UNIVERSAL MECHANISM 9

User's manual

Simulation of accumulation of rolling contact fatigue damage in railway wheels and rails

Extension of Universal Mechanism software for modelling of the processes of accumulation of the rolling contact fatigue damage in railway vehicle wheels and rails are considered

2020

Contents

25. Simulation of accumulation of rolling contact fatigue damage in railway wheels and rails

25.1. General information

Program package **Universal Mechanism (UM)** has been developed at the Laboratory of Computational Mechanics of Bryansk State Technical University (BSTU), Russia.

Modules **UM RCF Wheel (UM Rolling Contact Fatigue of Wheel)** and **UM RCF Rail (UM Rolling Contact Fatigue of Rail)** represents the additional program tools integrated in **UM Simulation** program. The indication of presence of the modules in current configuration of **UM** is sign «+» in the corresponding line of **About** window (command menu **Help | About program…**), Figure 25.1.

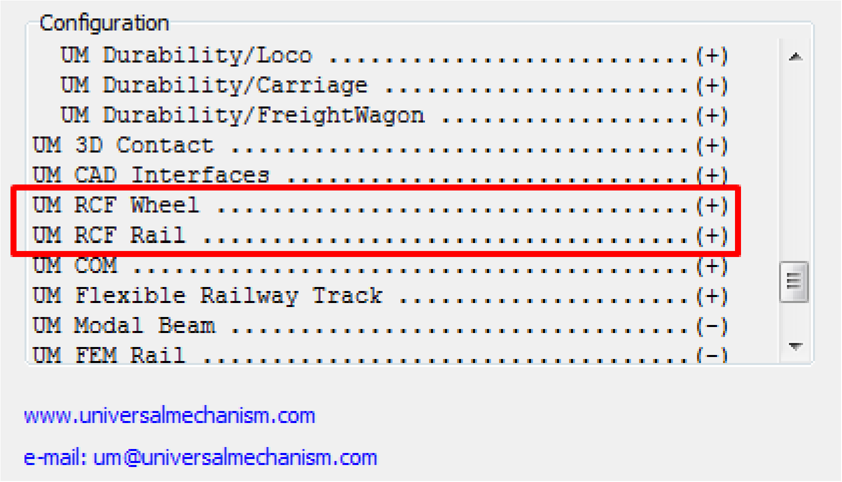


Figure 25.1 List of modules in **About program** window

Modules **UM RCF Wheel** and **UM RCF Rail** are developed for simulation of the process of accumulation of rolling contact fatigue (RCF) damage in railway wheels and rails. The modules may be used for implementing of multivariant comparative calculations, for example for solution of the problem of the wheel tread or rail profile optimization according to RCF criterion.

Module UM RCF Wheel allows the user:

- to define the velocity of the accumulation of RCF damage in wheels with the different tread surfaces by using the data which are obtained by simulation of the railway vehicle dynamics by means of the **UM Loco** (Chapter 8);
- to use the four RCF criteria for modelling of the accumulation of RCF damage:
  - criterion of amplitude of maximum shear stress;
  - Dang Van criterion;
  - Sines criterion;
  - combined criterion;
- to take into account wheel profile wear-out effect on the rate of accumulation of RCF damage using the **UM Loco/Wheel Profile Wear Evolution** tool (Chapter 16);

- to take into account wheel steel hardness effect on the rate of accumulation of RCF damage;
- to take into account residual stresses effect on the rate of accumulation of RCF damage;
- to represent processes of the stress modelling and the damage accumulation in the wheel with the help of graphic interface including isolines and coloring.

Module UM RCF Rail allows the user:

- to define the velocity of the accumulation of RCF damage in rails with the different tread surfaces by using the data which are obtained by simulation of the railway vehicle dynamics by means of the **UM Loco** (Chapter 8);
- to use the four RCF criteria for modelling of the accumulation of RCF damage:
  - criterion of amplitude of maximum shear stress;
  - Dang Van criterion;
  - Sines criterion;
  - combined criterion;
- to take into account rail profile wear-out effect on the rate of accumulation of RCF damage using the **UM Loco/Rail Profile Wear Evolution** tool (Chapter 16);
- to take into account residual stresses effect on the rate of accumulation of RCF damage;
- to represent processes of the stress modelling and the damage accumulation in the rail with the help of graphic interface including isolines and coloring.

For modelling of the process of RCF damage accumulation in the **UM RCF Wheel** and **UM RCF Rail** modules the following assumptions are used:

- the materials of the contacting bodies are uniform, isotropic and elastic;
- the geometric forms of the contact surfaces exert a determinative influence on conditions in contact and these conditions weakly depend from the structural forms of wheels or rails;
- the temperature level arising in the wheel or rail does not influence on a stress state and mechanical properties of their materials considerably.

For operation of the **UM RCF Wheel** module the **UM Loco** module and **UM Loco/Wheel Profile Wear Evolution** tool in the current configuration of **UM** are required.

For operation of the **UM RCF Rail** module the **UM Loco** module and **UM Loco/Rail Profile Wear Evolution** tool in the current configuration of **UM** are required.

This manual is compiled as follows. Practical issues are discussed in Sect. 25.4 and Sect. 25.6. More detail information about RCF and approaches to modelling of the accumulation of RCF damage, the description of the algorithms involved in the programs is considered in Sect. 25.2, Sect. 25.3 and Sect. 25.5.

25.2. Rolling contact fatigue

A reason of some widespread defects of the machine elements is rolling contact fatigue. Interaction of a wheel and rail is characterized by the following features. This is a very high pressure (more than 1000 MPa) in the contact that causes a plastic deformation of the subsurface layers under the big friction coefficients. Rolling the wheel on the rail especially at moving in curve sections is accompanied by the considerable longitudinal and lateral creep that exerts a large damaging effect. Besides, cases of the full sliding of the wheel along the rail are possible during braking. It results to thermomechanical damage of the rolling surface and rather fast spalling of the surface damaged layers under acting of the contact stresses. Due to high levels of the contact stress and temperature arising on the frictional surfaces of wheel the cracks appear. The cracks are typically either parallel or perpendicular to the wheel tread. With continued rolling the cracks grow, and when the material is weakened to the point of failure the spalls are formed. A network of the cracks may progress into the RCF shells [1].

25.2.1. Rolling contact fatigue damage of railway wheels

Surface RCF damage (Figure 25.2a) become apparent as inclined parallel cracks placed on the rolling circle and spalls from them. This type of defect is named thread checks. At the first stage hair-like cracks inclined about under equal angles to generatrix of the wheel cylindrical surface appear on the rolling surface. Angle of inclination of the cracks in different cases may change from 20–30 to 45 degrees. Section where such cracks are discovered may be situated at 25–40 mm from outside wheel face, on the middle of tread, in the region adjoining to flange. Within process of increasing of the run the small cracks begin to unite forming local spalls. Later the local spalls may generate a chain of entire spalls, which are objectionable defect and the wheel is liable to turning [1].

a)

b)

Figure 25.2. RCF damage of railway wheel: a – shallow; b – subsurface

Subsurface RCF damage (Figure 25.2b) appear at the depth 3–6 mm and more and cracks commonly develop towards tread causing spalling in form of shells. However, the crack may develop inside of the wheel that, in case of many times turned rim, under action of impact load may result in destruction of the wheel [2].

A spectrum of the forces dependent from wheel load, position of the wheel on rail, wheel and rail profiles, friction forces acts on the wheel tread. Unfavorable combination of the worn wheel and rail profiles at some positions of the wheel on rail is the factor which contributes to arising normal and tangential stresses and thus to arising rate of the damage accumulation [1].

One of the features of appearing subsurface RCF damage of wheel is that rim material under action of normal and tangential traction is in complicated stress state, when directions of the main stresses change during cycle and the stress components reach maximum values at the same time [3].

Often the subsurface damages progress in the presence of the inclusions (voids, blowholes) as well as of heterogeneities of structure. Modelling of an elastoplastic stress state using finite element (FE) analysis shows that local stresses increase in the region close to the defect and as a result the plastic deformation has place [4]. It causes residual stresses that may be tensile ones. It furthers the crack nucleation [1].

However RCF damages may arise in absence of evident material defects [5] at the depth about 4 mm from tread where the equivalent stresses reach the maximum value exceeding the fatigue limit of material at the considered point. Such damages arise under large stresses in cases of unfavorable combination of the worn wheel and rail profiles especially of the "trough-shaped profile of wear" [1], when the normal stresses under wheel load 120 kN increase from 1100 to 2300 MPa and more. The contact stresses also increase with decreasing the wheel diameter after multiple reprofiling of the wheel.

25.2.2. Rolling contact fatigue damage of rails

Contact fatigue damages caused by cyclic loads during the interaction of the wheel and rail are subdivided into those arising from the surface and emerging below the surface.

The initiation of fatigue damage in a rail occurs at the point where the maximum stress is reached. When this maximum stress causes increasing of the cyclic plastic deformation, the resulting damage will be contact fatigue damage. Since the point with the maximum stress in the contact between the wheel and the rail at high friction coefficients is located on the rolling surface, the corresponding type of contact fatigue damage emerging on the surface is called head check (parallel cracks on the rail head fillet) or squat, depending on the origin of the stress causing fatigue [6].

With decreasing the friction coefficient in contact the point with the maximum stress is lowered into the subsurface layer and the stress can reach the maximum in the region where there are inclusions in the material. The type of contact fatigue damage that occurs below the surface of a rail is known as shelling. Contact fatigue cracks occurring below the surface gradually grow towards the surface and cause chipping of metal of the rail head, usually from the rolling surface of the inner rail of a curve or from the fillet of the head of the outer rail, especially when it lubricated [6]. The type of such fatigue damage is shown in Figure 25.3a.

Spalling of the rail head (Figure 25.3b) is caused by RCF damage as a result of number of factors, such as non-conforming wheel and rail profiles, high adhesion coefficient, rail overloading due to the speed of movement that does not correspond to the equilibrium speed, increasing of the contact stresses due to false flange [6].

a) b)

Figure 25.3. RCF damage of the rail[1]: a – subsurface; b – superficial

The fatigue cracks can also occur under high loads in the presence of non-metallic inclusions or defects in the material, which are stress concentrators that cause crack initiation. The development of such cracks can lead to breaking of the rail and, as a result, to derailment.

25.2.3. Factors effecting the rolling contact fatigue of wheels and rails

The RCF resistance is related to plasticity, hardness, wearing rate and other important aspects. But first of all RCF durability of the railway wheels and rails depends on chemical composition of the steel.

Analysis shows that the compositions of the wheel steels in different countries do not differ greatly. The chemical composition of some of them is shown in Table 25.1. Such elements as nickel, copper are contained in an amount 0.25–0.3 wt%, molybdenum 0.08 wt%, phosphorus, sulfur 0.015–0.04 wt%, vanadium 0.01–0.05 wt% [7], [8], [9], [10], [11].

The carbon content has the big effect on the properties of the steel. It is contained in an amount 0.44–0.77 wt%. Increasing the carbon content increases the hardness, wear resistance and RCF resistance. Higher carbon content makes the wheel more sensitive to thermal effects in the case of tread braking, which result in thermomechanical damage.

The alloying elements such as manganese (Mn), silicon (Si), chromium (Cr) and other ones also have the effect on the ability of steel to resist RCF.

It is noted in the paper [12] the hardness is a factor having the biggest effect on the RCF resistance of the wheel steels.

[1] Photos courtesy of Prof. Zakharov S.M.

Table 25.1

Chemical composition of the wheel steels

Region	Standard	Grade of steel	Cast composition, wt%			
			C	Si	Mn	Cr
Russia, CIS countries	GOST 10791-2011	L	0.48-0.54	0.45-0.65	0.8-1.2	≤0.25
		2	0.55-0.63	0.22-0.45	0.5-0.9	≤0.3
		T	0.62-0.7	0.22-0.65	0.5-1.0	≤0.4
North America	AAR M-1071 M-208 2012	A	0.47-0.57	0.15-1.0	0.6-0.9	≤0.25
Europe	EN 13262:2004 +A2:2011	ER7	≤0.52	≤0.4	≤0.8	≤0.3
China	TB/T 2708 1996	CL60	0.55-0.65	0.17-0.37	0.5-0.8	≤0.25
Japan	JIS E5402_I 2015	C55	≤0.58	≤0.4	≤0.9	≤0.3

World manufacturers provide a wide variety of categories of rails of different types for different operating conditions. The steels with a high carbon content, which have a pearlite structure or close to pearlite structure, are used for the manufacture of rails. An increase in the strength properties of rails is achieved by alloying steels with manganese, nickel, chromium, molybdenum and vanadium, as well as using various manufacturing and heat treatment technologies. Table 25.2 shows the characteristics of rail steels of various types [6].

Table 25.2

Characteristics of rail steels

Types of rail steels	C, %	Microalloying	Tensile strength, MPa	Hardness, HB
Medium carbon	0,45 – 0,60	-	690 – 820	240 – 280
Medium carbon alloy	0,50 – 0,70	-	880 – 1080	270 – 300
With a high content carbon (standard)	0,72 – 0,82	-	900 – 1020	260 – 290
High carbon (medium strength)	0,60 – 0,82	Mo, Ni and/or V	1000 – 1200	310 – 330
High carbon (microalloyed)	0,65 – 0,82	V, Nb and/or Mo	1100 – 1300	330 – 400
High carbon (heat-strengthened)	0,65 – 0,82	Cr, V and/or Nb	1130 – 1300	340 – 380
Hypereutectoid (heat-strengthened)	0,85 – 1,00	-	1300 – 1400	380 – 430
Low carbon, martensitic structure	0,20 – 0,30	-	1450 – 1650	440 – 460
Medium carbon bainite	0,30 – 0,40	Mo and/or V	1300 – 1500	380 – 450

Increasing the carbon content increases the hardness of the steel, its wear resistance and resistance to contact fatigue, but decreases the impact strength of the material. Improvements in the technology of rail steel production in recent decades have led to the fact that harder rails have sufficient ductility and the impact strength.

25.2.4. Approaches to simulation of accumulation of rolling contact fatigue damage

A high-cycle fatigue calculation is performed according to the following scheme. A criterion of the RCF failure is chosen [13]. The parameters of the variable stresses cycle in the points of subcontact layer, when they pass through the region adjacent to the contact, are computed. The values of RCF criterion in the points by using these parameters are calculated. The point with the highest value of the criterion is regarded as dangerous. It is considered that the strength is provided if the criterion value at the dangerous point is smaller a certain value. The endurance limit in torsion is taken as such value. If the approach uses the limit stress diagram – a line dividing the diagram field into areas of safe and dangerous states – it is considered that the strength is provided if the corresponding point falls in the region of the safe states.

The quantities whose values are easily determined by testing of the specimens in bending and torsion are used in such approaches.

That RCF of a railway wheel may be considered as a high-cycle fatigue is confirmed by considering statistical data about reprofiling of the wheels due to RCF damages. The average traveled distance of the W3 wheel between reprofiling given in [14] is equal to $3.02 \cdot 10^5$ km. Thus the number of the loading cycles of the contact region is equal $1.1 \cdot 10^8$. The contact patch takes different locations along the width of the wheel tread during rolling. That is why number of cycles for the dangerous point is smaller than this value with taking into account the frequency of repeating the contact in the region of location of this point.

The RCF criteria considered below take into account the stress state only. At the same time there are other factors that have a great influence on RCF generation. The effect of lubricant located on the surfaces of the wheel and rail is one such factor. As noted in the papers [15] and [16] during testing of specimens of wheel steel under contact fatigue with pure rolling (without sliding) and supplying lubricant – oil or water – RCF damage is not observed even at very high pressures.

The possibility of using curves of RCF of the Weller's curves type is regarded in some papers [1], [17]. They are presented in the form typical for the low cycle fatigue

$$N = C\sigma_{eq}^{-m}, \tag{25.1}$$

where σ_{eq} – is the chosen RCF criterion characterizing the stress state in the contact region;

N – is the number of the cycles of variable stresses until arising fatigue defects;

C and m – are the material constants obtained experimentally.

The appearance of the RCF curve is shown in Figure 25.4.

The Eq. (25.1) allows calculating the number of cycles to material failure for a given value of the criterion σ_{eq}. Use of this approach is suggested by results obtained in some papers [15], [16]. These results show the RCF curves of wheel materials tending to an inclined asymptote, but not to an *x*-axis paralleled asymptote. That is, a physical limit of the RCF does not exist for them. The notion of a limited fatigue limit can be applied to them.

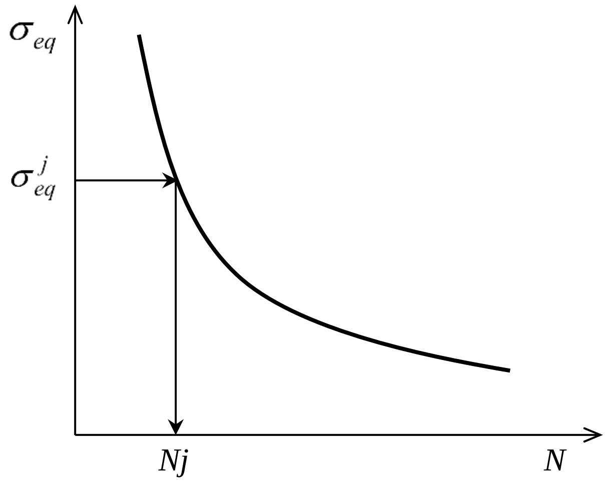


Figure 25.4. RCF curve

The RCF curve can be obtained only with help the experimental tests.

The equivalent stresses, according chosen RCF criterion, and the accumulated damage are calculated in the nodes of the wheel fragment FE model [18]. To study the accumulation of damage in the material of the wheel a linear summation model is used, which assumes that at each point of time the increment of damage does not depend on the accumulated damage. The RCF damage Q accumulated in the i-th node is calculated as

$$Q = \sum_{j=1}^{n} \frac{1}{N_i(j)}, \tag{25.2}$$

where $N_i(j)$ – is the number of the cycles of variable stresses until arising fatigue defects, when the value of criterion is σ_{eq}^{i};

n – is number of loading cycles (number of ingresses of the radial cross-section of the wheel into the region of contact).

In common practice $Q = 1$ is considered as the RCF failure condition.

25.2.5. Criteria of rolling contact fatigue damage

The peculiarity of the contact problem is as follows: the point, in which the process of damage accumulation is most intensive, is located on the surface of the contact patch or in the sub-contact layer of material where the stress state is multiaxial. When specimens are tested in tensile-compressive or bending to determine a fatigue limit, the point of maximum stress is located on the surface. For such tests the stress state is uniaxial and characterized by a normal stress σ. In torsion the stress state is biaxial and characterized by shear stress τ. Results from these tests are used to construct fatigue limit diagrams: the normal or shear stress vs. the number of cycles to specimen failure.

In the common practice it is accepted to construct the RCF curves as diagram: the maximum contact pressure p_0 vs. the number of cycles N until the appearance of the fatigue failure. The maximum contact pressure is taken as the criterion of the contact fatigue. There are several reasons why it cannot be used as a criterion for modelling of the processes of accumulation of RCF damage.

1. Use of this criterion does not take into account the creep, spin and tangential traction distributed over the contact surface and arising from the rolling of the wheels when braking or traction are applied. These factors significantly influence on the stress state.

2. The criterion cannot consider the residual stresses in the subcontact layer of the material.

3. The RCF tests are done under the high Hertzian pressures up to 3500 MPa. This gives rise to plastic deformations in the subcontact layer, which leads to a change of geometry of the contact surfaces, and a subsequent change of the contact pressure distribution.

In this connection, for modelling of the processes of accumulation of RCF damage more complex criteria accounting the peculiarities of the stress state in the contact of the wheel and rail is necessary to use.

25.2.5.1. Criterion of amplitude of maximum shear stress

The simplest criterion used in simulation of RCF damage accumulation is the amplitude of maximum shear stress τ_{max}^{a} [1]. Its suitability as a criterion is obvious when taking into account the dominant idea of the nature of fatigue failure: due to shifts of material on the planes of action of the maximum shear stresses. Through such shifts, the material becomes friable, and microcracks are formed. Repetition of this process leads to crack growth. The criterion allows taking into account the stresses caused by normal and tangential traction in the contact.

In [19], using the example of the uniaxial stress state, it is shown that the criterion allows take into account the residual stresses. The criterion does not take into account hydrostatic pressure, which plays an important part because the favorable conditions for the formation and development of the cracks are created when there is multiaxial tension in the material. The formation of cracks is more difficult with multiaxial compression.

The distribution of the values of the τ_{max}^{a} criterion in the region adjacent to the contact patch of elliptical shape is analyzed using the FE model of the parallelepiped (Figure 25.5).

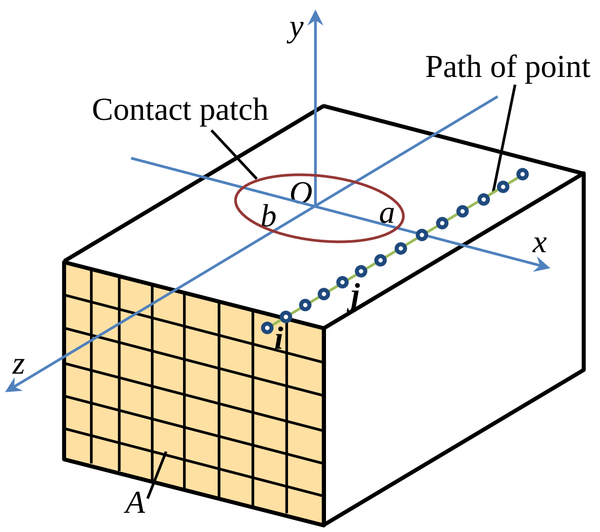


Figure 25.5. FE model of the fragment adjacent to the contact patch

The model consists of the eight-node finite elements with the edges size of 1 mm. The vertical forces are applied in the nodes located on the surface of the contact patch. Their values are determined in accordance with the elliptic law of distribution of pressures

$$p(x,z) = p_0\sqrt{1-\frac{x^2}{a^2}-\frac{y^2}{b^2}}, \tag{25.3}$$

where a, b are the semiaxes of the elliptical contact patch;

p_0 is the maximum pressure.

The coefficient of friction *f* and eccentricity of the elliptical contact patch *e* are varied during analyzing

$$e = \sqrt{1-\beta^2}, \tag{25.4}$$

where $\beta = b/a$.

The normal force equal to 120 kN and the maximum pressure p_0 1000 MPa were accepted for all variants.

The value of the criterion is determined for the nodes of the plane FE mesh *A*, which is dragged through the subcontact region along the *Oz* axis with steps equal to the length of edge of the finite element (Figure 25.5). The value of the maximum shear stress is determined for each position *j* occupied by the node *i* during the dragging of the mesh as

$$\tau_{max}^{j} = \frac{\left(\sigma_1^j - \sigma_3^j\right)}{2}. \tag{25.5}$$

In what follows, the point will be associated with node *i* of the plane FE mesh.

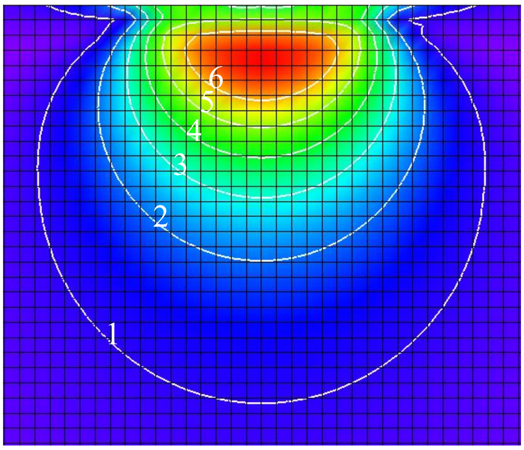
The value of the criterion for the node *i* of the plane FE mesh is calculated as the half of the difference between the maximum and minimum values of τ_{max} obtained for the positions *j*

$$\tau_{max}^{a} = \frac{\left(\tau_{max}^{max} - \tau_{max}^{min}\right)}{2}. \tag{25.6}$$

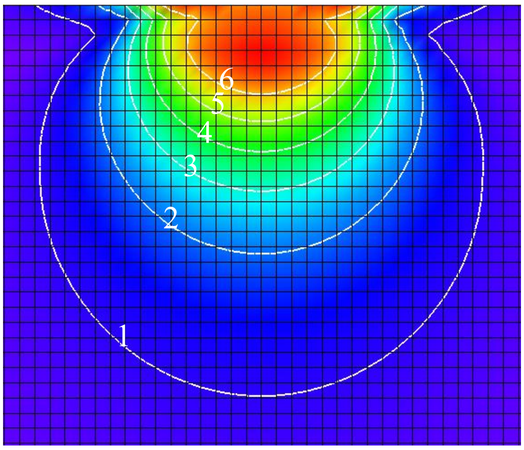
The effect of tangential traction occurring under full sliding of the wheel along the *Oz* axis is analyzed at the different values of the coefficient of friction *f*. The dangerous point, in which the criterion has the maximum value, is located at the depth of 4 mm below the contact surface at $f = 0.0$. The dangerous point is located at the depth of 3 mm below the contact surface at $f = 0.3$, in addition, the criterion value at the points located on the contact surface is close to the maximum.

The isolines of the values of the criterion in the nodes of the plane mesh dragged along the *Oz* axis through the region adjacent to the contact are shown in Figure 25.6. The dimensions of semiaxes are $a = 7.65$ mm and $b = 7.49$ mm for the ellipse of contact with the eccentricity $e = 0.2$ and $a = 11.41$ mm and $b = 5.02$ mm for the ellipse of contact with the eccentricity $e = 0.9$. In the all presented cases the number of isolines is the same – the seven, but the values of isolines are different. The maximum value of the criterion 163 MPa is obtained for the elliptical contact patch with the eccentricity $e = 0.2$ at $f = 0.0$ and 174 MPa at $f = 0.3$. The maximum value of the criterion 170 MPa is obtained for the elliptical contact patch with the eccentricity $e = 0.9$ at $f = 0.0$ and 181 MPa at $f = 0.3$.

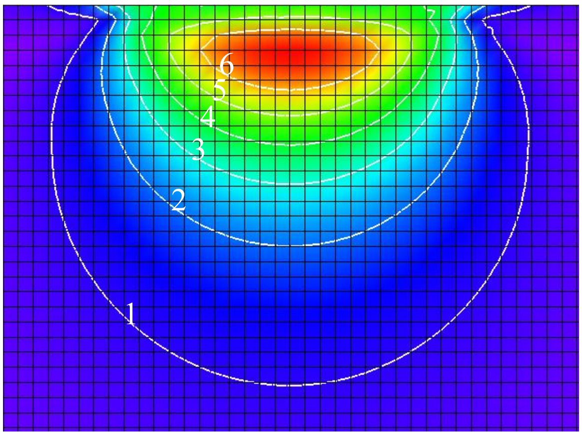
The eccentricity of the contact patch has little effect on the maximum value of the criterion.



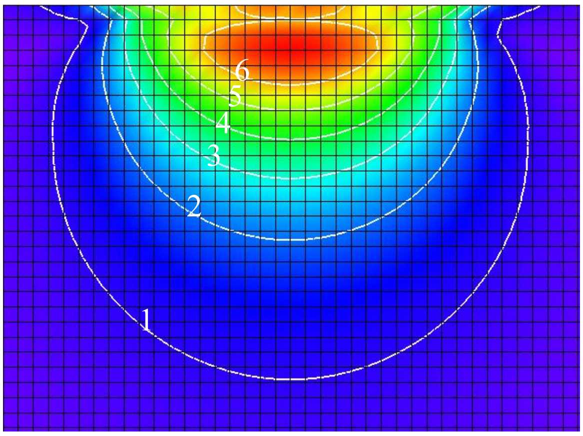
$e = 0.2; f = 0.0;$
Maximum value of τ_{max}^{a} is 163 МПа



$e = 0.2; f = 0.3;$
Maximum value of τ_{max}^{a} is 174 МПа



$e = 0.9; f = 0.0;$
Maximum value of τ_{max}^{a} is 170 МПа



$e = 0.9; f = 0.3;$
Maximum value of τ_{max}^{a} is 181 МПа

Figure 25.6. Isolines of values of the criterion of amplitude of maximum shear stress τ_{max}^{a} in the nodes of plane FE mesh dragged along the *Oz* axis through the region adjacent to the contact; 1-6 are isoline numbers

Use of the criterion of the amplitude of maximum shear stress allows considering the tangential traction. The distribution of τ_{max}^{a} is significantly changed with the increasing of the tangential traction: the region with the large values of the criterion increases and shifts to the contact surface.

25.2.5.2. Dang Van criterion

The Dang Van criterion [14] is represented by two inequalities

$$\tau_{EQ1} = \tau_a(t) + \alpha_{DV}\sigma_h(t) > \tau_e;$$
$$\tau_{EQ2} = \tau_a(t) - \alpha_{DV}\sigma_h(t) < -\tau_e, \quad (25.7)$$

where $\tau_a(t)$ is the time-dependent value of the shear stress at the specified material point and in the specified shear plane through this point. The value $\tau_a(t)$, called shear stress amplitude, is calculated as the difference between the current magnitude at time *t* of the rotating shear stress in the specified plane and its mean magnitude during one cycle;

$\sigma_h(t) = (\sigma_x + \sigma_y + \sigma_z)/3$ is the time-dependent value of the hydrostatic stress at the same material point, positive when tensile;

$\sigma_x, \sigma_y, \sigma_z$ are the normal stresses at the same point;

α_{DV} is the Dang Van coefficient – the dimensionless material parameter representing the influence of the hydrostatic stress, positive;

τ_e is the material parameter usually taken equal to the fatigue limit of the material in pure shear, positive.

The criterion is represented by the diagram in the form of two inclined lines corresponding to the inequalities (25.7). The hydrostatic stress is plotted along the abscissa axis, the "amplitude" of the shear stress along the ordinate axis and can be either positive or negative (Figure 25.7).

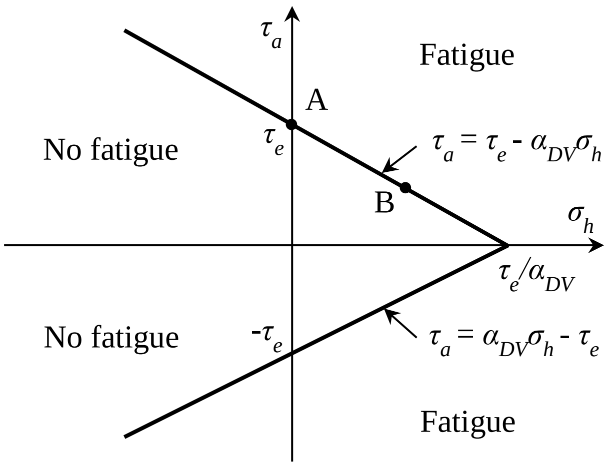


Figure 25.7. Dang Van diagram of contact fatigue under multiaxial stress state

The Dang Van coefficient is equal to the tangent of the slope angle of the cyclic fatigue diagram. The criterion is numerically equal to the length of the segment OC cut off by the line parallel to the inclined line of the diagram passing through point A with the coordinates $\sigma_h(t)$ and $\tau_a(t)$ (Figure 25.8). Then the ratio τ_e/OC can be considered as a safety factor. It is obvious, that for the cycles represented by points A and B the safety factors are the same.

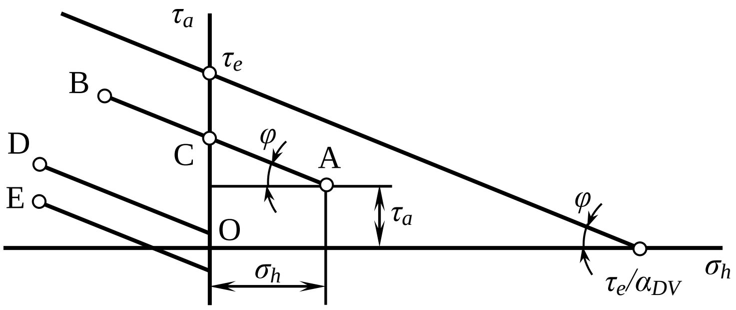


Figure 25.8. Determination of safety factor using fatigue diagram

The inclined straight line passing through the point D cuts off a short segment on the $\tau_a(t)$ axis, the safety factor takes a big value. If the line intersects the $\tau_a(t)$ axis at the point with the negative value of the criterion, the criterion is set to zero (the line passing through the point E). If only the absolute value of the shear stress $\tau_a(t)$ is taken into consideration in calculations, then only the upper part of the diagram can be used. The Dang Van criterion can be represented in the form

$$\tau_{DV} = |\tau_a(t)| + \alpha_{DV}\sigma_h(t). \tag{25.8}$$

The values $\tau_a(t)$ and $\sigma_h(t)$ are varied over time. In calculations using the FE models the time is converted to the coordinates of points *j* occupied by the node *i*, when it passes through the region adjacent to the contact (Figure 25.5).

To simplify the calculations of the $\tau_a(t)$ at the point, instead of the tangential stress at a certain point of the material acting on a certain plane passing through this point, the maximum tangential stress τ_{max}^i is used [14]. Then the mean maximum shear stress is equal

$$\tau_{mid} = \frac{\sum \tau_{max}^i}{n}, \tag{25.9}$$

where n is number of positions of the node in the one loading cycle.

The amplitude value of the shear stress is found as

$$\tau_a(t) = \tau_{max}^i - \tau_{mid}. \tag{25.10}$$

In the paper [20] the Dang Van coefficient is determined by dependence

$$\alpha_{DV} = 3(\tau_e/\sigma_e - 0{,}5), \tag{25.11}$$

where σ_e is ultimate fatigue strength of the material under a uniaxial stress state.

The recommended value of the coefficient of 0.38 is given in [21].

The distribution of the values of the Dang Van criterion in the region adjacent to the elliptical contact patch is analyzed using the FE model of the parallelepiped (Figure 25.5). The description of the model and the conditions of the test calculations are given in Sect. 25.2.5.1.

The value of the maximum shear stress τ_{max}^j is determined for each position *j* occupied by the node *i*, when the mesh *A* is dragged along the *Oz* axis (Figure 25.5). Then the mean value is determined

$$\tau_{mid} = \frac{\sum \tau_{max}^j}{n}, \tag{25.12}$$

where n is number of positions of the node *i*.

The value of the criterion for each node position is calculated as

$$\tau_{DV} = \tau_a + \alpha_{DV}\sigma_h, \tag{25.13}$$

where $\tau_a = \tau_{max}^j - \tau_{mid}$ is “amplitude” of shear stress;

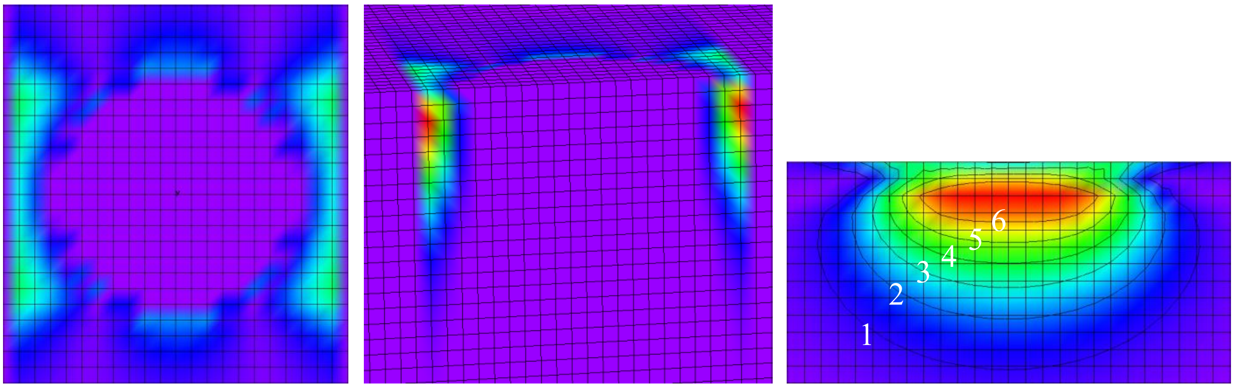
σ_h is hydrostatic stress in the node, when it occupies the position *j*.

The biggest value chosen from the values obtaining at the positions *j* is accepted as a value of criterion for node *i* of plane mesh.

The isolines of the values of the criterion in the region of the Hertzian contact with the eccentricity equal to 0.2 are shown in Figure 25.9. The semiaxes of ellipse of contact patch are

$a = 7.65$ mm and $b = 7.49$ mm. In Figure 25.9a the isolines picture on the contact surface is shown. The negative values of the criterion are obtained for all nodes located within the boundaries of the contact patch, so they are assumed to be zero.

The zero values of the criterion are obtained for the entire region located below the contact patch, as can be seen from the isolines picture for nodes located in the *yOz* plane (Figure 25.9b). Nonzero values are obtained in adjacent regions. The isolines picture for the plane FE mesh dragged along *Oz* axis through the subcontact region is shown in Figure 25.9c. The maximum value of the criterion 145 MPa is obtained for the node located on the *Oy* axis at the depth of 2 mm.



a) b) c)

Figure 25.9. Isolines of the values of the Dang Van criterion in the region of elliptical contact with the eccentricity $e = 0.2$ when the normal force is applied:
a – on the surface of contact; b – in the *yOz* plane; c – in the nodes of plane FE mesh dragged along *Oz* axis through the region adjacent to the contact; 1-6 are isoline numbers

The value of the criterion depends on the length of the selected path of the point (Figure 25.5). It determines the mean value of the maximum shear stress calculated by Eq. (25.9) and, accordingly, the "amplitude" value τ_a in Eq. (25.13).

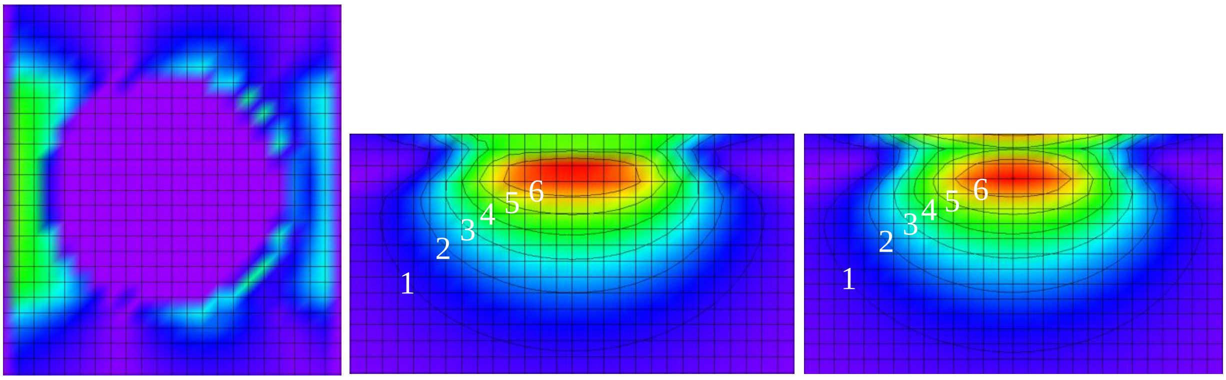
The effect of the tangential traction occurring under full sliding of the wheel along the *Oz* axis is analyzed for two values of the coefficient of friction $f = 0.17$ and $f = 0.3$. The isolines of the values of the criterion are presented in Figure 25.10: on the contact surface (Figure 25.10a) and in the plane mesh (Figure 25.10b,c). The last two figures show the same number of isolines – the seven, but the values of the isolines is different. The maximum value of the criterion at the $f = 0.17$ is 185 MPa and at the $f = 0.3$ it reaches of 211 MPa.

The same analysis was carried out for contacts with eccentricities $e = 0.6$ and $e = 0.9$. The similar pictures of distributions of the criterion values are obtained. The distributions of values of the criterion in the nodes of plane FE mesh for elliptical contact with eccentricity $e = 0.9$ are shown in Figure 25.11.

In Table 25.3 the maximum values of the criterion and the depth of the location of the point, in which this value takes place for the considered variants, are presented.

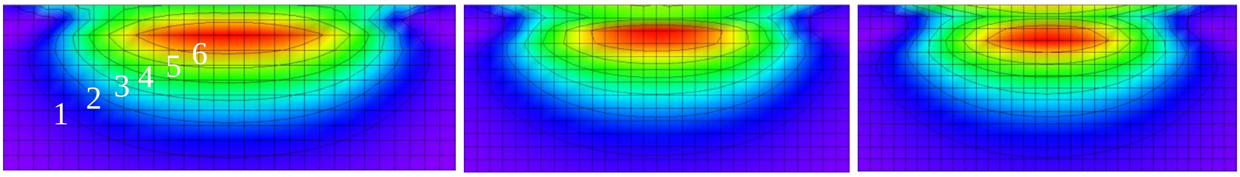
The shape of the contact patch has a small effect on the maximum value of the criterion.

Use of the Dang Van criterion leads to the increasing role of the tangential traction: with the coefficient of friction of 0.17 the maximum value of the criterion is increased by 25.3% and by 41.5% with the coefficient of friction of 0.3.



a) b) c)

Figure 25.10. Isolines of the values of Dang Van criterion in the region of elliptical contact with the eccentricity $e = 0.2$ when the normal force N and tangential traction $F_z = fN$ are applied:
a, b – $f = 0.17$; c – $f = 0.3$;
a – on the surface of contact; b, c – in the nodes of plane FE mesh dragged along *Oz* axis



a) b) c)

Figure 25.11. Isolines of the values of Dang Van criterion in the nodes of plane FE mesh dragged along *Oz* axis through the subcontact region of elliptical contact with eccentricity $e = 0.9$ when the normal force N and tangential traction $F_z = fN$ are applied:
a – $f = 0.0$; b – $f = 0.17$; c – $f = 0.3$

Table 25.3

The maximum value of the Dang Van criterion and coordinate *y* of the point, in which it takes place, depending on eccentricity *e* and coefficient of friction *f*

Eccentricity e		0,2		0,6		0,9	
		τ_{DV}, MPa	y, mm	τ_{DV}, MPa	y, mm	τ_{DV}, MPa	y, mm
Forces in contact N and F_z	$f = 0$	145	2	154	2	139	2
	$f = 0{,}17$	185	2	185	3	178	2
	$f = 0{,}3$	211	3	206	3	201	3

The Dang Van criterion allows taking into consideration the hydrostatic stress and residual stresses.

The Dang Van criterion is applied in calculations of strength at the variable stresses and multiaxial stress state. Estimation of the strength is carried out using a diagram, two branches of which are represented by inequalities (25.7). If the point representing the cycle of the variable stresses is located in the area of safe states, it is considered that strength is ensured and fatigue damage does not accumulate.

If the Dang Van criterion is used as a criterion of contact fatigue failure, it is necessary to determine the values of the criterion under the test loads for constructing RCF curve. It is represented by the left-hand side of inequalities (25.7) and can be interpreted as the length of segment of y axis cut off by a line parallel to the branch of the diagram, drawn through a point representing the cycle, in the stress scale. At the big negative hydrostatic stress inherent in the contact problem the value of the criterion is small and can even have a minus sign.

As a result, in the region with the biggest values of the stress components located under the contact patch the criterion takes the zero value. The peripheral regions adjacent to the contact region have the dominant influence on the value of the criterion.

25.2.5.3. Sines criterion

The Sines criterion [22] is described by the following inequality

$$\frac{1}{3}\sqrt{(P_1-P_2)^2+(P_2-P_3)^2+(P_1-P_3)^2}-A+\alpha\left(S_x+S_y+S_z\right)\le 0, \qquad (25.14)$$

where P_1, P_2, P_3 are the amplitude values of the cycle of the principal stresses;

S_x, S_y, S_z are the mean values of the cycle of the normal stresses;

A is a material constant proportional to the endurance limit in shear under a symmetrical loading cycle;

α is a positive dimensionless parameter of the material representing the influence of the hydrostatic stress.

The constants of the material A and α are determined experimentally.

This criterion contains a component of the type

$$[(\sigma_1-\sigma_2)^2+(\sigma_2-\sigma_3)^2+(\sigma_3-\sigma_1)^2]^{1/2}, \qquad (25.15)$$

associated with the specific strain energy due to distortion. It represents doubled modulus of the sum of the principal shear stresses [23]

$$\tau_1=\frac{\sigma_2-\sigma_3}{2},\quad \tau_2=\frac{\sigma_1-\sigma_3}{2},\quad \tau_3=\frac{\sigma_1-\sigma_2}{2}. \qquad (25.16)$$

This component represents an variable constituent of the stresses.

The last addend in the inequality (25.14) corresponds to the hydrostatic stress and represents a constant component of the stresses. This component is related to the specific strain energy due to change of volume.

The distribution of the values of the Sines criterion in the region adjacent to the elliptical contact patch is analyzed using the FE model of the parallelepiped (Figure 25.5). The description of the model and the conditions of the test calculations are given in Sect. 25.2.5.1.

The value of the octahedral shear stress is determined for each position j occupied by the node i during the dragging of the plane FE mesh A along the Oz axis (Figure 25.5)

$$\tau_{oct}=\frac{1}{3}[(\sigma_1-\sigma_2)^2+(\sigma_2-\sigma_3)^2+(\sigma_3-\sigma_1)^2]^{1/2} \qquad (25.17)$$

and the sum of the principal normal stresses

$$\sigma_{sum} = \sigma_1 + \sigma_2 + \sigma_3, \tag{25.18}$$

where $\sigma_1, \sigma_2, \sigma_3$ are the principal normal stresses.

Then the amplitude value of the octahedral shear stress is determined

$$\tau_{oct}^{a} = \frac{\left(\tau_{oct}^{max} - \tau_{oct}^{min}\right)}{2} \tag{25.19}$$

and the mean value of the sum of the principal normal stresses

$$\sigma_{sum}^{mid} = \frac{\left(\sigma_{sum}^{max} + \sigma_{sum}^{min}\right)}{2}. \tag{25.20}$$

The criterion value for the node *i* of the plane mesh is calculated as

$$\tau_S = \tau_{oct}^{a} + \alpha\sigma_{sum}^{mid}. \tag{25.21}$$

When the criterion has a negative value, then it is assumed to be zero.

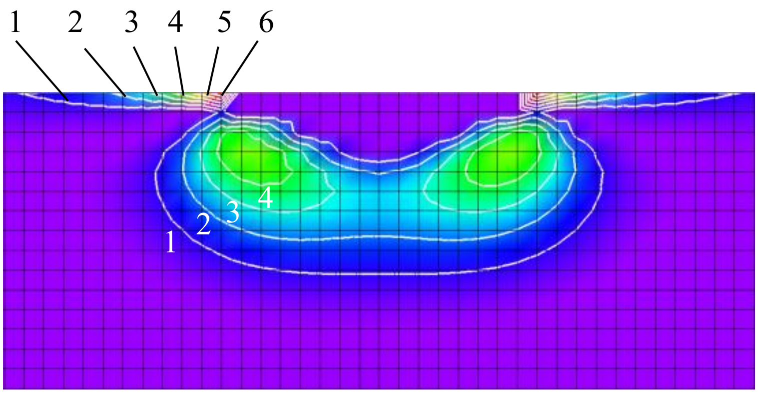
In the paper [24] the value of the coefficient α is recommended to be equal to 0.16.

The effect of the tangential traction occurring under full sliding of the wheel along the *Oz* axis is analyzed for two values of the coefficient of friction $f = 0.17$ and $f = 0.3$. The isolines of the values of the Sines criterion in the region of the Hertzian contact with eccentricity $e = 0.2$ for different coefficient of frictions are shown in Figure 25.12. The semiaxes of ellipse of contact patch are $a = 7.65$ mm and $b = 7.49$ mm. The maximum value of the criterion at the $f = 0.0$ is 42 MPa, at the $f = 0.17$ is 62 MPa and at the $f = 0.3$ is 82 MPa.

