# Numerical assignments using FEM

Solid mechanics with FEM (1MI170), VT 2024

Uppsala universitet
Institutionen för samhällsbyggnad och industriell teknik
Hållfasthetslära med FEM 5 hp
(2024/VT)
Författare: Kalle Axelsson

# Innehåll

# Numerical assignment 1

A cylindrical pressure vessel (pipe) intended for an internal pressure of 1 MPa is to be manufactured from sheet metal with a thickness of 1 mm. This vessel is clamped between two rigid walls. The Poisson's ratio of the metal is $\nu = 0.3$ and the elastic modulus is E=200 GPa, the yield strength of the material (flow stress) is$\sigma_y = 1000$MPa. What is the effective von Mises stress if the radius of the vessel is 50mm? You can also define your own material by selecting different $\nu$, $E$ and $\sigma_y$ .

1) Give the theoretical solution to the question above.
2) Simulated the problem with FEM in Solidworks.
3) How well do the numerical solution and theoretical solution match? Give detailed discussion. (Tips: a. the theoretical solution was derived assuming an infinitely long pipe; b. the level of matching depends on the length of the pipe you model and the position you examine.)

As for calculations, the following is the method for calculating the 3 principle stresses in a thin walled pressure vessel:

$$\sigma_\varphi = \frac{pa}{h}$$

$$\sigma_r \approx 0$$

$$\varepsilon_z = 0$$

$$\varepsilon_z = \frac{1}{E}[\sigma_z - v(\sigma_\varphi + \sigma_r) = 0$$

$$\sigma_z = v\frac{pa}{h}$$

$$\sigma_1 = \sigma_\varphi\,, \qquad \sigma_2 = \sigma_z\,, \qquad \sigma_3 = \sigma_r \approx 0$$

With these, the effective Von Mises stress can be calculated,

$$\sigma_e^{VM} = \frac{1}{\sqrt{2}}\sqrt{(\sigma_1 - \sigma_2)^2 + (\sigma_1 - \sigma_3)^2 + (\sigma_2 - \sigma_3)^2}$$

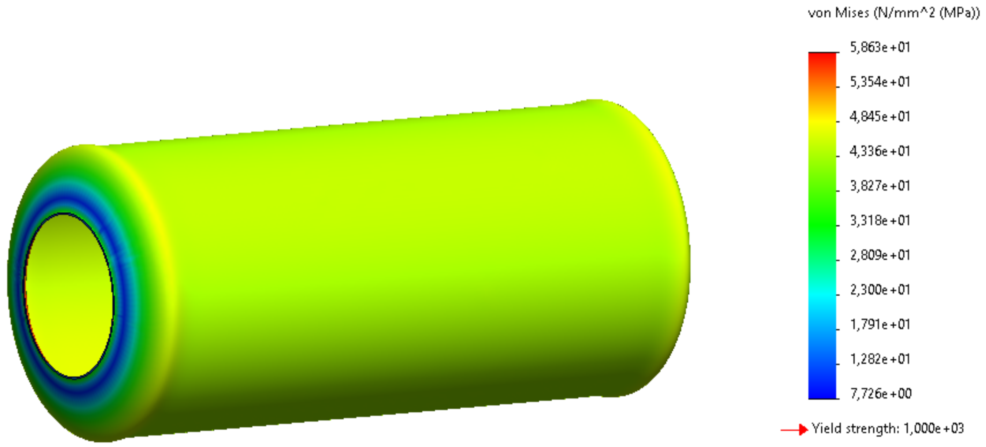
$$= \sqrt{\sigma_\varphi^2 + \sigma_z^2 - \sigma_\varphi\sigma_z}$$

$$= \frac{pa}{h}\sqrt{1 - v + v^2}$$

$$= \frac{1MPa * 50mm}{1mm}\sqrt{1 - 0.3 + 0.3^2}$$

$$= 44.44\ MPa$$

For the pipe, a simple 100 mm long, 100 mm diameter and 1 mm wall thickness pipe was designed. The vessel was in the simulation tool clamped between to walls and a 1 MPa pressure was applied along the inside.



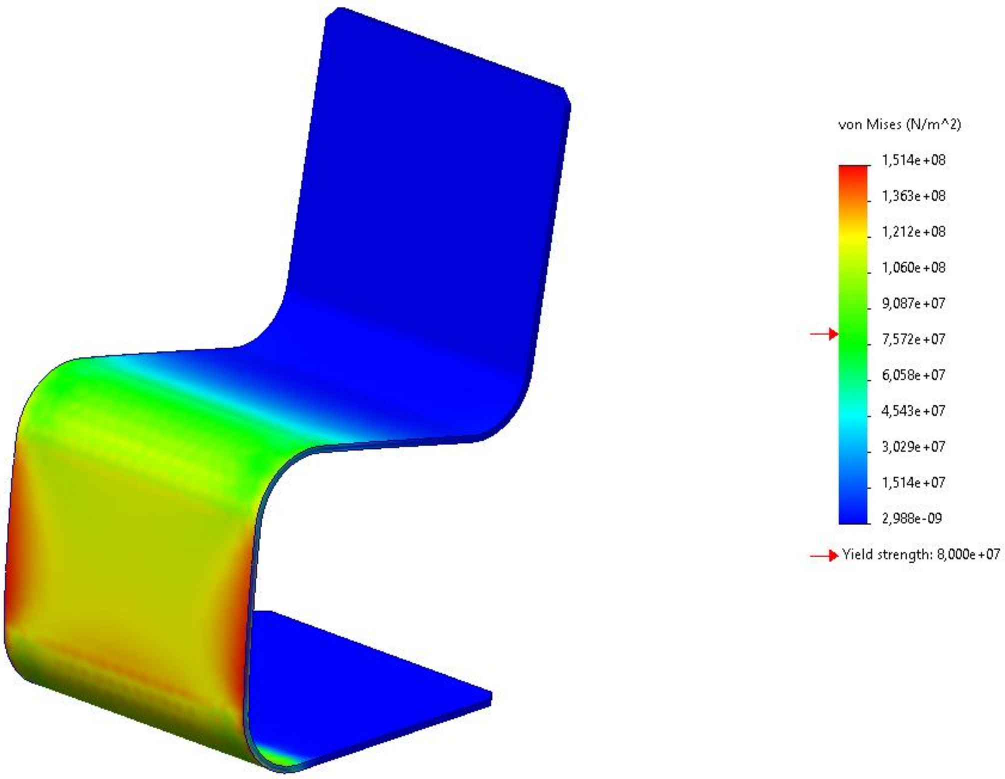

By probing the middle section of the pipe, an average of 45.04 MPa was measured. This value was very close to the theoretical solution. Where it does differ is at the maximum measured values, which in the simulation measured up to 58.36 MPa, much higher than the theoretical solution. These high stress points were found at the edges of the pipe and may lead to failure if not taken in to account while designing the pipe.

# Numerical assignment 2

A chair is constructed from aluminum plate. The elastic modulus and Poisson's ratio of the material are $E = 70\text{ GPa},\ \nu = 0.32$. The dimensions of the product are detailed in the chart attached in the next page. Use FEM to help solve the following questions:

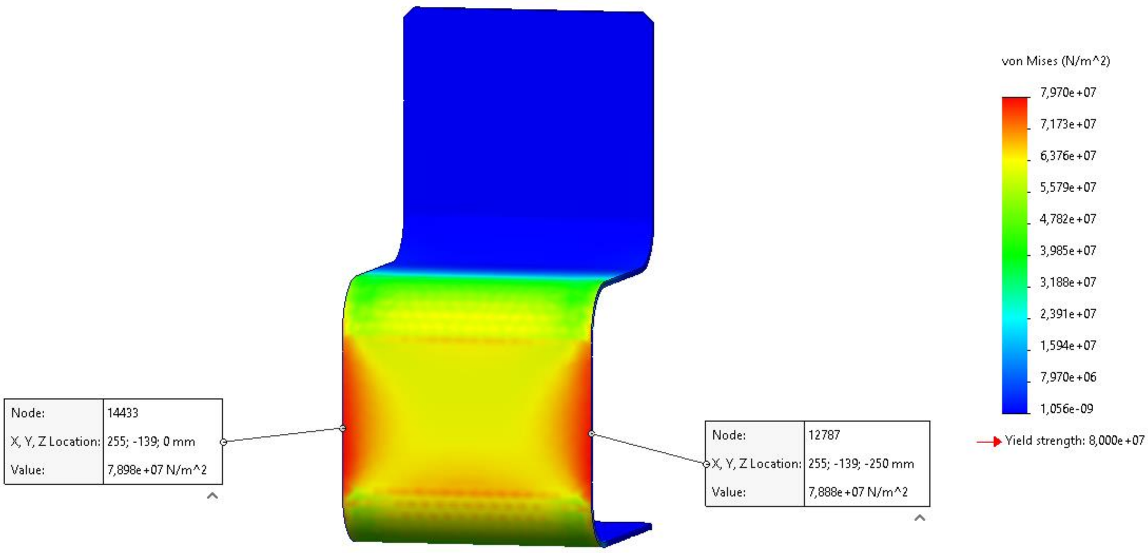
1. Suppose a person weighing 100kg sits on the chair, calculate the largest von Mises effective stress that arises in the chair.
2. Suppose the strength of the material is 80 MPa. Determine (approximately) the maximum weight of a person that can safely sit on the chair. If the chair is to fail, where does failure initiate?
3. You now want to change the thickness of the aluminum plate to save material and reduce cost, and the thickness of the plate is halved everywhere. What is the maximum weight the chair can bear now?
4. Suggest a better way of designing, so that the chair can bear a similar maximum weight as in question 2 but uses less material.

First, using simulation to find the largest von mises stress:

