Finite-Element-Analysis of Mechanical Characteristics of RTM-Tools

Dipl.-Ing. (FH) N. Erler, Dr.-Ing. M. Lünemann, Dipl.-Ing. (FH) M. Ströher
Airbus Deutschland GmbH
Center of Competence Jigs and Tools
Riesweg 151 – 155, D – 26316 Varel

Summary
Investigation to the mechanical behaviour of moulding-tools used for the production of aircraft-parts from carbon-fibre reinforced plastic (CFRP). Prediction of the deflection of the tool resulting in a geometrical inaccuracy of the part caused by heat and resin-pressure ANSYS Workbench FEA. Special attention has to be paid to the definition of the contact conditions set between the individual components of the tool.

Keywords
ANSYS, moulding tools, CFRP

1 Introduction

Composites gain an increasing importance in modern aviation industry; future aircraft families are anticipated to consist of CFRP-parts to a degree of 50 %. Responding to this development, also the respective jigs and tools have to be developed further since processing composites poses new requirements towards the tools as there is a significant interdependence between the characteristics of the tool and the achievable quality of the part.

Due to the higher complexity of the parts, this interrelation can no longer be predicted based on experience but has to be precisely forecasted, even, if the tool is merely intended for a test-part. Moreover, development cycles are shortened and do no longer allow time for iterative experimental trials to improve the design of the tools.

2 Object of interest

Airbus' Center of Competence Jigs and Tools reacts to this challenge by means of simulation such as the determination of the deformation of an RTM-tool caused by resin-pressure. RTM stands for Resin-Transfer-Moluding and describes the production-process for CFRP-parts by injecting liquid resin into a hermetically sealed tool after charging it with a dry preform and subsequent cure under the influence of pressure and temperature.

2.1 RTM-Tool

As an object of investigation a tool for a test-frame, which is shown in Fig. 1, was chosen. Such test-parts represent a cross-section of possible geometry features of structural parts and have to be produced to evaluate various combinations of materials, resins and geometry variations.

