Heat generation analysis of a rubber wheel using the steady-state transport analysis capability in Abaqus

R. K. Luo[1], X. P. Wu[2] and A. Spinks[1]

[1]Trelleborg IAVS, 1 Hoods Close, Leicester, UK , LE4 2BN

[2] School of Civil Engineering and Architecture, Central South University, Changsha, Hunan, China, 410075

E-mail: robert.luo@trelleborg.com

Abstract: A simulation is needed to investigate the heat generation on a rubber section of a wheel which is running at a constant speed. In parallel with this loading case a rig test was carried out to verify the finite element analysis. For analysis point of view a traditional way is to use the Lagrangian approach in which the mesh is used to discretize the structure (both materials and mesh are spatially fixed) and many revolutions are typically required to obtain the state of interest to the analyst, this approach is prohibitively expensive and cumbersome. Abaqus provides an approach based on the Eulerian theory in which the finite element mesh of the wheel does not rotate, instead the materials flow through the mesh. This analysis has used the following user subroutines: UMASFL to specify prescribed mass flow rate conditions; HETVAL to provide internal heat generation; USDFLD to to redefine field variables at a material point。There are two steps in the whole analysis. Firstly a mechanical analysis was carried out to find the deformed zone of the wheel due to the interaction between the drum in a test rig and the wheel. Secondly a heat transfer analysis was carried out based on the results from the mechanical analysis. However in order to use the multi-cpus capability provided by Abaqus the user subroutine UEXTERNALDB should be employed toinout the required data. The results have shown that by using the capability provided by Abaqus it is possible to simulate this complicated situation.

Keywords: Heat Transfer, Hyperelasticity, Rubber, Viscoelasticity

1. Introduction

A simulation is needed to investigate the heat generation on a rubber section of a wheel which is running at a constant speed. In parallel with this loading case a rig test was carried out to verify the finite element analysis. Figure 1 shows the rubber wheel against the drum on the test rig.

Figure 1 The rubber wheel in the test rig

For analysis point of view a traditional way is to use the Lagrangian approach in which the mesh is used to discretize the structure (both materials and mesh are spatially fixed) and many revolutions are typically required to obtain the state of interest to the analyst, this approach is prohibitively expensive and cumbersome. Abaqus provides an approach based on the Eulerian theory in which the finite element mesh of the wheel does not rotate, instead the materials flow through the mesh. From mechanical analysis point of view a sequentially coupled thermal-stress analysis is needed to obtain the temperature profile and then the stresses solutions.

However the heat generated in the wheel is due to the rubber deformation and the friction around the contact area. It is necessary to locate the contact area before the actual heat transfer analysis can be carried out. Figure 2 is a flow chart about the analysis procedure.

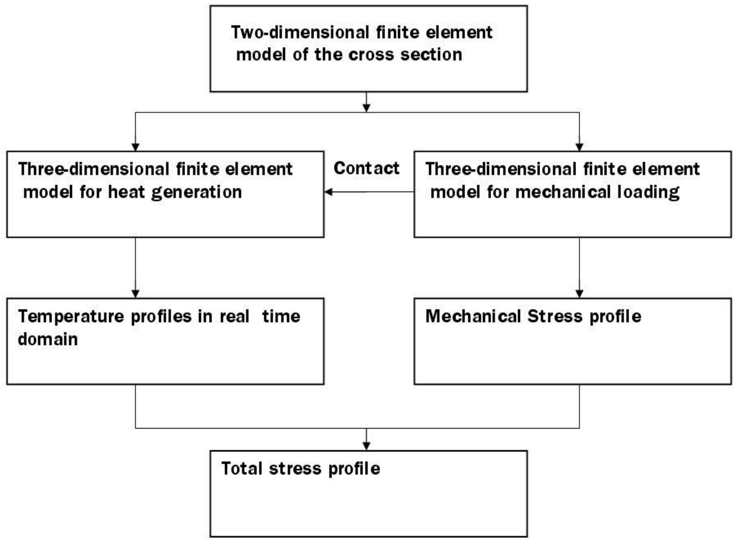

Figure 2 Flow chart showing the analysis procedure

2. Heat transfer

Heat generation in a rubber spring can be modelled with conductivity, internal energy movement, convection and radiation boundary conditions. The energy balance can be written as

$$\int_V \rho \dot{U} dV = \int_S q dS + \int_V r dV \qquad (1)$$

where V is a volume of rubber spring materials, S is a free surface of the spring, ρ is the density of the spring materials, $\dot{U}$ is the material time rate of the internal energy, q is the heat flux per unit area of the spring, flowing into the body; and r is the heat supplied externally into the spring per unit volume. The heat in the rubber wheel was generated through the contact patch between the wheel and the drum. In order to determine the amount of the heat input to the wheel, it was assumed that the external work done by the drum was transferred totally into the rubber part of the wheel as the heat energy.

