MASTER'S DEGREE IN
NUMERICAL METHODS IN
ENGINEERING

UNIVERSITAT POLITÈCNICA
DE CATALUNYA
BARCELONATECH

COMPUTATIONAL MECHANICAL TOOLS

Assignment 4: Nonlinearity using Abaqus

Authors:
Ivan Pérez

Professor:
Natividad Pastor

Academic Year 2019-2020

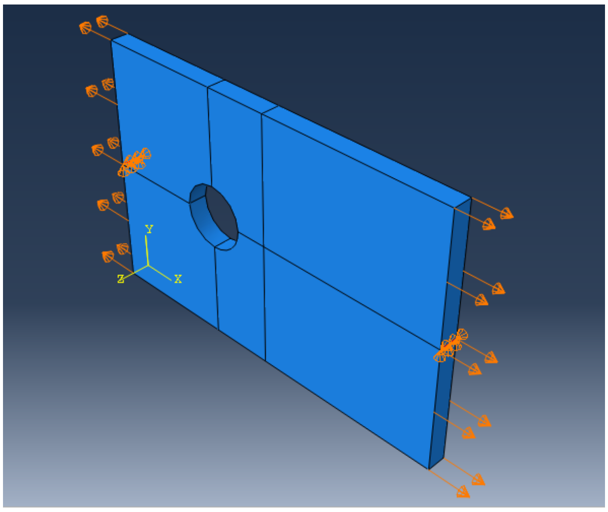
Contents

1 Introduction

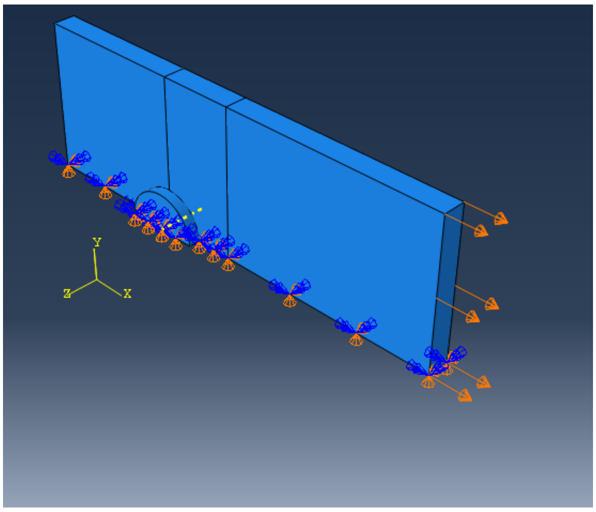
In this assignment it is required to analyse the stresses on a steel plate with a hole. To do so, it is used structural analysis CAE software called Abaqus. This plate will be simulated in 2 different environments: firstly is only involved the plate itself and a secondly a fixed pin has been inserted on the hole of the plate.

Furthermore, to understand how linearity and non-linearity of the materials affects to the inner stresses of the plate and the pin, it is needed simulate several cases by changing the material's properties. In the beginning, it is simulated both scenarios with perfectly elastic steel and in the upcoming cases they are simulated by introducing non-linearity of a more realistic steel material's curve.

Nevertheless it has to be noted that it is not possible to simulate the failure realistically with the used Abaqus solver. The plasticity of the material's curve has been introduced up to the yielding zone. For further stresses, the plate would yield endlessly without fracture.
It has been used Abaqus CAE softare to prepare the models (pre-processing) and to analyse the results (post-processing).



Case 1: Plate



Case 2: Plate with Pin (symmetry)

In the next 2 sections, firstly it is shown some glimpses of how the model is set up,which boundary conditions are introduced and how is the mesh before submit to calculate.

Finally the obtained results for elastic material (linear) and plastic materials (non-linear) are discussed.

2 Plate

Here it is explained in more detail the study of the plate with a hole for elastic and plastic steel.

2.1 Model definition

Several aspects for the preparation of the model to be simulated are described below.

