NUMERICAL SIMULATION OF THERMO-MECHANICAL PROBLEMS
BY COUPLING ABAQUS® AND FLUENT®
(Simulación numérica de problemas termo-mecánicos acoplando ABAQUS® y FLUENT®)


Abstract

The type of thermo-mechanical problem studied in this article is such that the thermal evolution of the problem affects the stress response, but the temperature field does not depend on the stress field. Consequently, the temperature history can be calculated in an uncoupled thermal analysis and, afterwards, introduced as a predefined field in the stress/displacement analysis. The software ABAQUS® has this capability but is quite limited when pretending to define a realistic thermal model. FLUENT® is used worldwide for robust simulation, visualization and analysis of fluid flow and heat transfer, with a wide capability for defining the thermodynamic characteristics of the processes. As a result more realistic thermal models can be accomplished using FLUENT®. The problem studied here was the cooling of a grooved rail (Ri60) in a cooling bed. These rails have constant cross-sectional geometries but different parts of the cross-sections have different thickness. Such asymmetry leads to non-uniform cooling and development of thermal stresses, which may be higher than the yield stress of the material at high temperatures. The situation leads to bending of the rail and development of residual stresses. Through the development of a Finite Element (FE) thermal-model in FLUENT® we could obtain more accurate temperature history of each rail’s nodes. Subsequently, these results were introduced into the FE-stress-model created in ABAQUS®, to calculate the rail’s stresses and deformations. This procedure is called sequentially coupled thermal-stress analysis. Nevertheless, in doing so, we faced the need of communication between FLUENT® and ABAQUS®. To make it possible we developed a results-conversion-program (named ‘conv_flu_abq’), using Linux shell programming (using scripts awk); R® (GNU software) and C programming. Using these tools, the conversion code had time and memory efficient execution. The development and structure of the code is presented. The code was tested successfully and the results are also provided.

Keywords: Thermo-mechanic problems, numerical simulation, FLUENT, ABAQUS.

Resumen

El problema termo-mecánico estudiado en este artículo es tal que la evolución de la temperatura define la respuesta de las tensiones, pero el campo de temperaturas no depende del campo de las tensiones. Por tanto, la historia de las temperaturas puede ser calculada mediante un análisis térmico desacoplado y luego ser introducida, como un campo predefinido, en un análisis de tensiones. El software ABAQUS® posee esta capacidad pero es bastante limitada a la hora de definir un modelo térmico realista. FLUENT® en cambio es un software ampliamente utilizado para realizar análisis de flujo de fluidos y transferencia de calor, con una amplia capacidad para definir las características termodinámicas de los procesos. El problema estudiado se refiere al enfriamiento de un rail ranurado (Ri60). Este
tipo de raíles presentan una geometría transversal constante pero su sección poseen diferentes espesores. Tales asimetrías hacen que el enfriamiento no sea uniforme y se desarrollen tensiones térmicas, que podrían llegar a ser mayores que la tensión de fluencia del material a altas temperaturas. Esta situación origina que el rail se curve y se desarrollen tensiones residuales. A través de un modelo térmico de Elementos Finitos (EF) desarrollado en FLUENT® pudimos obtener la historia de las temperaturas de los nodos del rail de una forma realista. Seguidamente, se introdujeron estos resultados en el modelo de tensiones de EF creado en ABAQUS®. Sin embargo, este enfoque originó la necesidad de comunicar FLUENT® y ABAQUS®. Para ello desarrollamos un programa de conversión de resultados (denominado ‘conv_flu_abq’), utilizando programación Shell de Linux; R® y C. A través de estas herramientas fue posible crear un código de conversión eficiente en tiempos y memoria empleada. En este artículo presentamos el desarrollo y la estructura de este código así como también resultados que demuestran su funcionamiento adecuado.

Palabras clave: Problemas termo-mecánicos, simulación numérica, FLUENT, ABAQUS.