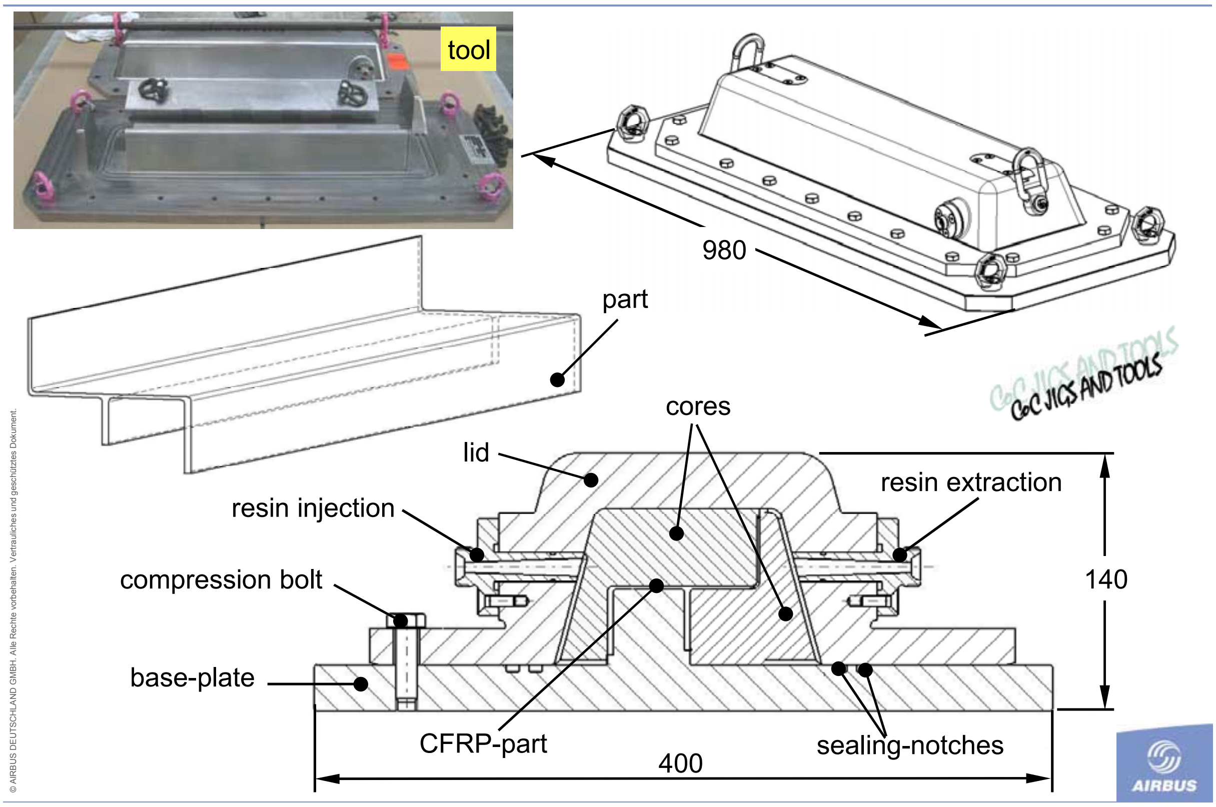

Fig. 1: Object of investigation

The tool mainly consists of a base-plate and a lid being held down by bolts to simulate the configuration during manufacture in a furnace environment. As the bolts tie down the lid, the slant of the contact-surface between the latter and the cores cause a movement of the cores towards each other resulting in a compression of the web in order to ensure the desired fibre-volume-content. Cores are necessary to allow for undercuts in the part. The seals and the resin-ports function as auxiliary elements.

In order to evaluate the design as far as structural shape and material-sizes are concerned, the deformation of the tool caused by the resin pressure at elevated temperature is of major interest.

2.2 Conditons arising from the use of the tool

The boundary conditions and influences which are taken into consideration in this study are enlisted in the following:

- Increase in temperature from 20 to 180 °C
- Difference in pressure: 7 bar
- Influence of the tool-components
- Contact conditions
- Compression of seals
- Resulting lid force
- Weight
- Cure in oven or under heated press

It becomes obvious, that the selection of the appropriate boundary conditions, e.g. the contact definition between the base-plate and the lid, has a decisive influence on the result. The determined deformation would cause the part to be outside the specified geometry.

3 Preparation of the FE-Model

3.1 Loading CAD-Geometry

The calculation discussed herein is based on the CATIA V5 model of the tool described above, which is transferred into the ANSYS Workbench environment using a gateway. Fig. 2 gives an impression of the process and shows how the product with its subordionate parts and elements such as axis systems and references.