3. Mechanical loading

Abaqus provides a specialized analysis capability to model the steady-state behaviour of a cylindrical deformable body rolling along a rigid surface. The capability uses a reference frame that removes the explicit time dependence from the problem so that a purely spatially dependent analysis can be performed. This choice of reference frame allows the finite element mesh to remain stationary so that only the part of the body in the contact zone requires fine meshing. On the current version only three-dimensional analysis with an axisymmetric geometry or a periodic geometry can use this capability. Stress relaxation during the dynamic loading has been modelled using a viscoelastic model. It can be achieved by directly specifying the Prony series parameters in a command line `*viscoelastic,time=prony`. It should be noted that the viscoelastic input is not affected a static analysis (otherwise use `*visco`).

4. Analysis procedure

The following steps were performed.

Step1: Create two-dimensional finite element model

The cross section of a two-dimensional finite element model was built up to form a base for a three-dimensional model. A command `*restart` should be used for the next three-dimensional analysis. `*user defined field` (update field variables at material points for user subroutine USDFLD), `*depvar` (specify numbers for solution-dependent state variables) and `*heat generation` (volumetric heat generation for user subroutine HETVAL) were defined in the input file. However at this stage there was no meaningful application for the user subroutines. The reason for these inputs is that the user subroutines can be used at later three-dimensional analysis. In a summary the following commands were used:

```
*USER DEFINED FIELD (update filed variables at a material point)
*DEPVAR (specify solution-dependent state variables)
*CONDUCTIVITY,DEPENDENCIES=1
*SPECIFIC HEAT
*DENSITY
*HEAT GENERATION (HETVAL required, volumetric heat generation)
*INITIAL CONDITIONS, TYPE=TEMPERATURE
*INITIAL CONDITIONS, TYPE=MASS FLOW RATE
```

This analysis has used the following user subroutines:
HETVAL ----- to provide internal heat generation in heat transfer analysis
USDFLD ---- to redefine field variables at a material point

However it should be noted that there is no actual meaning at this stage and this is the format request from Abaqus input. It must be modified in the next three-dimensional analysis. Figure 3 shows the finite element model of the cross section of the rubber wheel.