1. Introduction

In this article we present the simulation through finite element models of the cooling process of a grooved rail (Ri60). The problem is interesting because the cooling process is the first stage of the production in which the residual stresses start to appear. Residual stresses are those stresses that are locked in the object without the submission of any service or external loads. They influence mechanical properties of structural material and as a consequence facilitate the propagation of cracks [Rongbin et al., 1998]. Residual stresses can cause plastic deformation around the contact surface rail-wheel and modify the stress field near the running line and internally in the railhead [Webster et al., 1992] causing railways failures. The main reason why residual stresses develop at the cooling stage are thermal stresses. The grooved rail have constant cross-sectional geometries but different parts of the cross-sections have different thickness. Such asymmetries lead to non-uniform cooling and the development of thermal stresses, which may be higher than the yield stress of the material at high temperatures. The situation leads to bending of the rail and development of residual stresses.

The cooling process can be modelled as a sequentially coupled thermal-stress analysis because the thermal evolution of the problem affects the stress response, but the temperature field does not depend on the stress field. As a consequence, the temperature history can be calculated in an uncoupled thermal analysis and, afterwards, introduced them as a predefined field in a stress/displacement analysis. We have used FLUENT® package to develop the thermal-model [3]. FLUENT® is used worldwide for robust simulation of fluid flow and heat transfer because its capability for defining the thermodynamic characteristics of the processes. As a result a realistic thermal models can be accomplished using FLUENT® than using the thermal capabilities of ABAQUS®. Section 2 presets the development and results of the FE-thermal-model. The stress/displacement analysis was developed using ABAQUS® package to calculate the rail’s stresses and deformations. Nevertheless, in doing so, we faced the need of communication between FLUENT® and ABAQUS®. To make it possible we developed a results-conversion-program (named ‘conv_flu_abq’). The development and structure of the code is presented in section 3. The code was tested successfully and the results are also provided in section 4.
2. Problem Definition

2.1 The Cooling Bed

The cooling bed simulated is located indoors and in our case no artificial accelerated cooling system is applied. Nevertheless, the cooling bed has a wind entry with a principal stable direction along the rails longitudinal axis and with a maximum value of 1 m/s.

The input rail’s temperature is around 900°C and the output temperature is from 20°C to 70°C. The cooling time is approximately 2:30h.

2.2 Rail ri60

The profile and dimension of the Ri60 rail simulated is presented in the following figure.

![Figure 1. RI60 dimensions and ABAQUS's part.](image)

The rail is made of steel EN200 with the following thermal and mechanical properties [4]:

<table>
<thead>
<tr>
<th>°C</th>
<th>k (W/m°C)</th>
<th>cp (J/kg °C)</th>
<th>E(GPa)</th>
<th>alpha(e⁻⁵/°C)</th>
<th>σy</th>
</tr>
</thead>
<tbody>
<tr>
<td>20</td>
<td>41.7</td>
<td>461</td>
<td>212</td>
<td>1.19</td>
<td>540</td>
</tr>
<tr>
<td>100</td>
<td>43.4</td>
<td>479</td>
<td>207</td>
<td>1.25</td>
<td>450</td>
</tr>
<tr>
<td>200</td>
<td>43.2</td>
<td>499</td>
<td>199</td>
<td>1.3</td>
<td>334</td>
</tr>
<tr>
<td>300</td>
<td>41.4</td>
<td>517</td>
<td>192</td>
<td>1.36</td>
<td>255</td>
</tr>
<tr>
<td>400</td>
<td>39.1</td>
<td>536</td>
<td>184</td>
<td>1.41</td>
<td>180</td>
</tr>
<tr>
<td>500</td>
<td>36.7</td>
<td>558</td>
<td>175</td>
<td>1.45</td>
<td>130</td>
</tr>
<tr>
<td>600</td>
<td>34.1</td>
<td>587</td>
<td>164</td>
<td>1.49</td>
<td>90</td>
</tr>
<tr>
<td>700</td>
<td>31</td>
<td>625</td>
<td>150</td>
<td>1.52</td>
<td>50</td>
</tr>
<tr>
<td>800</td>
<td>26.5</td>
<td>674</td>
<td>134</td>
<td>1.55</td>
<td>30</td>
</tr>
<tr>
<td>900</td>
<td>21</td>
<td>738</td>
<td>115</td>
<td>1.58</td>
<td>20</td>
</tr>
<tr>
<td>1000</td>
<td>15</td>
<td>818</td>
<td>93</td>
<td>1.6</td>
<td>10</td>
</tr>
</tbody>
</table>

Tabla 1. Thermal and Mechanical properties.
2. FE-thermal-model

In the next graphic the grid of the FLUENT® FE-thermal-model is presented.

