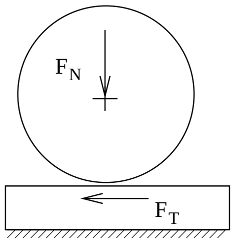
ENSMP 2nd/3rd year, finite element course, 22nd – 26st of november 2004

Study of the stresses in a wheel/rail contact



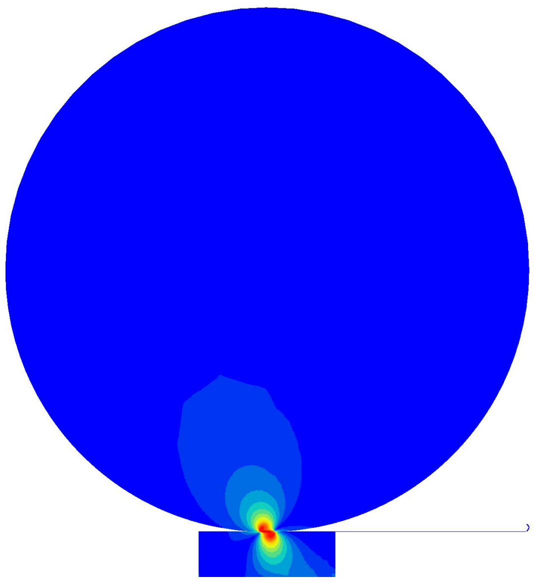
We model the wheel of a locomotive and a piece of rail by a cylinder and a plate. The weight of the locomotive puts a load of 9 tons on each wheel ($r = 575\,mm$). The weels and rails are both made of steel ($E = 210\,GPa,\ \nu = 0.3$) they have a cross section of $20\,mm$ and a friction coefficient of $\mu = 0.3$.

Our locomotive is standing in the train station. The contact between wheel and rail is loaded by the weight of the locomotive only. We can compute the contact stresses by the analytic Hertz solution as well as by the FE method. When the locomotive starts to pull the wagons the whole traction is transmitted through the wheel/rail contact. The maximum traction can be limited by the power of the engine, by the friction pair wheel/rail or the material properties of wheel and rail. We will have a closer look. And there is an other question : Can a contact slide and stick at the same time ?

In this miniproject we will compute the stresses near a wheel/rail contact with different loadings and compare them to the material resistance. We take a closer look at a tangentially loaded contact (when the train is excellerating). Finally we study the influence of the FE mesh refinement on the results.

Code utilisé : *ZeBuLoN*

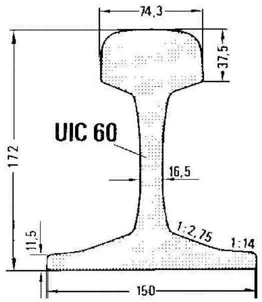
Mots-clés : *contact mechanics, normal load, tangential load, mesh refinement*



Methodology

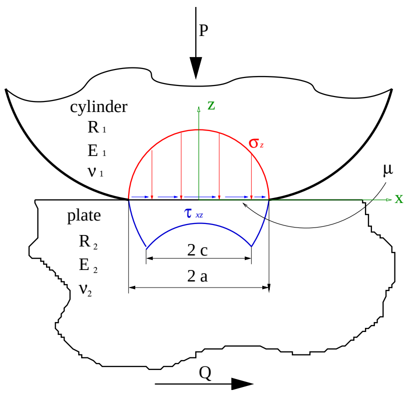
About the locomotive, its wheel and the rail

This is the train called "Transalpin". It is mainly used on the train connections Vienna–Graz and Salzburg–Graz. It's locomotive has a weight of 72 tons and stands on 8 wheels made of steel with a radius of 575 mm. The maximum load it can pull is 162 kN.



And this is the rail UIC60 with a tensile strength of 600 MPa. We assume that the material of wheel and rail starts yielding at 500 MPa. It is mainly used with highspeed trains. Even though the maximum speed of the Transalpin is only 150 km/h we take this rail. Let us assume that 20 mm of the rail head with a with of 72 mm are in contact with the wheel. The friction coefficient if $\mu = 0.3$.

