An example of panel solution in the elastic-plastic regime

Piotr Mika

May, 2014

1. Example – solution of the panel with ABAQUS program

The purpose is to analyze the elastic-plastic panel. The elastic solution of this panel is described in detail in the manual "Getting Started with ABAQUS and example solutions of panel". The dimensions and material data are given below, in Figure 1.

We are going to modify the previously prepared elastic model.

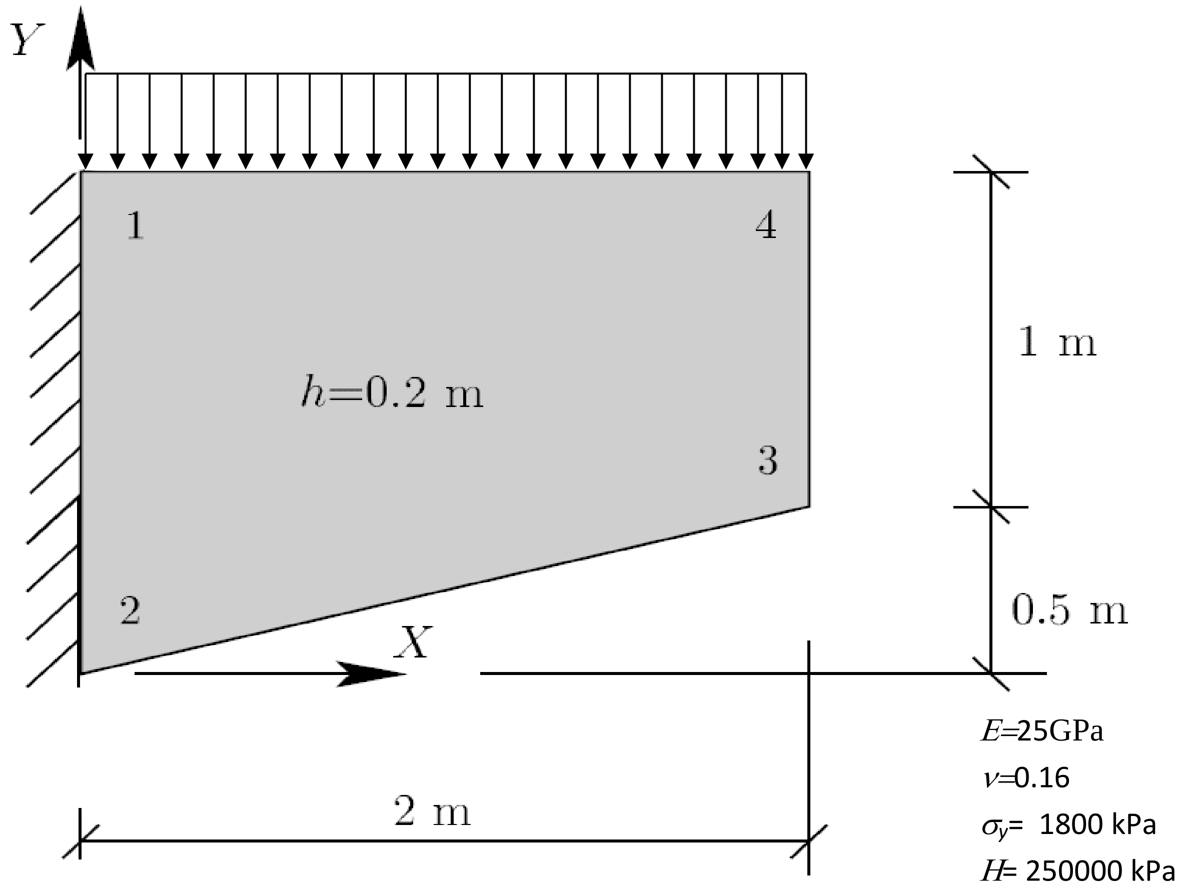
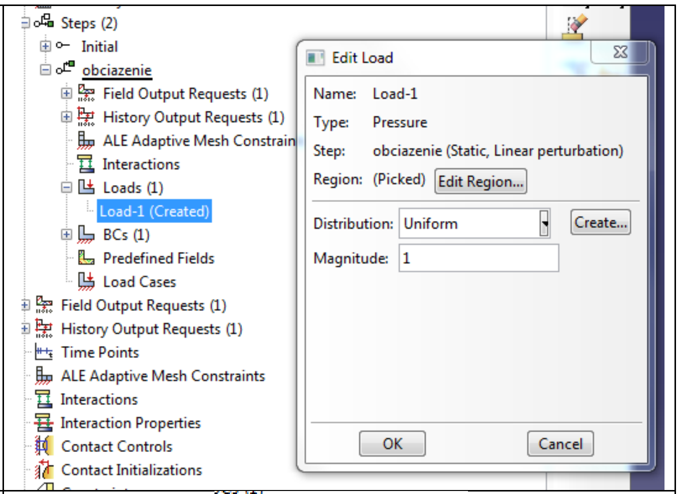