![Figure 2. FE-thermal-model in FLUENT® with velocity vectors.](image1.png)

The final temperature is presented in Figure 3.

![Figure 3. Final temperature in (K) on Ri60 after 2:36h. Temperature scale from 45 ºC to 69ºC.](image2.png)

Because of the wind in the front-to-end direction the front part of the rail is cooled faster than the end part. The hottest area is the center of the big head of the rail and the coolest is the foot tips.
The base is the part of the rail that cools down faster because of its less mass and larger surface in contact to the air.

The temperature evolution, experimental and simulated, of the rail’s head in the hottest part of the rail (1.5 m form rail’s ending edge) is displayed in next figure.

We can see that the cooling rate of the model follows very closely the experimental values.

In section 4, the residual stresses and deformation produced by the temperature evolution is presented. As we said before, the FE-stress-model was calculated in ABAQUS®.
3. Conversion Code to communicate FLUENT® results to ABAQUS®.

In the sequentially coupled thermal-stress analysis adopted we had the need to create a conversion code to communicate the temperature results of the FE-thermal-model, formulated in FLUENT® (v6.0), to the FE-stress-model, created in ABAQUS® (v6.4 or v6.5), in order to analyze the stresses and deformations produced by the temperature changes of each node.

The program was developed using:
• Linux shell programming (using scripts awk),
• R® (GNU software),
• C programming (gcc, also GNU software).

Using this tools, the conversion code presents time and memory efficient execution.

A general description of the sequentially coupled thermal-stress analysis using the conversion code is presented in the graphic bellow (Figure 6).

To simplify the communication, the rail’s mesh analyzed in FLUENT® and in ABAQUS® was the same (i.e. the nodes location were identical for both analysis). This was possible by creating the mesh using I-deas® (v9.2m), and exporting this unique mesh to FLUENT® format as well as to ABAQUS® format.

ABAQUS® is able to carry out a sequentially coupled thermal-stress analysis but by using its own thermal results files (‘.inp’, ‘.prt’, and ‘.fil’ files) [5]. Therefore, our general idea to develop the conversion code was to modify the thermal results files provided by a “fake” thermal simulation of the rail in ABAQUS®. These modifications were not too obvious because of the special arrangement of the ‘.fil’ file, which is the file that contains the node temperatures and times. (We worked with the ASCII version of the ‘.fil’ file, the ‘.fin’ file. The original ‘.fil’ file is written in binary format).

For memory store efficiency, ABAQUS write its results in a 80-character logical records, in the order described in the record definitions presented in the ABAQUS User’s Manual. For example, the line bellow represents a record for the temperature of the node “2”:

*I 14I 3201I 12D 8.019542800000005D+01

The beginning of each “record” is indicated by an asterisk (*). The first part “I 14”, indicates that this is a record of “4” words. The second part “I 3201” indicates the record key “201” which represents: node temperature. The third part “I 12” indicates the node number: “2”. And the last part “D 8.019542800000005D+01” indicates the temperature of that node as a floating point number (represented by the “D”). Each 80-character logical record is completely filled before the next one is started, so that any data item can be split. So, a general view of the ‘.fin’ would be:


After modify the ‘.fin’ file with the valid rail’s temperatures obtained by FLUENT, we were able to perform the stress analysis in ABAQUS, using as the temperature field this final ‘.fin’ file.
Figure 6. General description of the sequentially coupled thermal-stress analysis using the conversion code: conv_flu_abq.sh
3.1 Conversion code

The conversion code was called conv_flu_abq.sh and is composed by several subprograms. The list of the subprograms and their functions are presented bellow.