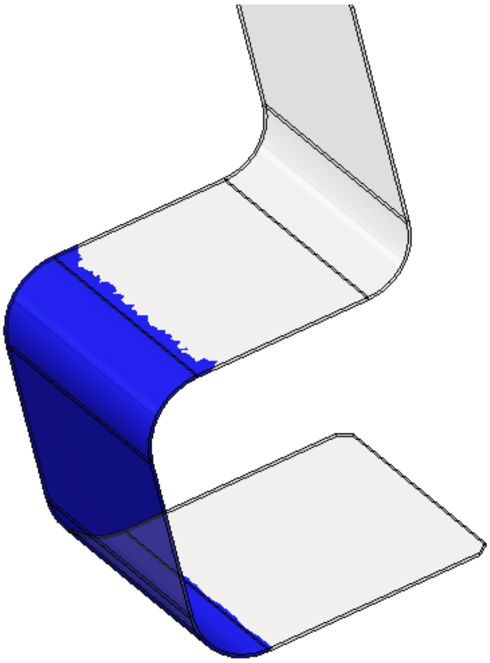


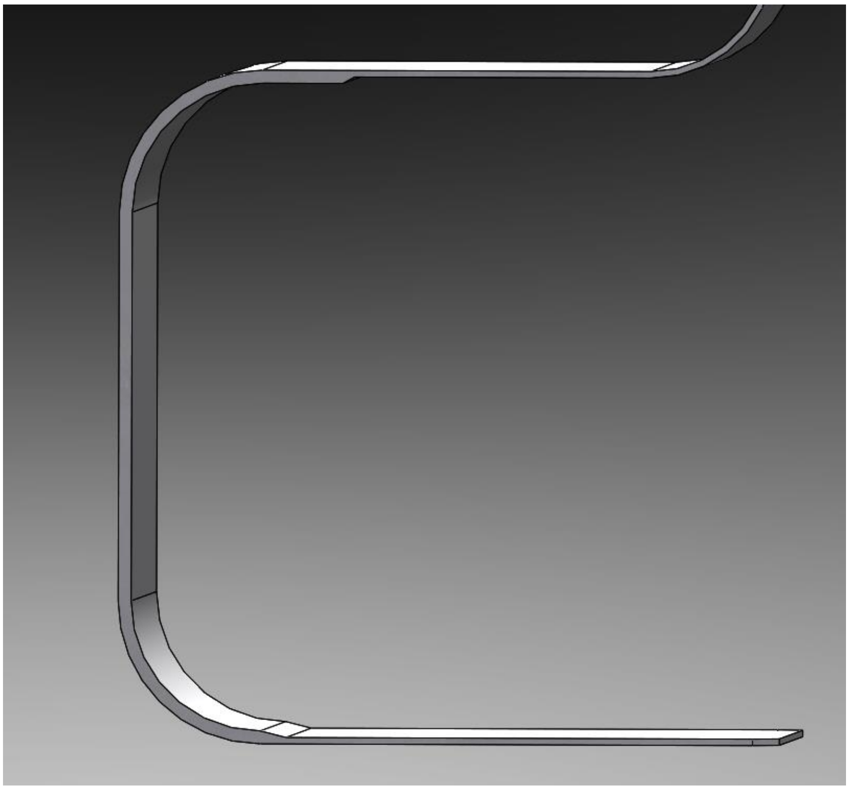

If a 100kg person sits on the chair, the largest von mises effective stress is 151 MPa. If the strength of the material is 80 MPa, the maximum safe force would be 550N, which translates to 56 kg. The failure would occur first at either side of the bottom half, shown in red.

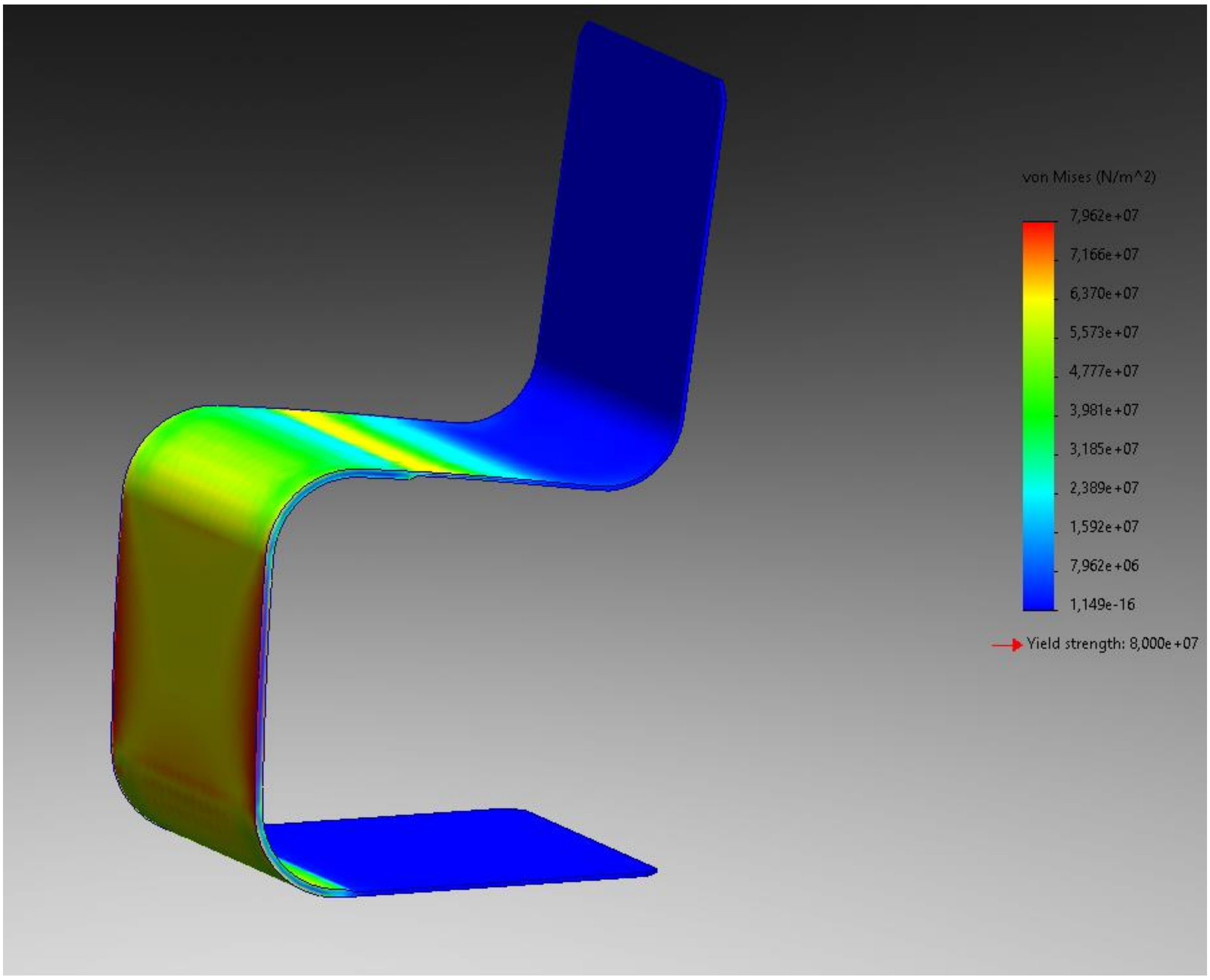
When halving the thickness of the chair, the largest safe force is 125N. that is equivalent to 12.73 kg



It is obvious where the largest load is when using the *Design Insight* tool. Meanwhile, the loading on the non-blue marked area is close to none. By reinforcing the area experiencing all of the load, the rest of the chair can remain with a thickness of 2.5mm.








which brings down the weight from 3134g to 2202g, 70% of the original value

# Numerical assignment 3

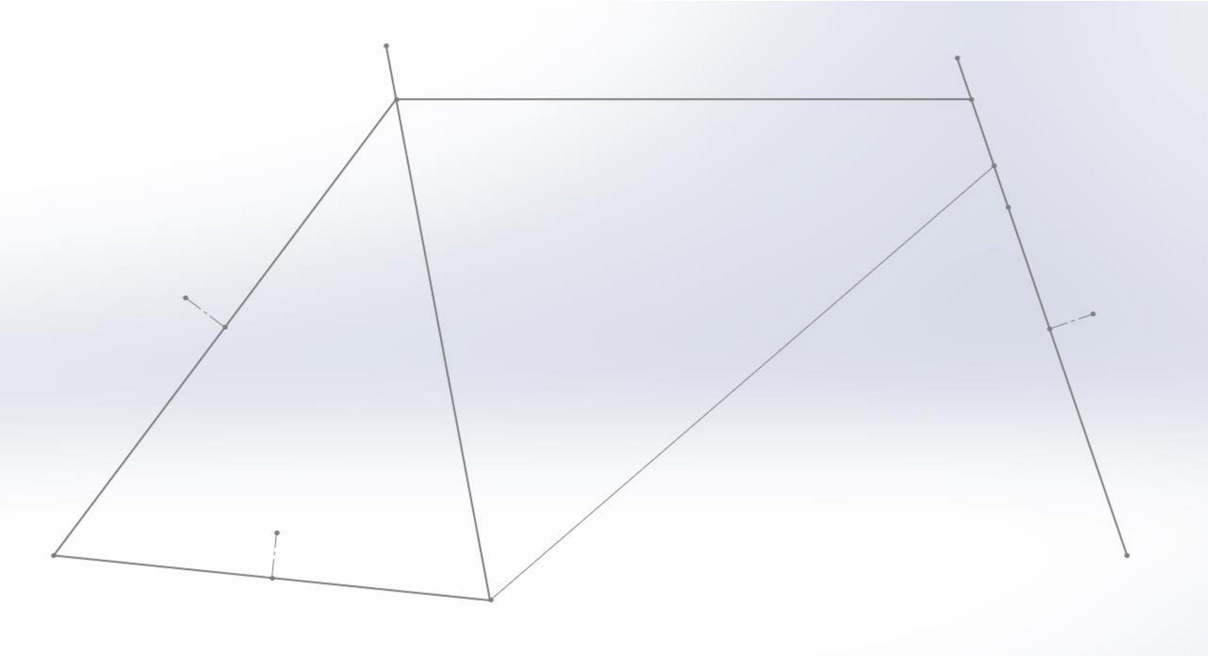
A bicycle frame is to be manufactured from Aluminum. The elastic modulus and Poisson's ratio of the material are $E = 70$ GPa and $\nu = 0.3$, the strength is $\sigma_f = 80$MPa and the density is $2700\text{kg/m}^3$. The frame shall be able to bear a weight of $m = 150$kg in the vertical direction when it is in use, and it can be assumed that this total weight is distributed as $P_1 = 0.4m$, $P_2 = 0.3m$, $P_3 = 0.3m$, as shown in the figure below.

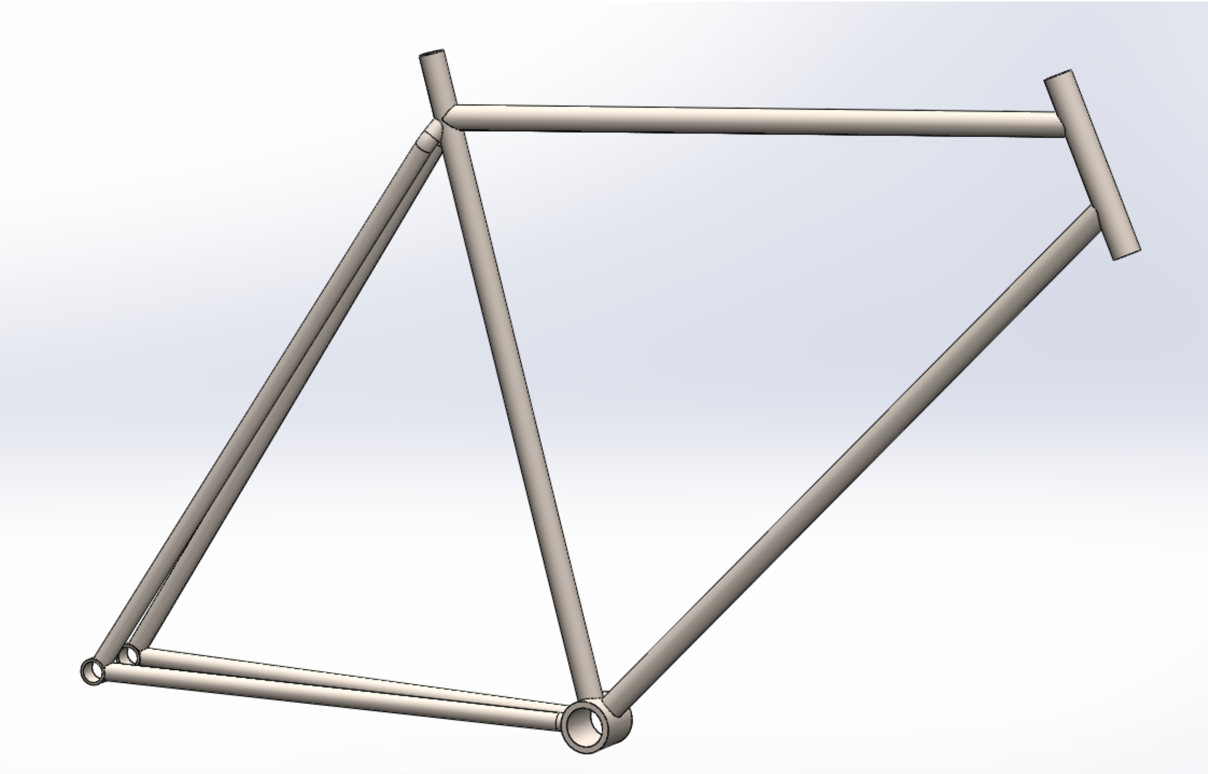
The key components of the frame are aluminum tubes with various wall thicknesses and radii. The lengths of the tubes have been specified in the figure below, where the lengths are in mm and the angle is in degrees. You are free to vary the other dimensions, e.g. the radius and wall thickness of the tubes. You can even slightly alter the shape of the tubes if you think it beneficial for the load bearing capacity.