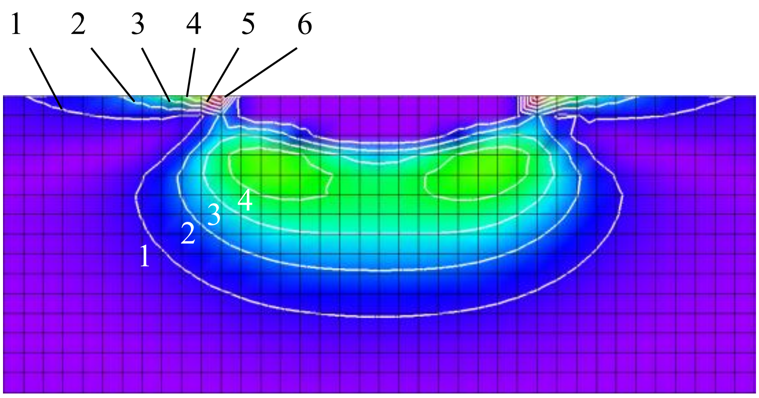
Variation of the eccentricity of the ellipse of contact patch showed the shape of the contact patch has little effect on the maximum value of the criterion.

Use of the Sines criterion leads to the increasing role of the tangential traction: with the coefficient of friction of 0.17 the maximum value of the criterion is increased by 47.6% and by 95.2% with the coefficient of friction of 0.3.

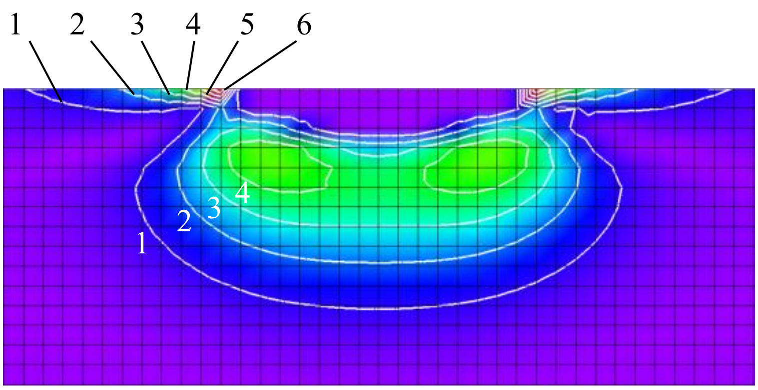
The Sines criterion allows taking into consideration the hydrostatic stress and residual stresses. The criterion, in particular, is used in the AAR standard [24] for estimation of the strength of a wheel plate by the criterion of fatigue. However, like the Dang Van criterion, in the region with the big values of the negative normal stresses located under the contact patch the Sines criterion takes a zero value. The values of the criterion in the nodes of plane mesh dragged along the *Oz* axis are formed due to the regions located beyond the bounds of the contact patch. In addition, the level of the criterion values is the order of magnitude lower than the level of stresses occurring in the contact of the wheel and rail.



a)



b)



c)

Figure 25.12. Isolines of the values of the Sines criterion in the nodes of plane FE mesh dragged along Oz axis through the subcontact region of elliptical contact with eccentricity $e = 0.2$ when the normal force N and tangential traction $F_z = fN$ are applied:
a – $f = 0.0$; b – $f = 0.17$; c – $f = 0.3$; 1-6 are isoline numbers

25.2.5.4. Combined criterion

Let the stress state at a point of a solid body be determined by the principal stresses σ_1, σ_2 and σ_3 (Figure 25.13a). Let us divide it into two states [25]. The state I with the stresses σ_h on the planes of the element causes a change of its volume (Figure 25.13b). The mean stress

$$\sigma_h = \frac{1}{3}(\sigma_1 + \sigma_2 + \sigma_3) \quad (25.22)$$

is called the hydrostatic stress. It causes uniform tension or compression.

The state II with the stresses

$$\begin{aligned} \sigma_1' &= \sigma_1 - \sigma_h, \\ \sigma_2' &= \sigma_2 - \sigma_h, \\ \sigma_3' &= \sigma_3 - \sigma_h \end{aligned} \quad (25.23)$$

causes distortion of the element (Figure 25.13c).

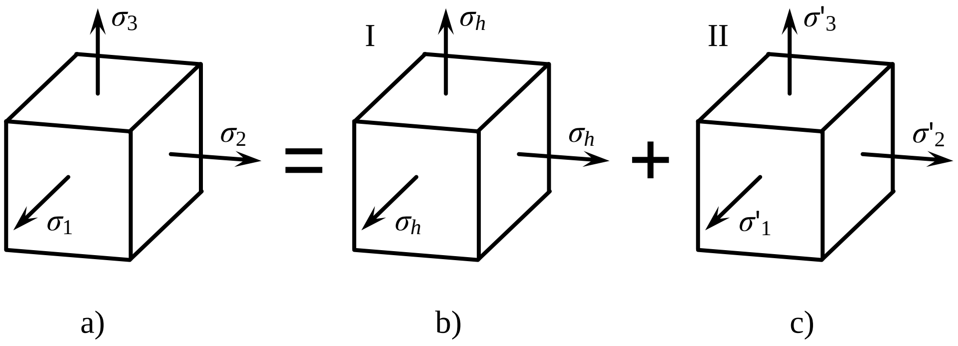


Figure 25.13. Separation of the stress state at a point by two:
I – causing change of volume; II – causing distortion

In accordance with this division the specific strain energy due to distortion of the element is

$$U_0^F = \frac{1+\nu}{6E}[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2], \quad (25.24)$$

and the specific strain energy due to change of its volume is

$$U_0^V = \frac{1-2\nu}{6E}(\sigma_1 + \sigma_2 + \sigma_3)^2, \quad (25.25)$$

where E is tensile modulus; ν is Poisson's ratio.

Taking into account that in the case of uniform compression the bodies are not destroyed Mises excluded the strain energy due to change of volume from consideration and obtained a criterion that takes into account the strain energy due to distortion

$$\sigma_{Mises} = \frac{1}{\sqrt{2}}[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2]^{1/2}. \quad (25.26)$$

The ratio between the energies depends on the stress state and can vary within wide limits. With pure shear the strain energy due to change of volume is zero. For uniaxial tension the energies are equal

$$U_0^F = \frac{1+\nu}{6E}2\sigma_{tens}^2, \quad (25.27)$$

$$U_0^V = \frac{1-2\nu}{6E}\sigma_{tens}^2. \quad (25.28)$$

The strain energy due to distortion is 6.5 times larger than the strain energy due to change of volume, when the Poisson's ratio ν = 0.3.

If the stresses equal in magnitude $\sigma_1 = \sigma_2 = \sigma_3 = \sigma_h$ act on the element of a body under consideration, the strain energy due to distortion and, hence, the equivalent stress σ_{Mises} is zero, regardless of whether they are tensile or compressive.

Davidenkov and Fridman proposed a unified strength theory that takes into account the mechanical state of material [26]. The material state diagram is constructed in the coordinates: the equivalent stress determined by the second theory of strength vs. the maximum shear stress (Figure 25.14).

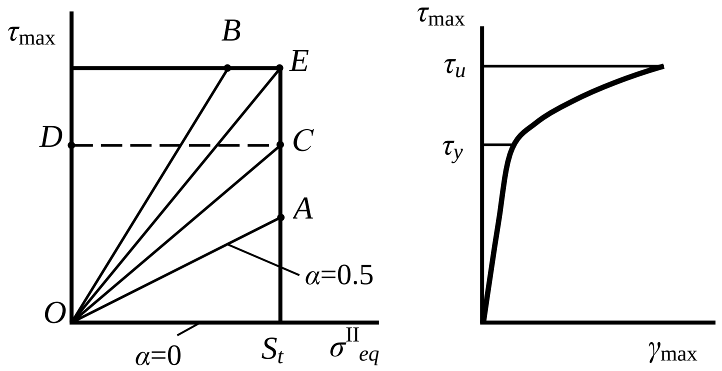


Figure 25.14. Diagram of a mechanical state of material

The stress state is characterized by the ratio $\alpha = \tau_{max}/\sigma_{max}^{II}$. The vertical line of the diagram cuts off a segment on the abscissa equal to the true resistance to the rupture S_t. If a ray whose slope angle is determined by the ratio of α intersects this line, the material is destroyed by disruption. Uniform tension is characterized by a ray passing along the abscissa axis. It represents the most severe loading, in which even plastic materials are destroyed by disruption. The contradictory result is obtained with estimation of this case using the theory of Mises – the equivalent stress is zero. The reason for this is the exclusion of the strain energy due to change of volume at the uniform tension from the analysis.

On the other hand, the contact problem has its own peculiarity. The uniform compression predominates at the points of the region adjacent to the contact. The ratio of energies changes in favor of the strain energy due to change of volume. This can be seen by the example of contact between the wheel and the rail. Let us take the following initial data: the diameter of running circle of the wheel of 950 mm, the radius of curvature of the rail profile of 500 mm, the load from the wheel on the rail is equal to 120 kN.

The stresses in the point located on the surface at the center of the elliptical contact patch are equal

$$\sigma_x = -0{,}797 p_0;\ \sigma_y = -0{,}8036 p_0;\ \sigma_z = -p_0,$$

where $p_0 = 1056.6$ МПа is the maximum pressure.

The strain energy due to change of volume is 26 times greater than the strain energy due to distortion. It turns out that the component, which is many times greater than the strain energy due to distortion, is not used when estimating the strength.

The Dang Van criterion (Sect. 25.2.5.2) and Sines criterion (Sect. 25.2.5.3) take into account the hydrostatic stress. In the Dang Van criterion the hydrostatic stress is multiplied by the coefficient α_{DV}, which is assumed to be 0.38 for wheel steels. At the high hydrostatic stress of the order of -916 MPa as in the considered example for points located in the region adjacent to the contact this negative term exceeds the absolute value of the "amplitude" of the maximum shear stress, which leads to the negative value of the criterion. The Sines criterion can also have a negative value for points located in the region adjacent to the contact.

The developers of the **UM RCF Wheel** and **UM RCF Rail** modules offer the combined criterion of contact fatigue obtained on the basis of estimation of the strength of a small element of body, on the planes of which the stresses causing its distortion and change of its volume act [27].

The criterion theories of strength are applied according to the following scheme.

The uniaxial stress state, which is equivalent to the multiaxial stress state, is determined using the appropriate criterion. Tresca or Mises criterion can be used to estimation of the strength of element subjected to the action of stresses that cause its distortion. The Mises theory of strength is preferable since it takes into account all three principal tangential stresses. Then the equivalent uniaxial stress is

$$\sigma_{eq}^{F} = \frac{1}{\sqrt{2}}[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2]^{1/2}. \quad (25.29)$$

The first or second theory of strength [25] can be applied to estimation of the strength of the element, on the planes of which the hydrostatic stress acts. For alloy steels the second theory of strength is applied.

Then the equivalent stress of uniaxial tension is

$$\sigma_{eq} = \sigma_1 - \nu(\sigma_2 + \sigma_3), \quad (25.30)$$

or, taking into account $\sigma_1 = \sigma_2 = \sigma_3 = \sigma_h$, we obtain

$$\sigma_{eq}^{V} = (1 - 2\,\nu)\sigma_h. \quad (25.31)$$

Then the uniaxial stress, which is equivalent to the multiaxial stress state determined by the principal stresses σ_1, σ_2 and σ_3, is equal

$$\sigma_{eq} = \frac{1}{\sqrt{2}}[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2]^{1/2} + (1 - 2\,\nu)\sigma_h. \quad (25.32)$$

The distribution of the values of the equivalent stress in the region adjacent to the elliptical contact patch is analyzed using the FE model of the parallelepiped (Figure 25.5) described in Sect. 25.2.5.1. The value of the equivalent stress σ_{eq} is determined for each position *j* of the node *i* using Eq. (25.32). The set of their values determines the cycle of variable stresses for the node *i*. The characteristics of the cycle of variable stress, which are equivalent to the multiaxial stress state, are determined as

$$\sigma_m = \left(\sigma_{eq}^{max} + \sigma_{eq}^{min}\right)/2, \quad (25.33)$$

where σ_m is the mean value of the stress of the cycle;

$$\sigma_a = \left(\sigma_{eq}^{max} - \sigma_{eq}^{min}\right)/2, \quad (25.34)$$

where σ_a is the amplitude value of the stress of the cycle.

In calculations in fatigue the Haigh diagram of limit stresses is used. Serensen and Kinasoshvili [28] proposed an approximation of this diagram (Figure 25.15).

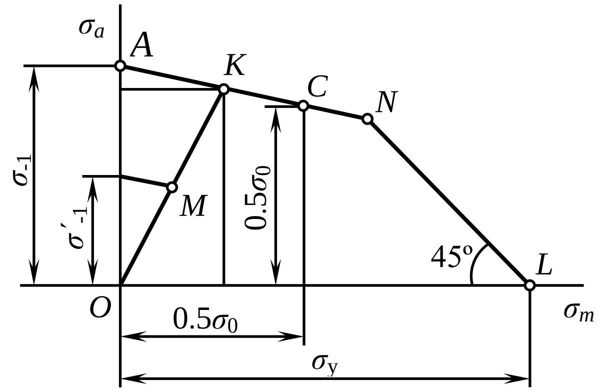


Figure 25.15. Serensen-Kinasoshvili approximated diagram

The segment limiting strength due to fatigue failure is constructed by the results of two tests. The tests with the symmetrical cycle allow obtaining the limit of endurance σ_{-1}. On the diagram it is represented by the point *A* (Figure 25.15). Testing with a pulsation cycle give the value of the endurance limit σ_0, which is represented by the point *C* with coordinates

$$\sigma_m = \frac{\sigma_0}{2}, \sigma_a = \frac{\sigma_0}{2}. \tag{25.35}$$

The straight line, which approximates the limiting curve, is drawn through the points *A* and *C*. The tangent of the slope angle of this line is

$$\psi_\sigma = \frac{2\sigma_{-1} - \sigma_0}{\sigma_0}. \tag{25.36}$$

The coefficient ψ_σ is called the coefficient of material sensitivity to the asymmetry of the cycle. The values of the coefficient ψ_σ depending on the ultimate strength of the material σ_u are shown in Table 25.4.

Table 25.4

The coefficient of sensitivity of material to the asymmetry of the cycle

σ_u, MPa	350-550	520-750	700-1000	1000-1200	1200-1400
ψ_σ	0	0,05	0,1	0,2	0,25

According to the data from Table 25.4 as a result of linear approximation using the least squares method the dependence is obtained

$$\psi_\sigma = 0{,}000301\sigma_в - 0{,}141. \tag{25.37}$$

The introduction of the coefficient ψ_σ allows to express the parameters of the cycle, which is represented by the point *K*, through the parameter of the symmetric cycle σ_{-1}. Similarly, the parameters of the cycle, which is represented by the point *M*, can be expressed through the parameter of the equivalent symmetric cycle σ'_{-1}.

Then the ratio

$$\frac{\sigma_{-1}}{\psi_\sigma \sigma_m + \sigma_a} \tag{25.38}$$

can be considered as a fatigue strength coefficient of safety, and

$$\sigma_{RCF} = \psi_\sigma \sigma_m + \sigma_a \tag{25.39}$$

as the criterion of contact fatigue.

The distribution of the values of equivalent stress and combined criterion in the region adjacent to the elliptical contact patch with eccentricity $e = 0.2$ is analyzed using the FE model of the parallelepiped (Figure 25.5). The description of the model and the conditions of the test calculations are given in Sect. 25.2.5.1.

The isolines of the values of equivalent stress in the *yOz* plane in the region adjacent to the Hertzian contact and on the surface ones are shown in Figure 25.16a. The maximum value of equivalent stress 480 MPa is obtained for the node located at the depth of 4.5 mm under the contact surface. The region with negative values of the stresses σ_{eq} is located near the center of the contact. In the center of the contact they reach of -118 MPa.

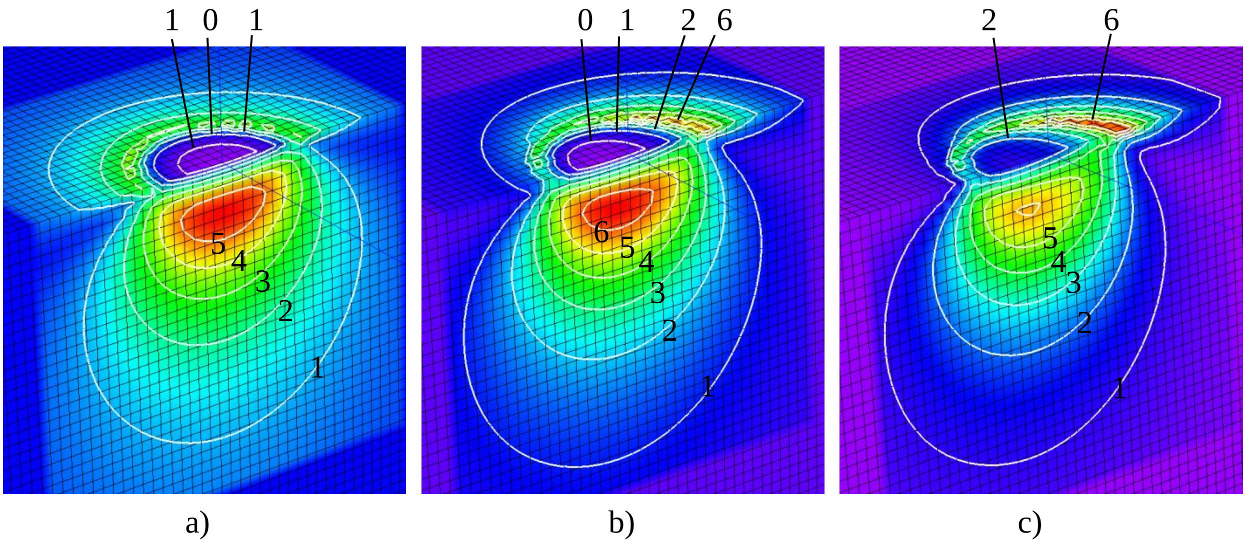


Figure 25.16. Isolines of the values of equivalent stress σ_{eq} in the *yOz* plane in the region of elliptical contact with eccentricity $e = 0.2$ when the normal force *N* and tangential traction $F_z = fN$ are applied: a – $f = 0.0$; b – $f = 0.17$; c – $f = 0.3$; 1-6 are isoline numbers

The effect of the tangential traction occurring under full sliding of the wheel along the positive direction of the *Oz* axis that are connected by direct proportionality with the pressures (25.3) is analyzed for two values of the coefficient of friction $f = 0.17$ and $f = 0.3$. The isolines of the equivalent stresses are shown in Figure 25.16b,c. The stress at the point located below the contact surface increases insignificantly with increasing the coefficient of friction: up to 487 MPa at the $f = 0.17$ and up to 507 MPa at the $f = 0.3$. For considered cases this point is shifted along the *Oz* axis by 1.05 mm and 2 mm, and by 4 mm along the *Oy* axis respectively. The equivalent stress on the contact surface increases sharply from -118 to 443 MPa at the $f = 0.17$ and up to 588 MPa at the $f = 0.3$. The point with the biggest stress is shifted to the contact surface and located on the trailing edge of contact.

These lines have the same maximum order of isolines – the seventh, but the values of the isolines are different. The maximum value of the positive equivalent stress for case with $f = 0.17$ is 560 MPa. At the coefficient of friction equal to 0.3 the point with the maximum positive stress of 650 MPa is located on the contact surface.

The set of values of equivalent stress at the points j occupied by the node i when dragging the plane mesh through the subcontact region (Figure 25.5) represents the cycle of variable stresses for this node. The maximum and minimum stresses as well as the mean σ_m (25.33) and amplitude σ_a (25.34) values is determined for the obtained cycle. The value of the criterion is obtained by substituting them in the Eq. (25.39). The isolines of the values of the combined criterion for the plane mesh A, obtained when applying the previously considered loads, are shown in Figure 25.17.

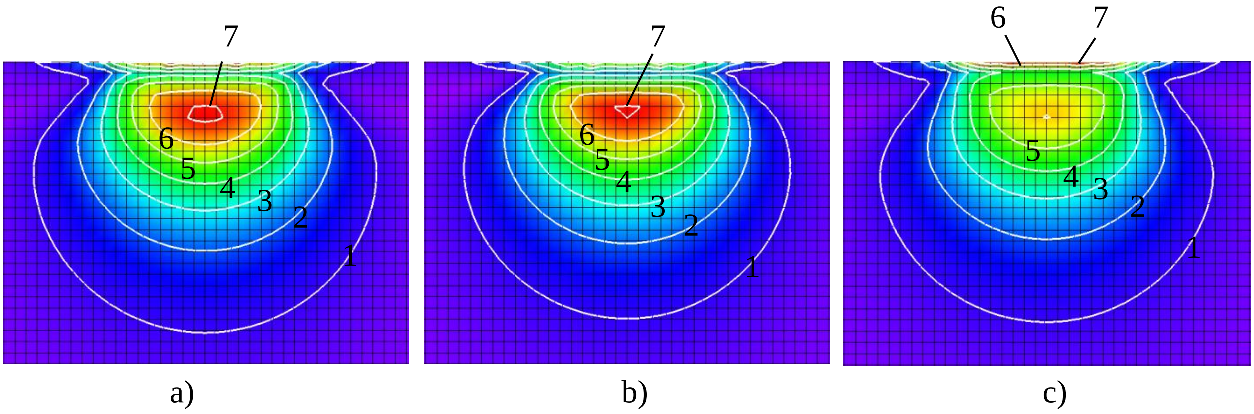


Figure 25.17. Isolines of the values of combined criterion σ_{RCF} in the nodes of plane FE mesh when the normal force N and tangential traction $F_z = fN$ are applied:
a – $f = 0.0$; b – $f = 0.17$; c – $f = 0.3$; 1-6 are isoline numbers

The biggest values of the criterion for the variants $f = 0.0$ and $f = 0.17$ are obtained for the points located at the depth of 4.5 mm under the contact surface. They amounted to 267 and 273 MPa respectively. With the coefficient of friction equal to 0.3 the point with the biggest value of the criterion of 328 MPa is shifted on the contact surface. The maximum value of the criterion in the subcontact layer at the depth of 5 mm is 277 MPa in this case.

Thus, the distributions of the values of equivalent stress, which do not contradict the estimates using the theory of Tresca and Mises, are obtained. The biggest equivalent stress is obtained at the point located in the subcontact layer when only normal force is applied to the bodies. The point with the biggest value of the equivalent stress is displaced on the contact surface when one body slides relative to the other with the coefficient of friction equal to 0.3. The negative value of the equivalent stress is observed in a small region adjacent to the contact patch.

The Dang Van (Sect. 25.2.5.2) and Sines criteria (Sect. 25.2.5.3), which allow analyzing a multiaxial stress state taking into account the hydrostatic stress, have found application in calculations of the machine parts of complex geometric shape, railway wheel plates. The problem of hydrostatic stress accounting is noticed in the works of A. Ekberg. It consists in the fact that for points located in the region of the contact patch the hydrostatic stress contained in the criterion is of big value and negative. For this reason this term can exceed in the absolute value the "amplitude" of the maximum shear stress. As a result, the criterion can have a negative value.

In the proposed combined criterion only one component of the hydrostatic stress plays a significant role, the other two enter into it with multiplying by "-ν".

The criterion allows taking into account the residual stresses in the material of the wheel, as well as the temperature stresses that occur during tread braking.

25.2.6. Rolling contact fatigue tests

Constructing RCF curves requires the solution of two problems – determination of the criterion value σ_{eq} for the different load levels and the material constants C and m to expression (25.1).

Specimens in the form of rollers or discs with a diameter 30-40 mm of thickness 8-12 mm are typically used for RCF testing of materials (Figure 25.18). One roller is made of the wheel steel; other roller is made of the rail steel. In some papers the roller made from rail steel is referred as a loading roller [29]. The loading roller may also be made of high-strength steel or the alloy WC8 (the Russian steel grade), which contains W 92 wt%, Co 8 wt% and has hardness of 88 HRA.

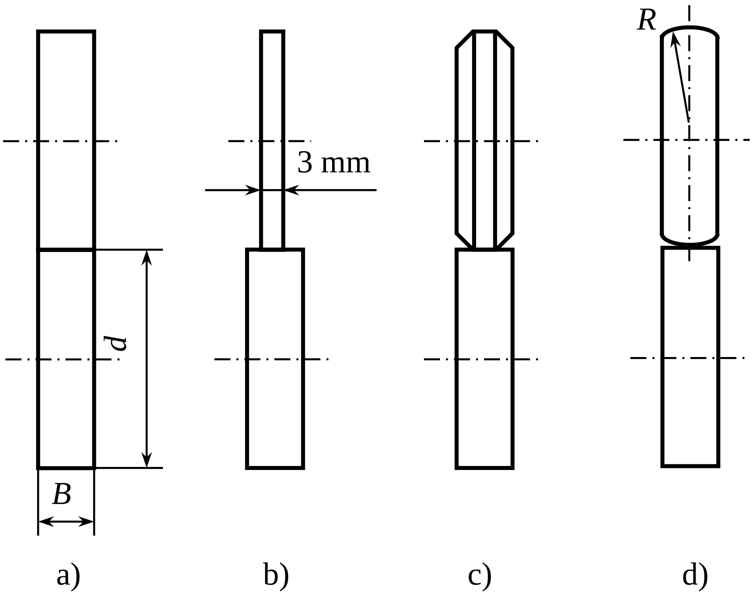


Figure 25.18. Specimens for RCF testing of the wheel steels

Specimen material in the contact region is in a state of plane strain. The stress state at the butt ends of the specimen is a plane stress, so the thickness of the loading roller must be sufficient so the influence of the butt ends can be ignored.

To obtain the maximum contact pressure of about 2000 MPa it is necessary to apply the load of 9.16 kN when the specimen sizes are d = 40 mm, B = 8 mm. The load is decreased by decreasing the contact length B (Figure 25.18a). In this case the loading roller of the thickness B = 3 mm (Figure 25.18b) or the roller with width of the rolling surface 3-4 mm (Figure 25.18c) is used. The rollers with the toroidal surface of rolling are also used (Figure 25.18d).

Another feature of RCF testing is the appearance of plastic deformation in the specimen material even at relatively low Hertzian pressures, the effect of which should be taken into account.

The results of the RCF tests provided by Prof. D.P. Markov were used in the development the **UM RCF** module. These tests were done at the Railway Research Institute (JSC "VNIIZhT", Russia). The test specimens were made from wheel steels having a different chemical composition. Nine experimental castings (EC) were produced for this.

As previously stated in Sect. 25.2.3, the factor having the biggest effect on the RCF resistance of the wheel steels is the hardness [12]. Therefore, the hardness was taken as a criterion of choosing steels. Three EC covering the hardness range from 260 to 320 HB were selected. In addition, they are close to Russian wheel steels of the grades 2, L and T [7] by their characteristics. Further in the manual the steels will be called by the numbers of EC.

EC6, EC2 and EC9 correspond to the carbon content of the steel grades 2, L and T respectively (Table 25.5).

Table 25.5

The characteristics of the materials of the experimental castings

Steel grade	The carbon content, wt%	Hardness, HB
EC6	0.57	269
EC2	0.54	295
EC9	0.73	322

The test specimens were cut out at a distance of 10 mm from the rolling surface of the wheel. The cylindrical specimens of diameter 40 mm were used. The toroidal roller of the diameter of 38.6 mm with a radius of the torus 18 mm made of hard alloy WC8 was used as the loading roller. The speed of rotation of the specimen was 1500 rpm. The specimens were rolling without sliding. The oil I-20 was applied to the rolling surfaces. The tests were carried out on five load levels. From 1 to 5 specimens were tested at each load level.

The test results are shown in Table 25.6.

Table 25.6

The dependence of the durability of the specimens on the load

Load, N	The durability, cycles$\cdot 10^5$		
	Steel EC6	Steel EC2	Steel EC9
120	8.71	11.73	18.90
300	5.12	5.86	9.14
600	2.82	3.21	5.78
900	2.90	2.80	3.87
1200	1.89	2.16	2.91

25.2.7. Stress-strain curves of wheel steels

The stress-strain curves of the wheel steels of the grades 2, L and T [7] are obtained for solving the contact problems in the elastoplastic formulation for the tested specimens. Tensile tests of the long cylindrical specimens with a diameter of 10 mm were done at the Vyksa Steel Works

(Russia). The obtained values of the yield strength, proof stress and the ultimate tensile strength of the steels are shown in Table 25.7.

Table 25.7

The characteristics of the strength and plasticity of the wheel steels

Parameter	The grade of the steel		
	L	2	T
The yield strength, MPa	655	624	747
ε_y ($\varepsilon_{0.2}$)	$3.28 \cdot 10^{-3}$	$3.12 \cdot 10^{-3}$	$3.73 \cdot 10^{-3}$
The ultimate tensile strength, MPa	931	887	1121

The polygonal approximation of the stress-strain curves is made for using them in the calculation. The relationship between stresses and strains for the *i* segment is represented as

$$\bar{\sigma} = a_i + b_i\bar{\varepsilon}, \tag{25.40}$$

where $\bar{\varepsilon} = \varepsilon_i/\varepsilon_y$ is the normalized intensity of the strains;

ε_i is the strain intensity;

ε_y is the strain corresponding to the yield strength;

$\bar{\sigma} = \sigma_i/\sigma_y$ is the normalized stress intensity;

σ_i is the stress intensity;

σ_y is the yield strength;

a_i, b_i are the dimensionless coefficients for the *i*-th segment of the curve.

The approximated stress-strain curves of the steels 2 and L are shown in Figure 25.19

The values of the coefficients of the polygonal approximation in the Eq. (25.40) for eight segments of the stress-strain curves of the steel grades 2, L and T are given in Table 25.8.

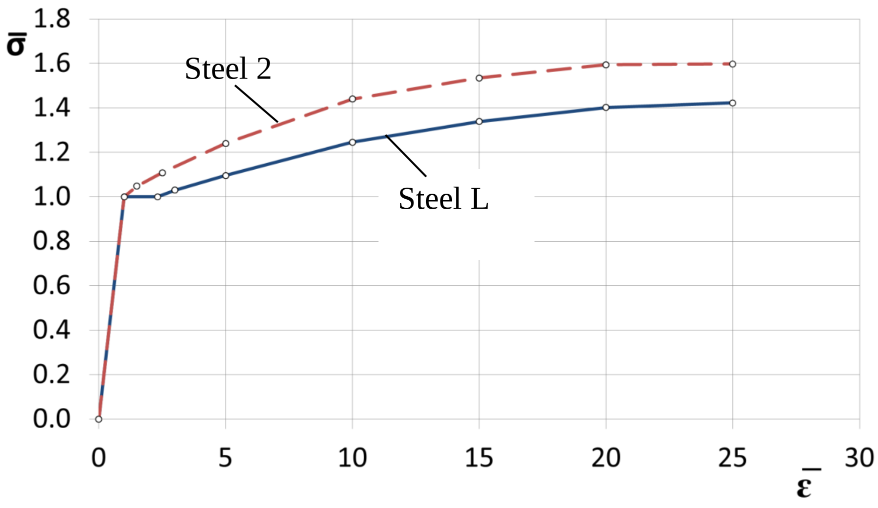


Figure 25.19. The approximated stress-strain curves of the wheel steels

Table 25.8

The parameters of the polygonal approximations of the stress-strain curves of the wheel steels

The steel of the grade L								
The strains intervals $\bar{\varepsilon}$	0-1	1-2.32	2.32-3	3-5	5-10	10-15	15-20	20-25
a_i	0	1	0.898	0.931	0.946	1.06	1.15	1.318
b_i	1	0	0.044	0.033	0.03	0.0186	0.0126	0.0042
The steel of the grade 2								
The strains intervals $\bar{\varepsilon}$	0-1	1-1.5	1.5-2.5	2.5-5	5-10	10-15	15-20	20-25
a_i	0	0.904	0.9565	0.978	1.04	1.25	1.358	1.578
b_i	1	0.096	0.061	0.0524	0.04	0.019	0.0118	0.0008
The steel of the grade T								
The strains intervals $\bar{\varepsilon}$	0-1	1-1.5	1.5-2	2-4	4-6	6-8	8-12	12-16
a_i	0	0.93	0.915	0.959	1.001	1.046	1.18	1.329
b_i	1	0.07	0.08	0.058	0.0475	0.04	0.0233	0.0108

25.2.8. Rolling contact fatigue curves of wheel steels

The stress-strain state of the specimens was analyzed by the FE method [18]. The scheme of testing and FE models of the fragments of the wheel steel specimen and loading roller are shown in Figure 25.20.

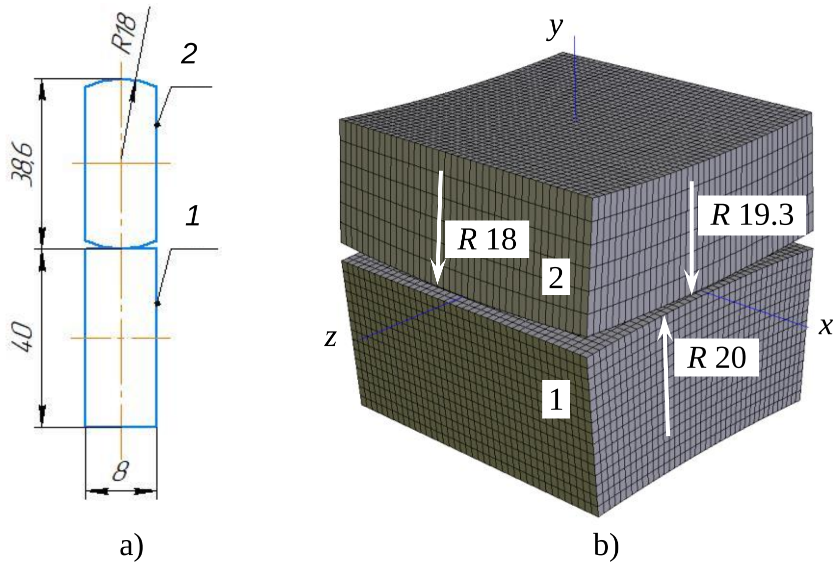
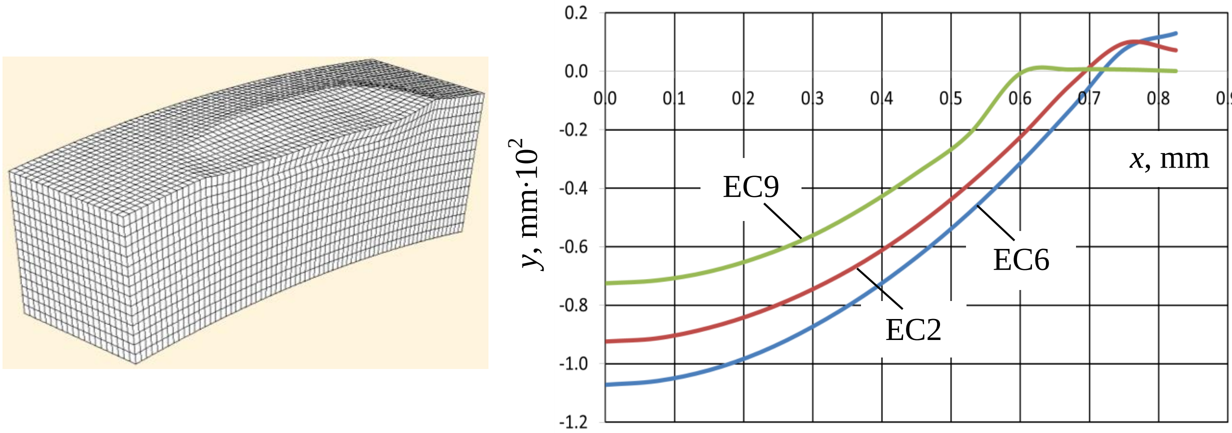


Figure 25.20. Wheel steel specimen of cylindrical shape (1) and loading roller of toroidal shape (2): a – scheme of testing; b – FE models

FE models of the fragments of the wheel steel specimen and loading roller had dimensions on the *Ox* and *Oz* axes of 3.15 mm, on the *Oy* axis of 1.5 mm. The finite elements of the serendipity family [30] with eight nodes and dimensions of the edges along the *Ox* and *Oz* axes of 0.075 mm and 0.1 mm along the *Oy* axis were used. The rollers had dimensions: the radius of the specimen of the wheel steel is 20 mm, the radius of the loading roller is 19.3 mm and the radius of the torus is 18 mm. The loading roller material had following parameters: $5.9 \cdot 10^{11}$ Pa for the tensile modulus and 0.202 for the Poisson's ratio. The wheel steel specimens material had following parameters: $2.0 \cdot 10^{11}$ Pa for the tensile modulus and 0.3 for the Poisson's ratio. The FE model of the wheel steel specimen had 26,896 nodes. The FE model of the loading roller had 11,767 nodes. The number of the finite elements was 24,000 and 9,600 respectively.

The rolling contact problem was solved in the elastoplastic formulation. The approximated stress-strain curves with the values of the coefficients a_i and b_i given in Table 25.8 were used for the wheel steel materials (Sect. 25.2.7). Rolling was simulated by using a sequential shift of the loading roller along the *Oz* axis at a step equal to the size of the finite element. 30 steps were used which corresponds to rolling along a distance of 2.25 mm. Such distance was sufficient to exclude the influence of edge effects on the stress state in the contact region. The contact problem was solved for each position of the loading roller. Solutions were made for the specimens of all three EC steels for five load levels (Sect. 25.2.6).

During testing a trough was formed on the cylindrical surface of the wheel steel specimen due to rolling of the loading roller. Its cross-section shape is shown in Figure 25.21.



a) b)

Figure 25.21. The trough formed on the cylindrical surface of the wheel steel specimen due to rolling of the loading roller: a – deformed state of the FE model in the formation of the trough (displacements are increased of 20 times); b – profiles of the cross-sections of the trough after the initial rolling of the loading roller with the load of 1200 N for specimens of EC6, EC2 and EC9 steels

The biggest depth of the trough was obtained for the specimen of EC6 steel. It amounted to 0.0107 mm at the load 1200 N. The radii of curvature of the troughs in cross-section were defined approximately. Their values of 24, 25.8 and 27.5 mm were obtained for the specimens of EC6, EC2 and EC9 steels respectively.

Typically the maximum theoretical Hertzian pressure is used to generate RCF curves. However, due to the formation of the observed trough the real contact pressure will be significantly lower than the maximum Hertzian pressure. For example, comparison of the values of the maximum theoretical Hertzian pressure and the pressure calculated by the FE method for the specimen of EC2 steel are shown in Table 25.9.

Table 25.9

The maximum pressure in the contact of the specimen of EC2 steel and the loading roller

Load, N	120	300	600	900	1200
Hertzian pressure p_0 without accounting the formed trough, MPa	1520	2059	2598	2978	3278
Pressure p_0 by FEA with accounting the formed trough, MPa	1471	1643	1936	2095	2195

The maximum pressure under the load 1200 N when considering the curvature of the trough from the Hertz solution in the elastic formulation is equal to 2345 MPa. It is much less than the pressure 3278 MPa calculated without taking into account the formed trough.

The initial rolling of the loading roller was simulated as follows. The loading roller was placed at the position *0*, the load was applied to it. The contact problem in the elastoplastic formulation was solved. Then the loading roller was moved in position *1* onto the distance equal to length of the finite element. The contact problem was solved again. This procedure was repeated until reaching the position *29*. The deformed state of the surface of specimen after the initial rolling is shown in Figure 25.21a. The figure shows that after the initial rolling on the surface of specimen the trough is formed. The field of residual stresses in the material of specimen after initial rolling was obtained.

Then the loading roller was placed at the position *14* at the middle of the raceway for simulation of repeated loading. The same load as for the initial loading was applied to it. The stresses obtained after repeated loading were used for determination of the values of RCF criterion.

The length of the segment of the raceway of the loading roller was chosen so that at this length it was possible to choose a region where the residual stresses did not change. The isolines of residual stresses by Mises after initial rolling are shown in Figure 25.22.

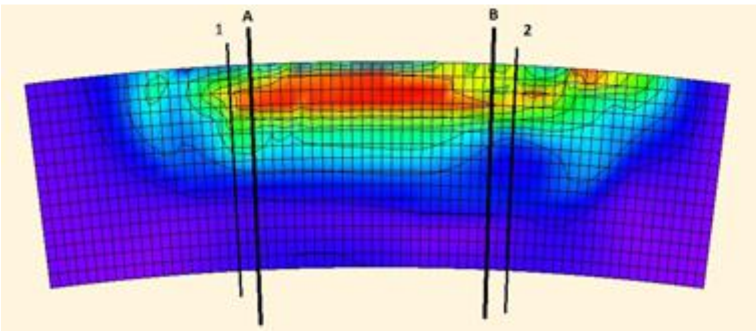


Figure 25.22. Section of fragment of the wheel steel specimen along the longitudinal plane of symmetry: 1, 2 are initial and final positions of the loading roller; AB is segment selected to determine the values of RCF criterion; coloring are levels of stresses by Mises

Variable stress state in the rolling direction takes place close to the initial position 1 and final position 2. Therefore, for determination of the value of RCF criterion the region of the model between the planes A and B was chosen (Figure 25.22). The stresses in all nodes of the FE model after repeated loading and the values of the RCF criterion at the nodes located in the radial section of the specimen were determined by the methods described in Sect. 25.2.5.

The isolines of values of the Dang Van criterion (Sect. 25.2.5.2) in the radial section of the specimen of EC2 steel under the loads 300 N and 1200 N are shown in Figure 25.23. At the lowest loads the biggest values of stresses are observed at the depth of 0.4 mm (Figure 25.23a). As the load increases, the points with the biggest values rise to the contact surface. There is the region with zero criterion values below the contact patch surface. This is the region with a big negative hydrostatic stress (Figure 25.23b). The maximum values of the Dang Van criterion in the material of the specimen of EC2 steel depending on the applied loads are given in Table 25.10.

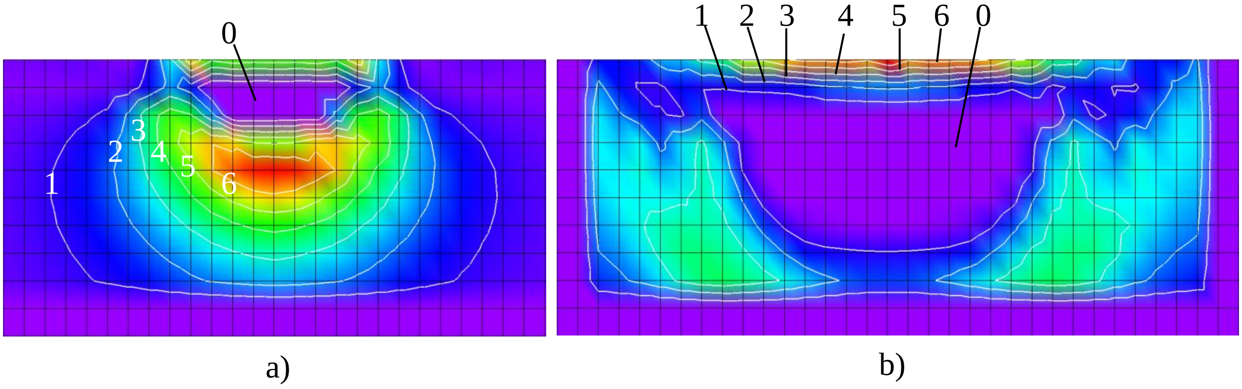


Figure 25.23. Isolines of values of the Dang Van criterion in the radial section of the specimen of EC2 steel under the load: a – 300 N; b – 1200 N; 1-6 are isoline numbers

Table 25.10

Maximum values of the Dang Van criterion in the material of specimen of EC2 steel and its durability depending on the level of the load

Load, N	1200	900	600	300	120
τ_{DV}, MPa	296	280	251	140	110
Durability, cycles$\cdot 10^5$	2.16	2.8	3.21	5.86	11.73

As a result of approximation of the obtained data in accordance with the expression (25.1) the dependence of the number of cycles until arising fatigue defects in the material of the specimen of EC2 steel on the value of the Dang Van criterion was obtained

$$N = 5{,}28 \cdot 10^9 \tau_{DV}^{-1{,}8}. \tag{25.41}$$

The dependence (25.41) is represented by the diagram in Figure 25.24.

The similar analysis was carried out for the specimens of EC6 and EC9 steels. The resulting RCF curves obtained using the Dang Van criterion for the specimens from the three experimental castings are shown in Figure 25.25.

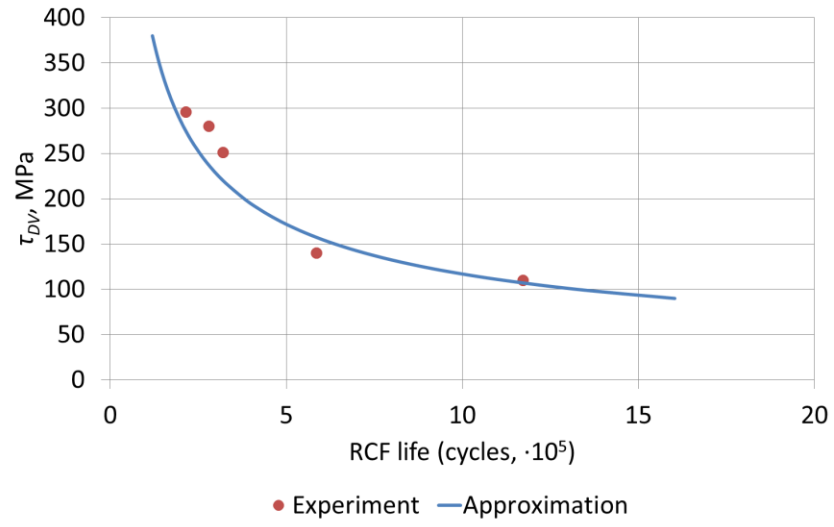


Figure 25.24. RCF curve for specimen of EC2 steel obtained using the Dang Van criterion

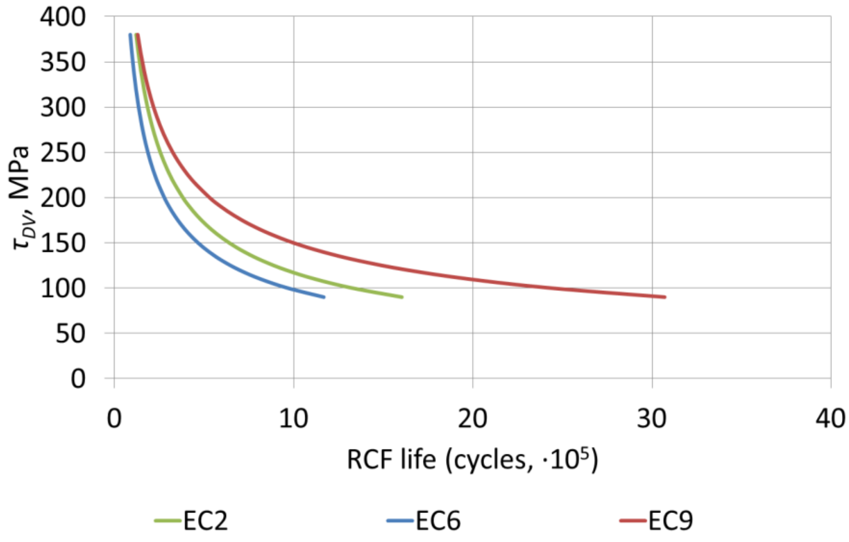


Figure 25.25. RCF curves for specimens of EC6, EC2 and EC9 steels obtained using the Dang Van criterion

The calculations using other RCF criteria were done similarly.

