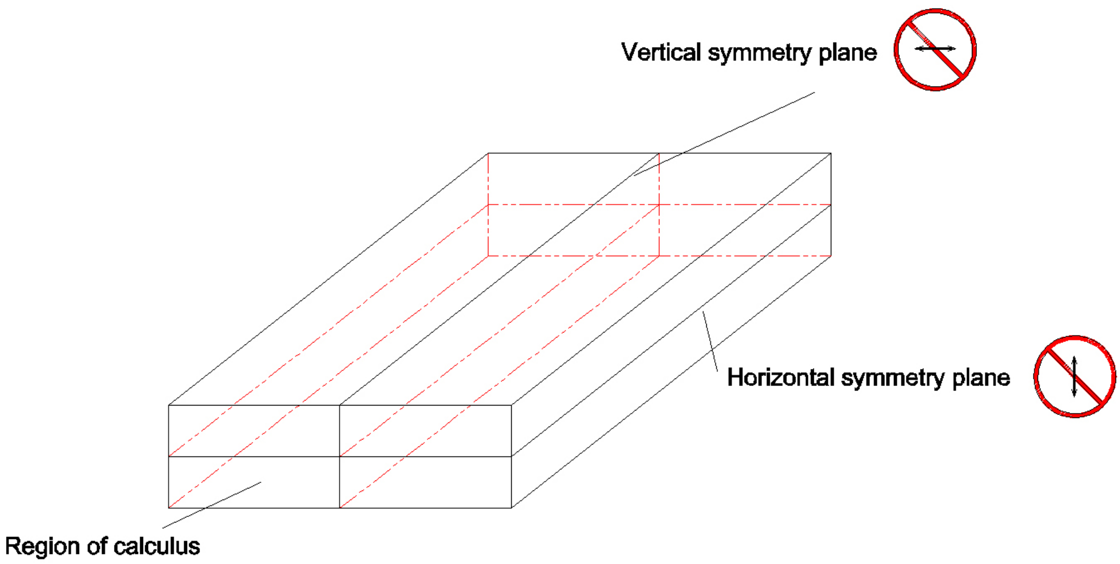
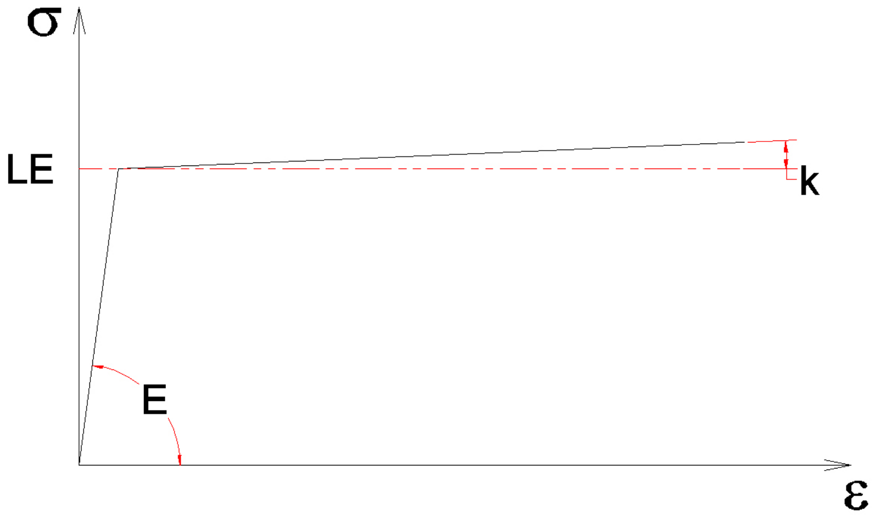
MATHEMATIC MODEL OF THE APPLICATION

This application divides a bar of rectangular section, and length equal to the contact arc, (calculated from the roll diameter and the reduction) in a finite element mesh. The elements are cubic, with nodes at the corners and at the middle of the edges. (picture 5). So, each element has 20 nodes. From the problem symmetry (pass flat-oval), it is only necessary to calculate the left bottom region. Nodes belonging to the vertical plane of the longitudinal section, splitting the bar in two halves, are then not allowed to move sideways. Nodes belonging to the horizontal plane of the longitudinal section passing through the center of the bar are not allowed to move up or down. (picture 1).



