
Faculty of Computing, Engineering & Science

Master of Science - Mechanical Engineering

Module

Finite Element Analysis (NG3H238B)

Assignment Topic

Hook Analysis

Submitted:

University Year 2016/2017

04th of April 2017

Submitted by:

Banholzer, Florian (16123182)

Examiner: Dr Rae Gordon, University of South Wales

Content

List of Figures

List of Tables

1. Introduction

For industrial and construction applications hooks are important components, which are generally used as a fixing element to lift heavy loads. Such crane hooks are extended with a pulley to fit in a rope or chain of the crane. The hooks should be sufficient enough to pick up ropes or chains to carry the corresponding load (e.g. manufactured parts). In most cases, hooks occur in trapezoidal, circular, rectangular or triangular shapes. Furthermore, it is important that hooks are manufactured and engineered regarding their purpose to deliver the maximum performance, reliability and safety. (Devaraj, 2015) (Fetvaci, et al., 2006)

Hence, hooks are very liable components. Based on a continuous loading and unloading as well as stresses of various degrees, crane hooks are always subjected to failure due to a summarisation of a large amount of stresses. (Thakur, et al., 2016) To pretend a hook of failure and to increase the period of application, finite element analysis can be used to improve it. Finite element analysis is a numerical method to simulate loading conditions on a complex engineering problem and determine the design's response to those conditions. (Cunningham, 2014)

The scientific objective of this assignment is to perform a computer aided finite element analysis to calculate a stress analysis and resulting displacements of a hook under a certain load. Moreover, the following analysis is realised with the software ANSYS 17.2 and a subsequent scientific research.

The analysis is consisting of two main aims. First of all, a calculation of occurring stresses and resulting displacements, based on an applied load onto a crane hook, is determined with a two-dimensional model. For this analysis, with an illustrated hook in Figure 1, following facts are given:

Property	**Element Solution**
Type of material	Alloyed Steel
Young's modulus of elasticity (E)	210 GN/m^2
Poisson's ratio (ν)	0.28
Tensile strength (S_{uts})	724 MN/m^2
Elastic strength (S_y)	620 MN/m^2
Applied load	1182 N

Table 1: Basic finite element model facts

Furthermore, the second aim is to develop the justification for the three-dimensional cross-section of the hook in Figure 1. For this examination, the trapezoidal cross-section shape will be assumed. In addition, this second research question is based on the realised finite element analysis as well as a scientific research.

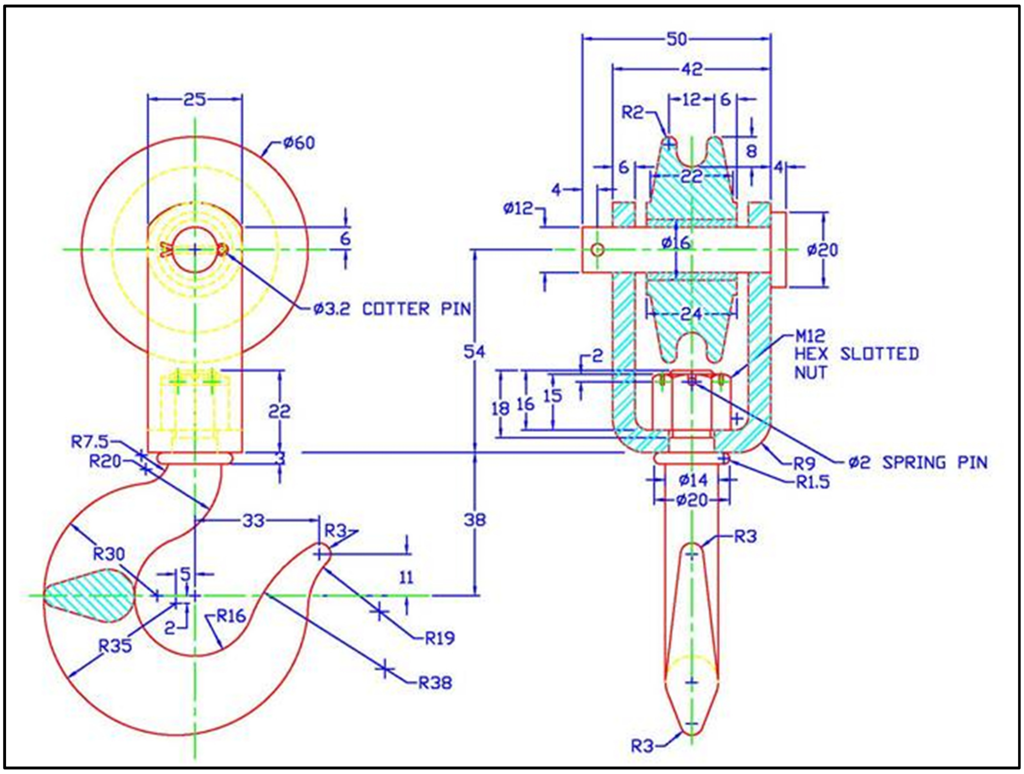

Figure 1: Proposed hook and pulley design

2. Assumptions

Initially, of the finite element modelling with ANSYS, some essential assumptions regarding the two-dimensional analysis of the hook have to be assumed. These assumptions were performed to simplify the analysis and to fill in the gaps of unknown values. This includes general assumptions, geometry assumptions, material assumptions and assumptions of physical conditions, which are described below.

2.1. General assumptions

To simplify the presented three-dimensional component, it can be reduced to a two-dimensional component analysis, in the case of an assumed constant thickness at any point. In this respect following general conditions have to be defined:

- The plane geometry is defined in the direction of the x-and y-axis as well as the thickness in the z-direction.
- The geometry or rather the cross-section is constant through the thickness.
- Plane stress can be assumed because the thickness of the model is relatively small compared with the other dimensions. Besides plane stress, plane strain is another finite element analysis approach for two-dimensional analysis.
- There are no changes in the geometry boundary conditions through the thickness.
- The force does not change through the thickness.
- No force in z-dimension (no out of plane force).
- Body forces in the plane of the structure are evenly distributed through the thickness of the structure.
- Loads cannot be applied in the plane of the hook.
- Shear effects are not considered. (Gordon, 1994) (Abbey, 2015)

2.2. Geometry assumptions

In fact, of the drawing in Figure 1 and the assumptions of a two-dimensional analysis, following additional assumptions concerning the geometry are assumed:

- The hook is a non-symmetrical geometry, it is necessary to include the whole geometry of the hook. Therefore, an axisymmetric problem cannot be assumed.
- According to a two-dimensional problem, the thickness has to be constant, as described in Chapter 2.1. Illustrated in Figure 1, such a simple crane hook has different thicknesses over the whole geometry. Based on the widest diameter of Ø14 mm and the

narrowest diameter of Ø6mm the simplified thickness can be determined with an arithmetic average. Therefore, the diameter of Ø6mm results from the radius R3. Moreover, following calculation can be assumed to determine a simplified thickness of the hook.:

$$thickness = \frac{\varnothing_{wide} + \varnothing_{narrow}}{2} = \frac{14\ mm + 6\ mm}{2} = 10\ mm$$

The simplification of the three-dimensional model towards the two-dimensional model with the calculated and assumed constant thickness of 10 mm can be visualised with Figure 2.

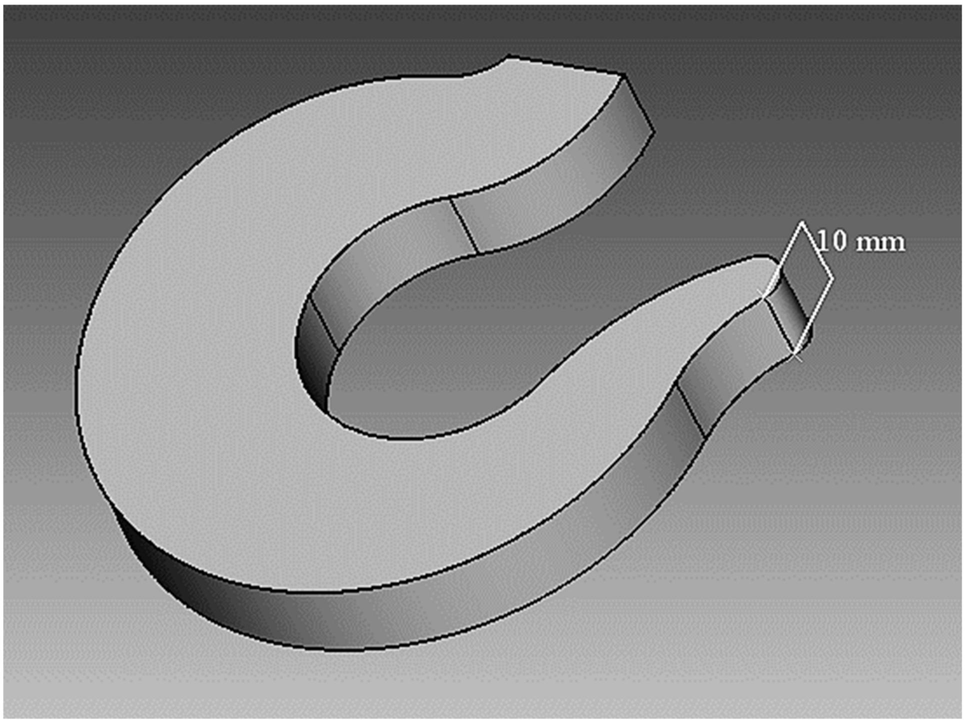

Figure 2: Drawing of the simplified hook

2.3. Material assumptions

Further assumptions about the material and its properties were adopted:

- The alloy steel of the hook is homogeneous with uniform consistency and properties through the whole material.
- In addition, no discontinuities and defects occur in the material.
- The material can be described as isotropic with the same properties in all directions.
- The material is linear elastic with no further deformations through external loads, it is completely reversible.