In accordance with the combined criterion (Sect. 25.2.5.4) for each position *j* of the node *i* of the FE model (Figure 25.5) the value of the equivalent stress σ_{eq} (25.32) is calculated. For the obtained cycle the mean stress σ_m (25.33) and amplitude σ_a (25.34) are calculated. The criterion value for the node *i* is calculated using the dependence (25.39).

The isolines of values of the combined criterion in the radial section of the specimen of EC2 steel under the loads 300 and 900 N are shown in Figure 25.26.

At the load equal to 300 N the biggest value of the criterion of 276 MPa is observed at the points located at the depth of 0.3-0.4 mm under the contact surface (Figure 25.26a). The biggest value of the criterion equal to 488 MPa at the load of 900 N is observed at the point located on the surface of the specimen (Figure 25.26b).

The maximum values of the combined criterion in the material of the specimen of EC2 steel and the durability of the specimens depending on the applied loads are given in Table 25.11.

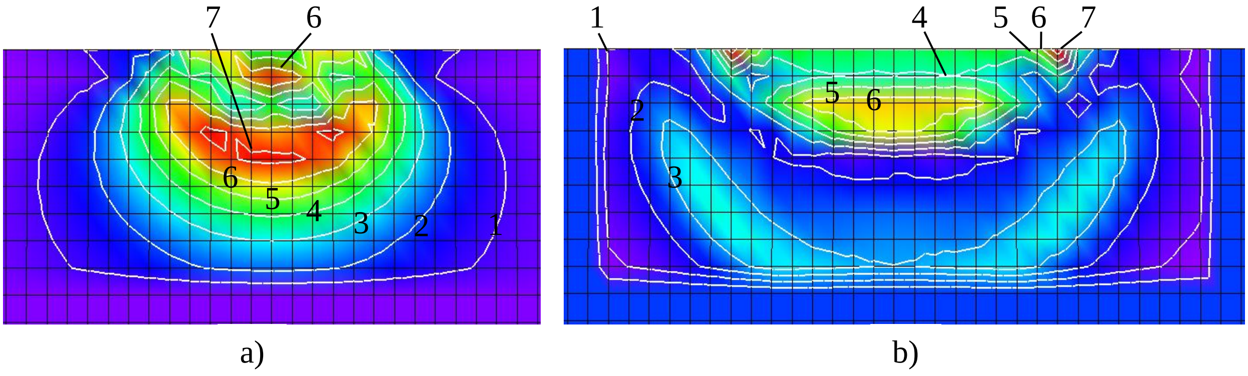


Figure 25.26. Isolines of values of the combined criterion in the radial section of the specimen of EC2 steel under the load: a – 300 N; b – 900 N; 1-7 are isoline numbers

Table 25.11

Maximum values of the combined criterion in the material of specimen of EC2 steel and its durability depending on the level of the load

Load, N	1200	900	600	300	120
σ_{RCF}, MPa	450	488	462	276	263
Durability, cycles·10^5	2.16	2.8	3.21	5.86	11.73

As a result of approximation of the obtained data in accordance with the expression (25.1) the dependence of the number of cycles until arising fatigue defects in the material of the specimen of EC2 steel on the value of the combined criterion was obtained

$$N = 3{,}61 \cdot 10^{10} \sigma_{RCF}^{-1{,}9}. \qquad (25.42)$$

The dependence (25.42) is represented by the diagram in Figure 25.27.

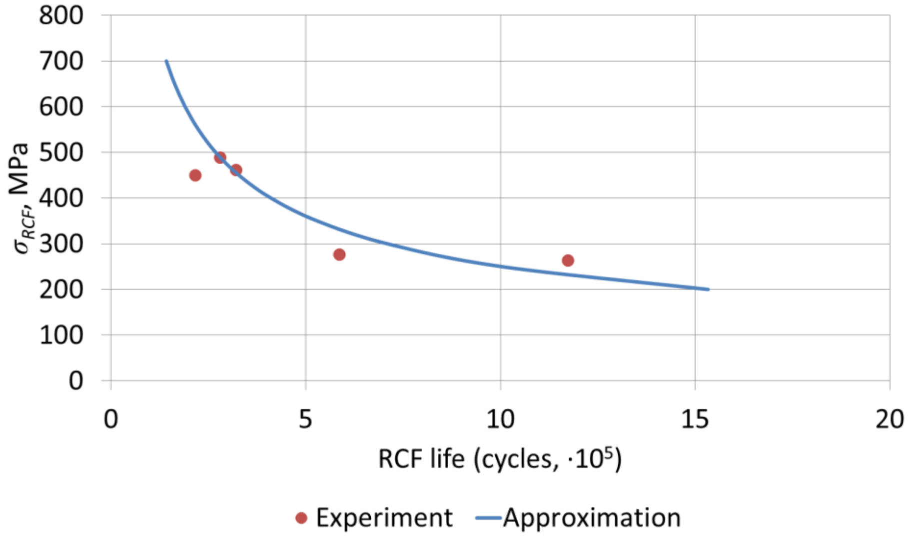


Figure 25.27. RCF curve for specimen of EC2 steel obtained using the combined criterion

The similar analysis was carried out for the specimens of EC6 and EC9 steels. The resulting RCF curves obtained using the combined criterion for the specimens from the three experimental castings are shown in Figure 25.28.

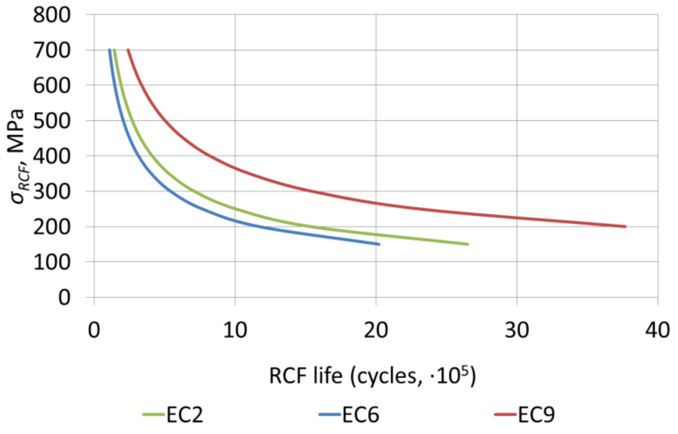


Figure 25.28. RCF curves for specimens of EC6, EC2 and EC9 steels obtained using the combined criterion

In accordance with the criterion of the amplitude of maximum shear stress (Sect. 25.2.5.1) for each position *j* of the node *i* of the FE model (Figure 25.5) the value of the stress τ_{max} (25.5) and then the criterion value τ^a_{max} (25.6) as the amplitude value of the cycle of the stresses τ_{max} are calculated.

The resulting RCF curves obtained using the criterion of the amplitude of maximum shear stress for the specimens from the three experimental castings are shown in Figure 25.29.

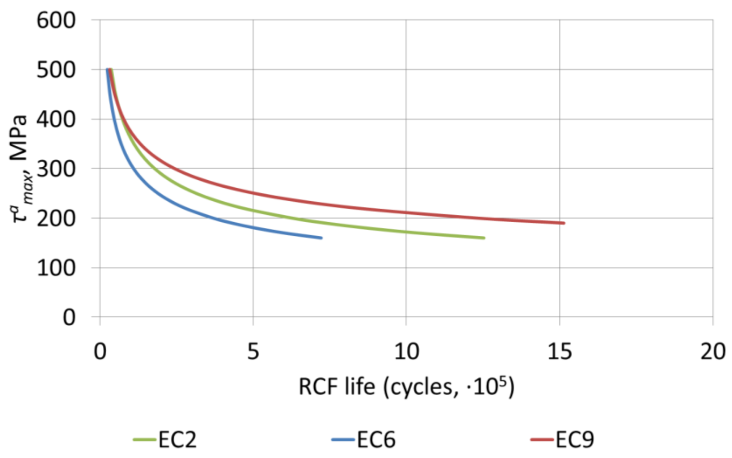


Figure 25.29. RCF curves for specimens of EC6, EC2 and EC9 steels obtained using the criterion of the amplitude of maximum shear stress

In accordance with the Sines criterion (Sect. 25.2.5.3) for each position *j* of the node *i* of the FE model (Figure 25.5) the value of the octahedral shear stress τ_{oct} (25.17) and of the sum of the normal stresses σ_{sum} (25.18) are calculated. For the obtained cycles of stresses τ_{oct} and σ_{sum} the amplitude value of the octahedral shear stress τ^a_{oct} (25.19) and the mean value of the sum of the normal stresses σ^{mid}_{sum} (25.20) are calculated. The criterion value for the node *i* is calculated using the dependence (25.21).

The resulting RCF curves obtained using the Sines criterion for the specimens from the three experimental castings are shown in Figure 25.30.

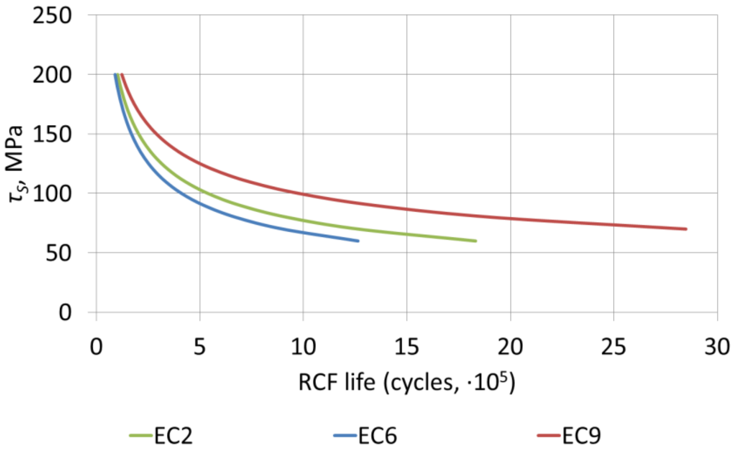


Figure 25.30. RCF curves for specimens of EC6, EC2 and EC9 steels
obtained using the Sines criterion

For ease of use the values of the material constants C and m in the expression of the RCF curve (25.1) for the four criteria considered above and the wheel steels of different hardness from the three experimental castings are summarized in Table 25.12.

Table 25.12

Values of the material constants for different wheel steels and RCF criteria

RCF criterion	Wheel steel	Hardness, HB	Material constants in Eq. (25.1) for RCF curve	
			C	m
The criterion of the amplitude of maximum shear stress	EC6	269	$2.955 \cdot 10^{12}$	3.0
	EC2	295	$8.522 \cdot 10^{12}$	3.2
	EC9	322	$1.972 \cdot 10^{15}$	4.0
The Dang Van criterion	EC6	269	$3.845 \cdot 10^{9}$	1.8
	EC2	295	$5.280 \cdot 10^{9}$	1.8
	EC9	322	$6.117 \cdot 10^{10}$	2.2
The Sines criterion	EC6	269	$1.031 \cdot 10^{10}$	2.2
	EC2	295	$3.390 \cdot 10^{10}$	2.4
	EC9	322	$9.765 \cdot 10^{11}$	3.0
The combined criterion	EC6	269	$2.753 \cdot 10^{10}$	1.9
	EC2	295	$3.610 \cdot 10^{10}$	1.9
	EC9	322	$4.347 \cdot 10^{11}$	2.2

The obtained RCF curves can be used for modelling of the process of accumulation of the RCF damage for the wheel steels that are close in characteristics to the characteristics of experimental castings.

25.3. Simulation of accumulation of rolling contact fatigue damage using UM RCF Wheel module

Rolling the wheel on the rail is a cyclic loading. Thus, the maximum stresses in any radial section of the wheel occur once per revolution of the wheel. For this reason, in the **UM RCF Wheel** module accumulated RCF damage is attributed to one radial cross-section of the wheel.

In the simulation of the process of accumulation of RCF damage it is impossible to indicate the point, at which the accumulation of damage will be occurred most intensively. Even if the contacted bodies will not have a mutual displacement, but the normal force is variable, for example, in the Hertz problem the sizes of the semiaxes of the elliptical contact patch and therefore also the positions of the dangerous points both on the contact surface and in the subcontact layer will be changed.

Consequently, calculations should be performed for the certain set of points located in the region adjacent to the contact. When the FE analysis is applied, the nodes of the FE model can be used as such points. For the correct solution of the contact problem the dimensions of the FE models of the bodies should be such that the condition of smallness of the contact patch size is fulfilled.

But with a large number of degrees of freedom of the system the considerable time to the contact problem solving and determination of the stresses at the nodes is needed. And it is necessary to perform such calculations many times. It is possible to select some cross-section of the wheel rim and watch for the force values at each moment when it comes into contact with the rail. The number of such cases should be sufficient so that the statistical data about the forces and positions of the wheel on the rail are representative.

Based on the considerations outlined above, in the module a fragment of the FE model of the wheel adjacent to the contact is used. The size of this localized fragment should be so that the stresses on the planes of localization are relatively small.

25.3.1. Input data

Necessary data for modelling of the accumulation of RCF damage in the **UM RCF** module is created during process of simulation of dynamics of a railway vehicle in the **UM Loco** module (Chapter 8) with help the **UM Loco/Wheel Profile Wear Evolution** tool (Chapter 16). In Figure 25.31 the **Object simulation inspector** window with settings of the evolution project of the wheel profile is shown.

The wheels, for which it is necessary to simulate the accumulation of damage, should be switched on at the tree in the **Rolling contact fatigue** group box. In result after completion of simulation of dynamics of railway vehicle in the folder specified in the **Directory for saving results** item the binary ***.wld** files and the text ***.rcf** file of RCF project will be created. In the same folder the files with results of simulation of wear and settings of the evolution project will be located. The RCF project file has the same name as the model. Name of the ***.wld** input data file contains the number of the wheelset and notation of the wheel. Notation "**l**" is left wheel, "**r**" is right wheel.

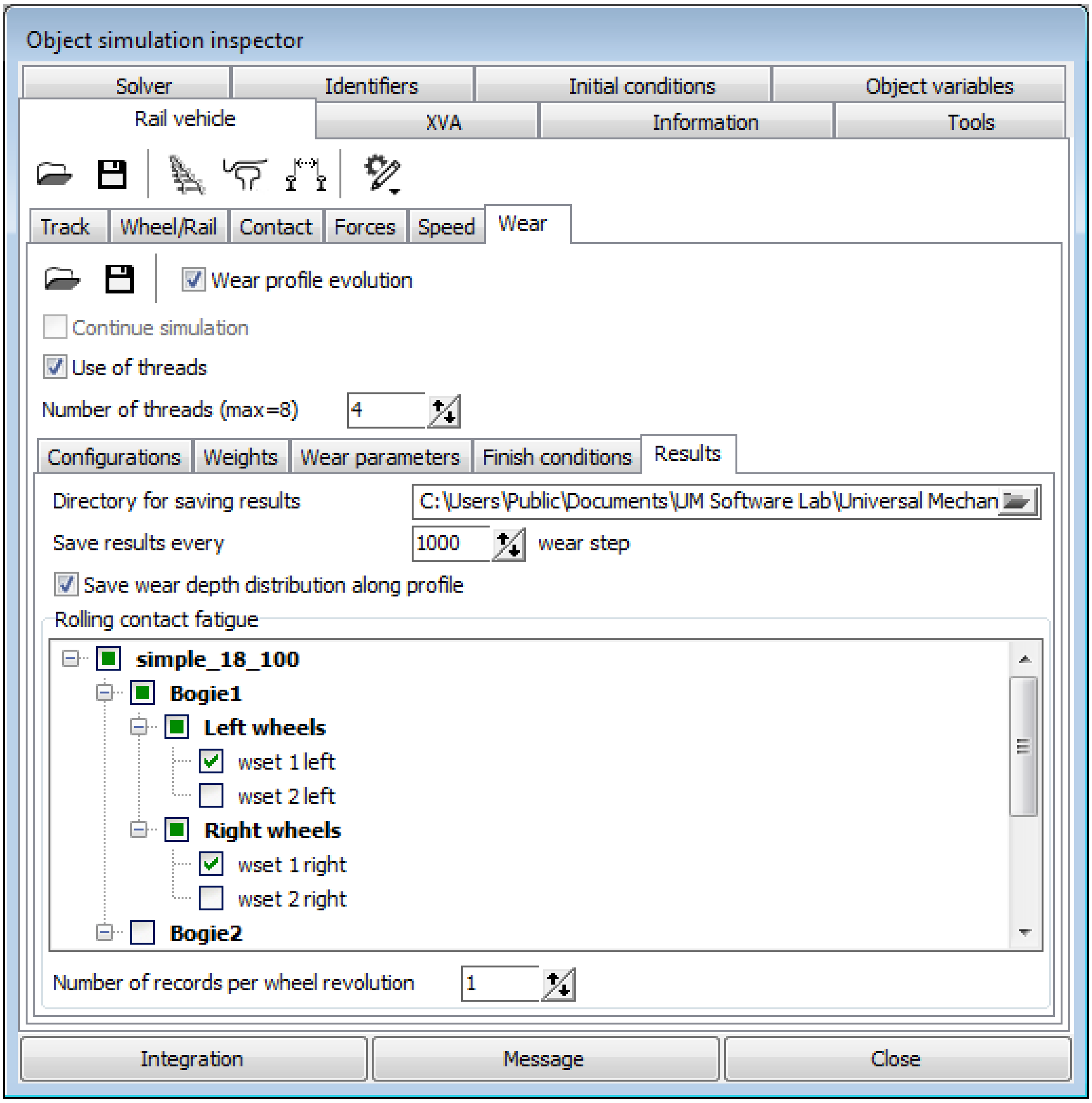


Figure 25.31. Settings of the evolution project of the wheel profile in **UM Simulation**

The wheel profiles obtained as a result of wear are saved in the ***.wld** initial data file. The number of the profiles depends on the number of the iterations specified in the evolution project. For each number of profile the following data is stored:

– the Young's modulus and Poisson's ratio of material of the wheel;

– the radius of the wheel;

– the number of the points of the wheel profile and their coordinates;

– the weight factors;

– the number of loading cycles (the number of revolutions of the wheel);

– the data about contact for each loading cycle – the coordinates of nodes of the mesh on the surface of contact patch and values of the forces in the nodes of mesh.

Замечание. There should be enough free space on the hard disk. Depending on the settings of the evolution project each ***.wld** file with the input data can take up to several gigabytes.

25.3.2. Construction of finite element model of wheel

For solving the contact problem of the wheel and rail the dimensions of the finite elements are chosen so as to ensure a minimum computation time at the sufficient precision of solution. The profiles of the rolling surfaces of the wheel and rail from input data file are specified by the coordinates of the points located along the contours with step of 0.1 mm. For analysis of the stress state in the contact region it turned out to be appropriate to apply the meshes with the size of the element of 1 mm for three-dimensional (3D) FE models. Nevertheless, when using the fragments that include regions adjacent to the contact surfaces of such sizes at which the stresses are small on the surfaces of extraction of the fragment, the FE model contains a big number of the degrees of freedom. In this connection, the FE model containing 10 layers of finite elements counted from the contact surface is used in the calculations.

Construction of the FE model starts with creation of the two-dimensional (2D) flat mesh composed of the quadrangular finite elements. It is created on the base of nodes placed on the wheel profile with step of 1 mm (Figure 25.32). Its dimension along the wheel profile is 105 mm. This dimension covers the profile section on which the contact has place at any possible positions of the wheel on the rail. Then 3D FE model is generated. 2D mesh is dragged along circular arc by revolution around an axis of the wheelset (Figure 25.33).Dimension of the FE model at the circumferential direction is 50 mm. This dimension is chosen with taking into consideration that contact patch length does not exceed 50 mm in any case.

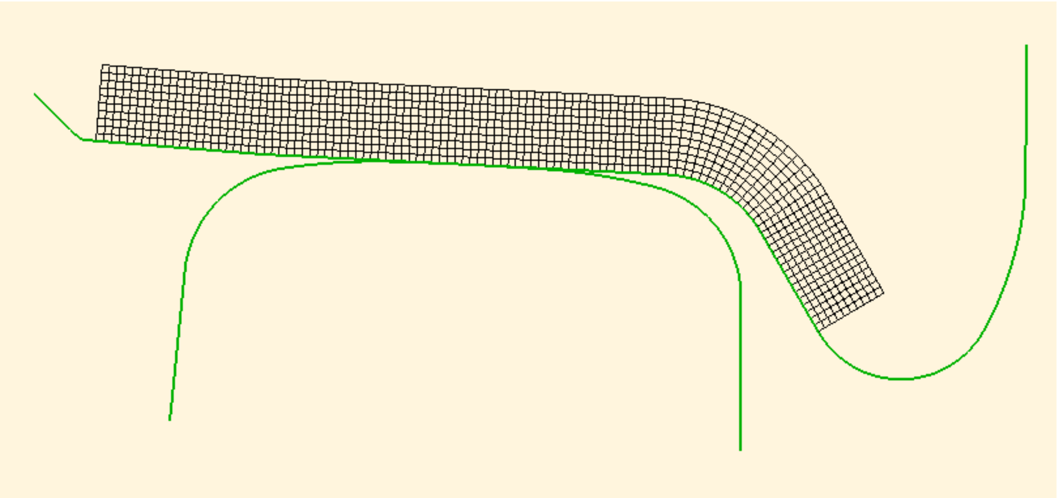


Figure 25.32. 2D mesh for construction of 3D FE model of wheel fragment

The thickness of the fragment of 10 mm ensures sufficient accuracy of the solution when using an elastic foundation. The most probable dimension of the semiaxis *a* of the contact patch does not exceed 10 mm and the point with the biggest shear stress is placed on the depth about 0.5*a*. It is possible to cover the region of the biggest stresses at the selected size of the fragment. Nevertheless, the obtained scheme is characterized by a big number of the degrees of freedom. For example, the FE model presented in Figure 25.33 contains 59 466 nodes and has 178 398 degrees of freedom. Therefore, in order to decrease the computation time a fragment of this FE model will be used in calculations (see Sect. 25.3.3).

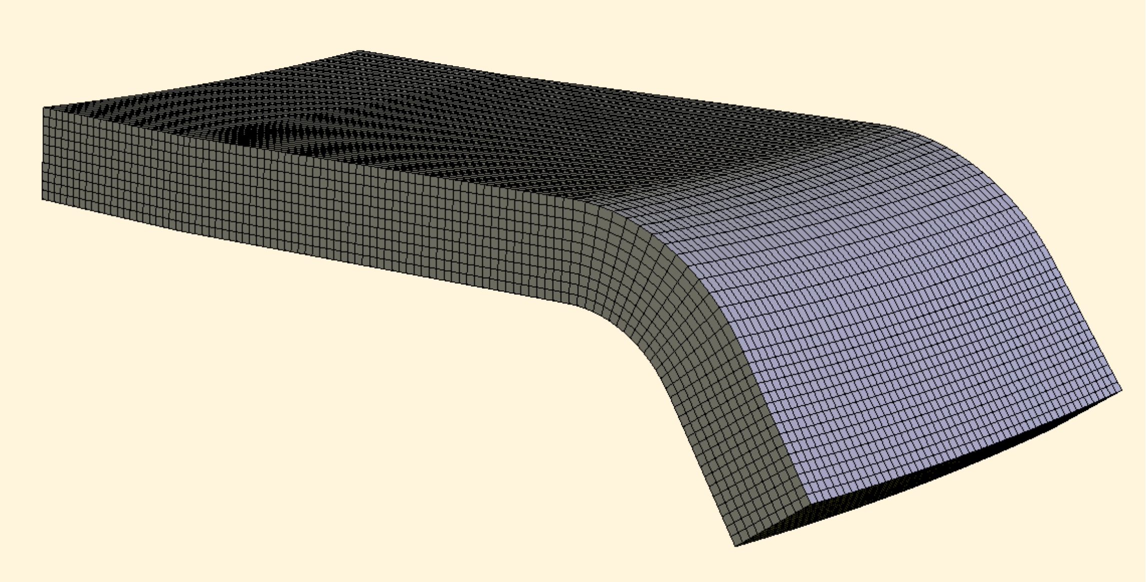


Figure 25.33. 3D FE model of wheel fragment

The stiffness matrix of the obtained 3D FE model of the wheel consisting of eight-node finite elements is formed. This concludes the preparatory stage and further calculations for loading cycles are started.

25.3.3. Calculation of stresses in the region of wheel and rail contact

For solution of the rolling problem in the **UM Loco** module the model of contact forces of W. Kik and I. Piotrowski [31] and the Kalker CONTACT model [32] are used (Chapter 8, Sect. 8.4.2.5.2. "Parameters of inertial rail contact"). The **UM Loco/Multi-point Contact Model** tool is required for the W. Kik and I. Piotrovski model and the **UM Loco/CONTACT add-on interface** tool for applying the CONTACT program.

Normal and tangential traction are calculated at the nodes of the 2D mesh located on the contact surface. The mesh is created as follows: the contact patch is divided into strips of the same width oriented along the rolling direction, and then each strip is divided into the same number of elements having the same dimensions within the strip (Chapter 8, Sect. 8.3.1.2.2.3. "FASTSIM"). The contact pressure on the plane of each element is determined. The calculated forces are attributed to the center of gravity of the element. The number of the strips and elements is set by the user. Since during the rolling of the wheel along the rail the contact patches change their shape and size, the step of the mesh in two directions also changes for each new contact.

For determination of the stresses in the nodes of the FE model of the wheel fragment it is necessary to apply the forces obtained as a result of solving the contact problem to the nodes located on the contact surface. However, the nodes of the mesh created during the solution of the problem by fast algorithms do not coincide with the nodes of the FE mesh. Therefore, after loading the initial data the forces in the nodes of the FE mesh located on the contact surface are calculated using a linear interpolation. At that the position of the meshes relative to each other is uniquely determined by the coordinates of the wheel profile points, on the basis of which these meshes are created.

The procedure for determining the stresses at the nodes of the FE model of the wheel for each loading cycle begins with the separating a fragment (Figure 25.34a) from the initial FE model (see Sect. 25.3.2). The dimensions of the mesh of the fragment adjacent to the contact surface are chosen so that they are 3 mm bigger than the contact patch in each direction. Separated fragment has a thickness as the initial 3D FE model.

The application of the FE fragment on the elastic foundation [33] as the design scheme allows to determine the stresses and the strains in 3D region adjacent to the contact surface without any type of simplified assumptions like these: a contact patch has shape of circle or ellipse, to selection of the critical plane, to assumption that it is in a state of plane deformation. Use of such fragment allows decreasing the computation time essentially [33], [34], [35], [36].

During construction of the initial FE model the components of its matrix of stiffness are calculated. To the nodes located on all surfaces of the separated fragment except the contact surface the elastic constraints are applied (Figure 25.34b). The stiffness of the constraints is assumed to be 10^7 N/m in accordance with the recommendations given in work [36].

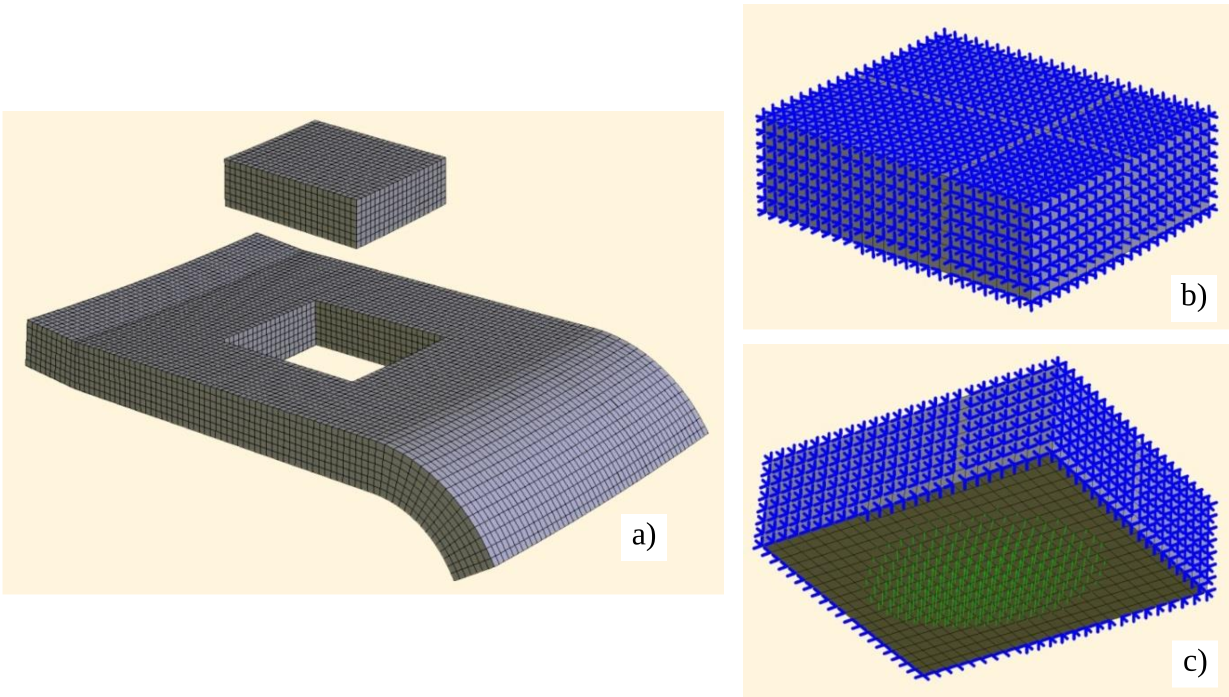


Figure 25.34. FE fragment on elastic foundation:
a) separation of small fragment from initial FE model;
b) applying of elastic constraints on surfaces of separated fragment;
c) applying of forces to nodes of contact surface of separated fragment

Components of the stiffness matrix of the finite elements ensemble adjacent to an internal node of the fragment are taken from the stiffness matrix formed for the initial FE model. For nodes located on the surfaces of separated fragment the components are halved, for nodes located on the edges they are divided into four.

Nodal contact forces are applied to the nodes located on the contact surface of the separated fragment (Figure 25.34c). Displacements of the nodes are determined with the method of node iterations [37].

The normal and shear stresses are calculated for all nodes of the separated FE fragment. The number of nodes of the fragment in the rolling direction is considered as the path of the i-th node of the plane radial section of the fragment (Figure 25.5). Then each node from this chain is the j-th position occupied by the i-th node as it passes through the region adjacent to the contact. The equivalent stress per cycle is calculated in accordance with the chosen RCF criterion (Sect. 25.2.5). The obtained values of the criterion are assigned to the nodes located in the plane of symmetry of the FE model of the wheel fragment. During simulation of the process of accumulation of RCF damage they are compared with damaging values from given RCF curve.

The areas of equal values of the combined criterion σ_{RCF} in the color and the isolines of the values of the criterion are shown in Figure 25.35. The biggest $\sigma_{RCF} = 263$ MPa is obtained for the node located at the depth of 4 mm below the contact surface.

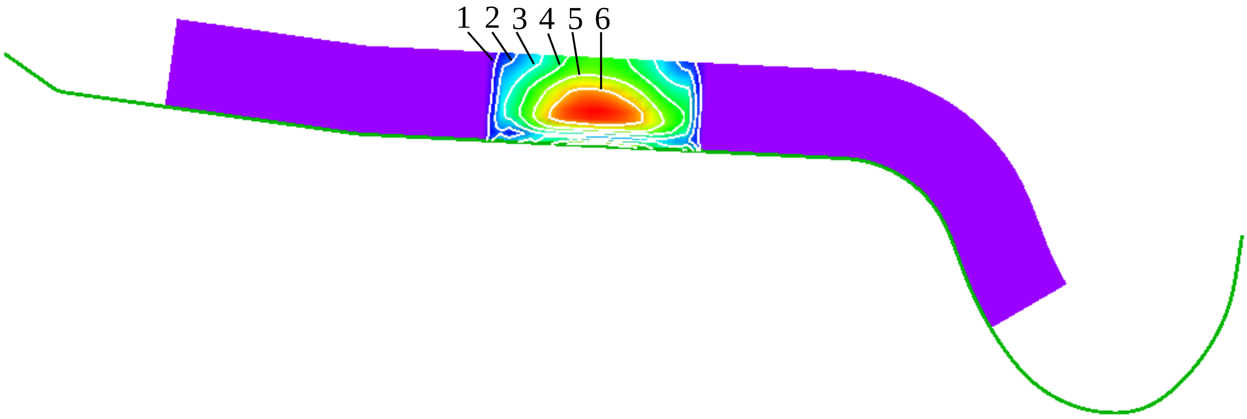


Figure 25.35. Areas of equal values of the combined criterion σ_{RCF} in the radial cross-section of the wheel contained in the plane of symmetry of contact; 1-6 are numbers of isolines σ_{RCF}, the value of isoline is 37.57 MPa

25.3.4. Accumulation of damage in the nodes of finite element model

The simulation of the dynamics of the movement of a railway vehicle is carried out on the sections of the track of short length in order to reduce the machine time costs. The mileage is scaled to obtain wear indicators for longer sections, for example, several thousand kilometers. The wear at each point of the wheel profile is calculated using a scaling factor. A similar approach is applied to the process of modelling of the accumulation of RCF damage in the wheel. Dependence (25.2) for calculating the RCF damage accumulated in the n-th node is transformed to the form

$$Q = \sum_{m=1}^{N_{it}} \sum_{i=1}^{N_C} \frac{N_{ws} km_{step}}{S_i - S_{b_i}} \alpha_i \sum_{j=1}^{N_{v_i}} \beta_{ij} \sum_{k=1}^{N_{LC_i}} \frac{1}{N_n(k)}, \quad (25.43)$$

where N_{it} is the number of iterations of the wear simulation;

N_C is the number of configurations in the wear project;

N_{ws} is the number of the wear steps (the number of profile changes during one wear iteration);

km_{step} is the mileage at one wear step;

S_i is the length of the track traveled by the vehicle during the simulation;

S_{b_i} is the distance after which the wear calculation starts;

α_i is the weight of the *i*-th configuration $\left(\sum_{i=1}^{N_C} \alpha_i = 1\right)$;

N_{v_i} is the number of the speeds in the *i*-th configuration;

β_{ij} is the weight of the *j*-th speed in the *i*-th configuration $\left(\sum_{j=1}^{N_{v_i}} \beta_{ij} = 1\right)$;

N_{LC_i} is the number of loading cycles for the *i*-th configuration;

$N_n(k)$ is the number of cycles until the appearance of fatigue failure when the criterion value is σ_{eq}^n.

The number of loading cycles N_{LC_i} for the *i*-th configuration is determined by the dependence

$$N_{LC_i} = \frac{S_i - S_{b_i}}{2\pi R} N_r, \tag{25.44}$$

where R is the wheel radius;

N_r is the number of records per wheel revolution.

All of the listed above parameters are set in the evolution project. For details, see Chapter 16 of the user manual.

The number of cycles until arising fatigue defects for the nodes located in the plane of symmetry of the FE fragment is calculated using the Eq. (25.1). Then the accumulated damage in these nodes is determined by summing the damage in accordance with dependence (25.43).

In the program settings the user can choose two variants for stopping the simulation of the damage accumulation.

1) Set the limiting value of the accumulated damage Q.

2) Select the traveled distance variant, i.e. damage will be accumulated without limitation until the wheel has covered the entire distance set in the evolution project.

User can also set the minimum value of the damaging stress in accordance with the selected RCF criterion. During simulation of the accumulation of RCF damage the cycle is considered damaging if the value of the criterion exceeds this limit. The minimum value by default is zero.

25.3.5. Consideration of wheel profile wear

Wear of the profile of the rolling surface of the wheel is one of the most important factors influencing the accumulation of RCF damage in the material of the wheel. In the process of wear the profile of the wheel is changed, which leads to the change of the dynamics of the vehicle and the redistribution of the pressure in the contact of the wheel and the rail. In addition, as the result of wear the surface layer of the material of the wheel, in which fatigue damage has already been accumulated, disappears, RCF damage begins to accumulate in the layer located deeper under the tread. Thus, the processes of the wear and accumulation of RCF damage are inseparably connected. For this reason, the data needed for modelling of the accumulation of RCF damage in the **UM RCF Wheel** module is created using the **UM Loco/Wheel Profile Wear Evolution** tool (Chapter 16).

Let us introduce the concept of *wheel wear iteration*. Within the framework of this section, the *wheel wear iteration* is a series of calculations within which the profile of the rolling surface of the wheel does not change.

For accumulation of RCF damage in the **UM RCF Wheel** module the FE model of the wheel fragment is used (Sect. 25.3.2). At the each *wheel wear iteration* this model is reconstructed. A new FE mesh is constructed on the basis of the worn wheel profile. The damages calculated by means interpolation of the damages accumulated in the nodes of the mesh at the previous *wheel wear iteration* are assigned to the obtained nodes of the new mesh. Then the procedure of accumulation of RCF damage in the nodes of the new FE mesh of the wheel is repeated.

Figure 25.36 shows the distribution of accumulated damage in the material of the wheel of a freight car with the GOST profile [7] after the traveled distance of 20, 100 and 200 thousand km.

The number of the *wheel wear iterations*, the total traveled distance and traveled distance at one *wheel wear iteration* are determined by the set of parameters specified in the wheel profile evolution project.

Let us consider the example of setting these parameters in **UM Simulation**.

The required group of parameters is located at the **Rail/Wheel | Wear | Wear parameters** tab in the **Object simulation inspector**.

- **Number of iterations** = 10;
- **Number of wear steps** = 2000;
- **Mileage (km)** = 5.

In accordance with dependence (16.12) of Sect. 16.1.1.3. " The wear simulation parameters. The profile-updating procedure" of Chapter 16 the total traveled distance of the wheel will be

$$km_{tot} = N_{it} N_{ws} km_{step} = 10 \cdot 2000 \cdot 5 = 100\,000 \text{ км}.$$

As a result, for the RCF project we will get 10 *wheel wear iterations*. The traveled distance of the wheel at one *wheel wear iteration* will be 10,000 km.

If user is going to model the accumulation of RCF damage in the wheel material, then at the stage of creation of the evolution project he needs to determine how many *wheel wear iterations* will be required to solve this task and the traveled distance at one iteration. The increase of traveled distance leads to the increase of wear and, consequently, to a redistribution of forces in the contact of the wheel and rail due to the change of the shape of the contacting surface of the wheel.

Notice that in the evolution project the number of changes of the wheel profile is assigned by the **Number of wear steps** parameter, and for each new wheel wear iteration the profile assigned by the **Number of iterations** parameter is saved. That is from the series of the evolutive profiles the profile obtained at the last step of the previous iteration of the calculation of the wear is chosen, and it is considered invariable at one *wheel wear iteration*. During simulation of the accumulation of RCF damage within one *wheel wear iteration* the forces obtained taking into account the evolution of the wheel profile in the evolution project are applied to the nodes of the mesh of FE model of the wheel. Such approach provides the acceptable accuracy of solution of the problem of determination of the stresses in the material of the wheel and allows setting the sufficiently big traveled distance at one *wheel wear iteration*.

On the other hand, as a result of wear, the surface layer of the material of the wheel is removed, which leads to the redistribution of the accumulated damage. This distribution changes significantly with increasing wear (see Figure 25.36). Therefore, it is not advisable to set a very big traveled distance at one *wheel wear iteration*. Based on these considerations, it is desirable to divide the total traveled distance of the wheel into a quite big amount of the segments, to each of which one *wheel wear iteration* will be corresponded.

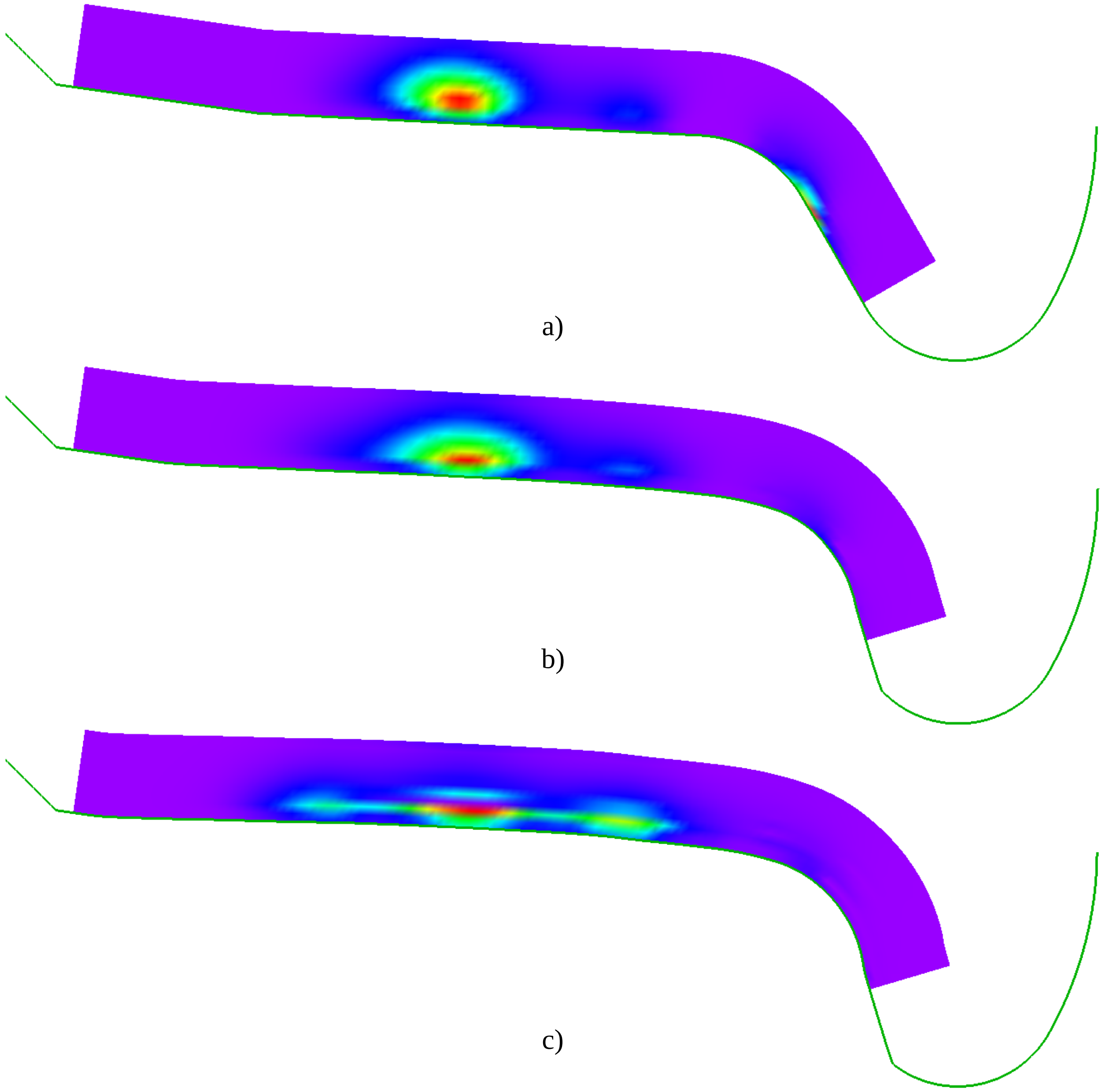


Figure 25.36. Isochromatic lines of accumulated damage in material of the car wheel with GOST [7] profile after traveled distance: a – 20,000 km; b – 100,000 km; c – 200,000 km

However, it should be taken into consideration that the excessive increase of the number of *wheel wear iterations* entails the increase of the computation time during simulation of the accumulation of RCF damage and often cannot bring the problem to a more exact solution. In addition, the size of the input data files (Sect. 25.3.1) increases, each of which depending on the evolution project settings can take up to several gigabytes on the hard disk.

The required number of *wheel wear iterations* and the traveled distance at one iteration are determined by testing a specific model. As a tentative value during simulation of the accumulation of RCF damage can be recommended the traveled distance at one *wheel wear iteration* of 5-10 thousand km.

The number of the contact problems that need to be solved at one *wheel wear iteration* during simulation of the process of accumulation of RCF damage is determined by the length of the way traveled by the vehicle when modelling the dynamics, and by the **Number of records per wheel revolution** parameter on the **Rail/Wheel | Wear | Results** tab in the **Object simulation inspector** (Figure 25.31). This parameter can accept only positive integer values greater than zero. With a value of 1 the number of contact problems is equal to the number of wheel revolutions at the given distance. As the value of the parameter increases, the number of contact problems increases by a multiple of this value. The increase of the value of the **Number of records per wheel revolution** parameter makes the set of statistics about the contacts of the wheel and the rail obtained as result of simulation of the vehicle movement more representative, but increases the computation time during simulation of the process of accumulation of RCF damage.

25.3.6. Consideration of presence of lubricant in wheel and rail contact

It has already been noted in Sect. 25.2.4 that during testing the specimens of the wheel steel under contact fatigue without the presence of the lubricant in the contact the damage is not observed even at very high pressures. This is due to the mechanism of generation of the cracks [16]. As a rule, the RCF tests of the wheel steel specimens in laboratory conditions give the understated number of cycles until arising fatigue defects, since they are carried out with continuously feeding the lubricant in the form of oil or water on the contact surfaces. Such conditions are harder than the real service conditions of the wheels. Under operating conditions rainwater is natural lubricant in the contact of the wheel and the rail. Obviously, depending on weather conditions such lubricant is not always present in the contact. Therefore, statistics show that the RCF durability of the wheels is higher than that of the laboratory specimens.

In order to obtain the results that are closer to the statistical operational data, the so-called "lubrication coefficient" K_{lub} is introduced in the **UM RCF Wheel** module. This coefficient is set by the user. It is dimensionless and ranges from 0 to 1. The coefficient is calculated for the specific operating conditions of the railway vehicle. For example, if a railway vehicle operates outdoors in a particular geographic region then it can be defined as the ratio of the number of rainy days per year, typical for that region, to the number of all days in the year. In other operating conditions you can take the ratio of the distance traveled by the vehicle with the presence of a lubricant in the contact between the wheel and the rail to the entire distance traveled by the vehicle.

With the introduction of the coefficient K_{lub} during simulation of the accumulation of RCF damage a modified form of the dependence (25.43) is used in the calculations

$$Q = K_{lub} \sum_{m=1}^{N_{it}} \sum_{i=1}^{N_C} \frac{N_{ws} k m_{step}}{S_i - S_{b_i}} \alpha_i \sum_{j=1}^{N_{v_i}} \beta_{ij} \sum_{k=1}^{N_{LC_i}} \frac{1}{N_n(k)}. \tag{25.45}$$

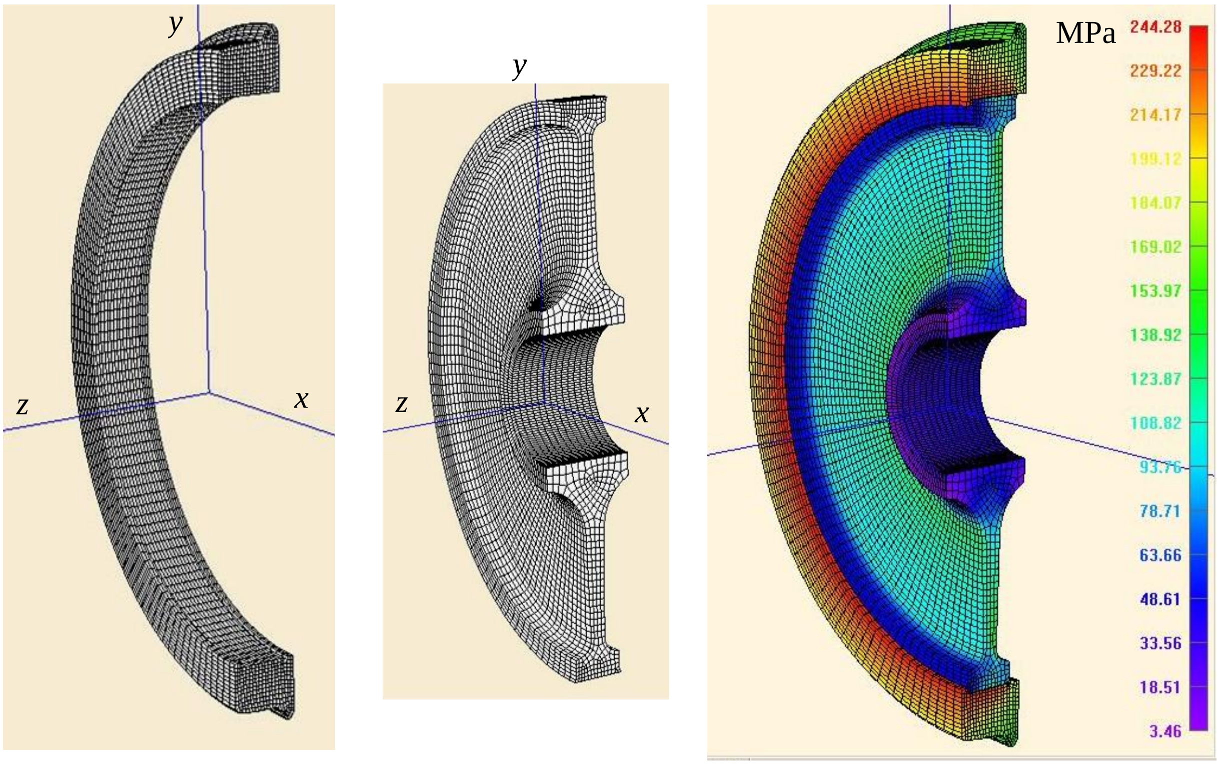
The default lubrication coefficient is 0.33.