Geometry definition

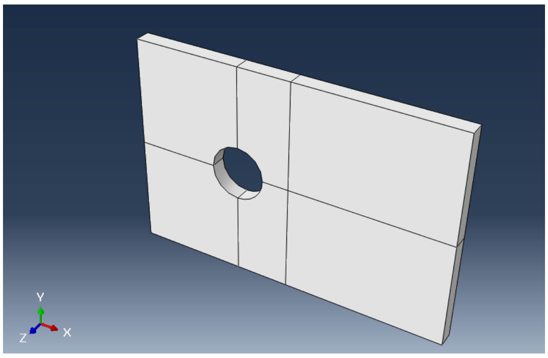
The plate dimensions are defined as lenght(X-direction), height (Y-direction) and thickness (Z-direction): 30mm, 20mm and 1.5mm respectively. And a hole of 5mm of diameter and which it center has been placed from 10mm of the left edge and from 10mm of the lower edge.

Boundary Conditions

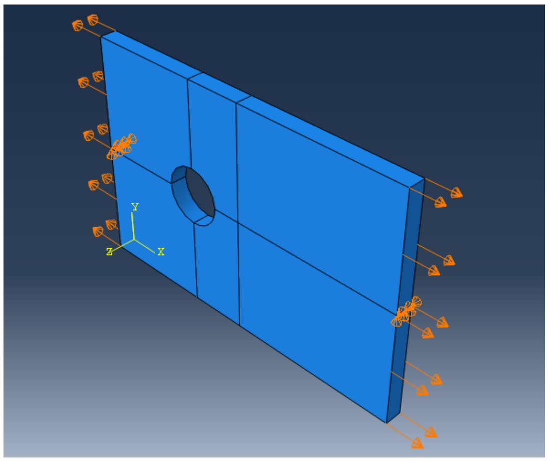
The degrees of freedom of the plate have been constrained to match the nature of the problem where the shape is stretched in X-direction.

- In each **leading-edge planes** of the plate have prescribed displacement of **+/-0.05mm in X-direction**.
- The **middle horizontal** line in the leading-edges of the plate are forced to move only in the X-direction by **not allowing displacements in Y or Z direction**.

In the below image can be seen the both constrains:



Geometry partitioned



Boudnary conditions

Meshing geometry

The geometry has been partitioned in order to get a good quality of structured mesh. Seeding on edges has been also modified to achieve the mesh density of cells equally in all the areas. Element type used is *C3D8R* in the plate.

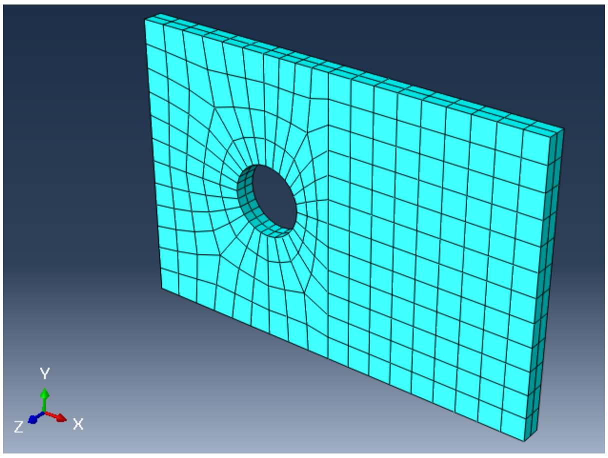


Figure 1: Mesh

2.2 Results

It has been simulated 4 different cases by changing the plastic-elastic characterization curve of the material. For all the cases the Young modulus of the material is the Steel young's modulus and is considered isotropic material.

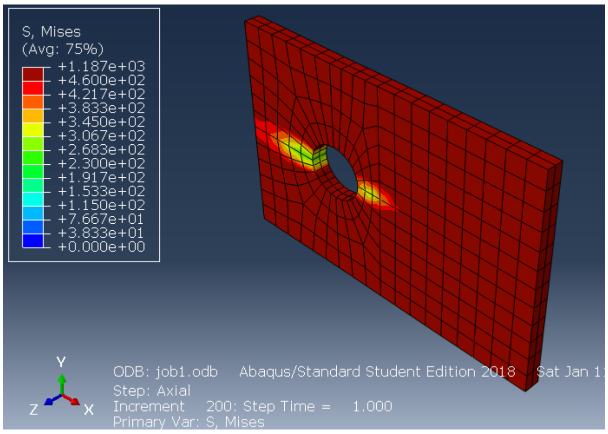
- **Case 1:** The Steel is modelled as infinitely elastic. Only the slope relation between Stres-strain curve is requiered.
- **Case A:** The steel is modelled with its Yield Stress point (460 MPa). After this point the material yields perfectly. The plastic stress-strain region is flat.
- **Case B:** A second point for plastic region is required. The yielding is not flat any more and is the one corresponding to the slope between these two points. After the second yielding point, the strain-stress curve becomes flat allowing perfect yielding.
- **Case C:** Analogous to the "Case B", but here the second yielding point has been moved from from displacement of 5e-3mm to a 2e-3mm meaning that "Case C" is less ductile than "Case B".