- The surrounding temperature has no influence on the hook and its material properties, which means the temperature is continuous.
- The simulation takes place in a continuous environment with no external influences.
- The yield strength will not be exceeded within the analysis.

2.4. Assumptions of physical conditions

In addition, following statements are assumed according to the physical conditions:

- At a certain point, there accrues no friction between the hook and the hanging load (e.g. rope).
- Body forces such as the dead weight of the components are not taken into account.

3. Modelling the analysis

Before the finite element analysis can be performed and examined, following boundary conditions and preferences have to be written down. These values are based on the assumptions of Chapter **Fehler! Verweisquelle konnte nicht gefunden werden.**, implemented into the modelling with ANSYS 17.2 and described in the following subchapters.

3.1. Element and Material

At the beginning of the modelling, the element type has to be chosen. As already described in Chapter **Fehler! Verweisquelle konnte nicht gefunden werden.**, the assumption of plane stress and two-dimensional analysis are the basis for the selected element type "PLANE183 - 2-D 8-Node Structural Solid". It is a higher order two-dimensional element with 8 nodes. Furthermore, it has quadratic displacement behaviour, is well suited to modelling irregular meshes and has two degrees of freedom (x-and y-direction). (SAS IP, Inc., 2016) Concerning the chosen element type, the element shape hast to be selected as quadrilateral and of course plane stress as the element behaviour.

Besides the element type, the material properties of alloyed steel can be extracted out of the problem description. Moreover, they are important to simulate the linear isotropic material of the hook. Nevertheless, the complete material properties and their values are shown in Table 2.

Property	**Value**
Young's modulus of elasticity (E) / EX	210 GN/m^2
Poissons's ratio (ν) / PRXY	0.28
Tensile strength (S_{uts})	724 MN/m^2
Elastic strength (S_y)	620 MN/m^2

Table 2: Material properties of alloyed steel

3.2. Geometry Structure

The geometrical structure of the hook is imported out of the "Hook.IGS" file into ANSYS 17.2. In addition to the given geometries out of this file, the thickness of 10mm, which is calculated in Chapter 2.2, has been added as a real constant. In the figure below, the basic two-dimensional shape of the hook, modelled with ANSYS 17.2, is illustrated.

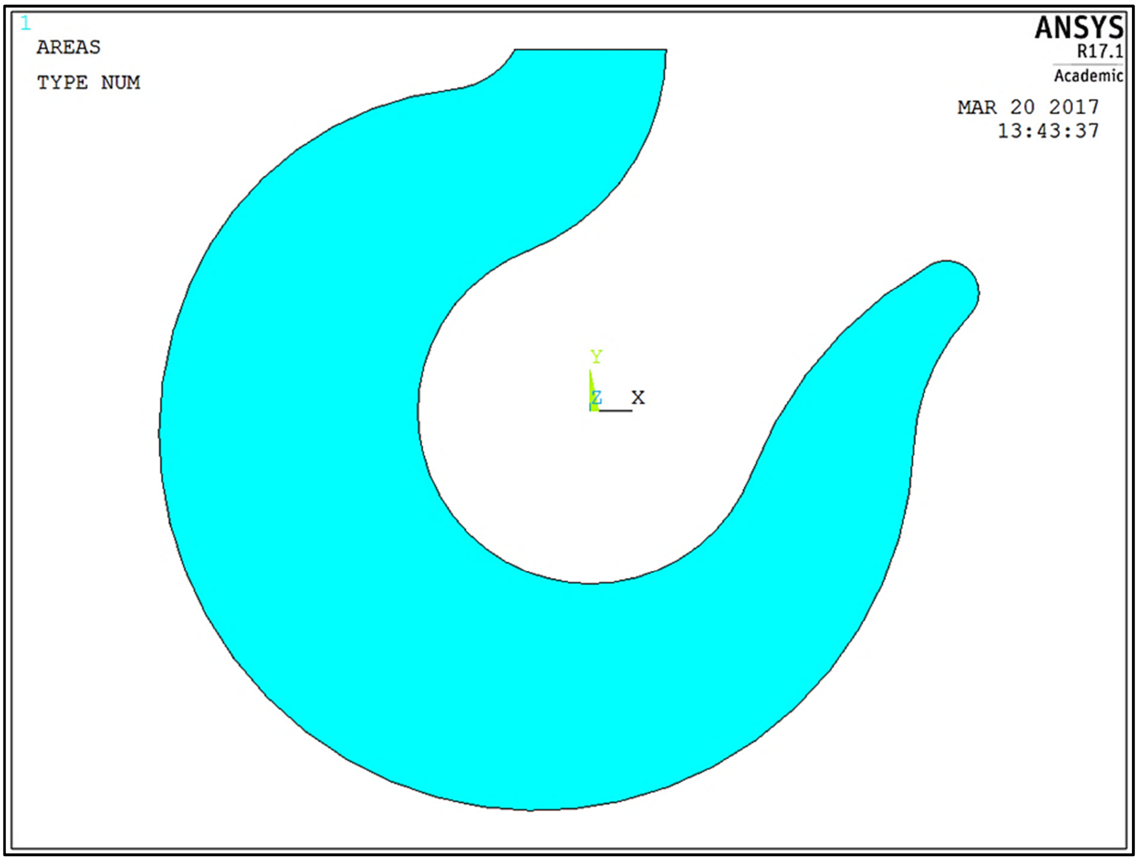

Figure 3: Two-dimensional shape of the hook

3.3. Boundary conditions

To apply a load, the boundary conditions simulate how the hook is fixed along its axes. In practical, the hook is fitted into a socket with a pulley, to hang down the crane. For this analysis, the hook is fixed on its upper line in all degrees of freedom (x-and y-direction). The conducted fixing of the hook with ANSYS is shown in Figure 4.

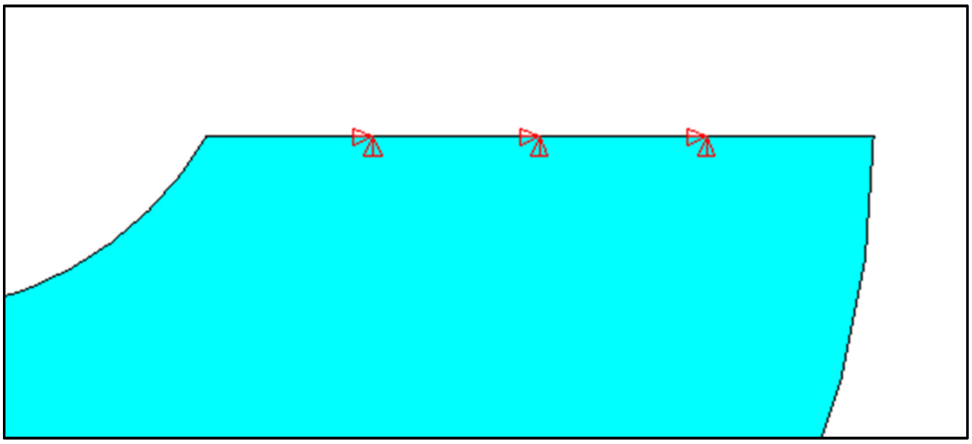

Figure 4: Boundary conditions of the hook in all dimensions of freedom (DOF)

3.4. Meshing

The total model and its area have to be mashed, to proceed further analysis. Therefore, the element edge length of the hook area was first considered with 3 mm as sufficient. After an optical assessment, the element edge length was finally adjusted to 2 mm. Furthermore, it was not necessary to refine the meshing to receive a subtle mesh in certain areas according to executed tests. The process of discretisation, which leads to the mesh, is important to divide the component into a number of small elements of defined sizes and nodes. Moreover, the mesh density and shape affect the solution accuracy and efficiency on a large scale. In addition, subtle mesh sizes are usually applied in areas with high stress/strain. (Tickoo, 2012) (Chen & Liu, 2014) In Figure 5Figure 1, the mesh of the hook is illustrated.

