in the thermal balance. Radiation and mass transfer were neglected in theoretical formulation but they are considered through adequate surface convection coefficients. The global equation of the thermal model, in matrix form, is (Cook et al. 1989)

\[ [K]\{T\} = \{Q\}, \]  

Here \([K]\) is the thermal characteristic matrix, \([T]\) is the vector of unknown nodal temperatures and \([Q]\) is the thermal load vector. The thermal characteristic matrix has components from conductivity and from surface convection, while \([Q]\) is obtained only from surface convection coefficients and bulk temperatures. The convective normal heat flow \(q\) is obtained from the relation

\[ q = h(T_a - T_b) \]  

where \(h\) is the film coefficient, \(T_b\) is the bulk temperature of the adjacent fluid and \(T_a\) is the temperature at the surface of the model. A steady-state thermal analysis may be either linear, with constant material properties, or nonlinear, with material properties depending on temperature. The thermal properties of most materials do vary with temperature, so the analysis is usually nonlinear. The static structural analysis is described by an equation similar to (1),

\[ [K]\{U\} = \{F\}, \]  

where \([K]\) is the stiffness matrix, \([U]\) is the vector of unknown nodal displacements, and \([F]\) is the global load vector obtained from internal pressure and thermal loading. Even for small displacements, the stiffness matrix is nonlinear because it is a function of mechanical material constants like Young’s modulus \(E\), Poisson ratio \(\nu\) etc., all of them depending of temperature. The thermal expansion coefficient \(\alpha\) of a material is function of temperature, and the thermal strain on a linear direction is obtained by

\[ \varepsilon_{th} = \alpha(T - T_{ref}) \]  

where \(\alpha\) is the secant coefficient of thermal expansion, \(T\) is the current temperature and \(T_{ref}\) is the reference (strain-free) temperature.

In the loop of improved convective loads module (see Fig. 2) of thermal analysis, we supposed that the temperature of fluids inside and outside the heater is unknown, also the convection film coefficients inside and outside the walls are unknown but their range correspond to the current literature (Baukal 2000; Gulder 1986; Heywood 1988; Incopera and DeWitt 2002; Kays and Crawford 2005). Starting with some guess values we can improve the unknowns to fit the estimated power of the heater. The project schematic of the analysis is presented in Figure 3.

![Diagram](Image)

Figure 3: Workflow of the sequentially coupled physics analysis

POWER OUTPUT ESTIMATION

Net heating value of light oil (or diesel) is round of 36 MJ/l. If the fuel consumption in the water heating system is around of 0.65 l/h, results an average net generation
power from fuel burning around of 6.5 kW. The desired heat exchanger power is 4.3 kW, so the effective power of the heat exchanger is estimated to be around of 66%.

MATERIAL DATA

Design work of modern auxiliary vehicle heater requires accurate determination of temperature, in order to optimize the correlations between size, shape and properties of materials used for parts on one hand, and the thermo-mechanical applications, on the other. Adopting the aluminum alloy for heat exchanger and its housing can effectively reduce the total weight but the elastic modulus of aluminum alloy is far less than that of cast iron, the deformation amount of the aluminum and the corresponding stress can be too large for a proper design. In the combustion chamber, there are high values of combustion temperature in the order of 2000K - 2500K (Lefbvre and Ballal 2010; Matarazzo and Laget 2011). The maximum temperature of the proximity material is much lower and the regions around the combustion chamber need to be securely cooled to prevent overheating. These are key information in order to optimize the design of the heater.

The combustion chamber has to be tested under these operational conditions. This procedure is necessary but expensive, so numerical simulation is used to reduce costs and prevent costly design mistakes. Important information was also found in (Serth 2007) which stated that the highest temperature of any point in each component must not go more than 66% of the melting point temperature of the component material. In the finite element model the heat exchanger was considered from an aluminum alloy but the flame guide and burner tube were considered from a refractory steel. Except the thermal conductivity and Young’s modulus which were considered as temperature functions (Figs. 4 - 7), the remaining properties were considered constant. The Poisson coefficient was considered 0.28 for steel and 0.33 for aluminum alloy, and the thermal expansion coefficient was considered 17*10⁻⁶ 1/K for refractory steel and 23*10⁻⁶ 1/K for aluminum alloy.

