STRUCTURAL SIMULATION

Case Description: The following simulation work aims to analyze the performance of a clip of automotive door panels. Through the use of finite element method can be established the behavior of the piece during the assembly and know the effort to accomplish it. In the second stage of the simulation will know the maximum traction force that the clip supports considering the initial stress state generated by the door panel fixing.

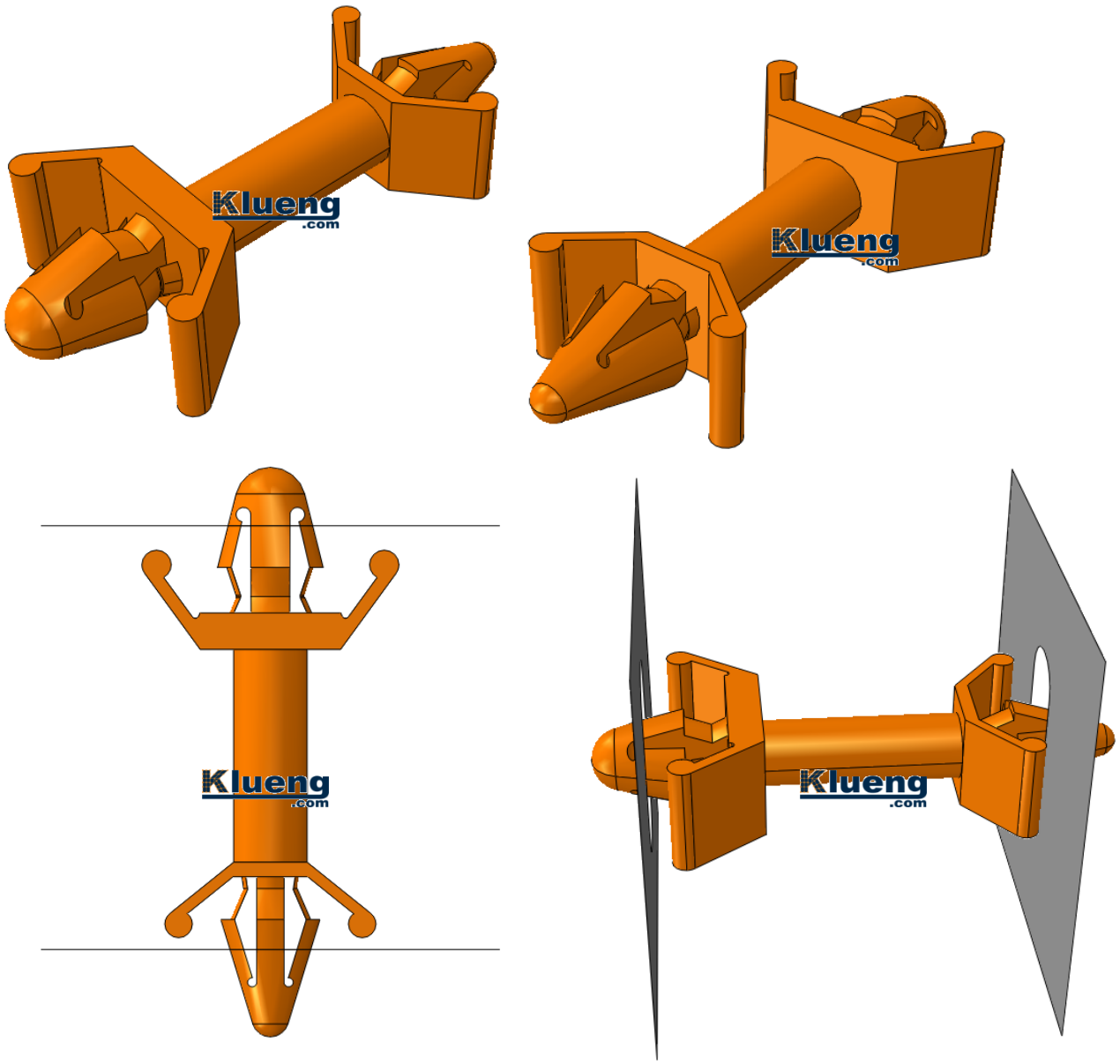

Fig. 1 – Geometry of the clip and the assembly simulated

Development: Fig. 2 shows the mesh of the finite element model. The mounting holes of the door panel and vehicle were modeled as rigid parts.