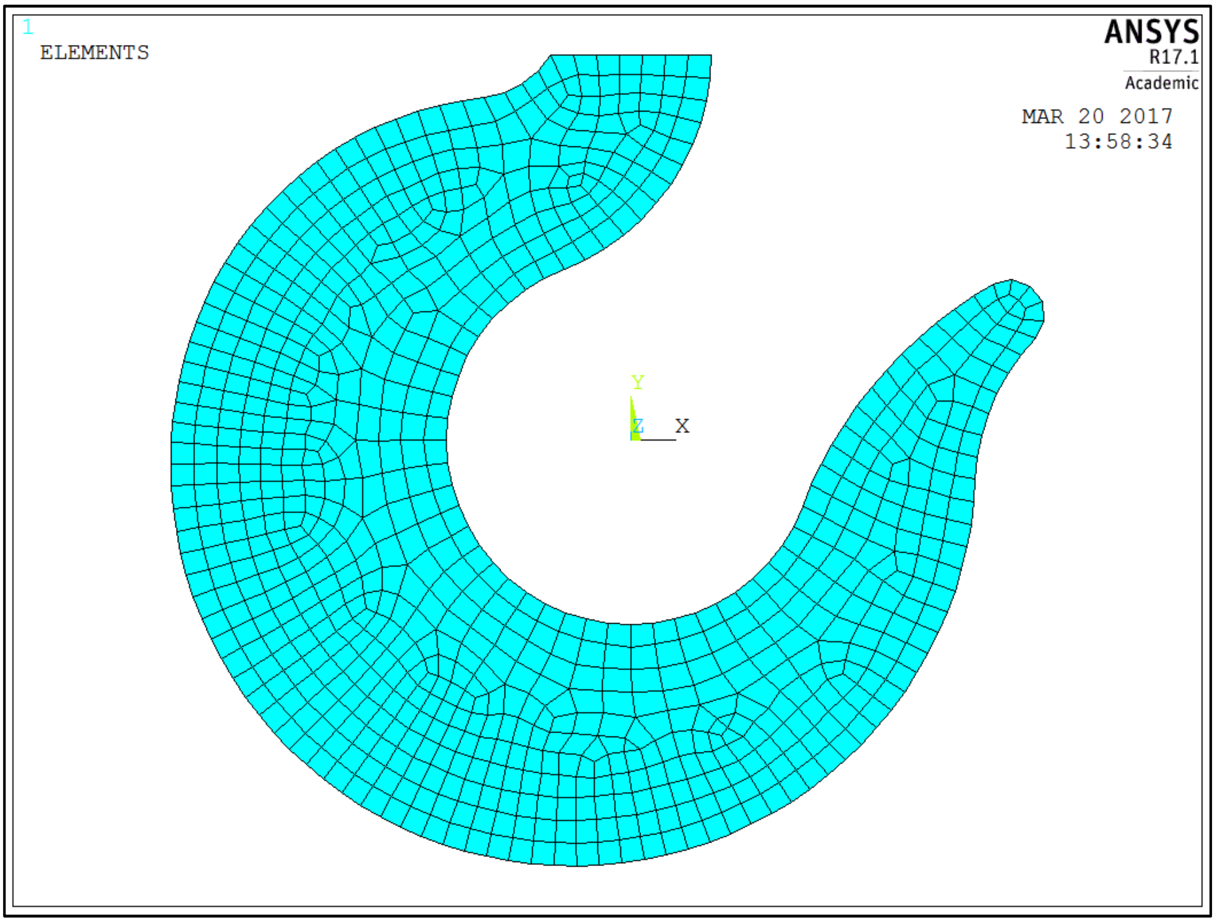

Figure 5: Meshed model of the hook

3.5. Loads

The final step of modelling is to apply the force onto the hook. To simplify the simulation, the load has been assumed as a constant load on a specific point (in the following referred to as a node). Hereof, the real thickness, for example of a rope, is neglected. The load acts with a value of 1,182 N in the y-direction on the lowest point of the inner radius, which is shown in Figure 6.

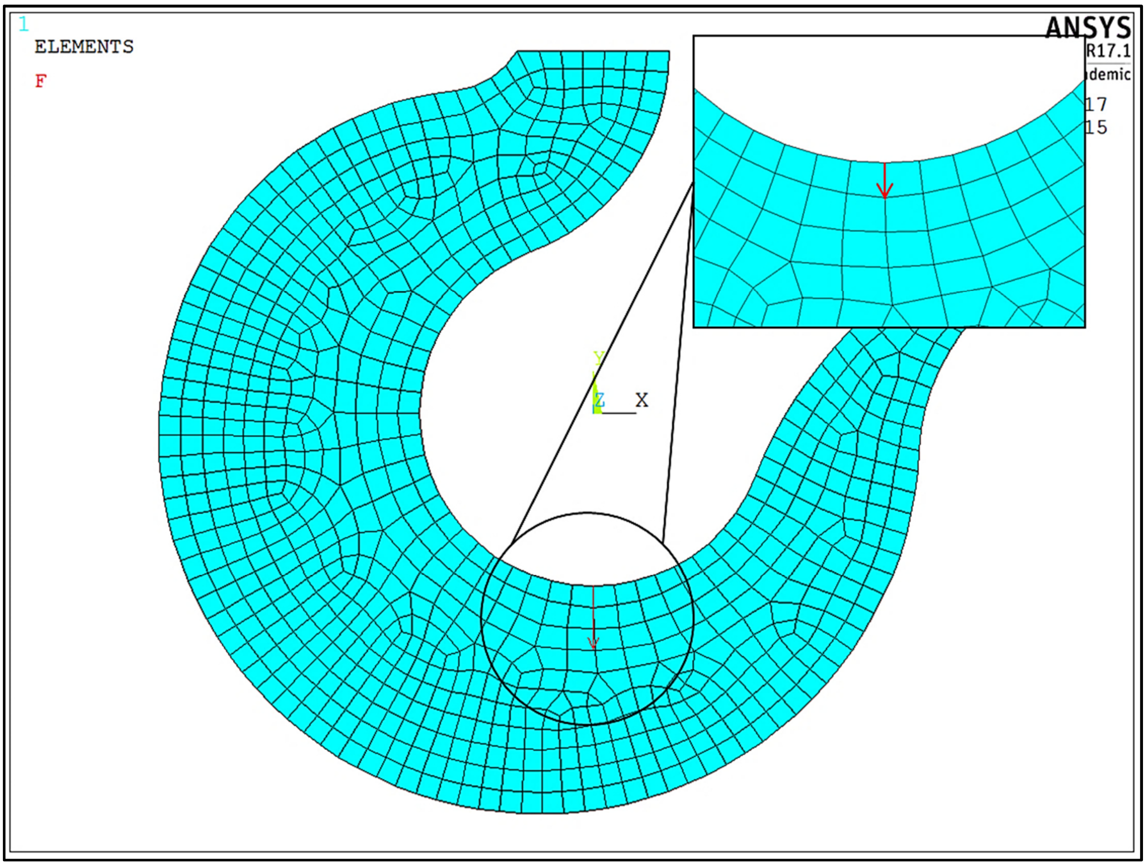

Figure 6: Load on the hook

4. Results

In this chapter, the results based on the modelling with ANSYS 17.2, described in Chapter **Fehler! Verweisquelle konnte nicht gefunden werden.**, are presented. The attention of the results is focused on the displacement and the stresses resulting on the hook based on a constant load.

4.1. Displacement

The displacement for the finite element analysis is calculated on the nodes for the applied load of 1,182 N. Assuming a two-dimensional analysis, the displacement of each node within an element consists of two components, in the x-and y-direction. (Modlen, 2008)

Figure 7, displays the maximum displaced structure of the hook, plotted over its original shape. It must be noted, that the deformation in this picture is scaled up, to visualise the shift. Moreover, the maximum displacement of 0.054977 mm or rather the vector sum of the x-and y-direction can be read out of this figure. This maximum displacement, received out of the list results, occurs in node number 177. Node number 177 is also marked in Figure 7.

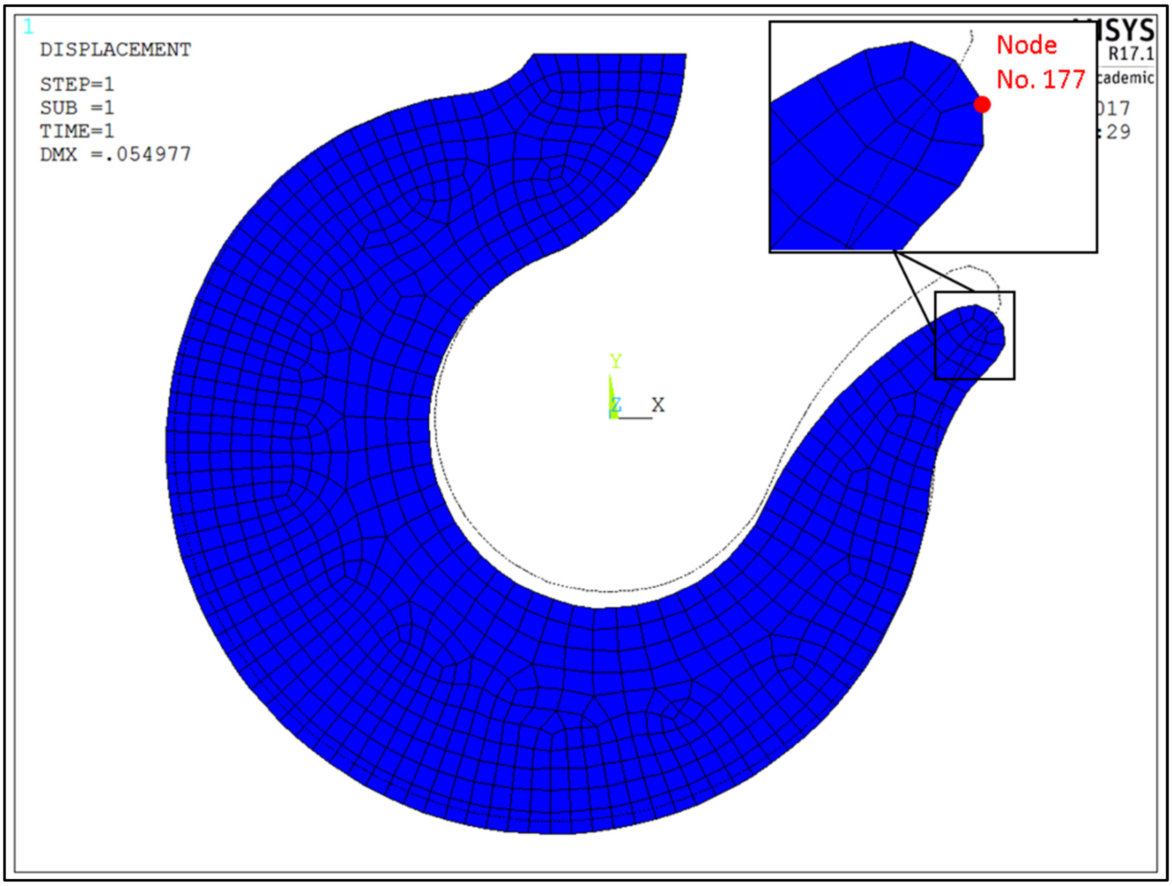

Figure 7: Maximum displacement of the hook

In addition to the explanations above, Figure 8 shows the course of the deformation. The deformation of the hook behaves as expected, consequently, the boundary conditions and loads have been added correctly. In this case, the main part of the radius has bent downwards and a displacement of around 0.025 mm (green) results. Furthermore, the end of the hook on the right side is moving downwards and shifts minimal to the right side with the maximum displacement (red). Around the upper area of the hook, there is no visible deformation (blue) because of the adopted boundary conditions and material properties.