Fig 1 Panel geometry and material constants

1.1. Preprocessing

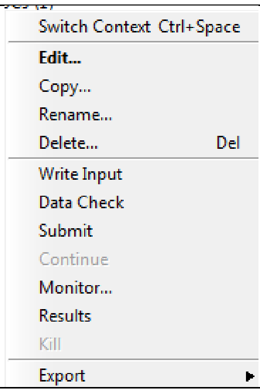
The steps leading to the calculation in ABAQUS program are described in the table on the following pages.

In order to determine the load at yielding initiation, we start with the calculation of the elastic regime with the unit load.
The load can be changed by developing *Loads* option in the first step (*Step-1*) and clicking on the name of the load (*load-1*). From the menu select *Edit* which opens a window where the value can be changed.



At the bottom of the model tree, click on Jobs, indicate the name of the task and run the calculations (click *Submit*).

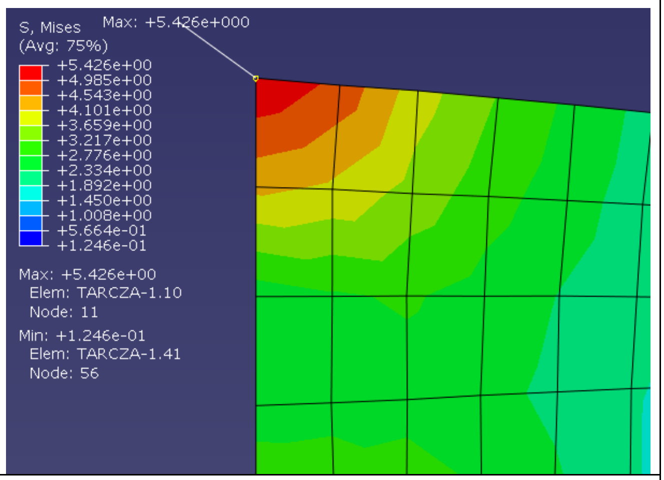
After switching to the *Visualization module* von Mises stress can be displayed.



Reading the maximum Mises stress values obtained by assuming the unit load allows for determining the value of the load that causes yielding of the material.
In our case (HMH yield criterion) 1800/5.426 ≅ 331.74 - exceeding that value causes yielding of the material.

TIP:

By clicking on the icon  and selecting the *Limits* tab, the location of the extremal value of the displayed variable is depicted.

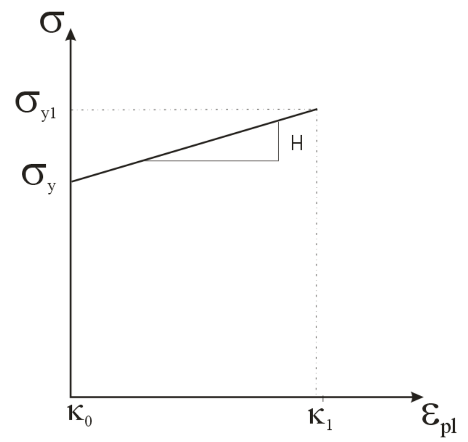


Further calculations will be carried out in two steps:
- Elastic, assuming the load, that generates the stresses close to yielding state and
- Plastic - assuming a much greater load