About the analytic description of contact stresses



We use the cylinder/plate contact as an idealized model of the wheel/rail contact. If a cylinder and a plate are pressed together with the force (P), then the half length of contact (a) and the maximum contact pressure (p_0) can be computed with the formulas 1, 2 and 3, 4.

$$\frac{1}{E^*} = \frac{1-\nu_1^2}{E_1} + \frac{1-\nu_2^2}{E_2} \tag{1}$$

$$\frac{1}{R} = \frac{1}{R_1} + \frac{1}{R_2} \tag{2}$$

$$a = \left(\frac{4PR}{\pi E^*}\right)^{1/2} \tag{3}$$

$$p_0 = \frac{2P}{\pi a} \tag{4}$$

The contact pressure in the whole contact is given by the following formulas.

$$x \leq a: \ \sigma_z = -p_0(1 - x^2/a^2)^{1/2} \tag{5}$$

$$x \geq a: \ \sigma_z = 0 \tag{6}$$

The maximum von Mises stress is $\sigma_{Mises} = 0.6 p_0$ at $x = 0$ and $z = 0.78a$.

Now we add a tangential force (Q). If the contact has a coefficient of friction (μ), then globally the two bodies will stick if $\frac{Q}{P} < \mu$ and slip if $\frac{Q}{P} > \mu$. But now we look at the local contact stress σ_z. We say, that locally the two bodies stick when $\frac{\tau_{xz}}{\sigma_z} < \mu$ and that locally the two bodies slip when $\frac{\tau_{xz}}{\sigma_z} > \mu$. With this description we allow the contact to stick in some locations while slipping in others. The analytic solution will provide us with information about where the contact sticks and where it slips. The analytic solution under such conditions gives :

1. If $x < c$ the local contact is in stick condition and $\frac{\tau_{xz}}{\sigma_z} < \mu$. The local contact shear stress is given by :

$$\tau_{xz} = \mu p_0(1 - x^2/a^2)^{1/2} - \frac{c}{a}\mu p_0(1 - x^2/c^2)^{1/2} \tag{7}$$

2. If $c < x < a$ the local contact is in slip condition and $\frac{\tau_{xz}}{\sigma_z} > \mu$.The local contact shear stress is given by :

$$\tau_{xz} = \mu p_0(1 - x^2/a^2)^{1/2} \tag{8}$$

The relation between stick zone (c) and contact zone (a) is given by equation 9.

$$\frac{c}{a} = \left(1 - \frac{Q}{\mu P}\right)^{1/2} \tag{9}$$

On the input files for FE computations

The mesher input file `wheel-rail.mast` : it contains the geometry of the wheel/rail assembly and the rules on how the geometry will be represented in a FE mesh by the meshing program. The file can be visualized by typing `Zmaster wheel-rail.mast`. With the program `Zmaster` the file can be edited and executed.

The mesh geometry file `wheel-rail.geof` : when the file `wheel-rail.mast` is executed this file is obtained. It contains the position of a the mesh nodes, the definition of the elements, all groups of elements and groups of nodes.

The computation input file `wheel-rail.inp` : contains commands to transform mesh geometry files (preprocessing), to control FE computations, to transform the results of FE computations (postprocessing).

The material behaviour file `steel.mat` : contains the elastic properties of steel.

The contact behaviour file `contact.mat` : contains the friction coefficient. The FE code can be used with a non constant friction coefficient. In this example the friction coefficient is constant and equal to $\mu = 0.4$.

Proposed work

A look at the input files

Open the file `wheel-rail.mast` using Zmaster, play with the meshing definitions and make the mesh using the button "Mesh Domain". Now add the upper half of the wheel to the mesh. You need to add an arc, define the "edge nodes", make a meshing domain.

Examine the resulting `wheel-rail.geof` (geometry, groups of nodes/elements). Exit Zmaster and edit the file `wheel-rail.geof`. Examine the node definitions, element definitions, groups of elements (`elset`) and groups of nodes (`nset`). See if the elements are linear/quadratic.

Edit the computation input file `wheel-rail.inp` .