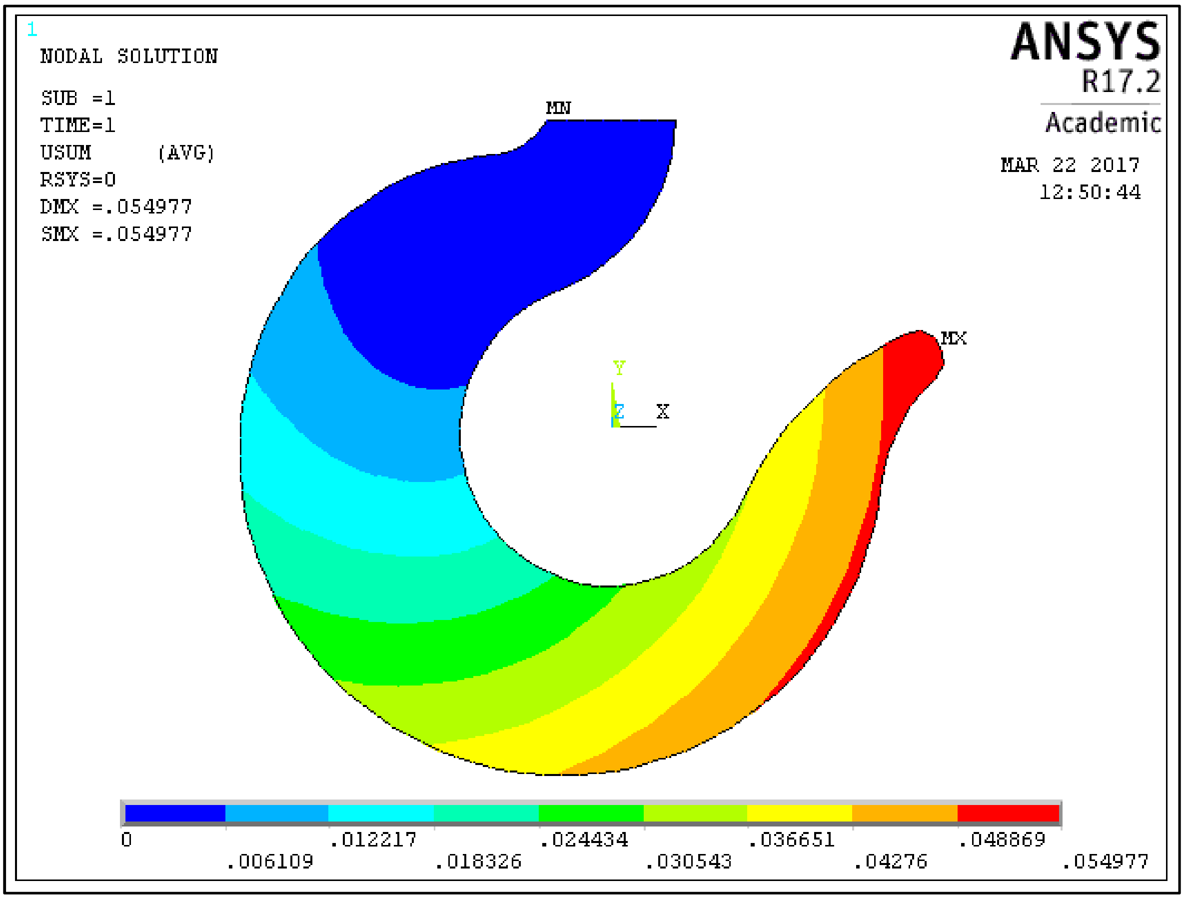

Figure 8: Nodal solution - Displacement vector sum

4.2. Operating stresses

After the displacement is obtained, the corresponding stresses can be calculated. Such stresses are described as the force of resistance, offered by the shape against the deformation. Therefore, stresses are calculated as a load per unit area (N/mm^2). The different stresses act in the direction of the global cartesian coordinates triggered through the applied loads. Stresses can be evaluated by elements or nodes. Nodal solutions are the averaged values of stresses at each node. In comparison, the element solution calculates the stress of each element with the average value of its corresponding nodes. For a comprehensive simulation, the results of the nodal solution have to be compared with the element solution. The comparative value is an indication of the right chosen mesh size. If the differentiation is too high, the mesh size is too coarse. (Tickoo, 2012) (3DVision Technologies, 2008)

The following subchapters describe the selected and most informative stress results with ANSYS 17.2. for the realised hook analysis.

4.2.1. Stress in x-Direction.

The stress course in Figure 9, shows the minimum (SMN) and maximum (SMX) stresses along the x-direction. As expected the maximum compressive stress occur in the area (1). However, this is based on the simplification of the simulation with the application of the load on one specific point. This means that the maximum compression in the area (1) can be neglected. Hence, it can be seen from the Figure 9, that the highest compressive stress occurs in the area (4) with approximately 25.2 N/mm^2 according to the colour scale. Additionally, around the area (5) less compression compared to the area (4) take place. Moreover, tensile stresses only develop on the inner radius of the hook, especially in the area (3) with 52.5 N/mm^2. Another area with tensile stress is the number (2), but its course operates less along the x-direction.

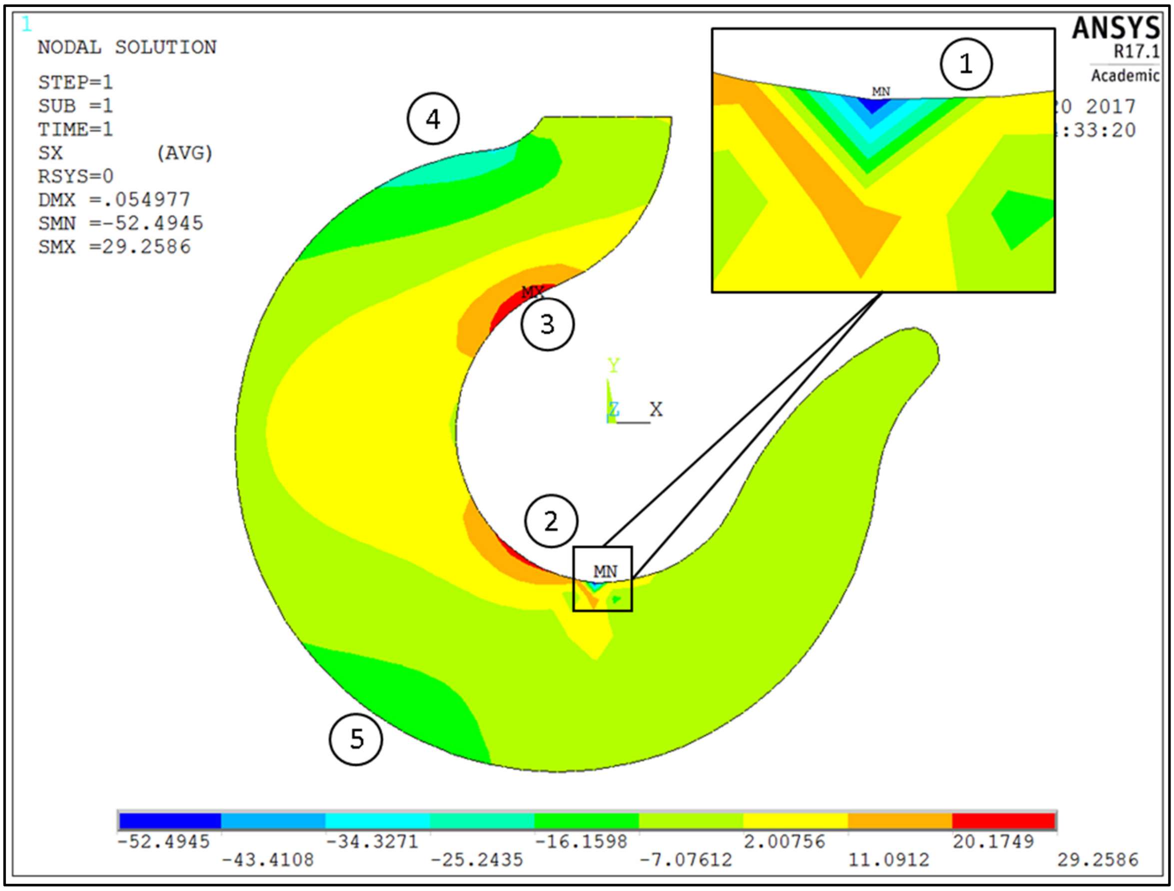

Figure 9: Nodal solution - X-Component of stress

As compared to the nodal solution, the element solution in Figure 10 looks almost the same. Regarding this perception, Table 3 illustrates the comparison of the respective maximum and minimum values. The stresses of the element solution divided by tensile and compression stress proceeds also like the nodal solution. More important, the evaluated values do not differ significantly. That is eventually the result of a fine enough mesh size.