ASSUMING THE DATA FOR PLASTIC HARDENING

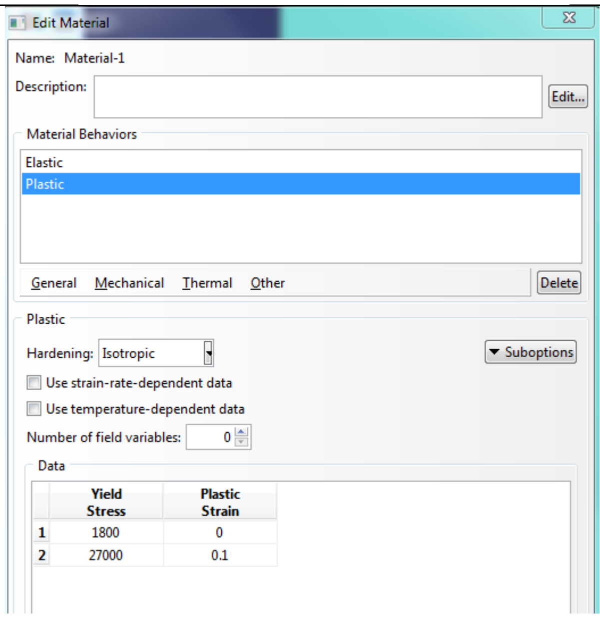
The formula $\sigma_{y1}= \sigma_y+\kappa_1*H$ is used, where H is accepted about $0.01*E$, and κ_1 about 0.1.

Final material data applied in calculations are given in the following figure.



DEFINITION OF MATERIAL - MODIFICATION

In Menu Tree / Materials click the "plus". Select the name of our material and click the *Edit* tab in *Mechanical/Plasticity/*Plastic. Assume: *Yield Stress* = 1800, *Plastic Strain* = 0 (a measure of plastic deformation) and add an extra line (by pressing enter). Then give the value of 27 000 for *Yield Stress* and 0.1 for *Plastic Strain*.