25.3.7. Consideration of residual stresses in wheel material

The residual technological stresses occurring in the process of manufacturing the wheels and the mounting stresses caused by the interference fit of the wheel rim on the wheel plate can be represented as stationary fields in the wheel material. Since the problems of determining such stresses are solved in the axisymmetric formulation, the obtained stress distributions are attributed to one radial section of the wheel.

In work [38] using the ANSYS software the residual technological stresses in the material of the all-rolled car wheel made of the steel grade 2 [7] were calculated.

The specialists of Laboratory of Computational Mechanics of BSTU performed the calculation of the mounting stresses in the locomotive wheel [39] caused by the interference fit of the wheel rim on the wheel plate. The stresses were calculated using the ITFEMCP software package oriented at solving the contact problems. Figure 25.37 shows the FE model of the wheel and the field of the equivalent stresses by the Mises criterion in coloration.



a) b) c)

Figure 25.37. Mounting stresses in the locomotive wheel caused by the interference fit of the wheel rim on the wheel plate: FE model of wheel rim (a) and wheel plate (b); regions of equal equivalent stresses by the Mises criterion (c)

For using in the process of simulation of accumulation of RCF damage the information about the distribution of the residual stresses in the radial section of the wheel is specified in the text file. The file has the following structure:

1) the number of points of the radial section of the wheel;

2) the coordinates of the points *x* and *y*;

3) the values of normal and shear stresses for each point – σ_x, σ_y, σ_z, τ_{xy}, τ_{yz}, τ_{zx}.

The origin is chosen at the point of the wheel profile located on the running circle. The wheel profile should be oriented so that the flange is located on the right. The positive direction of the *Ox* abscissa axis is to the right – to the flange, the positive direction of the *Oy* ordinate axis is upward – to the axis of rotation of the wheelset (Figure 25.38). Coordinates are given in millimeters, stresses – in megapascals.

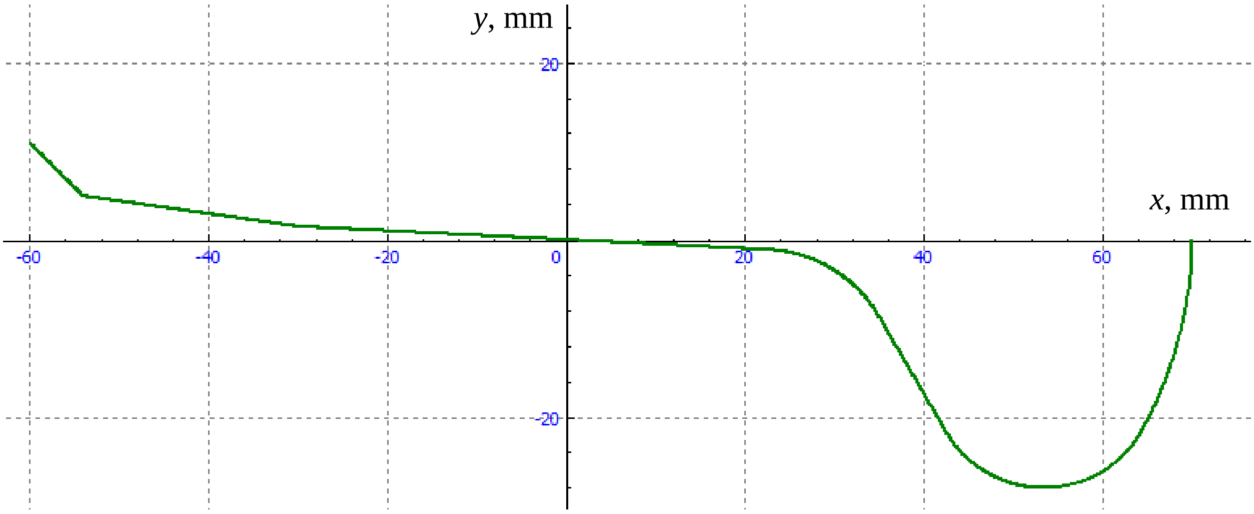


Figure 25.38. Reference system for assignment of coordinates of the points of profile of radial cross-section of the wheel

The **UM** library includes two files of the format described above. The first file contains data about the residual technological stresses in the material of the all-rolled car wheel from the steel grade 2 [7], wheel diameter is 954 mm:

{UM Data}\RCF\Wheel\Residual technological stresses - wheel D954 mm.txt

The second file contains data about the mounting stresses in the material of the locomotive wheel [39] caused by the interference fit of the wheel rim on the wheel plate, the wheel diameter is 1250 mm:

{UM Data}\RCF\Wheel\Interference fit stresses - wheel D1250 mm.txt

After loading the residual stresses data from the file into the **UM RCF Wheel** project the values of the residual stresses in the nodes of the flat FE mesh of the wheel fragment are calculated using interpolation.

Figure 25.39 shows the distributions of the circumferential, axial and radial residual technological stresses in the radial section of the car wheel after loading and processing in the **UM RCF Wheel** project.

The distributions of the circumferential, axial and radial mounting stresses caused by the interference fit of the wheel rim on the wheel plate in the radial section of the locomotive wheel after loading and processing in the **UM RCF Wheel** project are shown in Figure 25.40.

In process of simulation of the accumulation of RCF damage the field of the alternating loading stresses obtained from the solution of the contact problem is superimposed on the stationary field of the residual stresses, the normal and shear stresses of both fields are summed. Then the value of the criterion of the RCF failure in the nodes of the FE mesh located in the radial section of the wheel is calculated.

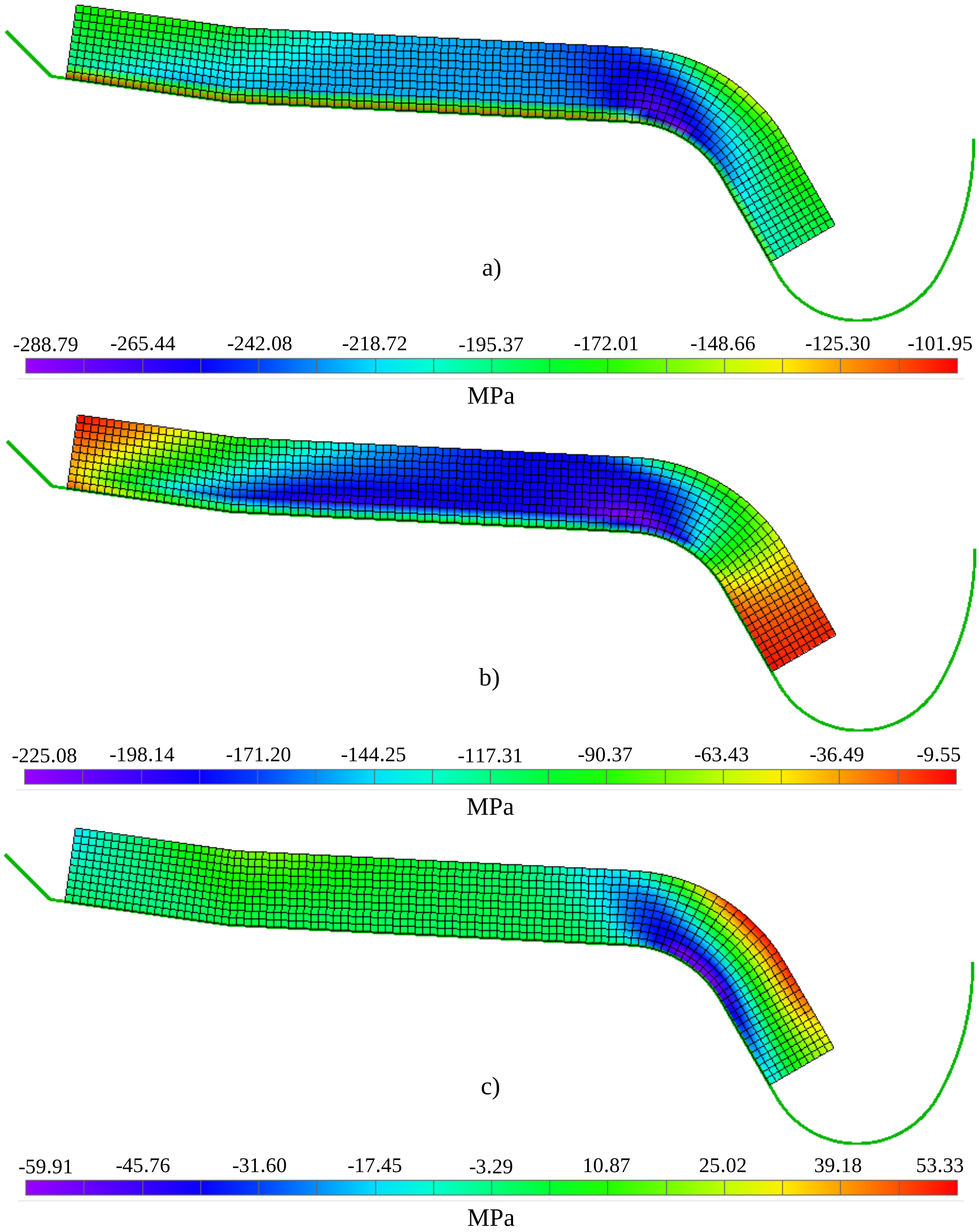


Figure 25.39. Distribution of residual technological stresses in the radial section of the car wheel: a – circumferential; b – axial; c – radial

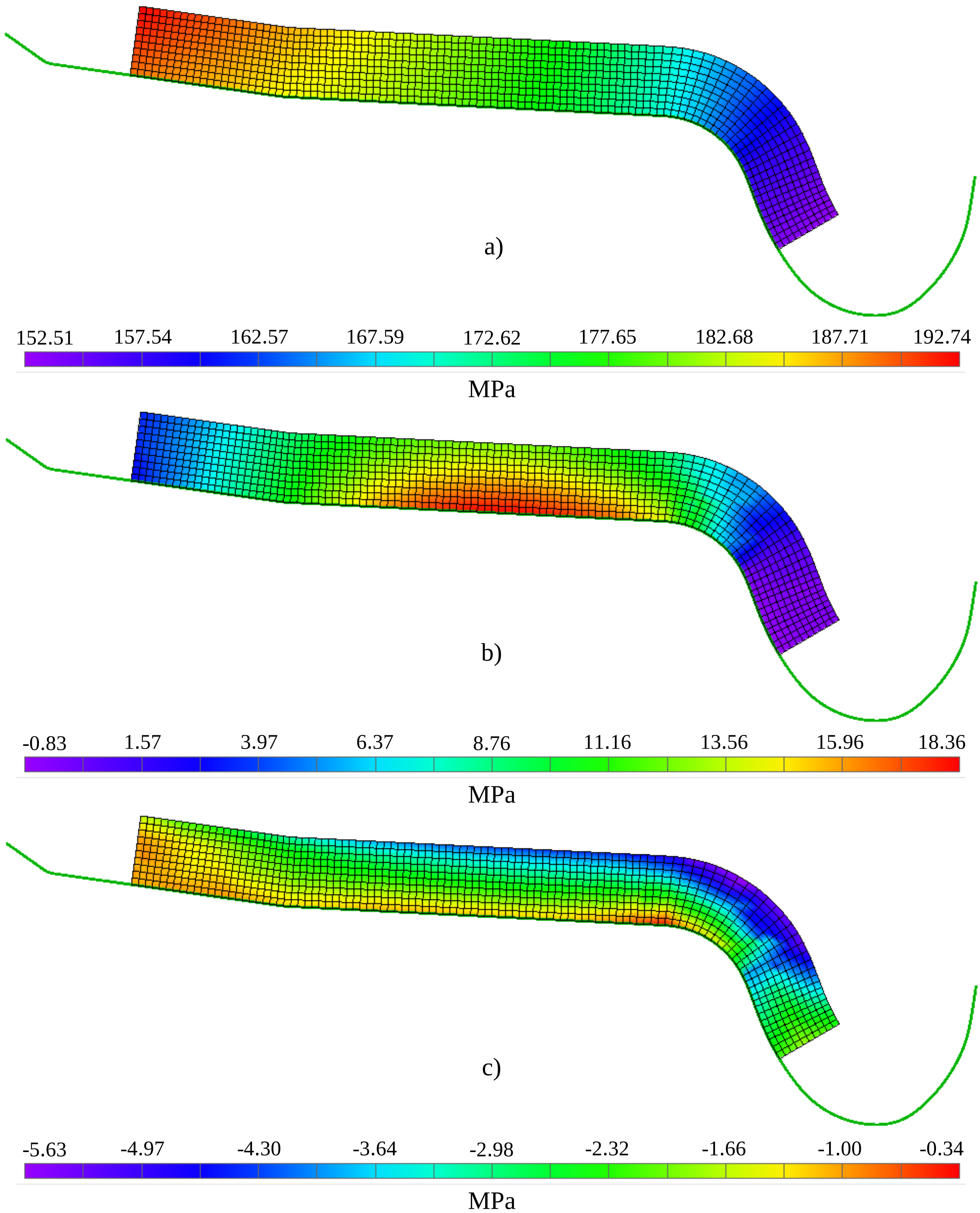


Figure 25.40. Distribution of mounting stresses caused by the interference fit of the wheel rim to the wheel plate in the radial section of the locomotive wheel:
a – circumferential; b – axial; c – radial

Figure 25.41a shows the areas of equal equivalent stresses by the combined criterion (Sect. 25.2.5.4) in the radial section of the car wheel for one of the variant of the location of the contact obtained from the solution of the contact problem without taking into account the residual stresses. The point with the maximum value of the criterion of 170 MPa is located at the depth of 3 mm below the contact surface. As seen from Figure 25.41b, accounting the residual technological stresses significantly changes the distribution of the equivalent stresses. Due to the big compressive circumferential stresses (Figure 25.39a) the equivalent stress in the point located at the depth of 3 mm decreases to 89 MPa. However, the point with the maximum value of the criterion of 161 MPa is now located on the contact surface.

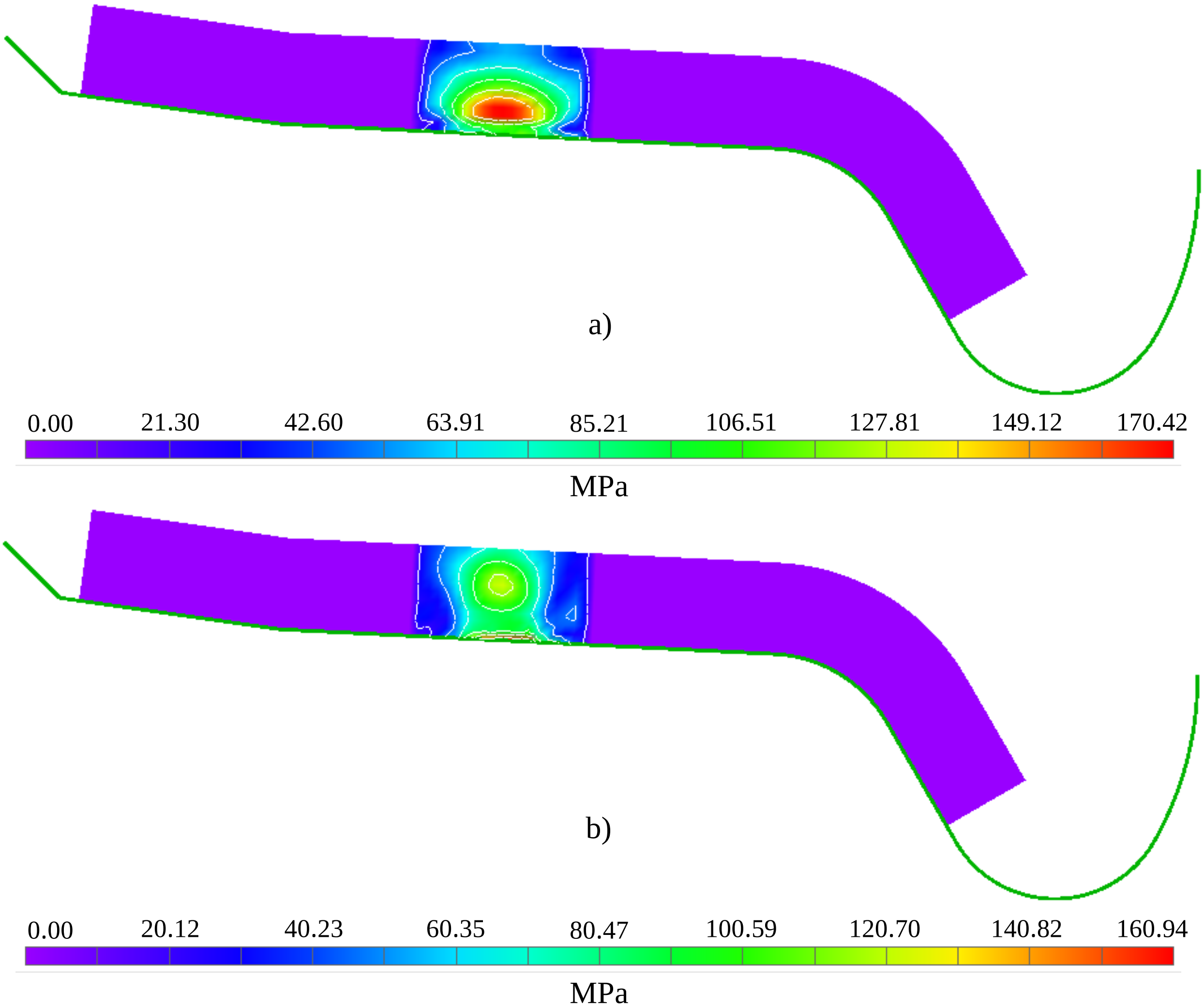


Figure 25.41. Areas of equal equivalent stresses by combined criterion in the radial section of car wheel: a – without accounting the residual stresses; b – with accounting the residual technological stresses

Figure 25.42a shows the areas of equal equivalent stresses by the combined criterion (Sect. 25.2.5.4) in the radial section of the locomotive wheel for one of the variant of the location of the contact obtained from the solution of the contact problem without taking into account the residual stresses. The point with the maximum value of the criterion of 242 MPa is located at the depth of 4 mm below the contact surface. Accounting the mounting stresses caused by the interference fit of the wheel rim on the wheel plate does not significantly change the distribution of equivalent stresses (Figure 25.42b), the point with the maximum criterion value remains at the same depth. However, due to the big tensile circumferential stresses (Figure 25.40a) the equivalent stress in this point increases to 295 MPa.

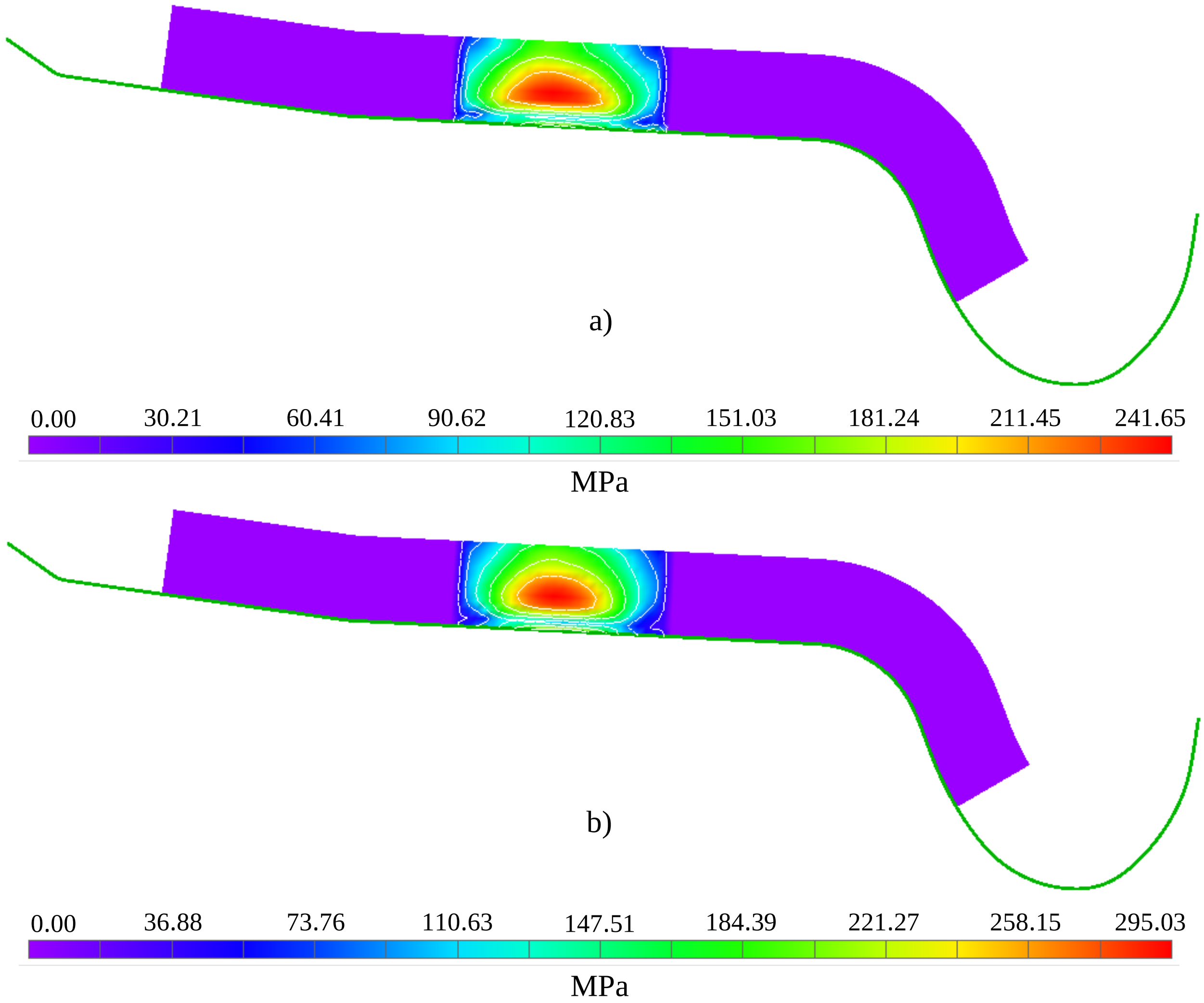


Figure 25.42. Areas of equal equivalent stresses by combined criterion in the radial section of locomotive wheel: a – without accounting the residual stresses; b – with accounting the mounting stresses caused by the interference fit of the wheel rim on the wheel plate

25.4. Working with UM RCF Wheel module

25.4.1. Running UM RCF Wheel module

The RCF project file is created in **UM Loco** (Chapter 8) module with help **UM Loco/Wheel Profile Wear Evolution** tool (Chapter 16), see Sect. 25.3.1.

During or after installation **UM** software on computer it is recommended to associate the files with ***.rcf** extension with RCF project files of **UM RCF Wheel** module. In this case the **UM RCF Wheel** module will be automatically run by double click on such files.

Another way to run the module is using **Start** button and then **All programs | Universal Mechanism 9.0 | Tools | UM Rolling Contact Fatigue of Wheel** menu item. After running **UM RCF Wheel** module user needs to open the RCF project file ***.rcf** with help **File | Open** menu item or  button on the **Standard** toolbar.

25.4.2. Interface of module

Interface of the **UM RCF Wheel** module is presented in Figure 25.43.

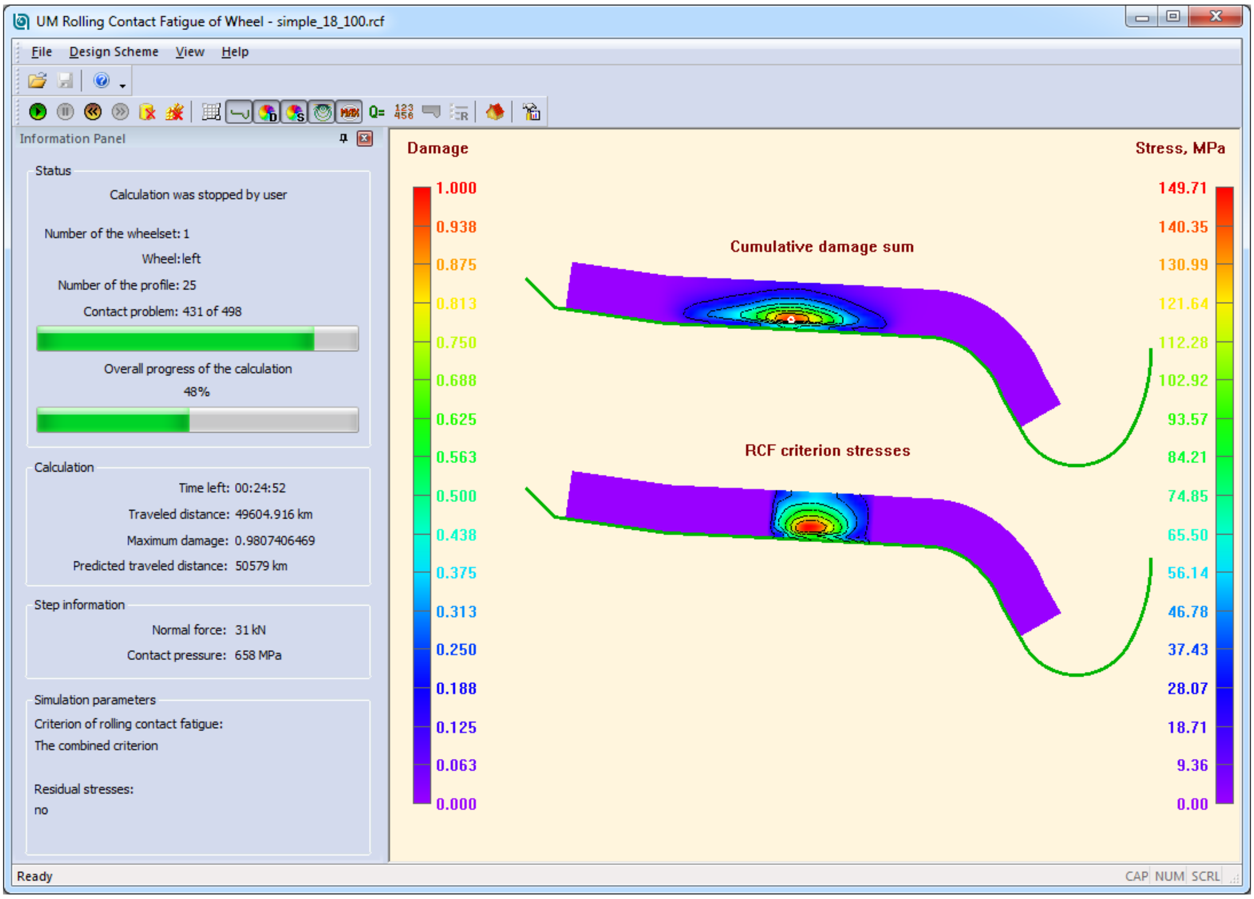


Figure 25.43. Interface of **UM RCF Wheel** module

In the top of the window the title bar, menu bar, **Standard** toolbar and **Design Scheme** toolbar are located. In the bottom the status bar is located. In the left side of the window the floating **Information Panel** is placed. The user can dock it to the left or right side of the window or hide it. By default, the **Information Panel** is attached to the left side of the window and is always visible whenever the module is run.

The user can show or hide the toolbars and **Information Panel** and change its settings by the **View | Toolbars and Docking Windows** menu or using the context menu by clicking the right mouse button. The user can also change the interface of the module window using the **View | Application Look** menu.

The workspace of window is intended for the graphical displaying of design scheme and results of calculations. If the data for calculation is loaded, in the center of the workspace always is displayed the wheel profile and FE mesh. All display modes can be switched on or off, for example, using the buttons on the **Design Scheme** toolbar. If during calculations the coloring modes are switched on, the equivalent stresses in the plane of symmetry of FE fragment of the wheel are shown in the center of the workspace. The cumulative RCF damage in the same plane is shown on top. The color scale for estimate of the level of accumulated RCF damage is on the left and the color scale for the estimate of the equivalent stresses is on the right.

25.4.3. Parameters

The **Parameters** option becomes available when the data for calculation is loaded. In Figure 25.44 the dialog box is shown, which appears on the screen with help the **Design Scheme | Parameters...** menu item or button .

The appeared dialog box has four tabs: **Calculation, Material**, **Residual stresses** and **Visualization**. The **Reset to default** button allows returning to the initial values of the parameters at the selected tab. The changed parameters are saved only after clicking **OK**. Otherwise, the changes will not be saved.

25.4.3.1. The «Calculation» tab

The **Calculation** tab is shown in Figure 25.44.

Modelling of the process of accumulation of RCF damage in the **UM RCF Wheel** module provides two variants of the finish of calculation:

1) the reaching of limit of the accumulated damage sum in the wheel point;

2) the passage of the entire distance specified in modelling the dynamics of the railway vehicle (traveled distance).

In the first case user has to set the limit value of cumulative damage sum. The accumulation of RCF damage at the point of wheel is calculated by using the Eq. (25.45), the accumulation of RCF damage in the nodes of FE scheme is reviewed in Sect. 25.3.4. After reaching of limit of the cumulative damage sum the calculation will be continued until the finish of the current *wheel wear iteration* (see Sect. 25.3.5), after that the calculation for the next wheel is started.

In the second case the calculation will be realized for all *wheel wear iterations*, for all wheels listed in the ***.rcf** project, regardless of the accumulated damage in the wheel.

By default, the first variant **Limited value of the cumulative damage sum** is chosen and the value of **1** is set.

The calculation of each new *wheel wear iteration* begins with the preparing the stiffness matrix of FE fragment of the wheel (see Sect. 25.3.2, 25.3.5). This operation can take the several minutes depending on computer speed. In this connection, the total computation time significantly increases, particularly if the calculation is carried out for tens or hundreds of *wheel wear iterations*. In any case, it fails to avoid these time-consuming operations at the first calculation. But this possibility appears if necessary to repeat the calculation for the same model with other conditions.

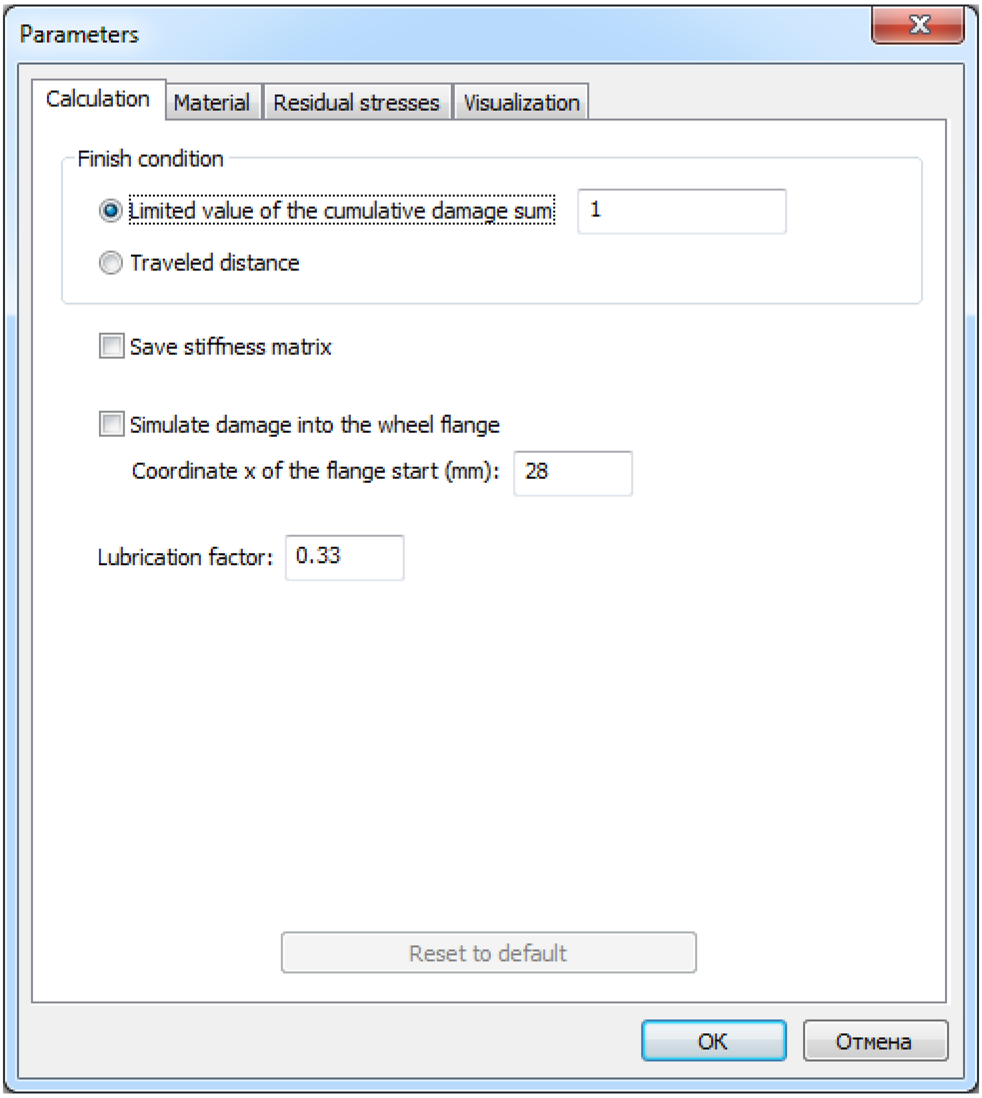


Figure 25.44. The **Parameters** dialog box, the **Calculation** tab

The checked option **Save stiffness matrix** means that the stiffness matrix for each new *wheel wear iteration* will be saved in the ***.smx** file. This file has a binary format and it is located in the same folder as the file with the initial data. Size of this file is greater than 100 MB. So it is needed to have enough free space in the partition of the hard drive where the files are stored. Now during the calculation the components of the stiffness matrix will be loaded from the file, rather than calculated, that minimizes the computation time.

However, use of this option carefully.

Following variables are used for calculating the components of the stiffness matrix:

- the number of nodes of the FE model;
- the coordinates of nodes of the FE model;

- the Young’s modulus *E*;
- the Poisson's ratio *μ*.

If the coordinates of points of the profile, the dimensions of FE model, the Young's modulus or Poisson's ratio were changed, before the repeated calculation the stiffness matrix must be formed anew. Accordingly, the previous matrix must be destroyed. The algorithm realized in the **UM RCF Wheel** module works as follows: first, an attempt to open the ***.smx** file corresponding to the initial data files is made – identification is done by file name. If such file exists, the components of the matrix are loaded from it, else the components of the stiffness matrix are calculated.

Note. If initial data was changed, it is necessary to delete the previous stiffness matrices before starting the new calculation. Otherwise it will lead to incorrect results.

The module reacts on changes of model parameters, but cannot react on all the possible actions of user. For example, if changes to the files of initial data are made outside of the **UM RCF Wheel** module or **UM** software. For this reason, the **Save stiffness matrix** option is inactive by default.

The files of stiffness matrices can be deleted by the **Design Scheme | Delete stiffness matrix** menu item or by  button.

The **Save stiffness matrix** option can be useful in two cases.

1) The calculation has been interrupted, the module is closed. After restarting the module and resuming the calculation is necessary to form the stiffness matrix for current *wheel wear iteration*. If before that it was saved in the file, then the components of the matrix will be quickly loaded from the file.

2) The calculation was carried out with the specific conditions. User decides to repeat the same calculation by changing the one or several parameters. For example, the RCF criterion is changed. In this case, it is needed to delete the result files and return to the start of the calculation (see Sect. 25.4.7). At that the files of stiffness matrices are not deleted. Repeated calculation will carry out faster.

The **Simulate damage into the wheel flange** item (Figure 25.44).

Slip in the contact of the wheel flange and the side surface of the rail leads to the big shear stresses on the surface of the wheel flange and fast accumulation of RCF damage on this surface. However, as a result of intensive wear the surface layer with accumulated damage quickly disappears. Statistical operational data confirms that RCF defects on the contact surface of the wheel flange are rare. Therefore the **UM RCF** module allows to exclude the simulation of the process of accumulation of RCF damage in the wheel flange – by default, the **Simulate damage into the wheel flange** item is not activated. In this case, the user needs to specify a section of the wheel profile, which will be considered as a flange. The **Coordinate x of the flange start (mm)** parameter is used for this purpose, see Figure 25.44. The default value of this parameter is 28 mm. The coordinate is assigned in the reference system shown in Figure 25.38. Thus, part of the FE model of fragment of the wheel whose *x* coordinates of nodes are greater than the specified coordinate *x* of the flange start is considered as the wheel flange.

The purpose of parameter **Lubrication factor** (Figure 25.44) is described in detail in Sect. 25.3.6.

If calculation parameters at the **Calculation** tab were changed, then after clicking the **OK** button a dialog box appears, see Figure 25.45.

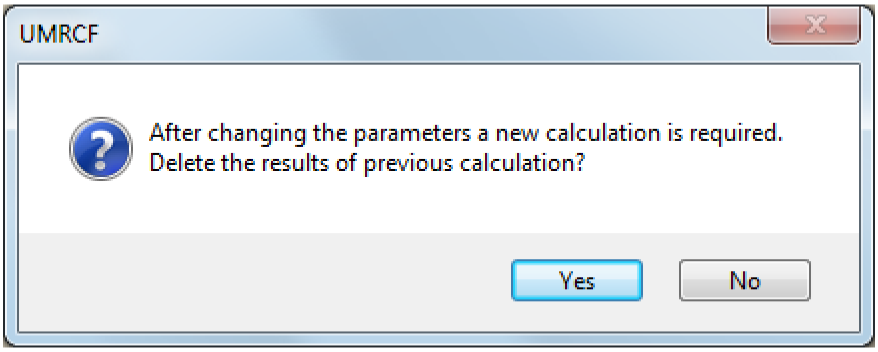


Figure 25.45. Dialog box – request to remove the results of the calculation

The **No** button means that the parameter changes will not be saved.

The **Yes** button means that the parameters will be saved and files with the simulation results will be deleted. The stiffness matrix files, if previously created, will not be deleted. After that the calculation with new parameters can be started.

25.4.3.2. The «Material» tab

The **Material** tab is shown in Figure 25.46.

Young's modulus *E* ($2\cdot10^{11}$ Pa is the value by default) and Poisson's ratio μ (0.3 is the value by default) can be assigned at the tab.

The RCF curve is approximated by the function (25.1), see Sect. 25.2.4. The values of the material constants *C* and *m* in this expression are assigned depending on the chosen RCF criterion. A criterion can be chosen in the **RCF criteria** group box. The criteria are described in the following sections:

1) The criterion of the amplitude of maximum shear stress – see Sect. 25.2.5.1.

2) The Dang Van criterion – see Sect. 25.2.5.2.

3) The Sines criterion – see Sect. 25.2.5.3.

4) The combined criterion – see Sect. 25.2.5.4.

The RCF curves and values of the material constants *C* and *m* for these criteria are given in Sect. 25.2.8.

The user can also set the minimum value of the damaging stress according to the selected RCF criterion. During modelling of the process of accumulation of the RCF damage the cycle is considered as damaging if the value of the criterion exceeds this limit. This minimum value is equal to zero by default.

If any of the parameters at the **Material** tab was changed, then after clicking the **OK** button the dialog box appears as shown in Figure 25.45.

The **No** button means the exit without saving the changed parameters.

The **Yes** button means the exit with saving the changed parameters. The result files will be deleted. If the Young's modulus or Poisson's ratio is changed, the stiffness matrices files will be also deleted.

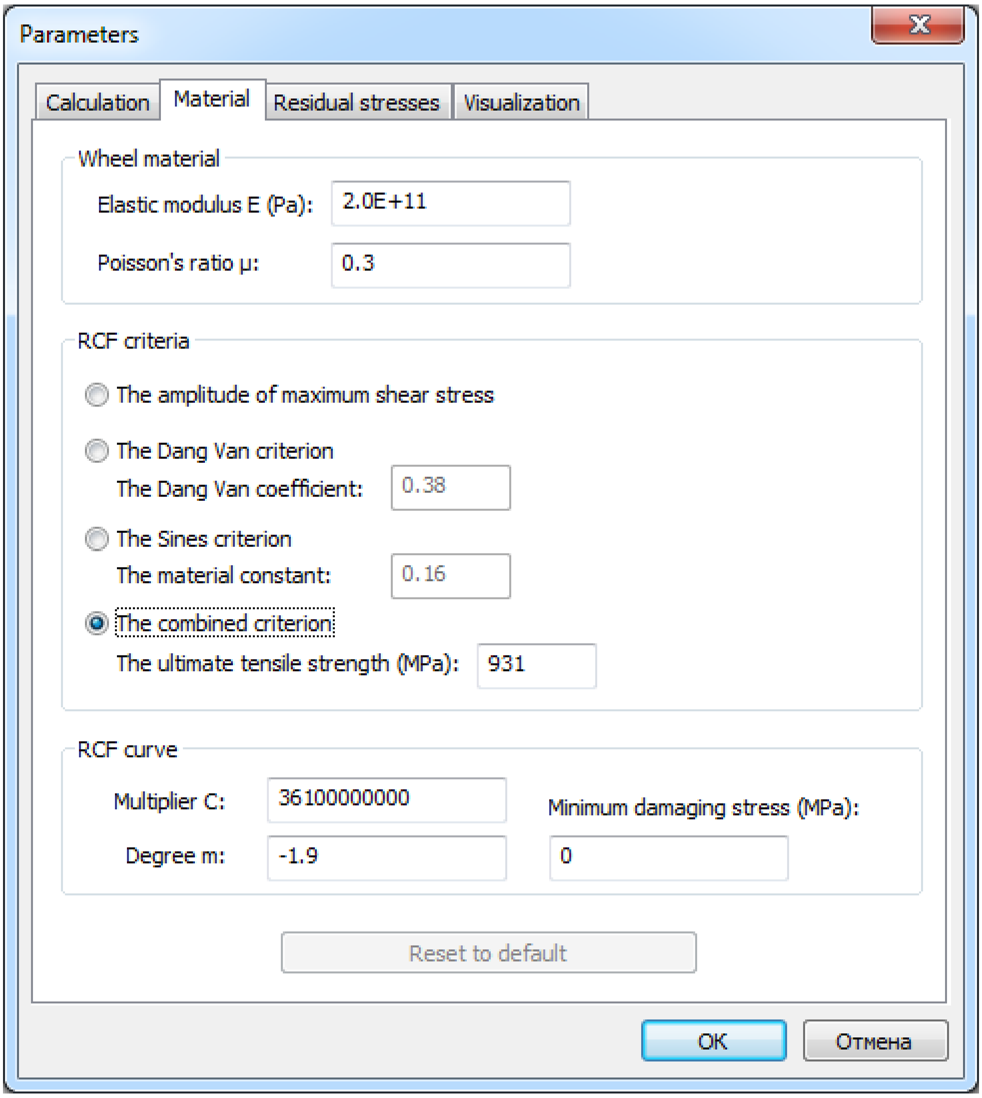


Figure 25.46. The **Parameters** dialog box, the **Material** tab

25.4.3.3. The «Residual stresses» tab

The **Residual stresses** tab is shown in Figure 25.47.

With help of this tab the user can include in the calculation two types of residual stresses: residual technological stresses in the wheel after manufacture and mounting stresses in the wheel caused by the fitting of the rim to the wheel plate. Residual stresses data is stored in the text files.

An approach to modelling of the process of accumulation of the RCF damage accounting the residual stresses in the wheel material is described in Sect. 25.3.7.

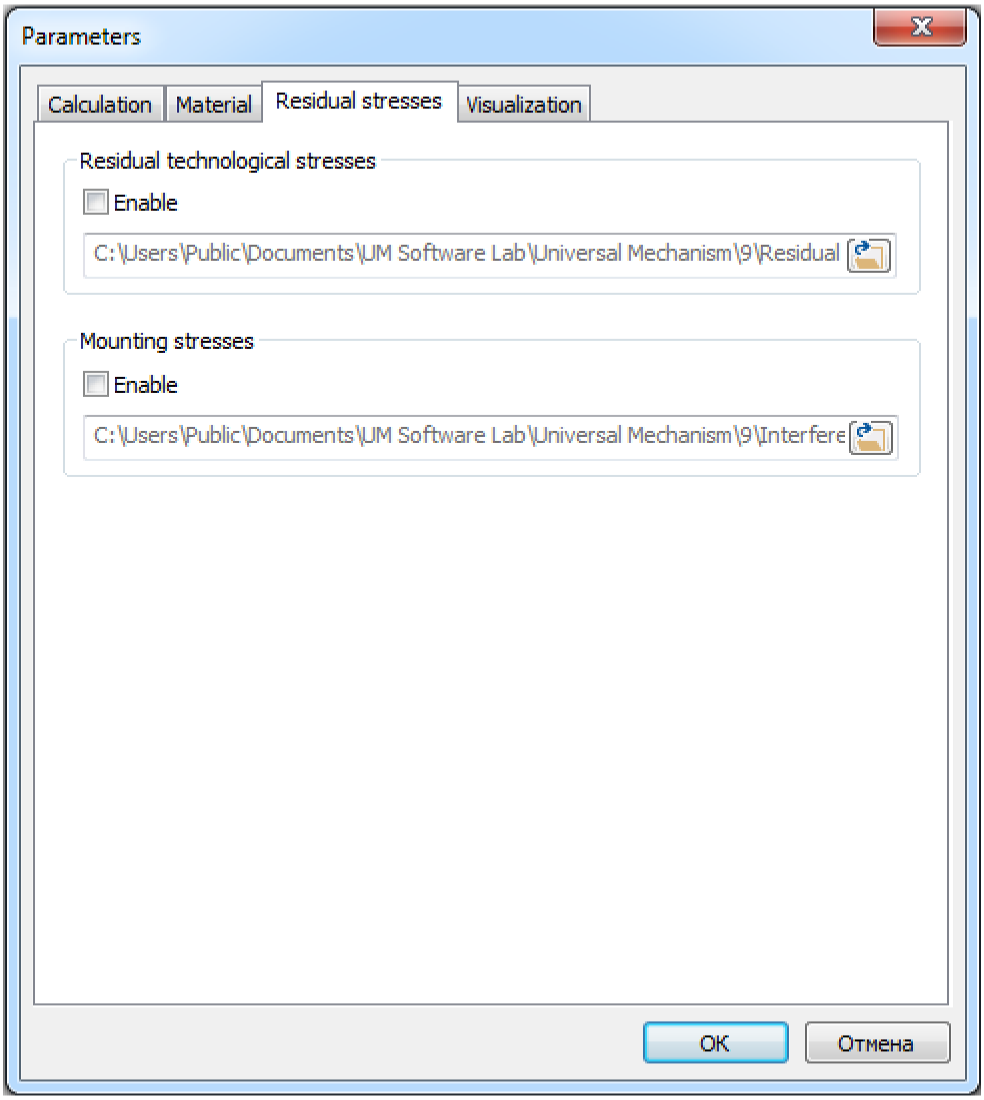


Figure 25.47. The **Parameters** dialog box, the **Residual stresses** tab

25.4.3.4. The «Visualization» tab

The **Visualization** tab is shown in Figure 25.48.