Comparison x-component of stress			
	Nodal solution	**Element Solution**	**Discrepancy**
SMN	- 52.49 N/mm^2	-52.17 N/mm^2	0.32
SMX	29.26 N/mm^2	29.38 N/mm^2	0.12

Table 3: Comparison x-component of stress

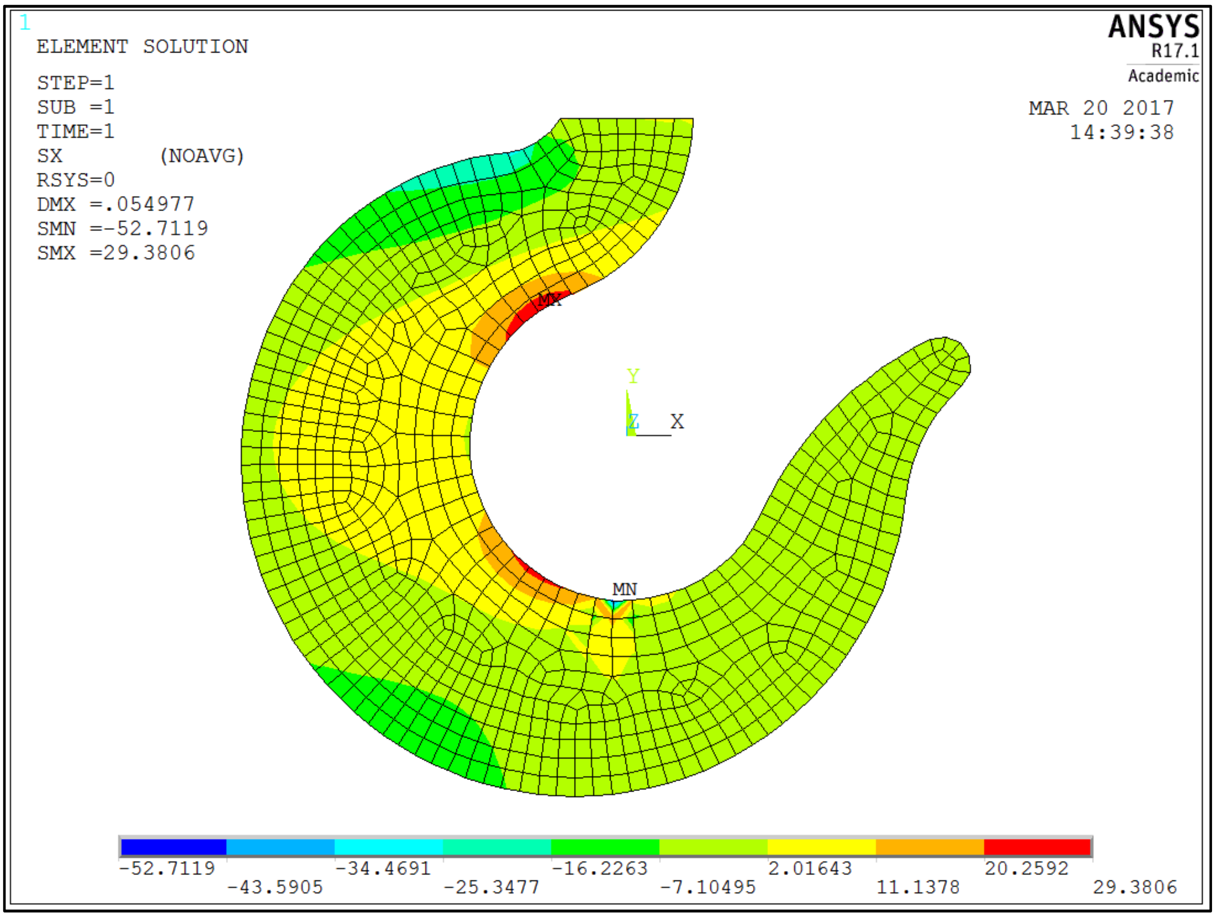

Figure 10: Element solution - X-Component of stress

4.2.2. Stress in y-Direction.

The stress results along the y-direction can be evaluated similarly to the x-direction. For the area (1) the same applies than described in Chapter 4.2.1. The values can be neglected because of the load applies on specific a node. Furthermore, it can be seen that the maximum compression stress with 99.5 N/mm^2 occurs in the area (2), due to the applied force on a specific node. This perception provides the highest tensile stress with 56.2 N/mm^2 in the area (2). Aforementioned explains that through pulling the hook in the y-direction, it will increase in length. Furthermore, in the area (3) occur the highest compression with a value around 30.3 N/mm^2. Yellow and light green areas, like area (4), show fewer stresses, in some parts right to virtually zero.

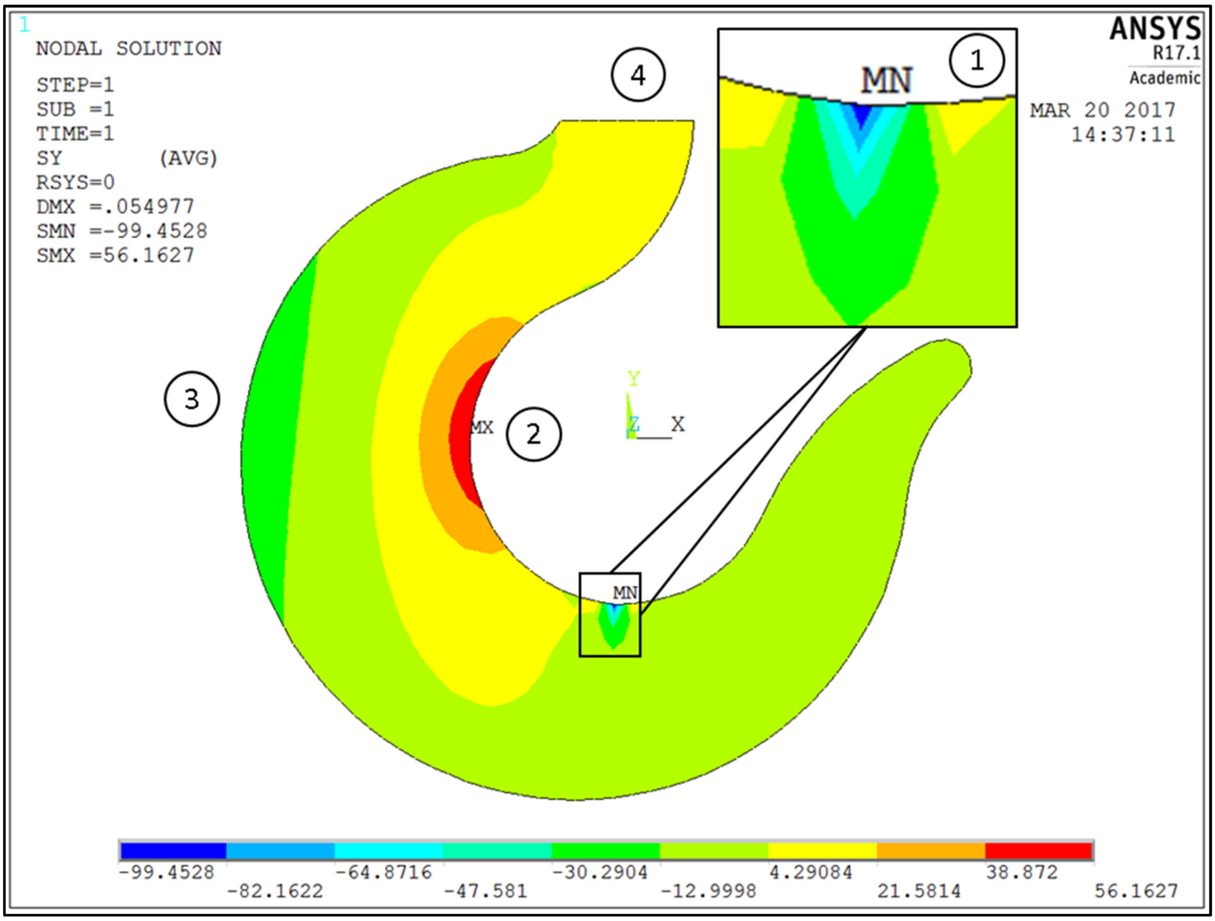

Figure 11: Nodal solution - Y-Component of stress

Figure 12 illustrates the element solution with the y-component of stress, it is the counterpart to Figure 11. As described in Chapter 4.2.1 the nodal and element solution for the y-direction also do not differ significantly. Regarding this perception, Table 4 illustrates the comparison of these values.

Comparison y-component of stress

	Nodal solution	Element Solution	Discrepancy
SMN	- 99.45 N/mm^2	-100.38 N/mm^2	0.93
SMX	56.16 N/mm^2	56.17 N/mm^2	0.01

Table 4: Comparison y-component of stress

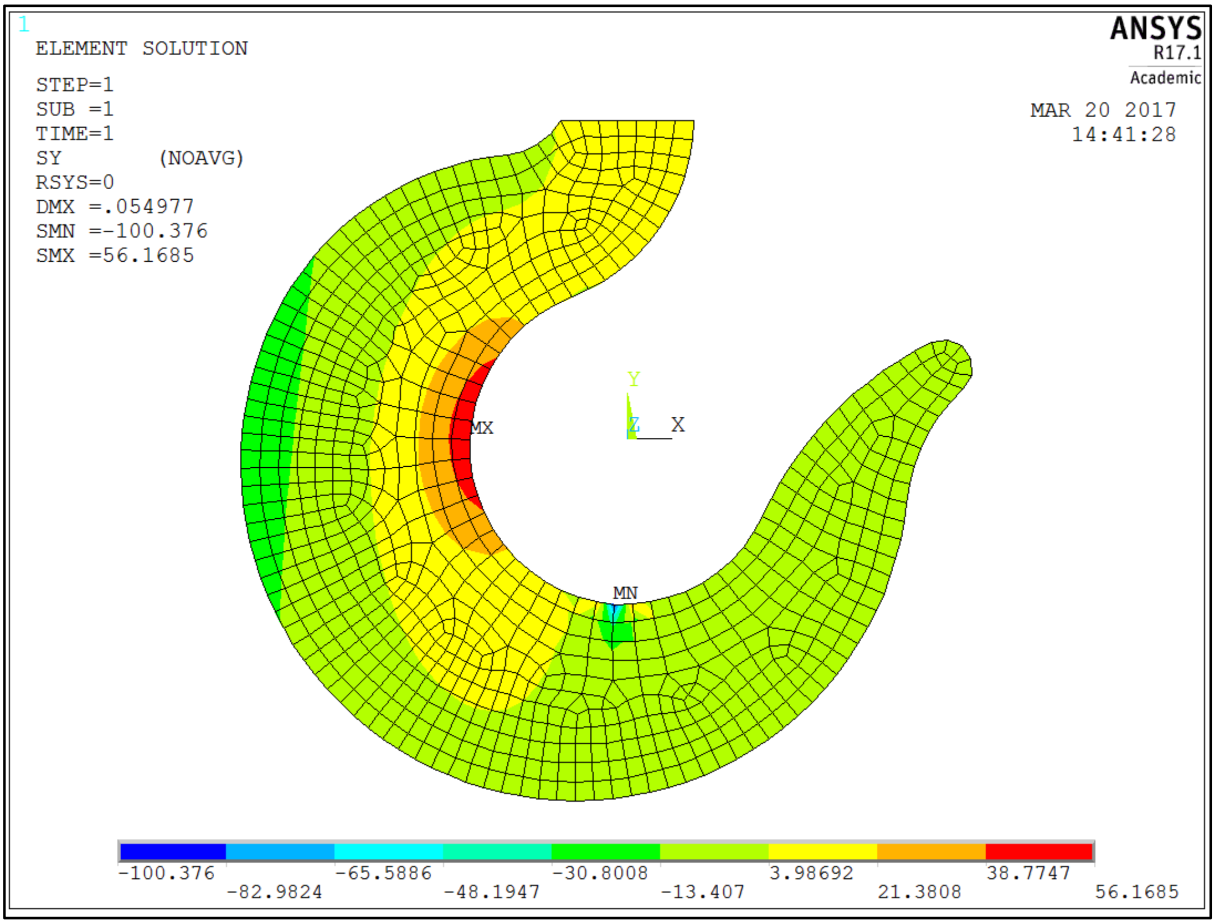

Figure 12: Element solution - Y-Component of stress

4.2.3. Stress in z-Direction.

To complete the analysis of stresses in all degrees of freedom, Figure 13 shows the graphical element solution in the z-direction. As expected according to the assumptions and a two-dimensional analysis, the z-component of stresses are negligible because they are zero. In this case, a detailed comparison of the element and nodal solution is not necessary.