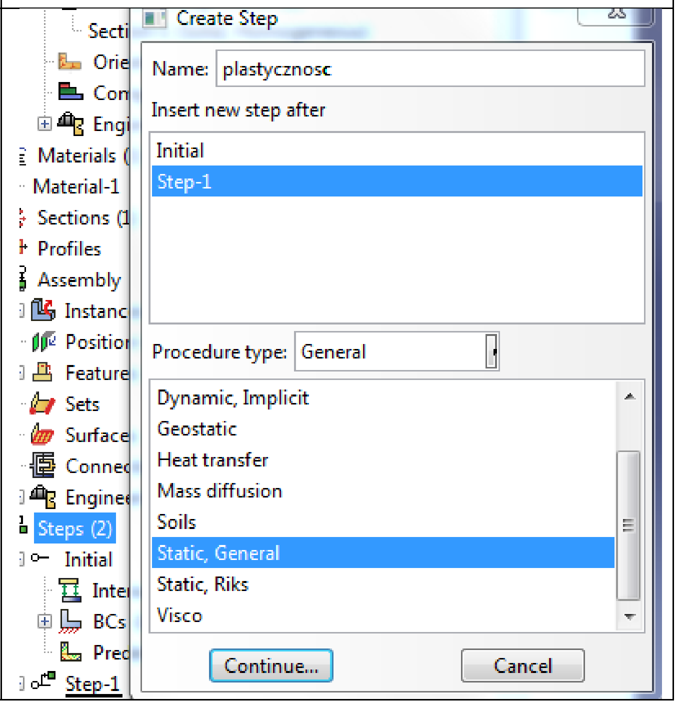


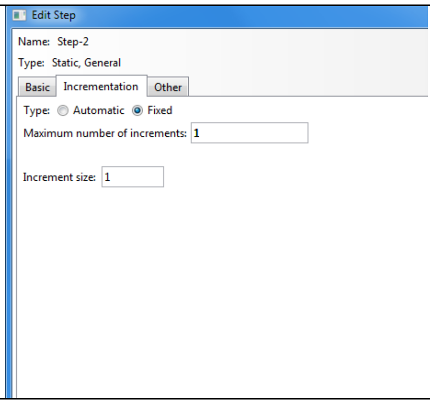
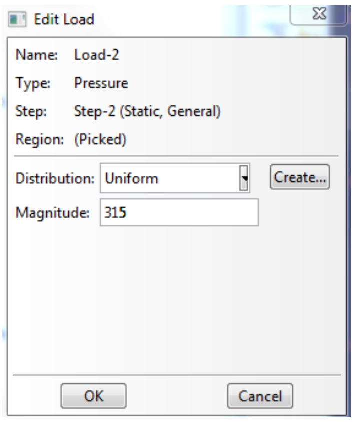
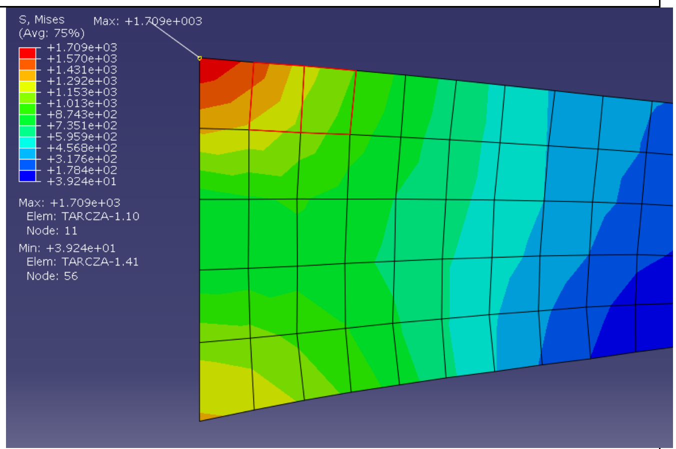
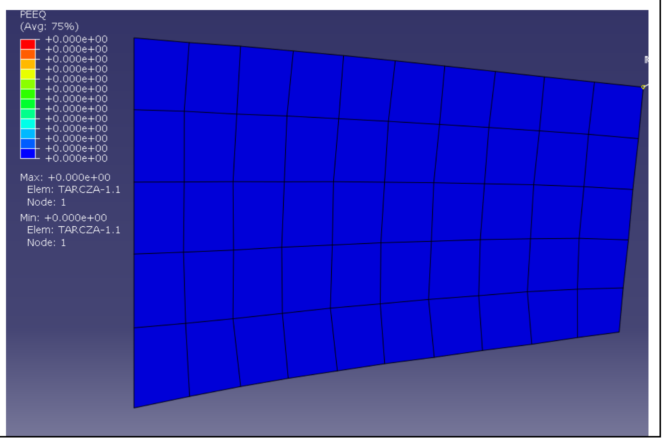
DEFINING CALCULATION STEPS

To run the non-linear analysis, we need to create another calculation step.

The current step (*Step-1*) *Linear perturbation/Static, Linear Perturbation* allows to determine the solution assuming linear-elastic material. This solution can be used to estimate the level of load that causes plasticity.

Double-click on the *Step - Create Step, Procedure Type: General, the type of analysis: Static, General*.

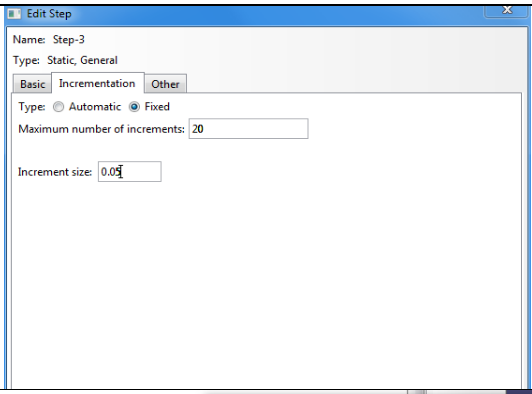


The *incrementation* card allows for manual specification of plastic step increment. In this case, as calculations are to stay the elastic range, one load increment without iteration is assumed.	
LOAD Develop step *Step-2*, click the *Loads*. In *create load* window select step, in which the load is to be applied (*Step-2*), category *Mechanical*, type *Pressure* and select *Continue*. Then choose the edge that will be loaded, click *Done* and enter the value 315 KN/m (slightly less than the value calculated from the ratio)	
When you run a task and go to the results, display Mises stress (maximum value is lower than the yield stress) Zero values equivalent plastic strain, marked in the ABAQUS as *PEEQ* prove that there is no yielding.	 