1. convert_abq.sh: takes the '.abq' files, exported from FLUENT FE-thermal- model, obtains its node's names and coordinates and store these values in "NODES_FLU.CSV". It also obtains the temperature history (temperature and its times) for each node and stores it in "TEMP.CSV".

2. nodes_inp.sh: takes the ABAQUS’s problem-definition-file, ‘.inp’ file, produced by the “fake” ABAQUS thermal analysis, obtains the node's names and coordinates and store these values in “NODES_INP.CSV”.

3. head_fin.c: takes the ABAQUS’s thermal-result-file, ‘.fin’ file, produced by the “fake” ABAQUS thermal analysis, obtains the header of this file and stores it in “HEAD_FIN.TXT”.

4. COMPARE_NODES.R: takes “NODES_FLU.CSV” and “NODES_INP.CSV” and generates “SALIDA_TEMP_0x.TXT” files with the data of times (represented in the name by “0x”), nodes and temperatures, in the ABAQUS’s 80-character logical records format.

5. change_fin.c: arrange the contents of each “SALIDA_TEMP_0x.TXT” files, in a unique file “TEMP_FIN.TXT”.

6. And finally, the same COMPARE_NODES.R code, adds “HEAD_FIN.TXT” and “TEMP_FIN.TXT”, to obtain the final and valid ‘.fin’ file: “Temp_results.fin”.

4. FE-stress-model results

The model developed in ABAQUS® is shown in next figure.

![Figure 7. FE-stress-model parts and assembly developed in ABAQUS®](image)

With the temperature results (stored in the ‘.aba’ files) we apply the conversion code to introduce the nodes’ temperature history in the ABAQUS stress analysis.
In the figure below we can see the simulated and experimental residual stresses values of the rail at cooling final time (2:36h).

![Graph showing experimental and simulated values comparison of residual stresses on Ri60 cross section.](image)

**Figure 8.** Experimental and simulated values comparison of residual stresses on Ri60 cross section.

We can say that the simulated values are very similar to the values obtained in the experimental test. There are some minor differences that can be caused by the measurement method (the saw cut) which have a tolerance error of 20MPa.

A general view of the thermal, residual stress and deformation results at final time is presented in Figure 9. The deformation are also the same as the observed in the real process.
Figure 9. Rail residual stress contour plot [Pa] and deformation of R60 at final time (2.36h).
Conclusions
We presented a FE-thermal-model of the cooling bed developed with the powerful CFD (Computational Fluid Dynamics) software FLUENT®. We were able to calculate the thermal variables of interest from a relatively simple problem definition: geometries, materials thermal properties, boundary conditions, operating conditions and ambient models (radiation model, convection model, etc.). The FE-thermal and the FE-stress models results have been compared with success against experimental test.

By using FLUENT® to model the thermal behaviour of the cooling bed we faced the problem to communicate the results to the stress analysis software ABAQUS®. After study the export and import results capabilities of both software, we developed a code that introduce the temperature results obtained by FLUENT® to the FE-stress-model created in ABAQUS®. The converter code, named conv_flu_abq.sh, was successfully tested with an 5m rail R60, and the thermal and stress results are presented.

References


Acknowledgment
We gratefully acknowledge support from the Ministerio de Educación y Ciencia de España, Direcció General de Investigación, by means of the DPI2004-07264-C02-01 research contract; from the “II Plan Riojano de I+D+i”; from the European Union by means of the CEUTIC INTERREG III A Spain/France trans-border cooperation project; and from the RFCS program by means of the RFS-CR-04023 and RFS-CR-04043.

Notes
Alpha Pernía Espinoza, Francisco Javier Martinez de Pisón Ascacibar, Fernando Alba Elías and Marina Corral Bobadilla teach at the Universidad de La Rioja. Ana González Marcos teaches at the Universidad de León. The EDMANS group encompasses professors and researchers from these two universities. Visit http://apiur.unirioja.es/edmans/ for more details.

Correspondence (For more information contact to):
Alpha Pernía Espinoza, Departamento de Ingeniería Mecánica, Universidad de la Rioja, c/ Luís de Ulloa 20, 26004 Logroño, LA RIOJA. Teléfonos: 941299232 / 636885169.