Find in the below table a summary of all simulated cases:

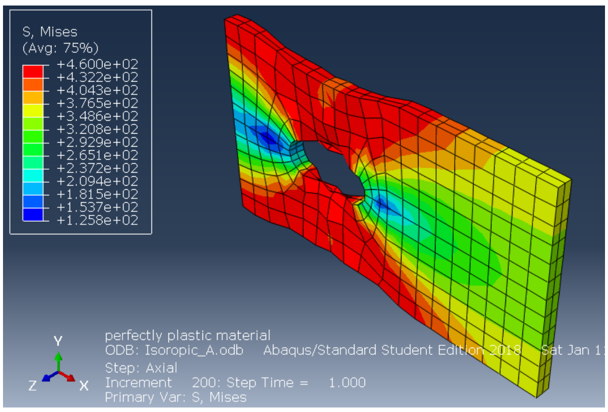
Case	**Material**	**Young modulus** $[N/mm^2]$	**Yield Stress 1** $[N/mm^2]$, $[mm]$	**Yield Stress 2** $[N/mm^2]$, $[mm]$
1	Perfectly Elastic	2.1e5	-	-
A	Perfectly plastic	2.1e5	460, 0	-
B	Plastic	2.1e5	460, 0	520, 5e-3
C	Plastic	2.1e5	460, 0	520, 2e-3

Stresses

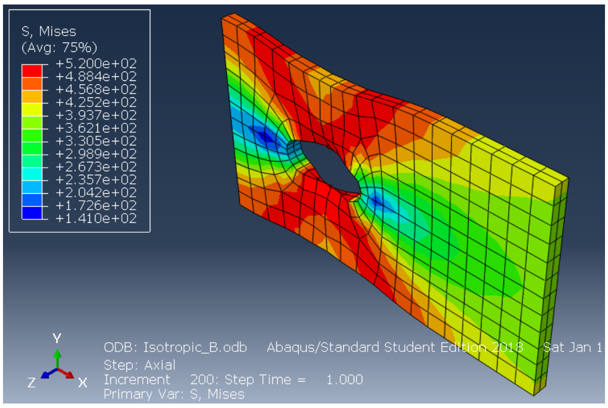
Find in the images below the stresses results of each case with the same scale.



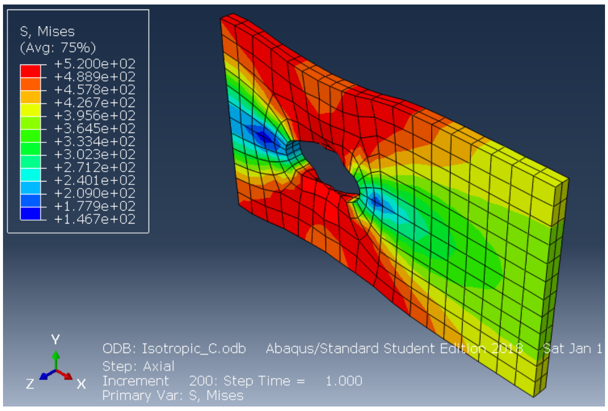
Case 1 Stresses



Case A Stresses



Case B Stresses



Case C Stresses

Force-Displacement of Node

It has been monitored the Force and displacement on X-direction in a node of the plate positioned in the mid plane and in the right leading-edge.