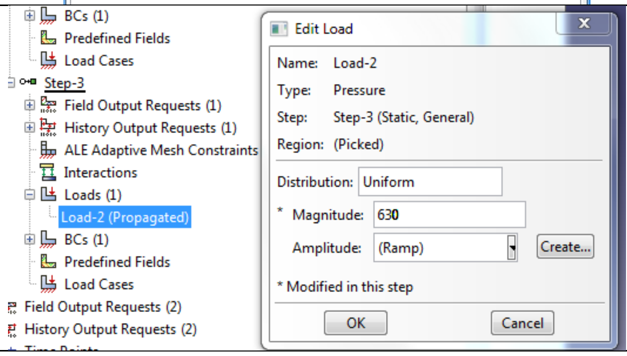
PLASTIC STEP

To create another step of calculation: double click on the *Step - Create Step, Procedure Type: General, the type of analysis: Static, General*.

In *incrementation tab*, we assume that the load is applied in 20 steps of 0.05 s



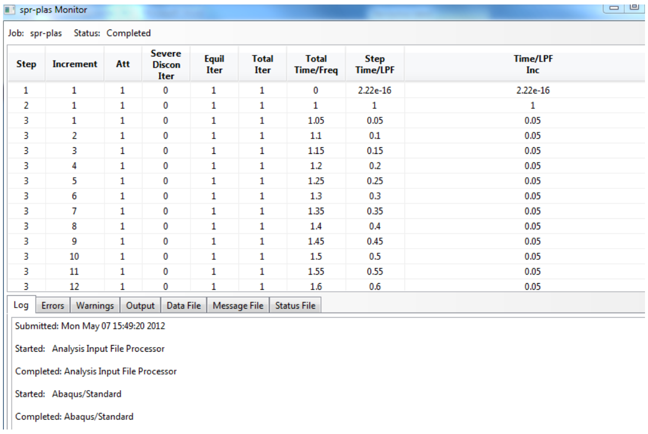
By selecting *step-3* it is noted that the assumed in the Step-2 load has been moved there.
Increase it twice to 630 KN/m



COMPUTING

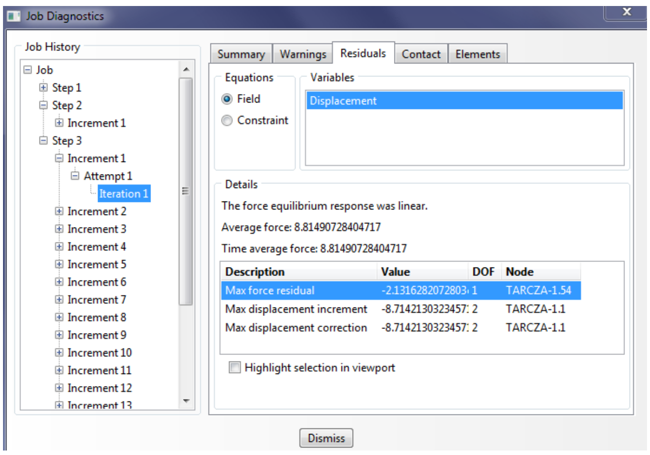
Using the menu *Jobs*, run calculations (click *Submit*).

Running the *Monitor* option in the manager of calculations, we can track the number of iterations in each increment within a calculation step.
The first column indicates the step number - in this case, we have three steps, and the second column gives the number of increments. Column 6 *Total Iter* gives the number of iterations needed to achieve a balance in each of the increments. Last but one column gives the total time, while the last one time increment.