1. The first part is a meshing procedure (`****mesher`). It is used to produce a linear version of the mesh `wheel-rail.geof`. This precedure can be started by typing `Zrun -m wheel-rail.inp`.
2. The definition of the FE computation starts with `****calcul`.
   - At first the `***mesh` is specified.
   - The part `***resolution` is used to specify the evolution of the loading. `**sequence 1` specifies that only one loading sequence is computed - the normal loading of the contact. Later we will add a second sequence - that of tangential loading of the contact. The load is linearly applied so that the full loading is achieved after 1 second (`*time 1.`). The FE problem is solved at 2 instances until the full loading is applied (`*increment 2`) - at the instances 0.5s and 1.0s. The part `**automatic_time` specifies what is done when conversion is not achieved in an increment.
   - In the part `***bc` the boundary conditions are defined. With `**impose_nodal_dof` the displacement of a group of nodes is defined in space (directions `U1, U2`) , the amount is specified, the time dependance is indicated by the name of a table containing the data. With `**impose_nodal_reaction` the force acting on each node of a group of nodes is set. As before the group of nodes, direction, amount and time need to be specified.
   - The part `***equation` allows us to specify a law for the common displacement of nodes. The type `**mpc1` specifies that the group of nodes `LIMI_wheel` has a common displacement in direction `U2`.
   - In the part `***table` the time tables used in the boundary conditions are defined.
   - The `***contact` is defined with the `**solve_method` and some solver parameters. The definition of the contact `**zone` is done

by assigning 2 surfaces as a *impactor / *target pair. The coefficient of friction is specified in the file contact.mat. The *warning_distance is that distance, within which the *target surface searches for nodes of the *impactor surface at the start of a computation increment. If a *impactor/*target pair is more distant than the indicated value, the contact will not be recognized in the increment.

– In the ***material part the material file is indicated.
– Next the ***output of the computation is specified. In our computation **extra contact data will be as well available as the standard **contour data, both can be visualized later in colorful pictures. The **test part specifies that an ASCI file with the name "wheel-rail.test" will be written. *nset_var means that the force acting on the indicated group of nodes in the indicated direction will be written out.

Edit the material behaviour file steel.mat and the contact behaviour file contact.mat and check their content.

When the locomotive is standing on the rail

Use the information on the train to complete the computation file wheel-rail.inp. One line for the horizontal displacement of the wheel needs to be added at **impose_nodal_dof. One line for the vertical loading of the wheel needs to be added at **impose_nodal_reaction. Then start the computation by typing Zrun wheel-rail. Look at the result by typing Zmaster wheel-rail. Examine the stresses, strains and contact parameters in contour maps and plot the contact stresses on the concerned groups of nodes.

Use the analyic solution of the Hertz contact to compute the contact halfwidth (a) and the peak contact pressure (p_0) and the maximum von Mises stress. Compare these values to the FE computation result.

Changing the computation find out - how much weight can be put on the wheel without irreversibly deforming wheel and rail ?

When the locomotive is accelerating or breaking

Now edit the computation file to add a second **sequence with a tangential loading of the contact. Your need to change *time and *increment too. You need to complete the sections ***table and ***bc, then execute the computation. Regulate the computation to see what shear and normal pressures persist in the contact when the locomotive pulls the train with its maximum force. Compute the corresponding analytic contact shear stress. Compare analytic and FE result. Then do the same for the case or an emergency brake. The wheels are blocked and slide on the rail.

Do wheel or rail get irreversibly deformed now ?

How the mesh refinement influences the FE computation results

Don't forget to note down the results you have gotten. Now we change the mesh from quadratic to linear elements using the preprocessing routine of the computation file. Type Zrun -m wheel-rail. Verify that the mesh in the file wheel-rail.geof is linear now. Repeat the computation now and compare the result to that with a quadratic mesh. Now the file wheel-rail.mast will be changed using Zmaster. Change the definitions of mesh domains to get a much coarser mesh of the contact. Use this file in a computation and compare the result the the previous ones.

How do normal and shear stresses in the contact respond to the mesh refinement ?