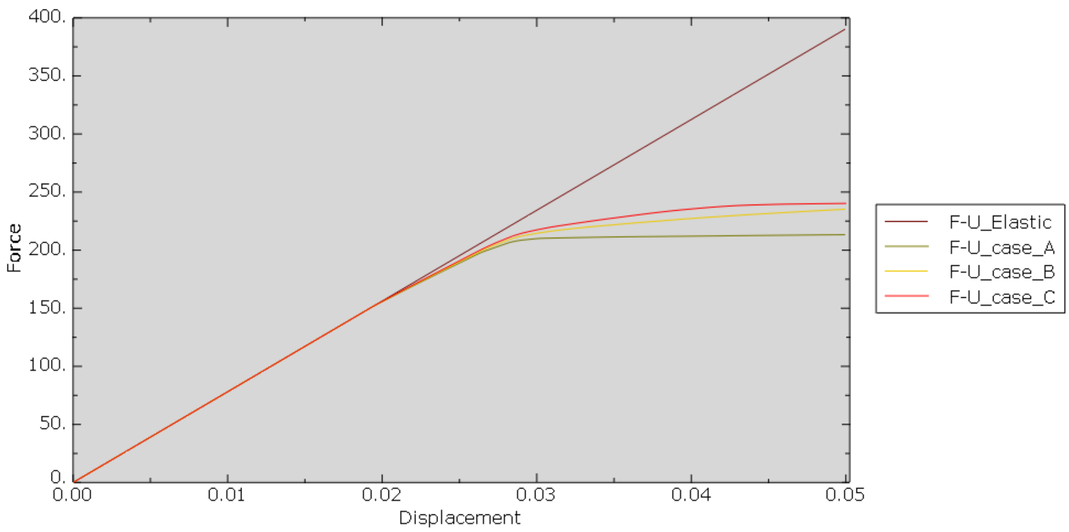


Figure 2: F-U curve of the cases

The first case is completely elastic and the stresses are very high since the material is not allowed to yield. The energy is completely stored as elastic deformation.

On the next cases there is considered plastic plate so the energy of the deformation cannot reversibly be stored and the material suffer permanent deformation when stresses in the material go beyond the 460MPa.

The case A, B or C are similar since they share the same yield point value, but they differ on the second yield stress point since the have different elastioplastic progression. In the case A the change from elastic to plastic zone happens suddenly and stress-strain slope becomes flat. In the case B and C they have a more progressive transition from elastic to plastic.

In all the cases the plate suffers severe permanent deformations that may lead to the complete failure before reaching the total deformation of 0.1mm in X-direction.

3 Plate with Pin

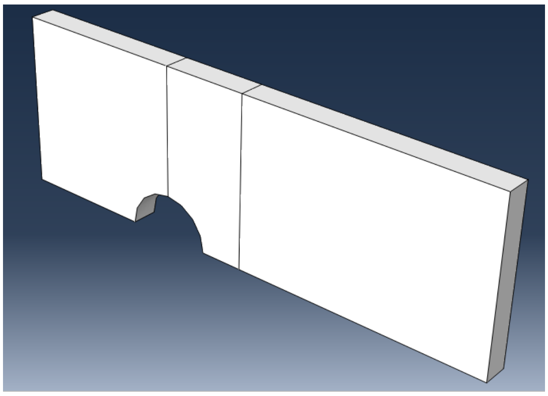
Here it is explained in more detail the study of the plate with a hole for elastic and plastic steel.

3.1 Model definition

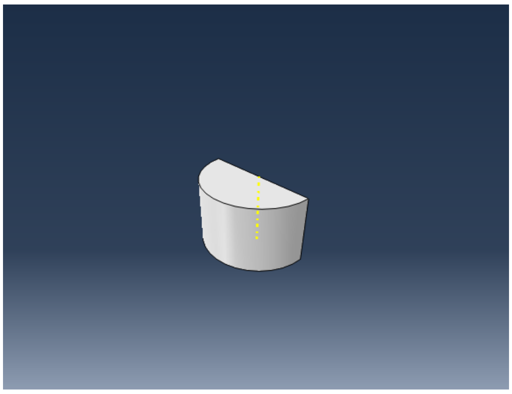
Several aspects for the preparation of the model to be simulated are described below.

Geometry definition

The plate is the same. Now it has been introduced a pin in the hole of 5mm of diameter and height of 3mm (twice the length of plate's thickness).