CONVERGENCE OF ITERATION

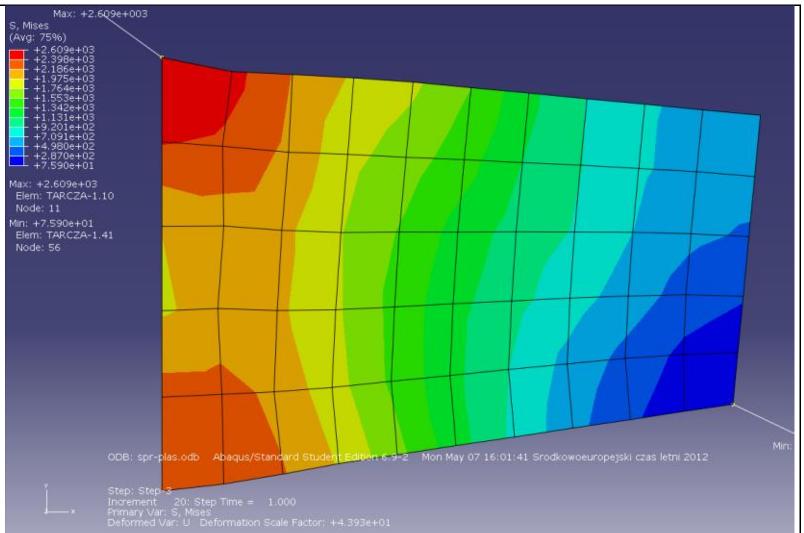
If the calculations are completed, the *Visualization module* and the *Job Diagnostic* from *Tools* menu can be started.
When you select *Attempt* in the *Summary* tab is selected, basic information about the number of iterations is obtained. In the model tree on the left side of the window, go to the iteration. *Summary* tab is used to check whether the iteration process is convergent, and if not, the reason why the iteration does not converge can be read from the *residuals* card. The max residual force, the increase in displacement and the



<table>
<tr><td>max displacement correction factor are given there.
Where these values are achieved in the modeled structure can be seen by marking the box Highlight selection in the viewport.</td><td></td></tr>
</table>

Control results

<table>
<tr><td>To check the results display the contour maps of Mises stress values, (menu Plot/Contours; the selection of variables and step is possible in - Results/Field Output).
The figure descriptions indicate the step and the number of the increment.</td><td>

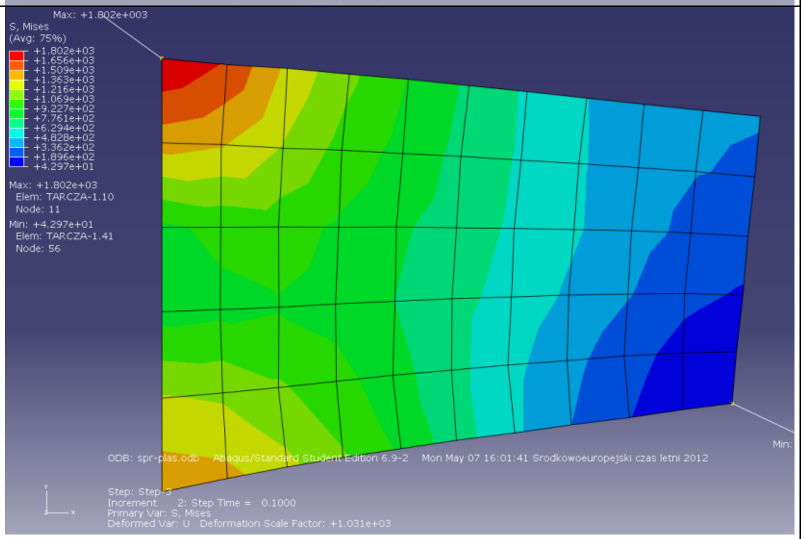


</td></tr>
<tr><td>By using the icons:

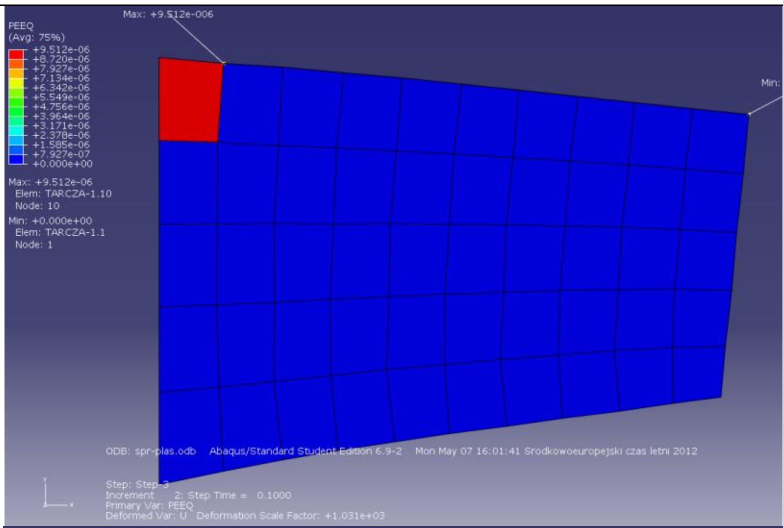
the results in different increments of time can be analyzed.

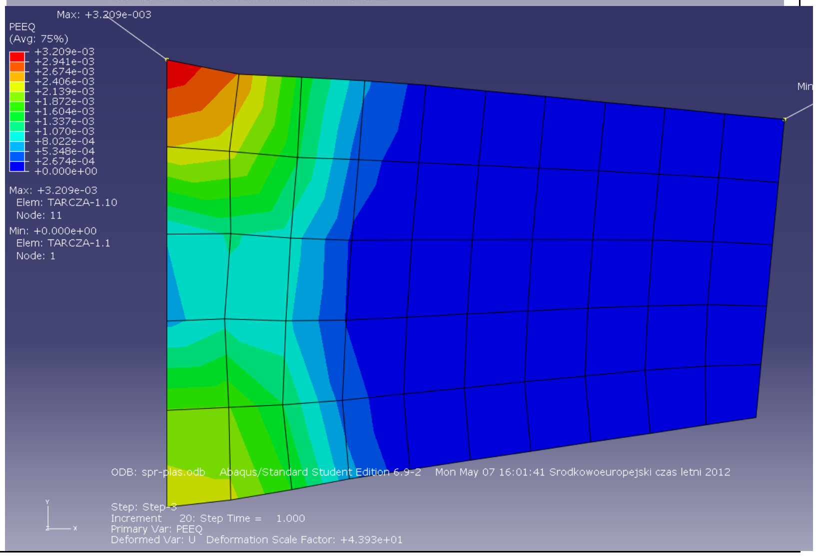
The yield strength is reached when a non-zero value of plastic deformation, marked in the ABAQUS as PEEQ symbol, is reached.

The next figure shows PEEQ achieved in the last increment (no. 20 - the current number is given in the description below the picture)</td><td>



</td></tr>
</table>


Max: +9.512e-006
PEEQ
(Avg: 75%)
+9.512e-06
+8.720e-06
+7.927e-06
+7.134e-06
+6.342e-06
+5.549e-06
+4.756e-06
+3.964e-06
+3.171e-06
+2.378e-06
+1.585e-06
+7.927e-07
+0.000e+00
Max: +9.512e-06
Elem: TARCZA-1.10
Node: 10
Min: +0.000e+00
Elem: TARCZA-1.1
Node: 1
Min:
ODB: spr-plas.odb Abaqus/Standard Student Edition 6.9-2 Mon May 07 16:01:41 Srodkowoeuropejski czas letni 2012
Step: Step-3
Increment 2: Step Time = 0.1000
Primary Var: PEEQ
Deformed Var: U Deformation Scale Factor: +1.031e+03


Max: +3.209e-003
PEEQ
(Avg: 75%)
+3.209e-03
+2.941e-03
+2.674e-03
+2.406e-03
+2.139e-03
+1.872e-03
+1.604e-03
+1.337e-03
+1.070e-03
+8.022e-04
+5.348e-04
+2.674e-04
+0.000e+00
Max: +3.209e-03
Elem: TARCZA-1.10
Node: 11
Min: +0.000e+00
Elem: TARCZA-1.1
Node: 1
ODB: spr-plas.odb Abaqus/Standard Student Edition 6.9-2 Mon May 07 16:01:41 Srodkowoeuropejski czas letni 2012
Step: Step-3
Increment 20: Step Time = 1.000
Primary Var: PEEQ
Deformed Var: U Deformation Scale Factor: +4.393e+01