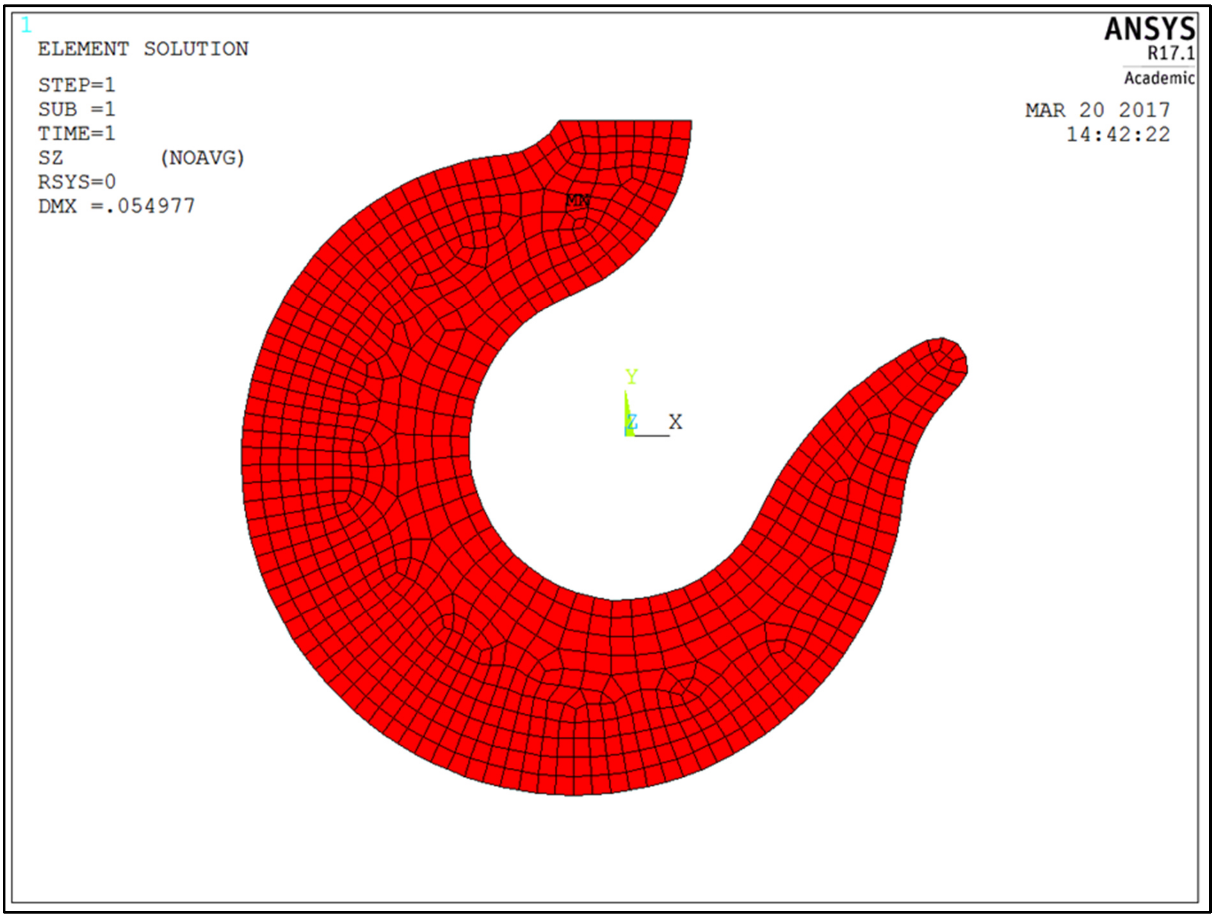

Figure 13: Element solution - Z-Component of stress

4.2.4. Von Mises Stress.

In this subchapter, after the stress analysis in all degrees of freedom, the more important stress analysis, called the von Mises stress, will be described. The von Mises stress, also known as equivalent stress considers the effect of all the stresses which operate in the degrees of freedom. It combines all these stresses to one value, the most important value for designing engineers. Following analysis gives a statement whether the model will fail concerning of too high stresses which exert an influence on the components. In other words, the material starts to yield at a point where the von Mises stress becomes equal or exceed the yield strength. In this case, through exceeding the yield strength, the material leaves the elastic range and starts to yield plastically. (Tickoo, 2012) (Kurowski, 2012)

As described in the previous subchapters, for the nodal solution of von Mises stress, the area (1) in Figure 14 and its maximum value of 87.2 N/mm^2 can also be neglected because of the applied load and previous assumption. Therefore, the maximal stress value occurs around the area (2) with approximate values between 47.9 N/mm^2 and 57.5 N/mm^2. Additionally, through a detailed analysis and list results, the highest stress value with 56.9 N/mm^2 occurs at node number 206. Moreover, in the area (3) stresses with 19.1 N/mm^2 can be read out, just like in area (4) with a small amount of 9.6 N/mm². More important are the dark blue areas, like area (1), where no stress occurs or rather approached zero.

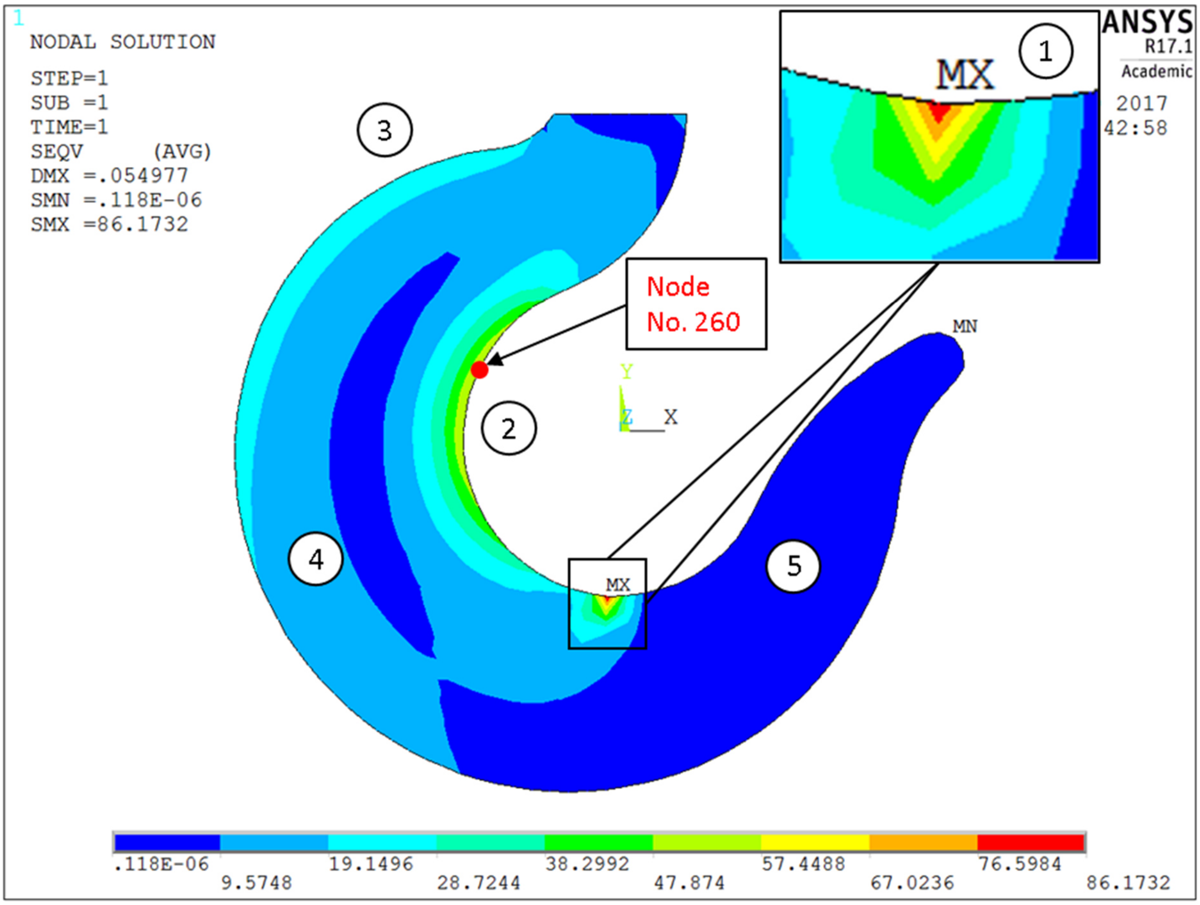

Figure 14: Nodal solution - Von Mises stress

First impressions about the element solution in Figure 15 are, that the stresses are higher than in the nodal solution. However, this can be explained by the different calculated colour scales. Closure inspection shows that the stress values and the areas they occur are almost the same. This means, that the same findings than in Chapter 4.2.1 apply. Nevertheless, in this case, the nodal solution gives a more detailed course of the stresses.