Half-plate geometry partitioned



Half-pin geometry

Boundary Conditions

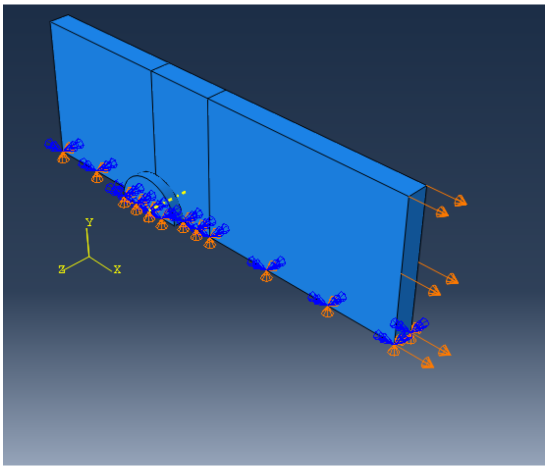
The degrees of freedom of the plate and the pin have been constrained to match the nature of the problem where the shape is stretched in X-direction. In addition contact conditions between the two geometries has been defined and different materials for each of them.

- The axis of the mid-pin has been fixed in the space with no possible movement or rotation.
- In the **right leading-edge plane** of the plate have prescribed displacement of **+0.1mm in X-direction**.
- The **middle horizontal** line in the leading-edges of the plate are forced to move only in the X-direction by **not allowing displacements in Y or Z direction**.
- **Contact conditions** has been prescribed between the plate and the pin with *friction coefficient*= 0.05.
- Finally the **symmetry** condition has been applied to the whole model in its middle plane.

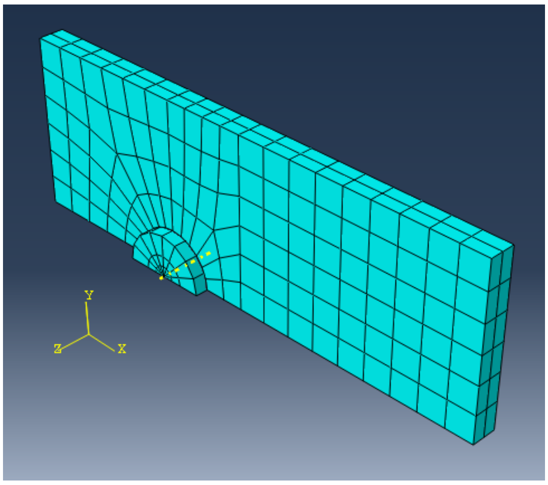
Meshing geometry

The geometry has been partitioned in order to get a good quality of structured mesh. Seeding on edges has been also modified to achieve the mesh density of cells equally in all the areas. Element type used is *C3D8R* in the plate. In this case, the pin has been meshed with elements types of *C3D6* around its revolution axis.

In the below images it can bee seen the boundary conditions and the mesh:



Boundary Conditions



Mesh

3.2 Results

It has been simulated 3 different cases by changing the plastic-elastic characterization curve of the material Pin. For all the cases the Young modulus of the material is the Steel young's modulus and is considered isotropic and plastic material. In the subsequent simulations the steel material of plate will remain constant while the pin steel will be varying its properties.

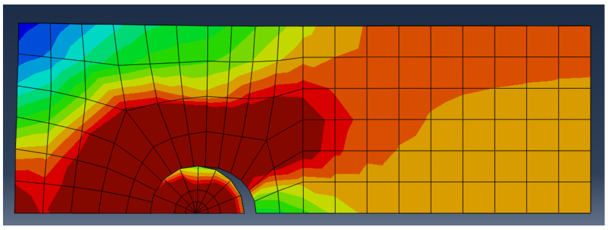
- **Case 1:** The material's properties are perfectly **elastic** for both, the **plate and pin**. No yield Stress point is required to characterize the material's curve.
- **Case A:** Now both parts are plastic. The **plate's yield stress** point is **460MPa** and the **pin's yield stress** point been set to **900MPa**. Note that, the pin is almost twice more strength than the plate.
- **Case B:** The plate material remains the same that in *Case A*. The **pin's yield stress** point has been decreased from 900MPa to **320MPa**. That means that the pin has almost three times less strength than the *Case A* so it is going to suffer more severe deformations for the same applied load.