Task: design such a bicycle frame that fulfills the load bearing requirement above and meanwhile is as light as possible.

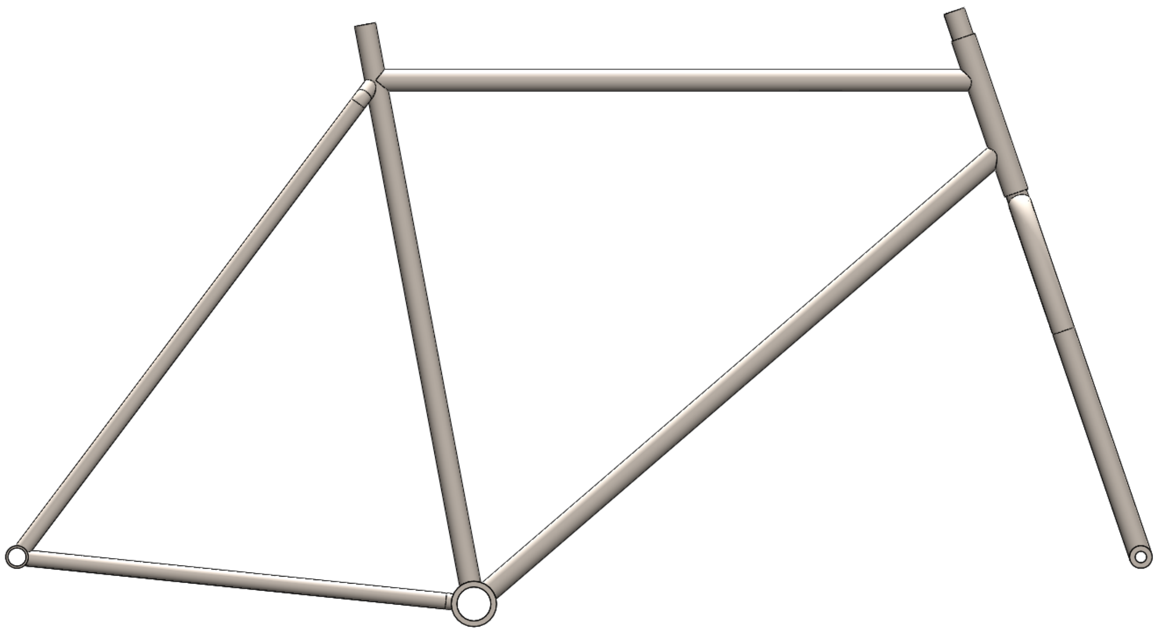
Tips: in strength analysis, the rear wheel attachment (point A) can be regarded as fixed ($u_x = u_y = 0$) while the front wheel attachment (point B) can be constrained by a roller condition so it is free in the horizontal direction ($u_y = 0$), i.e. the frame is free from external loading in the horizontal direction.

By applying the dimensions given from the assignment instructions, the following main sketch is drawn. This is what will be used as a guide for each individual tube.





The *main frame* was created by using the sweep tool, creating pipes that are 20mm wide in diameter with 2mm thick walls. Exception to this is the supporting frames for the back wheel, which are 14mm wide in diameter (but still 2mm thick).



The steering wheel frame is added, also using the sweep tool with pipes that are 20 mm wide in diameter and 2mm thickness.

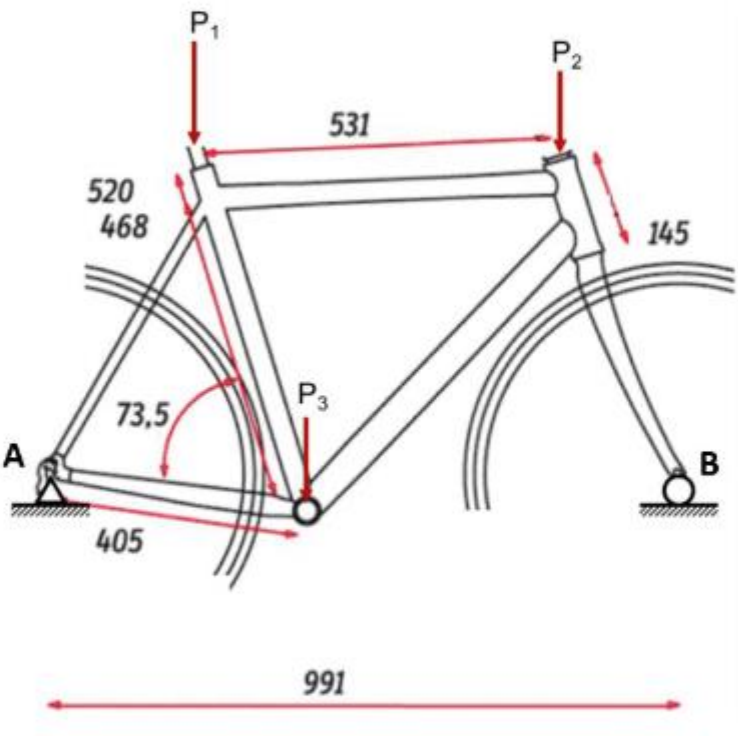
By applying the rear wheel attachment as a fixed point, the front wheel attachment as a *roller condition* and by adding three points of load as shown by the instructions, a strength analysis can be performed. The points of load are defined as vertical forces distributed as:

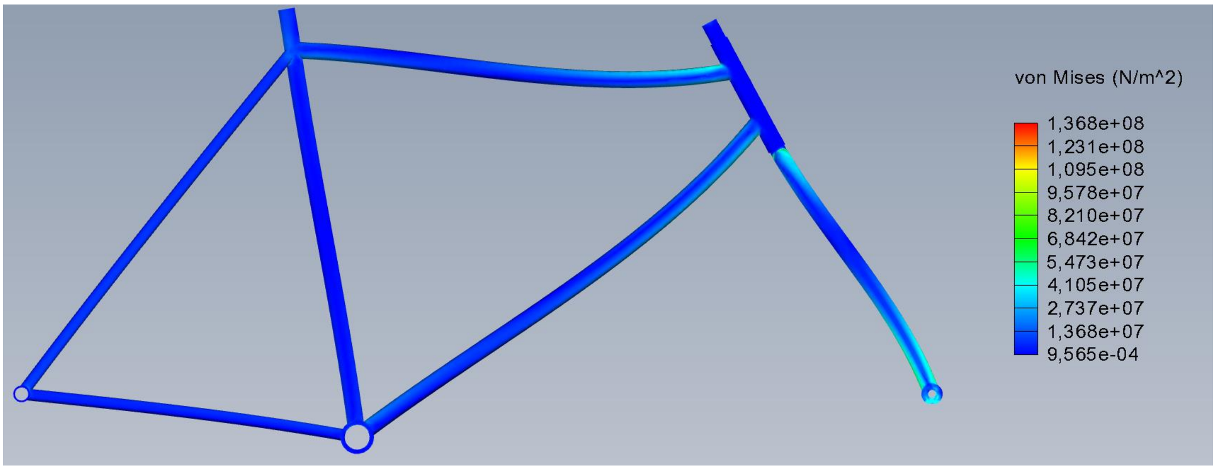
$$m = 150\ kg$$

$$P_1 = 0.4m$$

$$P_2 = 0.3m$$

$$P_3 = 0.3m$$

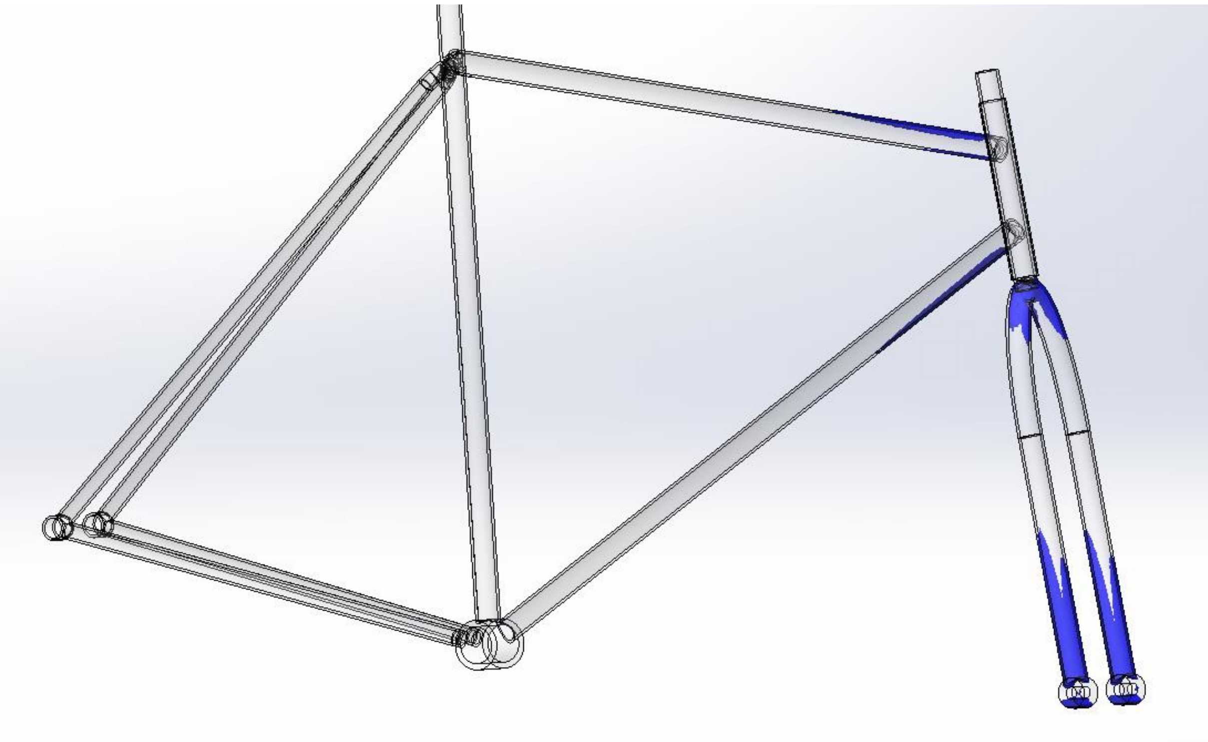



This will be used as a starting point, from which an attempt will be made to improve the design so that it can withstand the load, while minimizing mass. The current mass is 1240 g.