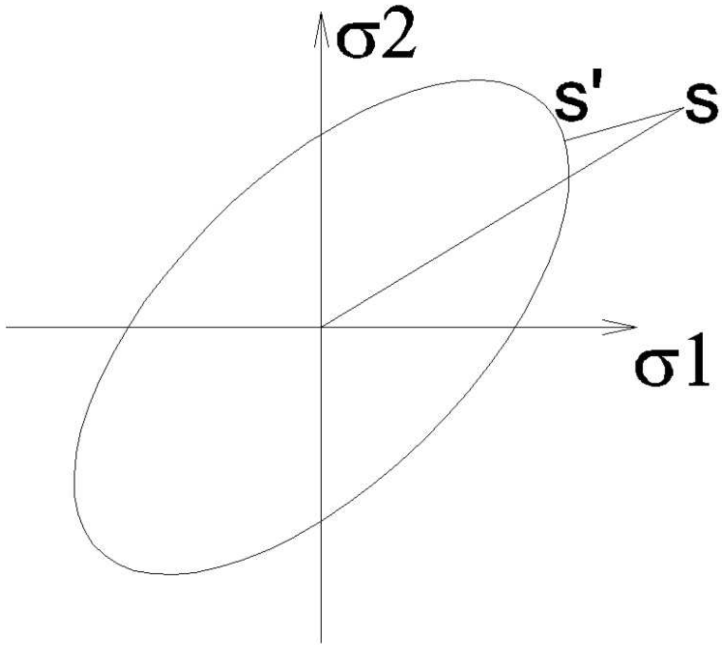
Picture 1

From the geometrical data of the flat bar and oval groove, the application calculates the nodal displacements for the first step, corresponding to a time interval just after the bite in the rolls. After knowing the new position of the nodes of that step, the next step is calculated and so on until the total reduction of thickness. The material parameters (Young modulus, hardening linear coefficient and yield point, besides the Poisson coefficient – Picture 2) are internal variables, but can be accessed and changed before execution, in the menu option *config*. Other parameters that can be changed in *config* are: friction coefficient, deformation step and the mesh configuration; (number of layers in the thickness, width and depth). The mathematical foundation of the finite element method in solids under elastic stress is detailed in the file feaDescription.pdf.



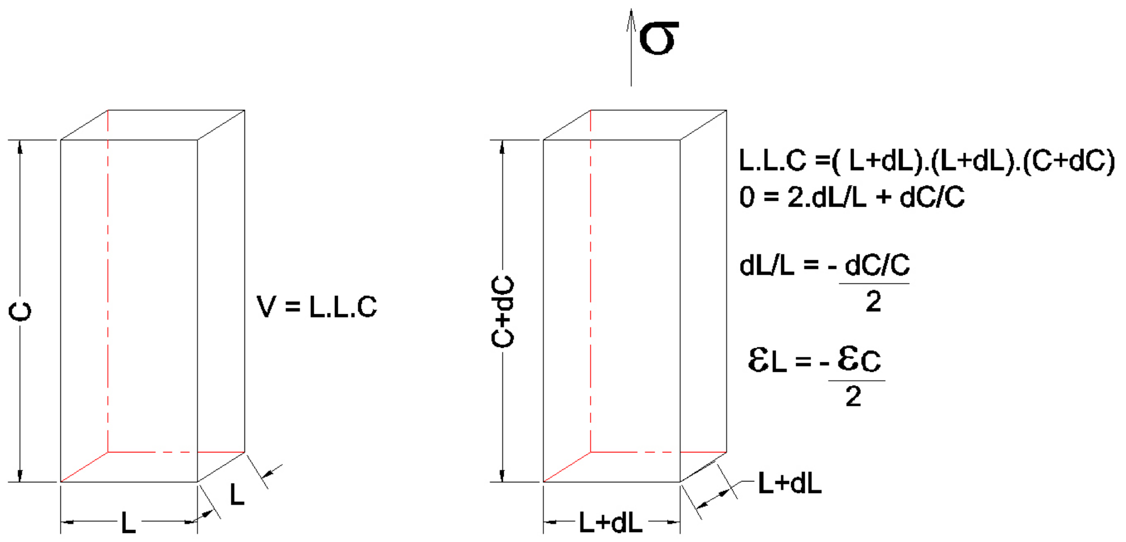
Picture 2

From the x, y and z displacements of each node, calculated initially supposing elastic behavior, the stresses are calculated inside each element. The components of the stress tensor are used as inputs to the Von Mises formula. If it is beyond the yield surface, the tensor is corrected by an algorithm, as shown the Picture 3 for the case of two dimensions. The components of the tensor corresponding to the point **S** are transformed to the point **S**'. Due to the strain hardening, the algorithm takes in account that after the deformation step, the yield point is a little bit greater.