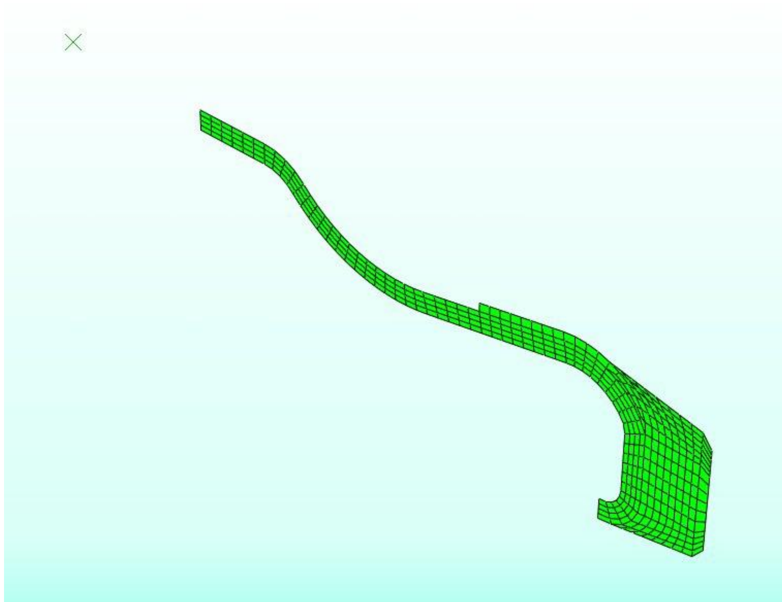

Figure 3 Finite element model of the cross section of the wheel

Step 2: Create three-dimensional finite element model

Figure 4 shows a three-dimensional finite element model used for a steady state transport analysis. An input file containing the following commands was used for three-dimensional heat analysis:

```
*SYMMETRIC MODEL GENERATION, REVOLVE, ELEMENT=, NODE=, FILENAME=
*SYMMETRIC RESULTS TRANSFER, STEP=1, INC=1
*PHYSICAL CONSTANTS, ABSOLUTE ZERO=-273.15,STEFAN BOLTZMANN=5.7E-11
*HEAT TRANSFER
*MASS FLOW RATE,USER
*SFILM
*SRADIATE
*NODE FILE
*NODE OUTPUT
```

User subroutines used in three-dimensional model:
HETVAL ----- to provide internal heat generation in heat transfer analysis
USDFLD ---- to redefine field variables at a material point
UMASFL --- to specify prescribed mass flow rate conditions

In the user subroutine, OPEN statement is needed to input previous saved data, e.g. different material density needs correctly assigned to the corresponding element sets .

OPEN (112,FILE='file name', STATUS='NEW, OLD,UNKNOWN')

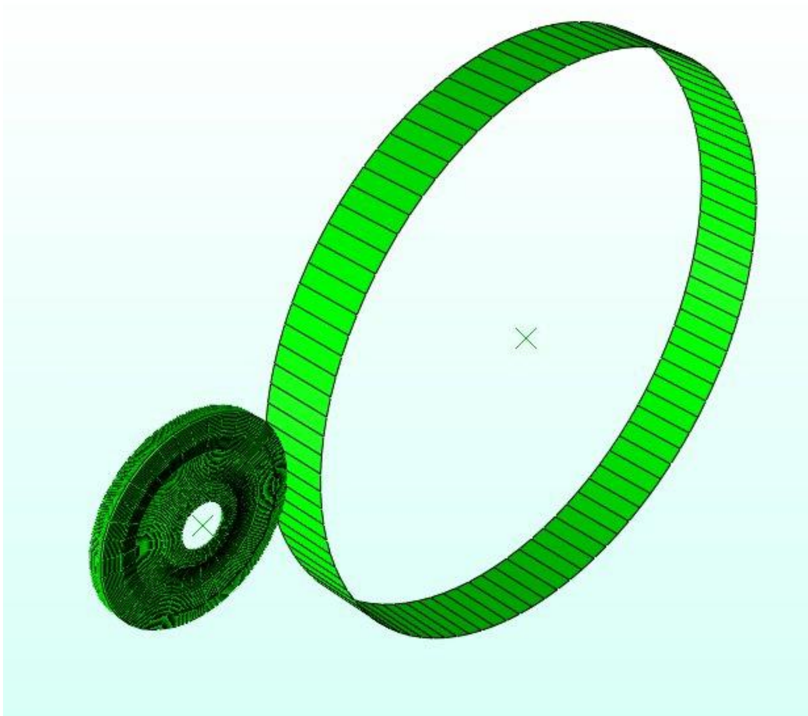

Figure 4 Three dimensional finite element model of the wheel

An example for the three-dimensional model generation is:

symmetric model generation, revolve, element=500***,node=***500***,filename=***pp*
0.0, 0.0, 0.0, 1.0, 0.0, 0.0
0.0, 0.0, 1.0 (coordinates for the axisymmetric cross section)
160., 16,1, G (segment angle, subdivisions ,bias ratio and general 3D elements)
40., 16.1, G
160., 16,1, G

Before the heat generation can be carried out it is necessary to determine the contact area between the rubber ring and the drum as well as the rubber volume which transfers the dynamic energy to heat energy. The contact area was calculated using the steady state transport capability provided by Abaqus. First a static load 5.9 kN was applied to the wheel through the drum. Second a steady state transport analysis was conducted. A free rolling state had to be obtained to simulate the test situation since the whole system was running at a constant speed. A hand calculation shows the free rolling speed of the wheel is around 50 Rad/s. A series of test calculations led a free rolling speed at 49.1812 Rad/s. The contacted area contributed more than 90% of the total rubber strain energy density. It was assumed that all heat would be generated via these elements. Figure 5 shows the strain energy density profile obtained from the steady state transport analysis and used for the heat energy input.