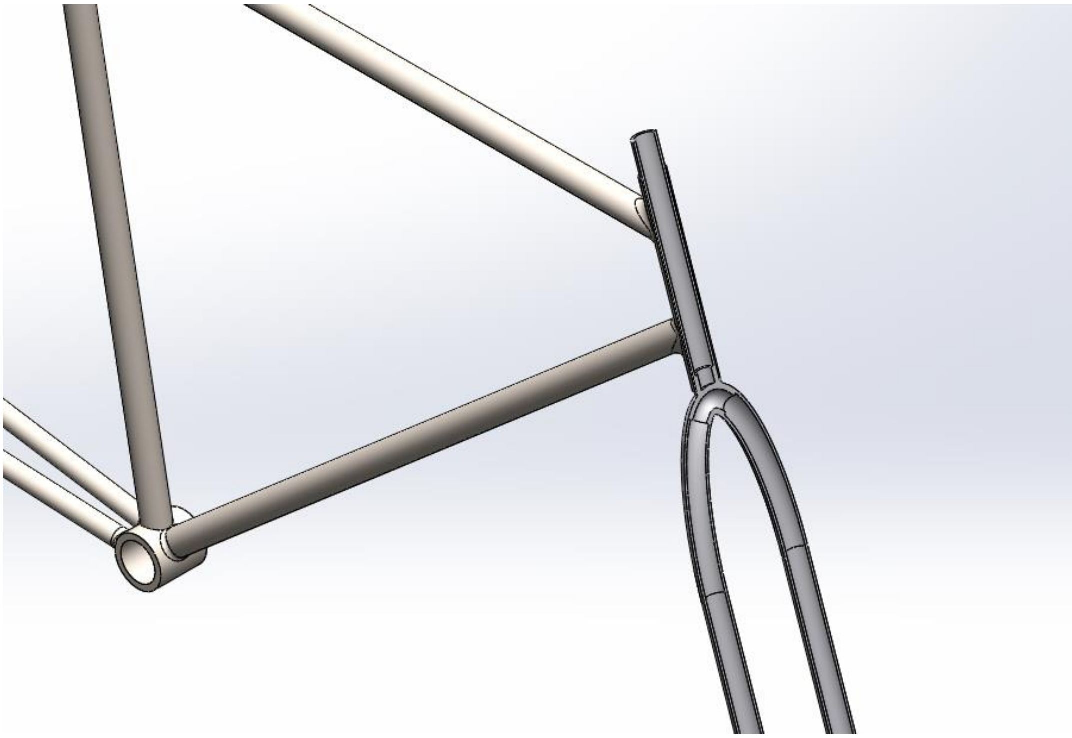
To clarify from the strength analysis, in the current design there are several points on the frame where the effective von mises stress is higher than the materials yield stress of $\sigma_f = 80MPa$. But large parts of the frame have an effective stress far below yielding, where material can be removed.

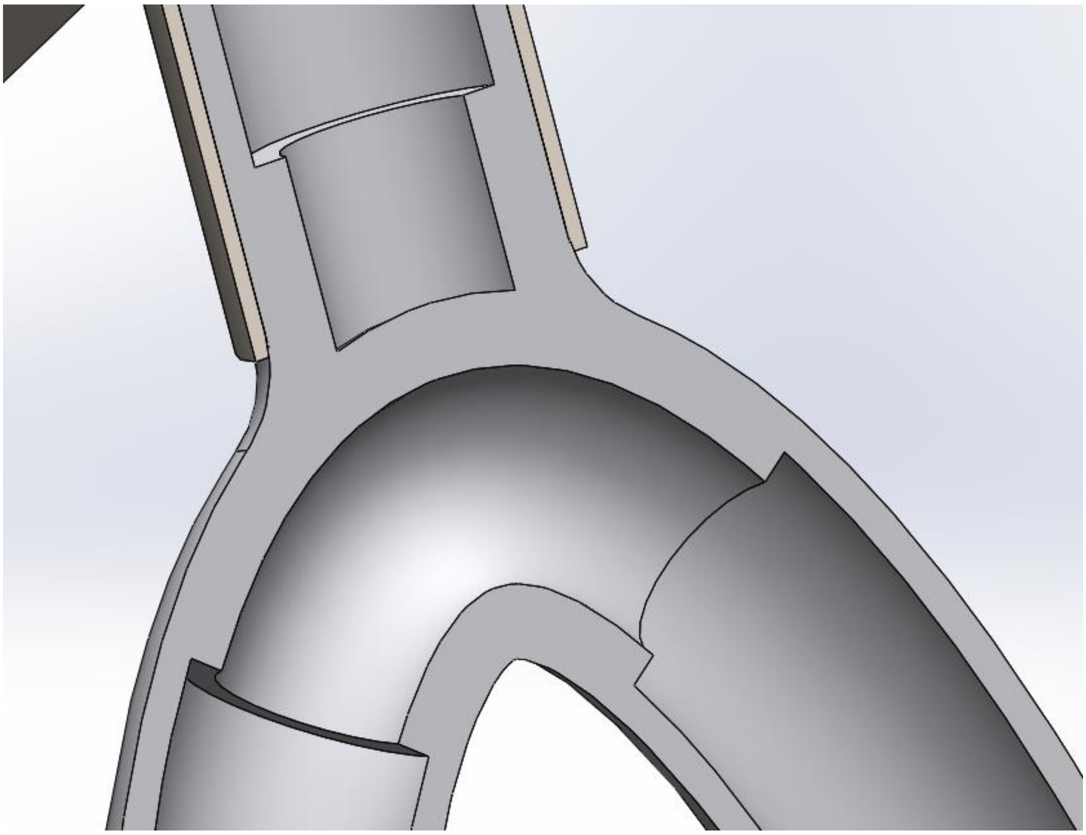
By analyzing where the stress is concentrated in the frame, and by using the solid works tool *Design Insight*, points of concentrated stress can be identified. As can be seen in the figure below those points are the front wheel frame and it's connection to the *main frame*.

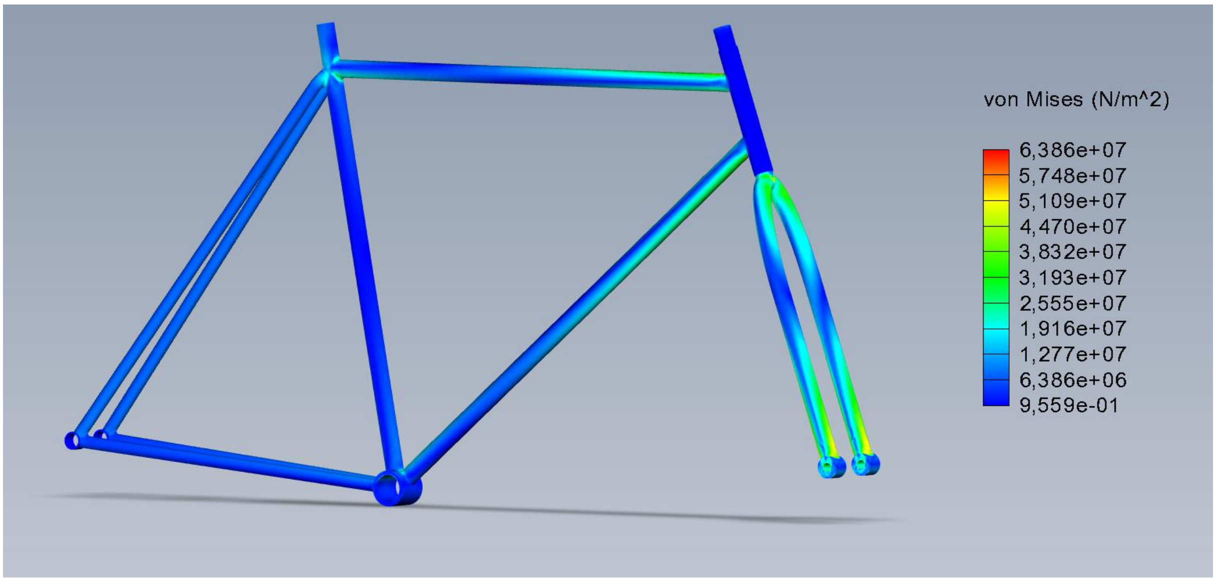


The highest stress concentration comes from sharp edges, these are dealt with by rounding the sharp edges with *fillets*. Fillets would come during production from welding the pipes together.

At the front wheel, where the steering handle meets the front wheel support, is one of the spots that need to be strengthened to withstand the load. This is done by thickening the the section with 2 additional millimeters in wall thickness. In hindsight, this would be challenging to manufacture, making the whole front wheel frame thicker would be more honest from the standpoint of *Design for Manufacturing*.



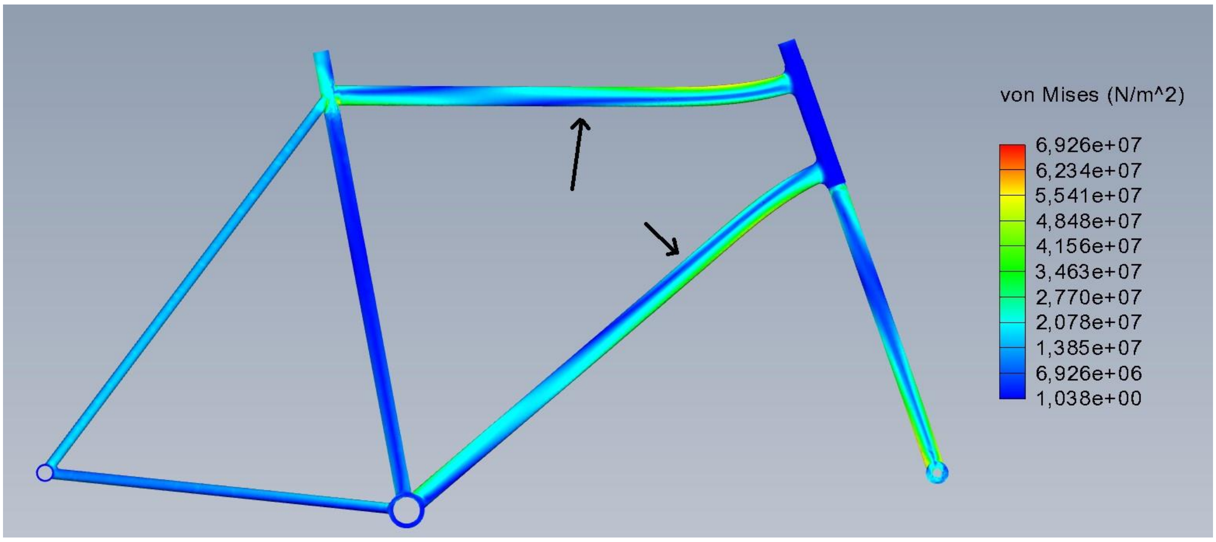




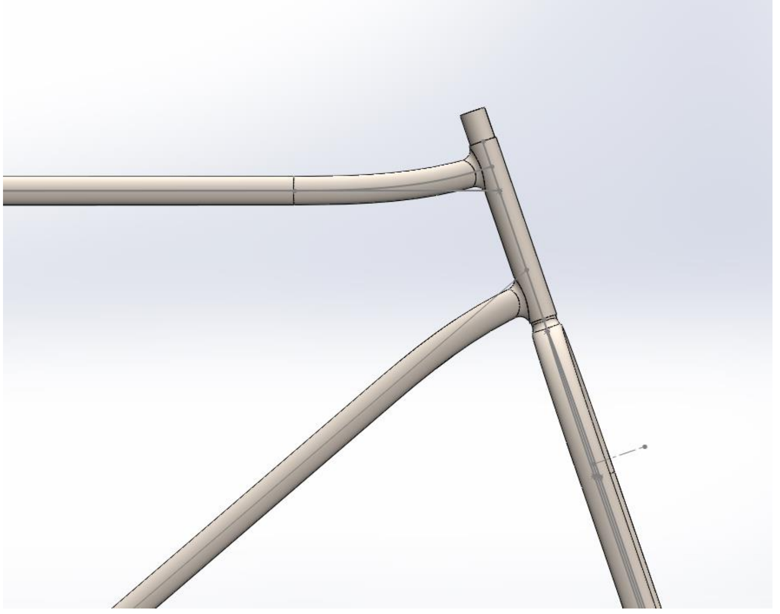

these changes, at the cost of adding 15 grams to the mass, make it so that the frame can withstand the load without reaching the yielding of 80 MPa, with the total mass being 1255g.