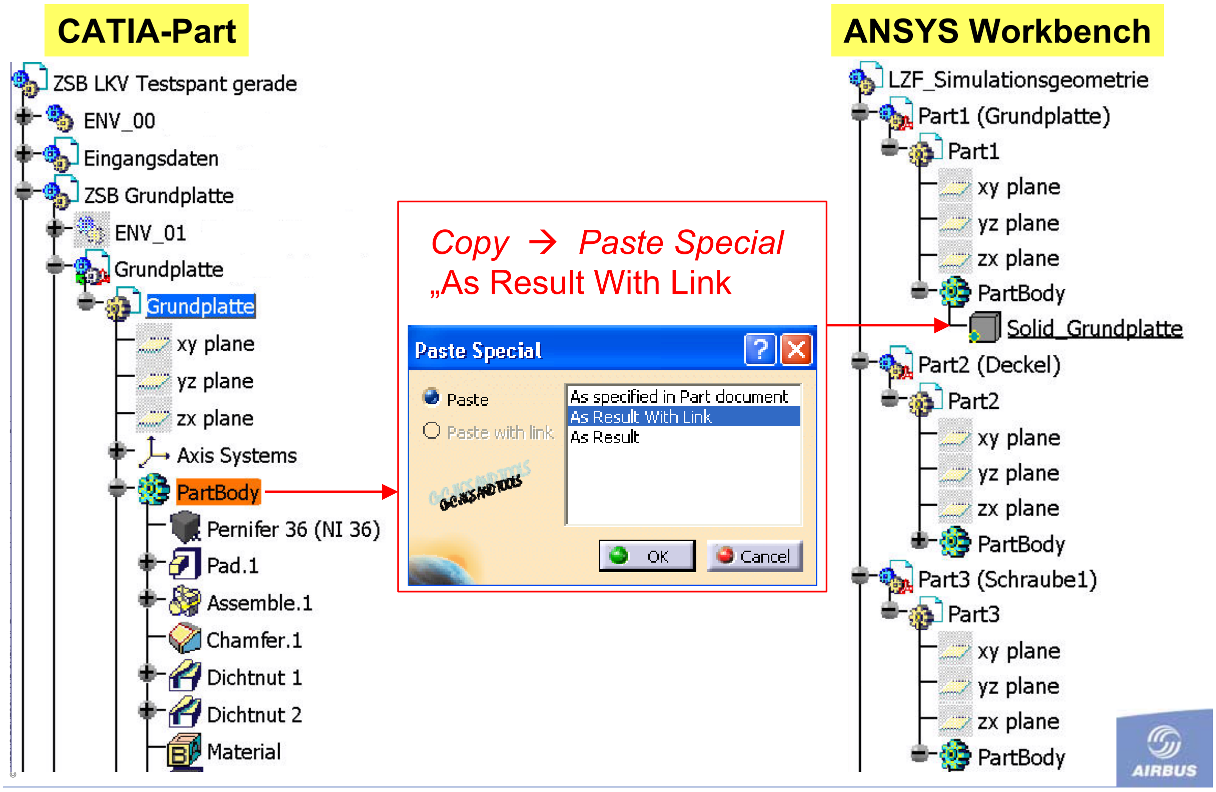

Fig. 2: Transfer of CAD-geometry

3.2 Refining Geometry

Once received in the simulation application, the model has to be refined from any geometry which is obsolete for the calculation. Particularly chamfers and bolt holes with a minor diameter as compared to the total dimensions of the entire object haved no influence on the result and can thus be omitted in order to reduce the number of elements.