The design scheme can be moved in the window in any direction by using the arrow keys on keyboard (or by the left mouse button), it can be zoomed by pressing the «Gray +» and «Gray -» keys on keyboard (or by the mouse wheel). All these operations are performed discretely with a determined step. The steps values can be changed at the **Visualization** tab. The steps are assigned in conventional units.

Also the background color of the workspace of window, the color and width of the lines for displaying the wheel profile, FE mesh and isolines can be assigned on the tab.

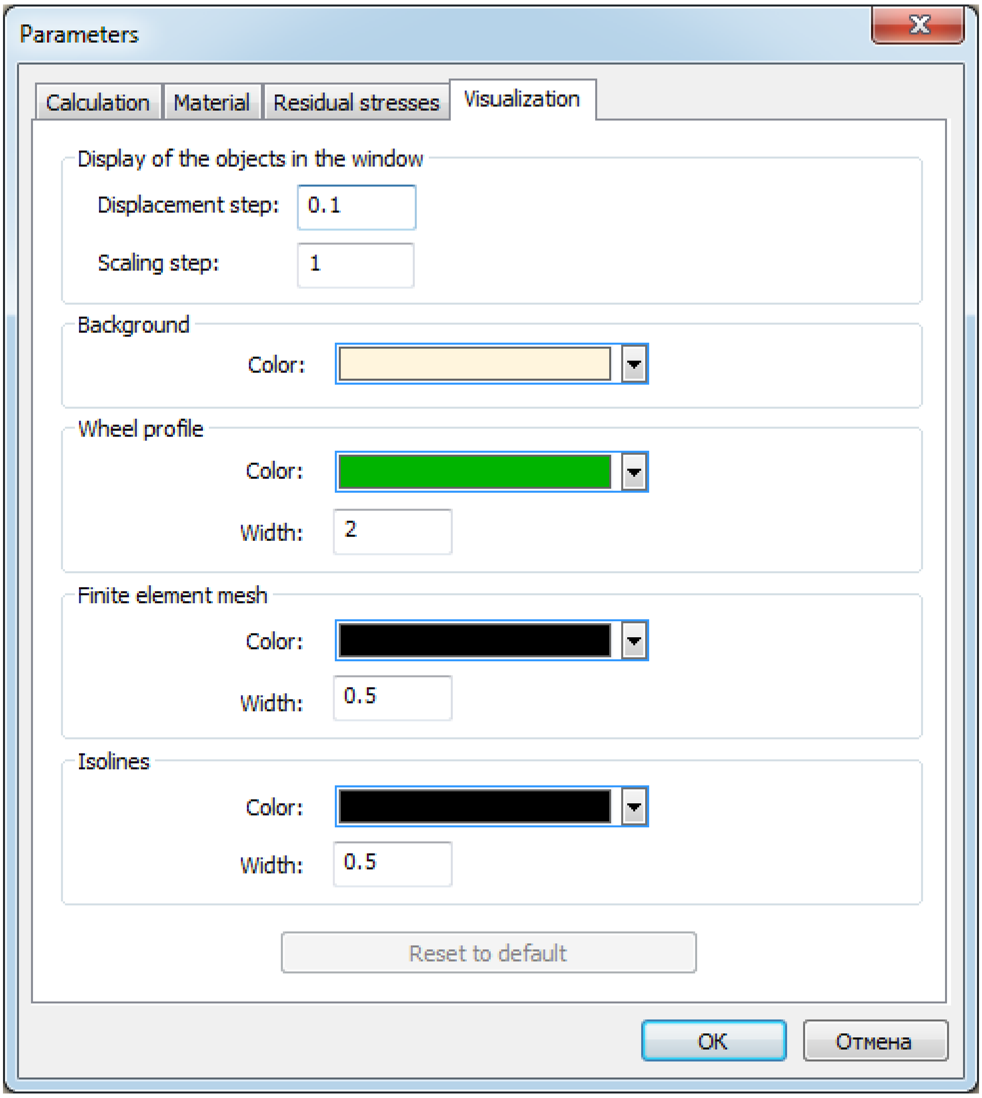


Figure 25.48. The **Parameters** dialog box, the **Visualization** tab

25.4.4. Imaging modes

The several types of the imaging modes of design scheme and calculation results, which can be switched on or off in the various combinations with each other, are implemented in the module.

If necessary, the design scheme in the window can be moved, zoom in or out. The scheme can be returned to its original position in the following ways:

– choose the **Design Scheme | Starting position** menu item;

– click the button  at the **Design Scheme** toolbar;

– click the **Home** key on the keyboard;

– right-click in the workspace of window, choose the **Starting position** item in the appeared shortcut menu.

25.4.4.1. The «Mesh» mode

The **Mesh** imaging mode can be enabled or disabled in three ways:

– choose the **Design Scheme | Mesh** menu item;

– click the button  at the **Design Scheme** toolbar;

– click the **1** key on the keyboard.

The mesh from quadrangular elements, which is located at the plane of symmetry of the FE fragment of wheel, is displayed in the center of workspace of the window (Figure 25.32). If with the enabled **Mesh** mode the **Color visualization of damage** mode is enabled, then the similar mesh is displayed in the top of window at the median plane of wheel fragment, where the process of accumulation of RCF damage is illustrated.

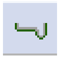
The mode is automatically activated when the data is loaded.

Note. The **Mesh** mode is always available if the data for calculation is loaded.

25.4.4.2. The «Wheel profile» mode

The **Wheel profile** imaging mode can be enabled or disabled in three ways:

– choose the **Design Scheme | Wheel profile** menu item;

– click the button  at the **Design Scheme** toolbar;

– click the **2** key on the keyboard.

The wheel profile is displayed in the center of workspace of the window (Figure 25.43). If with the enabled **Wheel profile** mode the **Color visualization of damage** mode is enabled, then the similar wheel profile is displayed in the top of window, where the process of accumulation of RCF damage is illustrated.

The mode is automatically activated when the data is loaded.

Note. The **Wheel profile** mode is always available if the data for calculation is loaded.

25.4.4.3. The «Color visualization of damage» mode

The **Color visualization of damage** imaging mode can be enabled or disabled in three ways:

– choose the **Design Scheme | Color visualization of damage** menu item;

– click the button  at the **Design Scheme** toolbar;

– click the **3** key on the keyboard.

In the top of window the radial cross-section of the wheel is displayed, in which the distribution of the accumulated RCF damage with help coloring is shown (Figure 25.43). The coloring is performed for each quadrangular element of flat mesh. The color of the node is chosen depending on the value of the accumulated RCF damage according to an estimation scale. Then the resulting colors from the four vertices are mixed in the plane of the element according to a linear law. In the left side of the workspace of window the color scale for estimation of the level of accumulated RCF damage is displayed.

Note. The **Color visualization of damage** mode is available if the data for calculation is loaded and the calculation has been done.

25.4.4.4. The «Color visualization of stresses» mode

The **Color visualization of stresses** imaging mode can be enabled or disabled in three ways:

– choose the **Design Scheme | Color visualization of stresses** menu item;

– click the button  at the **Design Scheme** toolbar;

– click the **4** key on the keyboard.

In the center of the workspace of window the radial cross-section of the wheel is displayed, in which the distribution of the equivalent stresses with help coloring is shown (Figure 25.43). The coloring is performed for each quadrangular element of flat mesh. The color of the node is chosen depending on the value of the equivalent stresses according to an estimation scale. Then the resulting colors from the four vertices are mixed in the plane of the element according to a linear law. In the right side of the workspace of window the color scale for estimation of the level of equivalent stresses is displayed.

Note. The **Color visualization of stresses** mode is available if the data for calculation is loaded and the calculation has been done.

25.4.4.5. The «Isolines» mode

The **Isolines** imaging mode can be enabled or disabled in three ways:

– choose the **Design Scheme | Isolines** menu item;

– click the button  at the **Design Scheme** toolbar;

– click the **5** key on the keyboard.

The mode is intended for displaying the lines of equal values of the component – isoparametric lines. The mode supports six isolines, that is, the component is divided into seven levels. If the **Color visualization of stresses** mode is enabled, then in the center of the workspace of window the radial cross-section of the wheel is displayed, in which the isolines of equivalent stresses are shown (Figure 25.43). If the **Color visualization of damage** mode is enabled, then in the top of the workspace of window the radial cross-section of the wheel is displayed, in which the isolines of accumulated RCF damage are shown (Figure 25.43).

Note. The **Isolines** mode is available if the data for calculation is loaded, the calculation has been done and the **Color visualization of damage** mode or **Color visualization of stresses** mode is enabled.

25.4.4.6. The «Maximum damage» mode

The **Maximum damage** imaging mode can be enabled or disabled in three ways:

– choose the **Design Scheme | Maximum damage** menu item;

– click the button  at the **Design Scheme** toolbar;

– click the **6** key on the keyboard.

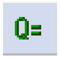
In the top of the workspace of window the radial cross-section of the wheel is displayed, in which the node with the biggest accumulated RCF damage is marked by the white circle (Figure 25.43). The value of the biggest damage can be viewed at the **Information Panel** in the **Calculation** parameters group box (Figure 25.50).

Note. The **Maximum damage** mode is available if the data for calculation is loaded, the calculation has been done, the **Color visualization of damage** mode is enabled and the biggest accumulated damage more than zero.

25.4.4.7. The «Accumulated damage» mode

The **Accumulated damage** imaging mode can be enabled or disabled in three ways:

– choose the **Design Scheme | Accumulated damage** menu item;

– click the button  at the **Design Scheme** toolbar;

– click the **7** key on the keyboard.

If the **Color visualization of damage** mode is enabled, then in the top of the workspace of window the radial cross-section of the wheel is displayed, in which the nodes of FE mesh are marked by the points, beside each node the numerical value of accumulated RCF damage in this node is shown. If the **Color visualization of stresses** mode is enabled, then in the center of the workspace of window the radial cross-section of the wheel is displayed, in which the nodes of FE mesh are marked by the points, beside each node the numerical value of equivalent stress in this node is shown.

Note. The **Accumulated damage** mode is available if the data for calculation is loaded, the calculation has been done and the **Color visualization of damage** mode or **Color visualization of stresses** mode is enabled.

25.4.4.8. The «Numbers of the nodes» mode

The **Numbers of the nodes** imaging mode can be enabled or disabled in three ways:

– choose the **Design Scheme | Numbers of the nodes** menu item;

– click the button  at the **Design Scheme** toolbar;

– click the **8** key on the keyboard.

In the center of the workspace of window the radial cross-section of the wheel is displayed, in which the nodes of FE mesh are marked by the points, beside each node its number is shown. If the **Color visualization of damage** mode is enabled, then the numbers are shown beside the nodes of radial cross-section of the wheel in the top of the workspace of window, where the process of accumulation of RCF damage is illustrated.

Note. The **Numbers of the nodes** mode is always available if the data for calculation is loaded.

25.4.4.9. The «Residual stresses» mode

The **Residual stresses** imaging mode can be enabled or disabled in two ways:

– choose the **Design Scheme | Residual stresses** menu item;

– click the button at the **Design Scheme** toolbar.

In the center of the workspace of window the radial cross-section of the wheel is displayed, in which the distribution of one of the components of residual stresses with help coloring is shown (Figure 25.39). The coloring is performed for each quadrangular element of flat mesh. The color of the node is chosen depending on the value of the stress component according to an estimation scale. Then the resulting colors from the four vertices are mixed in the plane of the element according to a linear law. In the right side of the workspace of window the color scale for estimation of the level of selected stress component is displayed.

The component of residual stresses for displaying can be chosen at the **Residual stresses** dialog box (Figure 25.49). To open the dialog box select the **Design Scheme | Choice of component** menu item or click the button at the **Design Scheme** toolbar.

Note. The **Residual stresses** mode is available if the residual stresses data is loaded (see Sect. 25.3.7, 25.4.3.3).

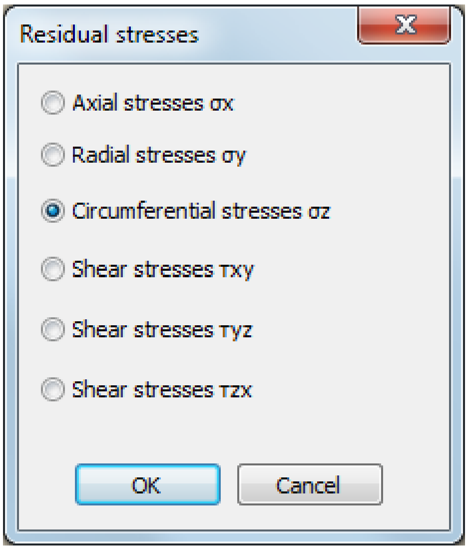


Figure 25.49. Selection of residual stresses components for displaying

25.4.5. Performing calculation

The calculation can be started in the following ways:

– choose the **Design Scheme | Run** menu item;

– click the button at the **Design Scheme** toolbar;

– click the **F9** key on the keyboard;

– right-click in the workspace of window, choose the **Run** item in the appeared shortcut menu.

The sequence of calculations is determined by the list of **wld** files in the RCF project **rcf** (see Sect. 25.3.1). Every **wld** file contains the input data for one wheel. Initially, the calculation taking into account the *wheel wear iterations* will be completely carried out for the first wheel, then for the second wheel and so on.

If the condition for the completion of the calculation is the reaching the limit value of the accumulated damage at the wheel point and it is reached earlier than the calculations for all *wheel wear iterations* were made, the remaining *wheel wear iterations* are skipped and the calculation for the next wheel is started. If the condition for the completion of the calculation is the travelled distance, the calculation is carried out for all *wheel wear iterations* of all wheels containing in the RCF project.

The **Information Panel** allows monitoring the status of the calculation (Figure 25.50).

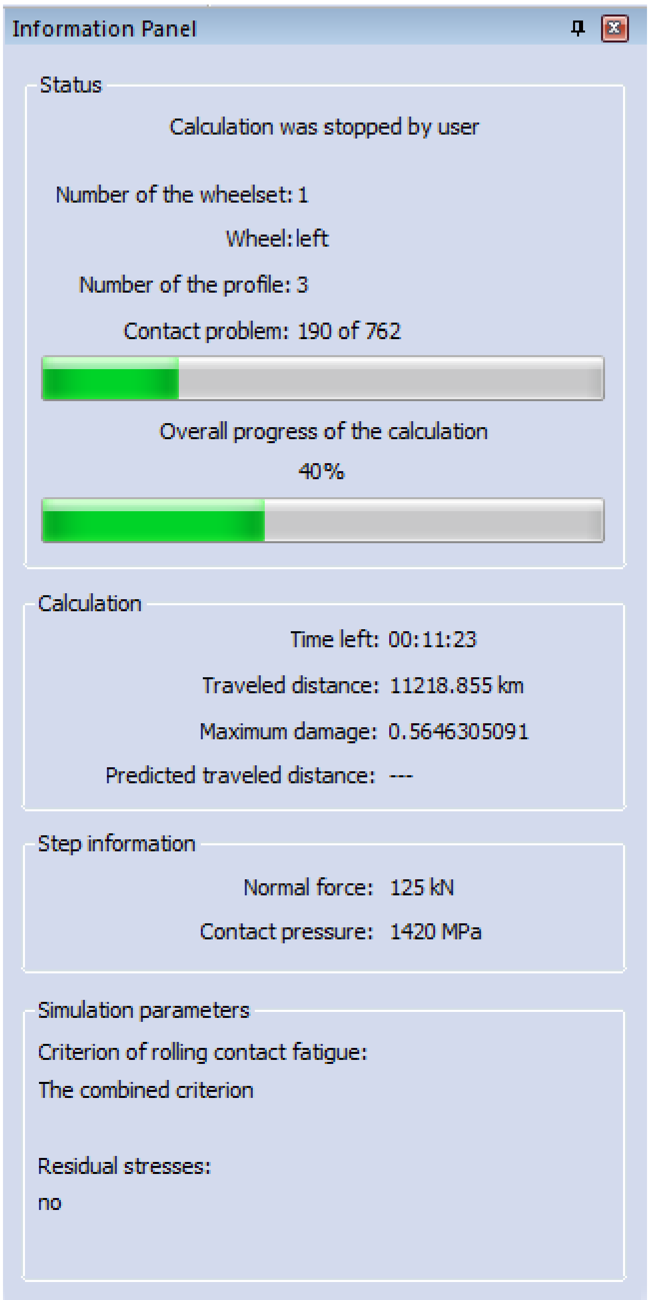


Figure 25.50. The **Information Panel**

After completion of the calculation the program displays a corresponding message in the dialog box on the screen.

The calculation can be stopped in the following ways:

– choose the **Design Scheme | Stop** menu item;

– click the button at the **Design Scheme** toolbar;

– click the **Esc** key on the keyboard;

– right-click in the workspace of window, choose the **Stop** item in the appeared shortcut menu.

25.4.6. Saving the results of calculation

The simulation results can be saved by means the **File | Save** menu item or the button on the **Standard** toolbar. This option is not available during the calculation running.

The calculation results are saved in two files. Both files have the same name as the RCF project file and are located in the source data folder.

The first file is binary and it has the ***.rcfresults** extension. The data from this file are used to display the calculation results in the **UM RCF Wheel** module and are loaded by the program automatically when opening the RCF project file.

The second file is a text file with the extension ***.csv**. This file allows the user to open the calculation results in **Excel** tables.

Note. The autosave of results after the completion of calculation for each *wheel wear iteration* is supported.

25.4.7. Deleting the results of calculation

The calculation results can be deleted by using the **Design Scheme | Delete results** menu item or the button on the **Design Scheme** toolbar. The option is available if the calculation was carried out.

Note. The ***.smx** files with saved components of stiffness matrices (if they exist) will not be deleted.

If it is necessary to delete the stiffness matrices files, the **Design Scheme | Delete stiffness matrix** menu item, or the button , or the **Ctrl+Del** hot key can be used (see Sect. 25.4.3.1).

25.4.8. Viewing calculation results

After completion of the calculation for the first *wheel wear iteration* the options are available allowing to navigate through the list of *wheel wear iterations* in two directions: from the first to the last and back.

These options can be used in the following ways:

– choose the **Design Scheme | Previous profile** or **Design Scheme | Next profile** menu items;

– click the button or at the **Design Scheme** toolbar;

– click the **PgDn** or **PgUp** key on the keyboard.

At that the calculation results for selected *wheel wear iteration* are displayed at the **Information Panel** and the workspace of window.

At the **Status** group box of the **Information Panel** the general information is displayed: the number of the wheelset, the wheel is left or right, the number of the profile is the number of *wheel wear iteration*, the total number and number of calculated contact problems for current *wheel wear iteration*, the percentage of overall progress of the calculation (Figure 25.50).

First of all, the value of biggest accumulated damage at the wheel point and the travelled distance are interest when analyzing the calculation results. These parameters can be seen at the **Calculation** group box of the **Information Panel** (Figure 25.50). If the calculation was carried out until reaching the limit of accumulated RCF damage and this limit has not been reached, the predicted traveled distance at the same group box shows how many kilometers the wheel could still run with the reached level of accumulated damage. The predicted traveled distance is determined by dividing the traveled distance to the biggest accumulated damage.

The wheel point with the biggest accumulated damage can be marked on the design scheme at the workspace of window with help the **Maximum damage** imaging mode (see Sect. 25.4.4.6). The depth at which this point is located under the rolling surface of the wheel can be determined using the **Mesh** imaging mode, starting from the fact that the side size of the finite element of the mesh is 1 mm (see Sect. 25.4.4.1). The **Color visualization of damage** mode (see Sect. 25.4.4.3) and **Isolines** mode (see Sect. 25.4.4.5) allow representing the distribution of accumulated RCF damage in the radial cross-section of the wheel.

File with ***.csv** extension (see Sect. 25.4.6) allows the user to open the calculation results in **Excel** tables and create graphs for preparing a report.

Using the results from the first data block, it is possible to plot the dependence of the accumulated damage at the dangerous point of the wheel material on the mileage. The example of such a graph is shown in Figure 25.51.

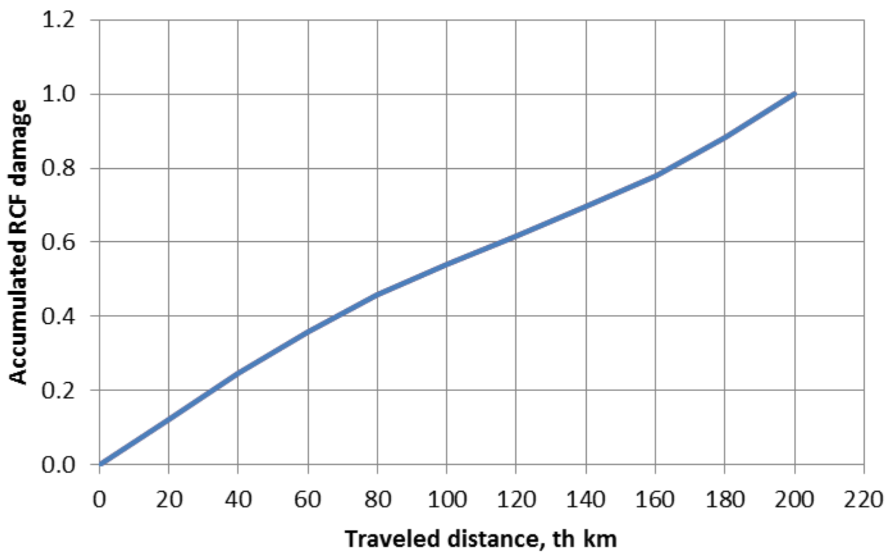


Figure 25.51. Dependence of accumulated damage at the dangerous point of the wheel material on the mileage

The data from the second block make it possible to plot the diagram of the accumulated RCF damage in the dangerous section of the wheel. By the dangerous section we will mean the circumferential section of the wheel containing the dangerous point, i.e. the point with the biggest accumulated RCF damage. To construct the diagram the values of the accumulated damage at the nodes of the FE mesh of the radial section of the wheel fragment belonging to the dangerous section are used. The example of such sequence of the nodes is shown in Figure 25.52a. The abscissa axis is the *x* axis in the coordinate system shown in Figure 25.38. The graph of the dependence of the accumulated damage in the nodes of the dangerous section of the wheel on their coordinate *x* is shown in Figure 25.52b.

The second block also contains the data allowing to plot the surface of accumulated damage in the radial section of the wheel fragment. The *x* coordinate of the node is plotted along the abscissa in accordance with the frame of reference shown in Figure 25.38. On the ordinate the depth of the node under the rolling surface is plotted. The value of the accumulated damage in the nodes of the FE mesh of the radial section of the wheel fragment is plotted along the applicate. The example of such surface is shown in Figure 25.53.

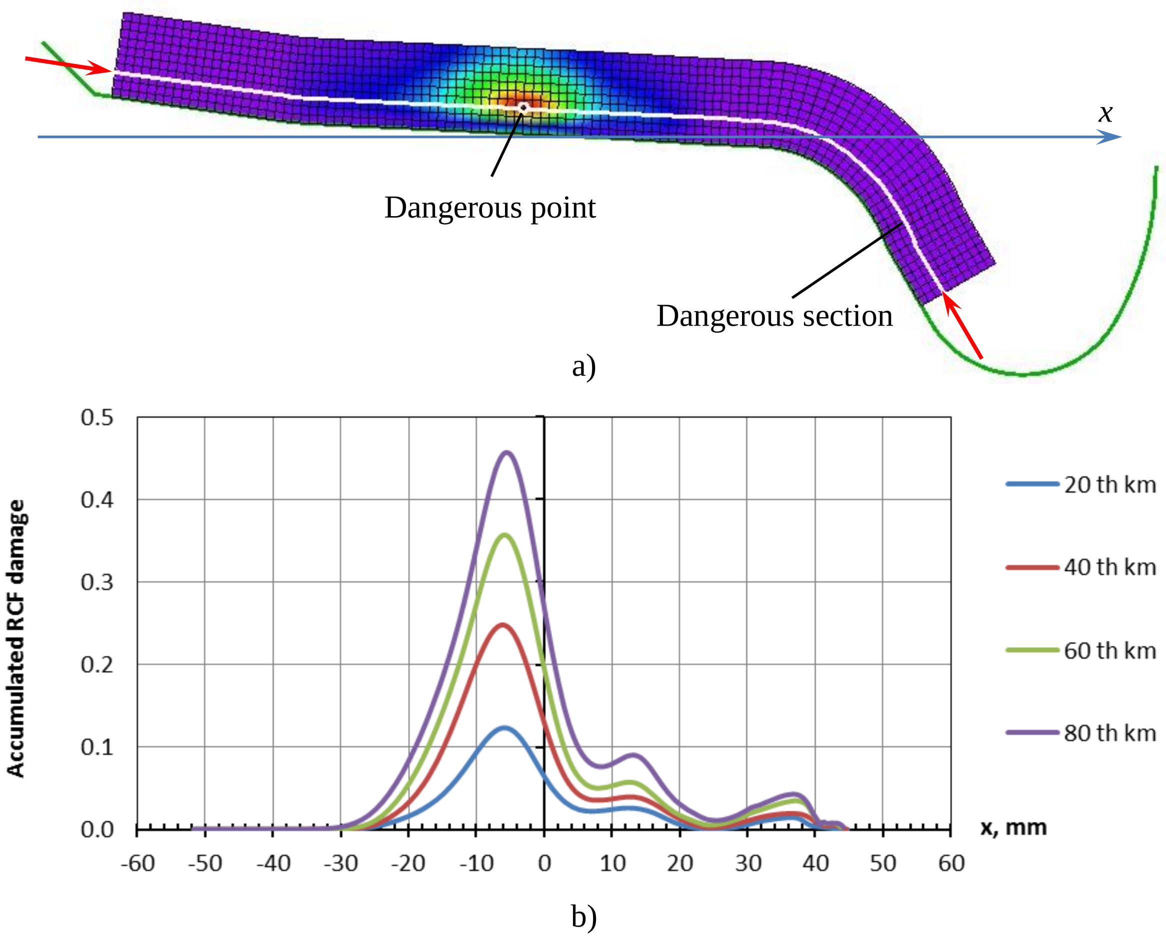


Figure 25.52. Diagram of accumulated RCF damage in dangerous section of wheel

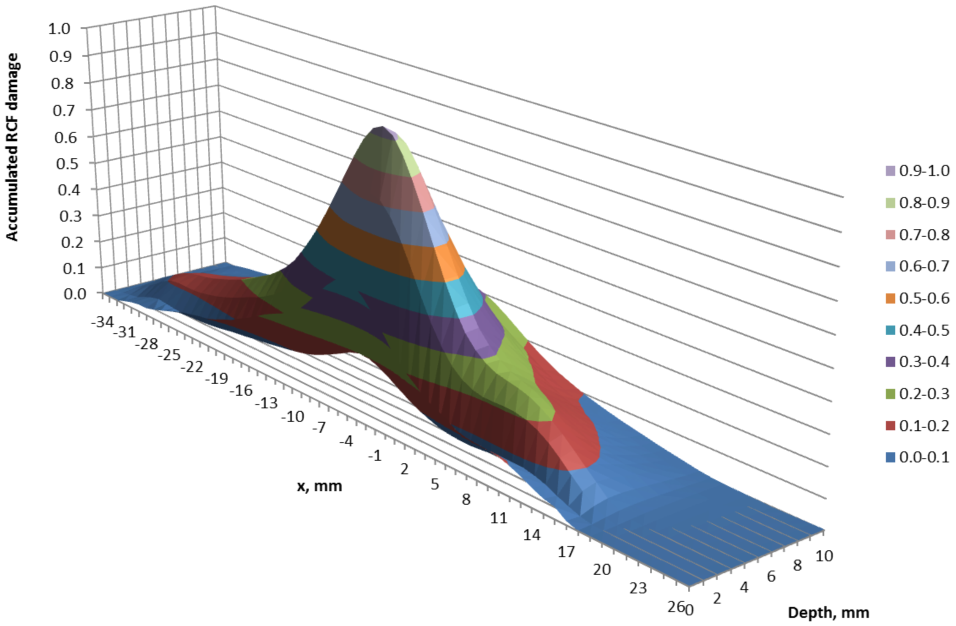


Figure 25.53. Accumulated damage surface in radial section of wheel fragment

25.4.9. Sample of simulation of accumulation of damage in the wheel

Let us consider the example of modelling of the process of accumulation of RCF damage in the wheel of the car with bogies of 18-100 model. The car model is supplied with the **UM** software and it is available in the library {UM Data}\samples\Rail_Vehicles\simple_18_100.

Load the model in **UM Simulation**. Open the **Object simulation inspector**.

At the **Solver | Simulation process parameters** tab set the following values:

- **Solver** = Park;
- **Error tolerance** = 1E-7;
- **Computation of Jacobian** = yes;
- **Jacobian for wheel/rail forces** = yes.

Go to the **Rail vehicle | Wear** tab and load the evolution project from *Wheel RCF Test.ecf* file. Set the required **Number of threads** and switch on the **Wear profile evolution** radio button (Figure 25.54). Run the simulation using the **Integration** button.

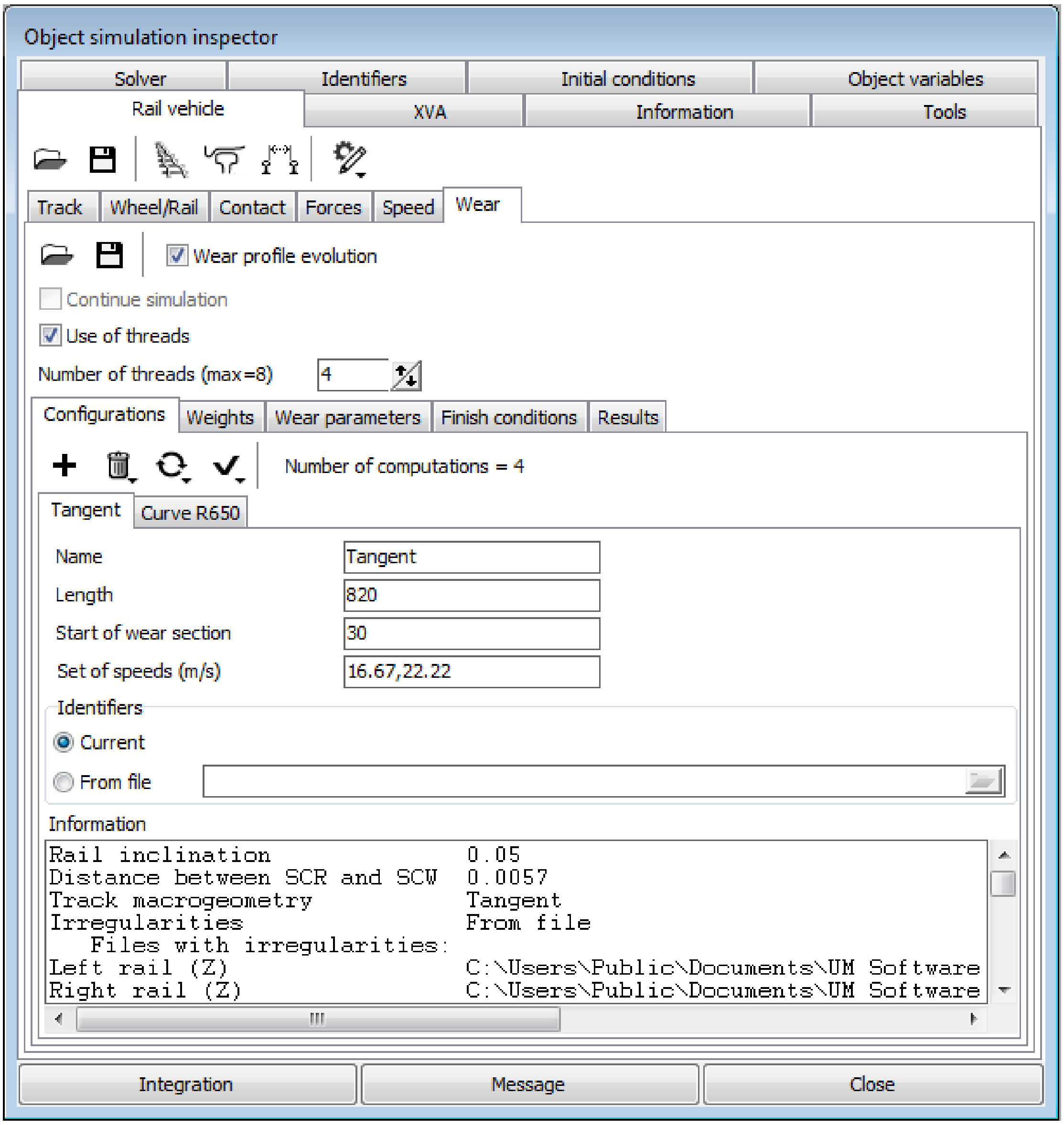


Figure 25.54. The settings of the project of wear profile evolution

According settings of the evolution project the simulation of moving the car is carried out along the tangent track section and the curve track section with radius of 650 m. On each track section the car moves at the speed of 16.67 and 22.22 m/s.

In result of the simulation the data for the five *wheel wear iterations* of the left wheel of the first wheelset were generated. The total travelled distance is 25 000 km. The travelled distance at each *wheel wear iteration* is 5 000 km. The RCF project file is located in the folder

{UM Data}\samples\Rail_Vehicles\simple_18_100\WearResultRCF\simple_18_100.rcf

Open the RCF project in **UM RCF Wheel** module in any of the ways specified in Sect. 25.4.1. Change the project settings at the **Calculation** and **Material** tabs as shown in Figure 25.55 and Figure 25.56. Keep the parameters at the **Residual stresses** tab unchanged. Save the project settings by click **OK** button at the **Parameters** dialog box. Run the simulation by the **Run** button (see Sect. 25.4.5).

Various possibilities of representation of the calculation results are described in Sect. 25.4.8. In Figure 25.57, Figure 25.58 and Figure 25.59 the results of simulation of the process of accumulation of the RCF damage in the material of the car wheel are shown.

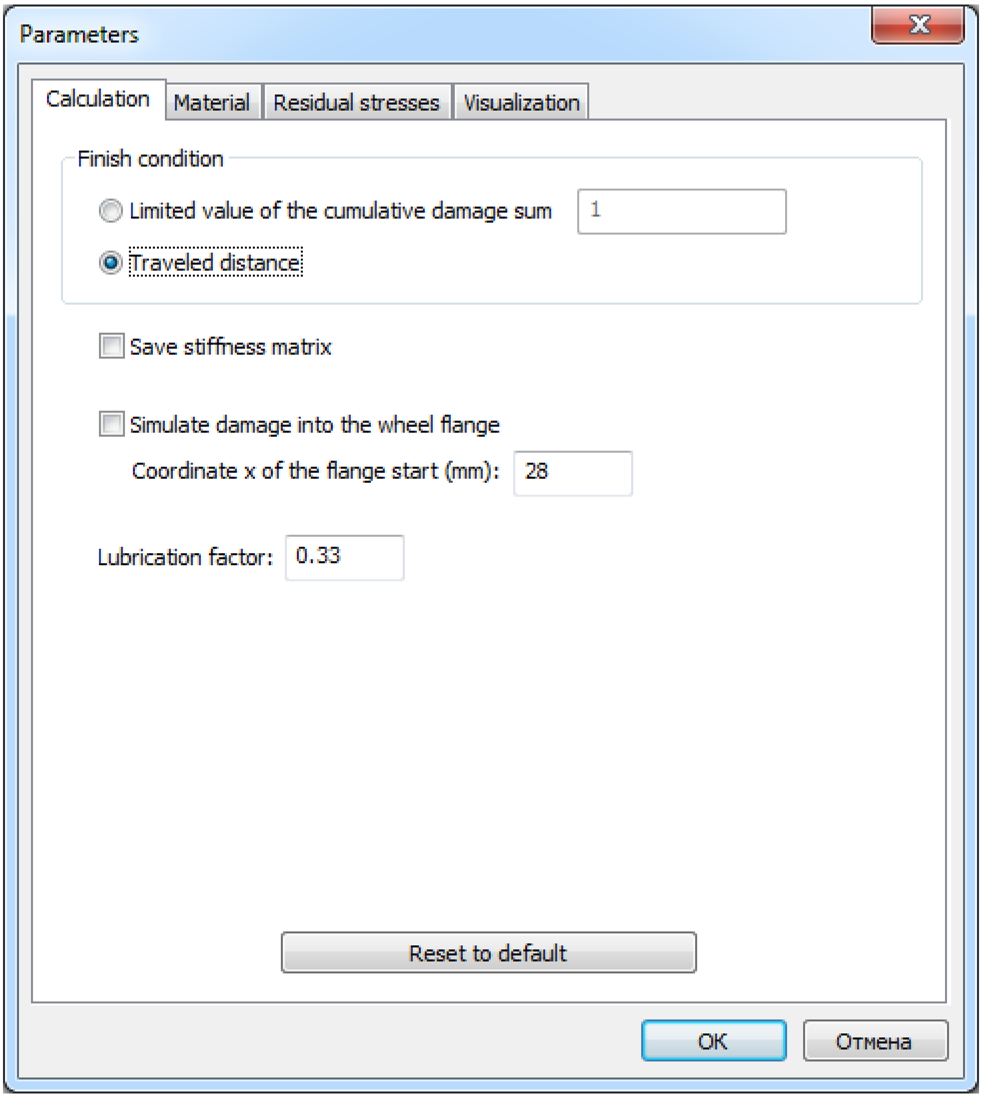


Figure 25.55. RCF project settings on the **Calculation** tab

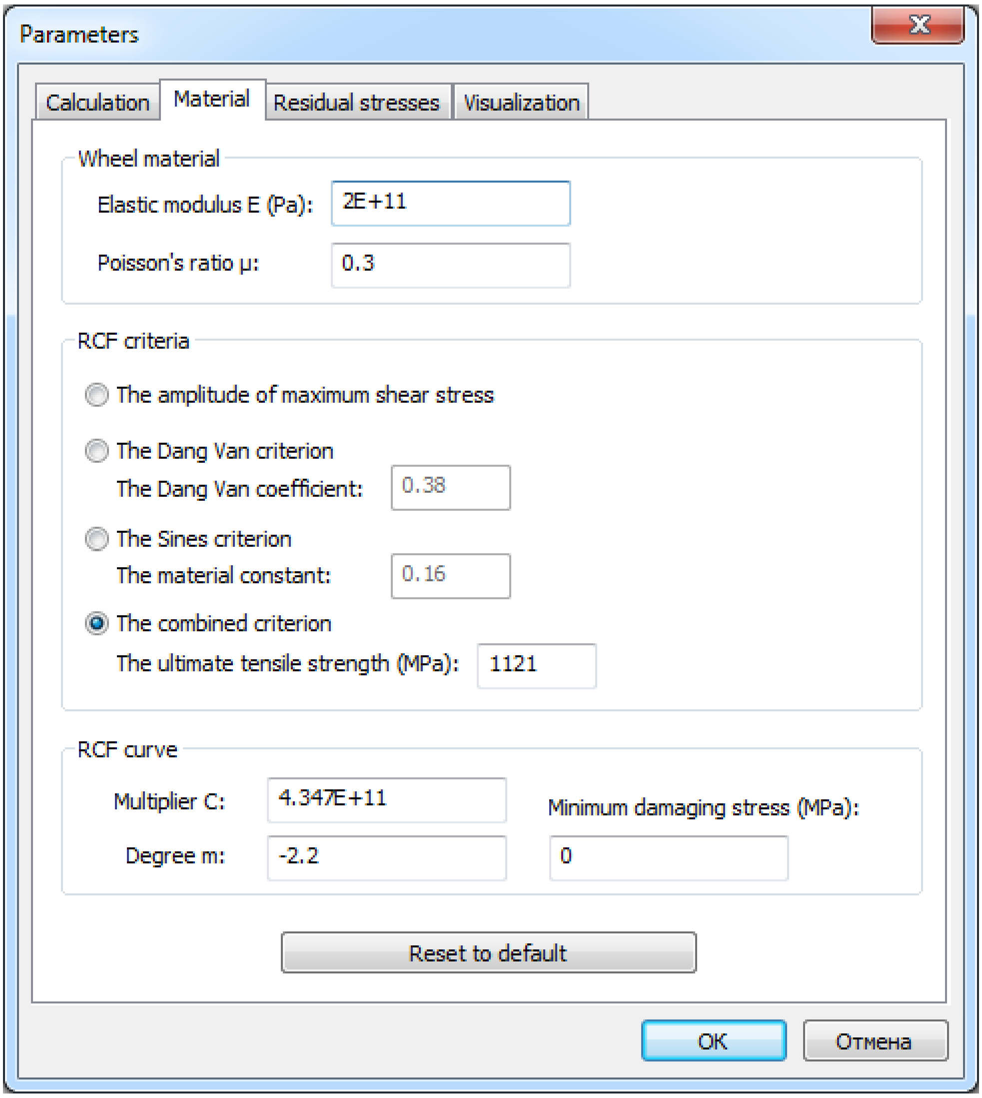


Figure 25.56. RCF project settings on the **Material** tab

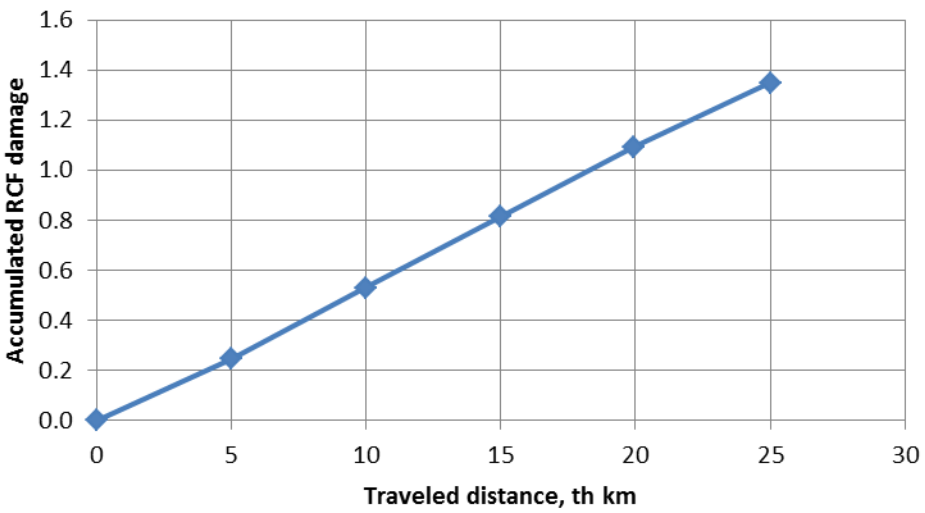


Figure 25.57. Accumulated RCF damage at the danger point of the wheel material

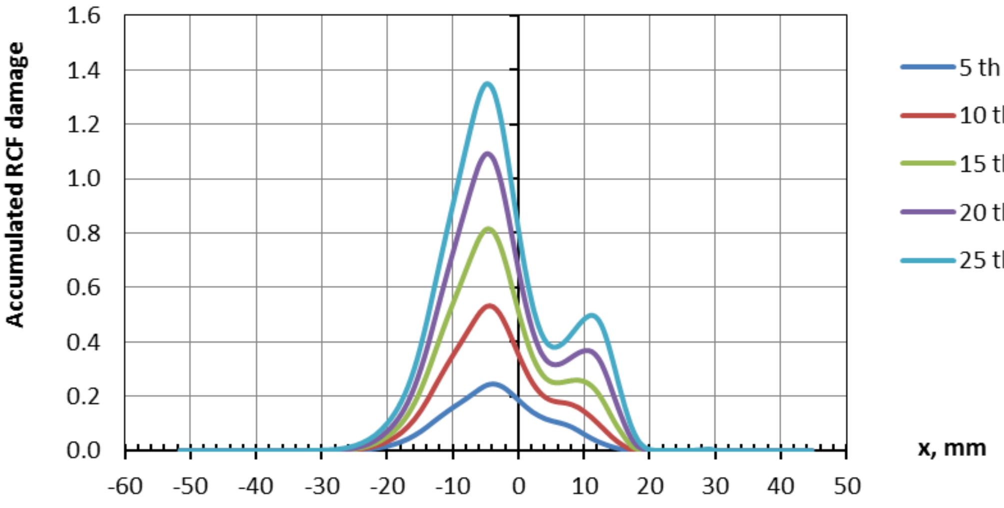


Figure 25.58. Diagram of accumulated RCF damage in the dangerous section of the wheel

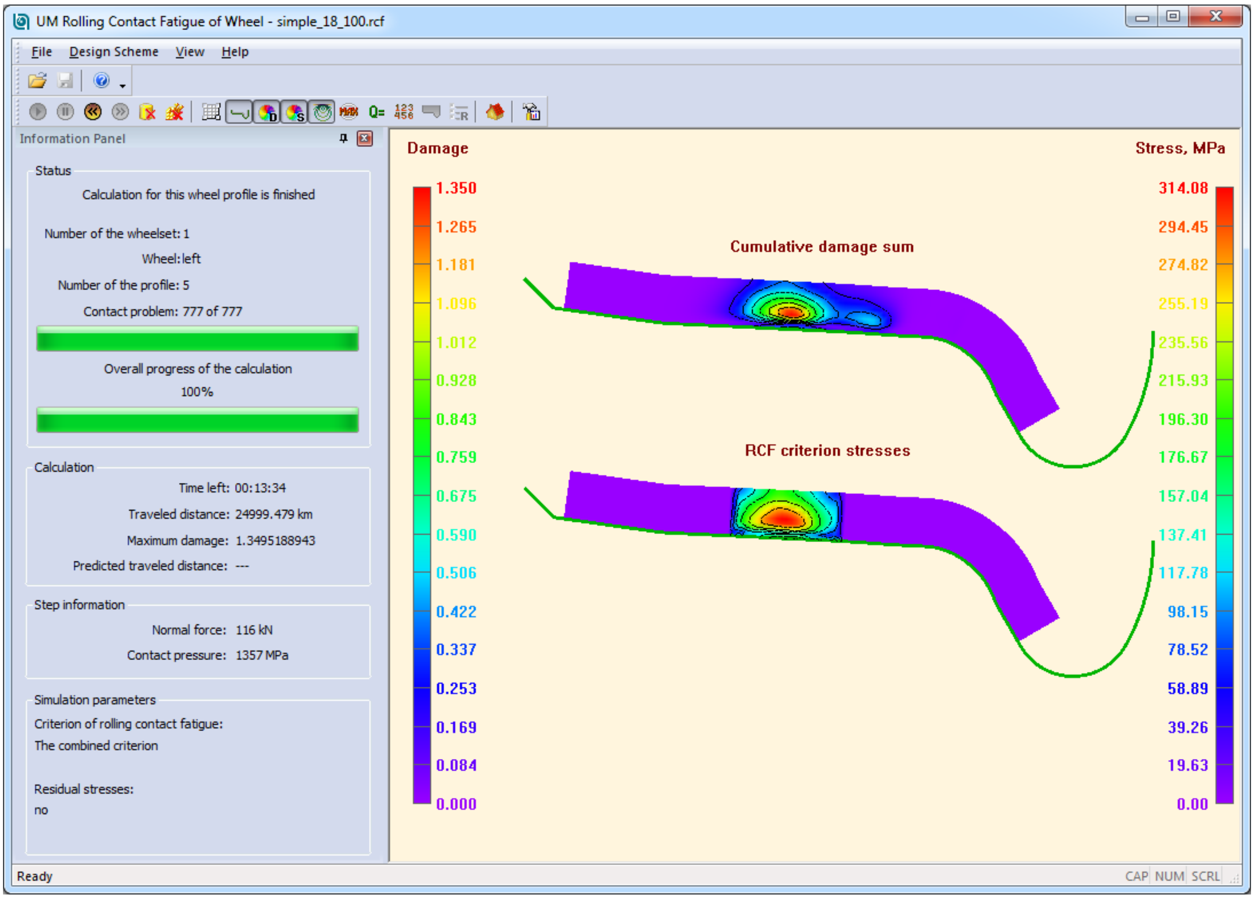


Figure 25.59. Results of simulation of the process of accumulation of RCF damage in the material of the freight car wheel

25.5. Simulation of accumulation of rolling contact fatigue damage using UM RCF Rail module

The formation of RCF damage in a certain section of the rail occurs as a result of cyclic loading and depends on the passed tonnage. The nature of loading at the same value of the passed tonnage can differ significantly depending on the types of railway vehicles passing through this section, their quantitative ratio, the degree of loading and the speed of movement. In addition, even vehicles of the same type may have different wheel diameters, profiles of wheel rolling surfaces and other parameters, which also affects the nature of the loading. Modelling the dynamics of the movement of railway vehicles in the **UM** allows the user to create the load blocks taking into account the above factors.