Find in the below table a summary of all simulated cases:

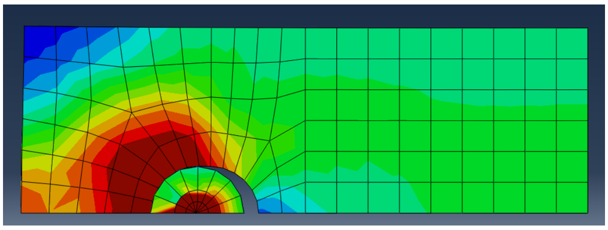
Case	**Piece**	**Material**	**Young modulus** $[N/mm^2]$	**Yield Stress 1** $[N/mm^2]$, $[mm]$	**Yield Stress 2** $[N/mm^2]$, $[mm]$
1	Plate	Elastic	2.1e5	-	-
A,B	Plate	Plastic	2.1e5	460, 0	520, 5e-03
1	Pin	Elastic	2.1e5	-	-
A	Pin	Plastic	2.1e5	900, 0	1000, 2e-3
B	Pin	Plastic	2.1e5	320, 0	400, 5e-3

Stresses

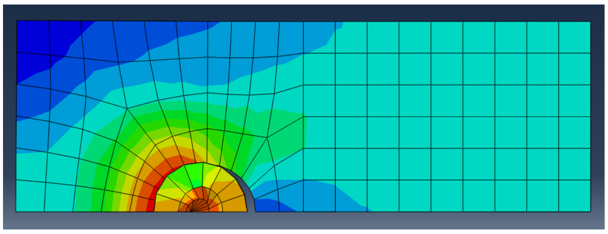
Find in the images below the stresses results of each case with the same scale.



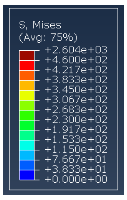
Case 1 Stresses



Case A Stresses



Case B Stresses



Legend

Force-Displacement of Node

It has been monitored the Force and displacement on X-direction in a node of the plate positioned in the mid plane and in the right leading-edge.

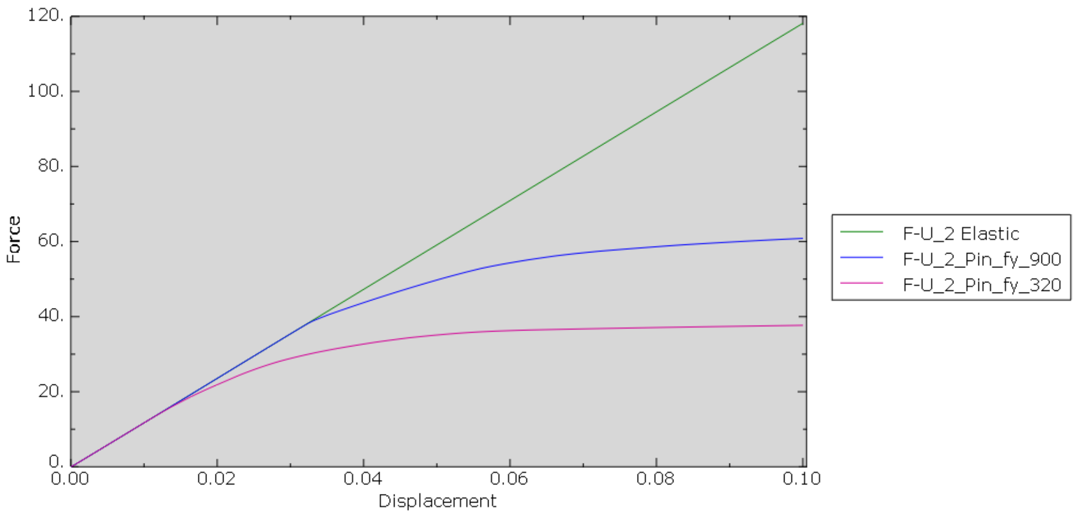


Figure 3: F-U curve of the cases

- In the case 1, both plate and pin are completely elastic and they absorb all the energy elastically. The pin and a great area of the plate are highly stressed.
- In the case A the pin is stiffer than the plate but it reaches its yield zone in its core. The yielding of the pin plus the yielding of the plate in the nearby contact area absorb great part of the energy so the rest of the plate is not too much stressed.
- In the case B the pin allows greats deformations and it supports great part of the energy without increasing too much the stresses transferred to the plate

4 Comparison

If we compare the first scenario without the pin against the second one, we can conclude that the pin has a great influence on the final force transmitted to the monitored point.
It is needed to understand that the boundary conditions are very different between the two scenarios. In the first scenario the "plate alone" is loaded across all their crossed section, while in the "Plate with pin" cases, the pin is fixed and this opposites the displacement of plate by the contact area in the hole. So great part of the stresses are supported by the pin which end up with the plate far less stressed.

The only scenario were the steel of the plate does not suffer plastic deformation is the very last simulated case: **case B ("Plate with Pin"**).