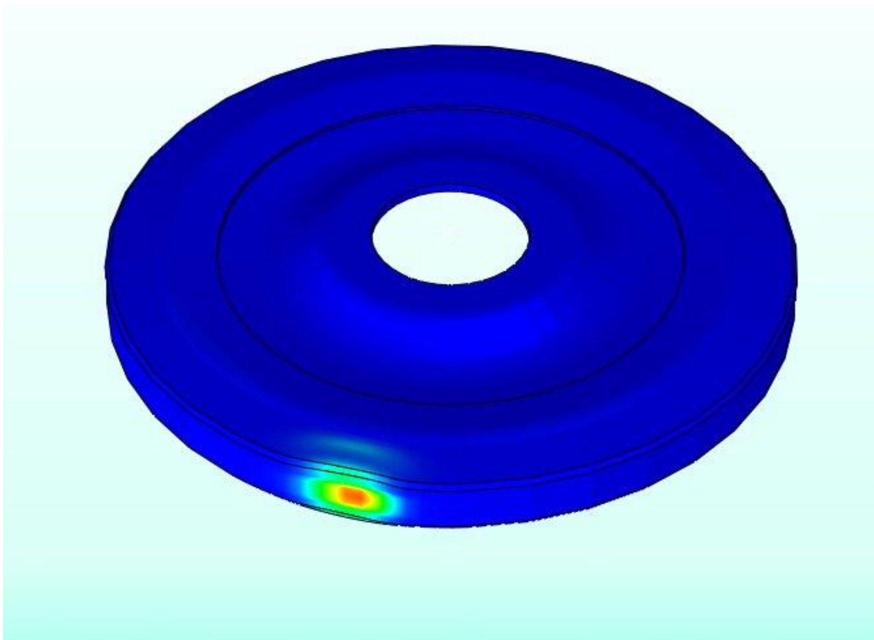

Figure 5 The strain energy density profile of the rubber wheel

The heat exchange occurs between the free surface of the wheel and the surrounding air. Both convection and radiation were applied to the free surface of the wheel as the thermal boundary conditions.

The following commands for the steady state transport analysis.

```
*SYMMETRIC MODEL GENERATION, REVOLVE, ELEMENT=, NODE=, FILENAME=
*SYMMETRIC RESULTS TRANSFER, STEP=1, INC=1
*CHANGE FRICTION (critical)
*STEADY STATE TRANSPORT, LONG TERM,INERTIA=YES
*TRANSPORT VELOCITY (angular velocity in Rad/S)
*MOTION,TYPE=VELOCITY,TRANSLATION
```

5. The multi-cpus capabilities

However in order to use the multi-cpus capability provided by Abaqus the user subroutine UEXTERNALDB should be employed to input the required data. It has been tested many times that the efficient way to carry out the analysis is to read in the required data through the user subroutine UEXTERNALDB and link these data with other user subroutines using COMMON statement. If multi-cpus specified, like `Abaqus J=JOB_NAME USER=USER SUBROUTINE CPU=4`, but no UEXTERNALDB used, the computer will hold the job without any cpu running.

6. Results and discussions

The two-dimensional analysis run 2 seconds (clock time) on a Intel Xeon CPU 5130 @ 2.00 GHz. The three-dimensional analysis run 1hour and 33 minutes (clock time). At current Abaqus version only 1 CPU is allowed for this type of analysis (if the user subroutine UEXTERNALDB not used). Figure 6 shows the temperature change during a very short period of the drum test. At beginning the increase of the temperature is limited to a very small area (about 1/14 circumference) and in a few seconds it quickly propagates to the rest of circumference surface. It should be a good practice to check the histories of the temperature change to make sure the model is generated correctly.