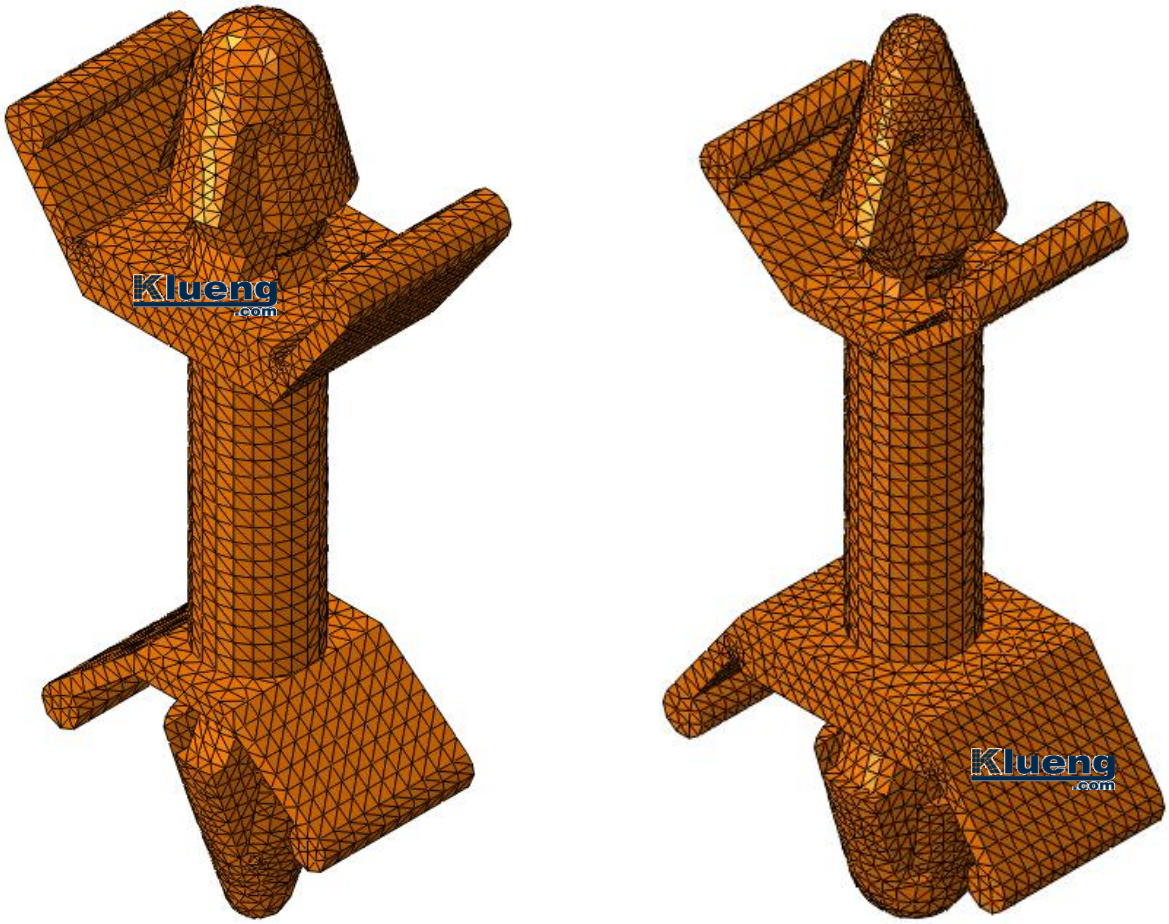

Fig. 2 – Finite element model mesh

The next step was to perform the simulation in which, at first, one end of the test piece engages and then the other, getting the necessary effort to carry out the assemble and knowing the stress state of the part to see if the assembly process produces the failure of the clip or not.

It was also simulated the limit effort which supports the clamping element when is pulled before its failure, for example, trying to separate the door panel from the car (closing operation of the door).