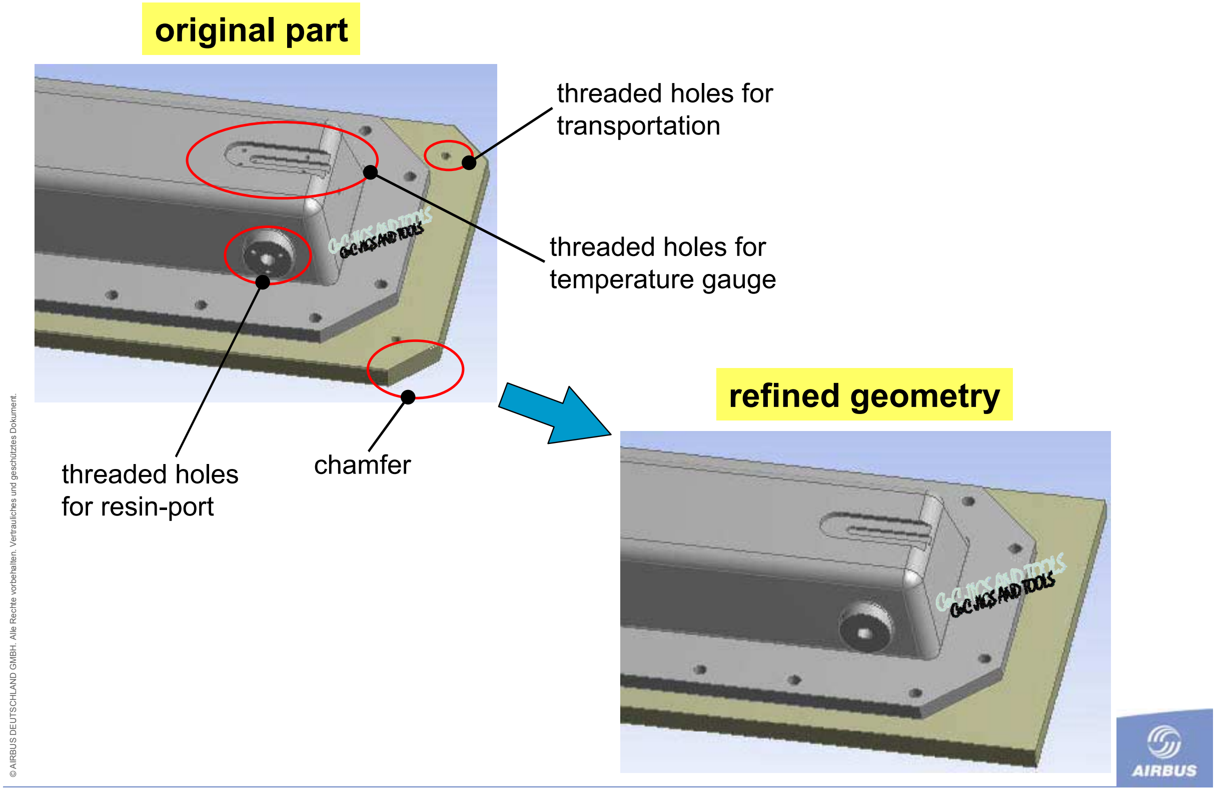

Fig. 3: Refinement of geometry

The elements being removed in this case are indicated in Fig. 3. The chamfers at the base-plate and likewise the threaded holes for transportation are irrelevant for this study. This also applies to the resin-port.

3.3 Setting Boundary Conditions

The boundary conditions have to be specified to ensure a convergent calculation. In this case the fixation of the base-plate preventing movement into any axis-direction as well as the contact condition between the two major components is of importance as shown in Fig. 4.