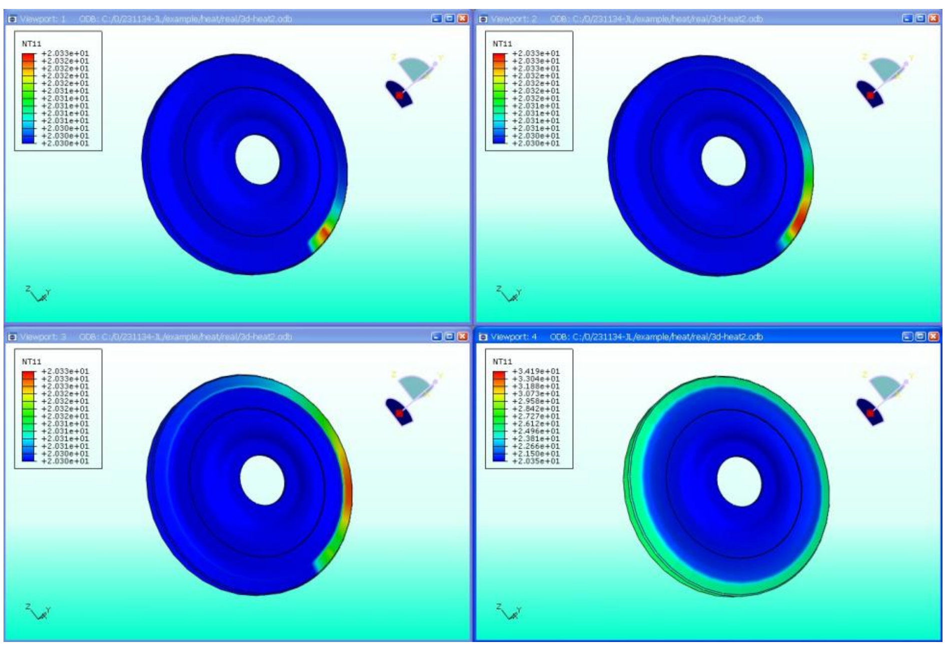

Figure 6 A short histories of the temperature change of the rubber wheel

Figure 7 shows the temperature profile on the cross section of the rubber wheel. It is clear that the highest temperature did not occur on the surface of the wheel; instead it located at the middle of cross section of the rubber part. The difference between them is around 95 degrees. The reason for this phenomena is that the rate of heat loss on the surface is much higher than that of the inside area.