The following images show the evolution of the stress field on the clip during the assembly (from left to right, top to bottom).

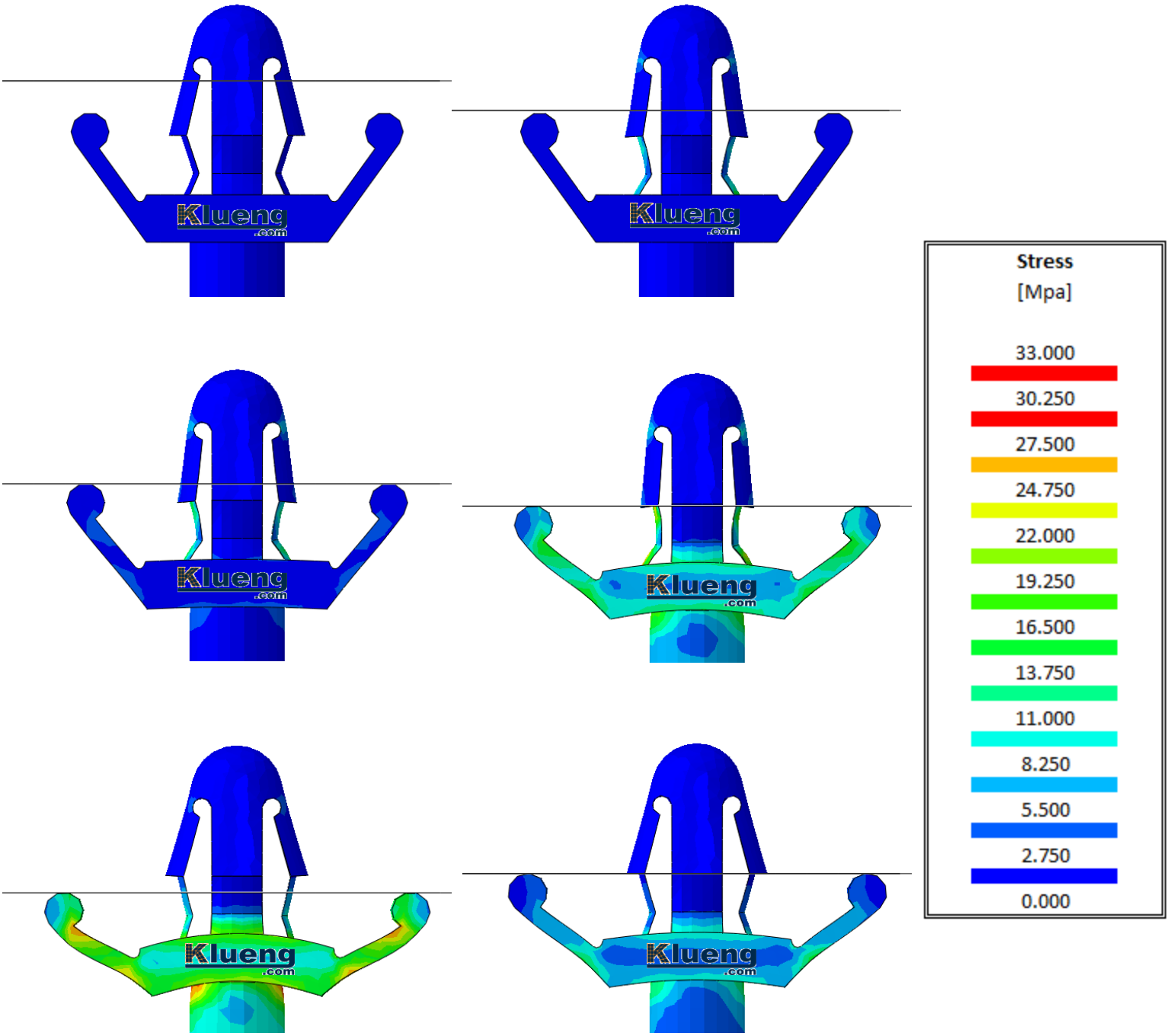