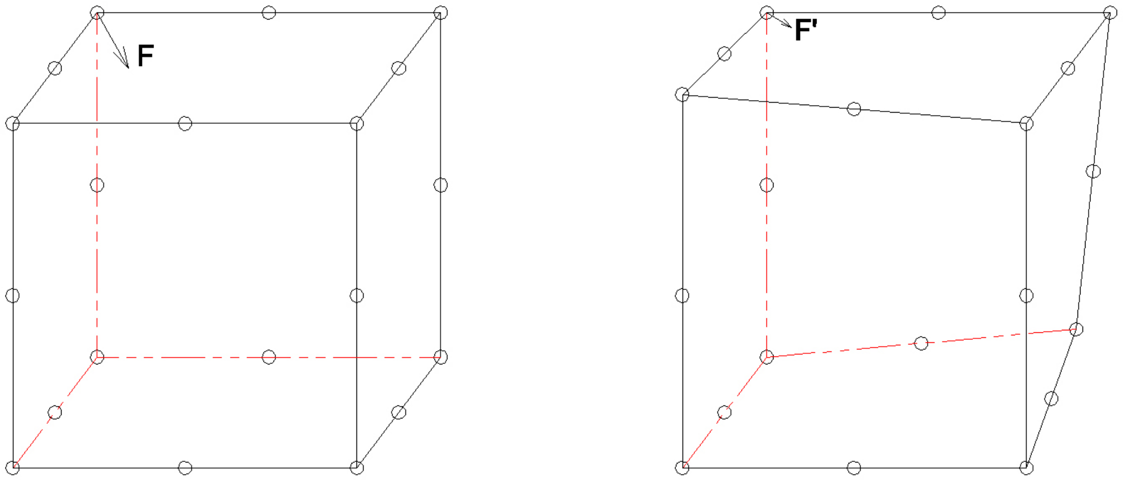
Picture 3

Other consideration to define that stress correction is the criteria, fair accurate for metals, of proportionality between the deviatoric stress tensor and the plastic strain tensor. The former is given from the subtraction of σxx, σyy e σzz by the average of that components. For the case of uniaxial tensile stress on a bar of squared section in the z direction, σxx e σyy are zero => $\boldsymbol{\sigma}_{\mathbf{dev}}$ = [-σzz/3, -σzz/3, 2. σzz/3, 0, 0, 0] = 1/3. σzz.[-1, -1, 2, 0, 0, 0]. Picture 4 shows how the components of the plastic strain tensor are proportional to $\boldsymbol{\sigma}_{\mathbf{dev}}$, because the strain component in the direction of the applied force is twice as big and has opposite sign of the side strains.



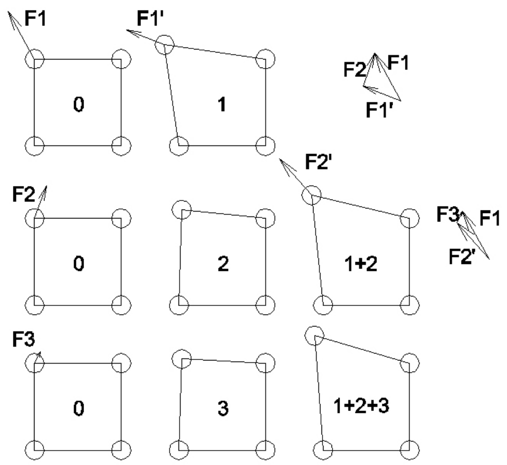
Picture 4

All nodal displacements have been calculated considering elastic behavior. In the finite element model, that required the emergence of nodal forces in the points that are forced to move with the rolls. On the other hand, internal elements had zero nodal forces. After the stresses are corrected to be on the Von Mises surface, the nodal forces that leads to that stresses are different from the original ones (Picture 5).



Picture 5

The application recalculates then the nodal displacements for elastic behavior, but now the nodal forces are the difference between the original and the recalculated ones. Due to the assumption of elastic behavior, the calculated displacements are always smaller than the actual ones for the applied forces. Each node has its displacement, for each coordinate direction, in that second turn, added to the previous one. From that new displacements, resulting from the sum of the first and second turn, the stresses are again calculated, and also the nodal forces that lead to them. (Picture 6 shows that sequence for 2 dimensions). Repeating the process several times, the difference between original and recalculated forces (residual force) becomes smaller and smaller. And the same for the displacement increment, to be added to the former ones. When that difference is smaller than a given value, we take the displacement as the actual one, and a new step of roll is calculated, until the final reduction.



On each step, the residual force applied to the nodes and the displacement increment become smaller, leading to the convergence of the calculus.

Picture 6