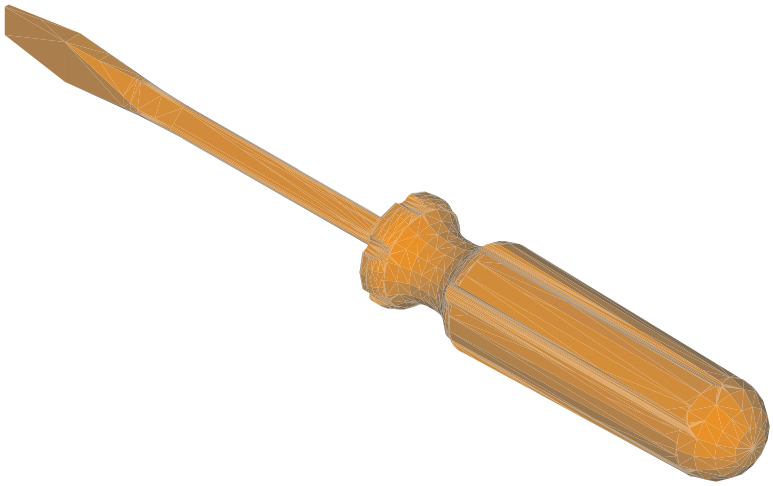
Problem description

It is desired to analyze the screwdriver shown in the following figure:



The geometry of the screwdriver is given in a Parasolid file.

The purpose of the analysis is to show how to modify the geometry using the discrete boundary representation feature (discrete BREP feature).

In this problem solution, we will demonstrate the following topics that have not been presented in previous problems:

- Importing the Parasolid geometry.
- Using the discrete boundary representation feature.

Before you begin

Please refer to the Icon Locator Tables chapter of the Primer for the locations of all of the AUI icons. Please refer to the Hints chapter of the Primer for useful hints.

Note that you must have an ADINA-M license to do this problem.

This problem cannot be solved with the 900 nodes version of the ADINA System because the 900 nodes version of the ADINA System does not include ADINA-M.

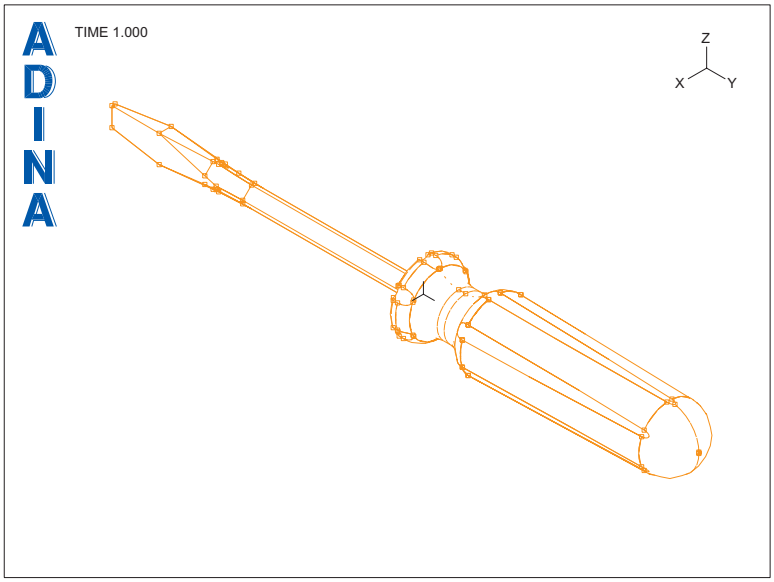
Much of the input for this problem is stored in files prob41.x_t and prob41_1.in. You need to copy files prob41.x_t and prob41_1.in from the folder samples\primer into a working directory or folder before beginning this analysis.

Invoking the AUI and choosing the finite element program

Invoke the AUI and choose ADINA Structures from the Program Module drop-down list.