Figure 4: Thermal conductivity of aluminum alloy

Figure 5: Thermal conductivity of refractory steel

Figure 6: Young modulus of aluminum alloy

Figure 7: Young modulus of refractory steel

MESH DESCRIPTION

To optimize the analysis, we included in the finite element model only the main parts 1-7 from Figure 1 which indeed are important for thermal and structural analysis. Some parts as burner motor, blower, fuel lines, plastic covers, electronic devices, sensors etc. were neglected because their effect is not very important. Combustion gases or heat transfer medium were not explicitly modeled. Their effect upon structural parts was considered only as
boundary conditions. Due to irregular shape of the parts we adopt for mesh a tetrahedral finite element type, Solid87 for thermal analysis and Solid187 for structural analysis. A total of 205491 of nodes and 112178 finite elements result after the automatic mesh using maximum relevance of the mesh algorithm (Fig. 8). Some parts were further refined using a 2 mm element size criterion. The same mesh was used for thermal and for structural analysis.

Figure 8: Mesh of the finite element model using ten nodes tetrahedral elements

CONTACT ELEMENTS

Thermal interactions between parts were modeled by perfect non thermal resistance contact elements, whereas the structural contact elements were considered according to real interactions, (i.e. bonded or no separation). Linear contacts were introduced between involved surfaces, for example, the Figure 9 marks linear no separation contact between port housing and burner tube. The element type of contact used in Ansys was Con tact174 and for target element was used Target170.

Figure 9: Example of contact between bodies

THERMAL BOUNDARY CONDITIONS

The thermal loads were applied iteratively on surfaces, with the updated values mentioned in Table 1. The model was iterated until the desired result was obtained - the heat exchange power for heat exchanger around of 4 kW.

Figure 10 shows the areas of convective boundary condition between combustion gases and the heat exchanger and Figure 11 shows the areas of convective boundary condition between heat transfer medium-water and the heat exchanger body.

<p>| Table 1. Convective boundary conditions |</p>
<table>
<thead>
<tr>
<th>Part</th>
<th>Location</th>
<th>$h$ [W/m$^2$\degree C]</th>
<th>$T_g$ [\degree C]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1-Blower cover</td>
<td>Interior</td>
<td>45</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>Exterior</td>
<td>25</td>
<td>0</td>
</tr>
<tr>
<td>2-Ports housing</td>
<td>Interior near motor</td>
<td>45</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>Exterior</td>
<td>25</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>To burner</td>
<td>50</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>To outlet</td>
<td>100</td>
<td>400</td>
</tr>
<tr>
<td>3-Heat exchange housing</td>
<td>Exterior to air</td>
<td>25</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>Interior to water</td>
<td>1500</td>
<td>100</td>
</tr>
<tr>
<td>5-Heat exchange body</td>
<td>Exterior to water</td>
<td>1500</td>
<td>100</td>
</tr>
<tr>
<td></td>
<td>Interior to combustion gases</td>
<td>150</td>
<td>700</td>
</tr>
<tr>
<td>6-Flame guide</td>
<td>Exterior to admission air</td>
<td>50</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>Interior to flame</td>
<td>200</td>
<td>720</td>
</tr>
<tr>
<td>7-Burner tube</td>
<td>Interior to admission air</td>
<td>50</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>Interior to combustion gases</td>
<td>150</td>
<td>700</td>
</tr>
<tr>
<td></td>
<td>Exterior to combustion gases</td>
<td>150</td>
<td>700</td>
</tr>
<tr>
<td></td>
<td>Flange to outlet</td>
<td>100</td>
<td>500</td>
</tr>
</tbody>
</table>

Figure 10: The convective boundary condition between combustion gases and the heat exchanger
Figure 11: The convective boundary condition between heat transfer medium-water and the heat exchanger body

THERMAL RESULTS

For the boundary conditions from Table 1 it was obtained a power of 4.53 kW for convective heat transfer between combustion gases and heat exchanger body and 4.16 kW for convective heat transfer between heat transfer medium (water) and heat exchanger body. The temperature distribution in structural parts varies between 77 °C and 700 °C (Figure 12).

Figure 12: Temperature distribution on all model. (Maximum temperature is inside; here the top legend is not uniform; it was modified for improving clarity)

Figures 13 - 15 present the temperature distribution in the main components of combustion chamber. The gray shadow correspond to unselected parts for which the results are not plotted.