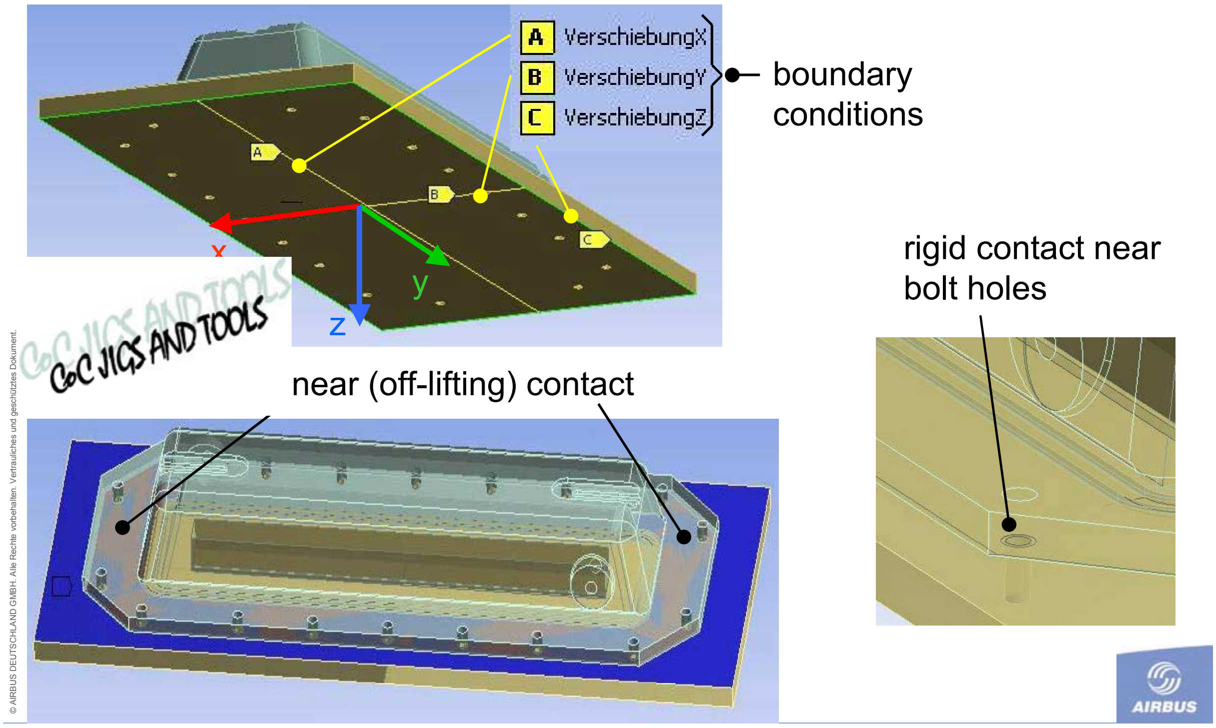

Fig. 4: FE-model with boundary conditions

The areas where and the way how the loads are applied are pointed out in Fig. 5. It is assumed further, that the tool is processed in a furnace and there is no limitation of the deflection by a heated press.