Importing the Parasolid geometry

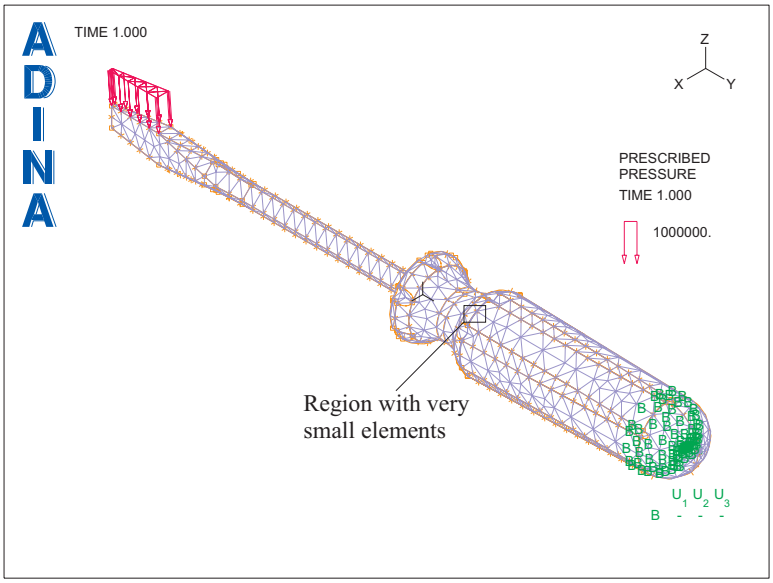
File prob41.x_t is a Parasolid file containing the geometry. Click the Import Parasolid Model icon  , choose file prob41.x_t and click Open. The graphics window should look something like this:



Defining the subdivision data, material, boundary conditions, loading and elements

We have prepared a batch file (prob41_1.in) which contains the rest of the model definition.

Choose File→Open Batch, navigate to the working directory or folder, select the file prob41_1.in and click Open. The graphics window should look something like the figure on the next page.



Zoom into the indicated region of the mesh. You will notice some very small elements. These elements are present because the geometry contains very narrow faces in this region of the model.

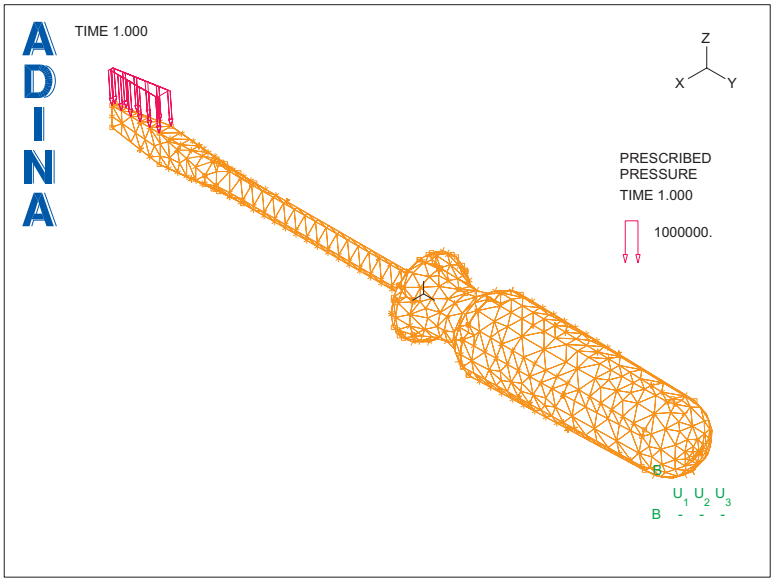
We would like to remesh, using a geometry without the narrow faces. Click the Delete Mesh icon  , set 'Delete Mesh from' to Body, enter 1 in the table and click OK.