To simulate the process of accumulation of RCF damage in the rail material, the dynamics of the movement of railway vehicles is simulated either on a tangent track section or on a curve track section. The number of rail cross-sections is set, in which the contact forces from each wheel of the passing vehicle are calculated. The number of such sections should be sufficient to provide representative statistics about the forces and position of the wheel on the rail. In the **UM RCF Rail** module the accumulated fatigue damage in these sections are summed up and assigned to one cross-section, which will be referred to below as the *control section of the rail*.

When modelling the process of accumulation of RCF damage, it is impossible to indicate the point at which the accumulation of damage will proceed most intensively. Even if the contacting bodies do not move mutually, but the normal force is variable, for example, in the Hertz problem the dimensions of the semiaxes of the elliptical contact patch will change, and hence the positions of dangerous points, both on the contact surface and in the subcontact layer of a material.

Therefore, the calculations must be carried out for a set of points located in the area adjacent to the contact. When using the finite element method, the nodes of the FE model can be used as such points. For the correct solution of the contact problem, the dimensions of the design scheme of the bodies must be such that the condition of the smallness of the contact patch size will be satisfied. For the correct solution of the contact problem, the dimensions of the FE models of the bodies must be such that the condition of the smallness of the contact patch size will be satisfied. But with a large number of degrees of freedom of the system, it will take considerable time to solve the contact problem and determine the stresses at the nodes, and such calculations must be performed many times.

Based on the above considerations, a fragment of the FE model of the rail adjacent to the contact is used in the module. Its dimensions are taken such that the stresses on the surfaces by which it cut is out will be relatively small.

25.5.1. Input data

Necessary data for modelling of the accumulation of RCF damage in the **UM RCF Rail** module is generated during process of simulation of dynamics of a railway vehicle in the **UM Loco** module (Chapter 8) with help the **UM Loco/Rail Profile Wear Evolution** tool (Chapter 16). In Figure 25.60 the **Object simulation inspector** window with settings of the evolution project of the rail profile is shown.

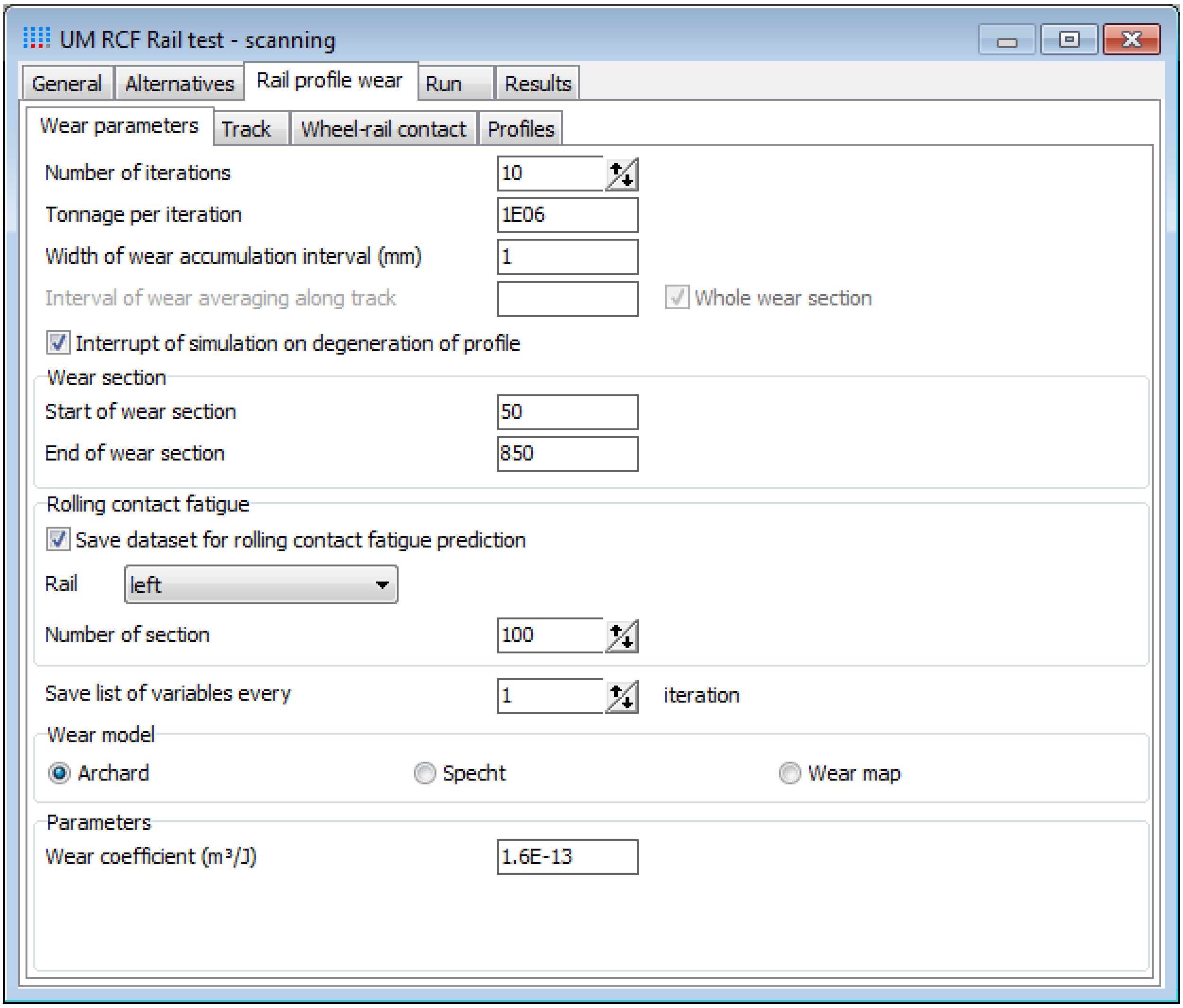


Figure 25.60. Settings of the evolution project of the rail profile in **UM Simulation**

It is necessary to enable the **Save dataset for rolling contact fatigue prediction** check box in the **Rolling contact fatigue** parameter group. In the drop-down list of the **Rail** item the user can select the rail for which the process of accumulation of RCF damage needs to be simulated: left, right or left and right. The item **Number of section** specifies the number of rail cross sections, in which the contact forces from each wheel of the passing vehicle will be calculated.

As a result, after the simulation of the dynamics of the railway vehicles is completed, a binary ***.rld** file will be created in the **RailWear** folder located in the rail profile wear evolution project. The same folder contains files with wear simulation results and evolution project settings. The file with input data ***.rld** includes in the title the name of the rail profile file and its identification – left "l" or right "r".

The initial data file ***.rld** stores the rail profiles obtained as a result of wear. The number of profiles depends on the number of iterations specified in the evolution project. The following information is stored for each profile number:

– the Young’s modulus and Poisson's ratio of material of the rail;

– the number of the points of the rail profile and their coordinates;

– the tonnage, number of numerical experiments, statistical weight of the experiment, mass passing through the calculated section;

– the number of loading cycles;

– the data about contact for each loading cycle – the coordinates of nodes of the mesh on the surface of contact patch and values of the forces in the nodes of mesh.

Note. There must be enough free space on the hard disk. Each ***.rld** file can be up to several gigabytes depending on the evolution project settings.

Figure 25.61 shows the frame of reference in which the coordinates of the points of the rail profile are set.

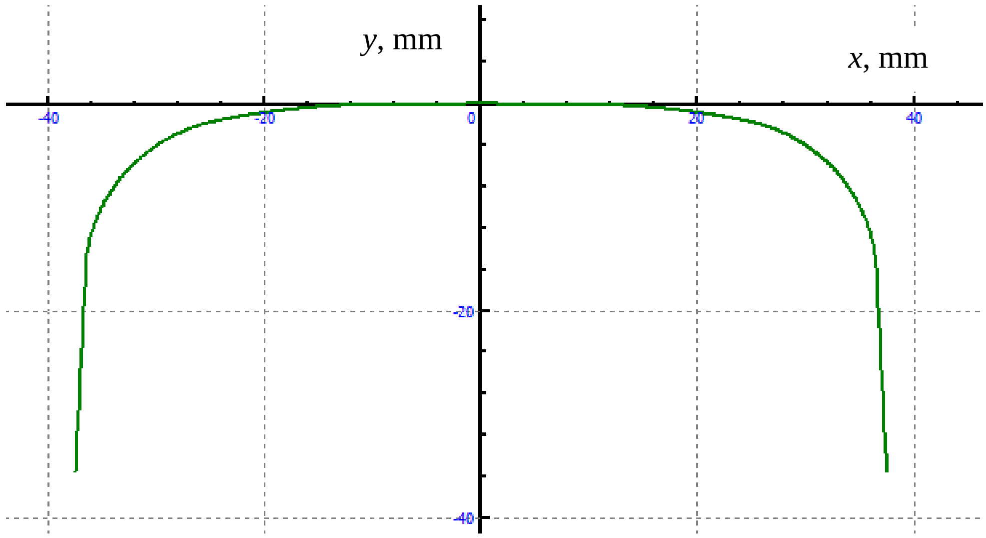


Figure 25.61. Reference system for specifying the coordinates of the rail profile points

25.5.2. Construction of finite element model of rail

To solve the contact problem of the wheel and rail the dimensions of the finite elements are chosen so as to ensure a minimum computation time with a sufficient precision of solution. The profiles of the rolling surfaces of the wheel and rail from the input data file are specified by the coordinates of points located along the contours with step of 0.1 mm. For analysis of the stress state in the contact region it turned out to be appropriate to apply the meshes with the size of the element of 1 mm for three-dimensional (3D) FE models. Nevertheless, when using the fragments that include regions adjacent to the contact surfaces of such sizes at which the stresses are small on the surfaces of extraction of the fragment, the FE model contains a big number of the degrees of freedom. In this connection, the FE model containing 10 layers of finite elements counted from the contact surface is used in the calculations.

Construction of the FE model of a rail fragment starts with the creation of a two-dimensional (2D) flat mesh composed of the quadrangular finite elements. It is created on the base of nodes placed on the wheel profile with step of 1 mm (Figure 25.62). Its dimension along the rail profile is 87 mm. This dimension covers the profile section on which the contact has place at any possible positions of the wheel on the rail. Then 3D FE model is generated. 2D mesh is dragged along the longitudinal axis of the rail (Figure 25.63).Dimension of the FE model at the longitudinal direction is 50 mm. This dimension is chosen with taking into consideration that contact patch length does not exceed 50 mm in any case.

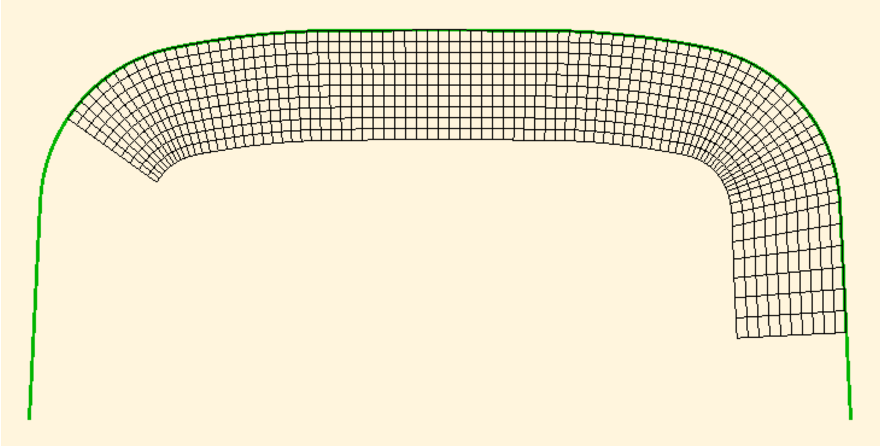


Figure 25.62. 2D mesh for construction of 3D FE model of rail fragment

The thickness of the fragment of 10 mm ensures sufficient accuracy of the solution when using an elastic foundation. The most probable dimension of the semiaxis *a* of the contact patch does not exceed 10 mm and the point with the biggest shear stress is placed on the depth about 0.5*a*. It is possible to cover the region of the biggest stresses at the selected size of the fragment. Nevertheless, the resulting scheme is characterized by a large number of degrees of freedom. For example, the FE model presented in Figure 25.63 contains 48 807 nodes and has 146 421 degrees of freedom. Therefore, in order to decrease the computation time a fragment of this FE model will be used in calculations (see Sect. 25.5.3).

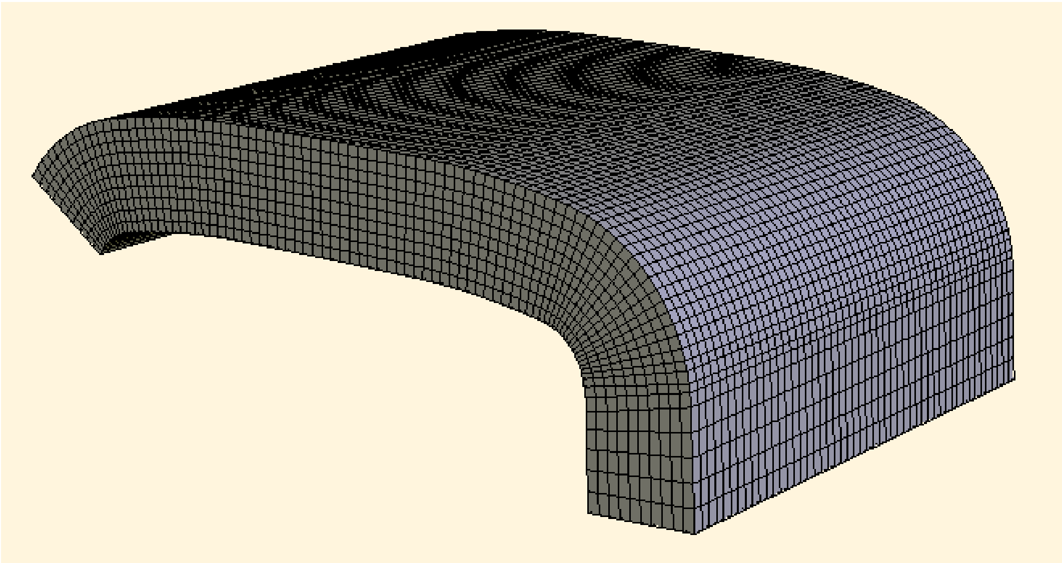


Figure 25.63. 3D FE model of rail fragment

The stiffness matrix of the obtained 3D FE model of the rail consisting of eight-node finite elements is formed. At this step, the preparatory stage is completed, and further calculations for loading cycles are started.

25.5.3. Calculation of stresses in the region of wheel and rail contact

For solution of the rolling problem in the **UM Loco** module the model of contact forces of W. Kik and I. Piotrowski [31] and the Kalker CONTACT model [32] are used (Chapter 8, Sect. 8.4.2.5.2. "Parameters of inertial rail contact"). The **UM Loco/Multi-point Contact Model** tool is required for the W. Kik and I. Piotrovski model and the **UM Loco/CONTACT add-on interface** tool for applying the CONTACT program.

Normal and tangential traction are calculated at the nodes of the 2D mesh located on the contact surface. The mesh is created as follows: the contact patch is divided into strips of the same width oriented along the rolling direction, and then each strip is divided into the same number of elements having the same dimensions within the strip (Chapter 8, Sect. 8.3.1.2.2.3. "FASTSIM"). The contact pressure and the forces acting on the plane of each element are determined. The calculated forces are attributed to the center of gravity of the element. The number of the strips and elements is set by the user. Since during the rolling of the wheel along the rail the contact patches change their shape and size, the step of the mesh in two directions also changes for each new contact.

For determination of the stresses in the nodes of the FE model of the wheel fragment it is necessary to apply the forces obtained as a result of solving the contact problem to the nodes located on the contact surface. However, the nodes of the mesh created during the solution of the problem by fast algorithms do not coincide with the nodes of the FE mesh. Therefore, after loading the initial data the forces in the nodes of the FE mesh located on the contact surface are calculated using a linear interpolation. At that the position of the meshes relative to each other is uniquely determined by the coordinates of the rail profile points, on the basis of which these meshes are created.

The procedure for determining the stresses at the nodes of the FE model of the rail for each loading cycle begins with the separating a fragment (Figure 25.64a) from the initial FE model (see Sect. 25.5.2). The dimensions of the mesh of the fragment adjacent to the contact surface are chosen so that they are 3 mm bigger than the contact patch in each direction. Separated fragment has a thickness as the initial 3D FE model.

The application of the FE fragment on the elastic foundation [33] as the design scheme allows to determine the stresses and the strains in 3D region adjacent to the contact surface without any type of simplified assumptions like these: a contact patch has shape of circle or ellipse, to selection of the critical plane, to assumption that it is in a state of plane deformation. Use of such fragment allows decreasing the computation time essentially [33], [34], [35], [36].

During construction of the initial FE model the components of its matrix of stiffness are calculated. To the nodes located on all surfaces of the separated fragment except the contact surface the elastic constraints are applied (Figure 25.64b). The stiffness of the constraints is assumed to be 10^7 N/m in accordance with the recommendations given in work [36].

Components of the stiffness matrix of the finite elements ensemble adjacent to an internal node of the fragment are taken from the stiffness matrix formed for the initial FE model. For nodes located on the surfaces of separated fragment the components are halved, for nodes located on the edges they are divided into four.

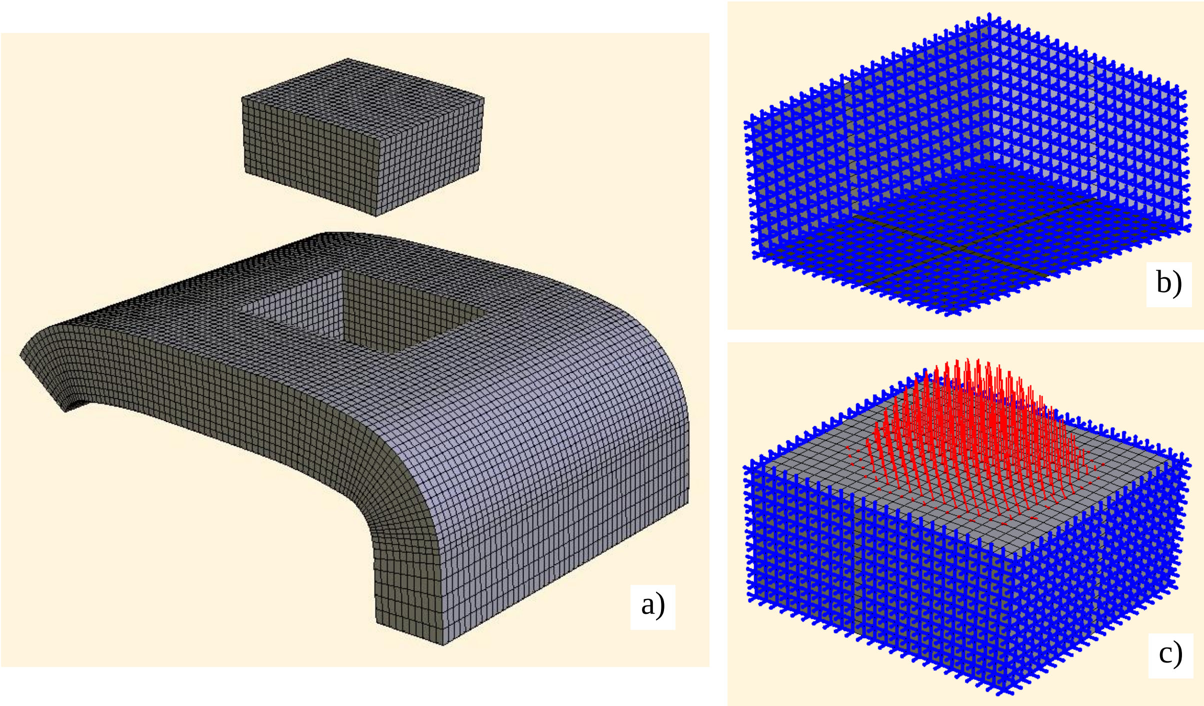


Figure 25.64. FE fragment on elastic foundation:
a) separation of small fragment from initial FE model;
b) applying of elastic constraints on surfaces of separated fragment;
c) applying of forces to nodes of contact surface of separated fragment

Nodal contact forces are applied to the nodes located on the contact surface of the separated fragment (Figure 25.64c). Displacements of the nodes are determined with the method of node iterations [37].

The normal and shear stresses are calculated for all nodes of the separated FE fragment. The number of nodes of the fragment in the rolling direction is considered as the path of the *i*-th node of the cross-section of the rail fragment (Figure 25.5). Then each node from this chain is the *j*-th position occupied by the *i*-th node as it passes through the region adjacent to the contact. The equivalent stress per cycle is calculated in accordance with the chosen RCF criterion (Sect. 25.2.5). The obtained values of the criterion are assigned to the nodes located in the plane of symmetry of the FE model of the rail fragment. During simulation of the process of accumulation of RCF damage they are compared with damaging values from given RCF curve.

The areas of equal values of the combined criterion σ_{RCF} in the color and the isolines of the values of the criterion are shown in Figure 25.65. The biggest $\sigma_{RCF} = 257$ MPa is obtained for the node located at the depth of 4 mm below the contact surface.

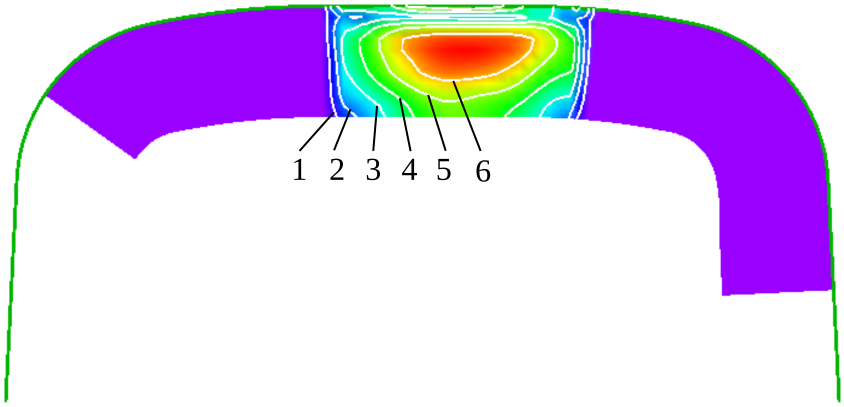


Figure 25.65. Areas of equal values of the combined criterion σ_{RCF} in the cross-sectionof the rail contained in the plane of symmetry of contact;
1-6 are numbers of isolines σ_{RCF}, the value of isoline is 36.71 MPa

25.5.4. Accumulation of damage in the nodes of finite element model of rail

In the process of modelling the wear of the rail profile the scaling is used, the purpose of which is to obtain significant wear with a small total weight of the vehicles passing along a given section of the track. In fact, the wear diagram is multiplied by a scale factor. A similar approach is applied to the process of modelling the accumulation of RCF damage in the rail material. Dependence (1.2) for calculating the RCF damage Q accumulated in the *n*-th node is transformed to the form

$$Q = \sum_{k=1}^{N_{it}} \sum_{i=1}^{N_{exp}} \sum_{j=1}^{N_i} \frac{M_{it}}{M_{sect} N_i} \alpha_i \frac{1}{N_n(j)}, \tag{25.46}$$

where N_{it} is the number of iterations of rail profile wear simulation;

N_{exp} is the number of numerical experiments;

N_i is the number of loading cycles for the *i*-th numerical experiment;

M_{it} is the tonnage assigned to one wear iteration;

M_{sect} is the mass passing through the calculated section;

α_i is the weight of the *i*-th numerical experiment;

$N_n(j)$ is the number of cycles until the appearance of fatigue failure, when the criterion value is equal to σ_{eq}^n.

The listed above parameters are set in the evolution project. For details, see Chapter 16 of the user manual.

The number of cycles until arising fatigue defects for the nodes located in the plane of symmetry of the FE fragment is calculated using the Eq. (25.1). Then the accumulated damage in these nodes is determined by summing the damage in accordance with dependence (25.46).

In the program settings the user can choose two variants for stopping the simulation of the damage accumulation.

1) Select the "Tonnage" option, i.e. the damages will accumulate without limits until calculations will be performed for all wear iterations specified in the evolution project.

2) Set the limiting value of the accumulated damage Q.

User can also set the minimum value of the damaging stress in accordance with the selected RCF criterion. During simulation of the accumulation of RCF damage the cycle is considered as damaging if the value of the criterion exceeds this limit. Its default minimum value is zero.

25.5.5. Consideration of presence of lubricant in wheel and rail contact

It has already been noted in Sect. 25.2.4 that during testing the specimens of the wheel steel under rolling contact fatigue without presence of a lubricant in the contact the damage is not observed even at very high pressures. This is due to the mechanism of generation of the cracks [16]. As a rule, the RCF tests of the wheel steel specimens in laboratory conditions give the understated number of cycles until the appearance of fatigue defects, since they are carried out with continuously feeding the lubricant in the form of oil or water on the contact surfaces. Such conditions are harder than the real service conditions of the wheels. Railway transport is widely used for the transportation of passengers and goods on mainline railways, tram lines and metro lines, in industrial zones, and in the mining industry. In outdoor conditions rainwater and condensate are the natural lubricants in the wheel-rail contact. Obviously, depending on weather conditions such lubricant is not always present in the contact. Therefore, statistics show that the RCF durability of the rails is higher than that of the laboratory specimens.

In order to obtain the results that are closer to the statistical operational data, the so-called "lubrication coefficient" K_{lub} is introduced in the **UM RCF Rail** module. This coefficient is set by the user. It is dimensionless and ranges from 0 to 1. The coefficient is calculated for the specific operating conditions of the railway track. For example, if a railway track is operated in the open air in a specific geographic region, then it can be defined as the ratio of the number of rainy days per year, typical for this region, to the number of all days in the year. In other operating conditions the ratio of the length of the track section with the presence of lubricant to the length of the entire track can be taken.

With the introduction of the coefficient Klub, when modelling the accumulation of RCF damage, a modified form of the dependence (25.46) is used in the calculations

$$Q = K_{lub} \sum_{k=1}^{N_{it}} \sum_{i=1}^{N_{exp}} \sum_{j=1}^{N_i} \frac{M_{it}}{M_{sect} N_i} \alpha_i \frac{1}{N_n(j)}. \tag{25.47}$$

The default lubrication factor K_{lub} is 0.33.

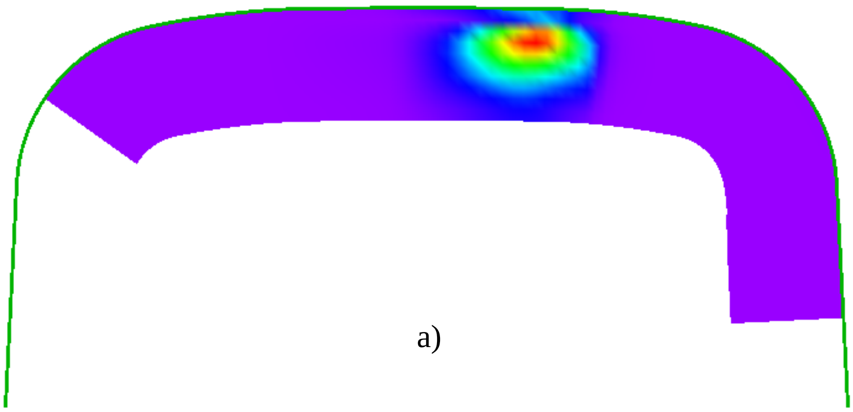
25.5.6. Consideration of rail profile wear

Since the processes of wear and accumulation of RCF damage are inextricably linked, the data needed for modelling of the accumulation of RCF damage in the **UM RCF Rail** module is created using the **UM Loco / Rail Profile Wear Evolution** tool (Chapter 16).

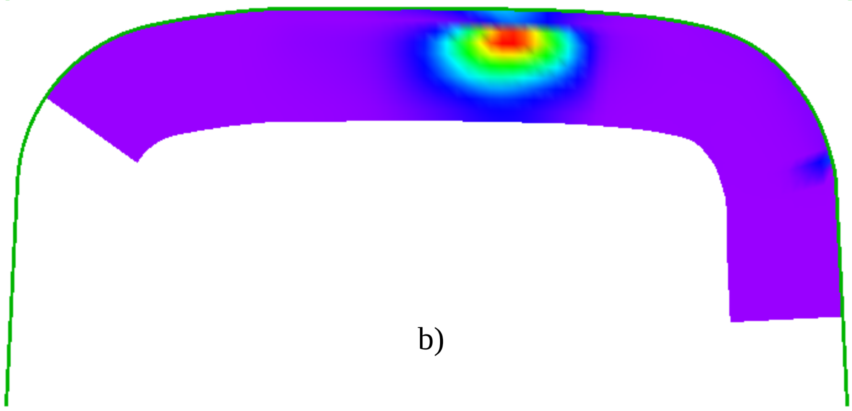
Let us introduce the concept of *rail wear iteration*. Within the framework of this section, the *rail wear iteration* is a series of calculations within which the profile of the rolling surface of the rail does not change.

For accumulation of RCF damage in the **UM RCF Rail** module the FE model of the rail fragment is used (Sect. 25.5.2). At the each *rail wear iteration* this model is reconstructed. A new FE mesh is created based on the worn rail profile. The damages calculated by means interpolation of the damages accumulated in the nodes of the mesh at the previous rai*l wear iteration* are assigned to the obtained nodes of the new mesh. Then the procedure of accumulation of RCF damage in the nodes of the new FE mesh of the rail is repeated.

The Figure 25.66 shows the distribution of accumulated damage in the *control section of the rail* R65 [40] on the tangent track section after the passed tonnage of 10 and 100 million tons.



a)



b)

Figure 25.66. Isochromatic lines of accumulated damage in the *control section of the rail* with R65 profile [40] after the passed tonnage: a – 10 Mt; b – 100 Mt

The number of *rail wear iterations*, total tonnage and tonnage at one *rail wear iteration* are determined by a set of parameters set in the rail profile evolution project.

The required parameter group is located on the **Rail profile wear | Wear parameters** tab in the scan project window (Figure 25.60).

- **Number of iterations** = 100;
- **Tonnage per iteration** = 1E06.

For example, for the given values in accordance with the formula (16.19) in Sect. 16.2.1. "Rail profile wear simulation" of Chapter 16 the total tonnage is

$$M_{tot} = N_{it} M_{it} = 100 \cdot 1E06 = 100 \text{ Mt}$$

The number of contact problems *N* that must be solved at one *rail wear iteration* when modelling the process of accumulation of RCF damage is determined by the dependence

$$N = \sum_{i=1}^{N_{exp}} N_{sect} N_w,$$

where N_{exp} is the number of numerical experiments;

N_{sect} is the number of cross-sections of the rail in which the contact forces from each wheel of the passing vehicle are calculated;

N_w is the number of wheelsets of vehicles included in the numerical experiment.

The number of calculated sections N_{sect} is set by the **Number of section** parameter on the **Rail profile wear | Wear parameters** tab in the scan project window. An increase in the value of this parameter makes the set of statistical data on wheel-rail contacts obtained as a result of modelling the movement of vehicles more representative, but increases the computation time spent on modelling the process of accumulation of RCF damage.

The required number of *rail wear iterations* and the passed tonnage per iteration are determined by testing a specific model. A big value of the **Tonnage per iteration** parameter can lead to a violation of the smoothness of the rail profile on the local section (Figure 25.67) or to the degeneration of the profile (see Sect. 16.1.1.3. "The wear simulation parameters. The profile-updating procedure" of Chapter 16). Therefore, in order to maintain the smoothness of the rail profile during the evolution process, when modelling wear, it is recommended to increase the number of iterations and decrease the passed tonnage per iteration. As indicative value of the **Tonnage per iteration** parameter 1 million tons can be recommended.

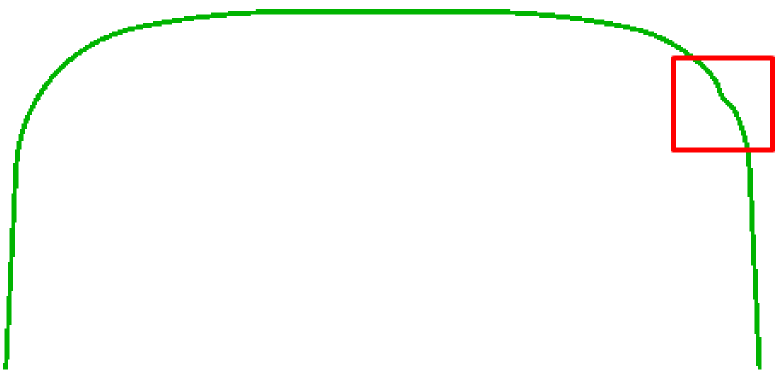


Figure 25.67. Violation of the smoothness of the rail profile in the local section

25.6. Working with UM RCF Rail module

25.6.1. Running UM RCF Rail module

An input data file for modelling of the accumulation of RCF damage in the rail is created in **UM Loco** (Chapter 8) module with help **UM Loco/Rail Profile Wear Evolution** tool (Chapter 16), see Sect. 25.5.1.

During or after installation **UM** software on computer it is recommended to associate the files with ***.rld** extension with the files of the **UM RCF Rail** module. In this case the **UM RCF Rail** module will be automatically run by double click on such files.

Another way to run the module is using the **Start** button and then **All programs | Universal Mechanism 9.0 | Tools | UM Rolling Contact Fatigue of Rail** menu item. After running **UM RCF Rail** module user needs to open the ***.rld** file with help **File | Open** menu item or  button on the **Standard** toolbar.

25.6.2. Interface of module

Interface of the **UM RCF Rail** module is presented in Figure 25.68.

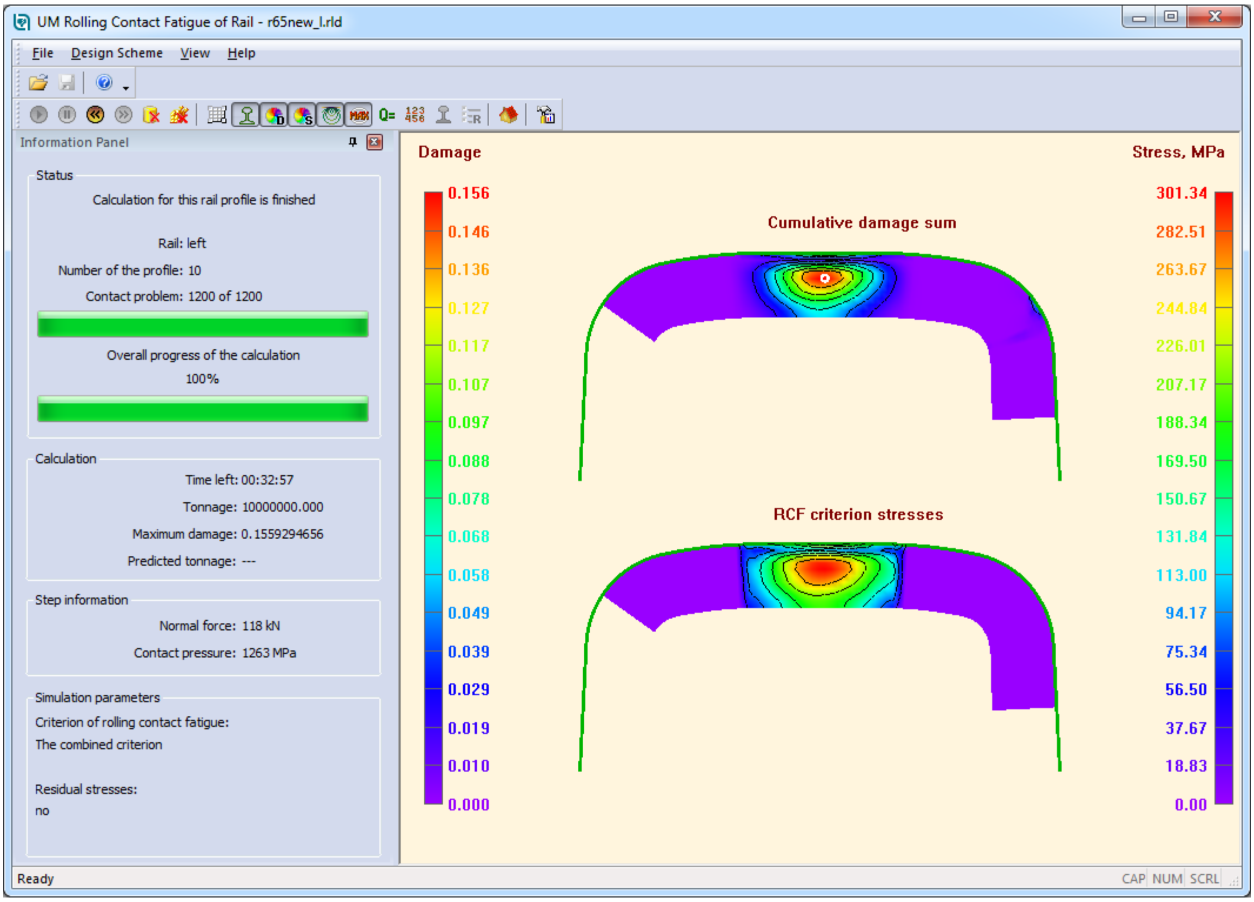


Figure 25.68. Interface of **UM RCF Rail** module

In the top of the window the title bar, menu bar, **Standard** toolbar and **Design Scheme** toolbar are located. In the bottom the status bar is located. In the left side of the window the floating **Information Panel** is placed. The user can dock it to the left or right side of the window or hide it. By default, the **Information Panel** is attached to the left side of the window and is always visible whenever the module is run.

The user can show or hide the toolbars and **Information Panel** and change its settings by the **View | Toolbars and Docking Windows** menu or using the context menu by clicking the right mouse button. The user can also change the interface of the module window using the **View | Application Look** menu.

The workspace of window is intended for the graphical displaying of design scheme and results of calculations. If the data for calculation is loaded, in the center of the workspace always is displayed the rail profile and FE mesh. All display modes can be switched on or off, for example, using the buttons on the **Design Scheme** toolbar. If during calculations the coloring modes are switched on, the equivalent stresses in the plane of symmetry of FE fragment of the rail are shown in the center of the workspace. The cumulative RCF damage in the same plane is shown on top. The color scale for estimate of the level of accumulated RCF damage is on the left and the color scale for the estimate of the equivalent stresses is on the right.

25.6.3. Parameters

The **Parameters** option becomes available when the data for calculation is loaded. In Figure 25.69 the dialog box is shown, which appears on the screen with help the **Design Scheme | Parameters...** menu item or button .

The appeared dialog box has four tabs: **Calculation, Material**, **Residual stresses** and **Visualization**. The **Reset to default** button allows returning to the initial values of the parameters at the selected tab. The changed parameters are saved only after clicking **OK**. Otherwise, the changes will not be saved.

25.6.3.1. The «Calculation» tab

The **Calculation** tab is shown in Figure 25.69.

Modelling of the process of accumulation of RCF damage in the **UM RCF Rail** module provides two variants of the finish of calculation:

1) passing through the *control section of the rail* of all vehicle configurations specified for the simulation of the dynamics of movement in the **UM Loco** module (passed tonnage);

2) reaching of limit of the accumulated damage sum in the rail point.

If the first option is selected, the calculation will be performed for all *rail wear iterations*, regardless of the accumulated damage at the rail point.

In the second case user has to set the limit value of accumulated damage sum. The accumulation of RCF damage at the point of rail is calculated by using the Eq. (25.47), the accumulation of RCF damage in the nodes of FE scheme is reviewed in Sect. 25.5.4. After reaching of limit of the cumulative damage sum the calculation will be continued until the finish of the current *rail wear iteration* (see Sect. 25.5.6).

By default, the first option is selected – the **Tonnage** item.

The calculation on each new *rail wear iteration* begins with the preparing the stiffness matrix of FE fragment of the rail (see Sect. 25.5.2, 25.5.6). This operation can take the several minutes depending on computer speed. In this connection, the total computation time significantly increases, particularly if the calculation is carried out for tens or hundreds of *rail wear iterations*. In any case, it fails to avoid these time-consuming operations at the first calculation. But this possibility appears if necessary to repeat the calculation for the same model with other conditions.

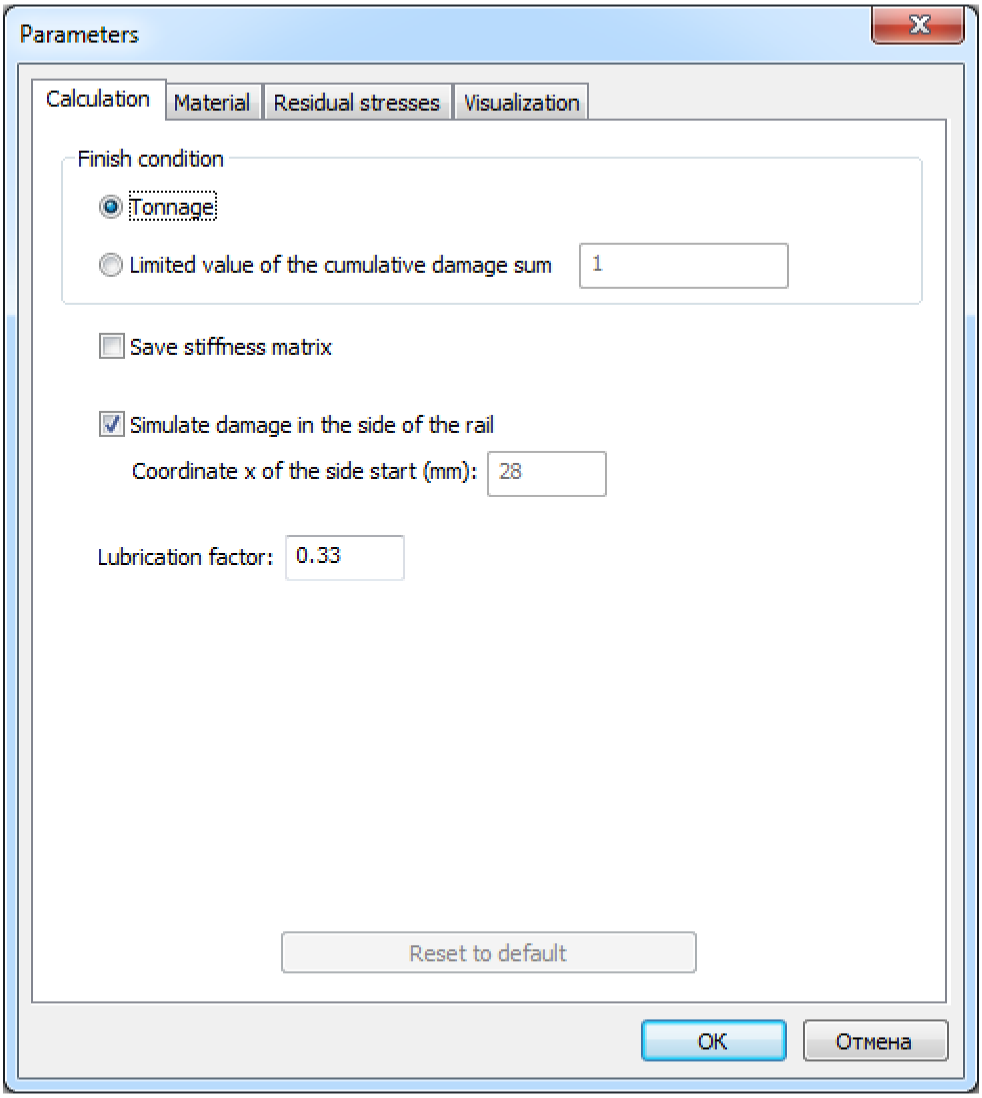


Figure 25.69. The **Parameters** dialog box, the **Calculation** tab

The checked option **Save stiffness matrix** means that the stiffness matrix for each new *rail wear iteration* will be saved in the ***.smx** file. This file has a binary format and it is located in the same folder as the file with the initial data. Size of this file is greater than 100 MB. So it is needed to have enough free space in the partition of the hard drive where the files are stored. Now during the calculation the components of the stiffness matrix will be loaded from the file, rather than calculated, that minimizes the computation time.

However, use of this option carefully.

Following variables are used for calculating the components of the stiffness matrix:

- the number of nodes of the FE model;
- the coordinates of nodes of the FE model;

- the Young’s modulus *E*;
- the Poisson's ratio *μ*.

If the coordinates of points of the profile, the dimensions of FE model, the Young's modulus or Poisson's ratio were changed, before the repeated calculation the stiffness matrix must be formed anew. Accordingly, the previous matrix must be destroyed. The algorithm realized in the **UM RCF Rail** module works as follows: first, an attempt to open the ***.smx** file corresponding to the initial data files is made – identification is done by the file names. If such file exists, the components of the matrix are loaded from it, else the components of the stiffness matrix are calculated.

Note.	If initial data was changed, it is necessary to delete the previous stiffness matrices before starting the new calculation. Otherwise it will lead to incorrect results.

The module reacts on changes of model parameters, but cannot react on all the possible actions of user. For example, if changes to the files of initial data are made outside of the **UM RCF Rail** module or **UM** software. For this reason, the **Save stiffness matrix** option is inactive by default.

If there is no need to save computer time, and there is no certainty that the files with the old stiffness matrices do not exist, it is better to delete the files with the old stiffness matrices before recalculation. This can be done using the **Design Scheme | Delete stiffness matrix** menu item or by  button.

The **Save stiffness matrix** option can be useful in two cases.

1) The calculation has been interrupted, the module is closed. After restarting the module and resuming the calculation is necessary to form the stiffness matrix for current *rail wear iteration*. If before that it was saved in the file, then the components of the matrix will be quickly loaded from the file.