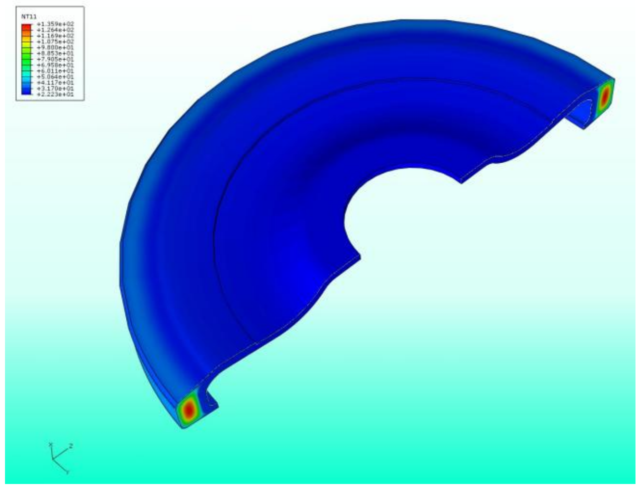

Figure 7 A temperature profile of the rubber wheel

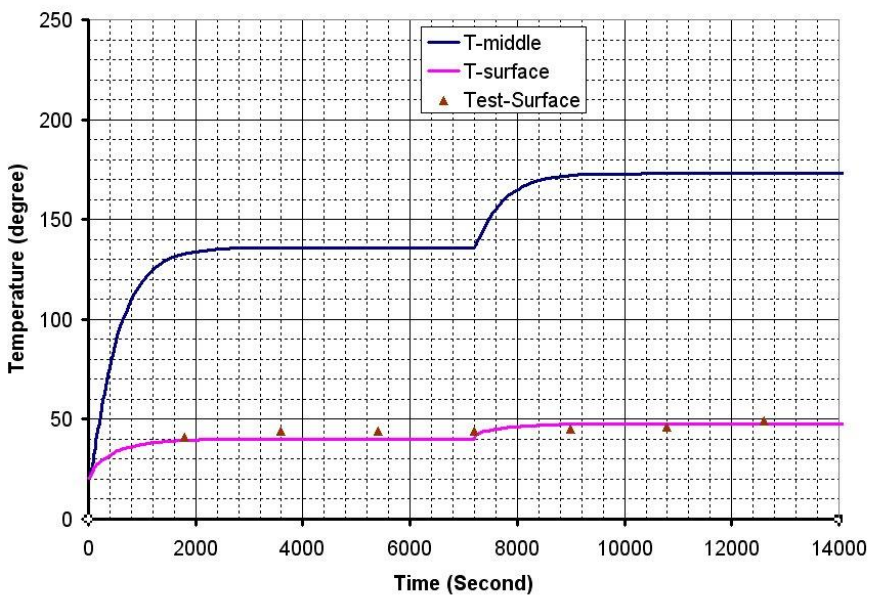

Figure 8 Comparison between the FE results and the test results

Figure 8 shows the temperature histories comparison between the finite element analysis and the test. The loading histories consist of two sections, first is 5.9 kN compressive load and the second is a 6.5 kN compressive load. The comparison between the simulation and the test is made on the surface temperature. It is shown that they matched very well after the steady state has arrived. In addition it is noted that the difference between the surface temperature and the inside temperature varies with the loading. The higher the loading the greater the difference is.

After the heat transfer analysis finished it is possible to carry out the stress analysis by combine the mechanical loading and the thermal loading together. The subsequent stress analysis is not the topic of this report. However, it is suggested that the temperature profile during the thermal stress analysis should be used against the same (or nearly the same) temperature file during the heat transfer analysis. By doing this it would help to secure the input of the correct temperature files. Figure 9 shows the two temperature profiles, which supposed to be identical. However due to different element types were used, it caused incorrect temperature input.

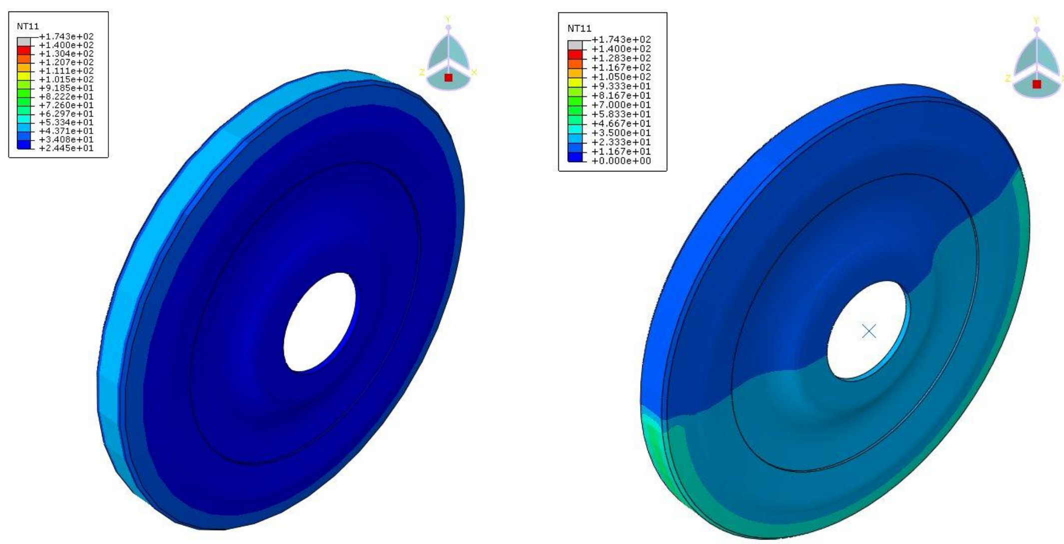

Figure 9 Mismatched temperature profiles

7. Acknowledgements

We thank Mr. Lake from Product Design Department and Mr. Jordan from Technical Department for their support. X.P. Wu, one of the authors, would like to thank the support from National Natural Science Foundation of China(50578160/50878214);the Key Grant Science and Technology Research Planning Projects of The Ministry of Railway of Peoples Republic of China (2008G032-12;2008G017-C)

8. References

1. R. K. Luo, W. X. Wu & W. J. Mortel, “A method to predict the heat generation in a rubber spring used in the railway industry”, Journal of Rail and Rapid Transit, 219, 2005, 239-244
2. R. K. Luo, W. X. Wu & W. J. Mortel, “Real time simulation and testing on heat generation of rubber spring under dynamic loading”, Constitutive Models for Rubber IV, (Proceedings of the Fourth European Conference on Constitutive Models for Rubber), A.A.Balkema, 2005
3. Abaqus Manual, version 6.8, 2008