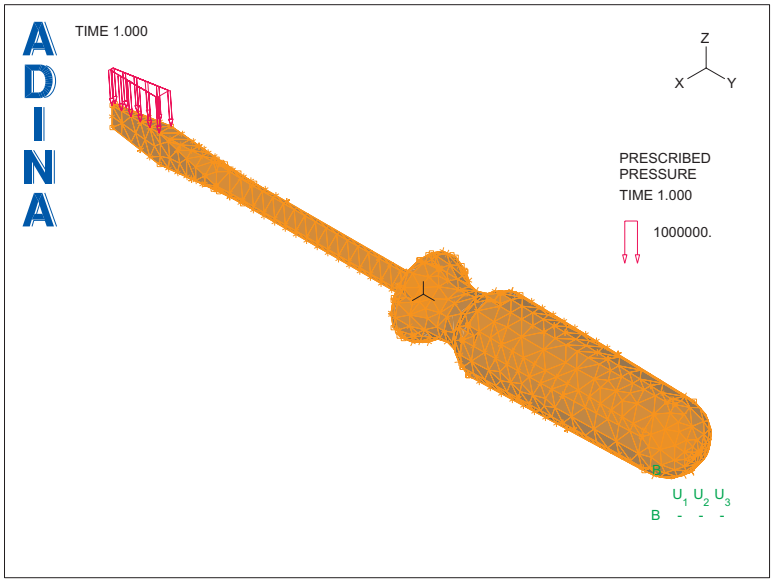
Creating a discrete boundary representation

Choose Meshing→Feature Removal→Discrete BREP, set the Body Number to 1 and click Create. Click Close to close the dialog box. The graphics window should look something like the top figure on the next page.

The orange lines are the boundaries of the discrete BREP triangles.

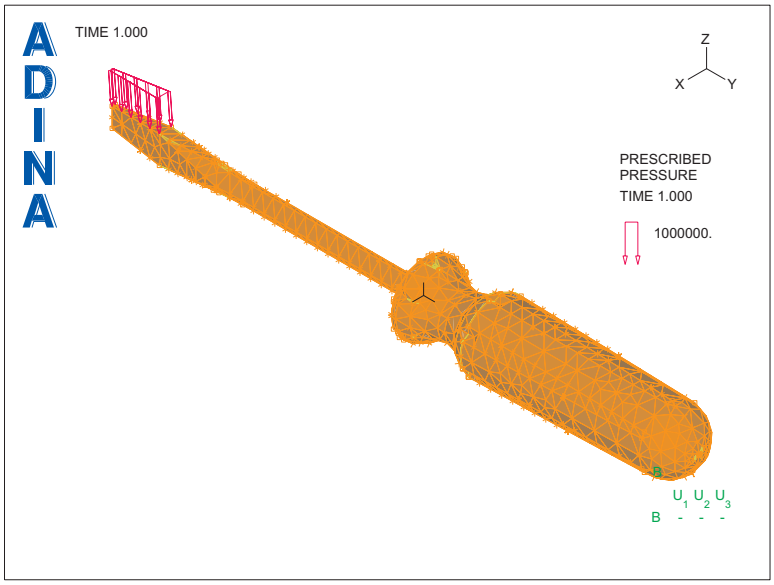
Click the Show Discrete BREP icon  to return to the original geometry plot, then click the Show Discrete BREP icon  again to show the discrete BREP. Click the Shading icon  . The graphics window should look something like the bottom figure on the next page.


ADINA
TIME 1.000
Z
X
Y
PRESCRIBED
PRESSURE
TIME 1.000
1000000.
U1 U2 U3
B - - -


ADINA
TIME 1.000
Z
X
Y
PRESCRIBED
PRESSURE
TIME 1.000
1000000.
U1 U2 U3
B - - -

Defeaturing the body

Click Meshing→Feature Removal→Body Defeature, set the Body Number to 1, set 'Remove Surface Triangles with Size below' to 0.001, set the 'Angle (in degrees) used in Coarsening' to 60 and click Preview. The graphics window should look something like this. The faces and edges that will be removed by the defeaturing are drawn in yellow.