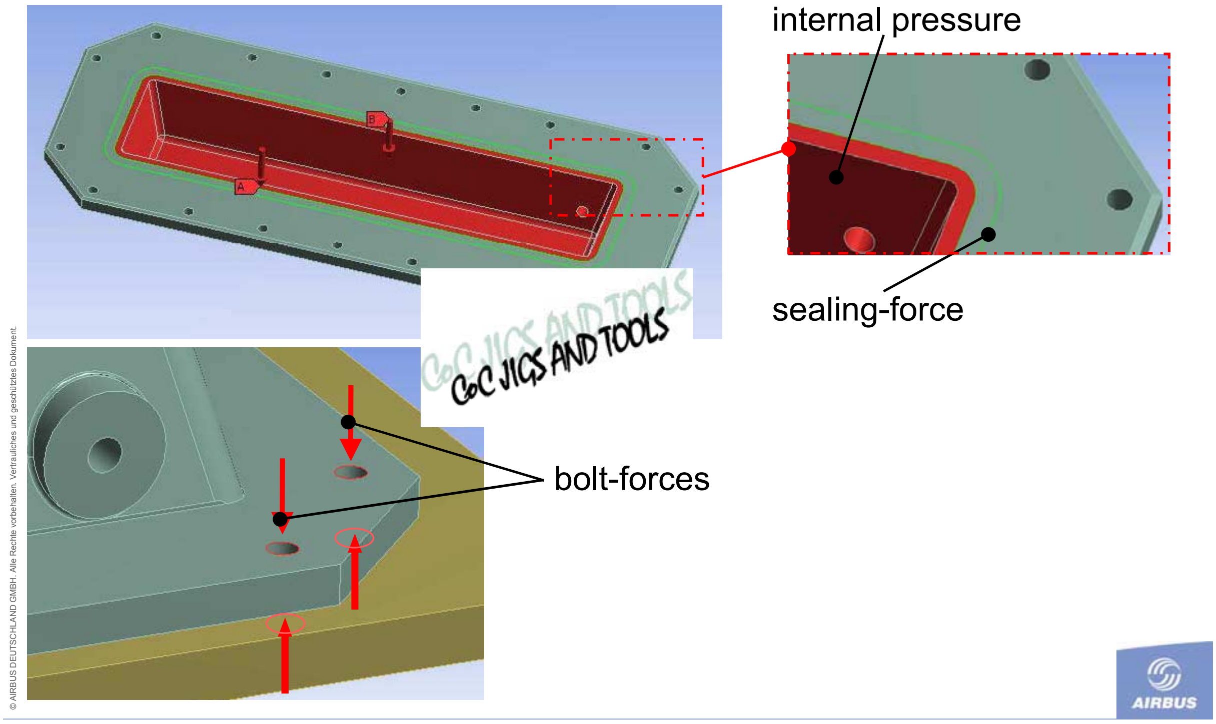

Fig. 5: Forces and loads applied to the model

Three different types of loads are taken into consideration in this case: the area load acting against the inner surface of the tool caused by the resin-pressure; the bolt-forces resulting from compressing the tool by preloading the bolts as well as the linear caused by compacting the seals. The resin-pressure impacts on all surfaces encompassed by the inner seal whereas the bolt-forces act on the edges of the bolt-holes in the lid.

4 Results

4.1 Stress and Deformation

During the study it becomes obvious, that the selection of the appropriate boundary conditions, e.g. the contact definition between the base-plate and the lid, has a decisive influence on the result and it is up to the expertise of the simulation engineer to decide which settings represent the reality most.

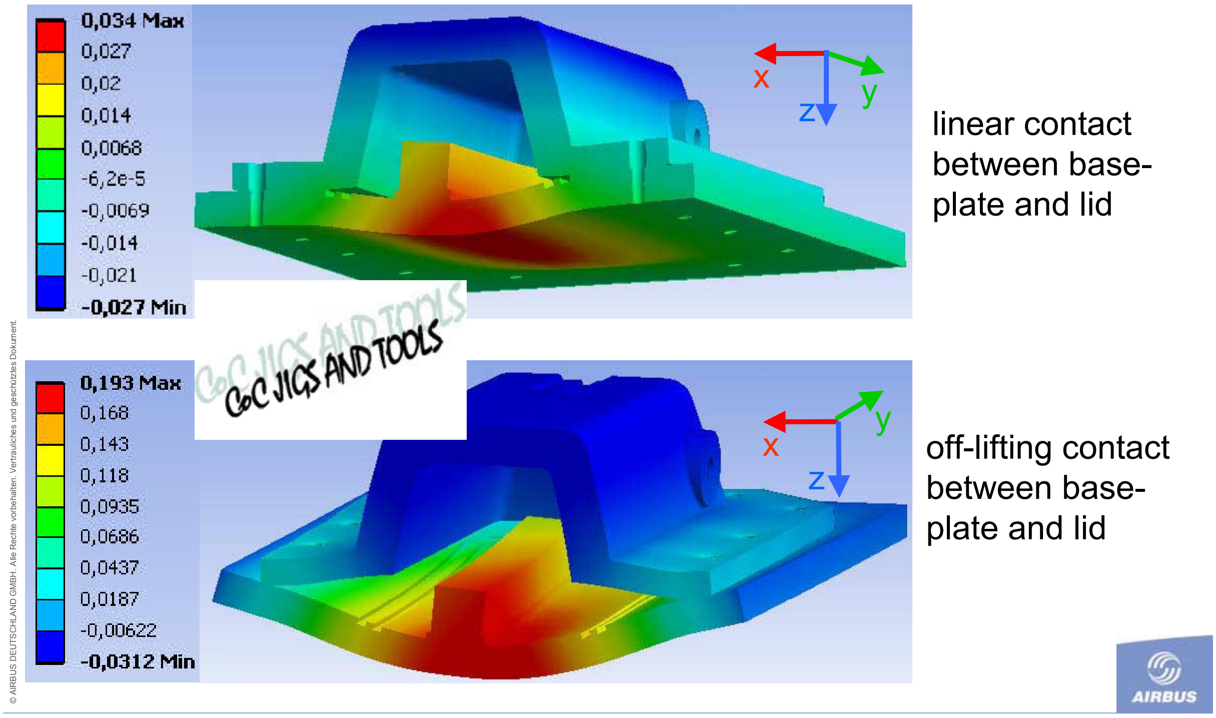

Fig. 6: Deformation of the tool-components in z-direction

A comparing visual impression of the deflection of the major components of the tool is conveyed in Fig. 6. It is comprehensible that the first definition supposes an ideal rigidity of the contact area whereas the second definition presumably resembles the reality more since the gaping between base-plate and lid can cause leakage which is observed from time to time.