By using an ellipse shaped tube instead of a circular pipe in the *main frame*, can strengthen the pipe as the loading is along the x-y plane, with no or minimal load in the z-direction. By letting the ellipse tube have an increased height, but decreased width, the tube may be strong enough that the tube walls can be thinner while still maintaining enough strength.

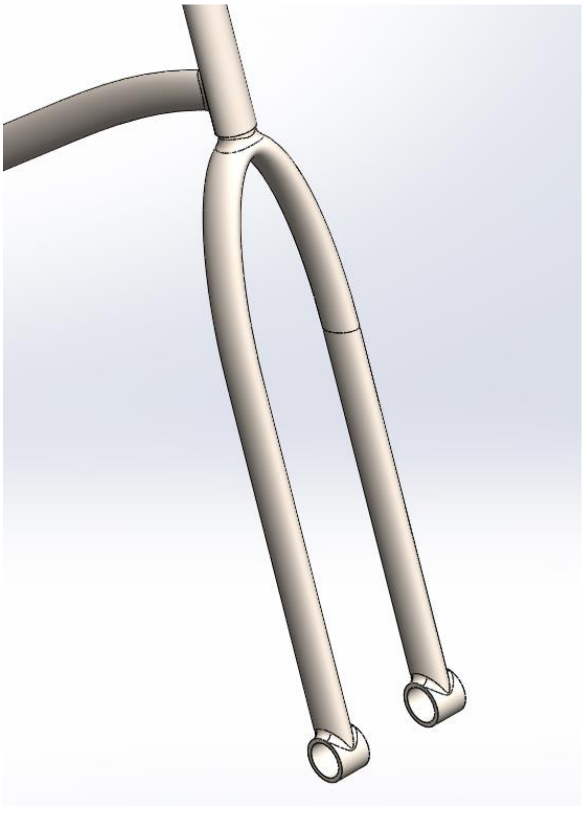
By changing the chape to an ellipse, 10mm width and 20 mm height, the wall thickness can be halved to 1mm. This was first applied to the bottom and upper tube, shown by the arrows in the following figure. This decreased the mass down to 796 grams.



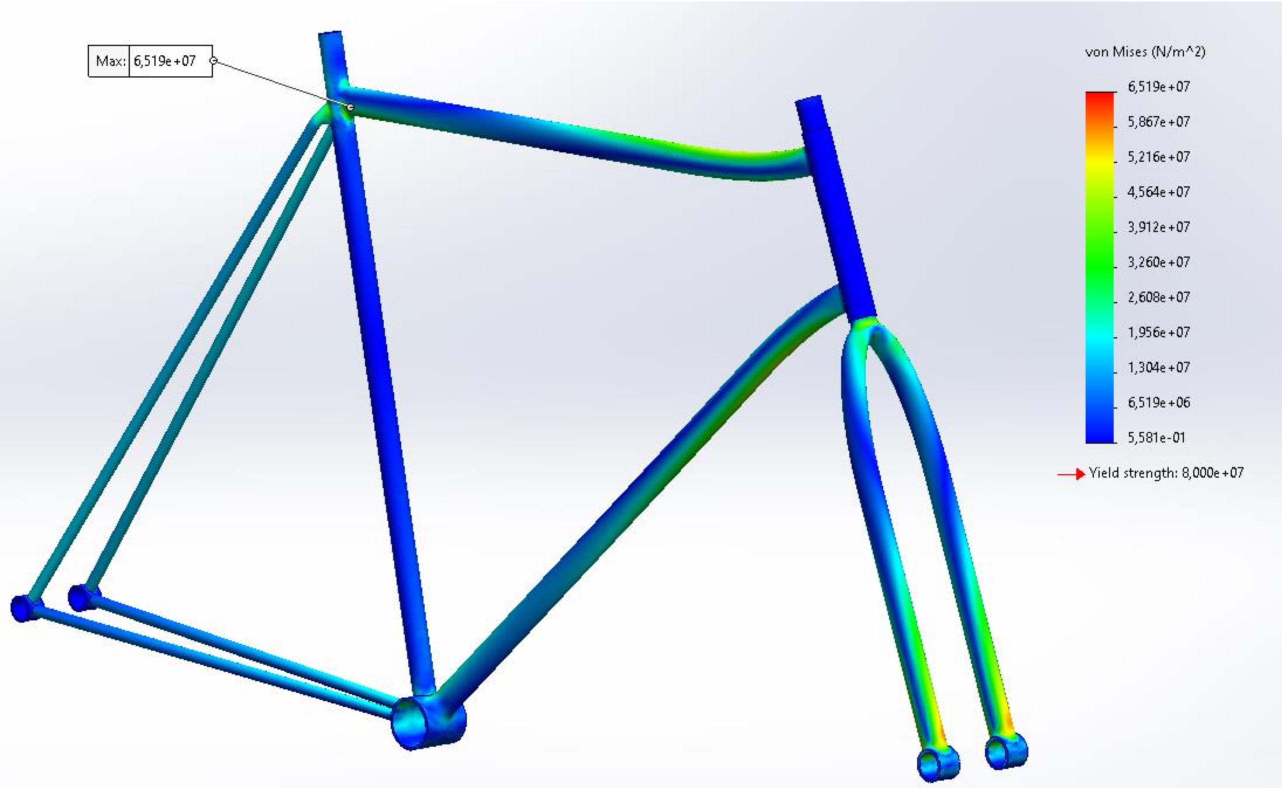
Also applied was a slight angle between the connection of the *main frame* and the front. The idea was to compensate for the load that can be seen on the top side of the upper tube, and the bottom side of the lower tube. This proved to be a positive change, and is very much doable in a manufacturing perspective, by simply bending the tubes.



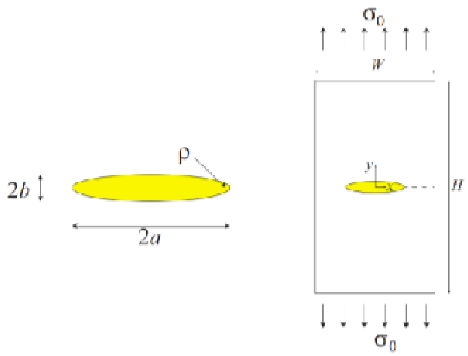
This ellipse shape of tubing was also applied to the tubes at the front, a similar 20mm height, 10mm wide shape with a 2mm wall thickness was found to be strong enough to withstand the load, while also bringing the mass down to 693 grams.



Here is a picture of the final designs simulation, with a maximum effective stress of 65.19 MPa. The final weight of the frame is 693 grams.



# Numerical assignment 4




A plate of length H and width W is under tension in the y direction, and the remote tensile stress is $\sigma_0$. At the center ($x = y = 0$) of the plate, there exists a defect which takes the shape of an ellipse with half-axes $a$ and $b$. When $a = b$, the defect is a circular hole, and when $b \to 0$, it becomes a crack.

The stress field around such an ellipsoidal defect has been derived analytically. At the tips of the ellipse, i.e. $x = \pm a, y = 0$, the stress in the loading (y) direction is approximately $\sigma_y = \sigma_0\left(1+2\sqrt{a/\rho}\right)$, where $\rho = b^2/a$ is the radius of curvature of the tip of the ellipse.

We further assume plane condition applies to this problem, and $a = W/10, H = 2W$. The elastic properties E and $\nu$ are selected by yourselves.

(1) Analyze the problem with FEM and examine how the stress $\sigma_y$ varies from the tip of the ellipse to the edge of the plate (i.e. along the dashed line in the figure, $y = 0, a \le x \le W/2$) for different ratios $b/a = \{1, 0.5, 0.25, 0\}$. For case $b/a = 0$, the defect can be modeled as a straight line/edge and corresponds to an ideal crack. Compare the FEM calculated values $\sigma_y$ at the tip of the ellipse ($x = \pm a, y = 0$) with the theoretical ones and comment on the results. What are the practical implications?

(2) Assume now that the yield strength $\sigma_f$ of the material is given by the stress obtained when a defect-free plate is loaded with $\varepsilon_y = 5\%$, i.e. $\sigma_f = 0.005E$. Suppose the plate fails when $\sigma_{y,\max} = \sigma_f$, determine approximately the failure moment for the four cases $b/a = \{1, 0.5, 0.25, 0\}$ and discuss the results. Particularly, discuss if the failure criterion is still applicable to the case of a crack ($b/a = 0$).

For numerical assignment 4, the plate was created simply with an extrude. The width was chosen to be 100mm, so the other dimensions per the assignment instructions became:

$$W = 100\ mm$$

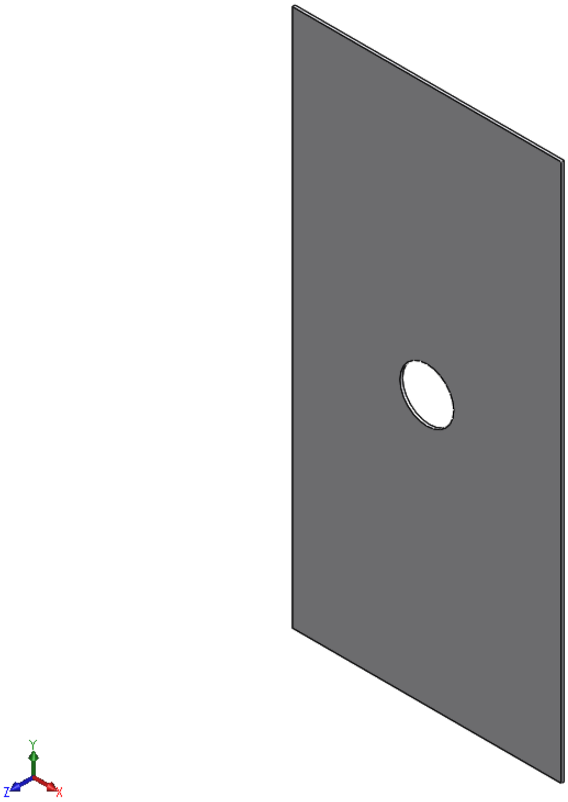
$$H = 2W = 200\ mm$$

$$a = \frac{W}{10} = 10$$

$$b = \{10, 5, 2.5, 0\}$$

The values of the elastic properties E and *v* were set to 70 GPa and 0.3 respectively.

The model was created by extruding the shape with a depth of 1 mm (z-direction), but because the simulations will be simplified per the instructions to assume place condition applies, the depth will be made not impact the results. The whole was created with a simple cut centered in the middle of the plate, 2a wide and 2b high.