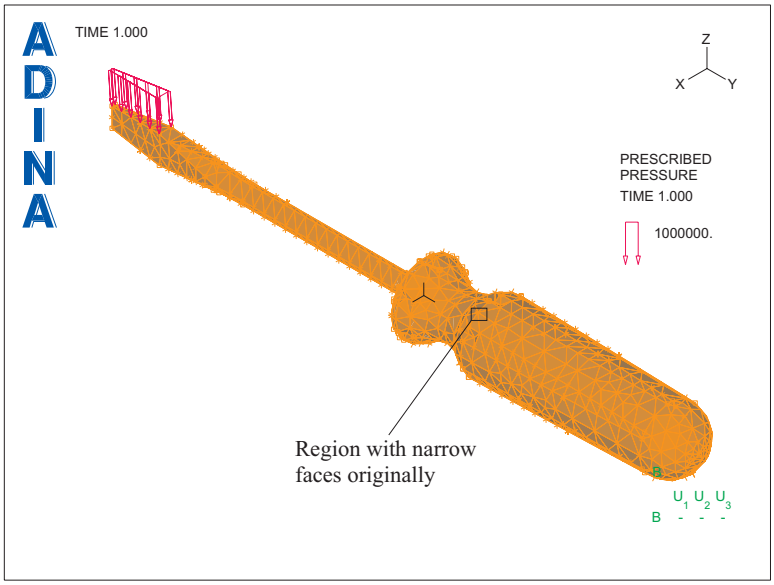
To defeature the body, click OK. The graphics window should look something like the top figure on the next page.

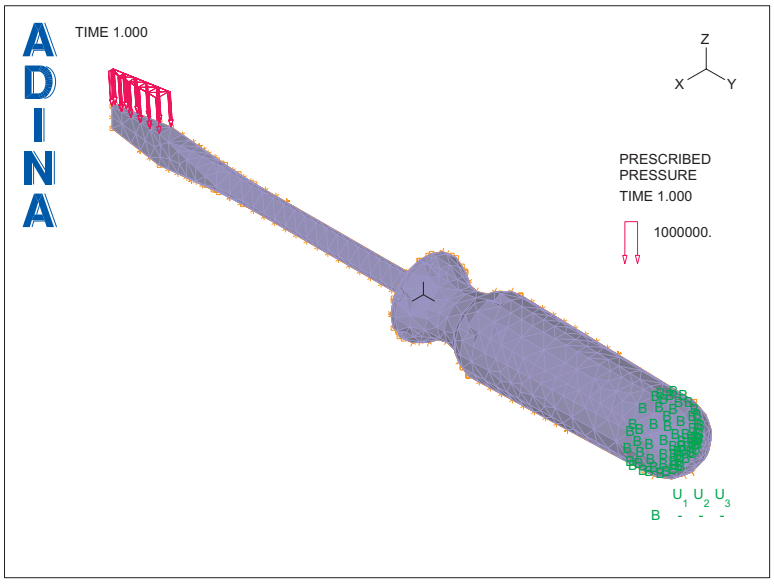
When you zoom into the indicated region, there are no narrow faces in the discrete BREP.

Remeshing

Click the Mesh Bodies icon  , set the Boundary Meshing to Delaunay, set the Nodes per Element to 4, enter 1 in the table and click OK. The graphics window should look something like the bottom figure on the next page.

Notice that the element faces on the surface match the discrete BREP of the body.


ADINA
TIME 1.000
Z
X
Y
PRESCRIBED
PRESSURE
TIME 1.000
1000000.
Region with narrow
faces originally
U1 U2 U3
B - - -