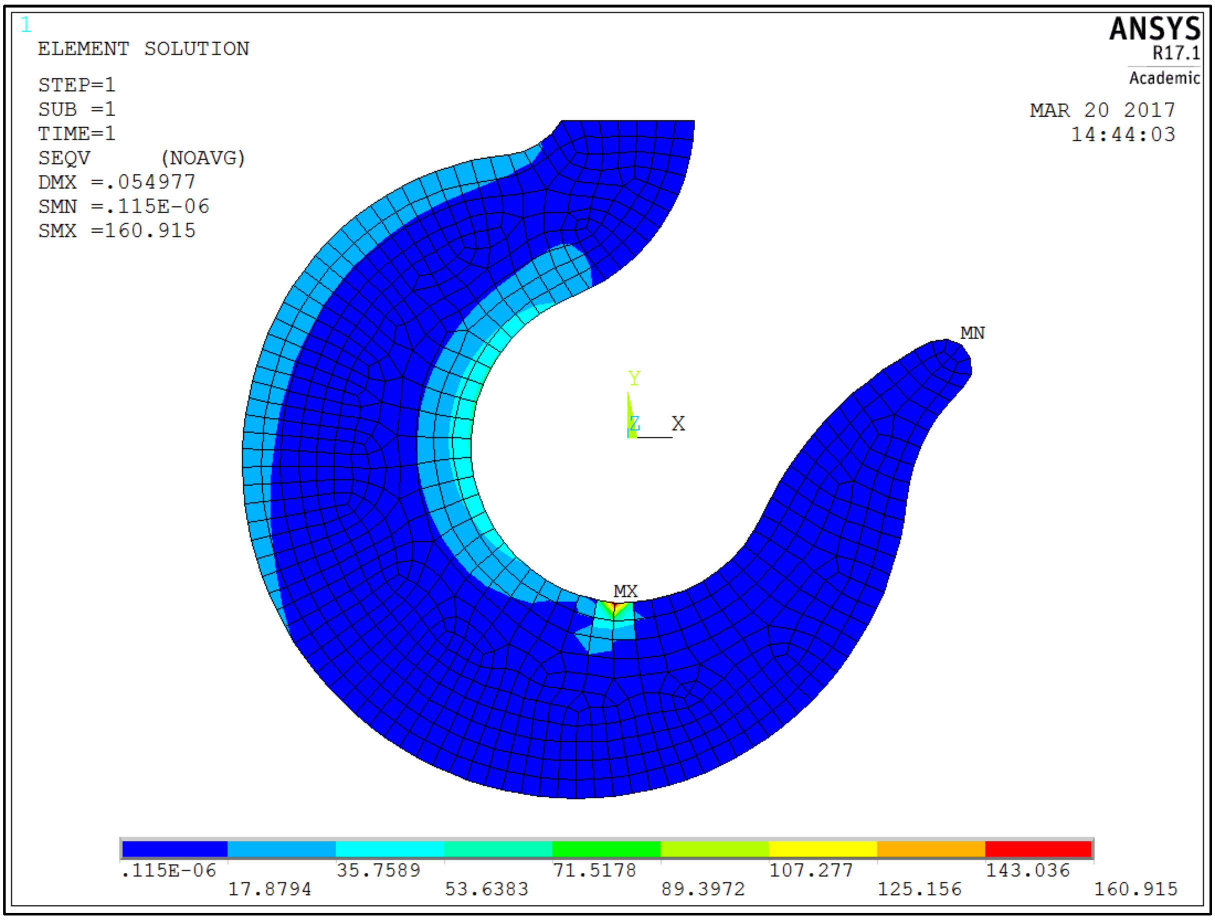

Figure 15: Element solution - Von Mises stress

5. Discussion

After evaluating the results in Chapter 4, they built the basic to substantiate the following discussion. First of all, the focus concentrates on the calculated and simulated results together with their interpretation. Afterwards, the discussion goes on with the second research question to analyse the three-dimensional cross-section for the chosen hook in Figure 1. Finally, the chapter ends with determined considerable limitations.

5.1. Results Verification

The first basic method to ensure the quality of the finite element analysis is to check the deformation with an objective view. Figure 7 shows the maximum displacement with a value of 0.055 mm. According to the applied load of 1182 N and the appointed boundary conditions, the hook deforms primarily in the y-direction and less in the x-direction. This solution leads to the realisation that the hook acts as expected and in a reasonable way. In addition, it is important to mention that the displacement of 0.055 mm is very small and almost negligible.

Furthermore, an objective view onto the stress results is important to perform a plausibility check. The stress analysis in the degrees of freedom show in general tensile areas on the inner radius and a compression area on the outer radius. Such tensile stresses and compression stresses result in bending, which is negligible with those material properties and the relatively small applied load. To sum up, it can be said that the stresses proceed as expected.

At least to verify the results, the mesh density has to be examined as described in Chapter **Fehler! Verweisquelle konnte nicht gefunden werden.**. Therefore, it can be assumed that the mesh size is sufficient and the simulation is correct. This is attributed to the almost similar values between the nodal and element solutions of the x-component of stress, y-component of stress and von Mises stress. If the mesh size would be too coarse, the rigidity increases and reduces the results.

5.2. Factor of safety

To look closer at the results of the von Mises stress, the appliance of the factor of safety is an appropriate approach. The factor of safety is a measurement to indicate that a structure resists loads. In other words, it displays a comparison of actual strength to the required strength. (Gere & Goodno, 2012) Following formula is used to calculate the factor of safety:

$$Factor\ of\ safety\ (FOS) = \frac{Yield\ strenght}{working\ stress}$$

The factor of safety is calculated with the evaluated maximal von Mises stress of 56.9 N/mm^2 in node 260 and the given elastic strength of 620MN/m^2 which is equal to 620 N/mm^2.

$$Factor\ of\ safety\ (FOS) = \frac{620\ ^{N}/_{mm^2}}{56.9\ ^{N}/_{mm^2}} = 10.9$$

Due to the calculated factor of safety with 10.9, it seems very high on the first view. Based on the stress analysis and the factor of safety, it can be concluded that the hook is over-designed for this case of application. Moreover, it is important to mention that in practical industrial applications usually a factor of safety of 5 is defined for lifting hooks. (North Sea Lifting Limited, 2014)

As a consequence of the over-designed hook, some optimisations can be considered. First of all, by maintaining these initial material properties and design, the hook can be used for higher loads. Therefore, the hook is more high-grade than used in this simulation. Secondly, by leaving the load by the same value of 1182 N, the hook can be made of another material with lower properties (e.g. structural steel S355 with a yield strength of 345 N/mm^2). (MEADinfo, 2015) Furthermore, by leaving the load by the same value again, through modifications of the hook design it can be saved up material or make it lighter. This can be relocated through removing material, especially in the area (4) on Figure 14, where the von Mises stress is against zero and do not affect the usage negatively. Lastly, the thickness of the hook can be reduced for the same intended use and load. For example, by reducing the thickness of the hook by 50% to 5 mm the maximum von Mises stress that occurs is around 114.9 N/mm^2. This calculation is illustrated in the following Figure 16. Additionally, a factor of safety with 5.4 can be achieved, which is sufficient according to the safety regulations for lifting hooks. In this respect, it can be saved 50% of the material.