The simulation was set up as a 2D simplification, to achieve plane conditions and simulation on a 2D shape. A pressure of 100 N/$m^2$ was applied as $\sigma_0$ on the bottom and the top side of the plate (y-direction)

## Part 1

*Analyze the problem with FEM and examine how the stress $\sigma_y$varies along the width of the plate, from the tip of the ellipse to the edge of the plate.*

From the instructions, the stress in the loading direction is approximately the following:

$$\sigma_y = \sigma_0(1 + 2\sqrt{a/\rho})$$

$$\rho = \frac{b^2}{a}$$

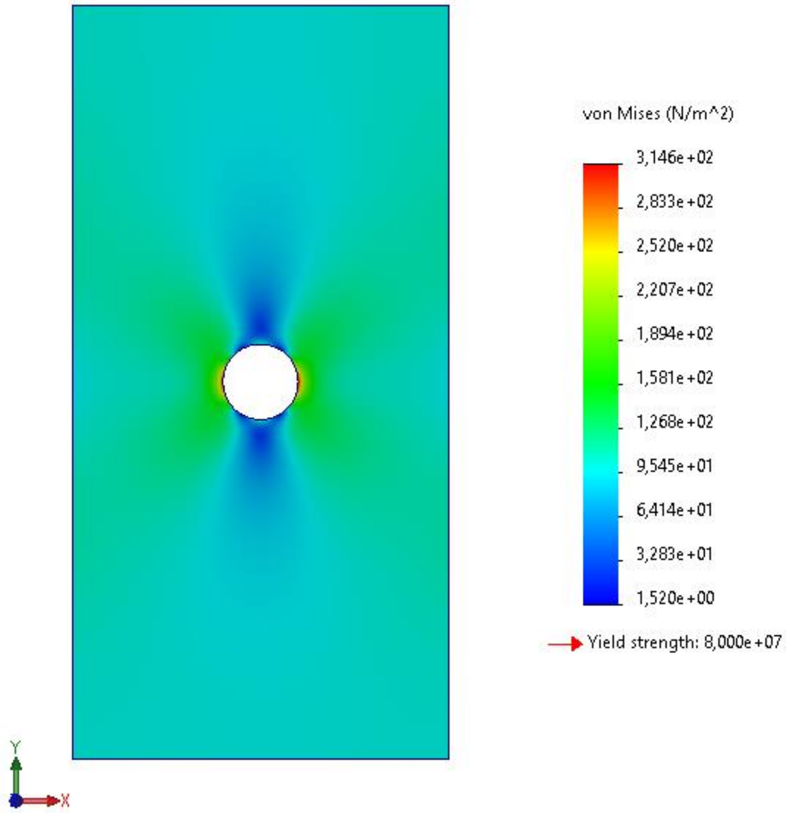
and for the different values of b, the following approximations can be calculated:

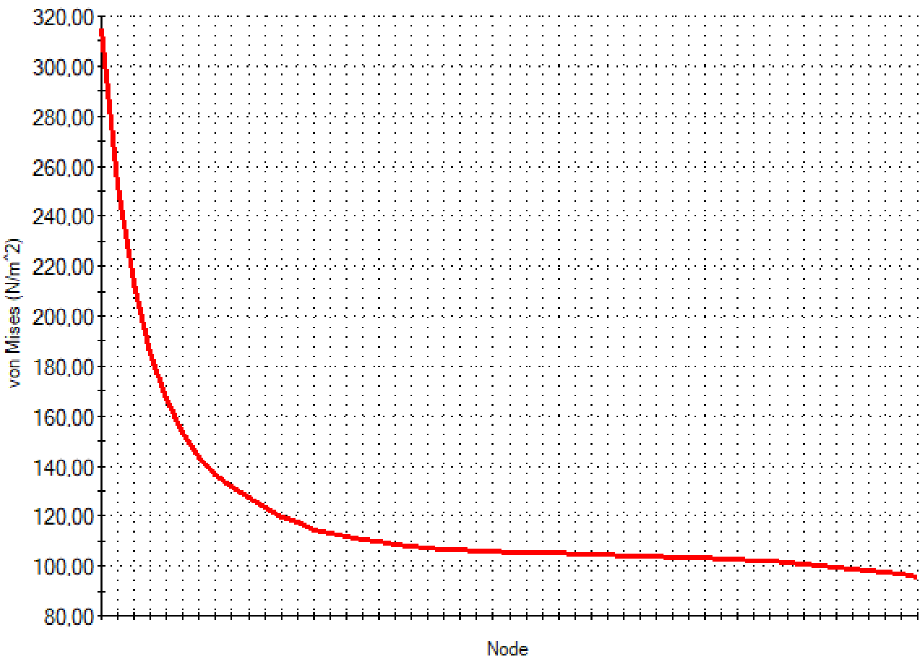
| a | b | $\sigma_0$ | $\sigma_y$ |
|---|---|---|---|
| 20 | 20 | 100 | 300 |
| 20 | 10 | 100 | 500 |
| 20 | 5 | 100 | 900 |
| 20 | 0 | 100 | Towards infinity |

Each case for each value of b will now be simulated and shown along with a graph that measures the stress along the x-axis using 50 evenly distributed points from the tip of the ellipse to the edge of the plate. The distance between the tip of the ellipse and the edge of the plate is 40 mm, that means each point is spaced 0.8 mm from each other This graph will help with analyzing how the stress varies.

## Case 1:

a = b= 10 mm, where the maximum simulated von Mises stress reached 315 Pa, compared to the calculated and approximate 300 Pa.

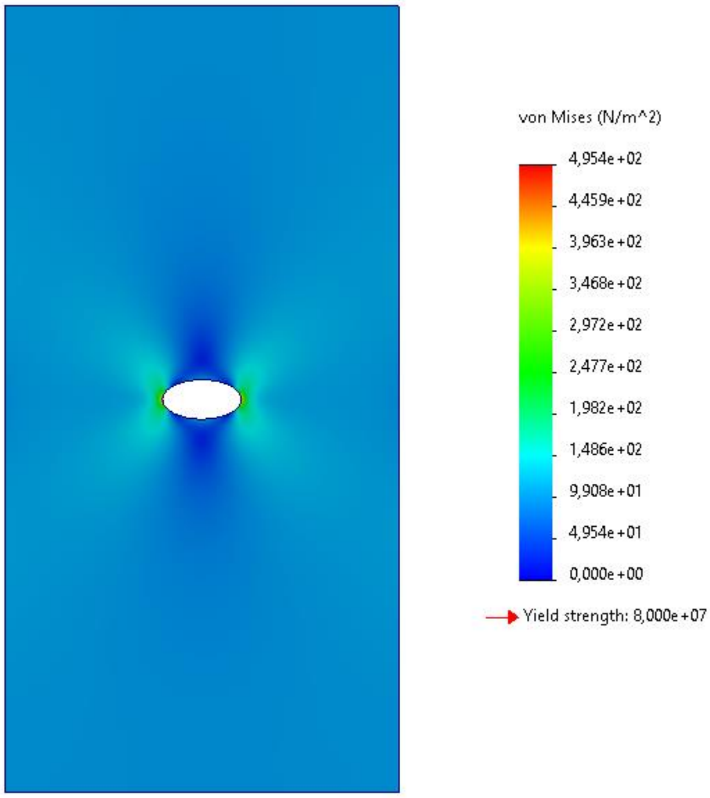


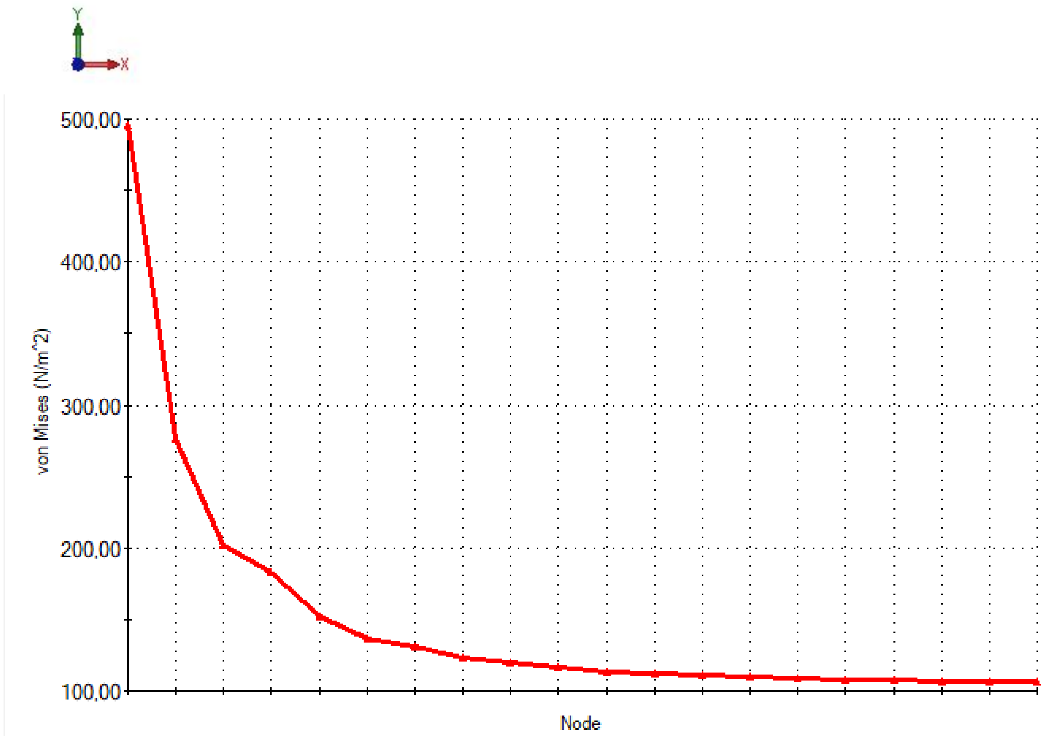




As can be seen on the graph, the stress is the highest closest to the ellipse, and quickly falls off the closer it reaches the edge of the plate, furthest away from the ellipse.