Although the difference in the absolute values of deformation because of different contact definitions is rather small, the determined deformation would cause the part to be outside the specified tolerances.

Fig. 7 reveals the distribution of equivalent stress in the tool. The strain of the major structure seems uncritical because of thee rather high wall-thickness whereas stress-intensifications appear in areas where the bolt-forces act on the surface.

***Equivalent stress (von MISES)* [MPa]**

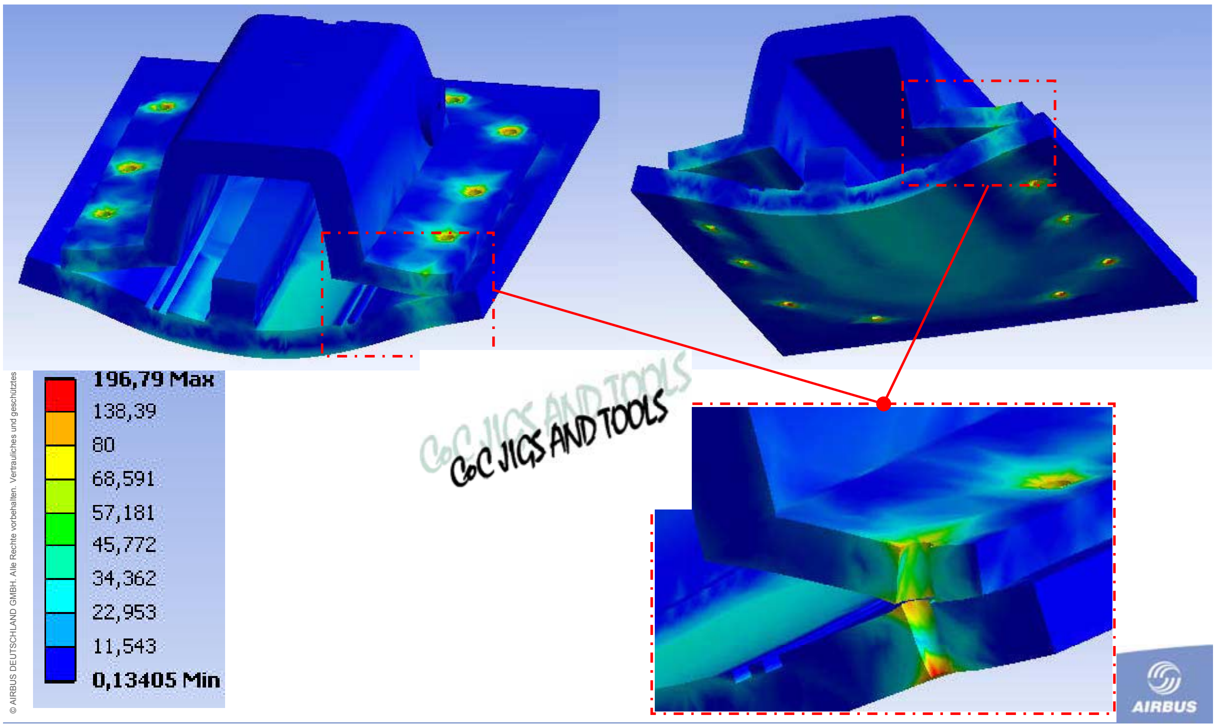

Fig. 7: Equivalent stress (von MISES) resulting from loads

4.2 Conclusions

The software program ANSYS is an efficient tool to evaluate variants of tool-concepts at a very early stage of the design-process and helps to prevent expensive rework.

5 References

[1] N.N.: Einführung in ANSYS DesignSpace 11.0, CADFEM GmbH, Grafing, 2007.