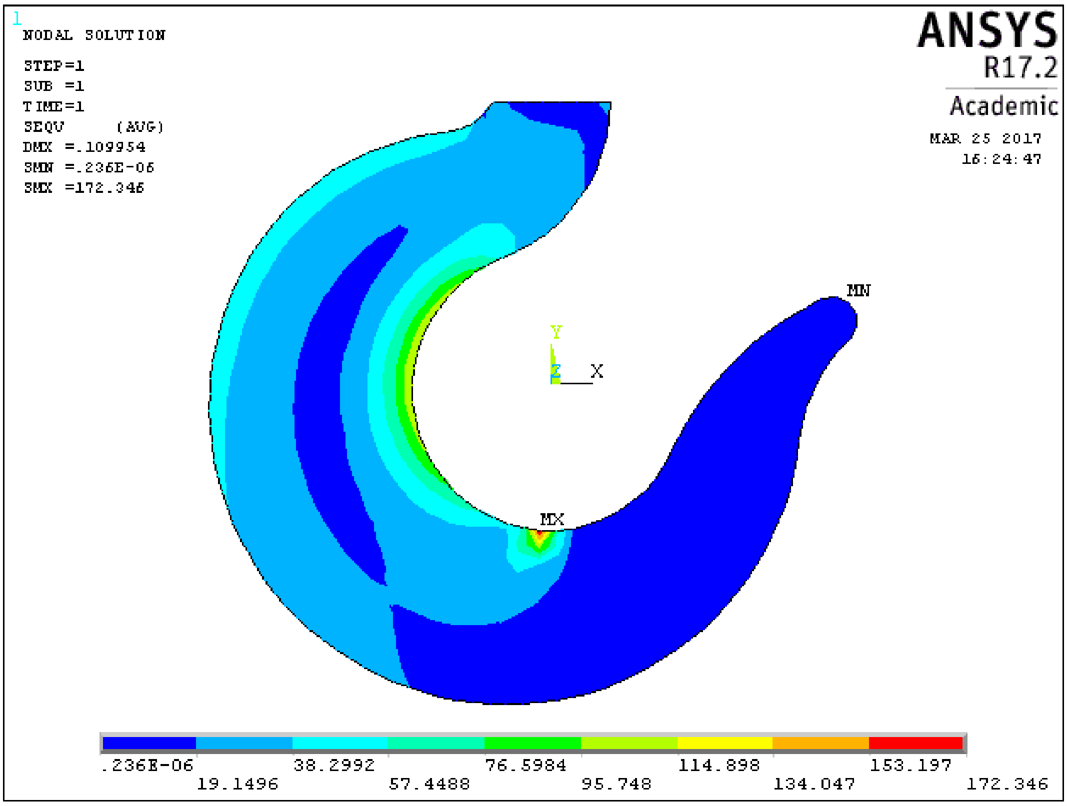

Figure 16: Von Mises stress analysis after reducing the thickness

5.3. Shape of the hook

This subchapter discusses the second research question to analyse the justification for the three-dimensional cross-section of the chosen hook. This investigation is based on the insights out of the finite element analysis. Figure 17 illustrates the different analysed sections of the hook. Thus, hooks, like crane hooks usually have a trapezoidal cross-section (1), which is wider on the inside and narrower on the outside. The wider inner side, thereof, is wider to carry the load of ropes or chains satisfactory. Furthermore, it can be inferred that through the rounded corners friction will be reduced and for example, the rope becomes protected. In addition, to design a wider inner surface means that the load can be better distributed instead of a narrow inside. A trapezoidal cross-section also provides a better material utilisation. As described in the subchapter above based on the von Mises stress plot, it is visible that there are some certain areas (2) with a very low stress level, where the shape becomes narrower. Besides, the narrow outside can be justified by a lower compression stress (3) compared with the wide inside (4) where a higher tensile stress occurs. This results in higher stress on the inside than the outside and finally deformation, even if the deformation is small and within the elastic region. Another important point is that the outer part (5) of the hook, with very slight stress, is eventually just to make sure to keep the caring load safe in the hook. Therefore, this indicates that the thickness of this part is much smaller and less material. Furthermore, the highest deformation in this part of the hook with 0.055 mm specifies the less importance concerning mechanical strength.

In addition to this subchapter, it should be noted that besides the trapezoidal shape are some other cross-section used in industrial applications like circular, rectangular and triangular cross-section. (Lanjekar & Patil, 2016)

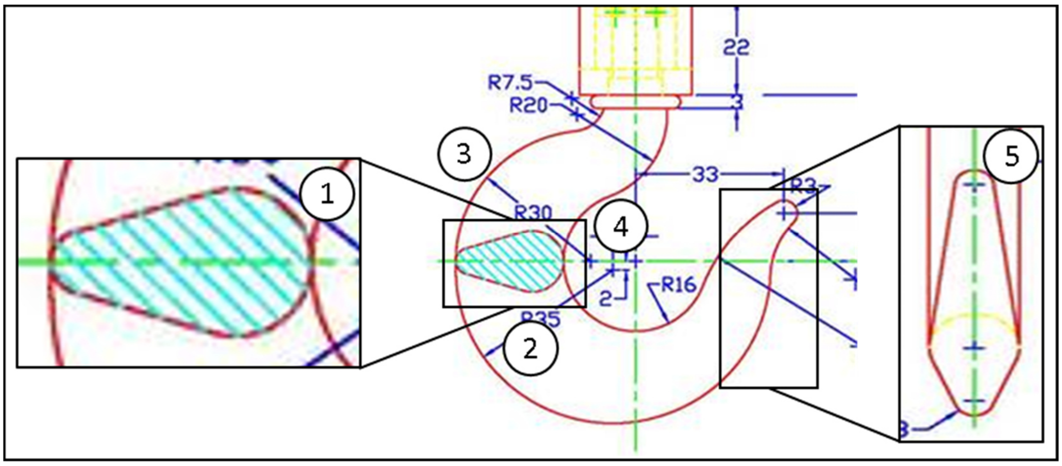

Figure 17: Different analysed sections of the hook

5.4. Limitations

The facts presented in this chapter describe limitations as a result of the performed finite element analysis in the previous chapters. First of all, it should be observed that the calculation and results are confined by the described assumptions in Chapter **Fehler! Verweisquelle konnte nicht gefunden werden.**. These assumptions relate the total model and the constitutive analysis. In addition, generally, they can also be seen as limitations. Furthermore, the finite element analysis and results only depict a two-dimensional analysis which does not represent the real conditions with at least three-dimensions. The modelled hook was set up very simple for simulation purposes whereby the results are very limited. For example, instead of a trapezoidal shape, the simulation used a rectangular shape to model the problem. Another serious limitation is that the applied load only acts punctual on one node and not on an area. An area would represent a rope or chain much better. This leads to inaccurate stresses around the node, such stresses would course in reality different, like on more elements. Based on this experience, further studies should be optimised in that respect. Furthermore, the punctual load on one node caused negative scaling of the colour scale, e.g. for the von Mises stress analysis. Additionally, material properties cannot be replicated exactly by physical properties without any defects. In terms of the limitations, it should also be remembered that the results with the finite element software ANSYS do not reflect the reality by one hundred percent. Due to the finite element analysis, it is only an approximate result and not the reality. This numeric procedure results in deviations from reality, as mentioned above. For example, elements in the simulation are always an average calculation. Finally, besides the advantages of finite element analysis like saving money and time, it does not replace real practical tests because they always show the reality. Those mentioned practical tests may be carried out as a realistic test on cranes or with testing equipment like a tensile testing machine. In this sense, a good method to verify and confirm a finite element model is to compare the simulation results with results of conducted practical tests.

6. Conclusions

Finite element analysis is a helpful method to analyse complex engineering designs or problems. With respect to this fact, first of all, a finite element analysis with ANSYS was performed regarding the central task of the assignment. Based on assumptions like the simplification from three-dimensional to two-dimensional, the design of a simple hook has been modelled. In addition to the mentioned assumptions, the material properties, boundary conditions and applied load were also indispensable conditions.

The results first return the calculated displacement of the hook under the load of 1182 N, which results in a maximum displacement of 0.055 mm. Due to the area of application and the relatively small value, it can be almost neglected. Further important results are the stress analysis within all degrees of freedom (x-and y-direction) and particular the von Mises stresses. On the one hand with the stress curve over the hook, it was illustrated that the causing stresses lead to tensile stress on the outer radius, compression stress on the inner radius and slight stresses in the outer hook part. Moreover, the highest von Mises stress occurs with 56.9 N/mm^2.

The situation became even more clear when the stresses were compared with the material properties and the calculated factor of safety of 10.9. It can, therefore, be concluded that total over-engineering exists, regarding the applied load. Furthermore, this knowledge should be observed more closely with further simulations in this field.

Secondly, besides the simulation, another focus was the analysis of the three-dimensional cross-section used for the hook, which is predominantly based on the stress analysis. The main developed characteristic was, that the trapezoidal cross-section is useful because of the occurred tensile and compressive stresses. Furthermore, it was noticeable that on the outer side slight stresses occur, just to make sure to keep the suspended load safe in the hook. Moreover, material utilisation is another point regarding the hook shape and its trapezoidal cross-section. Nevertheless, it was impossible to find additional insights out of literature besides the developed results.

Last but not least, it should be noted that finite element analysis does not replace practical tests under real conditions. However, it is a very fast and adaptable method to analyse complex engineering designs and to save money as well as time.

7. References

3DVision Technologies, 2008. *3DVision Technologies.* [Online] Available at: https://www.3dvision.com/blog/entry/2008/04/18/nodal-versus-elemental-stresses.html
[Accessed 22 March 2017].

Abbey, T., 2015. *Digital Engineering.* [Online] Available at: http://www.digitaleng.news/de/simplifying-fea-models-plane-stress-and-plane-strain/
[Accessed 20 March 2017].

Chen, X. & Liu, Y., 2014. *Finite Element Modeling and Simulation with ANSYS Workbench.* 1. ed. Boca Raton: Taylor & Francis Group, LLC.

Cunningham, P., 2014. *CAE Associates.* [Online] Available at: https://caeai.com/sites/default/files/CAEA_Plastics_Demo.pdf
[Accessed 20 March 2017].

Devaraj, A., 2015. Design of a Crane Hook of Different Materials and Stress Analysis Using ANSYS Workbench. *International Journal for Research in Applied Science & Engineering Technology*, July, pp. 310-314.

Fetvaci, M. C., Gerdemeli, I. & Erdil, A. B., 2006. FINITE ELEMENT MODELLING AND STATIC STRESS ANALYSIS OF SIMPLE HOOKS. *Trends in the Development of Machinery and Associated Technology*, 15 September, pp. 797-800.

Gere, J. M. & Goodno, B. J., 2012. *Mechanics of Materials.* 8. ed. Stamford: Cengage Learning.

Gordon, D. R. S., 1994. *ITTI Finite Element Training Project - Static I: Two Diemensional Analysis.* Manchester: Manchester Computer UMIST Support Unit.

Kurowski, P. M., 2012. *What is calculated in FEA?,* s.l.: s.n.

Lanjekar, K. & Patil, A. N., 2016. Weight Optimization of Laminated Hook by Topological Approach. *IOSR Journal of Mechanical and Civil Engineering*, October, pp. 7-20.

MEADinfo, 2015. *Mechanical Engineer's Information Hub.* [Online] Available at: http://www.meadinfo.org/2015/08/s355-steel-properties.html
[Accessed 27 March 2017].

Modlen, D. G., 2008. *Introduction to Finite Element Analysis,* Manchester: Manchester Computer Centre UMIST Support Unit.

North Sea Lifting Limited, 2014. *NSL.* [Online] Available at: http://nsl.ascoworld.com/sites/nsl/files/documents/loler_matrix_2014_colour.pdf [Accessed 25 March 2017].

SAS IP, Inc., 2016. *ANSYS Help Viewer - PLANE183 Element Description,* s.l.: s.n.

Thakur, V. M. et al., 2016. Study and Analysis of Crane Hook in Loading Area. *International Journal of Advanced Research in Education & Technology*, January, pp. 88-89.

Tickoo, S., 2012. *ANSYS Workbench 14.0: A Tutorial Approach.* 1. ed. Schererville: CADCIM Technologies.