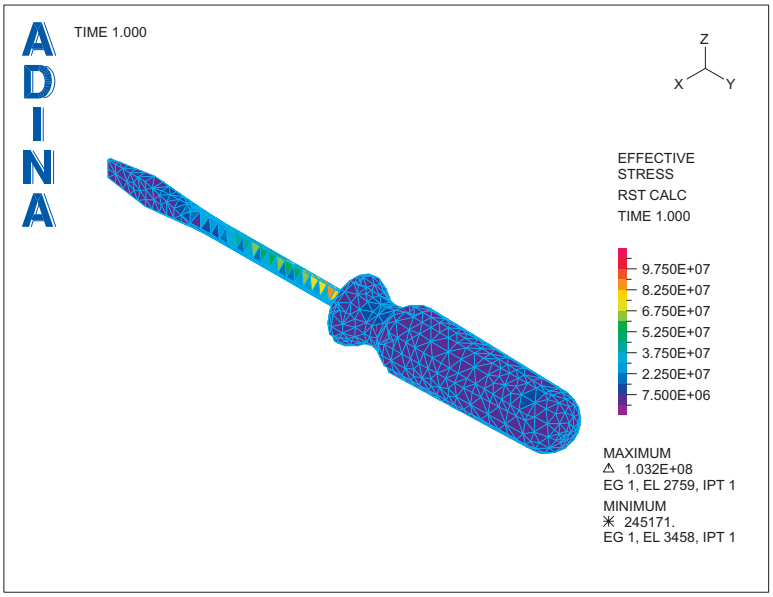

ADINA
TIME 1.000
Z
X
Y
PRESCRIBED
PRESSURE
TIME 1.000
1000000.
U1 U2 U3
B - - -

Generating the data file, running ADINA, loading the porthole file

Click the Save icon and save the database to file prob41. Click the Data File/Solution icon, set the file name to prob41, make sure that the Run Solution button is checked and click Save. When ADINA is finished, close all open dialog boxes, choose Post-Processing from the Program Module drop-down list (you can discard all changes), click the Open icon and open porthole file prob41.

Plotting the effective stresses

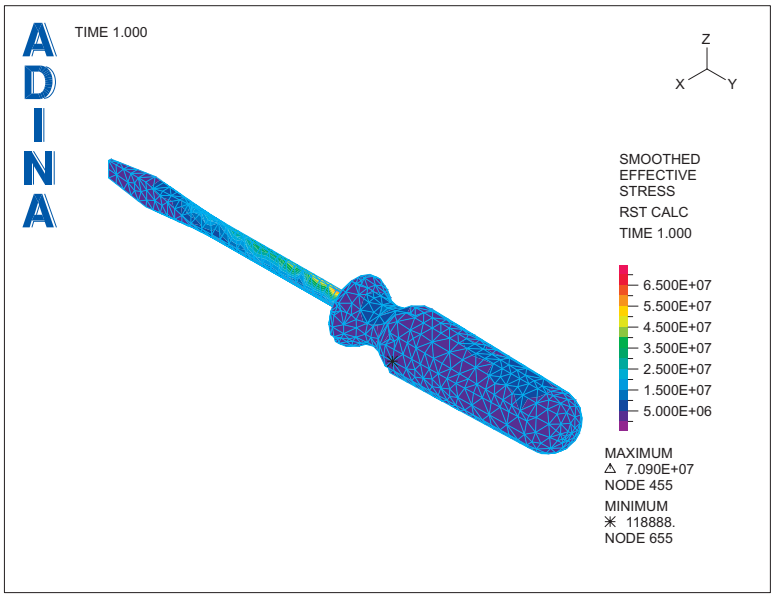
Click the Quick Band Plot icon. The graphics window should look something like this:



Since 4-node tet elements are used in the meshing, the stresses are constant within the elements. Click the Smooth Plots icon. The graphics window should look something like the figure on the next page.

Although the results look better, they are not more accurate. A refined model should be used to verify the results.

Exiting the AUI: Choose File→Exit (you can discard all changes).


ADINA
TIME 1.000
Z
X
Y
SMOOTHED
EFFECTIVE
STRESS
RST CALC
TIME 1.000
6.500E+07
5.500E+07
4.500E+07
3.500E+07
2.500E+07
1.500E+07
5.000E+06
MAXIMUM
△ 7.090E+07
NODE 455
MINIMUM
✳ 118888.
NODE 655