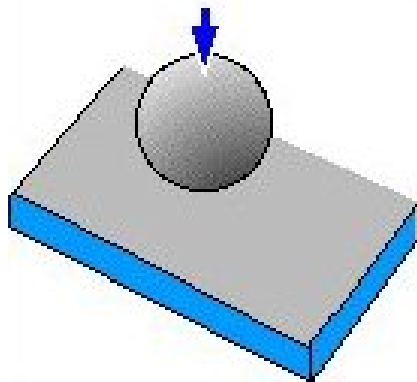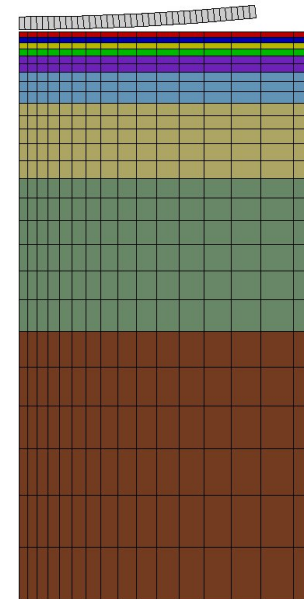# Study of a sphere–plate contact - Influence of the thickness of the coating

In automotive as well as aeronautical industries, coatings are generally applied on the surface of bodies in contact in order to protect them from different mechanical or chemical effects such as oxidation, corrosion, extreme temperatures, wear, etc. The mechanical behaviour of this coating can be evaluated by using indentation tests on the coated surface. This is the subject of this mini-project. The aim of the study is then the definition of the suitable conditions to avoid the bias in the characterisation of the characteristics of the coating. The main question is: what is the minimal thickness that allows the user to reach the intrinsic properties of the layer ?

A rigid sphere and a coated half space are considered. The coating behaviour is elasto-plastic with a nonlinear kinematic hardening, the material of the substrate is linear elastic. We propose an analysis of the maximum and residual stresses in the half space (Fig.1) during the indentation test, and the characterisation of the global load–displacement curves.

Fig 2: Axisymmetrical FE mesh of a sphere–plate contact model

Figure 2 shows the mesh available. Axisymmetric linear elements are used. A series of layers are predefined. They will be considered either as substrate or as coating for different analysis in order to build numerical models with different coating thicknesses.

Keywords: *Contact mechanics, coating, plasticity*

Fig.1: Illustration of the sphere–plate contact

Files available for FE computations

The files are given in a series of directories. The directory `0_substrate` corresponds to a contact between a sphere and a thick elastic cylinder: the properties of this cylinder will be kept later for the substrate. On the other side, the directory `9_coating` corresponds to a contact between a sphere and a thick plastic cylinder: the properties of this half space will be kept later for the coating. Between them, the eight directories `i_layer`, with $i = 1..8$ correspond to an increasing thickness of the coating layer (`1_layer` corresponds to 22.5 microns of the coating, `2_layer` - 45 microns, `3_layer` - 75 microns, `4_layer` - 105 microns, `5_layer` - 174 microns, `6_layer` - 300 microns, `7_layer` - 632 microns, `8_layer` - 1294 microns). The only file in each of these directories is the `calcul.inp` file, which contains:

– the definition of the analysis (after the line `****calcul`): the role of each section can be found in his name (`***resolution`, bc (the boundary conditions), `***contact`, `***material`, `***output`);

- The section `***resolution` is used to specify the algorithm. `**sequence 2` specifies that two loading sequences are prescribed – to load and unload. The evolution of the applied force during loading and unloading is linear; the full loading path is achieved after 2 seconds (`*time 2.`). The suggested number of increments is 20 in each sequence, nevertheless, `**automatic_time` will allow the code to dynamically optimize the increment size.

- In section `***bc`, the boundary conditions are defined. `**imposed_nodal_dof` allows to prescribe the displacement of node sets. The prescribed value is obtained by multiplying the base value by the value contained in the table. `**impose_nodal_reaction` allows to apply forces on each node or node set.

- The section `***equation` allows to specify multi-point constraints between nodes. The type `**mpc1` indicates that all the nodes in the group `load_plan` have a common displacement (here in direction `U2`).

- In section `***table`, the table that represents the loading path is defined. This table is used for the definition of the boundary conditions (in `***bc`).

- The contact conditions are given by `***contact`, with a subsection concerning the contact algorithm (`solver_method`) and a section concerning the definition of the contact zone (`**zone`). Zones are modeled by a node set acting as the impactor (slave) surface which may come in contact with a target (master) surface area, defined by a line in two-dimensional problems. The coefficient of friction is defined by `*friction`.

- In section `***material`, the materials are associated with element sets.

- Section `***output` is devoted to output management. The command `**extra contact` allows to store in the output files specific data relevant to the contact calculations, like contact stresses or relative slip between the bodies. The `**test` part specifies that an ASCII file named `calcul.test` will be written. It includes the resulting forces on a series of node sets, and the displacement of some nodes.

– the definition of a post-processing (after `****post_processing`), which will allow to draw some stress and strain components along a line in the half space, perpendicular to the surface, at the initial contact point between the two bodies.

The only difference between each directory is given by the way the materials are spatially distributed.

All the `calcul.inp` files have in common a mesh file and three material files, located in the directory `MatMesh`. In this directory, the file `mesh.geof` defines the nodes, the elements and the various groups used for applying the boundary conditions. The mesh can be displayed by means of the command

<p align="center" style="color:brown">Zmaster mesh.geof</p>

which allows, for example, to highlight the various layers. The files `substrate.mat` and `coating.mat` contain the mechanical behaviour of the two materials presented in the problem. The first one is elastic isotropic. The second one is plastic, with nonlinear kinematic hardening. A third material, `soft.mat` is used to simulate a soft spring, which is used to easily avoid rigid motion of the cylinder for a loading under prescribed force.

FE computations for different coating thicknesses

All the computations can be managed from the directory `Manage`. It contains:
– little shells that can be adapted according to users' wish to edit all the input files `calcul.inp` (`VI`), or all the `calcul.ut` files after the computations (`VIu`);
– a file to run all the computations (`RUN`);
– a file to run all the post-processings (`RUNpp`);
– three files to plot the results (`DRAW` to plot displacements from the `calcul.test` files; `DRAWsig` and `DRAWeto` to plot respectively the stresses and the total strain from the files `sig.txt` and `eto.txt` generated by the post-processing).

A series of guide lines are now given to the students.

1. Go to directory `Manage` and run all the computations by means of the command `RUN`.

2. Go to each directory (`0_substrate`, *i*`_layer` and `9_coating`) and run the command `Zmaster calcul` to check the computations. Draw the contour plots of the various stress components and of the total strain. Check the `calcul.test` files, to see what has been saved inside.

3. Go back to directory `Manage`. Check the file `DRAW` and adapt it to have relevant plots illustrating the differences between the various computations; run together the command `VI` and `VIu`: the aim is to adapt the output number in the postprocessing section, to draw the stress and strain profile at the proper time. Choose the relevant plots and draw the various figures by using `DRAWsig` and `DRAWeto`. Compute for each case the depth of the print remaining at zero load at the end of the test.

4. Plot the variation of the depth of the print *h* according to the thickness of the coating *e*.

5. This work can be made for various properties of the coating (change the constitutive equation in `coating.mat`).

Propose a synthesis of the computations according to two different points of view:

- what is the minimum thickness of the coating layer needed to get the true intrinsic properties of the coating by an indentation test ?

- what is the error made for a given thickness if one *forgets* that the layer thickness is too small.