## Case 2

a = 2b, a = 10 mm and b = 5 mm. The maximum simulated von Mises stress reached 495.4 Pa, compared to the calculated and approximate 500 Pa

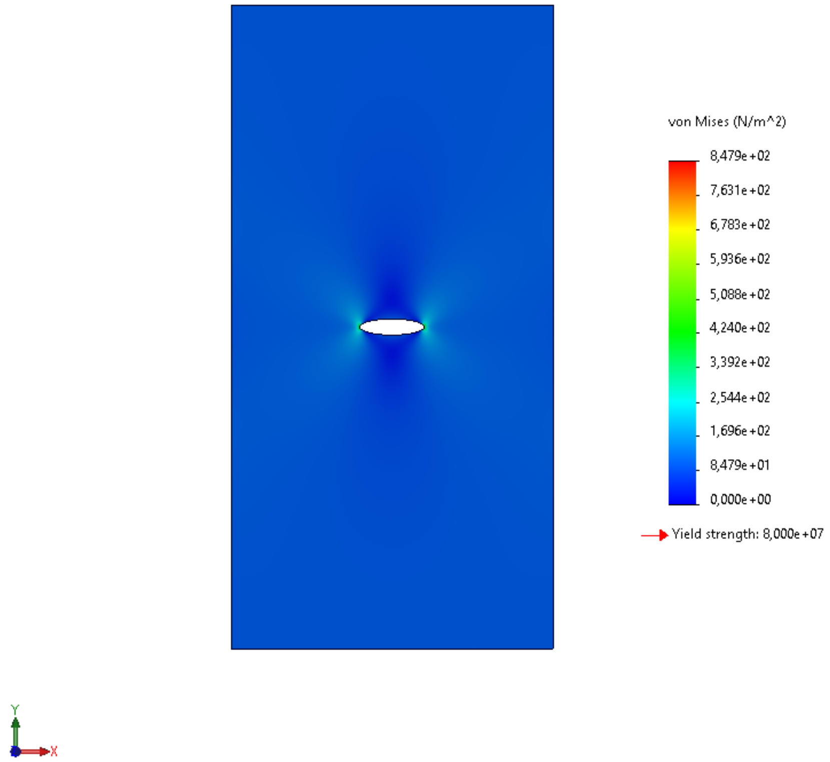


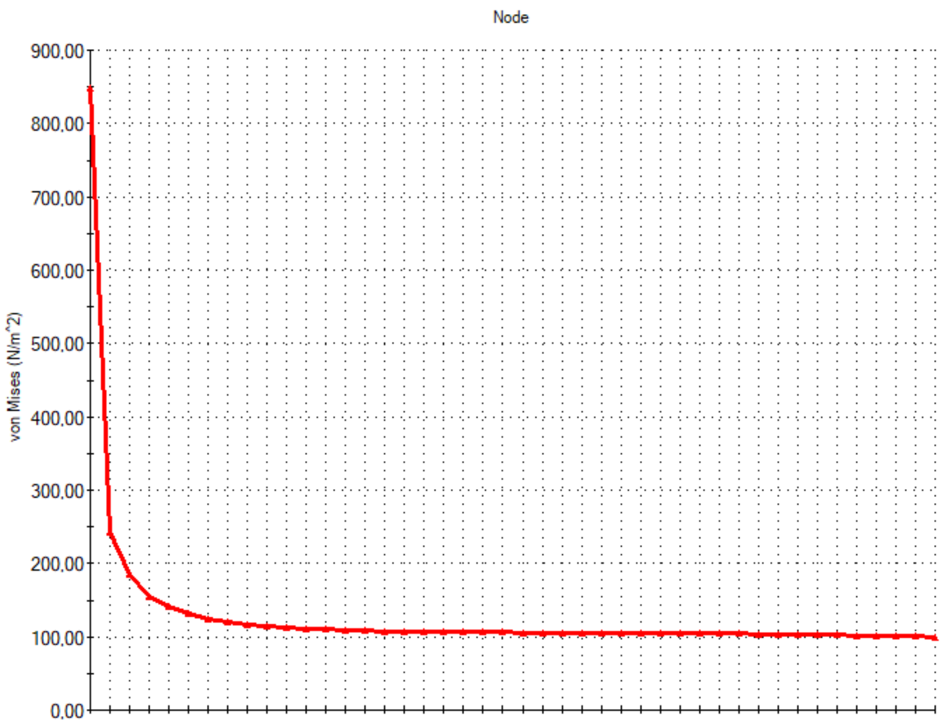




note: this graph only contains 20 points evenly distributed 2 mm apart. This was a mistake that was noticed too late. The purpose of the graph, to analyze the varying stress, is still fulfilled.

## Case 3

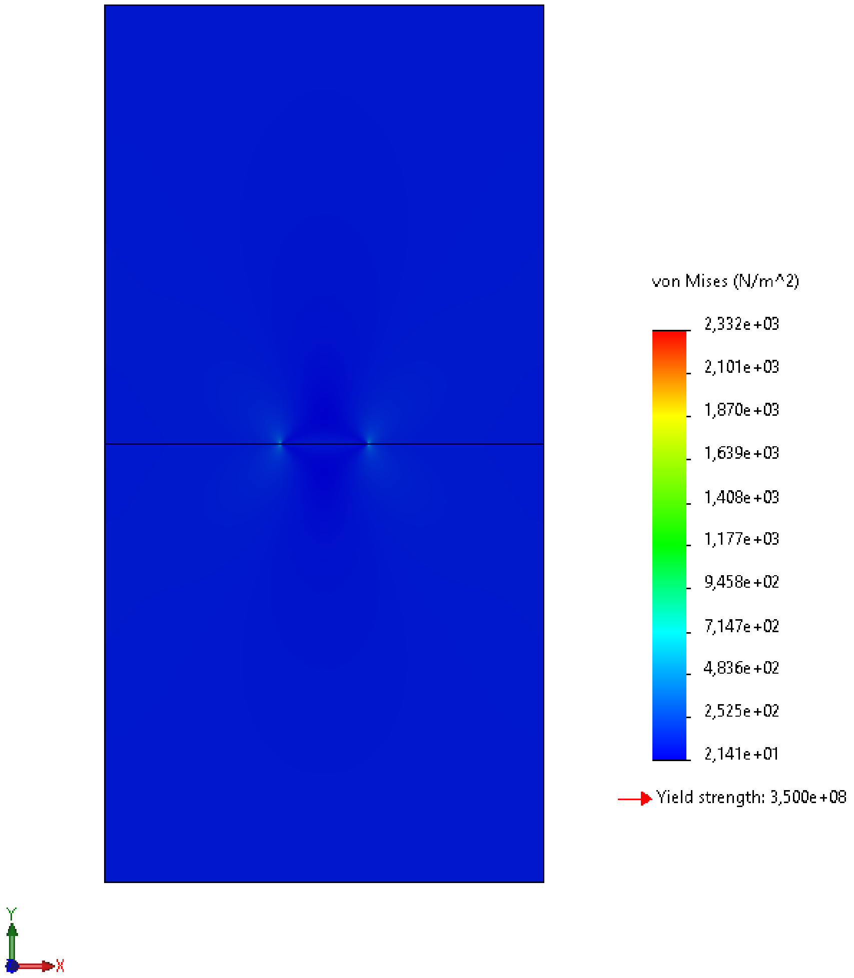
a = 4b, a = 10 mm and b = 2.5 mm. The maximum simulated von Mises stress reached 847.9 Pa, compared to the calculated and approximate 900 Pa

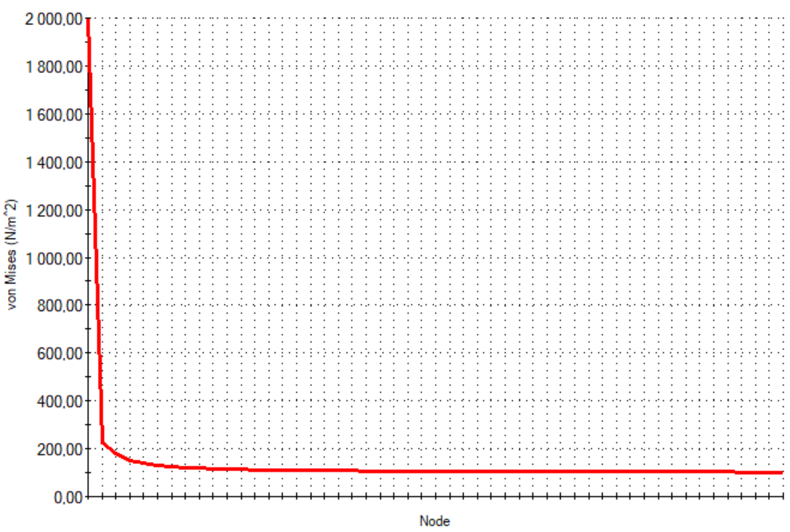


## Case 4

a = 10 mm and b = 0. The maximum simulated von Mises stress reached 2332 N/m$^2$, whereas the formula for the approximation contains a division by zero which means that the maximum stress would go towards infinity. Because of how meshing works, SolidWorks has to assume a certain size, even though you may limit that mesh size to say 0.01mm$^2$ cubes, it can never be zero and therefore the max stress will not reach values close to infinity.



## Thoughts and conclusion for part 1:

It does appear that as the value of *b* decreases which means the ellipse height decreases, the stress $\sigma_y$ gets more concentrated closer to the tip of the ellipse. The decrease of the value b also shows a change in the gradient and how quickly the stress falls off. This can be seen by comparing the graph of *Case 1* when a = b, compared to the *Case 4* when b = 0, where in the latter the line looks almost vertical. This shows how a larger but round hole/defect can be more safe compared to a much smaller crack (as in *Case 4*), which can have a much higher concentration of the load and hence be more prone to failure.

# Part 2

*Determine approximately the failure moment for the four cases and discuss the results. Particularly, discuss if the failure criterion is still applicable to the case of a crack*

The material has an Elasticity modulus of $7*10^{10}$ Pa. According to the instructions, the yield strength is given by the following:

$$\sigma_f = 0.005E$$

This would make the failure point at 3.5e+8 Pa, or 350 MPa.

$$\sigma_y = \sigma_f \rightarrow failure$$

Using the previously used analytical solution to approximate the stress at the tip of an ellipsoid,

$$\sigma_y = \sigma_0(1 + 2\sqrt{a/\rho})$$

We can approximate the loading $\sigma_0$ needed to achieve failure,

$$\rightarrow \sigma_0 = \frac{\sigma_y}{(1 + 2\sqrt{a/\rho})}$$

so for failure, if $\sigma_y = \sigma_f$:

$$\sigma_0 = \frac{\sigma_f}{(1 + 2\sqrt{a/\rho})}$$

$$\rho = \frac{b^2}{a}$$

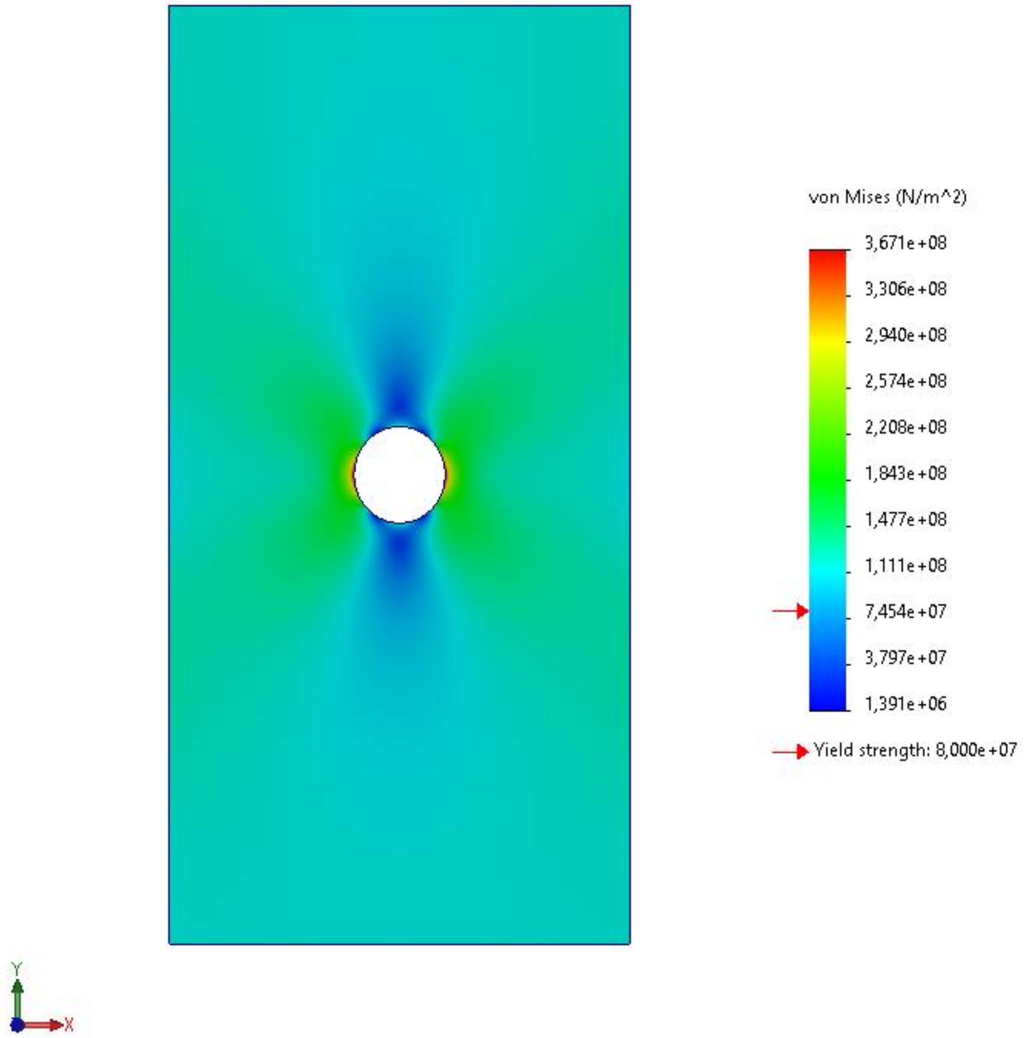
using this along with the calculated yield strength 350 MPa, the following table is calculated which shows the approximated loads for failure:

| b/a ratio | $\sigma_0$ |
|---|---|
| 1 | 116.67 MPa |
| 0.5 | 70 MPa |
| 0.25 | 38.89 MPa |
| 0 | Towards 0 Pa |

When inputting these values in each respective simulation, the following four cases happen:

## Case 1

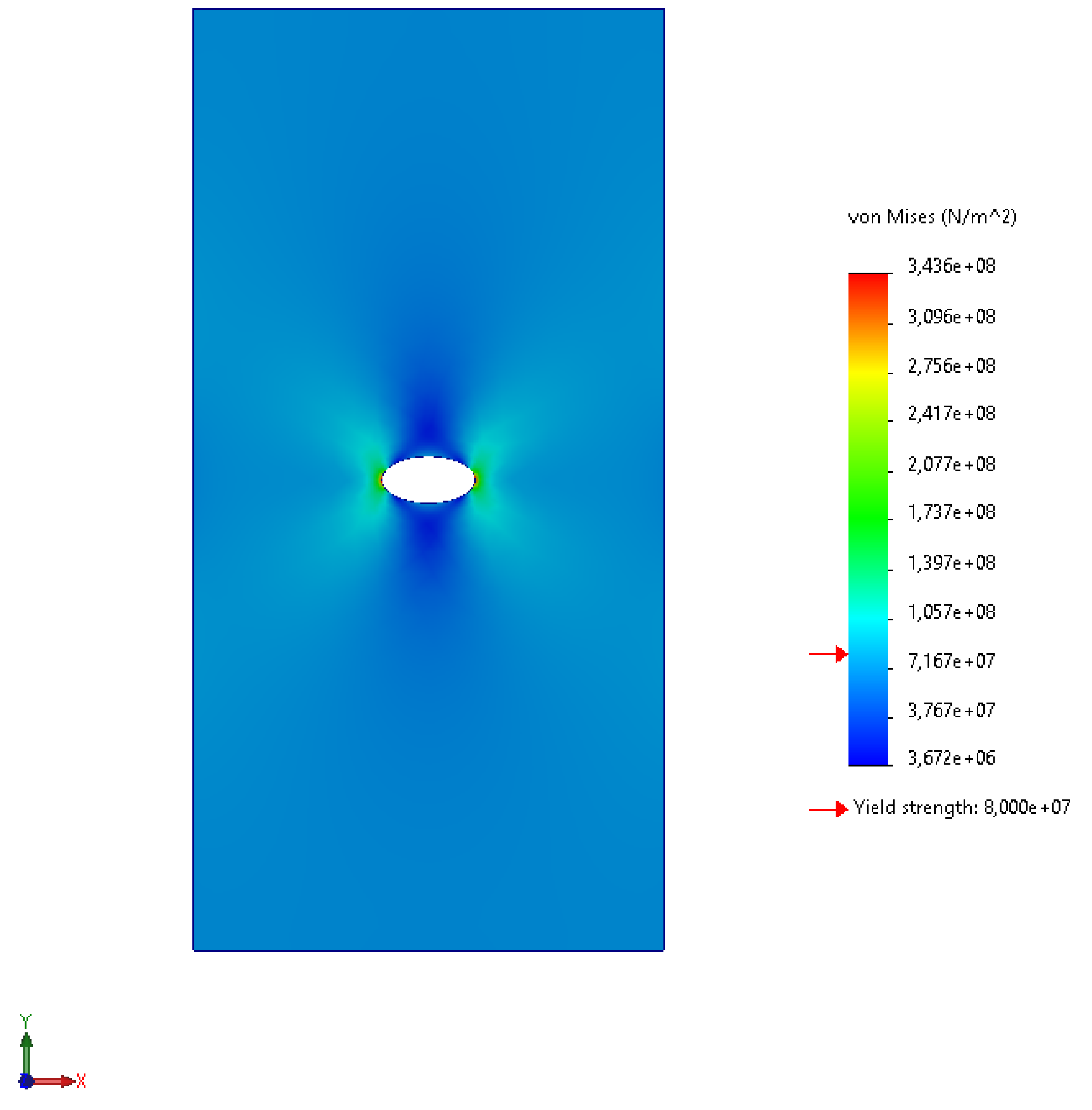
a = 10 mm, b = 10 mm. The approximated load needed for failure, 116.67 MPa, led to failure, as the highest recorded effective von Mises stress was 367.1 MPa compared to the 350 MPa needed for yielding and failure.




By instead applying $\sigma_0$ with a value of 111 MPa the resulting maximum stress was 349.3 MPa, very close to the failure moment. That means that the approximated value 116.67 was over 5% than what was needed by simulation

## Case 2

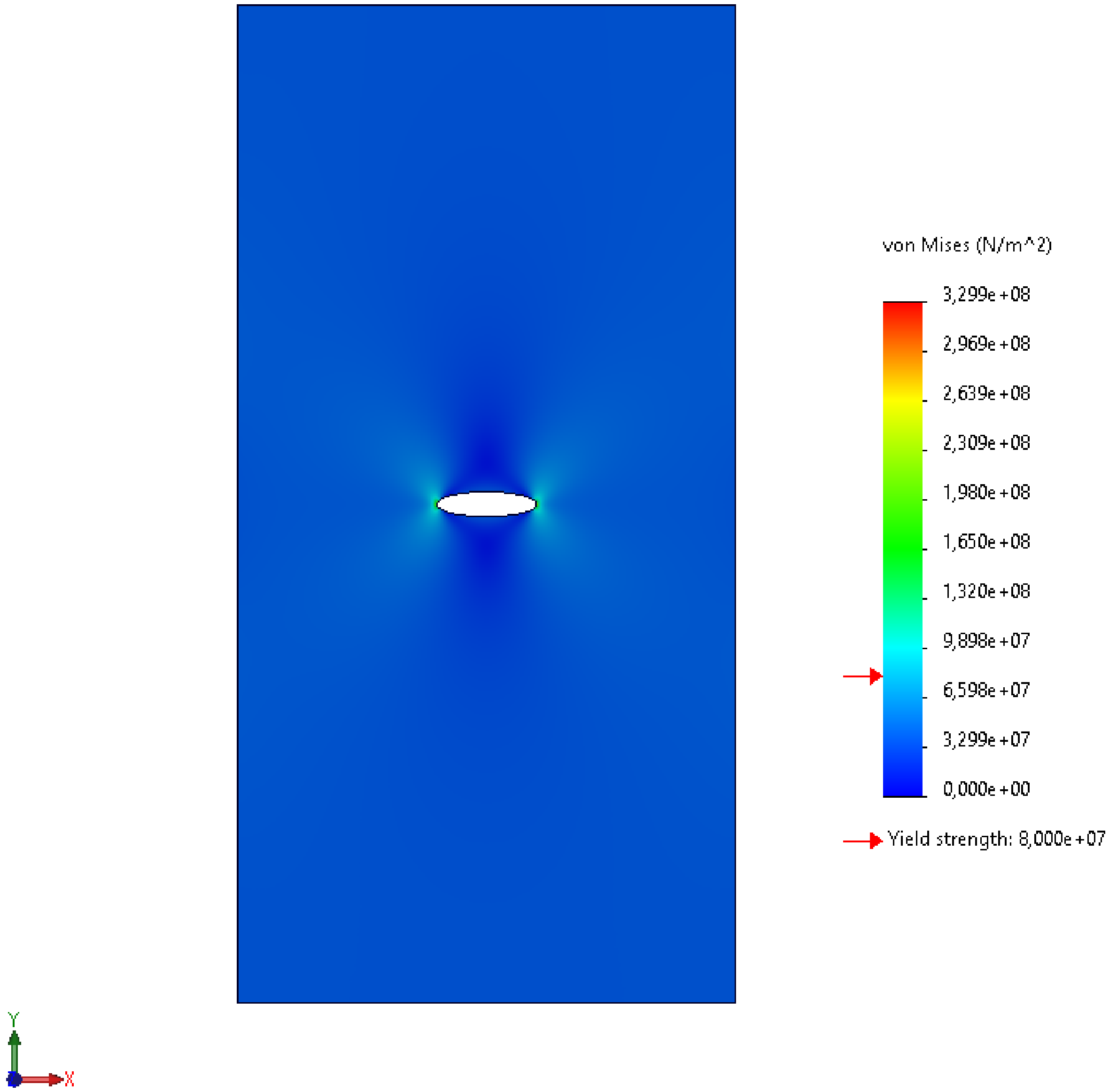
a = 10 mm, b = 5 mm. The approximated load needed for failure, 70 MPa, did not lead to failure, as the highest recorded effective von Mises stress was 343.6 MPa compared to the 350 MPa needed for yielding and failure.




while 71.3 MPa gave a resulting maximum stress of 350.0 MPa. The approximated value was very close to the simulated load needed for failure. The approximation was an underestimation this time.

## Case 3

a = 10mm and b = 2.5mm. The approximated load needed for failure, 38.89 MPa, did not lead to failure, as the highest recorded effective von Mises stress was 329.9 MPa compared to the 350 MPa needed for yielding and failure.




Using 41.2MPa during simulation gave a maximum recorded stress of 349.5 MPa, very close to the needed for failure, with 41.2MPa being close to 6 % of the approximate value 38.89.

## Case 4

a = 10mm, b = 0. Here, as the ellipsoid is 0 in height and therefore a perfect crack, the stress is even more concentrated at the tips of the crack. With the approximation formula, any value of $\sigma_0$ would result in a stress of infinity. This Is impossible to achieve, as there will always be some sort of height, how small that may be. Below, through a simulation with the smallest mesh size that the used computer could handle and calculate, 15 MPa was enough for failure. The limiting factor here is the mesh size during simulation. The smaller the mesh size, the higher the stress concentration at the points of the edges. This does show how the failure criterion is not applicable, or at least not suitable, for a crack with an infinitely small height/width like this case of b = 0 mm.