2) The calculation was carried out with the specific conditions. User decides to repeat the same calculation by changing the one or several parameters. For example, the RCF criterion is changed. In this case, it is needed to delete the result files and return to the start of the calculation (see Sect. 25.6.7). At that the files of stiffness matrices are not deleted. Repeated calculation will carry out faster.

The **Simulate damage in the side of the rail** item (Figure 25.69).

The **UM RCF Rail** module makes it possible not to simulate the process of accumulation of RCF damage in the side of the rail. By default, the **Simulate damage in the side of the rail** item is enabled. If the user deactivates this option, he needs to specify the section of the rail profile that will be considered side. The **Coordinate x of the side start (mm)** parameter is used for this purpose, see Figure 25.69. The default value of this parameter is 28 mm. The coordinate is set in the reference system shown in Figure 25.61. Thus, the part of the FE model of the rail fragment, the x coordinates of the nodes of which are greater than the given x coordinate of the start of the side part, is considered the side part of the rail.

The purpose of the **Lubrication factor** parameter (Figure 25.69) is described in detail in Sect. 25.5.5.

If the calculation parameters on the **Calculation** tab have been changed, then after clicking the **OK** button, a dialog box appears, see Figure 25.45.

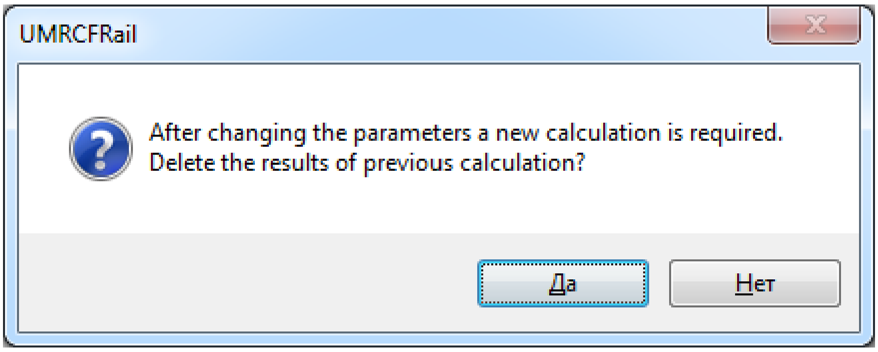


Figure 25.70. Dialog box – request to remove the results of the calculation

The **No** button means that the parameter changes will not be saved.

The **Yes** button means that the parameters will be saved and files with the simulation results will be deleted. The stiffness matrix files, if previously created, will not be deleted. After that the calculation with new parameters can be started.

25.6.3.2. The «Material» tab

The **Material** tab is shown in Figure 25.71.

Young's modulus E ($2\cdot10^{11}$ Pa is the value by default) and Poisson's ratio μ (0.3 is the value by default) can be assigned at the tab.

The RCF curve is approximated by the function (25.1), see Sect. 25.2.4. The values of the material constants C and m in this expression are assigned depending on the chosen RCF criterion. A criterion can be chosen in the **RCF criteria** group box. The criteria are described in the following sections:

1) The criterion of the amplitude of maximum shear stress – see Sect. 25.2.5.1.

2) The Dang Van criterion – see Sect. 25.2.5.2.

3) The Sines criterion – see Sect. 25.2.5.3.

4) The combined criterion – see Sect. 25.2.5.4.

The RCF curves for steels with different chemical composition and values of the material constants C and m for these criteria are given in Sect. 25.2.8.

The user can also set the minimum value of the damaging stress according to the selected RCF criterion. During modelling of the process of accumulation of the RCF damage the cycle is considered as damaging if the value of the criterion exceeds this limit. This minimum value is equal to zero by default.

If any of the parameters at the **Material** tab was changed, then after clicking the **OK** button the dialog box appears as shown in Figure 25.70.

The **No** button means the exit without saving the changed parameters.

The **Yes** button means the exit with saving the changed parameters. The result files will be deleted. If the Young's modulus or Poisson's ratio is changed, the stiffness matrices files will be also deleted.

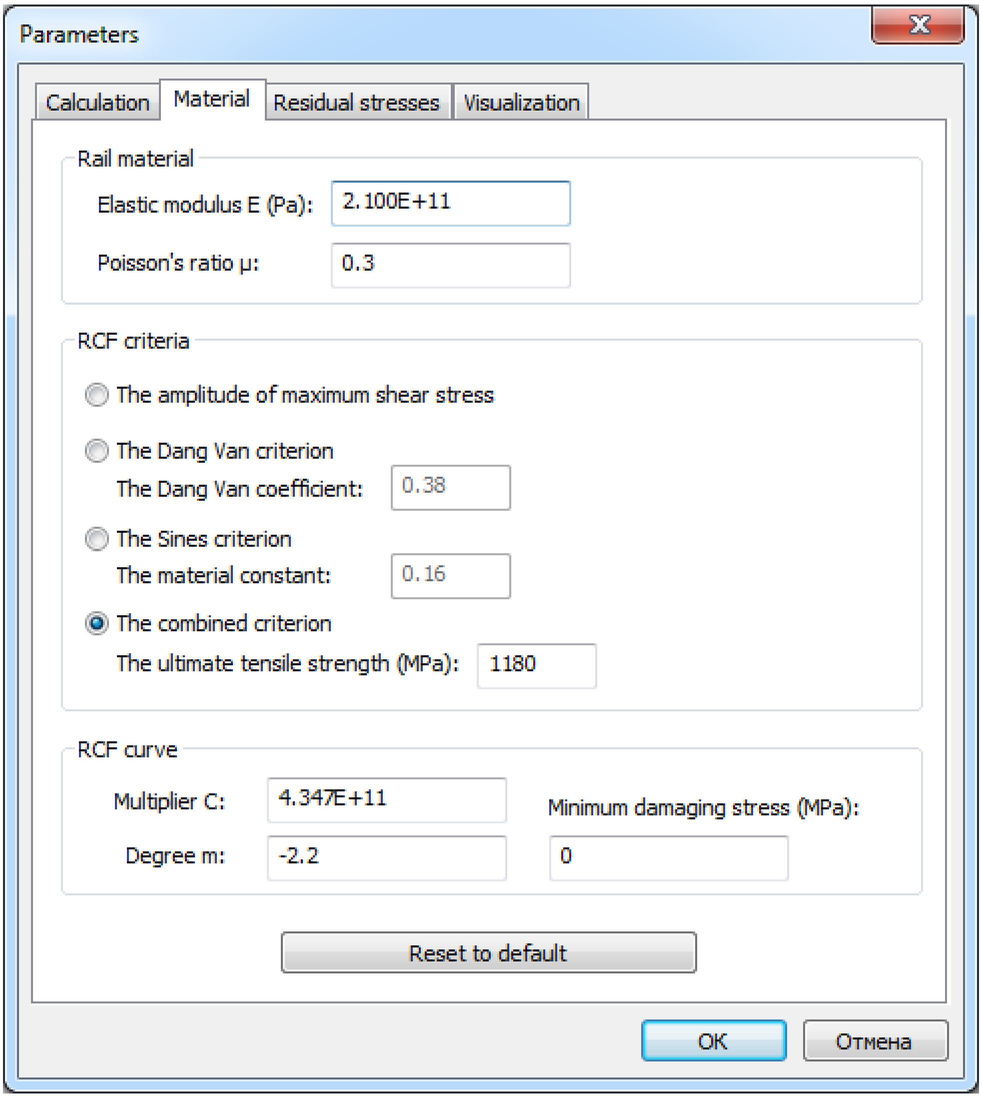


Figure 25.71. The **Parameters** dialog box, the **Material** tab

25.6.3.3. The «Residual stresses» tab

The **Residual stresses** tab is shown in Figure 25.72.

With help of this tab the user can include in the calculation two types of residual stresses: residual technological stresses in the rail after manufacture and any other type of residual stresses, for example, arising in the rail during operation. Residual stresses data is stored in the text files.

The approaches to modelling of the process of accumulation of RCF damage accounting the residual stresses for the wheel and the rail are similar. A description of such approaches is given in Sect. 25.3.7.

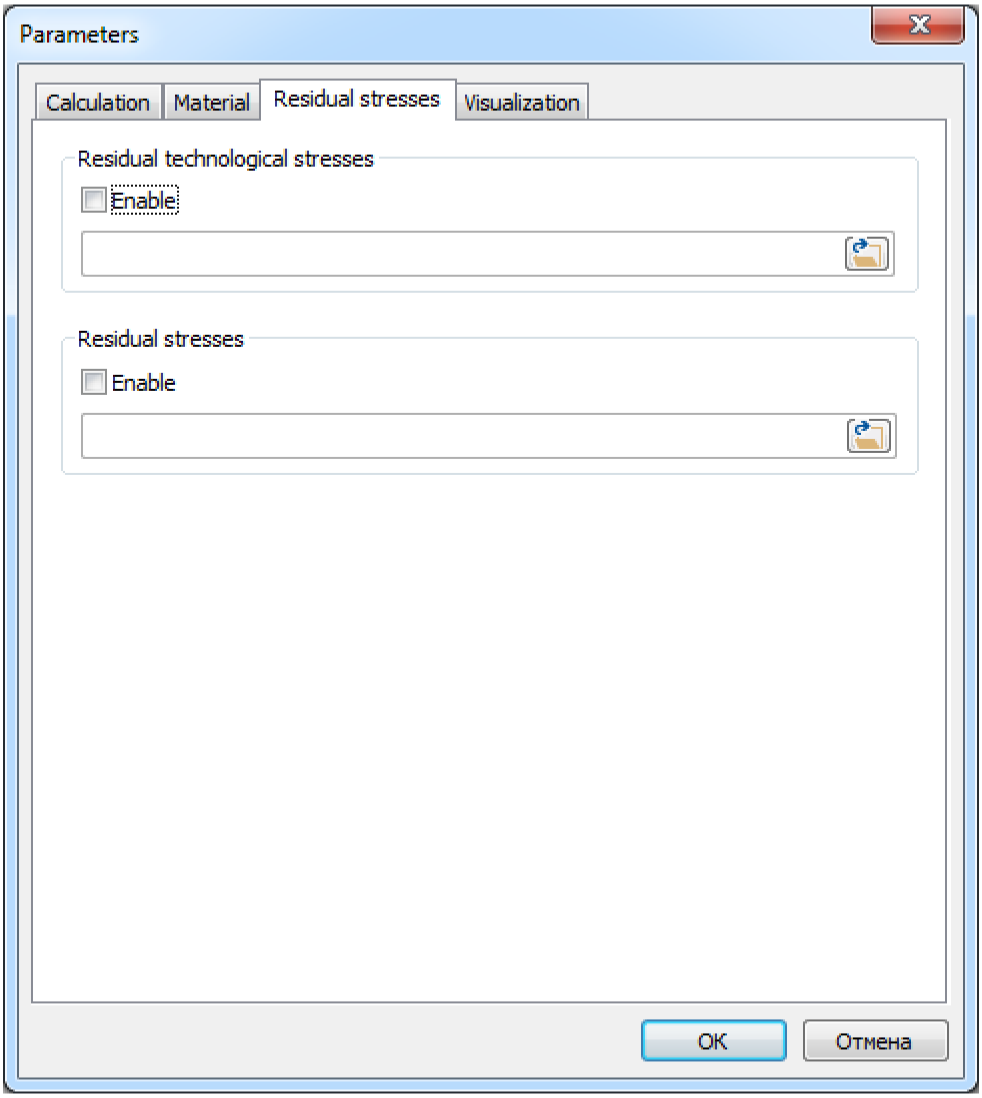


Figure 25.72. The **Parameters** dialog box, the **Residual stresses** tab

25.6.3.4. The «Visualization» tab

The **Visualization** tab is shown in Figure 25.73.

The design scheme can be moved in the window in any direction in the plane of the screen using the arrow keys on keyboard (or the left mouse button), it can be scaled using the «Gray +» and «Gray -» buttons on keyboard (or the mouse wheel). All these operations are performed discretely with a determined step. The step values can be changed at the **Visualization** tab. The steps are set in conventional units.

Also the background color of the workspace of window, the color and width of the lines for displaying the rail profile, FE mesh and isolines can be assigned on the tab.

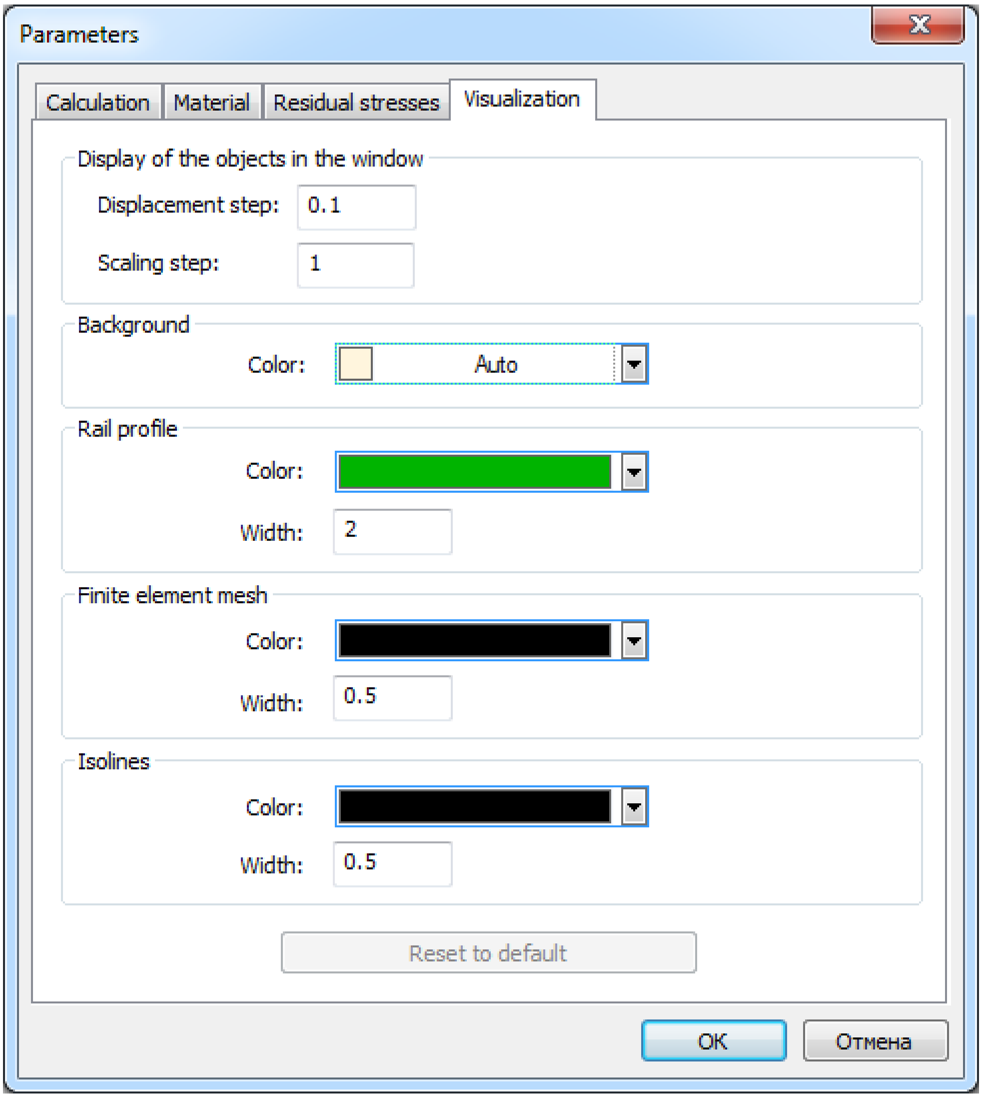


Figure 25.73. The **Parameters** dialog box, the **Visualization** tab

25.6.4. Imaging modes

The several types of the imaging modes of design scheme and calculation results, which can be switched on or off in the various combinations with each other, are implemented in the module.

If necessary, the design scheme in the window can be moved, zoom in or out. The scheme can be returned to its original position in the following ways:

– choose the **Design Scheme | Starting position** menu item;

– click the button  at the **Design Scheme** toolbar;

– click the **Home** key on the keyboard;

– right-click in the workspace of window, choose the **Starting position** item in the appeared shortcut menu.

25.6.4.1. The «Mesh» mode

The **Mesh** imaging mode can be enabled or disabled in three ways:

– choose the **Design Scheme | Mesh** menu item;

– click the button at the **Design Scheme** toolbar;

– click the **1** key on the keyboard.

The mesh consisting of quadrangular elements, which is located at the plane of symmetry of the FE fragment of rail, is displayed in the center of workspace of the window (Figure 25.62). If with the enabled **Mesh** mode the **Color visualization of damage** mode is enabled, then the similar mesh is displayed in the top of window at the median plane of rail fragment, where the process of accumulation of RCF damage is illustrated.

The mode is automatically activated when the data is loaded.

Note. The **Mesh** mode is always available if the data for calculation is loaded.

25.6.4.2. The "Rail profile" mode

The **Rail profile** imaging mode can be enabled or disabled in three ways:

– choose the **Design Scheme | Rail profile** menu item;

– click the button at the **Design Scheme** toolbar;

– click the **2** key on the keyboard.

The rail profile is displayed in the center of workspace of the window (Figure 25.68). If with the enabled **Rail profile** mode the **Color visualization of damage** mode is enabled, then the similar rail profile is displayed in the top of window, where the process of accumulation of RCF damage is illustrated.

The mode is automatically activated when the data is loaded.

Note. The **Rail profile** mode is always available if the data for calculation is loaded.

25.6.4.3. The «Color visualization of damage» mode

The **Color visualization of damage** imaging mode can be enabled or disabled in three ways:

– choose the **Design Scheme | Color visualization of damage** menu item;

– click the button at the **Design Scheme** toolbar;

– click the **3** key on the keyboard.

In the top of window the cross-section of the rail is displayed, in which the distribution of the accumulated RCF damage with help coloring is shown (Figure 25.68). The coloring is performed for each quadrangular element of the flat mesh. The color of the node is chosen depending on the value of the accumulated RCF damage according to an estimation scale. Then the resulting colors from the four vertices are mixed in the plane of the element according to a linear law. In the left side of the workspace of window the color scale for estimation of the level of accumulated RCF damage is displayed.

Note. The **Color visualization of damage** mode is available if the data for calculation is loaded and the calculation has been done.

25.6.4.4. The «Color visualization of stresses» mode

The **Color visualization of stresses** imaging mode can be enabled or disabled in three ways:

– choose the **Design Scheme | Color visualization of stresses** menu item;

– click the button  at the **Design Scheme** toolbar;

– click the **4** key on the keyboard.

In the center of the workspace of window the cross-section of the rail is displayed, in which the distribution of the equivalent stresses with help coloring is shown (Figure 25.68). The coloring is performed for each quadrangular element of the flat mesh. The color of the node is chosen depending on the value of the equivalent stresses according to an estimation scale. Then the resulting colors from the four vertices are mixed in the plane of the element according to a linear law. In the right side of the workspace of window the color scale for estimation of the level of equivalent stresses is displayed.

Note. The **Color visualization of stresses** mode is available if the data for calculation is loaded and the calculation has been done.

25.6.4.5. The «Isolines» mode

The **Isolines** imaging mode can be enabled or disabled in three ways:

– choose the **Design Scheme | Isolines** menu item;

– click the button  at the **Design Scheme** toolbar;

– click the **5** key on the keyboard.

The mode is intended for displaying the lines of equal values of the component – isoparametric lines. The mode supports six isolines, that is, the component is divided into seven levels. If the **Color visualization of stresses** mode is enabled, then in the center of the workspace of window the cross-section of the rail is displayed, in which the isolines of equivalent stresses are shown (Figure 25.68). If the **Color visualization of damage** mode is enabled, then in the top of the workspace of window the cross-section of the rail is displayed, in which the isolines of accumulated RCF damage are shown (Figure 25.68).

Note. The **Isolines** mode is available if the data for calculation is loaded, the calculation has been done and the **Color visualization of damage** mode or **Color visualization of stresses** mode is enabled.

25.6.4.6. The «Maximum damage» mode

The **Maximum damage** imaging mode can be enabled or disabled in three ways:

– choose the **Design Scheme | Maximum damage** menu item;

– click the button  at the **Design Scheme** toolbar;

– click the **6** key on the keyboard.

In the top of the workspace of window the cross-section of the rail is displayed, in which the node with the biggest accumulated RCF damage is marked by the white circle (Figure 25.68). The value of the biggest damage can be viewed at the **Information Panel** in the **Calculation** parameters group box (Figure 25.75).

Note. The **Maximum damage** mode is available if the data for calculation is loaded, the calculation has been done, the **Color visualization of damage** mode is enabled and the biggest accumulated damage more than zero.

25.6.4.7. The «Accumulated damage» mode

The **Accumulated damage** imaging mode can be enabled or disabled in three ways:

– choose the **Design Scheme | Accumulated damage** menu item;

– click the button at the **Design Scheme** toolbar;

– click the **7** key on the keyboard.

If the **Color visualization of damage** mode is enabled, then in the top of the workspace of window the cross-section of the rail is displayed, in which the nodes of FE mesh are marked by the points, beside each node the numerical value of accumulated RCF damage in this node is shown. If the **Color visualization of stresses** mode is enabled, then in the center of the workspace of window the cross-section of the rail is displayed, in which the nodes of FE mesh are marked by the points, beside each node the numerical value of equivalent stress in this node is shown.

Note. The **Accumulated damage** mode is available if the data for calculation is loaded, the calculation has been done and the **Color visualization of damage** mode or **Color visualization of stresses** mode is enabled.

25.6.4.8. The «Numbers of the nodes» mode

The **Numbers of the nodes** imaging mode can be enabled or disabled in three ways:

– choose the **Design Scheme | Numbers of the nodes** menu item;

– click the button at the **Design Scheme** toolbar;

– click the **8** key on the keyboard.

In the center of the workspace of window the cross-section of the rail is displayed, in which the nodes of FE mesh are marked by the points, beside each node its number is shown. If the **Color visualization of damage** mode is enabled, then the numbers are shown beside the nodes of the cross-section of the rail in the top of the workspace of window, where the process of accumulation of RCF damage is illustrated.

Note. The **Numbers of the nodes** mode is always available if the data for calculation is loaded.

25.6.4.9. The «Residual stresses» mode

The **Residual stresses** imaging mode can be enabled or disabled in two ways:

– choose the **Design Scheme | Residual stresses** menu item;

– click the button at the **Design Scheme** toolbar.

In the center of the workspace of window the cross-section of the rail is displayed, in which the distribution of one of the components of residual stresses with help coloring is shown. The coloring is performed for each quadrangular element of the flat mesh. The color of the node is chosen depending on the value of the stress component according to an estimation scale. Then the resulting colors from the four vertices are mixed in the plane of the element according to a linear law. In the right side of the workspace of window the color scale for estimation of the level of selected stress component is displayed.

The component of residual stresses for displaying can be chosen at the **Residual stresses** dialog box (Figure 25.74). To open the dialog box select the **Design Scheme | Choice of component** menu item or click the button at the **Design Scheme** toolbar.

Note. The **Residual stresses** mode is available if the residual stresses data is loaded (see Sect. 25.6.3.3).

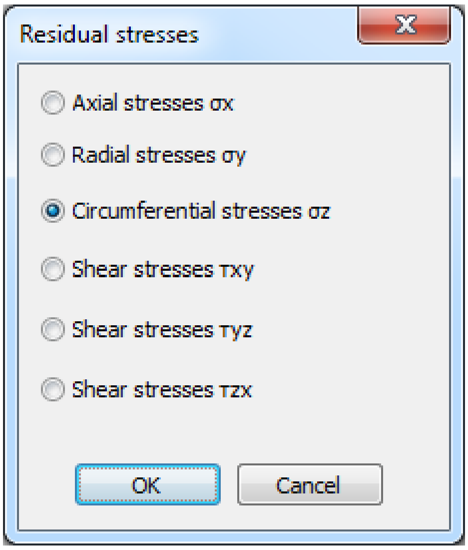


Figure 25.74. Selection of residual stresses components for displaying

25.6.5. Performing calculation

The calculation can be started in the following ways:

– choose the **Design Scheme | Run** menu item;

– click the button at the **Design Scheme** toolbar;

– click the **F9** key on the keyboard;

– right-click in the workspace of window, choose the **Run** item in the appeared shortcut menu.

The calculation is performed sequentially for each of the *rail wear iteration* in accordance with the initial data in the **rld** file.

If the condition for the end of the calculation is the achievement of the limit value of the accumulated damage at the point of the rail, and it is reached before the calculations were performed at all the *rail wear iterations*, then the procedure ends, and the calculation for the remaining *rail wear iterations* is not performed. If the condition for the end of the calculation is the passed tonnage, then the calculation is performed at all *rail wear iterations*.

The **Information Panel** allows monitoring the status of the calculation (Figure 25.75).

After completion of the calculation the program displays a corresponding message in the dialog box on the screen.

The calculation can be stopped in the following ways:

– choose the **Design Scheme | Stop** menu item;

– click the button at the **Design Scheme** toolbar;

– click the **Esc** key on the keyboard;

– right-click in the workspace of window, choose the **Stop** item in the appeared shortcut menu.

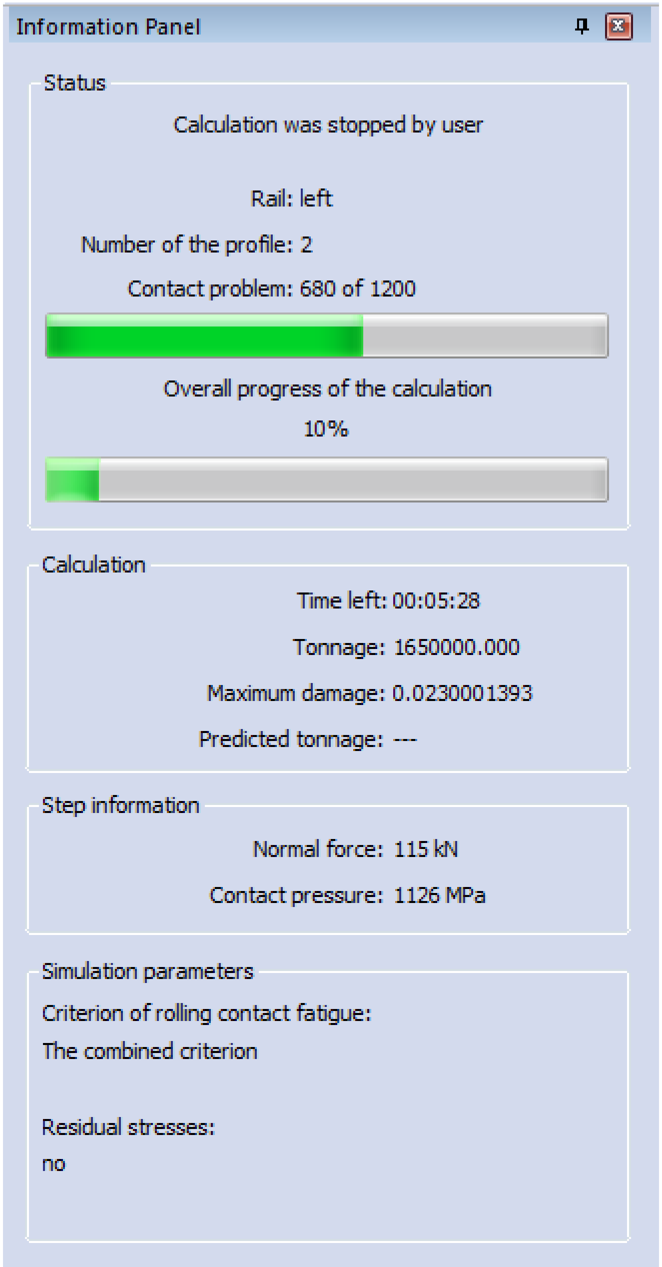


Figure 25.75. The **Information Panel**

25.6.6. Saving the results of calculation

The simulation results can be saved by means the **File | Save** menu item or the button on the **Standard** toolbar. This option is not available during the calculation running.

Note. The autosave of results after the completion of calculation for each *rail wear iteration* is supported.

The calculation results are saved in two files. Both files have the same name as the **rld** file and are located in the source data folder.

The first file is binary and it has the ***.rcfresults** extension. The data from this file are used to display the calculation results in the **UM RCF Rail** module and are loaded by the program automatically when opening the file with input data **rld**.

The second file is a text file with the extension ***.csv**. This file allows the user to open the calculation results in **Excel** tables.

25.6.7. Deleting the results of calculation

The calculation results can be deleted by using the **Design Scheme | Delete results** menu item or the button  on the **Design Scheme** toolbar. The option is available if the calculation was carried out.

Note. The ***.smx** files with saved components of stiffness matrices (if they exist) will not be deleted.

If it is necessary to delete the stiffness matrices files, the **Design Scheme | Delete stiffness matrix** menu item, or the button , or the **Ctrl+Del** hot key can be used (see Sect. 25.6.3.1).

25.6.8. Viewing calculation results

After completion of the calculation for the first *rail wear iteration* the options are available allowing to navigate through the list of *rail wear iterations* in two directions: from the first to the last and back.

These options can be used in the following ways:

– choose the **Design Scheme | Previous profile** or **Design Scheme | Next profile** menu items;

– click the button  or  at the **Design Scheme** toolbar;

– click the **PgDn** or **PgUp** key on the keyboard.

At that the calculation results for selected *rail wear iteration* are displayed at the **Information Panel** and the workspace of window.

At the **Status** group box of the **Information Panel** the general information is displayed: the rail – left or right, the number of the profile – the number of *rail wear iteration*, the total number and number of solved contact problems for the current *rail wear iteration*, the percentage of overall progress of the calculation (Figure 25.75).

First of all, when analyzing the calculation results, the value of the biggest accumulated damage at the point of the rail and the passed tonnage are of interest. These parameters can be seen at the **Calculation** group box of the **Information Panel** (Figure 25.75). If the calculation was carried out until reaching the limit of accumulated RCF damage and this limit has not been reached, the predicted tonnage at the same group box shows how many tons gross could still be passed through the *control section of rail* with the reached level of accumulated damage. The predicted tonnage is determined by dividing the tonnage by the biggest accumulated damage.

The point of the rail with the biggest accumulated damage can be marked on the design scheme at the workspace of window with help the **Maximum damage** imaging mode (see Sect. 25.6.4.6). The depth at which this point is located under the rolling surface of the rail can be determined using the **Mesh** imaging mode, starting from the fact that the side size of the finite element of the mesh is 1 mm (see Sect. 25.6.4.1). The **Color visualization of damage** mode (see Sect. 25.4.4.3) and **Isolines** mode (see Sect. 25.4.4.5) allow representing the distribution of accumulated RCF damage in the cross-section of the rail.

File with ***.csv** extension (see Sect. 25.6.6) allows the user to open the calculation results in **Excel** tables and create graphs for preparing a report.

Using the results from the first data block, it is possible to plot the dependence of the accumulated damage at the dangerous point of the rail material on the tonnage. An example of such a graph is shown in Figure 25.76.

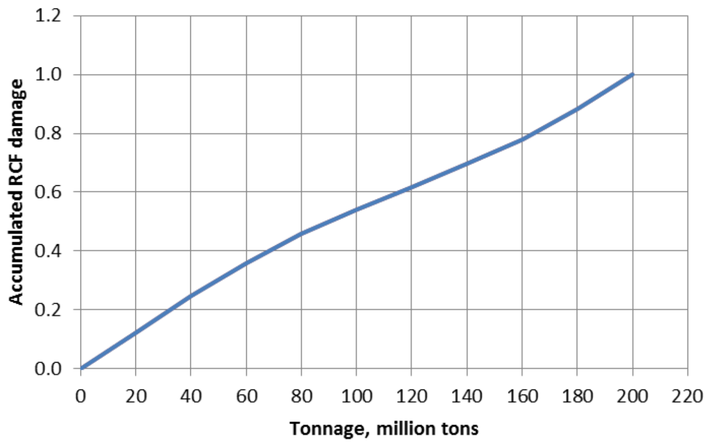


Figure 25.76. Dependence of accumulated damage at the dangerous point of the rail material on the tonnage

The data of the second block allows constructing a diagram of the accumulated RCF damage in the dangerous section of the rail. By the dangerous section we will mean the longitudinal section of the rail containing the dangerous point, i.e. the point with the biggest accumulated RCF damage. To construct the diagram the values of the accumulated damage at the nodes of the FE mesh of the cross-section of the rail fragment belonging to the dangerous section are used. The example of such sequence of the nodes is shown in Figure 25.77a. The abscissa axis is the *x* axis in the coordinate system shown in Figure 25.61. The graph of the dependence of the accumulated damage in the nodes of the dangerous section of the rail on their coordinate *x* is shown in Figure 25.77b.

The second block also contains the data allowing to construct the surface of accumulated damage in the cross-section of the rail fragment. The *x* coordinate of the node is plotted along the abscissa in accordance with the frame of reference shown in Figure 25.61. The ordinate is the depth at which the node is located below the rolling surface. The value of the accumulated damage in the nodes of the FE mesh of the cross-section of the rail fragment is plotted along the applicate. The example of such surface is shown in Figure 25.78.

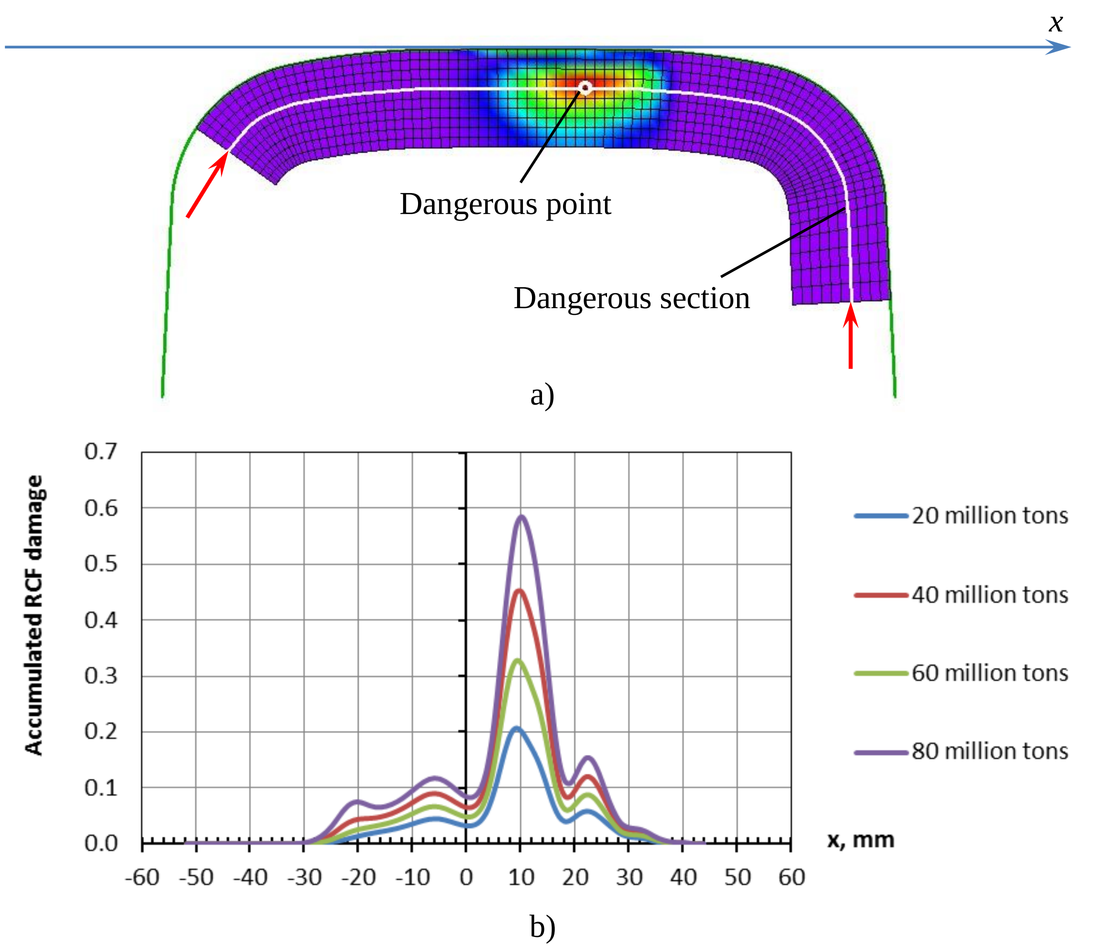


Figure 25.77. Diagram of accumulated RCF damage in the dangerous section of the rail

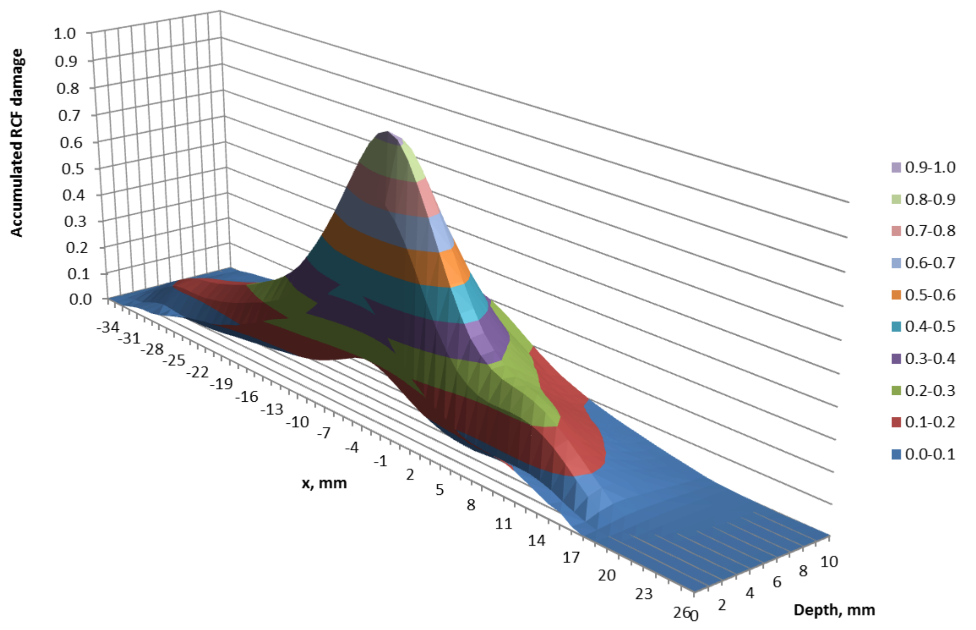


Figure 25.78. Accumulated damage surface in the cross-section of the rail fragment

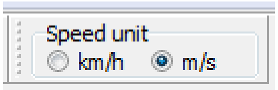
25.6.9. Sample of simulation of accumulation of damage in the rail

Let us consider an example of modelling of the process of accumulation of RCF damage in the rail.

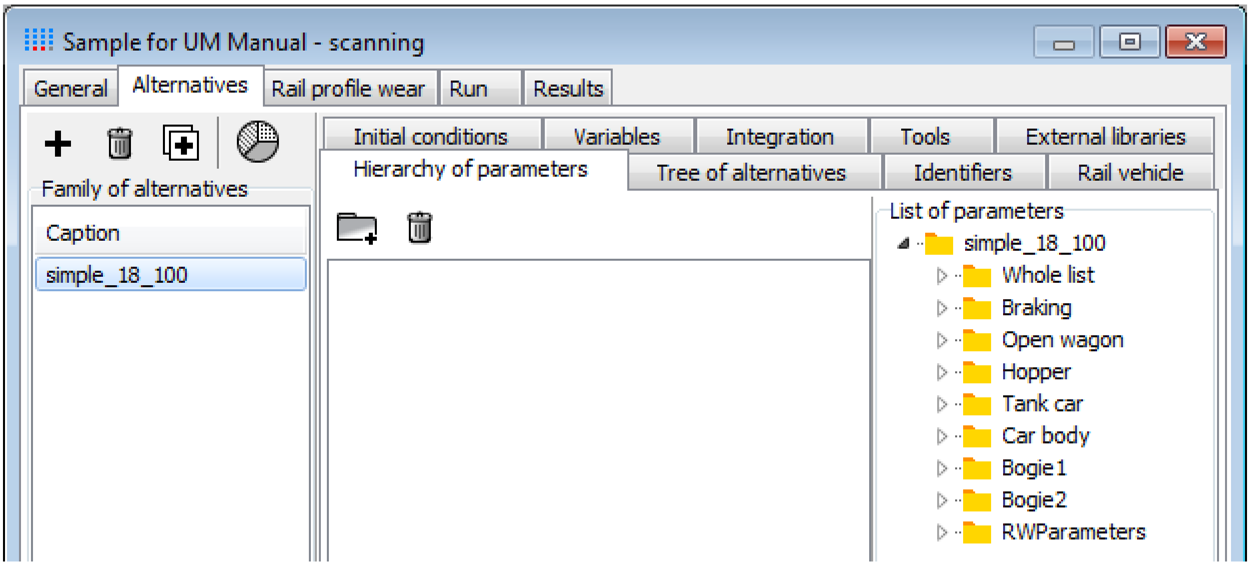
To prepare the initial data, let us simulate the wear process of the P65 rail [40] in a tangent section of the track when a loaded freight car passes along it on three-piece bogies of model 18-100 at speeds of 40, 60 and 80 km/h. The track irregularities correspond to the track of a bad quality according to the UIC standard. The car model is supplied with the **UM** software and it is available in the library {UM Data}\samples\Rail_Vehicles\simple_18_100.

Start **UM Simulation** and create a new scan project using the menu item **Scanning | New project - rail profile wear...** .

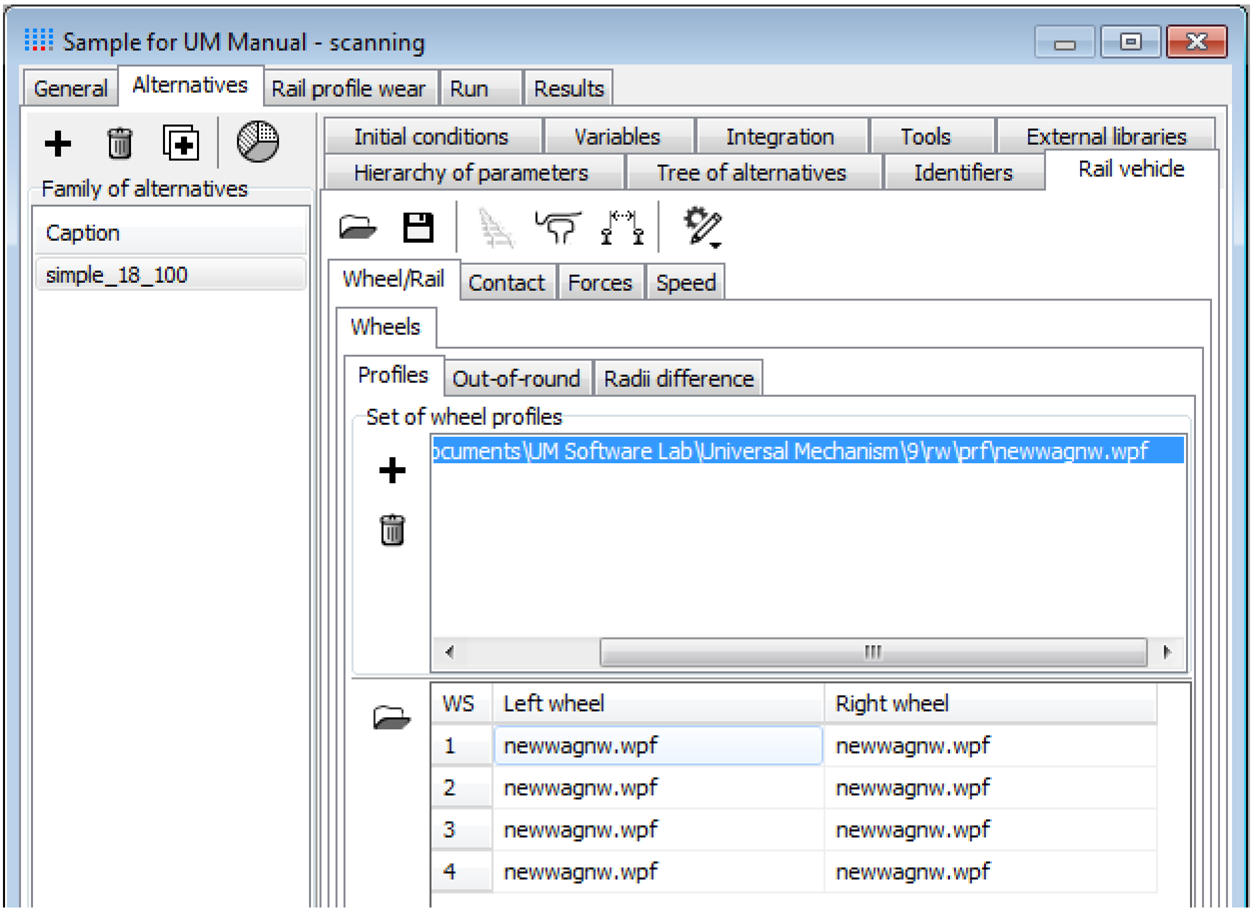
Make sure that the unit of measure for speed is m/s (meters per second) in the program. Set the **m/s** value on the **Speed unit** toolbar.



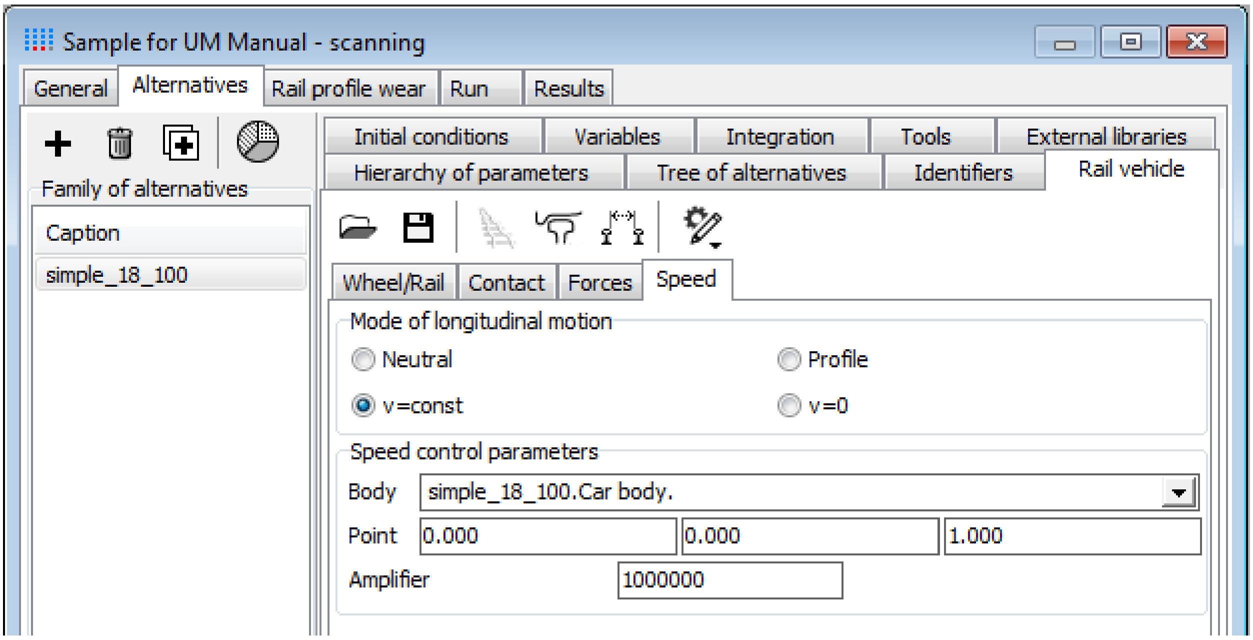
In the scanning project window open the **Alternatives** tab and add a new family of alternatives using the ✚ button. In the opened window select the model of the freight car *simple_18_100*. The selected model appears in the **Family of alternatives** list.