Fig. 3 - Top of the clip during assembly

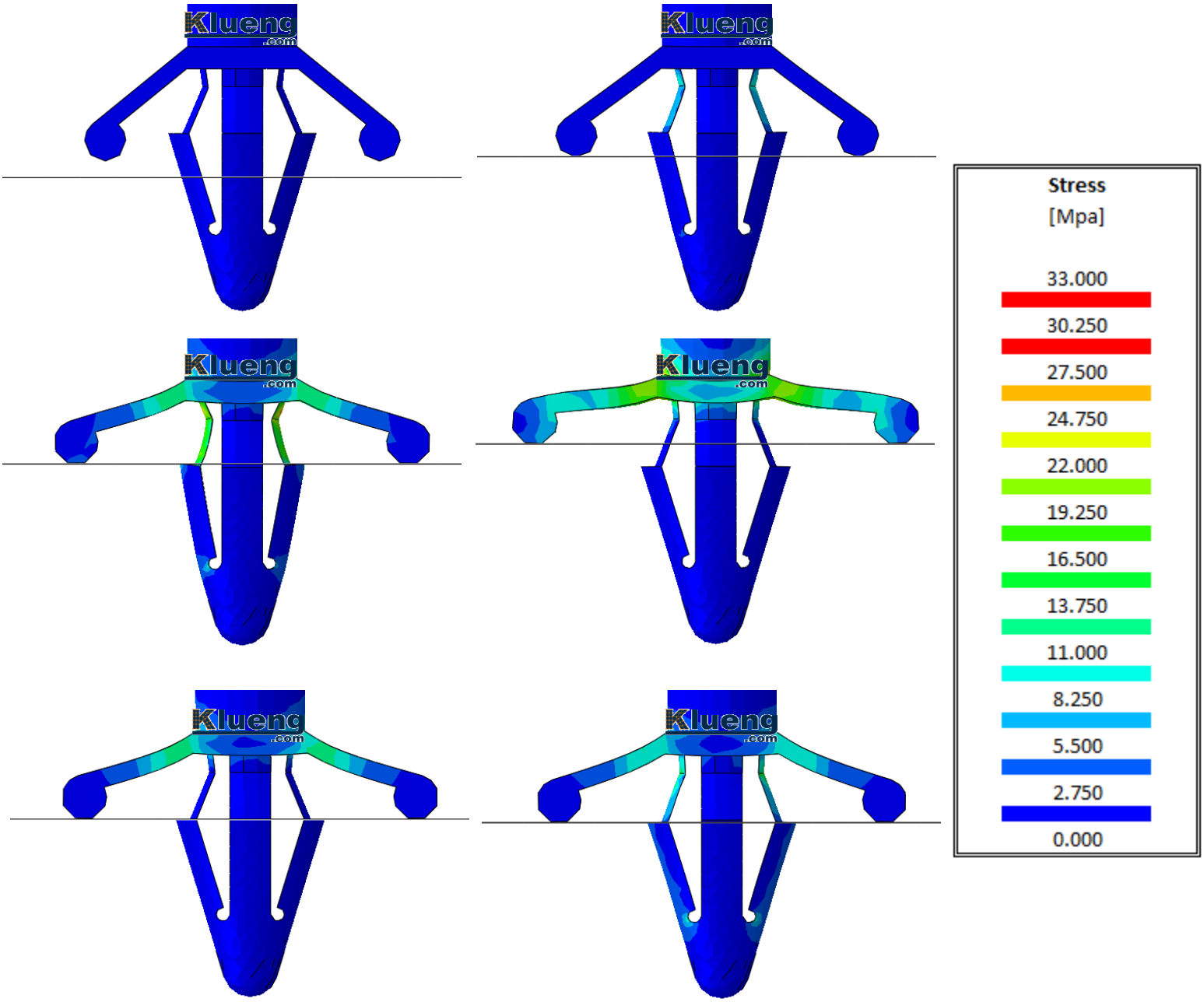

Fig. 4 - Bottom of the clip during assembly

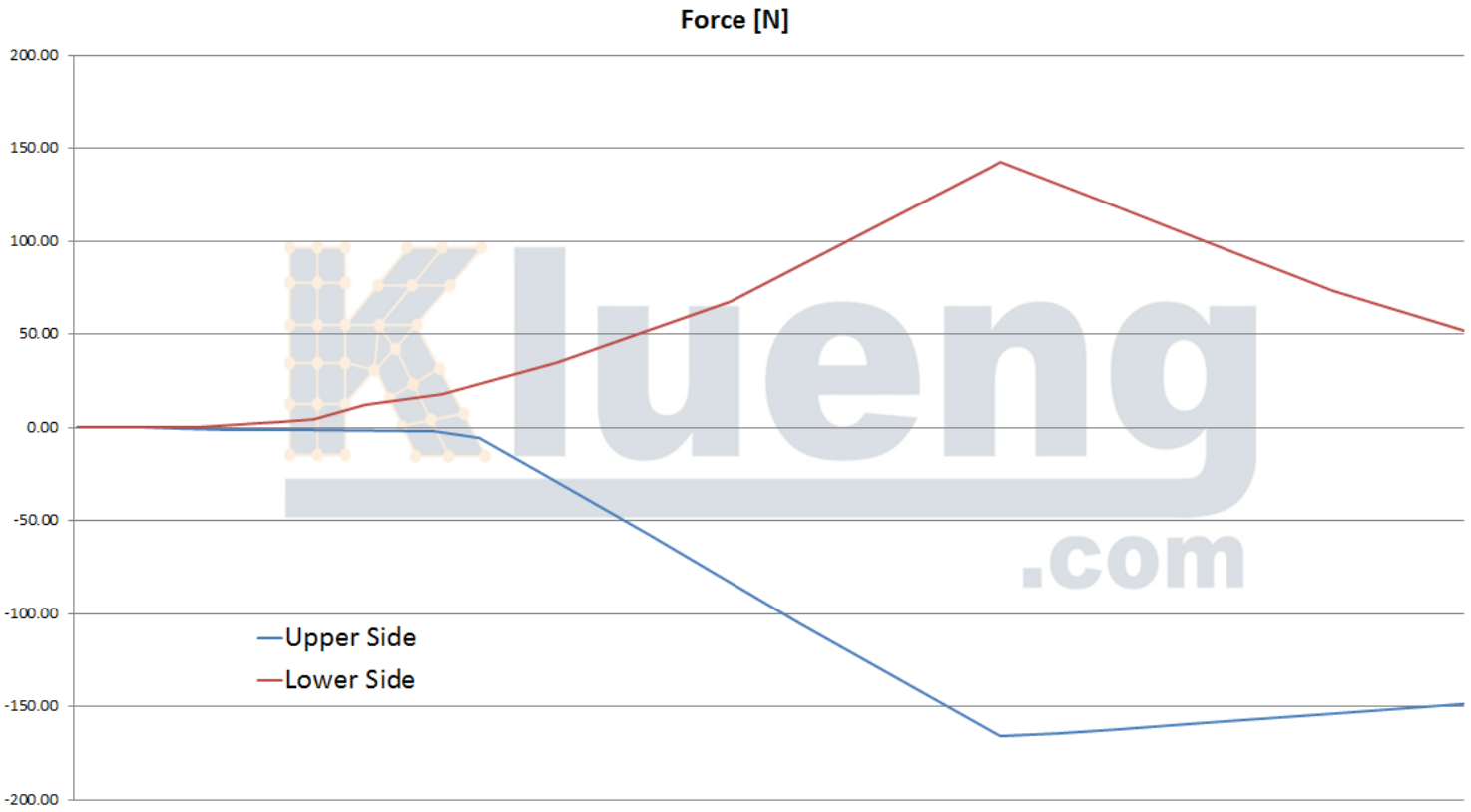

Fig. 5 - Force required to assemble the set

In a second step the maximum tractive effort that supports the clip before its failure was analyzed. The results are shown on the following page:

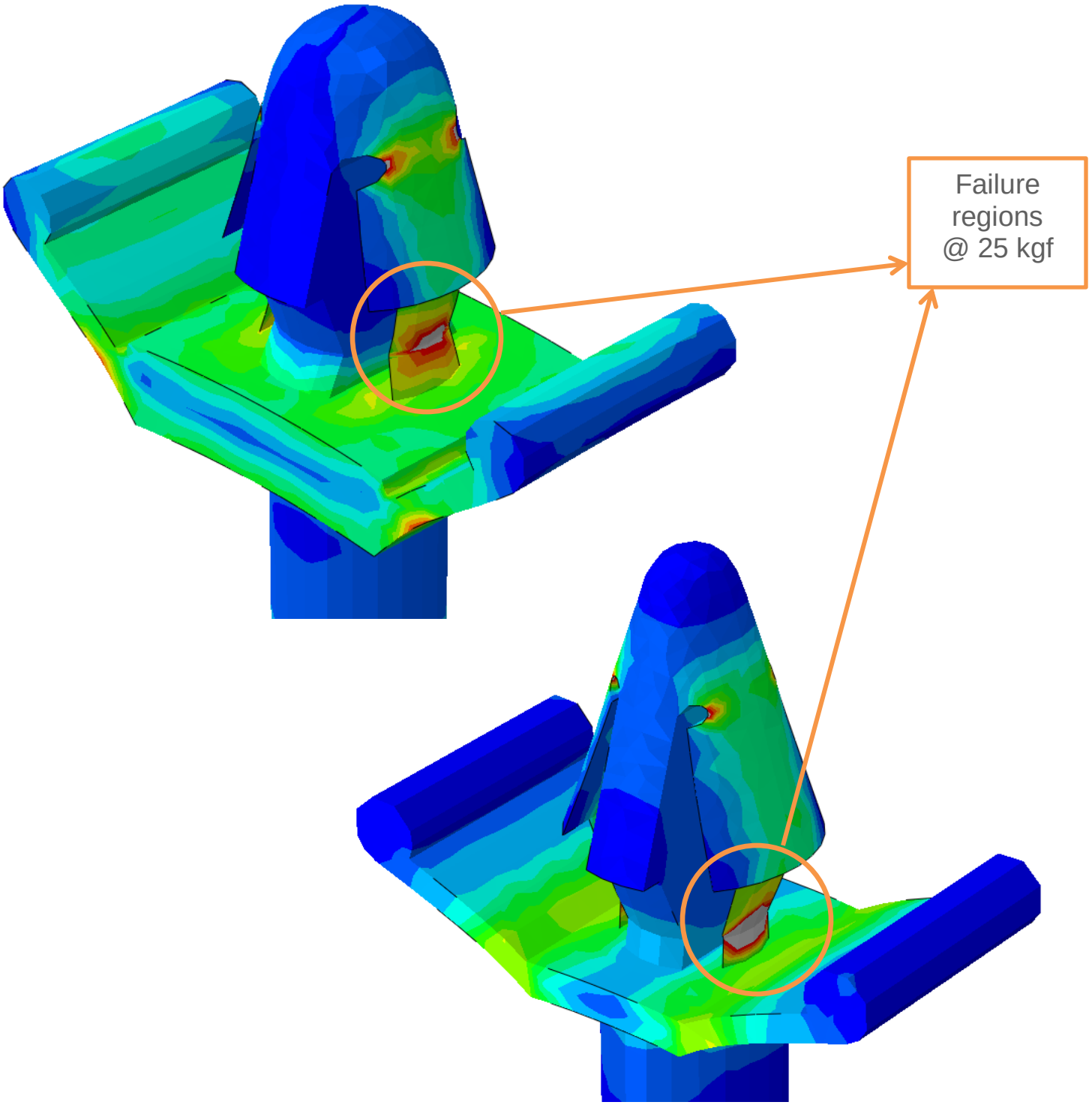

Fig. 6 - Regions where the failure occurs with 25 kgf applied

Conclusions: Through consulting FEM services was possible to establish in the virtual design stage:

- *Necessary effort to carry out the assembly of the clip (useful in the case of projecting an automatic assembly).*
- *Stress state in the clip during assembly operation.*
- *Critical load at which the element fails during operation and the region where it occurs (if necessary the element can be redesigned).*

With this information it is possible to optimize the product before making the injection mold to meet the specifications required by the customer.