STRUCTURAL BOUNDARY CONDITIONS

The mechanical connection using long bolts between blower cover and heat exchange housing were included as special connection using Beam188 for bolt and constraint equations to transfer loads to area of direct contact between parts.

Figure 13: Temperature distribution on the heat exchanger only

Figure 14: Temperature distribution on the flame support part

Figure 15: Temperature distribution on the burning tube part
Figure 16: The four studs used for two parts connection modeled as beam elements

To remove the rigid body motion of the model and to simulate a real mode of fixity of the heater without introducing supplementary mechanical strain due to the fixed area, four points, with remote displacement condition, were used (Fig. 17). Supplementary, the maximum pressure 2.5 bar of the heat transfer agent, was included in applied static structural boundary conditions. Because there are no friction in the model, to keep the burner tube in central position four weak springs (Combin14 element type in Ansys) were introduced between burner tube and heat exchanger house.

Figure 17: Position of the fixed points and the pressure load from the liquid in the heat exchanger

STATIC STRUCTURAL RESULTS

The reaction forces were very small (max 32 N) due to adequate fixing the model. The total displacement distribution for thermo-elastic analysis for all components is presented in figure 18 where the legend is not uniform because the maximum displacement is inside of the heater. It can be seen where one fixed point on three direction was chosen (dark blue area).

Figure 18: Total displacement distribution for thermo-elastic analysis

The axial deformation of the burner tube is around of 1 mm (Fig. 19) because the maximum temperature in the cylindrical tube is large.

Figure 19: Axial displacements for burner tube

The maximum von Mises stress in the heat exchanger body is not very large, only some picks of 103 MPa can be observed (Fig. 20), whereas in the burning tube, due to the high gradient of the temperature, the maximum von Mises stress is around of 306 MPa (Fig. 21).

Figure 20: Von Mises stress distribution for heat exchanger body
CONCLUSIONS

A very simple procedure, using finite element modeling and simulations, for temperature distribution in a water heater system is presented and the obtained results are in accordance with the current literature. Based on this temperature distribution, the stress distribution in the model is effortless to obtain. The presented methodology can be improved if some experimental data became available. Then the convection coefficients and bulk temperature can be considered spatially variables and an optimization module can be used for power balance of heat flow in the model. At this stage important results were obtained only by careful trials of different thermal boundary condition from the literature until the desired power output was reached (Jonathan et al. 2007).

REFERENCES


WEB REFERENCES

www.webasto.com
www.parkingheater.com
www.eberspaecher.com
www.ansys.com

ACKNOWLEDGEMENTS

The work has been funded by the Sectorial Operational Program Human Resources Development 2007-2013 of the Ministry of European Funds through the Financial Agreement POSDRU/159/1.5/S/134398.

BIOGRAPHIES

FLORIN RADOI received a Bachelor degree in Automotive Engineering (2011), and a M.Sc. in Advanced Hydraulics and Pneumatics Systems, both from University POLITEHNICA of Bucharest. He started to work at Parker Hannifin since early college, and now is serving as application engineer in the field of complex hydraulics and pneumatics automation systems. He is also Ph.D. candidate within UPB, where he is developing R&D activity in the field of automotive auxiliary heating systems.

ȘTEFAN SOROHAN received the M.Sc. degree in Mechanical Engineering, in the Transportation Faculty, Railway Systems from the University POLITEHNICA of Bucharest in 1989. He became a Ph.D. in Finite Element Model Reduction in 2001 and now he is associate professor at Strength of Materials Department within the same university. His main activity consist in finite element modeling of structural, thermal and multiphysics problems.

NICOLAE VASILIU graduated in Hydropower Engineering from University POLITEHNICA of Bucharest in 1969. He became a Ph.D. in Fluid Mechanics after a research stages in Ghent State University and Von Karman Institute from Bruxelles. He became state professor in 1994 and a member of the Romanian Technical Science
Academy in 2008. He was always involved in solving complex concurrent engineering problems in fluid power systems.

CONSTANTIN CALINOIU graduated in Hydropower Engineering from University POLITEHNICA of Bucharest in 1975. In 1980 he became a member of the Hydraulic Laboratory from the Romanian Aerospace Institute. In 1998 he became Ph.D. and associated professor in the Fluid Power Laboratory from U.P.B. He is working mainly in modeling, simulation, and identification of the hydraulic and electro hydraulic control systems.