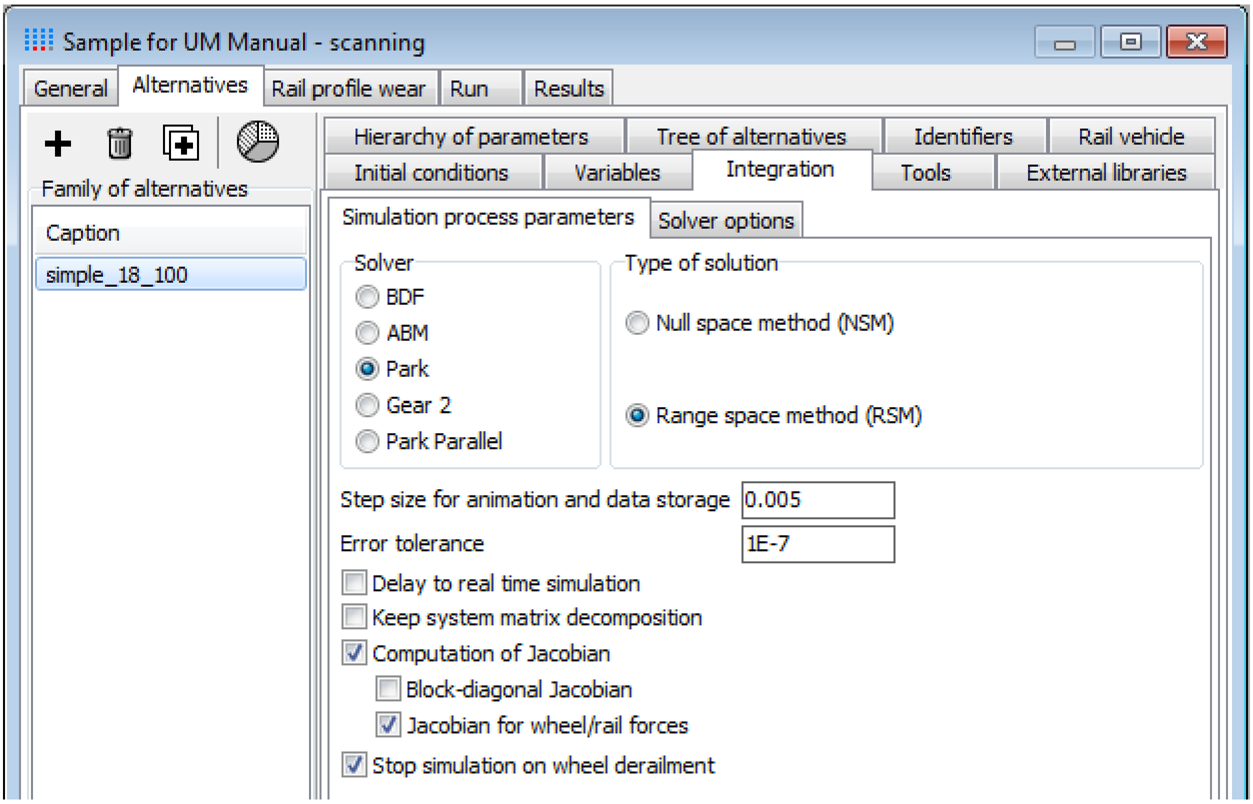
Open the **Rail vehicle | Wheel/Rail | Wheels | Profiles** tab. Clear the list of **Set of wheel profiles** by deleting the default profiles using the button 🗑. Add the *newwagnw.wpf* profile to the list **Set of wheel profiles** from the {UM Data}\rw\prf directory using the ✚ button and assign it to all wheels.



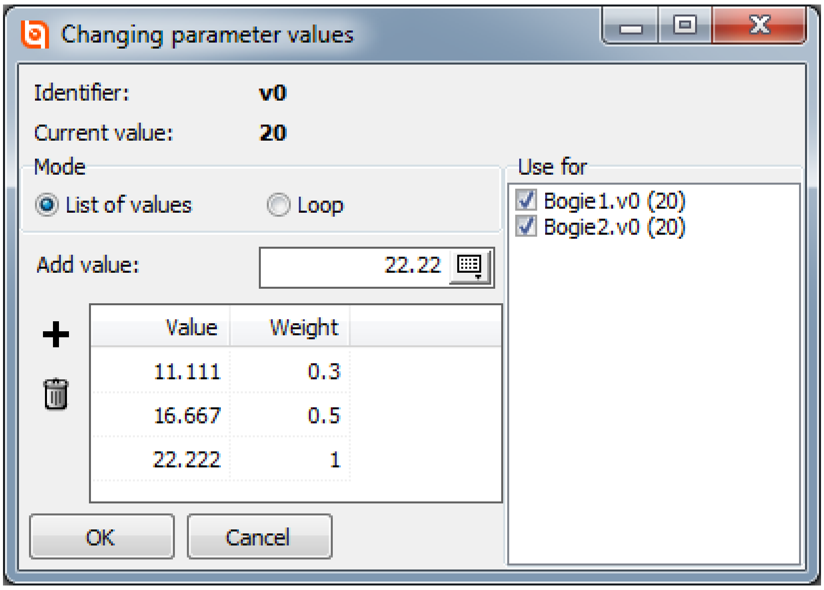
Open the **Rail vehicle | Speed** tab and select the mode of longitudinal motion **v = const**. In the **Body** item of the **Speed control parameters** section select the *simple_18_100.Car body* from the drop-down list.



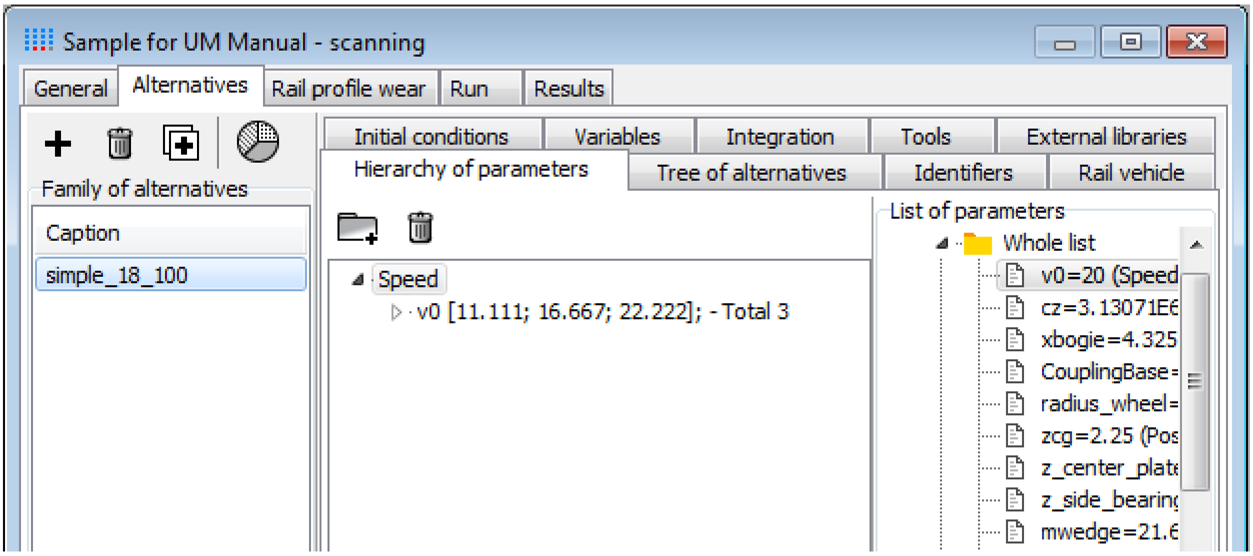
Open the **Integration | Simulation process parameters** tab. Set the value **1E-7** in the **Error tolerance** item. Enable the check boxes **Computation of Jacobian** and **Jacobian for wheel/rail forces**.



Open the **Hierarchy of parameters** tab. Add a new group of parameters using button and name it **Speed**. Click the *v0* parameter in the hierarchical list **List of parameters**. In the opened window **Changing parameter values** set the values 11.111, 16.667 and 22.222 (m/s) and the weights 0.3, 0.5 and 0.2 accordingly for these values, as shown in figure.

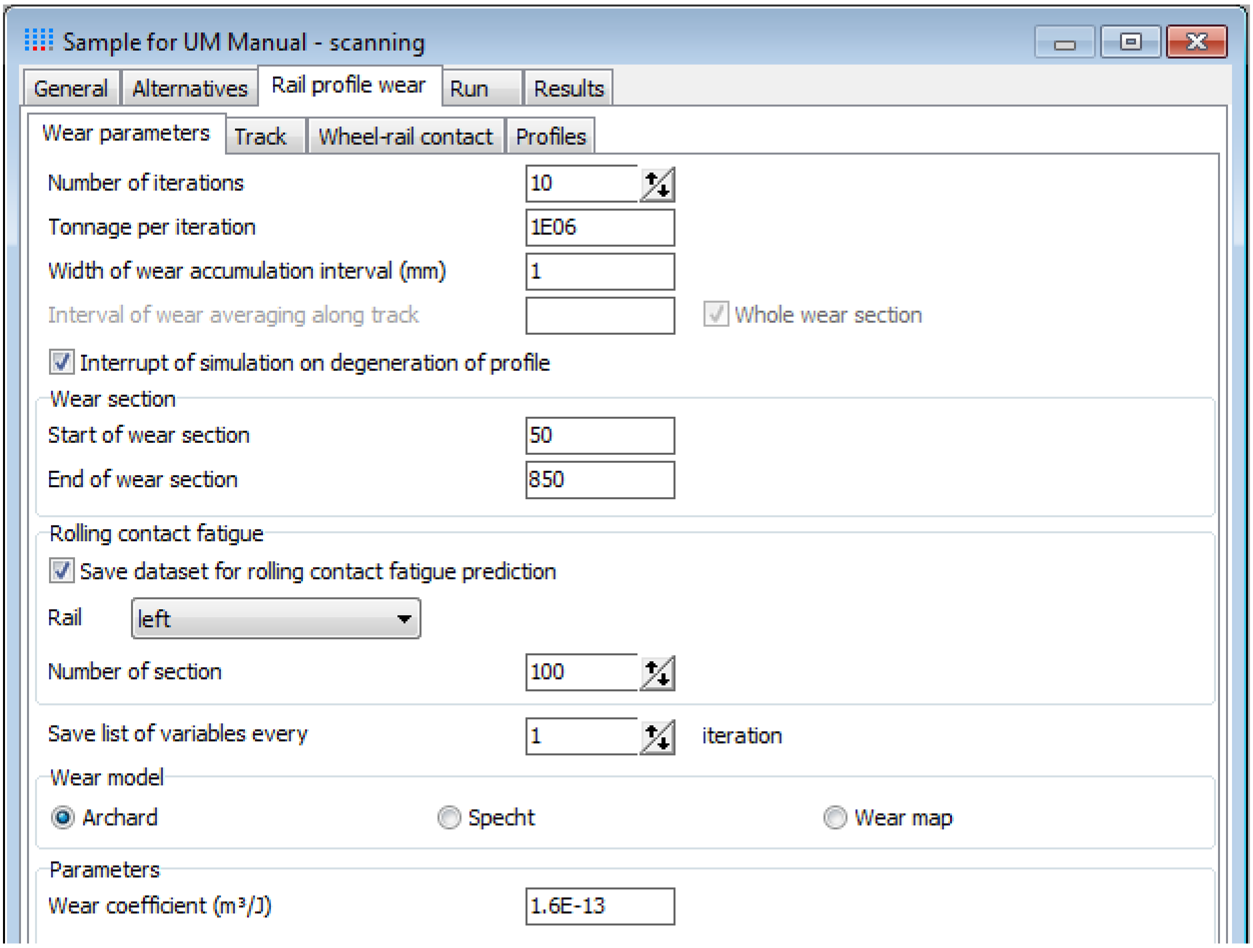


Close the **Changing parameter values** window and return to the scanning project window. After the performed actions the hierarchy of parameters for the family of alternatives *simple_18_100* will look as shown in figure.



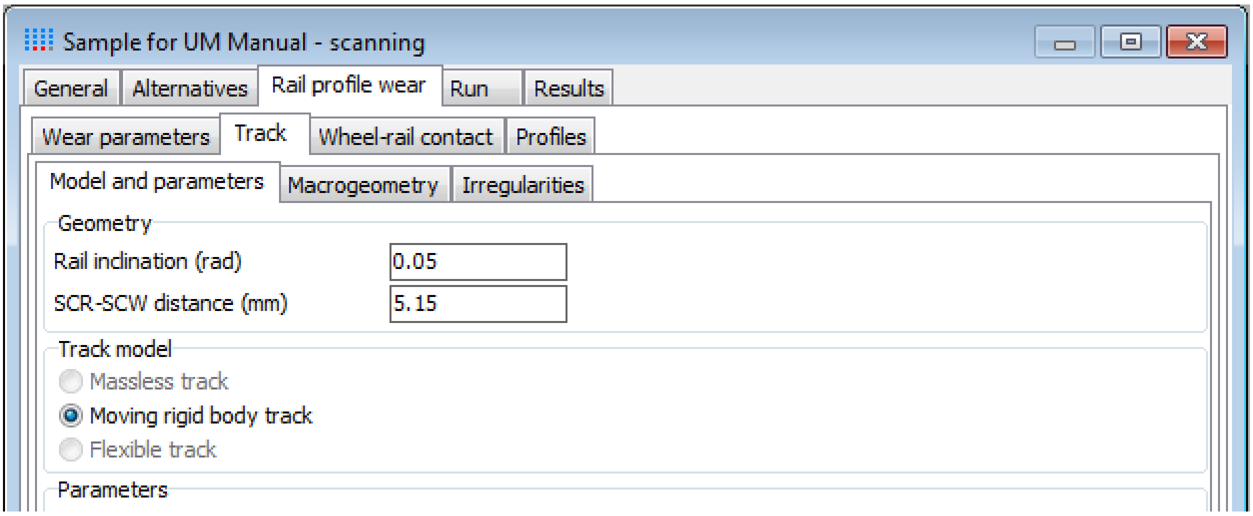
Open the **Rail profile wear | Wear parameters** tab and set the following values for the wear parameters:

- **Number of iterations** = 10;
- **Tonnage per iteration** = 10^6;
- **Width of wear accumulation interval (mm)** = 1;
- **Interrupt of simulation on degeneration of profile** = yes;
- **Start of wear section** = 50;
- **End of wear section** = 850;
- **Save list of variables every** = 1 iteration;
- **Wear model** = **Archard**;
- **Wear coefficient (m^3/J)** = $1.6\cdot10^{-13}$;
- **Save dataset for rolling contact fatigue prediction** = yes;
- **Rail** = left;
- **Number of section** = 100.

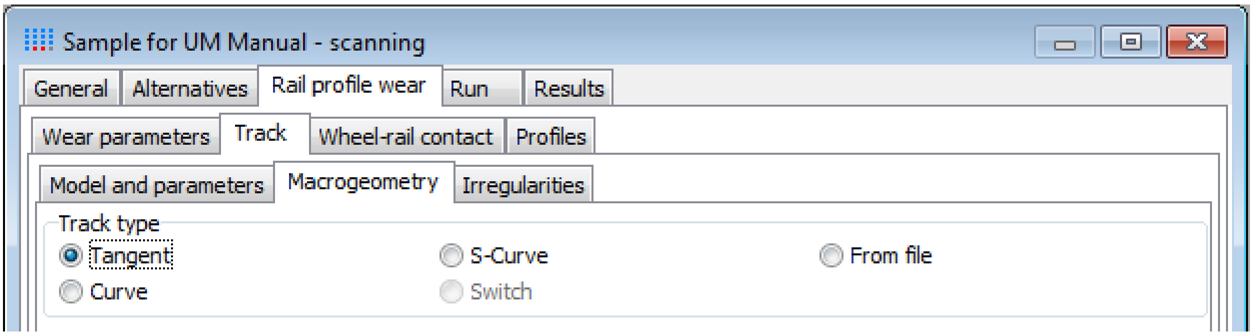


Open the **Rail profile wear | Track | Model and parameters** tab and set the following values for the parameters:

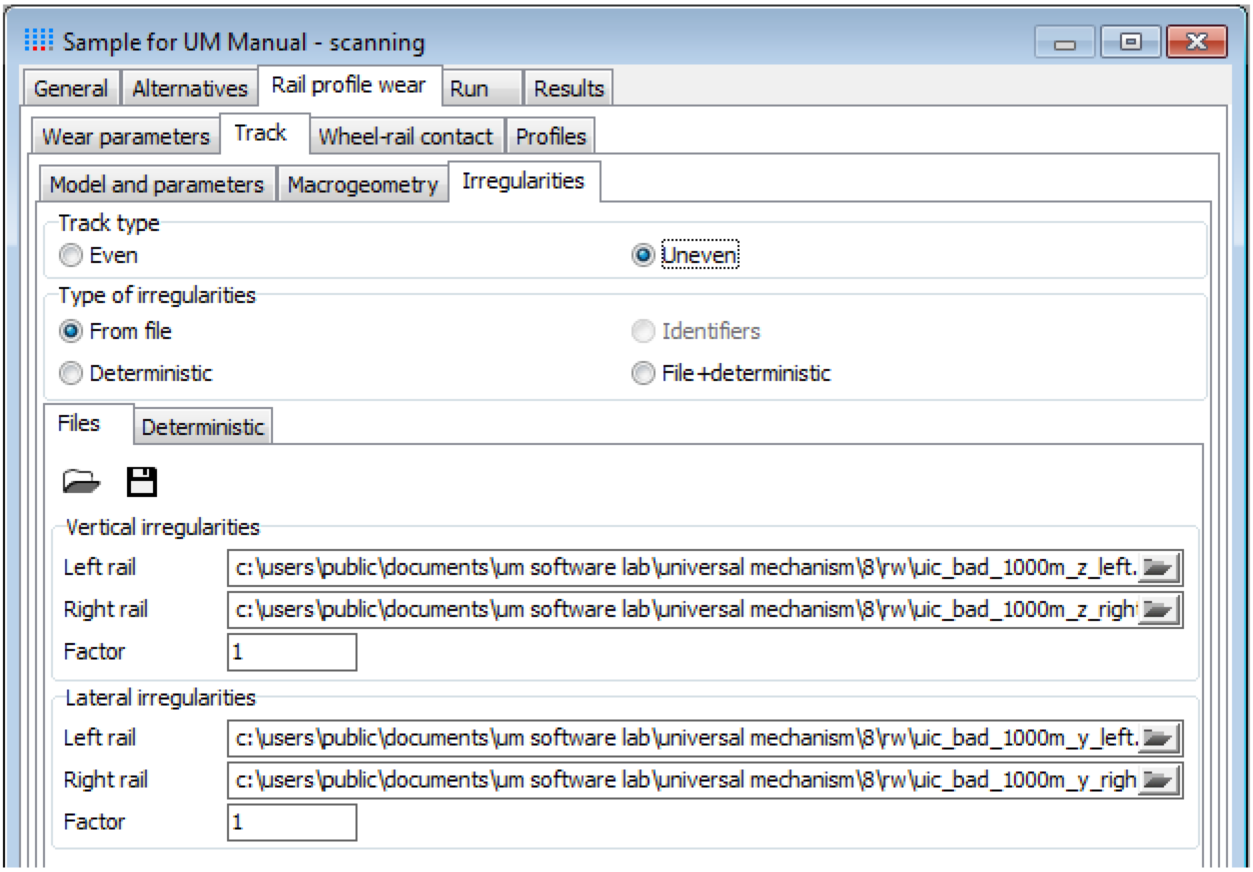
- **Rail inclination (rad)** = 0.05;
- **SCR-SCW distance (mm)** = 5.15;
- **Track model** = **Moving rigid body track**.



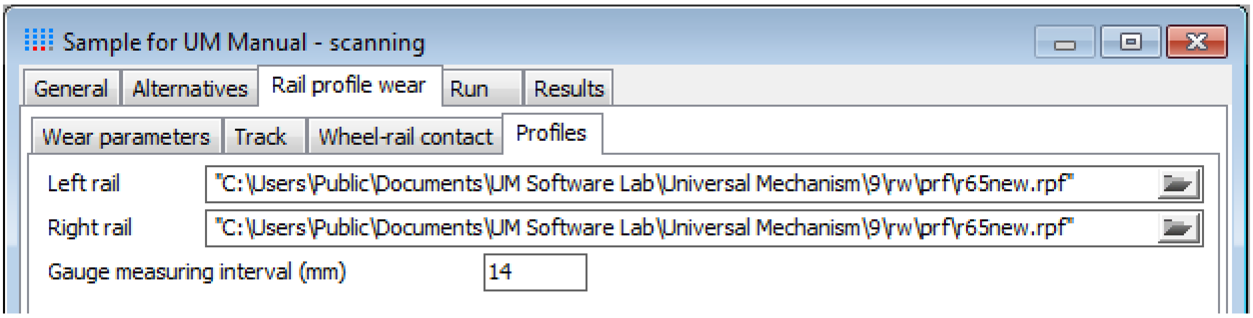
Open the **Rail profile wear | Track | Macrogeometry** tab and set the **Track type** to **Tangent**.



Open the **Rail profile wear | Track | Irregularities** tab, set the **Track type** to **Uneven** and **Type of irregularities** to **From file**. Set the group of irregularities *uic_bad_1000m.tig* for the rails from {UM Data}\rw directory using the **Load group of irregularities from file** button.

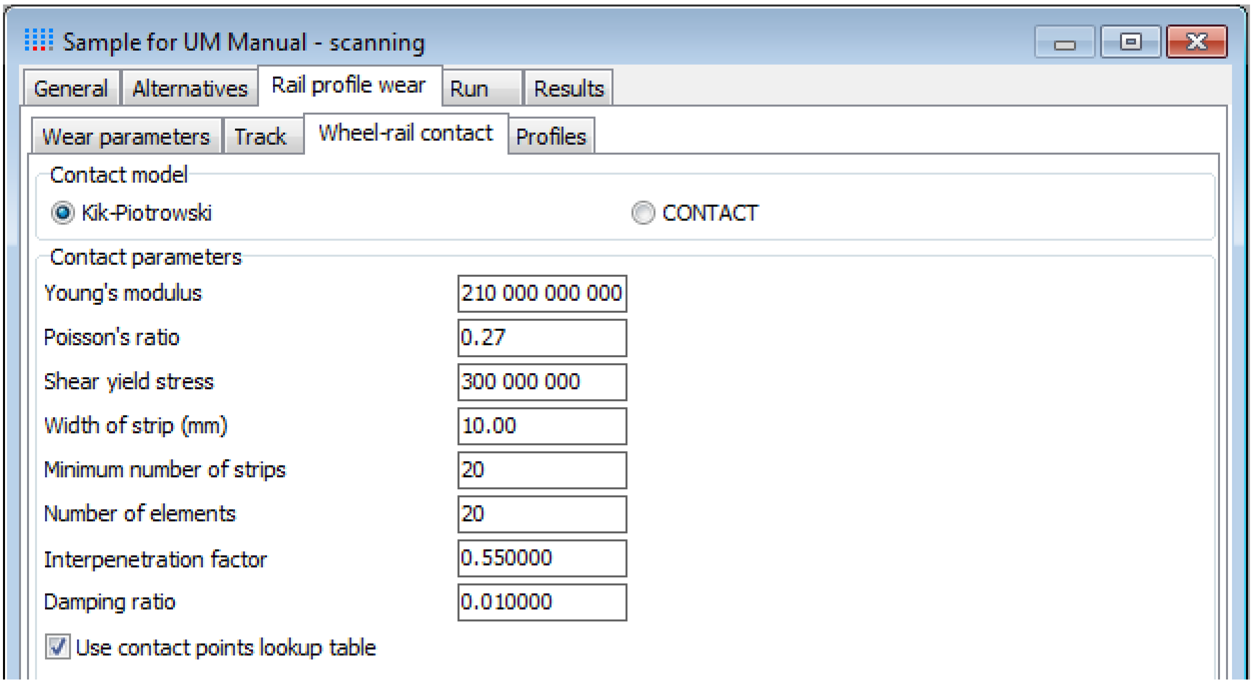


Open the **Rail profile wear | Profiles** tab and set the *r65new.prf* profile from the {UM Data}\rw\prf directory for the both rails. For more information on the assignment of rail profiles, see Chapter 8, Sect. "Assignment of rail profiles". Set the **Gauge measuring interval (mm)** to 14.



Open the **Rail profile wear | Wheel-rail contact** tab. Select the **Kik-Piotrowski** contact model and set the following values for the contact parameters:

- **Young's modulus** = $2.1 \cdot 10^{11}$;
- **Poisson's ratio** = 0.27;
- **Shear yield stress** = $3 \cdot 10^{8}$;
- **Width of strip (mm)** = 10;
- **Minimum number of strips** = 20;
- **Number of elements** = 20;
- **Interpenetration factor** = 0.55;
- **Damping ratio** = 0.01.



Wear project is ready. Open the **Run** tab. With the event log window make sure that scanning project contains no errors. Set the required number of processes based on the capabilities of your computer. Start the simulation by clicking the **Run** button.

As a result of the simulation, data were obtained for ten *rail wear iterations* of the left rail in the tangent section of the track; the total tonnage is 10 million tons, the tonnage passed at each iteration is 1 million tons.

Open the RCF project in **UM RCF Rail** module in any of the ways specified in Sect. 25.6.1. Change the project settings at the **Calculation** and **Material** tabs as shown in Figure 25.79 and Figure 25.80. Keep the parameters at the **Residual stresses** tab unchanged. Save the project settings by click **OK** button at the **Parameters** dialog box. Run the simulation by the **Run** button (see Sect. 25.6.5).

Various possibilities of representation of the calculation results are described in Sect. 25.6.8. In Figure 25.81, Figure 25.82 and Figure 25.83 the results of simulation of the process of accumulation of the RCF damage in the *control section of the rail* on the tangent track section are shown.

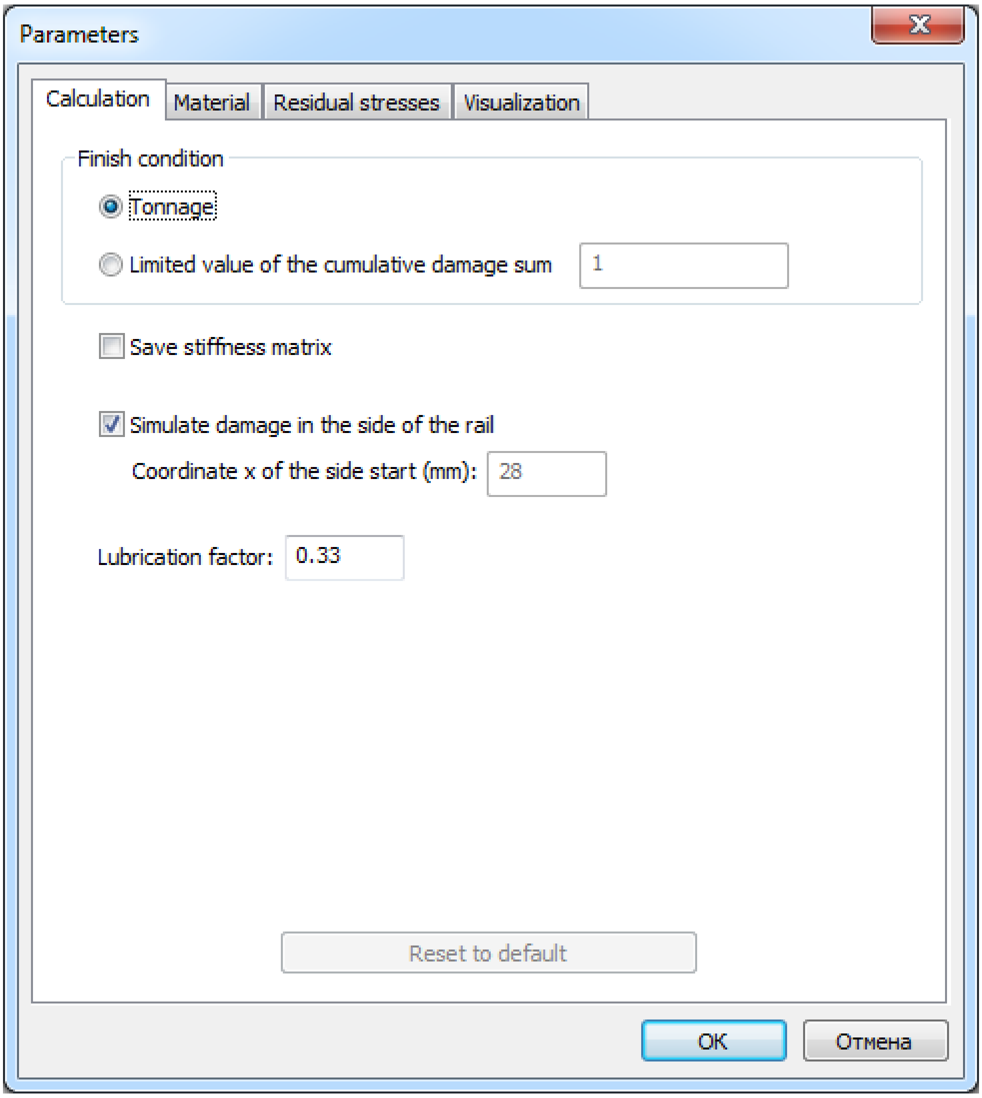


Figure 25.79. RCF project settings on the **Calculation** tab

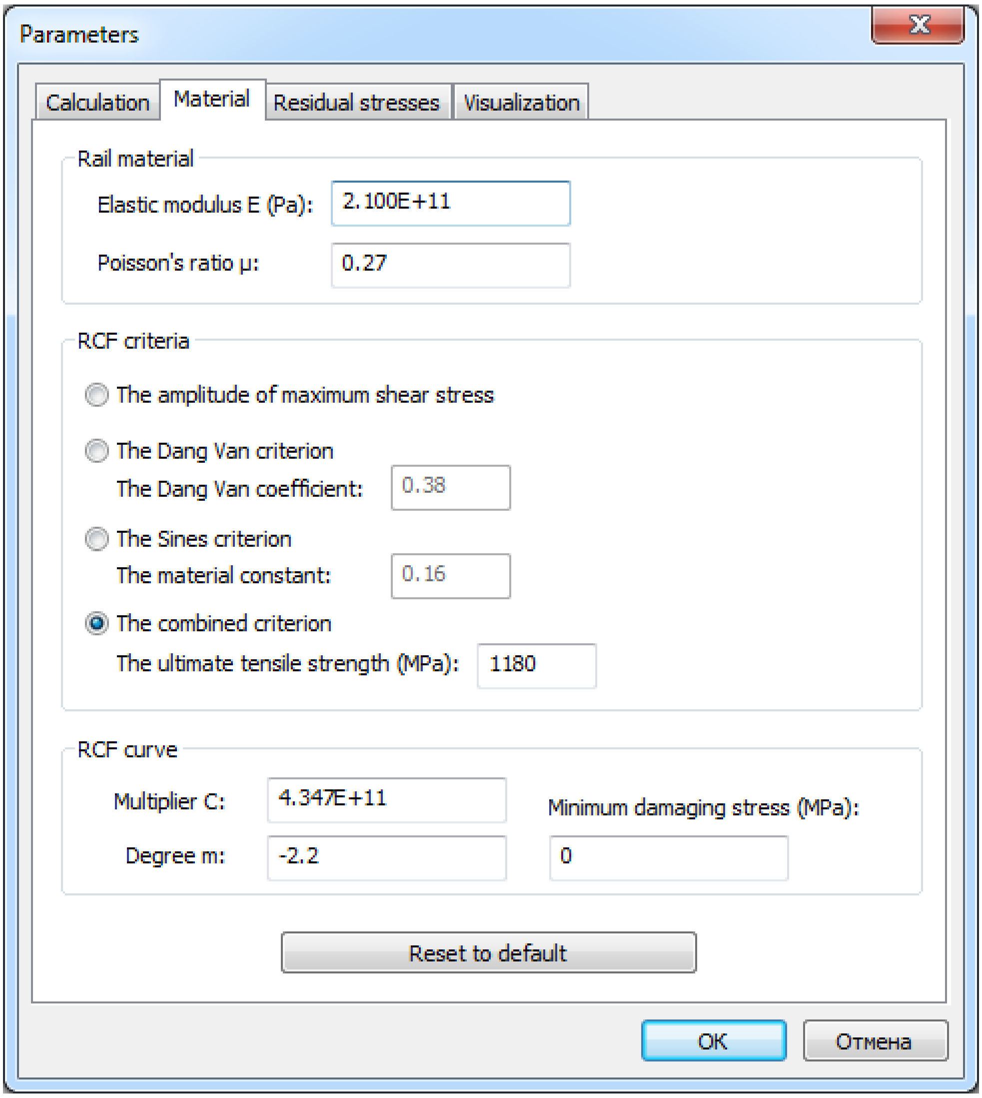


Figure 25.80. RCF project settings on the **Material** tab

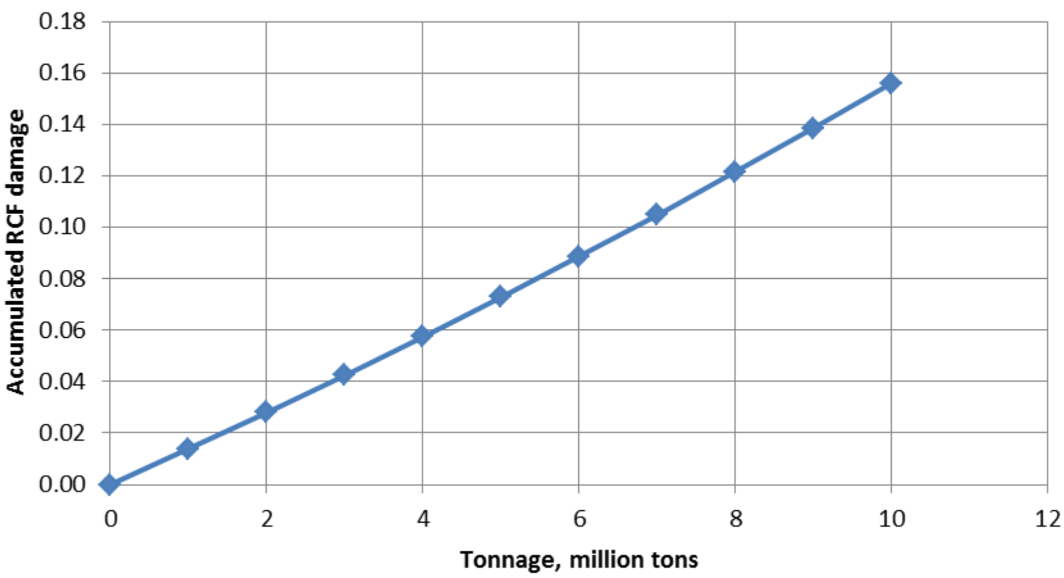


Figure 25.81. Accumulated RCF damage at the danger point of the rail material

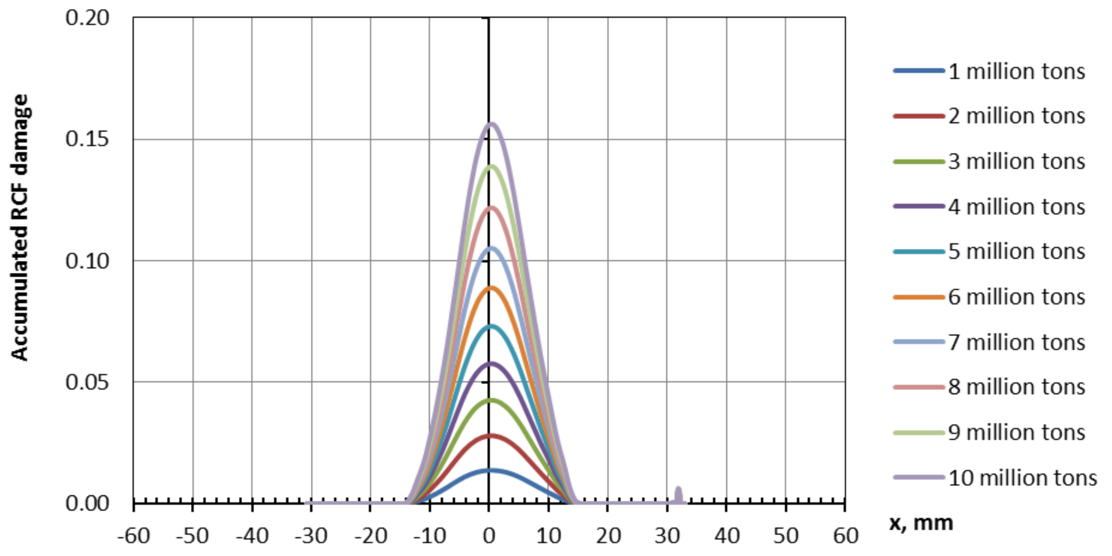


Figure 25.82. Diagram of accumulated RCF damage in the dangerous section of the rail

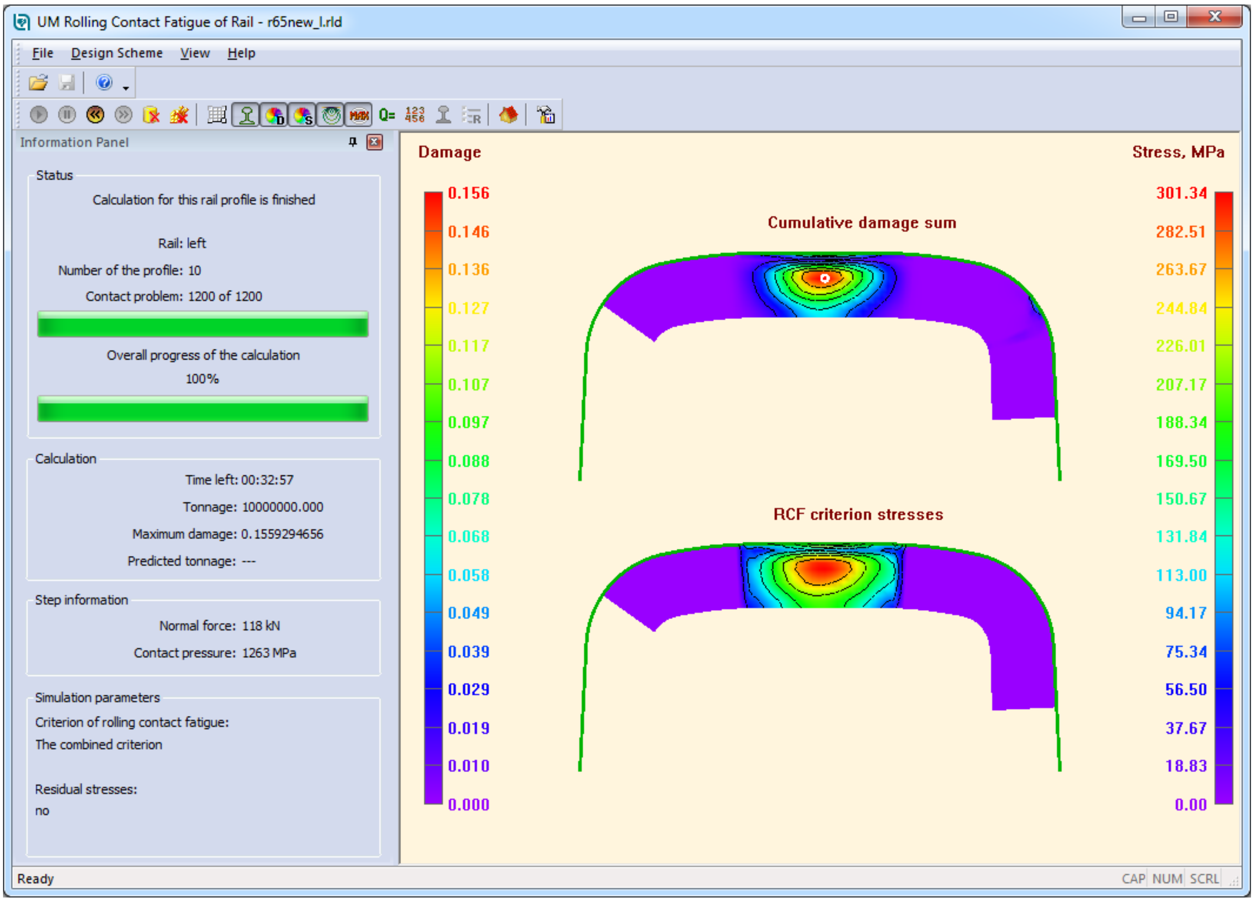


Figure 25.83. Results of simulation of the process of accumulation of RCF damage in the material of the rail

References

[1] Zakharov S.M. Rolling contact fatigue damages of freight cars wheels. In: Intext, Moscow; 2004 (in Russian).

[2] Shkolnik L.M., Markov D.P., Proydak U.S., et al. Increasing the durability of railway wheels in operation by carbonitride hardening of steel. Bull All Russia Sci Res Inst Railway Transport 1994;6:40–44 (in Russian).

[3] Ekberg A. Rolling contact fatigue of railway wheels. In: PhD Thesis Chalmers University of Technology, Gothenburg, Sweden, 2000.

[4] Ekberg A., Marais J. Effects of imperfections on fatigue initiation in railway wheels. Jornal of Rail and Rapid Transit 2000;214:1–18.

[5] Mutton P.J., Epp C.J., Dudec J. Rolling contact fatigue of railway wheels under high axle load. Wear 1997;211:280–288.

[6] Zakharov S.M. Generalization of the world experience of heavy haul traffic. Maintenance management of system wheel-rail. Moscow, Intext Publ., 2017, 420 p.

[7] GOST 10791-2011. Interstate Standard. All-rolled wheels. Specifications.

[8] AAR Manual of Standards and Recommended Practices Wheels and Axles. Wheels, Carbon Steel, Specification M-107/M-208, Adopted: 1962; Last Revised: 2012.

[9] European standard EN 13262:2004+A2:2011. Railway applications – Wheelsets and bogies – Wheels — Product requirements.

[10] TB/T2708-1996. Technical specifications of the rolled solid wheels for railway.

[11] JIS E 5402–1:2015. Rolling Stock – Solid Wheel – Part 1: Quality Requirements, Japanese Standards Association.

[12] Markov D.P. Tribological aspects of increasing of wear resistance and rolling contact fatigue resistance of railway rolling stock wheels. In: Thesis for a Doctor's Degree, RRI, Moscow, 1996 (in Russian).

[13] Sakalo V.I., Sakalo A.V. Criteria for predicting the initiation of rolling contact fatigue damage in the railway wheels and rails. Vestnik of the Railway Research Institute. 2019;78(3):141-148. (In Russ.).

[14] Ekberg A. Rolling contact fatigue of railway wheels – computer modelling and in-field data. In: Proc. of the 2nd mini conf. Contact mechanics and wear of rail/wheel systems, Budapest; 1996. p. 154–163.

[15] Clayton P., Su X. Surface initiated fatigue of perlitic and bainitic steels under wear lubricated rolling/sliding contact. Wear 1996;200:63–73.

[16] Larin T.V., Devyatkin V.P., Krivosheev V.N., Naumov I.V., Chalykh E.I. Solid-rolled railway wheels. In: Transzheldorizdat, Moscow; 1956 (in Russian).

[17] Afferrante L., Ciavarella M., Demelio G. A re-examination of rolling contact fatigue experiments by Clayton and Su with suggestions for surface durability calculations. Wear 2004;256:329–334.

[18] Sakalo V., Sakalo A., Tomashevskiy S., Kerentcev D. Computer modelling of process of accumulation of rolling contact fatigue damage in railway wheels. International Journal of Fatigue 2018;111:7–15.

[19] Goryacheva I.G., Torskaya E.V. Modeling of Contact Fatigue Damage Accumulation in Material with Residual Stresses during Rolling Friction. Friction and Wear. 2019;40(1):44-51. (In Russ.).

[20] Bernasconi A., Davoli P., Filippini M., Foletti S. An integrated approach to rolling contact sub-surface fatigue assessment of railway wheels. Wear 2005;258:973–980.

[21] Kim T.-Y., Kim H.-K. Three-dimensional elastic-plastic finite element analysis for wheel-rail rolling contact fatigue. International Journal of Engineering and Technology 2014;6:1593–1600.

[22] Talamini B., Perlman B., Gordon J. Development of a standard for new passenger car wheel designs. In: Proc. of IMECE 2006, int. mech. engineering congress and exhibition, Chicago, Illinois, November 5–10; 2006.

[23] Malinin N.N. Applied theory of plasticity and creep. In: Mashinostroenie, Moscow; 1968 (in Russian).

[24] AAR Manual of Standards and Recommended Practices. Wheels and Axles. Analytic Evaluation Of Locomotive Wheel Designs. Standard S-669. Adopted: 2011.

[25] Feodosev V.I. Strength of materials. In: MSTU publishing, Moscow; 2018 (In Russian).

[26] Fridman Y.B. Mechanical properties of metals. Part 2. Mechanical test. Structural strength. In: Mashinostroenie, Moscow; 1972 (in Russian).

[27] Sakalo V., Sakalo A., Rodikov A., Tomashevskiy S. Computer modeling of processes of wear and accumulation of rolling contact fatigue damage in railway wheels using combined criterion. Wear 432–433 (2019) 102900. DOI: 10.1016/j.wear.2019.05.015.

[28] Serensen S.V., Kogaev V.P., Shneiderovich R.M. Bearing capacity and calculation of machine parts for strength. In: Mashinostroenie, Moscow; 1975 (in Russian).

[29] Markov D.P. Tribology and its application in railway transport. In: Intext, Moscow; 2007 (in Russian).

[30] Zienkiewicz O.C. The finite element method in engineering science. In: McGraw-Hill, London; 1971.

[31] Kik W., Piotrowski J. A fast, approximate method to calculate normal load at contact between wheel and rail and creep forces during rolling. In: Proc. of the 2nd mini conf. Contact mechanics and wear of rail/wheel systems, Budapest; 1996. p. 52–61.

[32] Kalker J.J. A Fast Algorithm for the Simplified Theory of Rolling Contact. Vehicle system dynamics 1982;11:1–13.

[33] Sakalo A.V. Method of simulation of contact stresses by using finite element fragments on elastic foundation. Herald of the VSTU 2009;9:71–76 (in Russian).

[34] Sakalo V.I., Sakalo A.V. Numerical method for solving the contact problems for the solid bodies using the finite element fragments on the elastic foundation. In: Proc. of the 1st joint int. conf. on multibody system dynamics, Lappeenranta; 2010. p.366–373.

[35] Sakalo A.V. Simulation of accumulation of contact fatigue damages in railway wheel by using finite element fragments on elastic foundation. Vestnik of the Railway Research Institute 2011;4:44–49 (in Russian).

[36] Sakalo A.V. Improvement of the railway wheel tread profile according to a contact fatigue criterion. In: PhD Thesis Moscow State University of Railway Engineering (MIIT), Moscow, 2011 (in Russian).

[37] Sakalo V.I., Kossov V.S. Contact problems of railway transport. In: Mashinostroenie, Moscow; 2004 (in Russian).

[38] Kerentcev D.E. Increasing the structural strength of railway wheels of rolling stock. In: PhD Thesis Samara State Aerospace University, Samara, Russian Federation; 2016 (in Russian).

[39] GOST 11018-2011. Interstate Standard. Traction railway stock wheelsets of 1520 mm gauge railways. General specifications.

[40] GOST R 51685-2013. National Standard of the Russian Federation. Railway rails. General specifications.